Tekla Structural Designer 2021

Product Guides

May 2021

Contents

1 Installation and licensing workflow

To use Tekla Structural Designer, you need to have a license. The installation steps you need to take are different depending on the type of license you have: for server licensing, you need to install additional tools, which are not necessary when using online licenses.

Click to expand the section relevant to you and follow the links in the text for detailed instructions.

1.1 Tekla Structural Designer license types

Tekla Structural Designer cannot be used without a valid license. There are two types of license: online and server*

- An online license is connected to your Trimble Identity. The license is delivered directly to the Tekla Online Admin tool, where your company's Tekla Online account administrators can assign the online licenses to individual users. When Tekla Structural Designer starts, you log in to Trimble's cloud to reserve your license.
- If you have a server license, you need to install a license server on your computer or on a separate server in your internal network. Tekla Structural Designer connects to your local license server to check your license.

*Local and local USB legacy licenses are still supported, but are no longer available as new purchases.

1.2 If you manage your own installation of Tekla Structural Designer

NOTE To ensure a good experience using our software, make sure your computer meets the Tekla Structural Designer 2021 hardware recommendations (page 29).

If your organization has no main user who manages Tekla Structural Designer for you, your installation proceeds like this:

1. Make sure you have your Trimble Identity set up:

   a. If you have received an email invitation from Trimble to create a Trimble Identity, follow the instructions in the email to create your account to ensure you have the correct access rights.

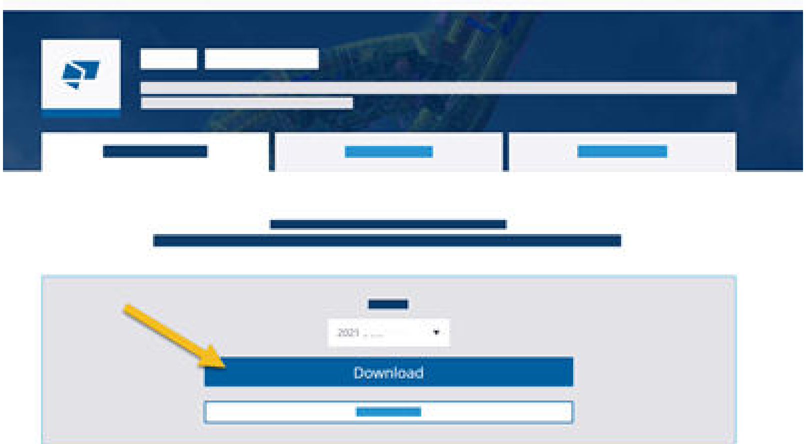
      If you wish to use a different email address for this account, create the account using your invitation first and then change the email address.

   b. If you have not received an invitation, you can create a new Trimble Identity to download the software. Click here to create a new Trimble Identity.

      To have access to your online licenses, you must be added to an organization group in Tekla Online by your organization's account administrator.

2. If you have a new account, **Log in** at https://account.tekla.com/, fill in all the required profile information, and click **Save**.

3. Download the installation package for Tekla Structural Designer from Tekla Downloads. At the site, click **Download** for a guided experience that ensures you have all necessary files.



4. If you have an online license, install the software:
    - Run the Tekla Structural Designer installer and make sure the installation finishes successfully.

      See Install and license Tekla Structural Designer (page 33) if you need more detailed instructions.
    - After you have completed the Tekla Structural Designer installation, you can now start Tekla Structural Designer
5. If you have a server license, install the software:
    - At the Server - Install the license server software on the server and activate your licenses.

      See also: How can I centrally deploy Tekla software? (page 38)
    - At the Client PCs - Run the Tekla Structural Designer installer and make sure the installation finishes successfully.

      See also: Install and license Tekla Structural Designer (page 33) if you need more detailed instructions.
6. Start learning how to use Tekla Structural Designer:
    - For a brief overview, see Get familiar with Tekla Structural Designer (page 43).

1.3 If someone manages Tekla Structural Designer for you

If your organization has a Tekla Structural Designer administrator (IT administrator or main user), you should follow their instructions for installation and licensing. You may still need to consider the following points:

- You need an account to access Tekla online services and online licenses. If your administrator has not invited you to your organization and assigned online licenses to you, ask to join so that you have access to all Tekla online services and licenses:

  Click here to create a new Trimble Identity
- In most cases, your Tekla Structural Designer administrator will prepare an installation package for you or install the software for you. Ask your administrator for further instructions.

Start learning how to use Tekla Structural Designer:

- For a brief overview, see Get familiar with Tekla Structural Designer (page 43).

1.4 Tekla Structural Designer 2021 hardware recommendations

System requirements for effective operation

CPU: Multi core Intel i5 Series or above, Xeon or AMD equivalent

- Highest affordable performance recommended.

Memory: 16GB (32GB or more recommended)

- Memory requirements are highly dependent on model content.

OS: 64-bit Microsoft Windows 8.1 / 10

- Operating systems must be running the latest service packs / updates.
- **Graphics**: 1600 x 900 resolution (1920 x 1080 or higher recommended)
    - 1GB or higher of dedicated RAM.
    - Utilizes HOOPS Visualize, a third party graphics engine available from Tech Soft 3D. To check your adapter’s compatibility visit the HOOPS developer website for HOOPS 25.x, DirectX and OpenGL requirements.
- **Disk space**: 1GB or more of free space for installation
    - Operational disk space requirements are highly dependent on model content.

Internet connection: Required for access to Online Services and some documentation.

License Service:

- Tekla Structural License Service 3.00 including Sentinel RMS 9.5

License Server: The latest version of the Tekla Structural Licence Service, at time of release, is shipped and installed with the software. If you have chosen to have a separate licence server, it is always our recommendation that you also run the latest version of the Tekla Structural License Service on it to ensure compatibility. Please see System Requirements for specific version details.

Test environments

The application is tested and supported on the following business versions of Microsoft Windows with the latest updates applied:

- Windows 10 64-bit
- Windows 8.1 64-bit

1.5 Create your Trimble Identity

You need a Trimble Identity to download Tekla Structural Designer and to use your online licenses. The Trimble Identity is connected to a Tekla Online organization (a user group for your physical organization).

Creating a Trimble Identity on your first license purchase

- If you did not have an existing Trimble Identity, Trimble sends you an email with an invitation to complete your account creation. Create your account using the link in this email and make sure you fill in all of the required user profile information.
- If you are a company's named contact, you are invited to your Tekla Online organization by Trimble when the organization is created in Tekla Online.

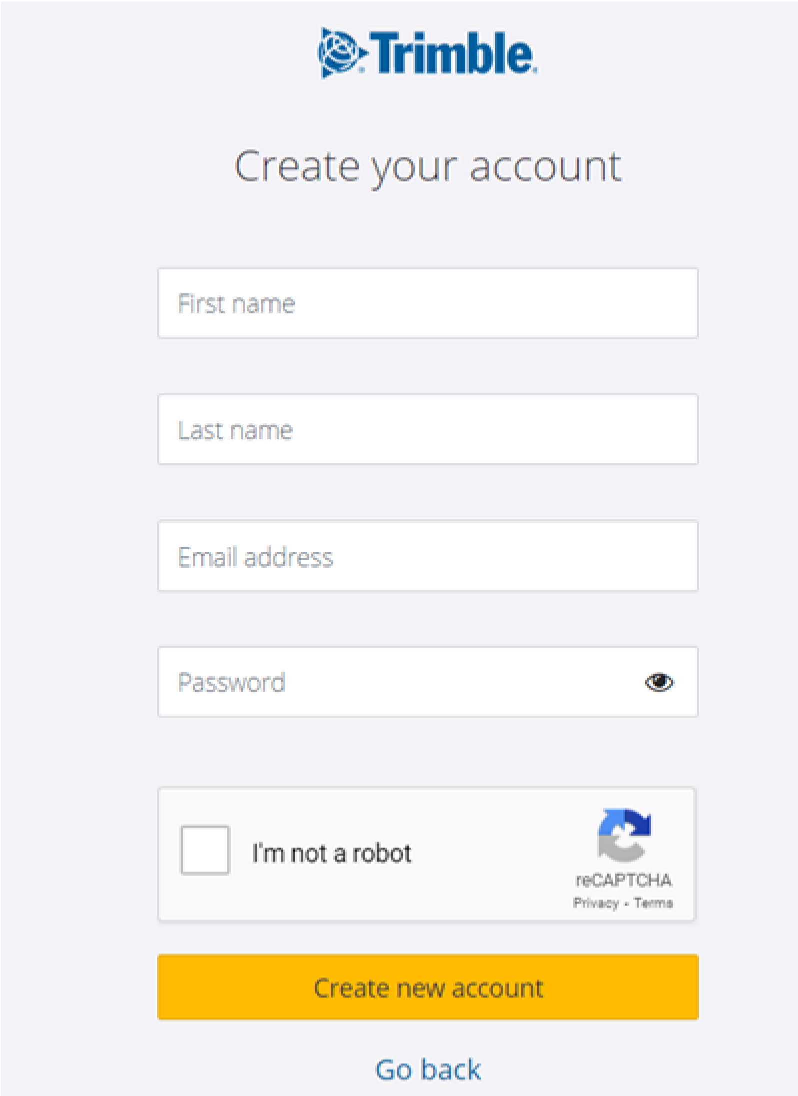
  You will receive an email to accept membership in your new Tekla Online organization. You are then responsible for managing the organization together with other administrators that you may assign.

Creating a Trimble Identity to join an existing organization

To create a new Trimble Identity account:

1. If you have received an email invitation from Trimble to create a Trimble Identity, follow the instructions in the email to create your account to ensure you have the correct access rights. Otherwise, Click here to create a new Trimble Identity.
2. Complete the **Create new account** form and click the **Create new account** button.

   If you have several different email addresses, **use your company email address**



3. Look for a verification email in your inbox and click the link provided to verify you account. You must verify your account to access your Trimble Identity.

   NOTE If you do not receive this email in your inbox, check your Spam/ Junk email folder.

4. **Log in** with your new account, fill in all the required profile information, and click **Save**.
5. Join your Tekla Online organization in one of the following ways:
   a. Switch to the **Organization** page on your user profile page, select an organization that you would like to join, and click **Send request**. If

there are no organizations listed, it means your email address does not match with any existing organization's email address.

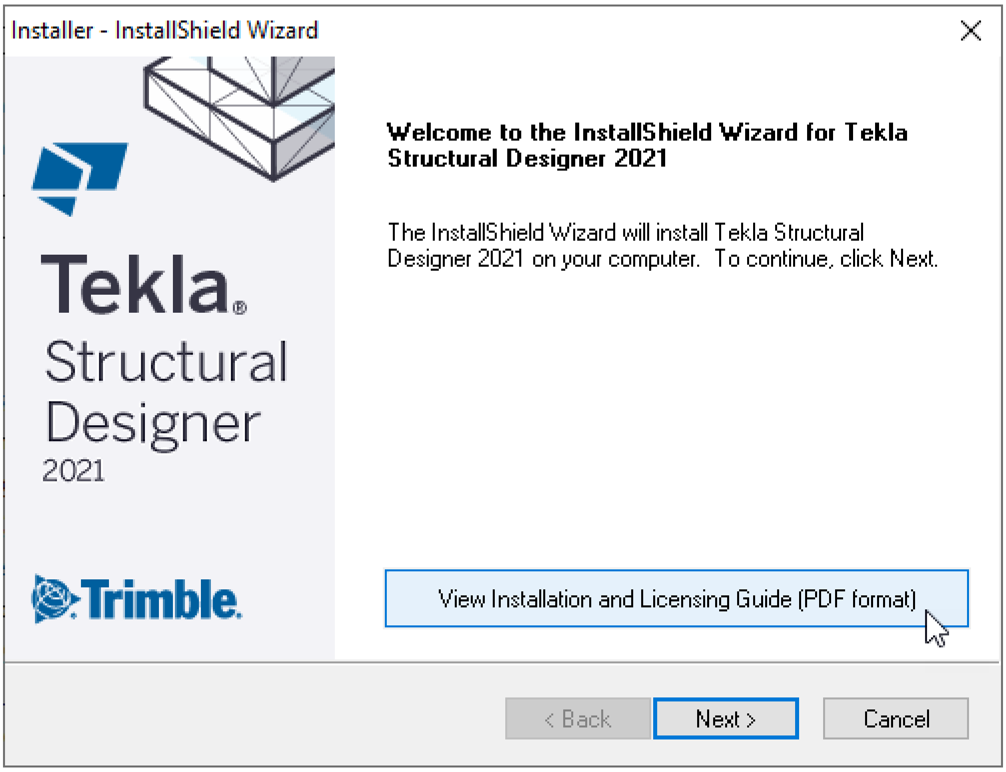
b. Ask your company's Trimble Identity administrator to invite you, and accept the invitation when it arrives via email or on your user profile pages at https://account.tekla.com/.

Your Trimble Identity is now active, and you can install and license Tekla Structures.

NOTE Membership in an organization can also affect your access to your organization's cloud-stored data, such as Tekla Model Sharing models. Make sure you do not switch between organizations unnecessarily. When available, use your company email address with your Trimble Identity.

1.6 Install and license Tekla Structural Designer

When you install Tekla Structural Designer, you will be asked to select your license method from a range of options. These are fully explained in the "Tekla Engineering Software - Installation and Licensing Guide 2021" which can be viewed during the installation process by clicking the button shown below.

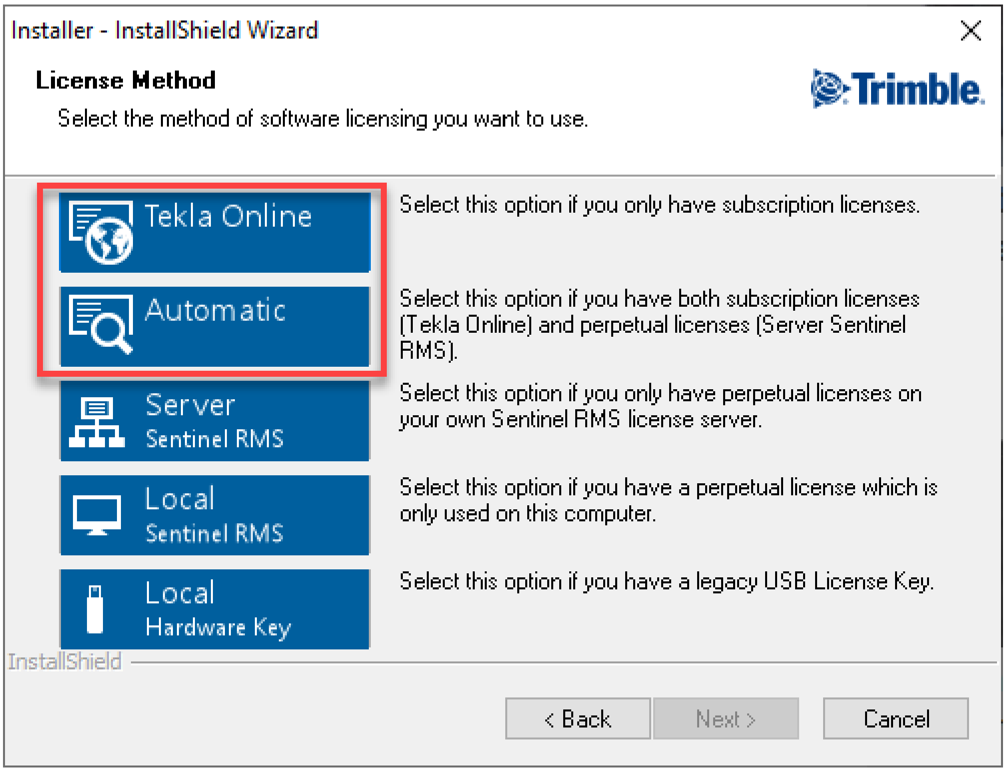


To install Tekla Structural Designer:

1. Download the installation file from Tekla Downloads to your computer.
2. Double-click the installation file to run the installation.
3. Follow the steps in the installation wizard to complete the installation.

Licensing & installation information specific to Tekla Structural Designer 2021

- **Licensing:**
  - **Perpetual Licenses** - for perpetual licenses using Sentinel RMS (both local and server), Tekla Structural Designer 2021 will require the activation of a **new 2021 version license**. You should already have received your 2021 Product Activation Key (PAK) as these are usually distributed prior to the software release. Please contact your local Service Department **now** if you do not have your PAK. To minimize any down time, we recommend you activate your PAK BEFORE installing this release.
    - **License Server Version** - if you are using **Sentinel RMS Server Licenses**, the Tekla Structural License Service on your license server **must** be updated to the new version 3.2.x or later (incorporating Sentinel RMS 9.7) to be compatible with this release. The installation for this can be obtained from Tekla Downloads. Sentinel RMS server licensing will not function correctly if this update is not performed!
      - For more information about this please see the TUA Article Tekla Structural Design Releases for 2021 & Sentinel RMS Server Licensing - the License Server must be updated and the associated FAQ Article.
  - **Subscription Licenses** - if you have **Subscription Licenses** they will use the new **Tekla Online** License Method and the 2021 license(s) should already have been added to your Tekla Online Organization. Select the “Tekla Online” license method option when installing the program as shown in the picture below (only select the “Automatic” license method if you have both subscription licenses and Sentinel RMS server licenses as described in this TUA Article).



    - In order to use a Tekla Online license you will need: a verified **Trimble Identity** & **Tekla OnlineAccount**; to be a member of your company's **Tekla Online Organization;** have an Online License assigned to you by your Organization's Administrator; to **Sign In** to your Tekla Online account when you run the program.
        - For more guidance on using Tekla Online Licensing see the following videos; Using Tekla subscription licenses and Managing Tekla subscription licenses.
- **Installation** - no previous installation of Structural Designer is required. To aid with transition, this release will install alongside existing versions and does not overwrite them.
    - **Integration**
        - **Tekla Structures** - if you wish to integrate with Tekla Structures you should install both this release and Tekla Structures 2021 for optimum performance.
            - Note that there have been significant improvements to integration - especially for rebar - in the 2021 releases. For more information about this please see the Tekla Structures Help Topic Tekla Structural Designer Import and Export.
        - **Tekla Portal Frame and Connection Designer 21** - if you wish to integrate this release with Tekla Portal Frame Designer and/or Tekla

Connection Designer (or if you use these stand-alone) you should install the new Tekla Portal Frame Designer 21 and/or Tekla Connection Designer 21 available from the Tekla Download Service (note that these too will also need a new license).

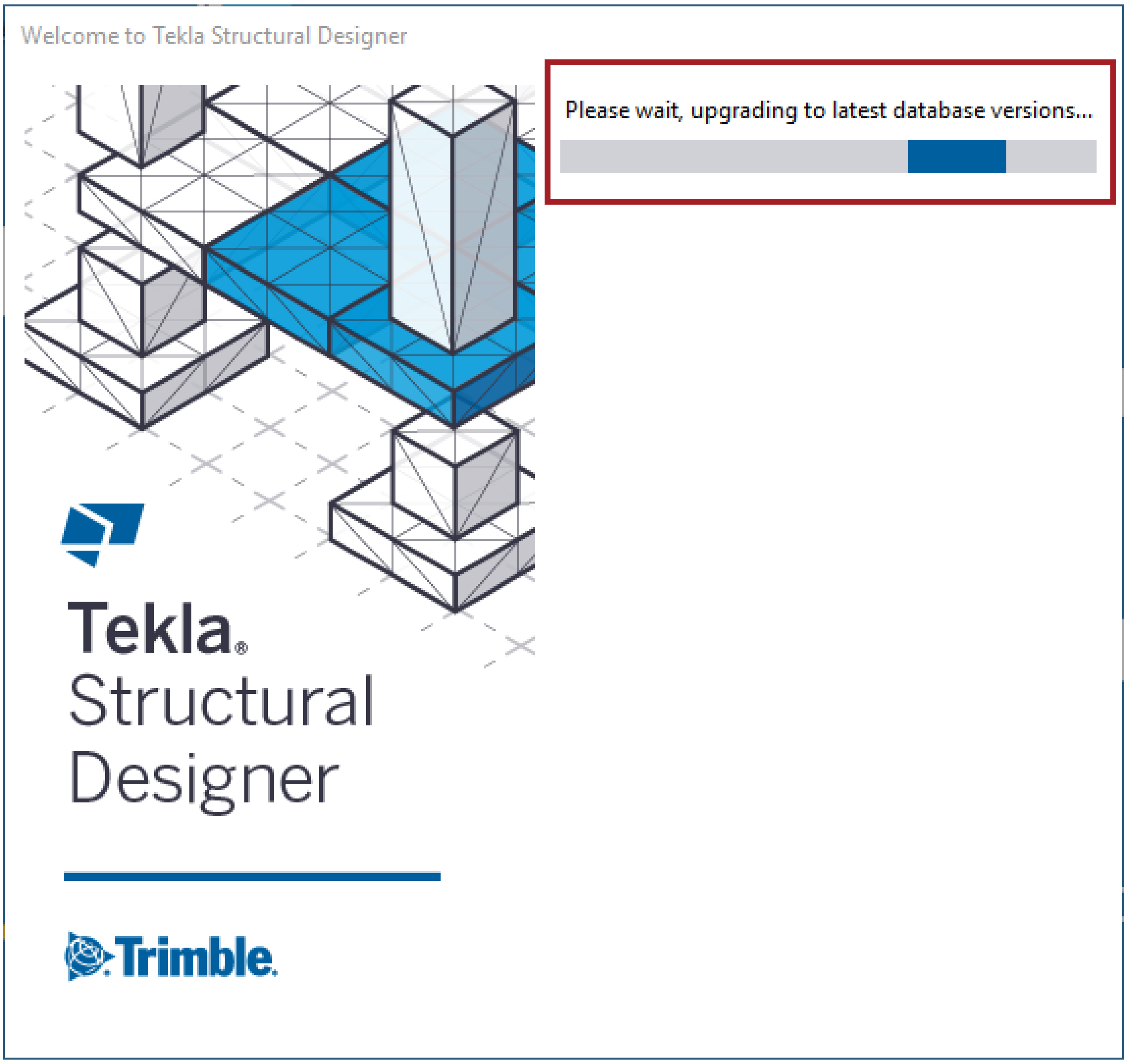
    - Note that Tekla Portal Frame Designer release 21 features a significant enhancement for the Eurocode to improve the classification of haunches for the case when they may be Class 4 slender as detailed in this release note. For full details please see the Tekla Portal Frame Designer 21 release notes.

  - **Tekla Tedds** - for integrated design using Tekla Tedds you should install the 2021 release for Tekla Tedds.

- **Previous Versions and file compatibility** - files from all previous versions can be opened in this release however, note that, once saved, they cannot then be opened in a previous release. If you wish to retain this option we therefore recommend using the File > Save As... option to save a copy of the file in the previous release and retain the original.

- **Databases** - Some Databases are updated in this release. For an existing installation of a previous release, your local Databases will automatically be updated (it is no longer necessary to do this manually via Home > Materials). This process works as follows:

  - Providing all databases can be updated automatically without user intervention, when this release is first run then the updates are applied and a message is displayed stating "updating to latest database versions" as shown below. On successful completion the message closes and the program will open.



Further help and update information

Tekla User Assistance

The Tekla User Assistance services are 24/7 online support channels for fast self-service, where you can find all the product guides, additional knowledge-base articles and instructional videos.

Tekla Discussion Forum

The Tekla Discussion Forum is the place to meet other users and discuss topics related to Tekla products. You can ask questions, contribute to the community by sharing your knowledge or get answers from support personnel.

Helpdesk

The Tekla Helpdesks support your daily operations so that our systems function as expected and any problems are solved as quickly as possible.

Software Update Service

The Software Update Service allows you to get the latest improvements to Tekla Structural Designer as soon as they are available. Providing you have an active internet connection you will be notified by Tekla whenever a new update is available.

The Software Update Service is enabled via Settings> General> Update Service on the Home ribbon.

You can also check for updates at any time via Check for Updates on the Home ribbon.

Previous versions

You can find details of requirements, enhancements and fixes for all previous releases in Tekla User Assistance (TUA) and Tekla Downloads via the links below:

- Tekla User Assistance Main version release notes
- Tekla User Assistance Service Pack release notes
- Tekla Downloads

How can I centrally deploy Tekla software?

I want to centrally deploy Tekla Structural Designer what do I need to do to be able to do this?

Tekla installations are designed to support distributed deployment scenarios using Windows Group Policy or proprietary Software Management Systems (SMS). The SMS software will vary depending upon your chosen deployment process and therefore we are unable to offer detailed technical assistance on how to implement this. What we can provide is a list of requirements / components and how to install them to allow you to package up a centralized install in your chosen deployment process.

Where is the information found?

Every release of the software has different dependencies so the best way for us to distribute this information is to ship it in the installation download. Because the vast majority of users will never need to know this we do not provide links to the information via the installation wizard - you have to unpack the download to find the information.

How do I unpack the install to extract the Distributed Deployment details?

1. Run the downloaded installation package
    - Allow the unpacking process to complete and the Installation wizard to launch
2. Cancel the Installation wizard
3. Open a Windows explorer and navigate to
    - C:\Program Files (x86)\Common Files\Tekla\Structural\Install Cache
4. In here there should be a sub folder related to the product and release you are trying to install, e.g. "tekla_structural_designer_YYYY_MMYYYY"
5. Open this subfolder and open
    - Distributed Deployment.htm

1.7 Tekla Structural Designer service packs

Service packs update the existing installed Tekla Structural Designer version software. You do not need to separately install previous service packs in order to install the most current service pack. For example, you can install service pack 2 without installing service pack 1.

- Service packs can include new features, and improvements and fixes to existing features. We recommend that all users install the latest service pack.

You can find the service packs in Tekla Downloads.

Install a Tekla Structural Designer service pack

You can install a service pack to update a version or a previous service pack. Service packs can contain new features, and improvements and fixes to existing features

If you have the related Tekla Structural Designer version or a previous service pack installed on your computer, you do not need to remove it before installing a new service pack.

To install the service pack

1. Download the service pack software installation file from Tekla Downloads to your computer.
2. Double-click the installation file to run the installation.
3. Follow the steps in the installation wizard to complete the installation.

1.8 Upgrade Tekla Structural Designer to a new version

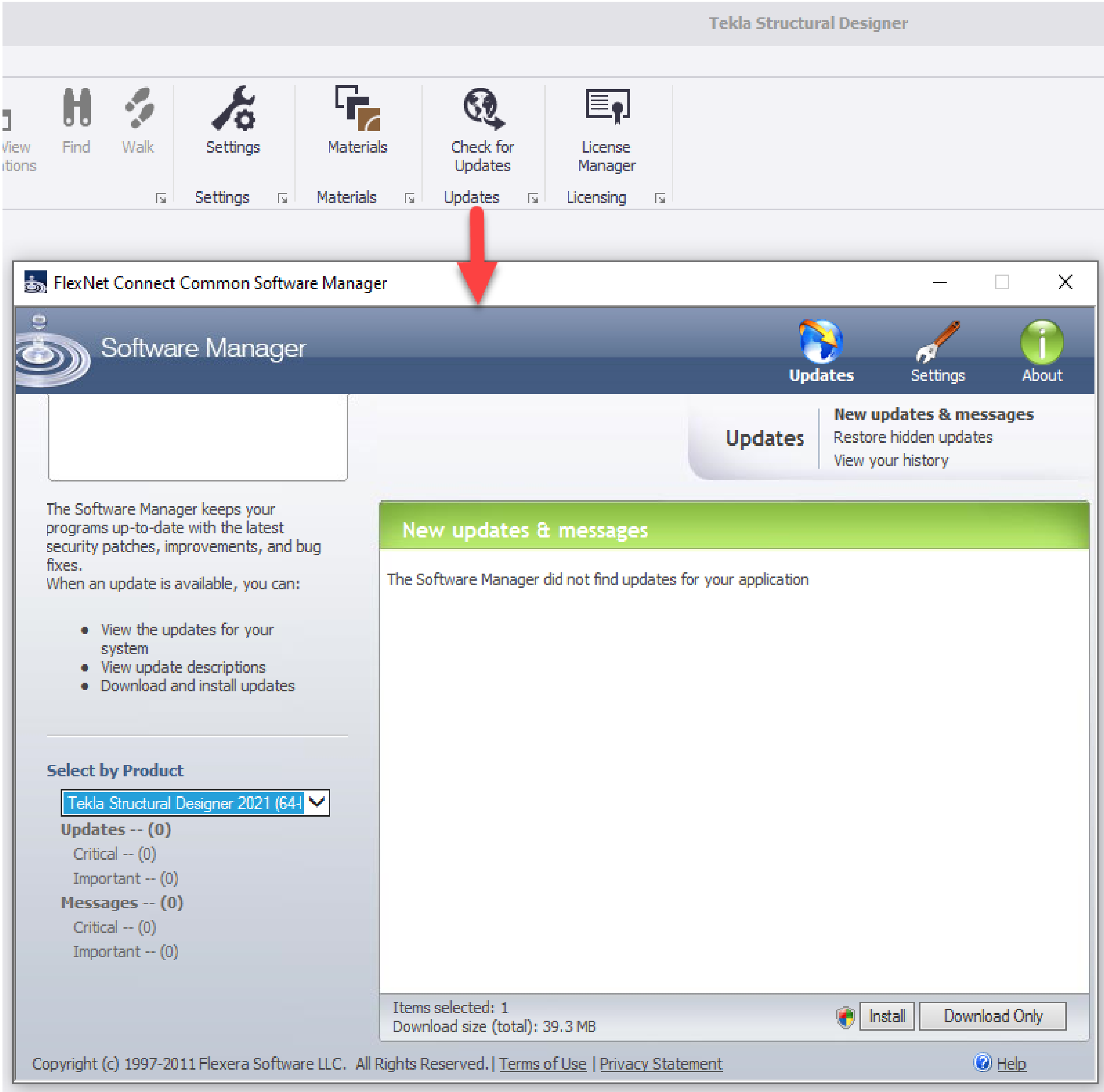
If you already have Tekla Structural Designer installed and are upgrading to a new major version please note that you can have multiple Tekla Structural Designer versions on your computer. When you install a new version you are not required to immediately uninstall the previous version.

Service packs are cumulative updates to major versions - these replace the previous service pack installation for the same Tekla Structural Designer version. So you should only have the latest service pack installed per version.

To upgrade your version, simply proceed to Install and license Tekla Structural Designer (page 33) as if you were a new user.

The Update Service

Using the Update Service is the easiest way to keep your Tekla Structural Designer installation up to date.



You can check for updates at any time by selecting the "Check for Updates" button on the Home tab of the Ribbon.

You can then install them directly from the list of updates in the Software manager.

2 Get familiar with Tekla Structural Designer

Tekla Structural Designer is an integrated model-based 3D tool for analysis and design (of both concrete and steel members) in multi-material structures.

Features include interactive modeling, automated structural analysis and design, drawing, and report creation.

Multiple design codes are supported:

- ACI/AISC
- Eurocodes
- British Standards
- Indian Standards
- Australian Standards

2.1 Philosophy

The aim is to allow you to rapidly build your model, apply loads, and design the model for an appropriate set of design forces.

On a day to day basis, you do not need to be involved with the underlying analysis models to achieve this. Instead, you can focus on the design results.

To make this possible, Tekla Structural Designer automatically creates and analyzes multiple solver models, each one being based on a different but widely accepted approach.

By designing for the forces from all the solver models, you can be confident that each scenario has been catered for.

2.2 Tekla Structural Designer way of working

Tekla Structural Designer differs slightly from traditional modeling, analysis and design process.

Traditional modeling, analysis and design process

The traditional modeling, analysis and design process can be summarized in the following phases:

1. Provide a way to input or describe the model.
2. Analyze the model.
3. Design the model.
4. Produce calculations.
5. Produce drawings.

Process in Tekla Structural Designer

In Tekla Structural Designer, the analysis and design phases are merged into a single process. As a result, the workflow is as follows:

1. Input the geometry and loads

   A key requirement these days is BIM integration, or the ability to be able to transfer data from one application to another. Tekla Structural Designer has tools to automatically import model data from Neutral Files and from 3D DXF to facilitate BIM integration.

   Of course, you can also build the model directly.

   Once the physical model has been created, the next step is loading it. Tekla Structural Designer allows you to apply a wide range of loads in a flexible system of loadcases. The system contains, among other things, a wind load generator available to automatically create wind loadcases. You can also generate load combinations automatically.

   You should also consider pattern loading. You can create patterned beam loads automatically, and create patterned slab loads manually for design of slabs.

2. Analyze and design the model

   Tekla Structural Designer automatically performs the analyses required to enable member design to proceed. This means that analysis and design are combined into a single automated process. The only exception to this rule is slab design.

3. Produce reports

   You can create a wide range of calculations. You can also tailor the calculations extensively to meet your requirements.

4. Produce drawings

   You can produce beam and column detail drawings, and member schedules.

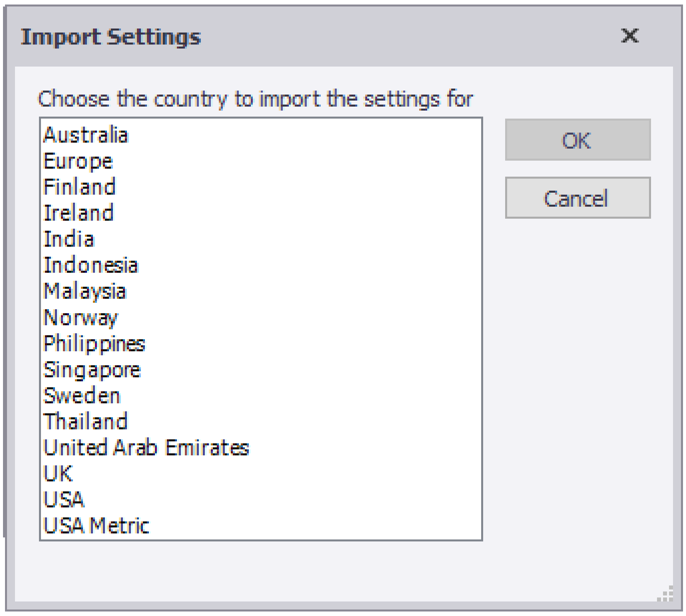
2.3 Start Tekla Structural Designer

When you start Tekla Structural Designer for the first time, you are asked to choose a country or region. Based on your selection various region specific settings such as design codes, units and default section sizes are configured. These settings, which can be changed to meet your company requirements, are applied when you start each new project.

Choose the country for your settings

1. Start Tekla Structural Designer by selecting it from the Windows Start menu or by double-clicking the desktop icon.

   A dialog box where you choose your settings appears.

2. Select a country that fits the region where your project is done.
3. Click **OK** The Tekla Structural Designer interface appears, with settings configured for the selected country.

Check or change your settings

You can check or change the settings that get applied to new projects at any time.

1. On the **Home** ribbon, click **Settings** The Settings dialog opens, displaying the settings that have been configured for the selected settings set.

2. Change the settings as needed.

NOTE Any changes that you make only apply to *new* projects, *existing* projects are unaffected. For more information about changing settings, see: Manage settings sets (page 886).

Modify project details

The **Project Wiki** allows you to modify project details, view revision history, and record revisions.

Modify project details and view revision history

You set the initial project details as you create a new project. If you want to change the project details later, or view other project parameters, see the following instructions.

1. On the **Home** tab, click **Project Wiki**.

   The **Project Wiki** dialog box opens.

   In the **Project Wiki** dialog box, you can:

   - Manage the parameters that are included in the output reports on the **Project Summary** page.
   - Track revision history on the **Revisions** page.
   - View the time at which each revision was started and last saved on the **Sessions** page.
   - View the changes associated with each revision on the **Changes** page.
   - View statistics related to the model size on the **Metrics** page and its sub-pages.
2. On the **Project Summary** page, modify the project details according to your needs.
3. Click **OK**.

Record revisions

1. Save the project before recording revisions.
2. In the file menu, click **Start Revision**.

   The **Start Revision** dialog box opens.
3. Type the revision ID and add additional notes related to the revision according to your needs.

4. In order to keep a record of the changes made in the revision, select the **Track Changes** option.
5. Click **OK**.
6. Continue to develop the model and save the model under a new name.
7. Repeat steps 2–5 as required.

Apply revision ID as an attribute for each modified element

1. On the **Home** tab, click **Project Wiki**.
2. Click **Changes**.
3. Select an appropriate UDA from the Revision Attribute droplist.
4. Either:.
    a. Click **Apply automatically** in order for the revision ID to be automatically recorded against the selected UDA on each change, or
    b. Click **Apply** to manually record the revision ID against the selected UDA for the currently elements listed in the dialog.
5. Click **OK**.

Use templates in new projects

If you are creating projects that share a common start point, you can use templates to avoid having to repeat inputting information.

In order to use a template, you must first create one.

A template can contain as much or as little information on the model as you consider applicable to serve as the start point for subsequent models.

For example, you can create a template that only contains a simple grid and the height of the first floor construction level.

See also

Create a new template (page 47)

Create a new project based on a template (page 48)

Create a new template

You can create a new template to simplify creating new models that have the same starting point. For more information, see the instructions below.

1. Start a new project (page 49)
2. Create the model data that you want to include in the template.

3. On the **Home** tab, click **Save As**.

   The **Save As** dialog opens.

4. Change **Save as type** to **Tekla Structural Designer template file (*.tsmdt)**.
5. Name the template and ensure that it is saved in the right location.
6. Click **Save**.

See also

Create a new project based on a template (page 48)

Create a new project based on a template

You can use an existing template as the base of your project. For more information, see the following paragraphs.

NOTE Before creating a new project, ensure that an appropriate settings set is active.

For more information, see Use settings sets.

---

1. On the **Home** tab, click New.

   A list of any templates in the current folder is displayed, as well as an **Open Template** option which enables you to browse in other folders for a template.

2. In the list, select the required template, or open a template from another folder.

   The template opens.

   To continue, the template should now be saved as a project.

3. On the **Home** tab, click **Save As**

   The **Save As** dialog opens.

4. Change the **Save as type** to **Tekla Structural Designer project files (*.tsmd;*.cscbd)**.
5. Name the file and ensure that it is saved in the right location.
6. Click **Save**.
7. Add the necessary building objects and loads in order to complete the project.

See also

Create a new template (page 47)

2.4 Work with projects

Start a new project

1. On the **Home** toolbar, click **New**

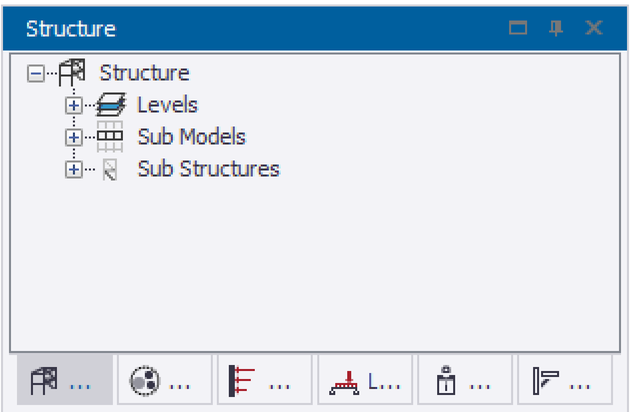
   NOTE Once familiar with the program, some users might prefer to set up a template (page 47) to use as the basis for new projects - particularly when there is a lot of commonality between projects.

   Your new project opens and you will see:

   - Two scene views (page 57): **Structure 3D** and **St. Base (Base) 2D**
   - A tabbed (page 59)
   - A (page 60)

2. Click the **Structure** tab under the **Project Workspace**

   A **Structure** tree is displayed in the **Project Workspace**.



3. In the **Structure** tree, click **Structure**

   The **Properties** window displays the **Structure** properties.

4. In the **Structure** tree, click **Levels**

The **Properties** window changes to display **Levels** properties - at this point there is only only level in the model.

NOTE The units in which the levels are displayed, (along with other settings), have been copied into the model from the settings set that was active when the project was started. These are known as 'model settings'. Model settings apply to the current project only and can be edited to suit the model as required.

What to do next:

- Check or change model settings in your project (see below).
- Try setting out some grid lines (page 179) in the **St. Base (Base) 2D** view.
- Next, double-click **Levels** in the **Structure** tree to create some levels (page 175).
- With grids in place and levels established, you can then open views of the other levels (page 86).
- You are now ready to begin creating the model (page 174).

Open a project

You can have one project open at a time. If you open a project and already have one open, Tekla Structural Designer prompts you to save the first project.

NOTE You can quickly open a recently used project, simply be selecting it from the 'Recent documents' list that is displayed on the **File** menu.

---

1. On the **Home** toolbar, click **Open**
2. Browse to the required folder, then select a project in the list.
3. To open the selected project, click **Open** or double-click the project.

   Your project opens and you will see:

   - The scene views (page 57) and (page 60) as they were displayed the last time the project was saved.
   - The (page 59) as it was displayed the last time the project was saved.

Check or change model settings in your project

You can check or change the model settings in the current project at any time.

1. On the **Home** ribbon, click **Model Settings**

   The **Model Settings** dialog opens, displaying the model settings that have been copied in to your project from the active settings set.

2. Change the model settings as needed.

NOTE Any changes that you make only apply to the *current* project, *new* projects are unaffected. For more information about changing model settings, see: Apply and manage model settings (page 877).

2.5 Work with autosave and backups

Backups ensure that whatever happens there can be previous versions of the model that can be reverted to.

An added benefit of backups is that they provide a limited (per user setting) history of the model.

Following a program error, an autosave provides a single shot at reverting back to a working model.

The difference between autosave and backup

By having both features the user can be confident that they always has a file with the most recent changes and a backup if that fails.

Autosave	Backup
The auto save file is a single temporary file constantly overwritten after a specified interval.	Multiple backup files are created which are only overwritten when a specified maximum number is reached. Each backup is only created if, after the specified interval, there have been changes since the last backup.
Provides a single shot at reverting back to a working model following an error.	Can be opened as and when required.
Once the model becomes corrupt the autosave file won't help.	Backups can be used to restore versions of the model prior to corruption.
The autosave file is a core save of the model.	Similar to autosave, backups are also a core save of the model.
Typically saved at shorter intervals than backup.	Typically saved at longer intervals than autosave.

Enable autosave

1. On the **Home** toolbar, click **Settings**
2. Under **Autosave**, click **General** > **Autorecover**
3. Check the **Enabled** box and set the **Interval** as required.
4. Click **OK**.

Enable backups

1. On the **Home** toolbar, click **Settings**
2. Click **General** > **Autorecover**
3. Under **Backups**, check the **Enabled** box and set the **Interval** as required.
4. Accept the default location, or browse to another location if required.

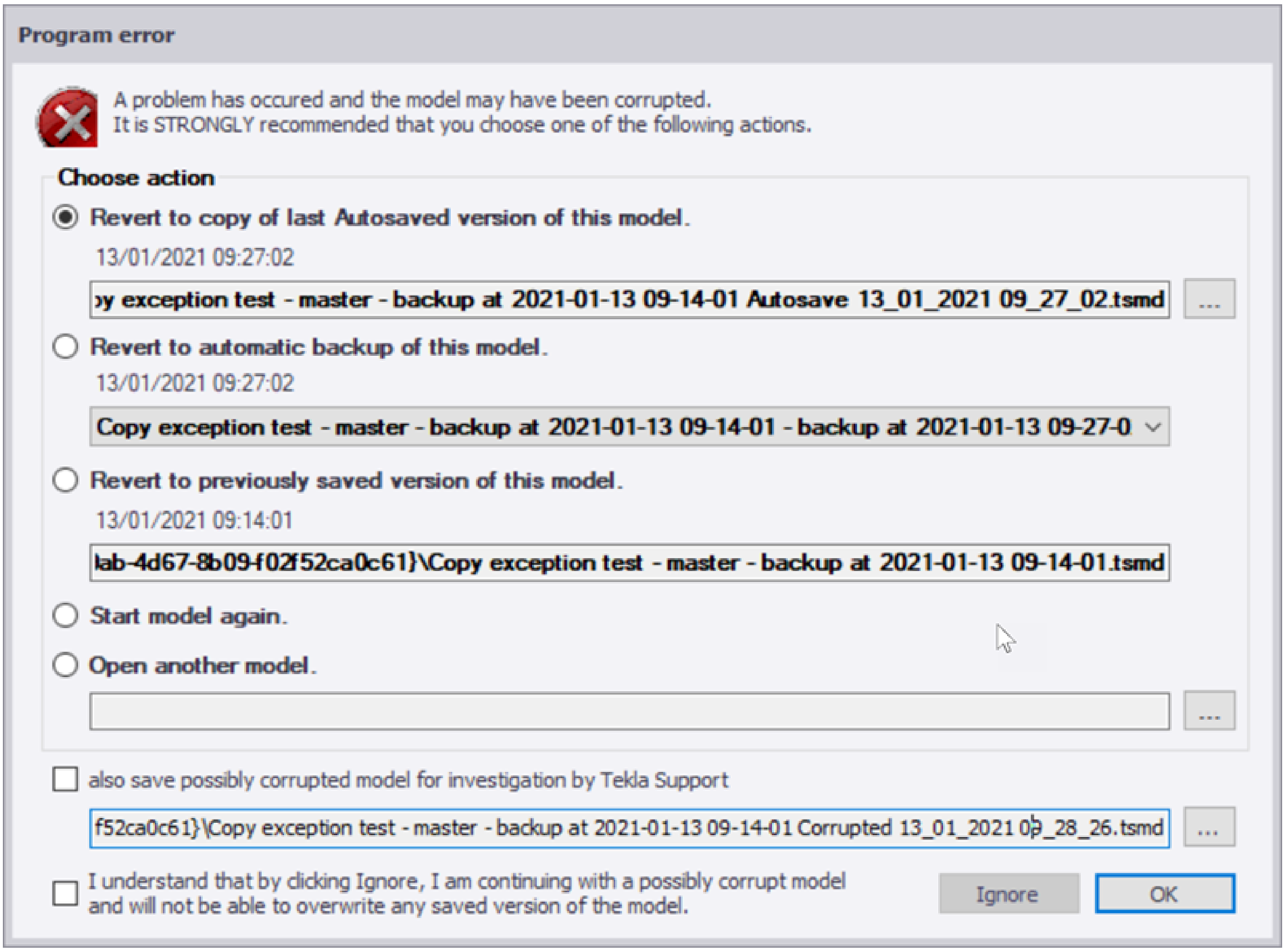
   Both local and network locations are allowed.
5. Click **OK**.

   NOTE Backup files are saved with read-only permission.

   NOTE If required you can adjust the limit on the number of backups. Once the limit is reached the older backups start to be automatically deleted.

Automatically restore an autosave or backup following a Program error

To minimize the potential for lost work and possible model data corruption, a 'Program error' dialog can be displayed when an error is detected. As shown below, when an error is detected you are presented with a number of action options which cater for the potential situations of the model.



We strongly recommend that you **do not click Cancel**, as you would then be working with a potentially corrupted file. Instead we advise that you always choose one of the action options and then **OK**.

- Where an autosave file exists, the top option will be available and is typically the recommended choice, (although depending on the intervals you have set, if both an autosave and a backup exist, it is possible the backup could be the most recent version of the model, in which case the second option should be used).
- If a previously saved version of the file exists, the third option will be available (whether or not autosave or backup files exist) listing the last

previous save of the model. This is the recommended choice if no autosave or backup file exists.

- If there is a preferred previous file version the engineer wishes to revert to, this can be selected via the option “Open another model”.
- If the model has only just been started, then none of the above scenarios apply and so the remaining option “Start model again” is the recommended choice.
- There is a final option to save the possibly corrupted model for submission to your local Support Team for investigation. We would be most grateful if engineers could use this option to help us with continuing to improve the program. When submitting such a file, please try and include as much information as possible about the steps undertaken in the program just prior to the error.

Manually restore a backup

Provided you have the backup facility enabled you will have previous versions of the project available to revert to if required. Backups work by periodically saving the model after a specified interval. Each backup is saved to a new name up to the point when the user definable maximum number of backups is reached, after which the older backups are overwritten.

1. On the **File** menu, click **Model Backups** The available backups for this Model ID are listed.

   NOTE The list is for information only, backups cannot be restored directly by selecting them from the list.

2. Click on the Backup path: to open the folder containing these backups.
3. Double-click on the required backup file to open it in another session of Tekla Structural Designer (you may need to exit from the previous session at this point if you only have a single license.) The backup model opens, but note that this is read-only.
4. Click **Save As** from the **File** menu to save the backup to a new name.

2.6 Get familiar with the user interface

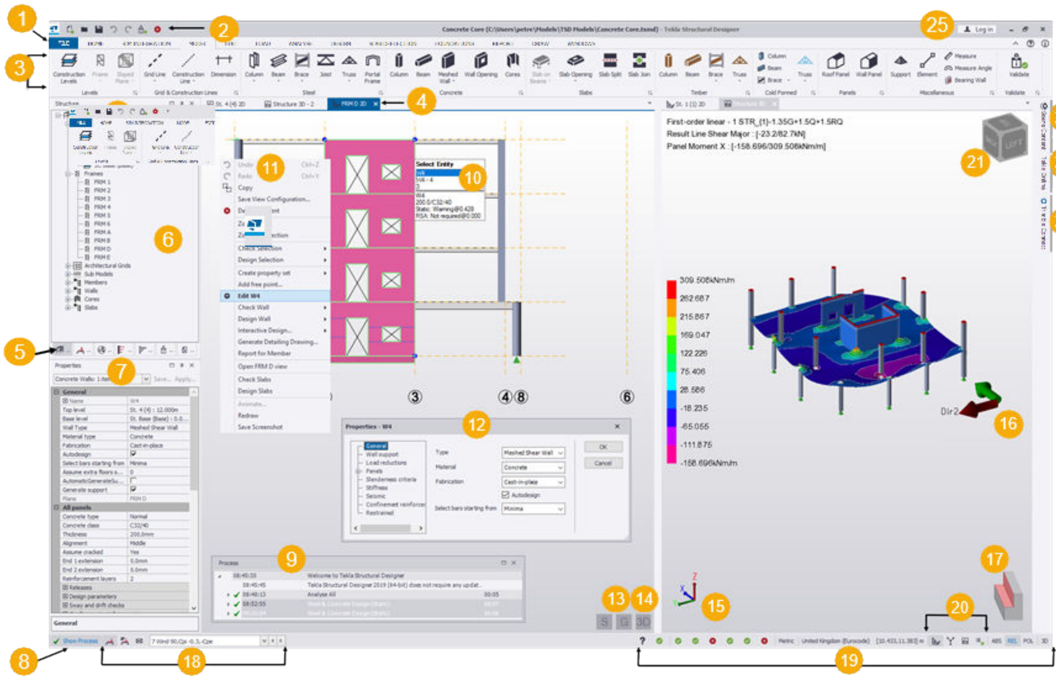
Before creating a new project, we recommend you take a moment to familiarize yourself with the user interface as explained in the following topics:

- Interface components (page 55)
- How to use the project workspace (page 70)
- How to manage scene views, view modes and scene content (page 85)

- How to hide, re-display and move windows (page 98)
- Keyboard shortcuts (page 100)

Interface components

The main components of the Tekla Structural Designer user interface are labeled in the image below:



1. File menu

The **File** menu contains those commands that you can use to perform file-related operations.

Button	Description
New	Creates a new blank project from scratch.
Open	Opens an existing project.
Save	Saves the currently open project.

Button	Description
Save As	Saves the currently open project with a new name, or as a template.
Save Model Only	Saves smaller model-only files without the analysis results. The model-only files can easily be shared amongst the project team.
Close	Closes the currently open project.
Print	Prints the currently open project.
Start Revision	Records changes to this revision of the project.
Send As Email	Creates a new email with the project attached.
Contact Support	Automatically uploads your model file and sends this, together with your issue description and key information (needed by Support to assist you) about your installation and system, to your Local Support Team as defined in your Tekla Online Profile.
Exit	Prompts you to save any open project, and then closes Tekla Structural Designer.

2. Quick access toolbar

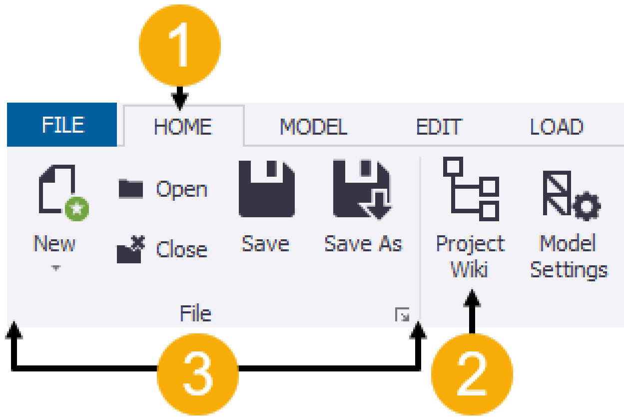
The **Quick Access** toolbar displays commonly used commands, listed below:

- **New**
- **Open**

- 
  Save
- 
  Undo
- 
  Redo
- 
- 
- 
  Delete

3. Ribbon

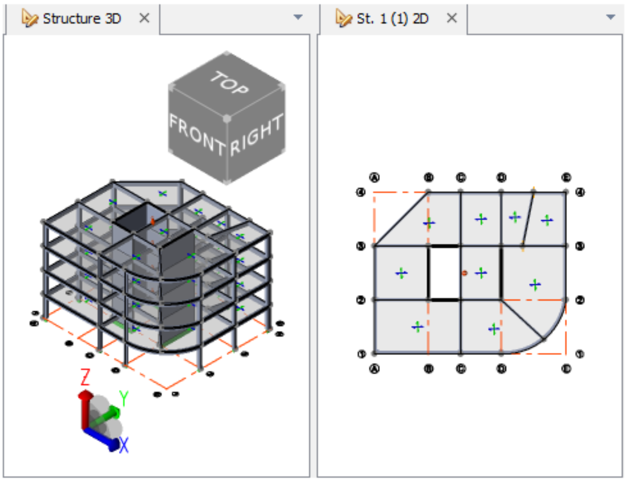
The ribbon consists of a number of toolbars, placed on various tabs. Each toolbar contains related commands organized into logical groups.



1. Toolbar tab
2. Command
3. Group

4. Scene views

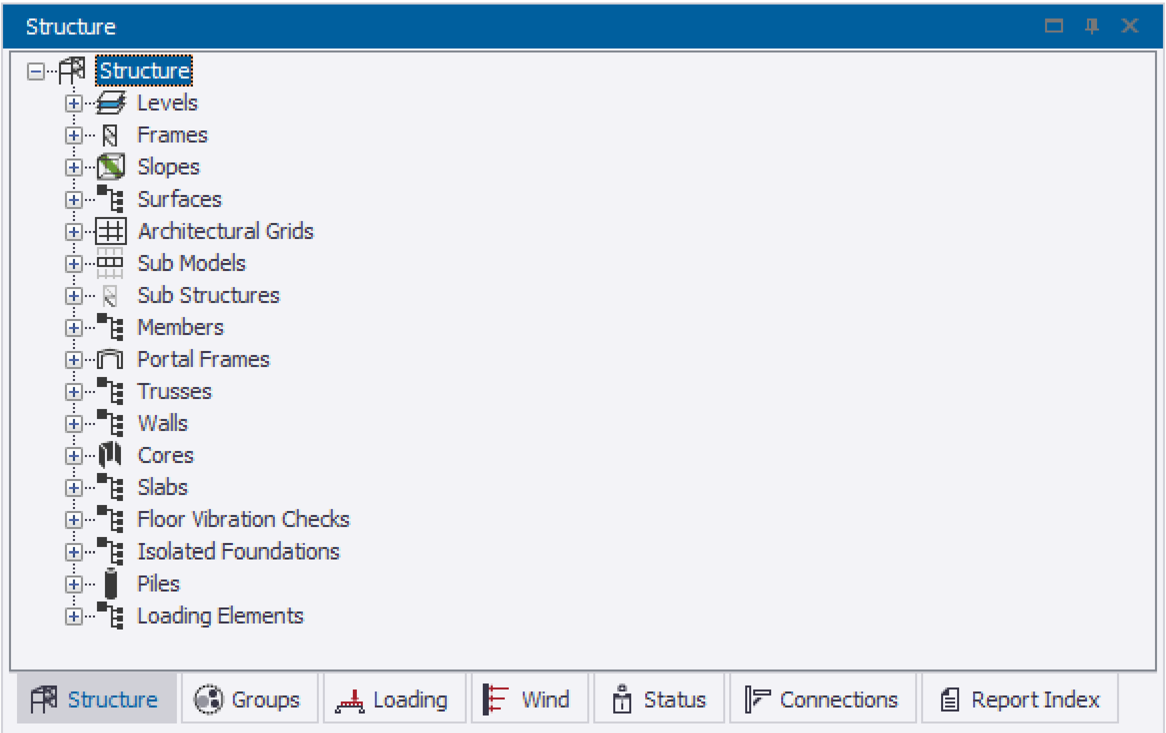
2D views, 3D views and solver views, of the model, sub models, frames, construction levels and individual members can be displayed in tabbed windows.



For more information, see: How to manage scene views, view regimes and scene content (page 85)

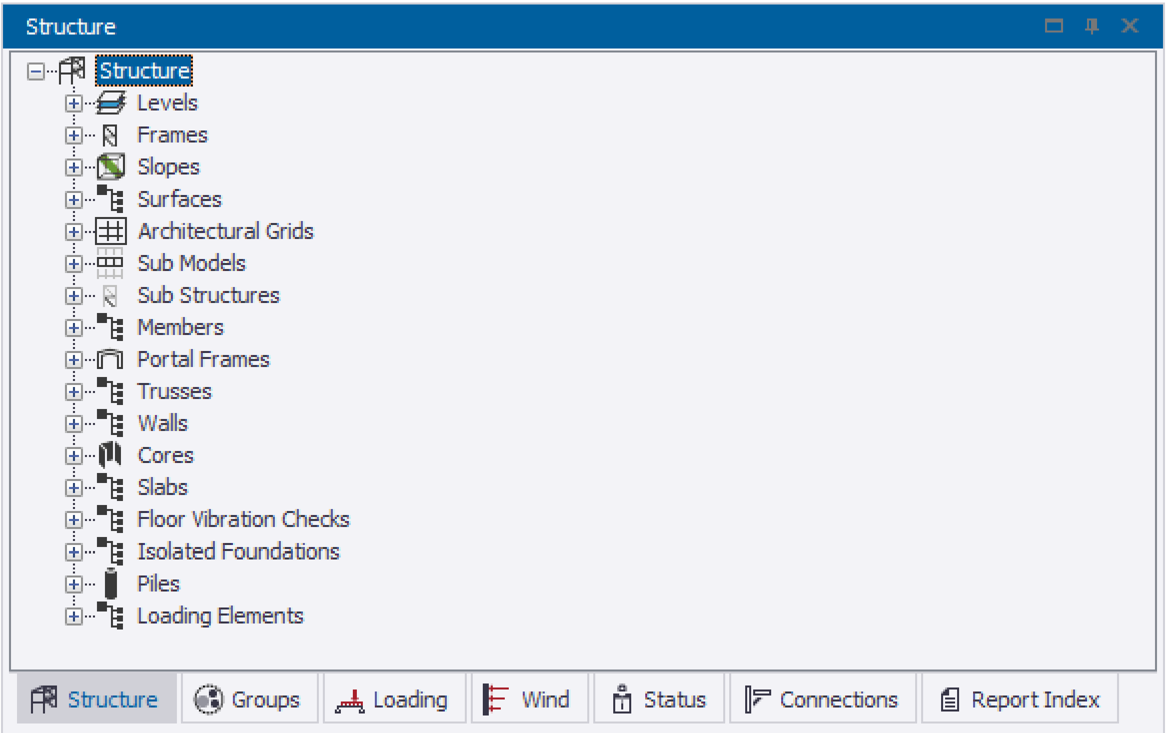
5. Structure tree

The **Structure** tree refers to the hierarchy of information displayed on the Structure tab in the **Project Workspace** . Similar 'trees' are displayed for the other tabs also.



6. Project Workspace

The **Project Workspace** is a central location of organizing the entire model into a hierarchical structure.

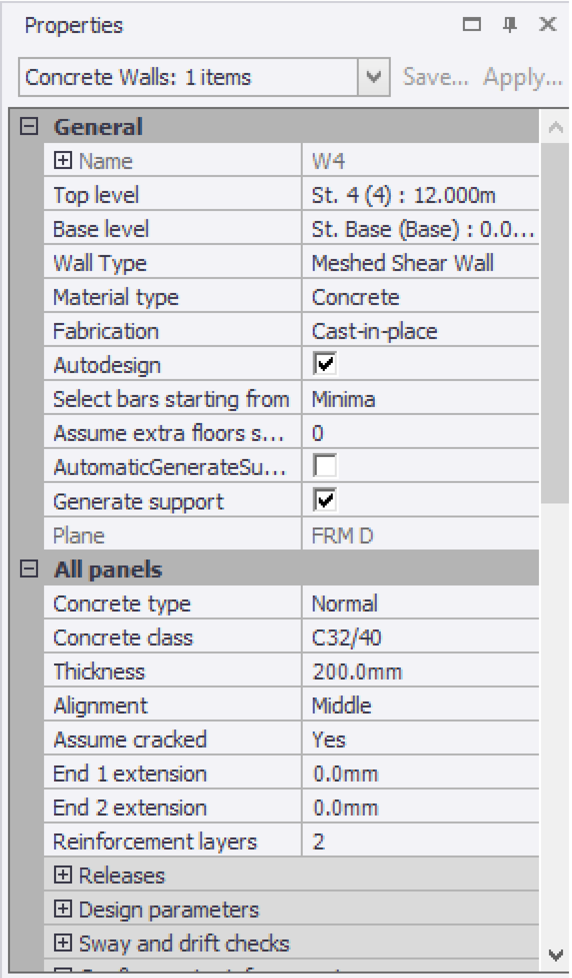


It provides access to a range of functions and is a key tool in creating and controlling your model.

For more information, see: How to use the project workspace (page 70)

7. Properties window

The **Properties** window facilitates the input, review and editing of model properties.



The **Properties** window is used to:

- input data when a command is run from a toolbar.
- review or edit existing properties when individual or multiple items are selected in the active scene view.
- review or edit properties when a branch of the **Properties** window is selected.

The properties displayed vary according to the selection. You can edit all of the properties, unless they are greyed out.

A key feature of the **Properties** window is that it enables the editing of multiple items at the same time. Existing properties of selected items are only displayed when if are identical for all selected items. If the properties differ, the property field is left empty.

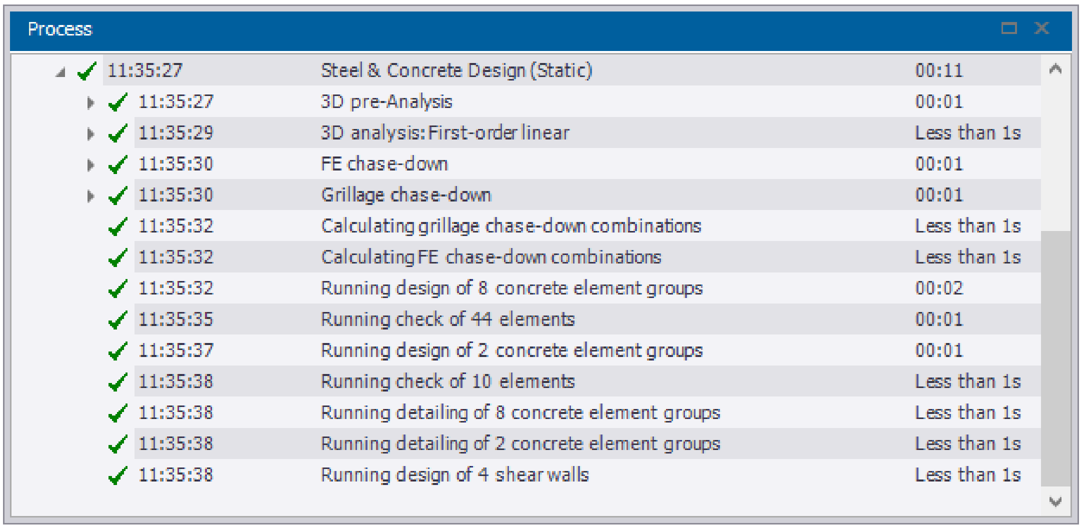
By default, the **Properties** window is docked at the bottom left of the main window, but you can reposition it if required.

8. Show Process button

Click to display the **Process Window**, described below.

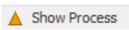
9. Process Window

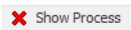
Initially, the **Process Window** is minimized. It can be displayed by clicking the **Show Process** button, located at the left end of the **Status bar**.



When you analyze or design the model, each step of the process is logged and displayed in the window.

When the window is minimized in the **Status bar**, any detected warning and errors are flagged. See the flags below:

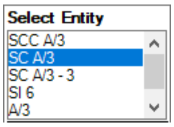
- 

- 

Such warnings and errors should always be fully investigated, as they may have an adverse affect the design.

10. Select Entity tooltip

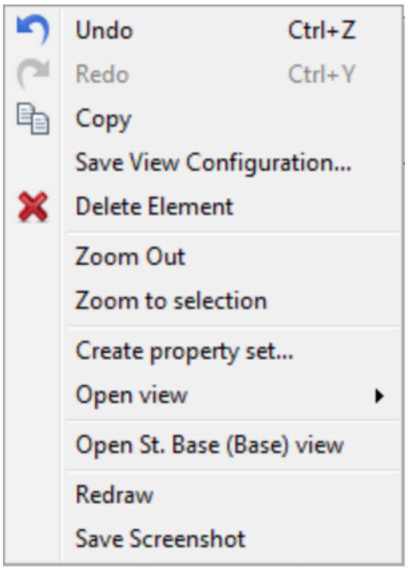
The program is automatically in ‘select mode’ when no other commands are being performed. In this mode you can hover the cursor over an entity and its name will be displayed in the **Select Entity** tooltip.



When the correct entity is displayed, click the entity to select it. If several entities are displayed, you can select one by using the Tab key or Up/Down arrow keys.

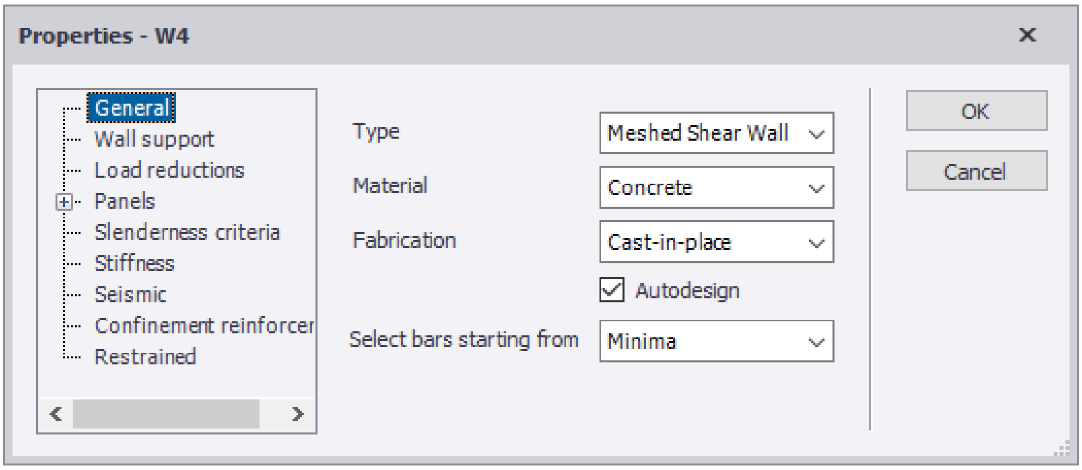
11. Context menu

You can right-click anywhere within a 2D or 3D view to display a menu that is context-sensitive to the item that is currently highlighted.



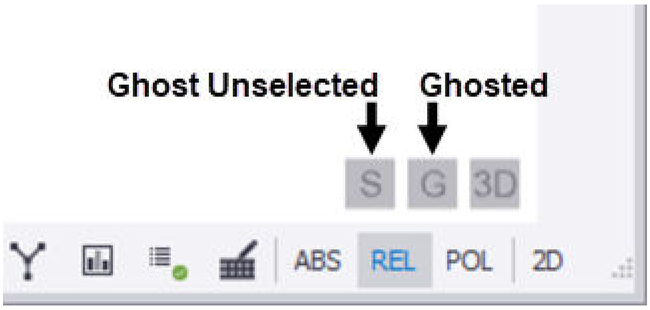
12. Properties dialog

You can use the **Properties** dialog box for viewing and editing parameters associated with an individual model object.



The **Properties** dialog box is displayed by right-clicking an object in the graphical display and selecting the **Edit** option from the context menu that appears.

13. Ghost Unselected and Ghosted toggle buttons



- **Ghost Unselected** button is displayed in all 2D and 3D Views. It is used to toggle the display of selected and unselected objects, making it easier to focus on a particular subset of objects within the model.

  See: Use Ghost Unselected to focus on the selection (page 165)

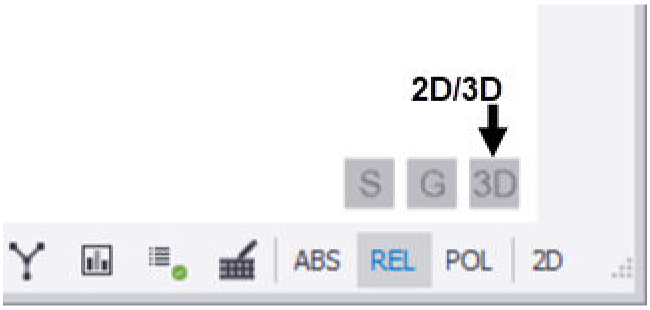
- **Ghosted** button is displayed in Sub Structure and Sub-model Views. It is also displayed in Level, Frame, and Slope Views when they have been toggled into 3D (via the 2D/3D toggle button). It is used to toggle the display of a ghosted view allowing you to see the current view in the context of the whole structure.

  See: Use Ghosted to see the view in the context of the whole model (page 924)

Related video

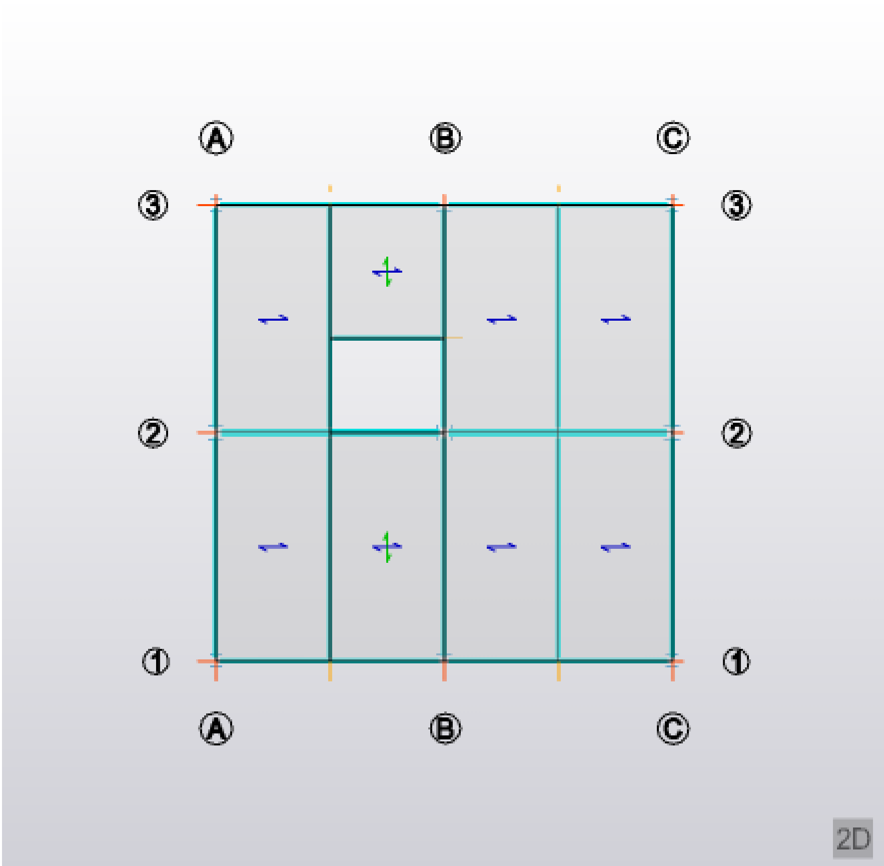
Ghosted Structure view

14. 2D/3D toggle button

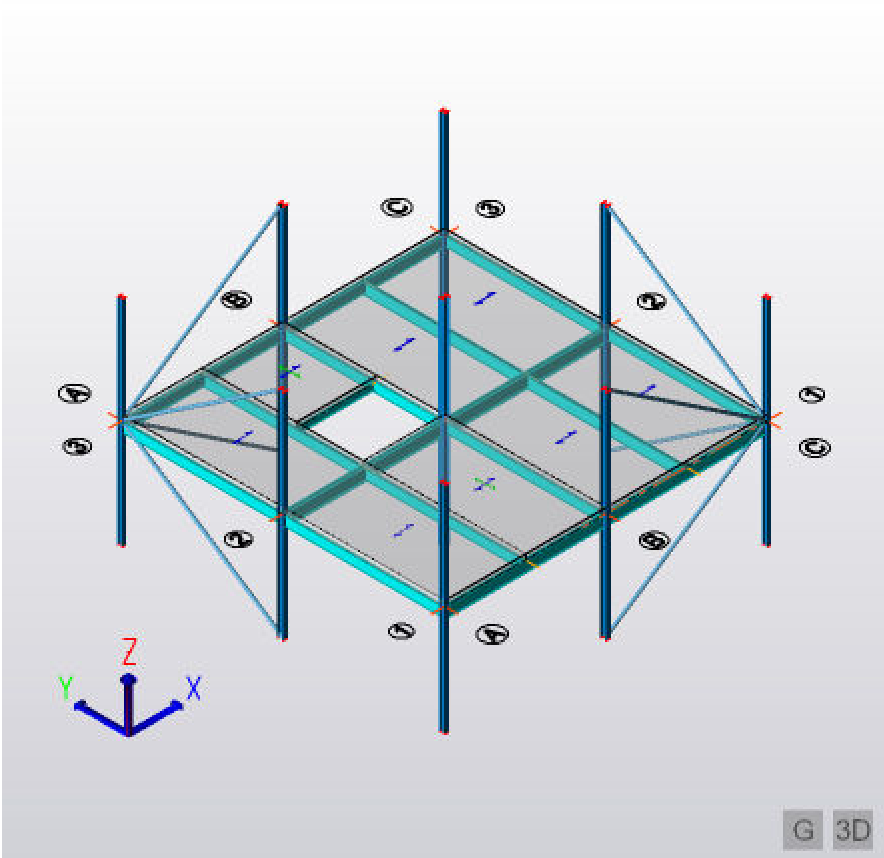


2D/3D button is displayed in 2D Scene Views only. It is used to toggle the view between 2D and a 3D isometric projection.

Example of a level view in 2D:



The same level view when toggled to 3D:



15. Global XYZ axes

The global XYZ axis system within which all other systems exist.

16. Building directions

The principle axes of the building can be rotated about global Z if required.

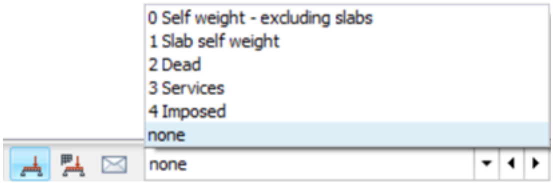
The building direction axes can be switched on/off (and their labelling contriolled) in Structure Properties (page 931).

17. Cutting planes

The **Cutting Planes** symbol is displayed in the bottom right corner of the scene view when any of the cutting planes are currently active.

18. Loading List

The **Loading** list is permanently docked at the bottom edge of the main window.



The **Loading** list has two main functions:

- Selecting a specific load case to add loads into
- Selecting a specific load case, combination, or envelope to view the results for.

When viewing results, first click the **Loadcase**, **Combination**, or **Envelope** button according to your needs. The, choose the specific loadcase, combination, or envelope name in the list.

19. Status bar

The **Status bar** is permanently docked at the bottom edge of the main window. The **Status bar** performs a number of different functions.

The **Status bar** provides feedback by indicating:

1. The analysis and design validity
2. The units system
3. The design code
4. The coordinates of the cursor relative to the global origin in 2D views
5. The view regime buttons that allow you to switch the view regime applied to the active scene view, see **View Regime buttons** below.
6. Coordinate display that allows you to set the display method for coordinate tooltips:
    - **ABS** (Absolute)
    - **REL** (Relative)
    - **POL** (Polar)
7. **2D**/**3D** toggle button
    - **2D**: the content of the 2D view is displayed in plan.

- **3D**: the content of the 2D view is displayed in isometric.

20. View regime buttons

The view regime buttons allow you to switch the view regime applied to the active scene view:

-  **Structural View** shows the geometry and loading of the structure.
-  **Solver View** shows the analysis model.
-  **Results View** shows the analysis results.
-  **Wind View** shows the wind model.
- **Review View** graphically examines the model properties or status.
-  **Slab Deflections View** shows the slab delection analysis results.

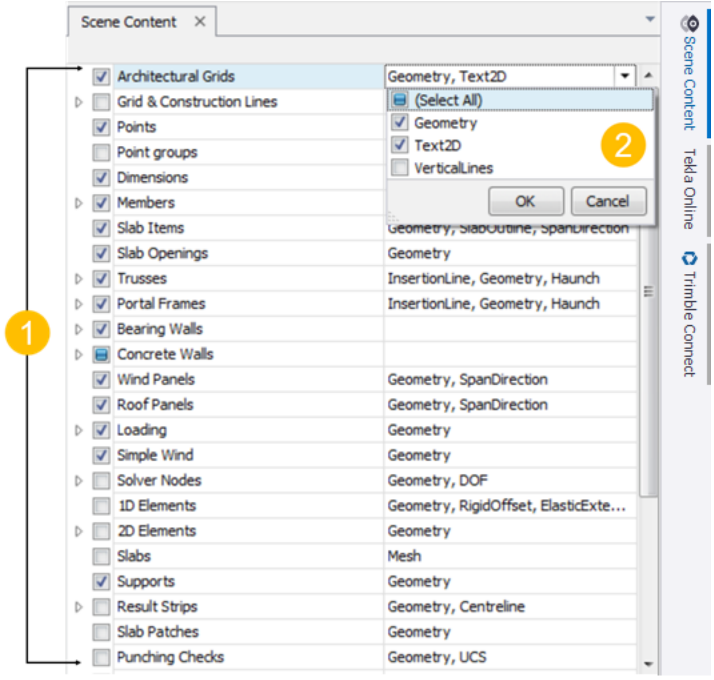
21. ViewCube

In 3D views, you can click a vertex, edge, or face of the **ViewCube** to rotate the model to a preset view.

For more information, see: Zoom, pan, rotate and walk through scene views (page 156)

22. Scene Content

The **Scene Content** sidebar on the right side of the screen is used to control the displayed content in the 2D and 3D Scene Views.



The sidebar contains:

1. Entity categories (with check boxes)

   Some categories have arrow symbols to their side, indicating sub categories. Click the arrow symbol in order to see the sub categories.

   The check box controls whether the entity category and its associated information is displayed. You can check the entities that you want to display and clear the ones that you do not.

2. Entity information controls

   Entity information controls list the information in each category that will be displayed. When clicked, they expand to lists that allow you to select the information you want to display in the model view.

For more information, see: Manage scene content information (page 90)

23. Tekla Online side pane

The Tekla Online side pane is used to access Tekla Online services provided for users of Tekla Structural Designer, for example Tekla User Assistance and Tekla Discussion Forum.

24. Trimble Connect side pane

The Trimble Connect side pane is used to access the the Trimble Connect project collaboration tool.

For more information, see: Working collaboratively with Trimble Connect (page 117)

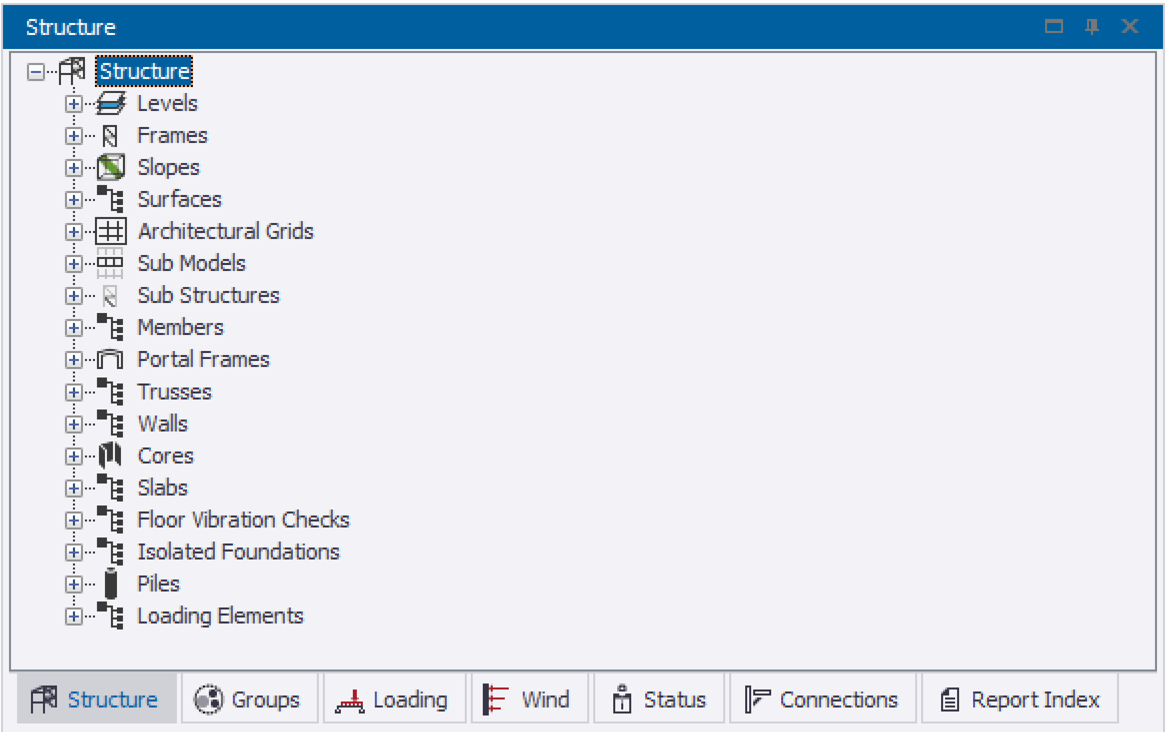
25. Sign in

Signing in to your Trimble Identity gives you access to a greater number of online resourses.

You can sign in to one service and then browse to another online service without a need to log in again. Find the services landing page here: http://www.tekla.com/services

How to use the project workspace

The **Project Workspace** is the central control area for your model.

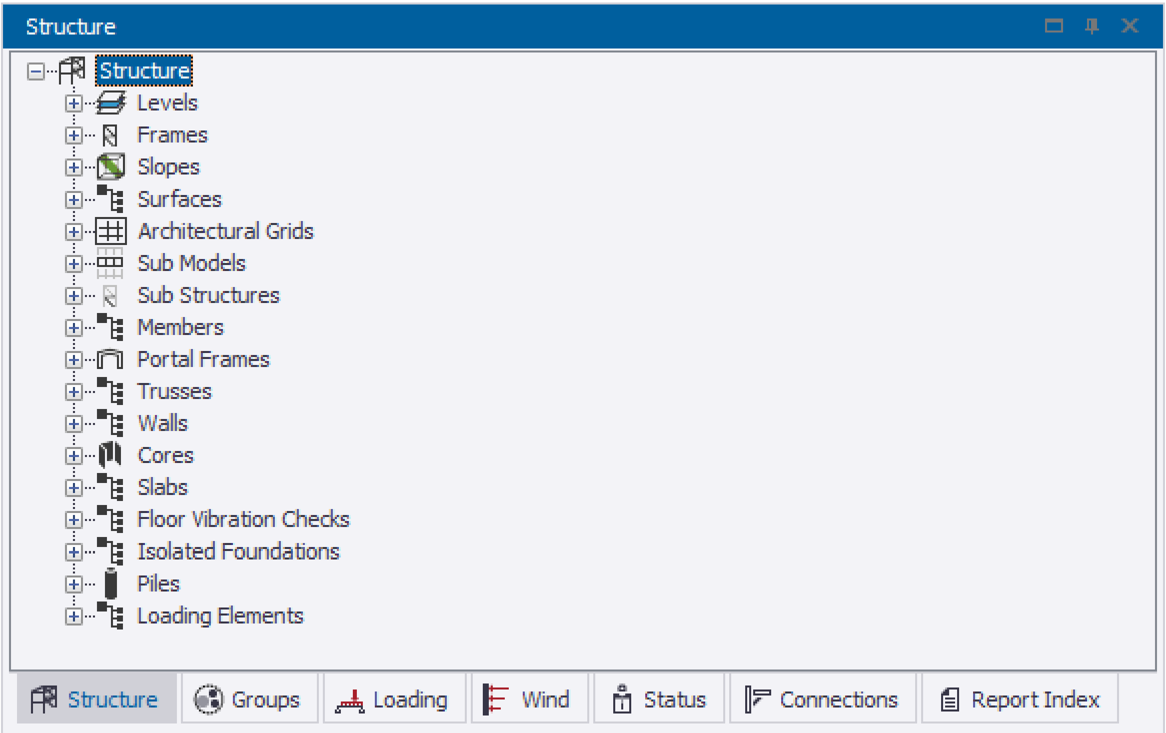


Its tabs are used to access a range of functions:

Icon	Tab	Content
	Structure	Structure (page 71) tree: for opening scene views, viewing and modifying model properties, controlling grids and sub models.
	Groups	Groups (page 77) tree: for organizing members into design and detailing groups and for displaying UDAs.
	Loading	Loading (page 80) tree: for viewing load status.
	Wind Model	Wind Model (page 81) tree: for viewing and modifying wind model properties
	Status	Status (page 82) tree: for highlighting validation issues and other aspects of the model status, i.e. • Model Geometry • Wind Model • Meshing • Decomposition • Solver • BIM • Drift, Sway, Wind Drift, Seismic Drift and Displacements
	Connections	Connections (page 83) tree: for viewing and designing steel connections
	Report Index	Report Index (page 836): contains bookmarks that can be used to move around the report when a report view is displayed.

View and modify model properties in the Project Workspace

The **Structure** tree organizes the model geometry in a hierarchical way. It also allows you to view and modify model properties.



When you open a new model, the **Structure** tree contains two sub-branches: **Levels** and **Sub Models**. As the model geometry develops new branches are added, as shown above.

When a branch or sub-branch is selected, the common properties of the selected branch are viewed in the **Properties** window. You can then modify the common properties in the **Properties** window.

View and modify structure properties

Structure properties, or building parameters, control the principal direction and default meshing properties of the building.

1. In the **Structure** tree, click **Structure**.

   The structure properties are viewed in the **Properties** window.
2. Modify the structure properties according to your needs.

View and modify construction level properties

- In the **Structure** tree, do one of the following:

To	Do this

View and modify the properties common to all construction levels	1. Click **Levels**. The construction level properties common to all levels are viewed in the **Properties** window. 2. Modify the properties according to your needs.
View and modify the parameters of a single construction level	1. Click the **+** sign next to **Levels**. All construction levels in the model are viewed. 2. Click the desired construction level. The level properties are viewed in the **Properties** window. 3. Modify the level parameters according to your needs.

View and modify frame properties

The **Frames** branch is added in the **Structure** tree when you create the first frame in the model.

1. In the **Structure** tree, click the **+** sign next to **Frames**.

   All frames in your model are viewed.
2. Click the desired frame.

   The frame properties are viewed in the **Properties** window.
3. Modify the frame properties according to your needs.

View and modify the slope properties

The **Slopes** branch is added in the **Structure** tree when you create the first frame in the model.

1. In the **Structure** tree, click the **+** sign next to **Slopes**.

   All slopes in your model are viewed.
2. Click the desired slope.

   The slope properties are viewed in the **Properties** window.
3. Modify the slope properties according to your needs.

View and modify grid properties

- In the **Structure** tree, do one of the following:

To	Do this
Modify the color, name, or visibility of a grid	1. Click the **+** sign next to **Architectural Grids**. All architectural grids in your model are viewed. 2. Click the desired grid. The grid properties are viewed in the **Properties** window. 3. Modify the color, name and visibility of the grid according to your needs.
Renumber all grids	1. Right-click **Architectural Grids**. 2. In the context menu, click **Renumber**.

View and modify sub model properties

1. In the **Structure** tree, click the **+** sign next to **Sub Models**.

   The sub models in your model are viewed.
2. Click the desired sub model.

   The sub model properties are viewed in the **Properties** window.
3. Modify the properties according to your needs.

TIP To open the **Sub Models** dialog dialog, double-click **Sub Models**.

View and modify sub structure properties

See: Manage sub structures (page 921)

View and modify member properties

Tekla Structural Designer classifies members by material and type, and further by fabrication and by shape. Members are classified by material and type, then further classified by fabrication and then by shape.

- In the **Structure** tree, do one of the following:

To	Do this
View and modify common properties for members of a particular type and fabrication	1. Click the **+** sign next to **Members**. The existing member types are viewed.

	2. Click the **+** sign next to a member type. The existing fabrication types are viewed. 3. Click a fabrication type. Common properties of all members of the fabrication type are viewed in the **Properties** window. 4. Modify the properties according to your needs.
View and modify common properties for members of a particular type, fabrication and shape	1. Expand the **Members** branch and its sub branches by clicking them. 2. Click the **+** sign next to the desired fabrication type. The shapes of the fabrication type are viewed. 3. Click a shape. The common properties of all members of the selected shape are viewed in the **Properties** window. 4. Modify the properties according to your needs.
View and modify the properties of an individual member	1. Expand the **Members** branch and its sub branches by clicking them. 2. Click the **+** sign next to the shape type. The member references of the shape are viewed. 3. Click a member reference. The properties of the member are viewed in the **Properties** window. 4. Modify the properties according to your needs.
View a member in a new window, select it in visible views, delete it, or modify it in the **Properties** dialog box	1. Expand the **Members** branch and its sub branches by clicking them. 2. Right-click a member reference. 3. In the context menu, select the desired option.

View and modify slab properties

- In the **Structure** tree, do one of the following:

To	Do this
View and modify properties common to all slabs	1. Click **Slabs**. The common properties to all slabs are viewed in the **Properties** window.

	2. Modify the properties according to your needs.
View and modify the properties of a parent slab	1. Click the **+** sign next to **Slabs**. The existing parent slabs are viewed. 2. Click the desired parent slab. The properties of the parent slab are viewed in the **Properties** window. 3. Modify the properties according to your needs.
Modify the properties of a parent slab, or delete it	1. Click the **+** sign next to **Slabs**. The existing parent slabs are viewed. 2. Right-click a parent slab. 3. In the context menu, select if you want to modify the parent slab in the, or delete it.
View and modify the properties of a slab item	1. Click the **+** sign next to **Slabs**. The existing parent slabs are viewed. 2. Click the **+** sign next to a parent slab. All the slab items within the parent slab are viewed. 3. Click a slab item. The properties of the slab item are viewed in the **Properties** window. 4. Modify the properties according to your needs.

View and modify wall or roof panel properties

1. In the **Structure** tree, click the **+** sign next to **Walls** or **Roofs**.

   The currently defined walls and roof panels are viewed.
2. Click the desired wall or roof panel.

   The properties of the selected wall or roof panel are viewed in the **Properties** window.
3. Modify the properties according to your needs.

View and modify result strip properties

1. In the **Structure** tree, click the **+** sign next to **Result Strips**.

   All the currently exiting result strips are viewed.

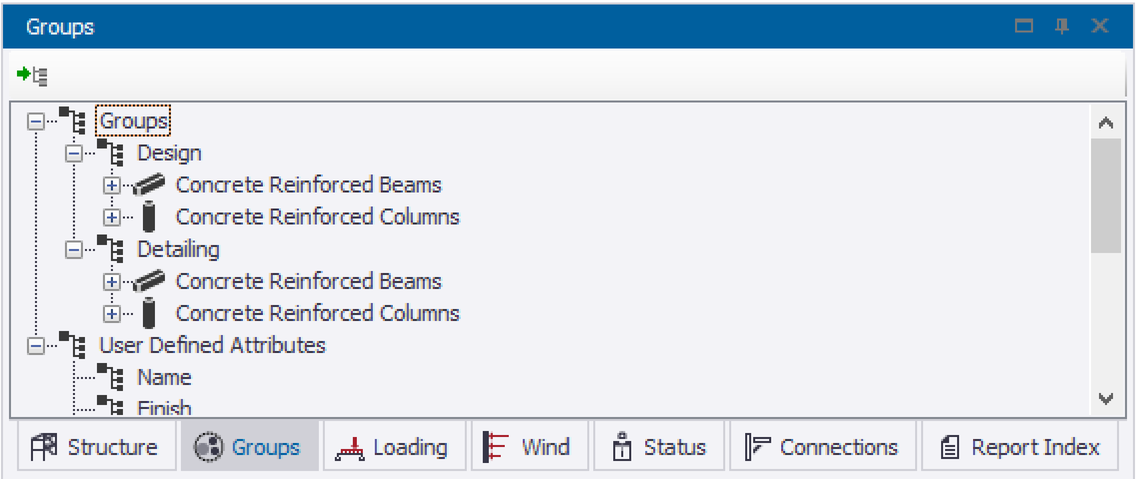
2. Click the desired result strip.

   The properties of the selected result strip are viewed in the **Properties** window.

3. Modify the properties according to your needs.

Manage groups in the Project Workspace

The **Groups** tree organizes the members in your model into design groups. It also allows you to manage design groups according to your needs.



If you have defined concrete members, they are also organized into further groups for detailing purposes. The application of grouping is most useful when handling concrete structures.

User defined attributes (UDAs) are also listed, allowing you to see which members have been assigned specific UDAs.

NOTE You can right-click on a group name (or UDA) in order to select all members in the group, (or with that UDA), simultaneously in the visible views.

Regroup a specific member type

1. In the **Groups** tree, right-click the member type you want to re-group.
2. In the context menu, select **Regroup Members**.

   The selected member type is regrouped.

Re-group all member types

WARNING Re-grouping all member types undoes any manual grouping you may have done.

- Click the Re-group ALL Model Members button on the top left corner of the **Groups** tree.

Add group

1. In the **Groups** tree, select the appropriate branch within which you want to manually create a new group.
2. In the context menu, select **Add Group**.

   A new group empty group is created.

Set as default group

When a new empty group is set as default, if you create a new member it will be placed in that group in preference to another empty group.

1. In the **Groups** tree, select an empty group.
2. In the context menu, select **Set As Default Group**.

   An asterix is placed next to the group name to indicate that it is the default group for new members.

Split a member group into smaller groups

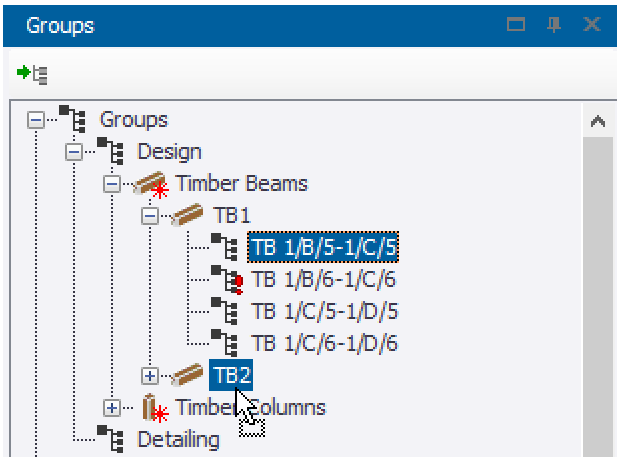
1. In the **Groups** tree, right-click the name of the group that you want to split.
2. In the context menu, select **Split Group**.

   All members of the group are ungrouped and placed into individual groups.

Move members from one group to another

1. In the **Groups** tree, open the group that contains the member(s) to be moved.
2. Click the first member name to select it, then if you want to add further members to the selection, hold down **Ctrl** while clicking on their names.
3. Drag the selection over the group name where you want to move it.

Provided that the selection meets the geometric criteria to belong to the group, a small rectangle will be displayed alongside the cursor.



NOTE You must drag over the **group name** as shown above, rather than over the group content for the rectangle to appear.

4. Release the mouse button.

TIP You can also change the group to which members belong by selecting them and then updating the group name in the **Properties** window.

Rename a member group

1. In the **Groups** tree, right-click the group that you want to rename.
2. In the context menu, select **Rename Group**.
3. Type the new name of the group and press Enter.

Modify the group name defaults for future projects

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**, and select the settings set that you want to edit.
3. Go to the **Grouping** page.
4. Review and modify the default group names according to your needs.
5. To save the changes, click **OK**.

Check a member group

1. In the **Groups** tree, right-click the member type you want to check.

2. In the context menu, select **Check Group**.

   The selected member group is checked.

Check a member group using Tekla Tedds

1. In the **Groups** tree, right-click the member type you want to check.
2. In the context menu, select **Check Group**.

   The selected member group is checked using Tekla Tedds.

Design a member group

1. In the **Groups** tree, right-click the member type you want to check.
2. In the context menu, select **Design Group**.

   The selected member group is designed.

Remove a member group

1. In the **Groups** tree, right-click the name of the group you want to remove.
2. In the context menu, select **Remove Group**.

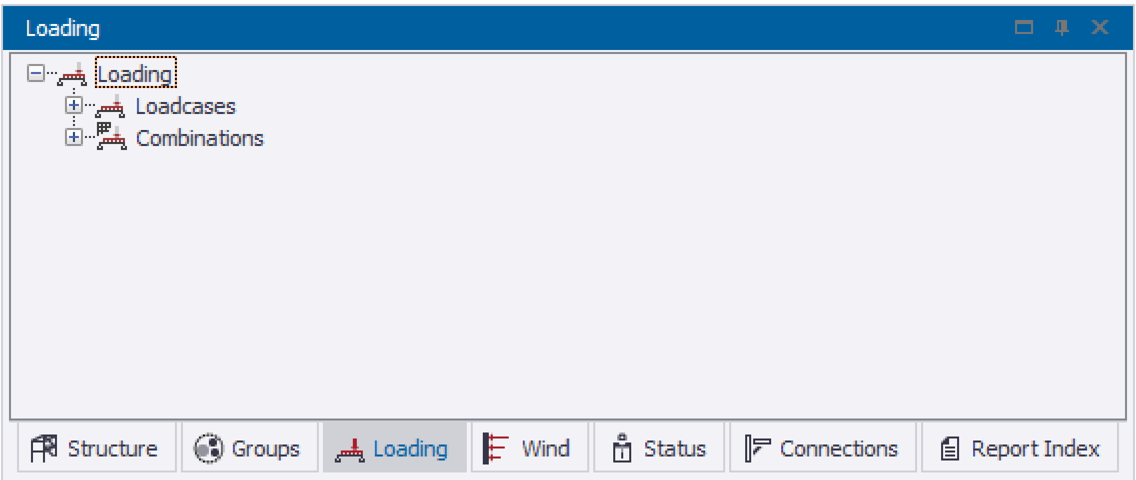
   All members are ungrouped and placed into individual groups.

See also

Create and manage user-defined attributes (page 916)

View load status in the Project Workspace

The **Loading** tab lists the individual loads that have been applied in each loadcase. After analysis it sums the applied loads in each combination and checks these against the sum of reactions.



See the status options in the following table:

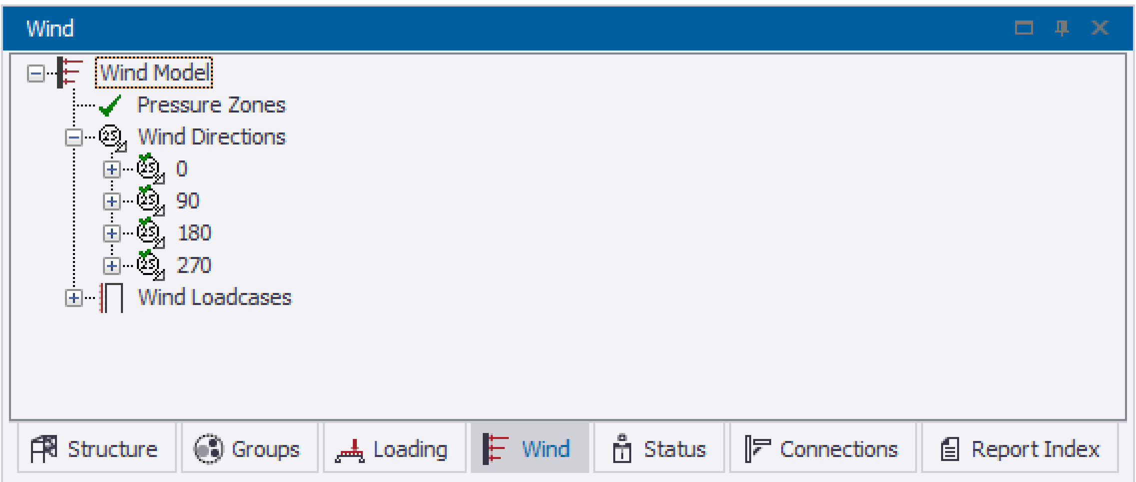
Icon	Meaning
✓	**Total Reaction** is in equilibrium with the **Total Load on Structure**.
✗	**Total Reaction** is not in equilibrium with the **Total Load on Structure**.
?	**Total Reaction** is unavailable.

1. In the **Project Workspace**, go to the **Loading** tab.
2. Click the desired loadcase.

   The properties of the loadcase are viewed in the **Properties** window.

View and modify wind properties in the Project Workspace

The **Wind Model** tree is used after completing the wind modeling process to display wind direction views, modify wind directions, and modify wind loadcases.



View a wind direction view

1. In the **Wind Model** tree, click **Wind Directions**.
2. Right-click the desired wind direction.
3. In the context menu, select **Open View**.

Modify wind direction properties

1. In the **Wind Model** tree, click **Wind Directions**.

2. Click the desired wind direction.

   The properties of the wind direction are viewed in the **Properties** window.

3. Modify the properties according to your needs.

View and modify wind loadcases

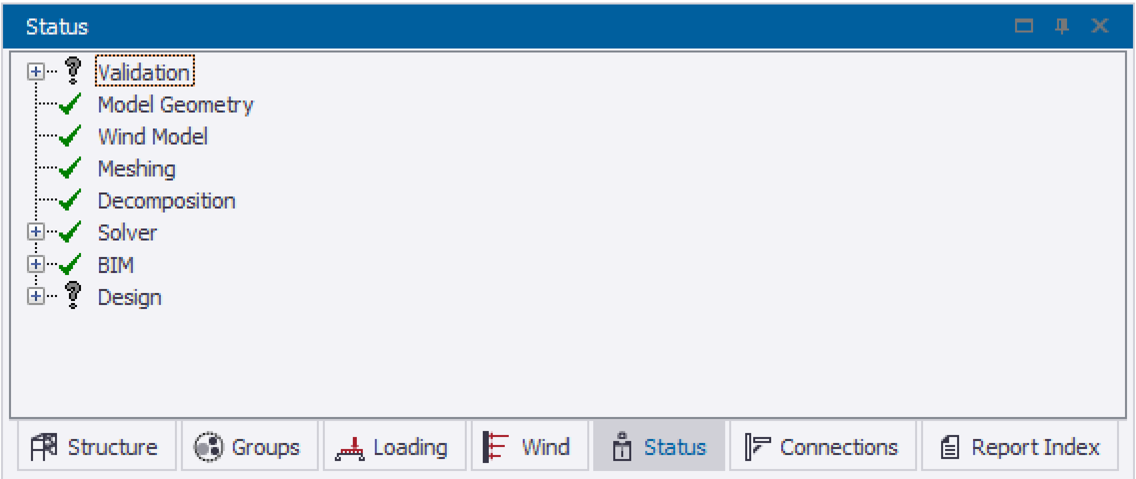
1. In the **Wind Model** tree, click **Wind Loadcases**.
2. Click the desired wind loadcase.

   The properties of the wind direction are viewed in the **Properties** window.

3. Modify the properties according to your needs.

View model status in the Project Workspace

The **Status** tree is used to review the validation messages and other model status indicators.



You can review:

- The validation status for the model

  For more information, see: Validate the model (page 338)

- The validation status for the analysis

  The analysis model validation is performed automatically when you analyze the model.

- The decomposition status

  The **Decomposition** branch shows if load decomposition has been successfully completed.

- The solver status

  The **Solver** branch lists the analyses that have been performed.

- The BIM validation

  If the model has been imported or exported, the **BIM** branch lists any warnings and errors that relate to the import or export process.

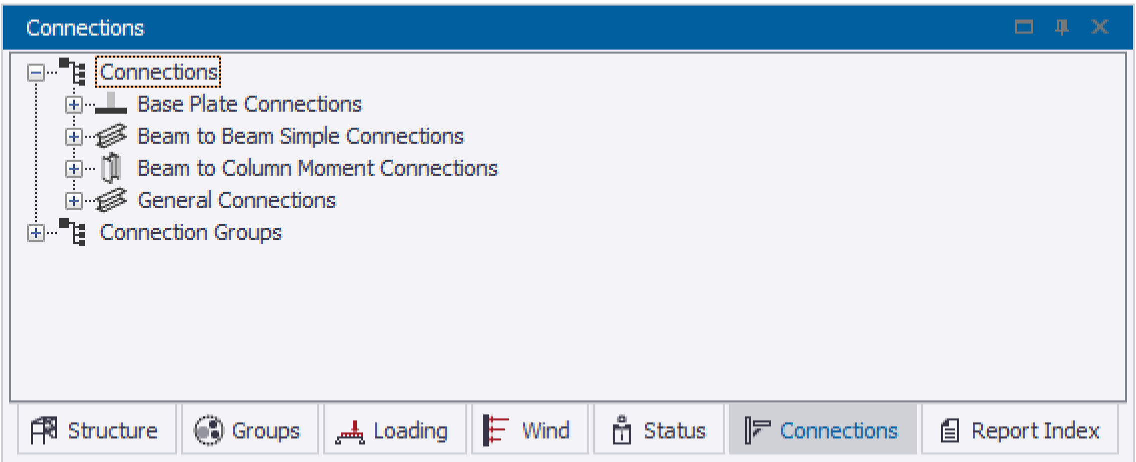
- The status of drift, sway, wind drift, seismic drift checks and displacements

  Each of these are reported automatically when the appropriate analysis has been performed. For more information, see: Drift, sway, seismic drift, wind drift, and overall displacements (page 662)

NOTE
- ✓ indicates that the analysis or process has been successfully completed.
- ✗ indicates that the analysis or process has resulted in an error.
- indicates that the analysis or process has resulted in a warning.

Manage and design connections in the Project Workspace

The **Connections** tree organizes the model's steel connections in a hierarchical way, by Type (i.e. Type/Reference), and by Group (i.e. Type/Section size/Reference).



When opened for the first time, or if the model has been edited in a way that affects the connection configurations within it, you must run **Update Connections** in order to determine the valid connections.

This applies a set of rules (page 658) to create and organize connections into the following Types:

- Base plate connections
- Beam to Beam Moment Connections

- Beam to Beam Simple Connections
- Beam to Column Moment Connections
- General Connections

Valid connections are displayed in the scene views and are also listed in the **Connections** tree

Update connections

To create connection objects for the first time, or to recreate connections after changes to the model:

1. Click on the **Connections** tree in the **Project Workspace**
2. Right-click the Connections branch.
3. Choose **Update Connections** from the context menu.

   This applies a set of rules (page 658) to determine all valid steel connections in the model. The resulting connections are listed in the Connections Tree and are also shown by bounding boxes in the Scene Views.

   NOTE If the model has been changed so that new connections exist, these are not created automatically; you are required to update connections as required.

Edit connection

- In the **Connections** tree, right click on the appropriate connection reference and choose **Edit...** from the context menu. A dialog opens allowing the connection to be edited.

Select connection in visible views

- In the **Connections** tree, right click on the appropriate connection reference and choose **Select in visible views** from the context menu. The connection is selected if it exists in the active scene view.

Design connection

- In the **Connections** tree, right click on the appropriate connection reference and choose **Design connection** from the context menu. The connection is designed using Tekla Connection Designer.

Export connection to Tekla Connection Designer

- In the **Connections** tree, right click on the appropriate connection reference and choose **Design connection** from the context menu. The connection is designed using Tekla Connection Designer.

See also

Create and check steel connections (page 635)

How to manage scene views, view regimes and scene content

Scene views

A scene view displays the model or a part of it in a tabbed window. It might display: the entire structure, a sub structure, or an individual member in 3D; a specific level, frame, or plane in 2D. A separate load analysis view is also available for displaying force and moment diagrams for individual members.

Click the following links to learn how to manage and manipulate scene views:

- Open, close and save scene views (page 86)
- Create and modify scene view tab groups (page 89)

View regimes

For each scene view, you should select a view regime from the **Status bar** appropriate to the task being performed:

Icon	View regime	Used for
	Structural View	Creating the structure geometry and applying loads
	Solver View	Displaying the analysis model
	Results View	Viewing analysis results
	Wind View	Viewing the wind model
	Review View	Examining the design status or reviewing specific model properties
	Slab Deflections View	Viewing slab deflection analysis results

Click the following link to learn how to switch between view regimes:

- Change the view regime (page 89)

Scene content

You control the level of information displayed in each scene view by switching items on/off in the **Scene Content** dialog.

Click the following links to find out more about scene content:

- Manage scene content information (page 90)
- Scene Content entity categories (page 91)

Open, close and save scene views

You can display multiple different scene views simultaneously as tabbed windows within the main window. See the following instructions to open, close and save scene views.

Open 3D views

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, do one of the following:

To	Do this
Open a 3D view of the entire structure	• Double-click **Structure**.
Open a 3D solver view of the entire structure	a. Right-click **Structure**. b. In the context menu, select **Open solver view**.
Open a 3D view of a sub model	a. Click the **+** sign next to **Sub Models**. The existing sub models are viewed. b. Double-click a sub model.
Open a 3D solver view of a sub model	a. Click the **+** sign next to **Sub Models**. The existing sub models are viewed. b. Right-click a sub model. c. In the context menu, select **Open solver view**.
Open a 3D view of a sub structure	a. Click the **+** sign next to **Sub Structures**. The existing sub structures are viewed. b. Double-click a sub structure.
Open a 3D solver view of a sub structure	a. Click the **+** sign next to **Sub Structures**. The existing sub structures are viewed. b. Right-click a sub structures. c. In the context menu, select **Open solver view**.
Open a 3D view of a single member from within another view	a. Hover the mouse over the desired member to highlight it. b. Right-click the member. c. In the context menu, select **Open [element name] view**.

Open a 3D view of a single member in the **Project Workspace**	a. Click the **+** sign next to **Members**. b. Expand the required sub branches similarly. c. Right-click the required member. d. In the context menu, select **Open view**.

Open 2D views

1. In the **Project Workspace**, go to the **Structure** tab.
2. In the **Structure** tree, do one of the following:

To	Do this
Open a 2D view of a construction level	a. Click the **+** sign next to **Levels**. The existing construction levels are viewed. b. Double-click the desired level.
Open a 2D solver view of a construction level	a. Click the **+** sign next to **Levels**. The existing construction levels are viewed. b. Right-click the desired level. c. In the context menu, select **Open solver view**.
Open a 2D view of a frame	a. Click the **+** sign next to **Frames**. The existing frames are viewed. b. Double-click the desired frame.
Open a 2D solver view of a frame	a. Click the **+** sign next to **Frames**. The existing frames are viewed. b. Right-click the desired frame. c. In the context menu, select **Open solver view**.
Open a 2D view of a sloped plane	**NOTE** Before you can view a 2D view of a sloped plane, you must create a sloped plane in your model. a. Click the **+** sign next to **Slopes**. The existing slopes are viewed. b. Double-click the desired slope.

Open a 2D solver view of a sloped plane	**NOTE** Before you can view a 2D view of a sloped plane, you must create a sloped plane in your model. a. Click the **+** sign next to **Slopes** The existing frames are viewed. b. Right-click the desired slope. c. In the context menu, select **Open solver view**.

Save, open and delete view configurations

Once a scene view has been adjusted to show a specific area of the model, you can save the scene view to a view configuration.

NOTE You can use view configurations in two ways:

- Include them as views in model reports.

  In this case, the view configurations retain the scene content settings that were in place when you saved the view configuration.

- Re-open them in a new scene view at a subsequent time.

  In this case, the view configurations adopt the scene content settings that are currently in place in the scene view.

- According to your needs, do one of the following:

To	Do this
Save a view configuration	1. Right-click anywhere in the view. 2. In the context menu, select **Save View Configuration...** The **View name** dialog box opens. 3. Name the view. 4. Click **OK**.
Open a saved view configuration	1. On the **Home** tab, click **Manage View Configurations**. 2. Select the desired view configuration. 3. Click **Open View**. 4. Click **OK**.
Delete a view configuration	1. On the **Home** tab, click **Manage View Configurations**. 2. Select the desired view configuration. 3. Click **Delete**. 4. Click **OK**.

Close views

- Click **x** on the top right corner of the view tab.

Create and modify scene view tab groups

When you have created multiple scene views, only the active view is visible by default. However, it is often useful to display views side by side or one below another. You can do this by creating tab groups.

Create a new tab group from an existing view tab

1. Right-click an existing view tab.
2. In the context menu, select **New Horizontal Tab Group** or **New Vertical Tab Group** according to your needs.

Create a new tab group using the docking control

1. Start dragging a view tab.

   A docking control appears in the middle of the view.

2. Do one of the following:

To	Do this
Create a new vertical tab group	• Drag the view over the left or right button of the docking control and release the mouse.
Create a new horizontal tab group	• Drag the view over the top or bottom button of the docking control and release the mouse.

Move a view from one tab group to another

1. Right-click the view tab.
2. In the context menu, select **Move to Next Tab Group**.

Change the view regime

- In the **Status bar** at the bottom of the main window, click the desired view regime button.
  -  **Structural View** to show the geometry and loading of the structure.

- **Solver View** to show the analysis model.
- **Results View** to show the analysis results.
- **Wind View** to show the wind model.
- **Review View** to graphically examine the model properties or status.
- **Slab Deflections View** to show the slab delection analysis results.

Manage scene content information

Different entity types have different levels of information associated with them. You can select how much of this information is displayed in each of the different scene views and view regimes. Scene content selections are saved independently with each scene view.

For example, in a **Solver View** regime, it is generally sufficient to represent beams by their insertion lines. However, in a **Structural View** regime, you are likely to also include their geometric outlines. In either of the views, you may also choose to display their direction arrows and possibly their reference texts also.

View the Scene Content window

1. Do one of the following:

If	Do this
Scene Content is set to **Auto Hide**	The **Scene Content** tab is docked on the edge of the main window. • Click the tab to expand the **Scene Content** window.
The **Scene Content** has been closed	• On the **Windows** tab, click **Scene Content**.

Select items in the Scene Content window

- In the **Scene Content** window, do one of the following:

To	Do this
View the sub categories of an entity category	• Click the arrow sign on the left side of the category name.
View a category in the model view	• Select the check box on the left side of the category name.

Hide a category in the model view	**NOTE** If you hide an entity category, you can no longer perform some commands that affect that entity category. For example, if you hide slab items, you cannot define slab or area loads on a floor because there are no slab panels to select. • Clear the check box on the left side of the category name.
Adjust the information viewed in the model view	1. Click the cell on the right side of the category name. A list of possible details opens. 2. Select the details you want to view. 3. Click **OK**.

Reinstate the default Scene Content selections

You can discard your current selections and reinstate the default **Scene Content** selections at any time.

See the following instructions.

1. Close the current view.
2. Re-open the view using the **Structure** tree.

See also

Open, close and save scene views (page 86)

Scene content entity categories

The following paragraphs describe some of the most important entity categories of **Scene Content**.

Architectural Grids

Allows you to show or hide architectural grids in 3D views. When selected, architectural grids are displayed at the lowest level of the model.

Grid & Construction Lines

Allows you to show or hide grid and construction lines at individual levels in both 2D and 3D views.

You can control the levels at which grids are or are not displayed in the **Properties** window for each level. Grids are only displayed in 3D views if the **Show grids in the 3D view** option is selected and in 2D views if the **Show grids in plane view** option is selected.

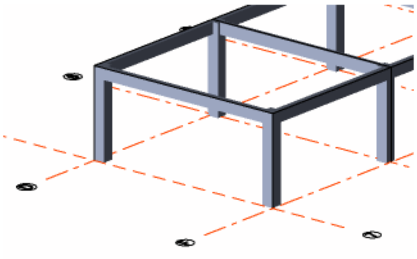
Members

Allow you to manage how each member type is displayed. For each of the mentioned elements, you can display the following information:

Geometry

Displays the faces of the member by shading them according to the member type.

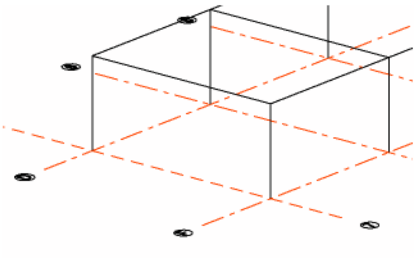
The following image displays a concrete beam and column example with the **Geometry** and **Architectural Grids** options selected.



InsertionLine

Displays a solid line between the start and end node of the member.

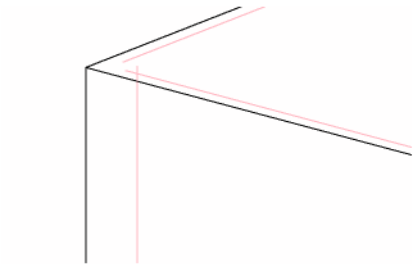
The following image displays the concrete beam and column example with the **InsertionLine** and **Architectural Grids** options selected.



LoadingLine

Displays a solid line through the center of the member. Any member loads are applied along the line.

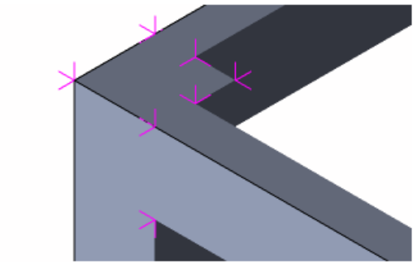
The following image displays the concrete beam and column example zoomed with the **InsertionLine** and **LoadingLine** options selected.



Normals

Displays the normal directions at each corner of the member.

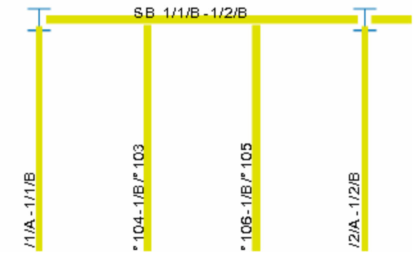
The following image displays the concrete beam and column example zoomed with the **Geometry** and **Normals** options selected.



Text

Displays the member name.

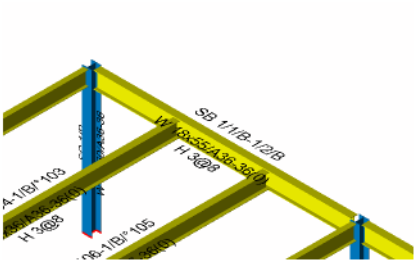
The following image displays a composite steel beam example with the **Geometry** and **Text** options selected.



Text2D

Displays the member name, section and class in the 2D plane of the member.

The following image displays the composite beam example with the **Geometry** and **Text2D** options selected.

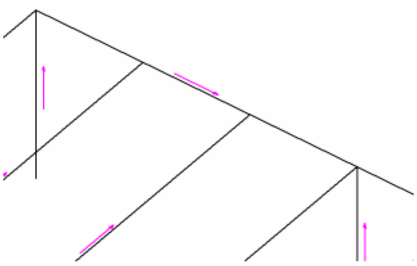


Direction

Displays the direction from end 1 to end 2 of the member.

If the direction is incorrect, go to the **Edit** tab and click **Reverse**.

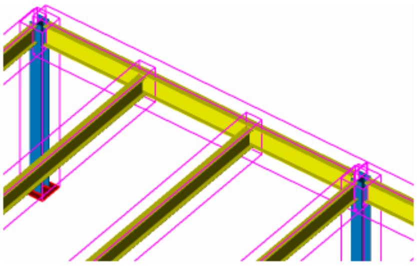
The following image shows the composite steel beam example with the **InsertionLine** and **Direction** options selected.



BoundingBox

Displays the bounding box of sections. Selecting the option may simplify right-clicking sections of a small cross section.

The following image displays the composite steel beam example with **Geometry** and **BoundingBox** selected.



Decking

Displays the strip of decking that is connected to the member.

The following image displays the composite steel beam example with **Geometry** and **Decking** selected.

Plan

NOTE The **Plan** entity category is only available in 2D views.

The initial display for 2D views is configured for modeling purposes, so by default, it does not display all the information that will be output when drawings are created. However, you can select the **Plan** entity category to overlay one of the following planar drawing categories on the 2D view:

- General arrangement
- Beam end forces
- Foundation layout
- Loading plan
- Slab/mat layout

You can also select the individual drawing layers to be overlaid. The **Plan** category can be very useful, as it allows you to display information that would otherwise not be available whilst modeling.

To use the **Plan** category, do the following:

1. Open a 2D view.
2. In **Scene Content**, select **Plan**.
3. In the list on the right side of the **Plan** category, you can select the drawing category.
4. Click the arrow on the left side of the **Plan** category to view the different drawing layers that you can optionally display.

   Only the layers that exist in the previously mentioned planar drawing categories can be displayed. The most important layers are:

Name	Description
General	Allows you to display grids, construction lines, and dimensions as they would appear in drawings.
Members	Allows you to display the various different member types labeled as they would appear in drawings.
Walls	Allows you to display concrete walls labeled as they would appear in drawings.
Slabs/Mats	Allows you to display concrete slabs labeled as they would appear in drawings.
Other	Allows you to display various other items that can be output to drawings. **NOTE** It is not yet possible to overlay beam end forces or foundation reactions on a scene view. However, you can display them in general arrangement drawings.
Tables	Allows you to display tables of information that can be output to drawings.
Connections	Allows you to display connection names, attributes, and reactions as they would appear in drawings.
Beam End Forces	Allows you to display beam end forces as they would appear in drawings.

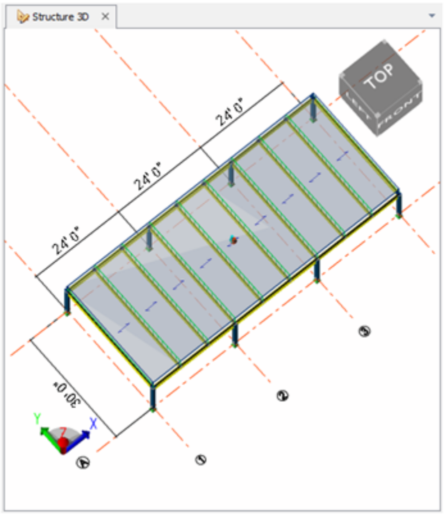
   By default, the layers that are initially selected match those in the first layer configuration for the drawing category in **Model Settings** --> **Drawings** --> **Layer Configurations**.

5. Select or clear the layers according to your needs.

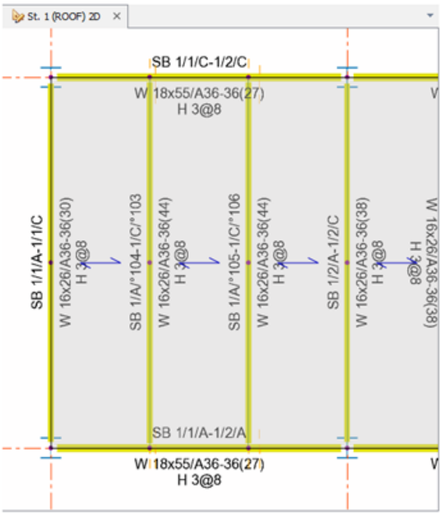
   TIP Layer content can be further controlled in **Model Settings** --> **Drawings** --> **Layer Configurations**.

Example

To illustrate how the **Plan** category would typically be used, consider the following composite beam example designed to the AISC 360 ASD head code:



In the following floor view, the standard beam labeling for modeling is applied. The labeling consists of the beam name, section, grade, number of connectors, and transverse reinforcement.

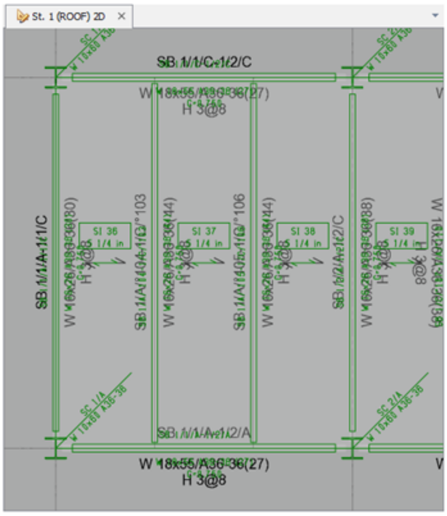


When you produce drawings, additional design information would probably also be conveyed. The information can contain, for example, the amount beam camber required. The camber is included in one of the drawing layers, so you can include it in the 2D scene view, as long as you know the layer it belongs to.

To include the camber, do the following:

1. Open a 2D scene view containing the beams.
2. In **Scene Content**, select **Plan**.

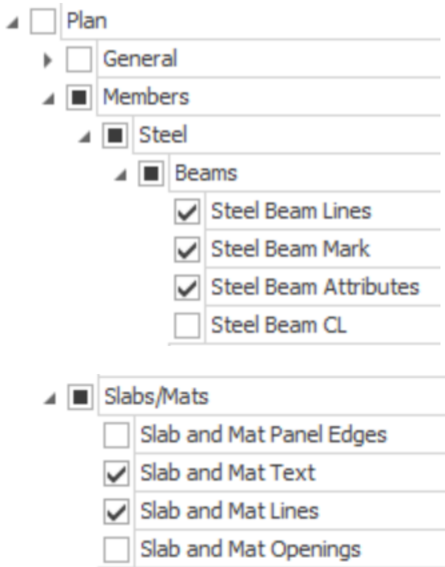
Depending on the current scene content selection, there may be a substantial duplication of axes, members, labeling, and so on.



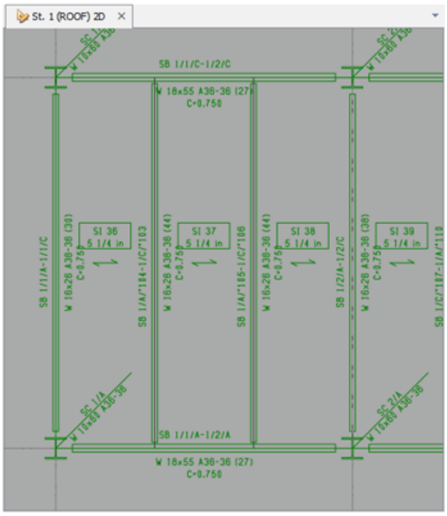
3. Expand the **Plan** category and do the following:

   a. Clear the **Members** and **Slabs/Mats** options.

   b. Select options according to the following images:



The scene view should now appear less cluttered, as shown in the following image.



4. On the **Draw** tab, click **Settings** and adjust the drawing settings as follows:

   a. Go to **Options** --> **Planar Drawings** --> **General Arrangement**.

   b. Go to the **Beams** tab.

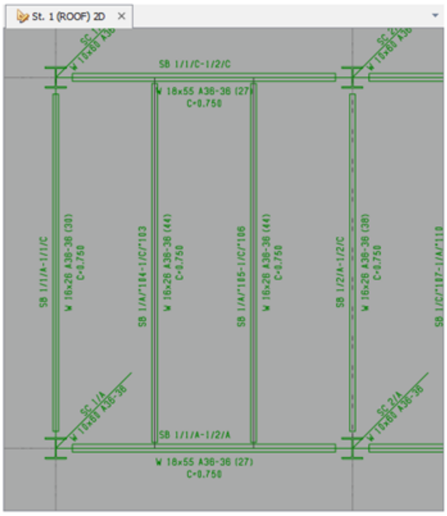
   c. Ensure that the **Camber** option is selected.

   d. Go to the **Slabs/Mats** tab.

   e. Clear all **Panel Labelling** options and the **Include panel span direction symbol** option.

   f. Click **OK**.

   The drawing is updated to match the new drawing options.



How to hide, re-display and move windows

The**Properties** window, the , **Scene Content**, and each of the **Project Workspace** tabs are displayed in windows that can be resized and repositioned, or docked to an edge of another window.

Auto-hide a window

To increase the area available for graphical display, you can choose to auto-hide any of the windows.

- In the desired window, click **Auto Hide**.

  The window immediately contracts to a tab.

- Click the window tab to expand it.

  You can only expand a window if it is docked against an edge of the main window.

Close a window

- Click **Close** at the top right corner of the window.

Re-display a closed window

1. Click the **Windows** tab.
2. In the **View** group, click the window name.

   If the window is already displayed, its control is highlighted in the **View** group.

Move a window

- Select the title bar of the window, and drag the window to its new location.

  If you place the selected window over an edge of the main window, or over a divider within the main window, the window docks to that edge or divider.

Dock a window as a tabbed page in another window

- Select the title bar of the window, and drag the window over the title bar or the tab group of the other window.

Open a tabbed page in another window

- Select the tabbed page, and drag the page to its new location.

  The tabbed page opens in a new window.

Dock a window using the docking control

1. Select the title bar of the window, and start dragging the window.

   The window docking control appears in the middle of the main window.

2. Drag the window over the docking control button that indicates the desired location.

Dock a window to a tabbed page of the current window

1. Select the title bar of the window, and drag the window over the central button of the docking control.
2. Release the left mouse button.

Keyboard shortcuts

Keyboard shortcuts in Tekla Structural Designer are described in the tables below.

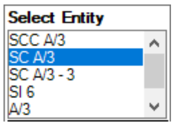
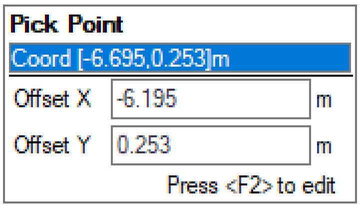
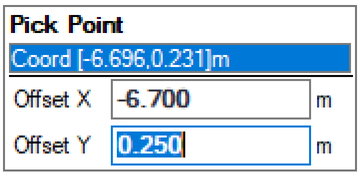
General keyboard shortcuts

Keys that perform general functions in Tekla Structural Designer are listed below.

Key	Function
F1	Displays the Tekla Structural Designer Help.
Ctrl	Holding down Ctrl while selecting entities adds the new entities to the current selection.
Esc	Cancels the current command.
Ctrl F	Opens the Find dialog
Ctrl O	Open a file
Ctrl N	Starts a new file

Keyboard shortcuts in 2D and 3D Views

The following keys function as described below in the Structural View and other graphical views.

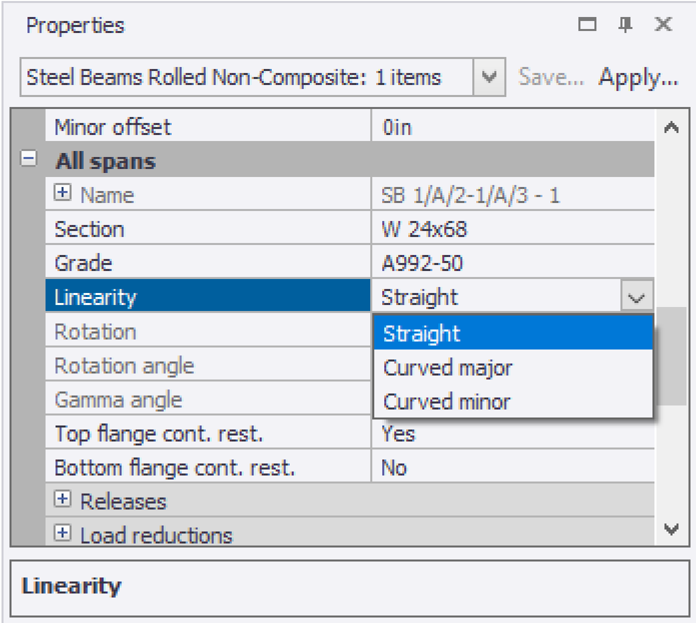
Key	Function
ZA	Zooms to the extents of the model.
ZS	Zooms to the selected entities. **NOTE** Prior to using this shortcut, at least one entity must be selected in order for the command to zoom to it.
Z1	Zooms to 1m (3ft) area around the cursor.
Z2	Zooms to 2m (6ft) area around the cursor.
Z3	Zooms to 3m (9ft) area around the cursor.
Up/Down arrow keys	Allows you to scroll between entities in the **Select Entity** tooltip when Tekla Structural Designer has detected multiple entities. 
Ctrl	Holding down Ctrl while selecting entities adds the new entities to the current selection.
F2	Whenever you are prompted to *pick a point*, the **Data Entry** tooltip displays the co-ordinates at the cursor position.  Pressing **F2** enables keyboard input of the exact co-ordinates required. 

Key	Function
F8	Switches between displaying and hiding the **ViewCube** in 3D views. 

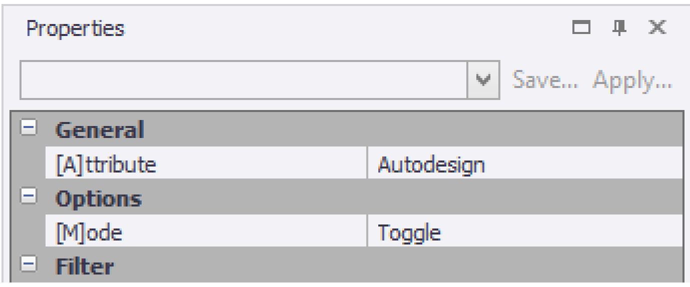
Keyboard shortcuts in Properties windows

Keys that perform specific functions in any **Properties** window are described below.

Key	Function
Up/Down arrow keys	Move between different properties and navigate within combo-box drop-down lists.
Tab	Tab - Enter or leave (and save) property editing mode.
Space	Space - Toggle check-boxes and expand combo-boxes.

<table>
<tr><th>Key</th><th>Function</th></tr>
<tr><td></td><td>
</td></tr>
<tr><td>Enter</td><td>Select and save property editing.</td></tr>
</table>

When a **Review View** is active, the following **Properties** window shortcuts are also available for **[A]ttibute** and **[M]ode**.

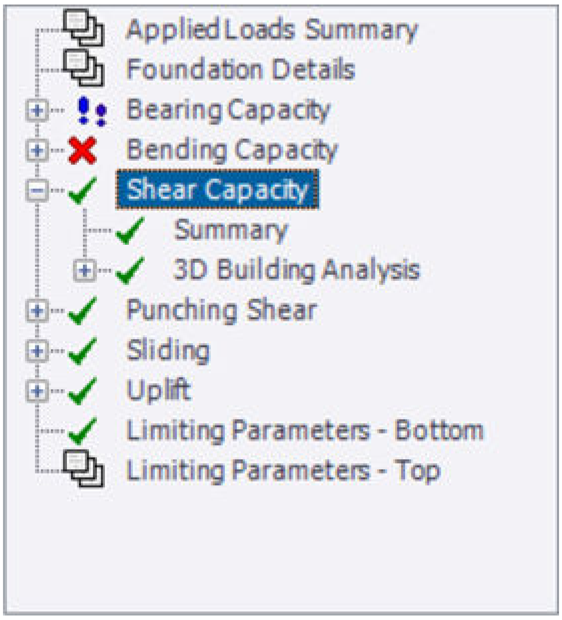


Key	Function
A	Press **A** to cycle through attributes in the **[A]ttibute** list. **TIP** If this doesn't work it is probably because the focus is in the **Properties** window. Click the **Review View** to makes it active and try again.

Key	Function
M	Press **M** to cycle through modes in the **[M]ode** list. **TIP** If this doesn't work it is probably because the focus is in the **Properties** window. Click the **Review View** to makes it active and try again.

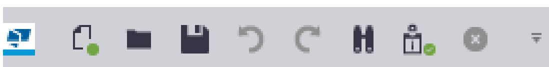
Keyboard shortcuts in tree structures

Keys that perform specific functions in any tree structure are described below.

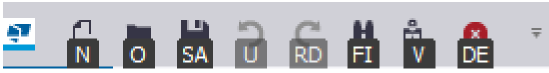
Key	Function
Right/Left arrow keys	Expand and collapse headings. 

Keyboard shortcuts to the Quick Access Toolbar

The Quick Access Toolbar is normally displayed as below.



If you press the **Alt** key letters are superimposed to enable each command to be accessed directly from the keyboard.



Command	Shortcut
New	**Alt N** (or, you can also use **Ctrl N**)
Open	**Alt O** (or, you can also use **Ctrl O**)
Save	**Alt SA**
Undo	**Alt U**
Redo	**Alt RD**
	Alt FI (or, you can also use **Ctrl F**)
	Alt V
Delete	**Alt DE**

Keyboard shortcuts to ribbon commands

All commands on the ribbon toolbars can be accessed from the keyboard after pressing the **Alt** key.

1. Press **Alt** then the appropriate letters (summarized in the table below) to open the required ribbon toolbar.

2. Press the appropriate letter(s) for the command you want to run.

Example: Press **Alt HFN** to open the **Home** toolbar and run the **Find** command shown above.

Toolbar	Shortcut
File	Press **Alt AP** then the letter(s) shown against the required command.
Home	Press **Alt H** then the letter(s) shown against the required command.
BIM Integration	Press **Alt B** then the letter(s) shown against the required command.
Model	Press **Alt M** then the letter(s) shown against the required command. .
Edit	Press **Alt E** then the letter(s) shown against the required command.
Load	Press **Alt LO** then the letter(s) shown against the required command.
Analyze	Press **Alt AN** then the letter(s) shown against the required command.
Design	Press **Alt DE** then the letter(s) shown against the required command.
Slab Deflection	Press **Alt SL** then the letter(s) shown against the required command.
Foundations	Press **Alt FO** then the letter(s) shown against the required command.
Report	Press **Alt RE** then the letter(s) shown against the required command.
Draw	Press **Alt DR** then the letter(s) shown against the required command.
Windows	Press **Alt W** then the letter(s) shown against the required command.
Loading Analysis	Press **Alt LA** then the letter(s) shown against the required command.
Results	Press **Alt RS** then the letter(s) shown against the required command.

Toolbar	Shortcut
Review	Press **Alt RV** then the letter(s) shown against the required command.
Review Data	Press **Alt RV** then the letter(s) shown against the required command.

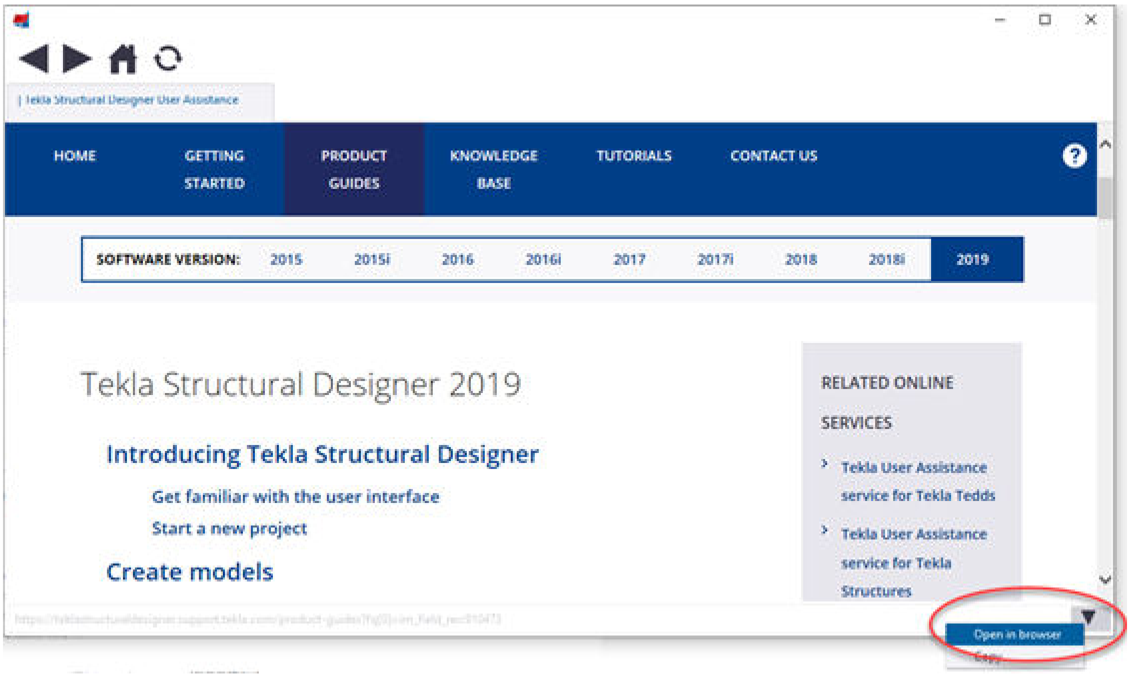
2.7 NOTE: Steps to take if the Help Viewer appears to be inactive

Please note that, since Help is displayed within a Tekla Structural Designer dialog, it will become inactive when another dialog is opened.

Should this occur, the following workarounds are suggested:

- Simply close the other dialog to continue.
- Alternatively you can access Tekla Online Help (TUA) for Structural Designer directly within any browser.

The Help can also be opened in a browser from any page by clicking the 'arrow' button at the bottom of the page as shown below:



3 BIM integration

To simplify your work, Tekla Structural Designer allows you to both export data to different file formats, and import files to Tekla Structural Designer.

Click the following links to find out more:

BIM fundamentals (Playlist)

3.1 Import model data

Tekla Structural Designer allows you to import model data from other applications.

Click the following links to find out more about importing a model into Tekla Structural Designer:

- Import a project from a Structural BIM Import file (page 109)
- Import a project from a TEL file (page 110)
- Import data from a 3D DXF file (page 116)

Import a project from a Structural BIM Import file

1. On the **BIM Integration** toolbar, click **Structural BIM Import**.

   The **BIM Integration** wizard opens.

2. Use the **[...]** button to browse and select the appropriate CXL file.

   The remainder of the fields are populated with the settings read directly from the file.

3. Click **Next** to continue.

   The **Import Options** page shows the settings related to the file, it will indicate if this is a first time import or an update, and allows the user to choose Metric or Imperial for the units, a design code and the default level type. The level type can later be adjusted in the construction levels dialog if required.

4. Click **Next** to continue.

   The **Relocate Import Model** page allows for models to be moved to around the datum position in Tekla Structural Designer from a real world co-ordinate. The extent of the model can be seen from the information displayed on the right hand side of the dialog.

5. Click **Next** to continue.

   The **Integration Filter** will be displayed. The five options shown here allow you to verify if grids, levels etc. are to be imported. Please note that the settings held in the Delete Existing... and four Update... columns should not be used when performing a First Time Import and are only used when synchronizing changes into an existing Tekla Structural Designer model.

6. Click **Next** to continue.

   Mapping of materials to recognised grades will be shown now. Here you can view the different materials recognised in the incoming CXL file and the options to alter materials with a more preferred grade.

7. Click **Next** to continue (if available).

   Following on from Material mapping, the dialog for Decking mapping will be shown (if applicable). Again this allows users to view the decking types being imported and have the option of overwriting the details if required.

8. Click **Finish**

   After the file has completed processing, the model will be displayed within the Structural 3D view and you can then proceed with validation of the structure before applying any analytical information.

BIM fundamentals (Playlist)

Integration with Tekla Structures (Playlist)

Integration with Autodesk Revit (Playlist)

Revit-TSD Integration

Import a project from a TEL file

Before importing a project from a TEL file, you are advised to be aware of the restrictions. Then, you can proceed to follow the detailed instructions to perform the import.

Restrictions

When importing TEL files, note the following points:

Round tripping	Does not exist in TEL file imports. This means that the import data is used to create new objects in the model, not to update existing ones. All existing objects and data are maintained.

Data that is imported	• Project summary (new models only)
	    • Project name, engineer, etc.
	• Support conditions
	    • Any associated UCS
	    • Spring supports, including linear and non-linear
	
	NOTE In S-Frame, for non-linear spring the default is F_max = 0. This does not

mean that the spring has zero capacity, but F_max is simply ignored.

This is not the case in Tekla Structural Designer. In Tekla Structural Designer, F_max =0 means the spring has zero capacity. Hence for models with compression-only springs imported from S-Frame, all nonlinear spring supports with F_max = 0 will need editing, or analysis will fail.

- 1D elements
    - Imported as analysis elements. However, contiguous elements are not merged into members (straight or curved).
    - Replicated by additional relatively stiff 1D elements.

    NOTE Automatic supports are not created, for example, under columns.

- Panels - area load only
    - Created as roof or wall panels without openings
- Panels - shell: tri, quad, or mixed
    - Created as meshed concrete walls (vertical planes only) or 2-way spanning slab items
    - Material properties are mapped manually during the import.
    - Thickness
- Panels - rigid or independent diaphragm
    - Created as 1-way spanning slab items.

	• Material properties are mapped manually during the import. • Thickness • Panels - holes • Created as slab or wall openings. **NOTE** The holes must be rectangular for walls and rectangular or circular for slabs. Otherwise, Tekla Structural Designer will generate a warning. • Loadcases (linear only) • Nodal loads • Settlement loads • 1D element loads, including uniform temperature loads • Area panel loads, not including uniform temperature loads • Combinations
Exclusions	• Units: the Tekla Structural Designer model units are not changed to match the TEL file units. However, values are converted to the Tekla Structural Designer model units. • There is no special handling for 2D files. 2D files are imported in the same plane as they are defined in S-Frame (the X-Y plane), and default constraints are not imported. • Default constraints are not imported, and no warning is generated. Default constraints are supports applied to all nodes without exception internally during analysis. Default constraints are not displayed in the S-Frame interface. For models

marked as 2D, default constraints restrict displacement to the X-Y plane and are as follows: Fz, My and Mz fixed. Default constraints may be manually applied in 3D models, and the S-Frame model can be examined to confirm their nature. The import does not replicate default constraints. To ensure equivalence, default constraints must be applied manually to all nodes, either in the S-Frame model prior to import, or in Tekla Structural Designer subsequently.

- The import does not create any physical members. S-Frame physical members are treated like any other 1D element and imported as a single analysis element. In particular, Tekla Structural Designer does not merge contiguous elements into members (straight or curved) or identify columns, beams, and so on. No warnings are generated.

  In addition, for S-Frame physical models please note the following:

  - Intermediate nodes that do not form the ends of other elements are not imported. If such nodes have supports applied, the model will not be equivalent and should be adjusted to ensure equivalence.
  - If physical members have tapered sections, the sections should be sub-divided in S-Frame before importing to Tekla Structural Designer to produce an equivalent model.
  - Alternatively, the S-Frame model can be converted to an analytical model in S-Frame prior to import, using the S-Frame command for this.

- Staged construction data is not imported, and no warning is generated. Typically, the entire model is imported, representing the last stage in which the model is complete. Otherwise, turn the **Staged Construction** setting in S-Frame off prior to import. This will remove all stages but the last one, and issue a warning to this effect. The model is then non-staged, and so should be valid for import.
- Although the following can be modeled as single objects in Tekla Structural Designer, no attempt is made to import them as single objects from collections of S-Frame objects:
  - Mid-pier walls
  - Trusses
  - Portal frames
- Inactive elements are imported as inactive analysis elements of the beam type.

  NOTE Inactive elements are quite likely to originate from tension-only cross bracing in Fastrak Building Designer models. In this situation, we recommend that you click the warning to identify the relevant part of the model, delete both "braces", and create new braces using the specific X-brace pair.

- Wall and strip integration lines are not imported. A warning is issued to this effect.
- Tapered sections: a 1D element is imported. However, no tapered section dimension data is imported. A warning is issued to this effect.

- Prestress data for 1D & 2D elements is not imported.
- Percentage fixity data for 1D elements is not imported.
- Non-linear spring data by graph: 1D elements and supports are imported, but the spring stiffnesses are set to 0. A warning is issued to this effect.
- Non-structural alignment and offsets (cardinal point data) are ignored, and warnings are issued if they are non-zero.
- Panels: general diaphragms, mat foundations, membranes and plates are excluded, and a warning is issued.
- Diaphragm panel node exclusions are ignored.
- 2D elements are excluded, and a warning is issued.
- Meshing properties for panels are excluded without warning.
- Shear Walls: only quadrilaterals can be created.
- Diaphragm constraints are excluded, and a warning is issued to this effect. Any diaphragm constraints must be replicated in Tekla Structural Designer to ensure equivalence.
- Slaved nodes are excluded, and a warning is issued.
- Lumped mass is excluded.
- Groups are excluded.
- Notional load factors: NHF and EHF are added to combinations, with the sign indicating positive or negative for each direction. However, the actual value is ignored. The standard notional load calculation method and default strength factor are used.

	• Non-zero gravitational factors for Global X & Y are excluded, and a warning is issued. • Thermal gradient loads for 1D and 2D elements are excluded. • Moving loads are excluded. • Time history loads are excluded. • 2D elements are excluded, and a warning is issued. • RSA data is excluded.

Instructions

1. On the **BIM Integration** tab, click **'TEL' File Import**.

   The **'TEL' File Import** button is not active unless you have a Tekla Structural Designer document open.

   The **BIM Integration** wizard opens.
2. Click **...** to browse the .TEL file, and click **Next**.
3. If necessary, adjust the location and rotation of the import model, and click **Next**.
4. Select an appropriate material type and grade for each material using the lists.
5. Click **Finish**.

The model is now imported. Any associated warning messaged can be reviewed in the **BIM** sub-branch of the **Status tree**.

Import data from a 3D DXF file

See the following paragraphs for the restrictions of 3D DXF import, and the instructions on importing data from a 3D DXF file.

Restrictions

Note the following points when importing 3D DXF files:

- Round tripping does not exist in TEL file imports. This means that the import data is used to create new objects in the model, not to update existing ones. All existing objects and data are maintained.
- Analysis elements are created from line segments in the selected layers of the DXF file as follows:
  - All lines in these layers become 1D analysis elements.

- Arcs and circles in the selected layers are excluded without a warning.
- Blocks are not handled, and no warnings are issued.
- Polylines in the selected layers are excluded without a warning.
- 3D solids in the selected layers are excluded without any warning.
- 2D faces for 3D objects in the selected layers are excluded without a warning.
- All ends of lines in the selected layers become nodes.
- Nodes are not introduced in the intersections of crossing lines.
- The following elements are not included in the import:
  - Reading of any text
  - Intelligence on "through members"
  - Gridlines and construction lines
  - 2D elements
  - Supports
  - Section properties
  - Materials
  - Loads
  - Combinations

Instructions

1. On the **BIM Integration** tab, click **3D DXF Import**.

   The **3D DXF Import** can only be selected in a 3D view.

   The **Open** dialog opens.
2. Browse to the required .dxf file.
3. Click **Open**.

   The **DXF Import Wizard** opens.
4. Select the layers and colors that you want to import.
5. If necessary, apply offsets and rotate the model before data is imported.
6. Click **OK**.

Any line segments found in the selected layers are now imported as analysis elements.

3.2 Working collaboratively with Trimble Connect

Trimble Connect is a project collaboration tool allowing project stakeholders access to reliable, up-to-date project information. It is available as a cloud-based platform (Trimble Connect Web) and a Windows application (Trimble Connect for Windows). Projects are synchronised between the Windows app and the cloud.

NOTE To learn more about Trimble Connect, see:

- https://connect.trimble.com/
- https://trimbleconnect.support.tekla.com/

NOTE You need to have a Trimble Identity before you can start using Trimble Connect

Trimble Connect Project Explorer is used within Tekla Structural Designer to control the flow of information between the open model and a Trimble Connect project.

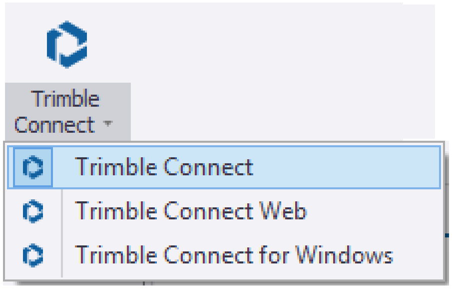
With Trimble Connect Project Explorer you can:

- link a Tekla Structural Designer model to a Trimble Connect project
- create and rename folders in a Trimble Connect project
- view a file list and rename files in a Trimble Connect project
- upload an IFC of the model to a Trimble Connect project
- upload drawings
- upload reports

NOTE A Tekla Structural Designer model can only be linked to a Trimble Connect project from the Trimble Connect Project Explorer and not from the web or Windows apps. The model must be linked before any information can move between Tekla Structural Designer and Trimble Connect.

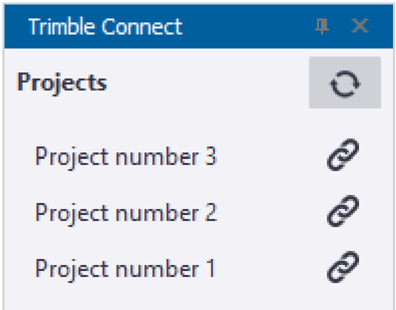
Launch Trimble Connect Project Explorer

1. Log in using your Trimble Identity, (if not already signed in).
2. On the **BIM Integration** tab, click the **Trimble Connect** droplist.
3. Select **Trimble Connect**



Trimble Connect Project Explorer opens, either as a docked window, or as a tab on the right of the interface.

Available projects are listed with link icons as below.



TIP If the project in which you want to collaborate is not shown, try clicking the refresh button to synchronise with the cloud. If it still not shown you would need somebody with the appropriate permissions to create it, or grant you access, using Trimble Connect Web, or Trimble Connect for Windows.

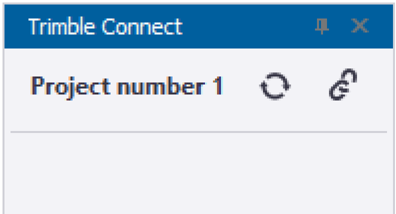
TIP If a 'Cannot find linked project' message is displayed, this indicates either that you do not have permission to view the project; or, that the project to which the model was previously linked has been deleted - in this situation you would need to click 'Unlink' before you are able to link the model with another project.

Link or unlink a project

1. To link to a project:

1. Click next to the project you want to link to.

   When the model is linked, the project name appears at the top of the window and the icon changes as shown below.

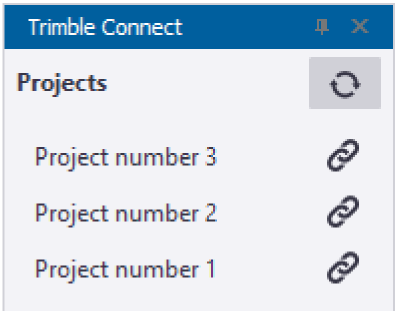


Any folders or files in the Trimble Connect project that you have *read* access to are also displayed.

2. To unlink from a project:

2. Click  to unlink the project.

   The full list of available projects is displayed once more.



Create folders, rename folders, rename files

When you link to a Trimble Connect project the existing project folder structure is displayed. You can add to this, if required.

NOTE You can only create but not delete folders in Trimble Connect Project Explorer. Folders can only be deleted in Trimble Connect Web, or Trimble Connect for Windows.

You can only rename folders and files to which you have *write* access.

---

1. To create a folder:
1. Right click in the Trimble Connect Project Explorer window.
2. Select **Create folder** and enter the folder name.
2. To create a subfolder:
3. Right click on an existing folder.
4. Click Create folder and enter the folder name.

3. To rename a file or folder:
5. Right click on the file/folder.
6. Select **Rename**

Upload an IFC file of a model

1. Right-click on the folder where you want to create the IFC.
2. Select **Upload New IFC file**
3. Follow the steps in the Export to IFC wizard to create the ifc file.
    a. Adjust the location and rotation of the model, and click **Next**
    b. Specify the export names of material grades, and click **Next**
    c. Specify the ifc file name

    The ifc file will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the IFC model in Trimble Connect.

NOTE If the model is subsequently updated, you can upload a new revision of the ifc file. To do this, right-click on the ifc file name and select **Upload new version of IFC**.

Upload a multi-member drawing

1. Open a view displaying the part of the model that you want to include in the drawing.
2. On the **Draw** ribbon, click the drawing type.

    The **DXF Export Preferences** dialog box opens.
3. Select **Upload to Trimble Connect**

    NOTE The above option is only available after the model has been linked to a Trimble Connect project.

    A default drawing name for the upload is displayed.

    If required, the drawing name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.

    The **Create link between model member and uploaded drawing** option is not applicable to multi-member drawings, so this option cannot be selected.

4. Click **OK**

   The drawing will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the drawing in Trimble Connect.

NOTE The **Defaults** button on the **DXF Export Preferences** dialog box can be used to set a default file location, output target and link creation option. The options defined here will be true for the whole model.

Upload a single member drawing

1. Highlight the member for which you want to create a drawing.
2. Right click, then from the context menu select **Generate Drawing**

   The **DXF Export Preferences** dialog box opens.
3. Select **Upload to Trimble Connect**

   NOTE The above option is only available after the model has been linked to a Trimble Connect project.

   A default drawing name for the upload is displayed.

   If required, the drawing name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.
4. To link the model member and drawing, click **Create link between model member and uploaded drawing**.

   See the *Link a drawing or report to an existing IFC* topic below for more details about this option.
5. Click **OK**

   The drawing will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the drawing in Trimble Connect.

NOTE The **Defaults** button on the **DXF Export Preferences** dialog box can be used to set a default file location, output target and link creation option. The options defined here apply for the whole model.

Upload a model report

1. Open the required report in Tekla Structural Designer.
2. On the **Report** ribbon, click **PDF Upload**

NOTE	Both **PDF Upload** and **Upload Settings** are only available after the model has been linked to a Trimble Connect project. (The latter can be used to set a default file location, output target and link creation option that applies for the whole model.)

The **Upload Report** dialog box opens and a pdf name for the report is displayed.

If required, the pdf name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.

The **Create link between model member and uploaded report** option is not applicable to model reports, so this option cannot be selected.

3. Click **OK**

   The pdf will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the report in Trimble Connect.

Upload a member report

1. Highlight the member for which you want to create a report.
2. Right click, then from the context menu select **Report for Member**

   The report is displayed in Tekla Structural Designer.
3. On the **Report** ribbon, click **PDF Upload**

NOTE	Both **PDF Upload** and **Upload Settings** are only available after the model has been linked to a Trimble Connect project. (The latter can be used to set a default file location, output target and link creation option that applies for the whole model.)

   The **Upload Report** dialog box opens and a pdf name for the report is displayed.

   If required, the pdf name and upload folder can be changed by clicking **[...]** to open the **Trimble Connect File Browser**.
4. To link the model member and report, click **Create link between model member and uploaded report**.

   See the *Link a drawing or report to an existing IFC* topic below for more details about this option.
5. Click **OK**

   The pdf will appear in the selected folder, then after an 'assimilation' process it will be possible for collaborators to view the report in Trimble Connect.

Link a drawing or report to an existing IFC

When drawings and reports are uploaded they are automatically linked to an existing IFC file as long as the appropriate option is selected (*Create link between model member and uploaded report / Create link between model member and uploaded drawing).*

It is important to **save** the Tekla Structural Designer model after exporting the IFC to Trimble Connect so that the link between the two is stored in the TSD model.

Linking reports

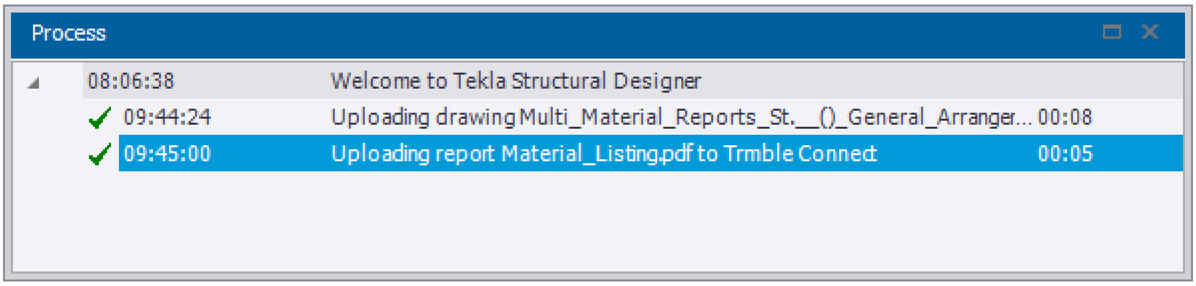
Linking of reports occurs for member reports only. If there are 3 IFC objects associated with one member (eg a 3 span beam) there will be a link created on each of the 3 IFC objects.

Linking drawings

Linking of drawings occurs for member drawings only. If there are 3 IFC objects associated with one TSD member (eg a 3 span beam) there will be a link created on each of the 3 IFC objects.

Check linking progress in the Process Window

The progress of the linking is logged in the process window. This will show the number of successful/unsuccessful links that have been created.



There are a few reasons why linking could be unsuccessful:

1. The member associated with the report/drawing does not exist in IFC model
2. Connection to the Trimble Connect project is lost
3. IFC model was not successfully uploaded

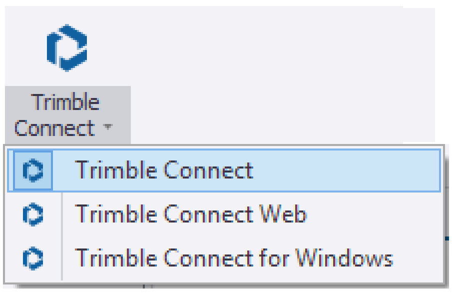
Open Trimble Connect to a model view for an IFC

1. In the Trimble Connect Project Explorer window, right-click on the IFC file to be viewed.
2. Select **Open in Trimble Connect Web**, or **Open in Trimble Connect for Windows** as required.

   Trimble Connect opens the linked project and displays a view of the selected ifc.

Open Trimble Connect

1. To open Trimble Connect from the ribbon:
1. If not already signed in, click log in (at the top right of the interface).
2. On the **BIM Integration** tab, click the **Trimble Connect** droplist.
3. Select select **Trimble Connect Web**, or **Trimble Connect for Windows** as required.



   - **Trimble Connect Web:** This launches the in-browser web app. If the open model is associated with a Trimble Connect project, the project itself is opened, otherwise a default Trimble Connect page is opened.
   - **Trimble Connect for Window:** This launches the Windows application. Again, if the open model is associated with a Trimble Connect project then that project is opened.
2. To open Trimble Connect from **Trimble Connect Project Explorer**:
4. Right-click on a folder in the Trimble Connect Project Explorer window.
5. Select **Open in Trimble Connect Web**, or **Open in Trimble Connect for Windows** as required.

   The in-browser web or Windows app is launched and opened at the selected folder.

3.3 Export to Trimble applications

Click the following links to find out more about exporting to other Trimble applications:

- Export a model to Tekla Structures (page 126)
- Export to Tekla Connection Designer (page 127)
- Export to Tekla Portal Frame Designer (page 128)
- Export to Tekla Tedds (page 132)

Export a model to Tekla Structures

To export a model to Tekla Structures, see the following instructions.

1. Create your model in the usual way.
2. Ensure that in Model Settings > Structural BIM > Export (page 1047) the settings are as you require.

   NOTE For optimum performance of rebar transfer to Tekla Structures ensure that the following are enabled:

   - Separate objects for each stack > Concrete columns
   - Separate objects for each span > Concrete beams
   - Separate objects for each panel > Shear walls

3. On the **BIM Integration** tab, click **Tekla Structures Export**.

   The **BIM Integration** wizard opens.
4. On the first page, click **Next**.
5. Adjust the location and rotation of the model, and click **Next**.
6. Select the items that are included in the model, and click **Next**.
7. Specify the export names of material grades, and click **Next**.
8. Specify the file name and location.
9. Select whether the file is exported for the first time, or whether you want to update an existing model.
10. Click **Finish**.
11. Start Tekla Structures and open the file to see your project.

NOTE Prior to running the export you can control which levels will be included by setting **Include in export** as required in the Level Properties (page 933). This allows individual levels to be:

- only included if the level is a floor
- always included

- never included

Integration with Tekla Structures (Playlist)

Export to Tekla Connection Designer

Before exporting connections to Tekla Connection Designer, you are advised to be aware of the limitations and also the recommended workflows for the different connection types.

NOTE Only valid base plate, column splice and moment connections listed in the Project Workspace Connections (page 83) tree can be exported.

To export a single connection

NOTE Export requires a Tekla Connection Designer licence.

1. Right-click the connection in a Scene View.
2. Choose Export *(connection name)* to Tekla Connection Designer from the right-click menu.

   The Tekla Connection Designer application opens to allow the selected connection to be designed.

To export multiple connections

NOTE Export requires a Tekla Connection Designer licence.

1. Select the connections to be exported in a Scene View.
2. On the **BIM Integration** tab, click **TCD Export**.

   The Tekla Connection Designer application opens to allow the selected connections to be designed.

To return connection data from Tekla Connection Designer

Connection data from Tekla Connection Designer can be returned to Tekla Structural Designer as follows:

1. In Tekla Connection Designer click Connection> Return Connection to Tekla Structural Designer

NOTE In the current release, Moment Connection data from Tekla Connection Designer is only partially returned. (e.g. Bolt layouts and endplate dimensions are not returned).

Export to Tekla Portal Frame Designer

Export to Tekla Portal Frame Designer workflow

It is important to follow the correct definition, loading and design sequence when frames are to be transferred for design to Tekla Portal Frame Designer.

1. Define frames/loading etc inTekla Structural Designer.
2. Run Analyse All (Static).

   Analysis is necessary in order for the loading to be included in the export to Tekla Portal Frame Designer.
3. Export the frame to Tekla Portal Frame Designer to design it and check member stability.
4. Return revised sections/grades and haunch geometry back to the Tekla Structural Designer model.

NOTE Only section sizes, haunch dimensions and steel grades are returned to Tekla Structural Designer. Any other changes to the model data and loading are not returned.

How loading, restraints and supports are handled in the export

Loadcases and combinations

When you open the Loadcases dialog in Tekla Portal Frame Designer the first four loadcases are automatically created and named as follows:

- Self Weight
- Frame Dead Load
- Frame Service Load
- Frame Imposed Load

The loading in these loadcases will be taken from Tekla Structural Designer loadcases (providing the loadcase names have been specified exactly as per the below table):

Tekla Structural Designer Loadcase	Tekla Portal Frame Designer Loadcase
Self weight - excluding slabs	Self weight

Tekla Structural Designer Loadcase	Tekla Portal Frame Designer Loadcase
Dead	Frame Dead Load
Services	Frame Service Load
Imposed	Frame Imposed Load

Every other Tekla Structural Designer loadcase that applies in-plane loading to the frame gets exported to Tekla Portal Frame Designer retaining it's original name.

If a particular Tekla Structural Designer loadcase does not apply any in-plane load to the frame being exported, the loadcase is not exported and it is also removed from the Tekla Portal Frame Designer load combinations.

If the Tekla Structural Designer model does not contain loadcases named "Dead", "Services" or "Imposed"; the "Frame Dead Load", "Frame Service Load" and "Frame Imposed Load" loadcases are created regardless, (defaulting to the loading values specified in the Tekla Portal Frame Designer Preferences). However, since these loadcases are not included in any load combinations they will have no affect on the design.

If the Tekla Structural Designer model contains loadcases named "Dead", "Services" or "Imposed" which do not apply any in-plane load to the frame being exported; then in the same way as any other loadcase that does not apply any in-plane load, the loadcase is removed from the Tekla Portal Frame Designer load combinations. (In this situation the "Frame Dead Load", "Frame Service Load" or "Frame Imposed Load" loadcase is still created regardless (defaulting to the loading value specified in the Tekla Portal Frame Designer Preferences), but once again since it does not get included in any load combinations it will have no affect on the design.

Once loadcases that do not apply in-plane load have been removed from a load combination, if the resulting combination is empty, or if it only contains self weight, the combination itself is removed.

Lateral Restraints

The Tekla Structural Designer lateral restraints are exported, strut restraints are ignored.

NOTE The below mappings assume default rotations of the rafters and columns. (Changing the rotation by 180 degrees flips the restrained flange in Tekla Portal Frame Designer.)

Rafter Restraints

These are exported to Tekla Portal Frame Designer restraints as follows

Tekla Structural Designer Lateral Restraint		Tekla Portal Frame Designer Restraint
Top Flange	Bottom Flange	
X	X	Torsional

Tekla Structural Designer Lateral Restraint		Tekla Portal Frame Designer Restraint
Top Flange	**Bottom Flange**	
X	-	Outer
-	X	Inner
-	-	No restraint

Column Restraints

These are exported to Tekla Portal Frame Designer restraints as follows

Tekla Structural Designer Lateral Restraint		Tekla Portal Frame Designer Restraint
Face A	**Face C**	
X	X	Torsional
X	-	Outer
-	X	Inner
-	-	No restraint

NOTE The rafter top flange and column face A can be identified graphically, for details expand the Axis systems topic and then refer to the **Object Orientation** section within it.

Supports

The Fy, Fz and Mx Tekla Structural Designer support properties are used to configure the Tekla Portal Frame Designer base fixities as follows

Tekla Structural Designer Support	Tekla Portal Frame Designer Base Fixity	
Fy Fixed	Horiz. Stiffness (ULS, SLS, Stability): Restrained	
Fz Fixed	Vertic. Stiffness (ULS, SLS, Stability): Restrained	
Mx Fixed	ULS:	Rot. 100%, Cap. 100%
	SLS:	Restrained
	Stability:	Restrained
Fy Free-Release	Horiz. Stiffness (ULS, SLS, Stability): 0kN/m	
Fz Free-Release	Vertic. Stiffness (ULS, SLS, Stability): 0kN/m	
Mx Free-Release	ULS:	Rot. Free, Cap. 1%
	SLS:	Free
	Stability:	Free
Fy Free-Spring Linear: XkN/m	Horiz. Stiffness (ULS, SLS, Stability): XkN/m	
Fz Free-Spring Linear: XkN/m	Vertic. Stiffness (ULS, SLS, Stability): XkN/m	

Tekla Structural Designer Support	Tekla Portal Frame Designer Base Fixity	
Mx Free-Spring Linear: XkNm/rad	ULS:	Rot. XkNm/rad, Cap. 1%
	SLS:	XkNm/rad
	Stability:	XkNm/rad
Fy Free-Spring Non Linear: -ve=XkN/m, +ve YkN/m	Horiz. Stiffness (ULS, SLS, Stability): Max (X, Y) kN/m	
Fz Free-Spring Non Linear: -ve=XkN/m, +ve YkN/m	Vertic. Stiffness (ULS, SLS, Stability): Max (X, Y) kN/m	
Mx Free-Spring Non Linear: -ve=XkNm/rad, +ve YkNm/rad	ULS:	Rot. Max (X, Y) kNm/rad, Cap. 1%
	SLS:	Max (X, Y) kNm/rad
	Stability:	Max (X, Y) kNm/rad
Mx Free-Nominally Pinned: 10%	ULS:	Rot. Free, Cap. 1%
	SLS:	20%
	Stability:	10%
Mx Free-Nominally Pinned: X%	ULS:	Rot. X%, Cap. 1%
	SLS:	X%
	Stability:	X%
Mx Free-Nominally Fixed: X%	ULS:	Rot. X%, Cap. 100%
	SLS:	X%
	Stability:	X%

To export a single frame

NOTE Export requires a Tekla Portal Frame Designer licence.

1. Right-click the frame in a Scene View.
2. Choose Export *(frame name)* to Tekla Portal Frame Designer from the right-click menu.

   The Tekla Portal Frame Designer application opens to allow the selected frame to be designed.

To export multiple frames

NOTE Export requires a Tekla Portal Frame Designer licence.

1. Select the frames to be exported in a Scene View.
2. On the **Home** tab, click **TPFD Export**.

   The Tekla Portal Frame Designer application opens to allow the selected frames to be designed.

To return revised sections from Tekla Portal Frame Designer

If the section sizes have been revised in Tekla Portal Frame Designer they can be returned to Tekla Structural Designer as follows:

1. In Tekla Portal Frame Designer click **Portal Frame**> **Return sections to model**

Export to Tekla Tedds

Once they have been designed using Tekla Tedds, precast and/or timber members can then be exported to Tedds in order to output the calculations.

NOTE	Export requires a Tekla Tedds licence.

Understanding each of the export options

Command	Description
Export to Tekla Tedds > Model	Launches Tekla Tedds and: • Creates a new Tedds Project containing seperate documents for each member in the model. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > Member	This option only appears for a highlighted member if the option to design using groups is not active. Launches Tekla Tedds and: • Creates a new Tedds Project containing a single document for the member. • The data exported is taken from the last design run (either Member

Command	Description
	or Selection) associated with the member. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > Group	This option only appears for a highlighted member if the option to design using groups is active. Launches Tekla Tedds and: • Creates a new Tedds Project containing a single document for the group. • The data exported is taken from the last design run (either Member, Group, or Selection) associated with the group. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > Selection	This option only appears if one or more members are selected. Launches Tekla Tedds and: • Creates a new Tedds Project containing seperate documents for each member or group in the selection. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.
Export to Tekla Tedds > *<Substructure name>*	Launches Tekla Tedds and: • Creates a new Tedds Project containing seperate documents for each member in the sub structure. Changes made in the Tedds Project are not returned to the Tekla Structural Designer model.

To export all timber and precast members

1. In the **Project Workspace**, click **Group** tab.

2. In the tree, right-click **Groups**.
3. In the context menu, select **Export using Tekla Tedds > Model**

   Tekla Tedds opens to allow the calculations for the all the exported members to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a single member

1. Right-click the member in a Scene View.
2. Choose **Export to Tekla Tedds > Member** from the right-click menu.

   Tekla Tedds opens to allow the calculations for the highlighted member to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export multiple members

1. Select the precast and/or timber members to be exported in a Scene View.
2. Right-click and choose **Export to Tekla Tedds > Selection** from the context menu.

   Tekla Tedds opens to allow the calculations for the selected members to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a group

1. Right-click any member of the group in a Scene View.
2. Choose **Export to Tekla Tedds > Group** from the right-click menu.

   Tekla Tedds opens to allow the calculations for the highlighted group to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

To export a substructure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to export.
3. In the context menu, select **Export using Tekla Tedds > <*Sub structure name*>**

   Tekla Tedds opens to allow the calculations for the substructure to be output or recalculated.

   NOTE If exported calculations are recalculated, the revised results are **not** sent back to the Tekla Structural Designer model.

3.4 Export to and import from other applications

Tekla Structural Designer allows you to import and export data to different file formats.

Click the following links to find out more about importing to and exporting from other applications:

- Export a model to Autodesk Revit (page 135)
- Export a model to IFC (page 136)
- Export to and import from Westok Cellbeam (page 137)
- Export to and import from FBEAM (page 138)
- Export a model to ADAPT (page 143)
- Export a model to STAAD (page 147)
- Export a model to Autodesk Robot Structural Analysis (page 148)
- Export a model to the cloud (page 149)
- Export to One Click LCA (page 149)
- Export to IDEA StatiCa Connection Design (page 152)

Export a model to Autodesk Revit

To export a model to Autodesk® Revit®, see the following instructions.

1. Create the model as usual.
2. On the **BIM Integration** tab, click **Autodesk Revit Export**.

   The **BIM Integration** wizard opens.
3. Adjust the location and rotation of the model, and click **Next**.

4. Select the items that are included in the model, and click **Next**.
5. Specify the export names of material grades, and click **Next**.
6. Specify the file name and location.
7. Select whether the file is exported for the first time, or whether you want to update an existing model.
8. Click **Finish**.

NOTE Prior to running the export you can control which levels will be included by setting **Include in export** as required in the Level Properties (page 933). This allows individual levels to be:

- only included if the level is a floor
- always included
- never included

NOTE To open the exported file in Revit, you will need to have installed the Tekla Structural Designer Integrator for Autodesk Revit. This is available for download from the following link: https://download.tekla.com/tekla-structural-designer/for-businesses/all-downloads?field_package_reference_tid_selective=5713

Integration with Autodesk Revit (Playlist)

Revit-TSD Integration

Export a model to IFC

To export your model to IFC, see the following instructions.

1. Create the model as usual.
2. On the **BIM Integration** tab, click **IFC Export**.

   The **BIM Integration** wizard opens at the **Relocate Export Model** page.

   The **Relocate Export Model** page allows for models to be moved from the datum position in Tekla Structural Designer to a real world co-ordinate.
3. Move and rotate the export model if required, and click **Next**.
4. On the **Integration Filter** page, select the items that are included in the export model, and click **Next**.
5. On the **IFC Export File** page, select whether the file is exported for the first time, or whether you want to update an existing model. Then specify the file format, name and location.
6. Click **Finish**.

NOTE Prior to running the export you can control which levels will be included by setting **Include in export** as required in the Level Properties (page 933). This allows individual levels to be:

- only included if the level is a floor
- always included
- never included

Export to and import from Westok Cellbeam

To export and import beams to and from Westok Cellbeam, see the following restrictions and instructions.

RESTRICTION Cellbeam import and export are only available for BS and Eurocode head codes.

Export to Cellbeam

1. On the **BIM Integration** tab, click **Cellbeam Export**.

   The **Export Westok Beams** dialog box opens. Any Westok Cellular beams in the model are listed in the dialog.
2. Select the beams that you want to transfer to Westok Cellbeam, and click **Next**.
3. Select the export format and name the file.
4. Click **Next**.
5. Ensure that you have selected the design combinations whose results you want to use in the Westok beam design.
6. Click **Next**.
7. Select the folder where you want to place the Westok files, and click **Export**.
8. Click **Finish**.

Import from Cellbeam

NOTE In order to import Westok beams into your model, you must have previously created the beams in your Tekla Structural Designer model, and have exported them to Westok design. When you do this, each beam is given a unique identifier. This means that when you import the results from the Westok file, Tekla Structural Designer knows to which beam in your model the imported details apply.

1. On the **BIM Integration** tab, click **Cellbeam Import**.

   The **Import Westok Beams** dialog box opens.
2. Click **Add...**
3. Select the files that you want to import.
4. If necessary, change the file names.
5. Click **Open**.
6. Click **Import**.

   Provided that Tekla Structural Designer recognizes the identifier, the details in the files are imported to the associated beam in your model.

Export to and import from FBEAM

To export to and import from FBEAM, see the following instructions.

TIP Create substructures for your FABSEC® beams to make it easier to review and manage the FBEAM workflow.

Overview

FBEAM is software from Fabsec for the design of composite and non-composite plain and cellular beams.

The workflow for taking FABSEC® beams from a Tekla Structural Designer model and designing then within FBEAM is as follows:

- Select the FABSEC® beams in your Tekla Structural Designer model and click **FBEAM Export**.

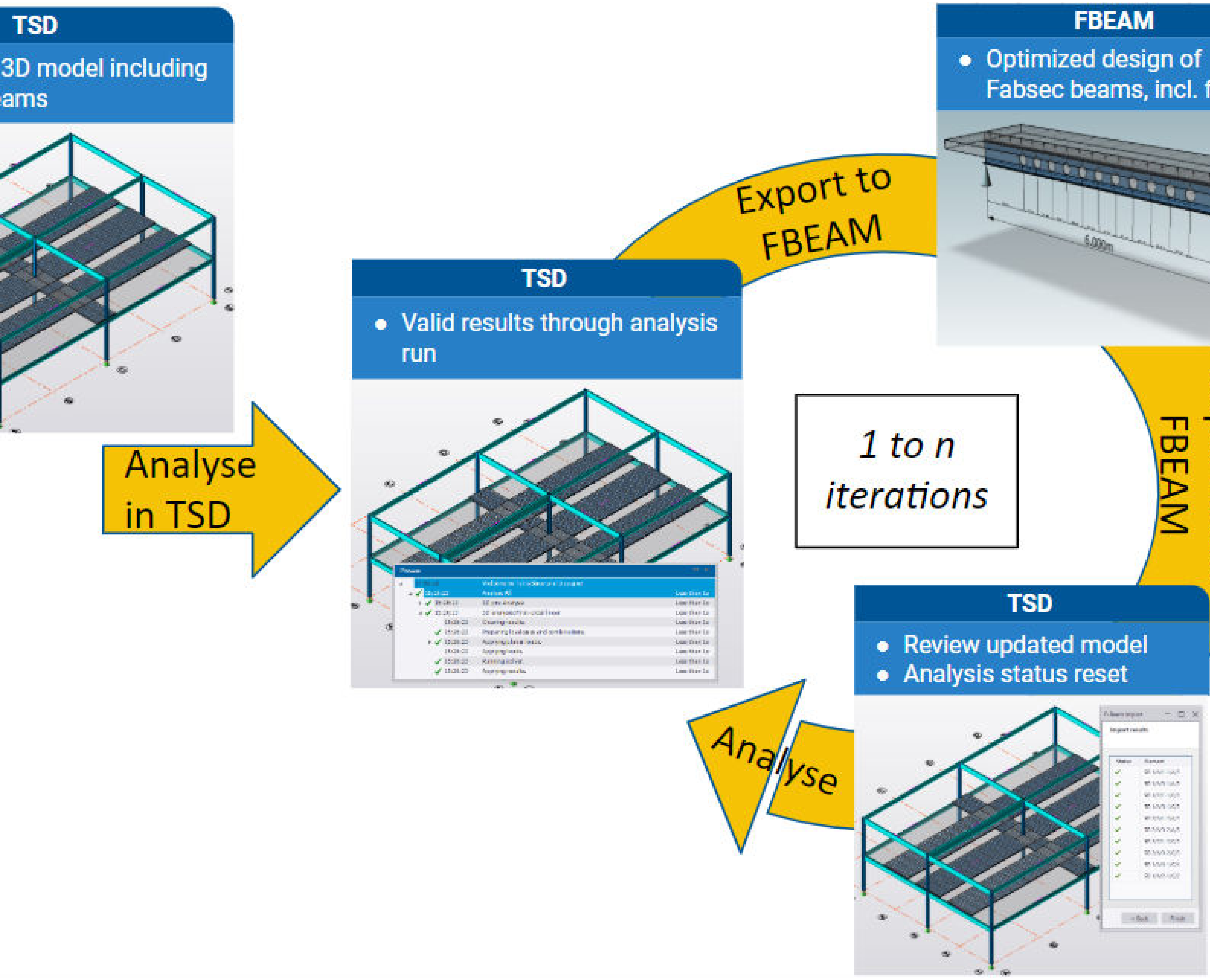
  This generates an XML file containing all the required properties required for each member.
- You can then launch FBEAM and open the project you wish to import the beams into.
- You then click the import button which starts FBEAM's import work flow. This will allow you to load the previously created XML file and select a subset or all of the beams in the file to import.

  Once the process is complete FBEAM will have created all of the members and their associated properties.
- You can then design the FABSEC® beams and specify fire protection coatings.

- Once complete, click the export button in FBEAM.

  This generates another XML file that can be imported back into Tekla Structural Designer.
- In Tekla Structural Designer, (with the model that was used to export the beams open), click the **FBEAM Import** to start the import workflow. This will then update each of the previously exported FABSEC® beams with the properties contained within the FBEAM generated XML file.

The above workflow can be summarized as follows:



Limitations

This following limitations apply to the import/export of FABSEC® beams:

1. Consider the case where multiple combinations are exported and a loadcase of a particular load type e.g Dead appears in one combination but not all. In FBEAM the analysis will include all loadcases of the same type in each combination. This can overestimate the load significantly.
2. When Wind loadcases are exported to the 'Additional Load Set' for a particular beam, Tekla Structural Designer searches for the Wind loadcase that has the largest positive end force and exports that value to the 'Additional Load Set' and gives it a 'Type', 'Wind' in FBEAM. Similarly for the maximum negative value in Tekla Structural Designer to the 'Type', 'Wind Up' in FBEAM. This loadcase will be used in the design of that beam for all positive wind combinations and all negative wind combinations respectively. Whether a wind combination is considered negative is determined from the start and end values of the distributed (Wind) load applied to that beam (or point loads). If both are negative then the wind combination is negative (uplift).
3. The special loadcase in Tekla Structural Designer 'Self weight - excluding slabs' (i.e. beam self weight) is assumed always to be included in design combinations. Consequently, the setting in FBEAM, 'Include the beam self weight' is always switched on.
4. In any combination if any load type e.g. Dead or Imposed appears more than once with different load factors, only one value of load factor is exported. This situation is unusual but might occur for imposed loads for example when different psi factors might apply.
5. Any load types or direction not supported by FBEAM are not passed through. This includes minor axis loading, axial loading, trapezoidal loads and torsion.
6. Imposed loads are typically entered into the 'protected' Construction Stage load combination in Tekla Structural Designer to represent live loads during construction e.g. due to 'heaping'. These are exported to FBEAM as Load Type 'CL'. They are given a load factor that will be the maximum from all imposed loadcases included in the Construction Stage combination although typically only one is included and with a load factor of 1.5.
7. Beam self weight loads are typically entered into the 'protected' Construction Stage load combination in Tekla Structural Designer. Normally these would be represented by the Load Type ' D' in FBEAM. However, in order to avoid taking into account all of the dead loads, including those at composite stage, the Construction Stage load combination in FBEAM is populated with the beam self weight as a 'cladding' load, 'C'.

8. The Slab Wet loads exported from Tekla Structural Designer should set the 'Construction Stage Floor type' in FBEAM to 'CL' - typically this will ensure that a load factor of 1.5 is used in the design combination for Construction Stage. For EC this is correct but for BS it should be set to 'D'. This setting is not accessible to the export from Tekla Structural Designer and so must be changed manually if BS 5950 design is to be used.
9. For composite beams when the wet weight of the concrete is included in the Construction Stage combination in Tekla Structural Designer, the results in FBEAM will only be correct in the following circumstances:
    a. Automatic Loading is checked on in the loadcase dialogue when Slab Dry is selected as the load Type,
    b. Automatic Loading is checked on in the loadcase dialogue when Slab Wet is selected as the load Type,
    c. At least on combination includes the Slab Dry loadcase,
    d. The Construction Stage combination includes the Slab Wet loadcase.
10. Beams can be rotated in Tekla Structural Designer but in FBEAM are always assumed to have their web vertical i.e. unrotated. Consequently any beam rotation is not passed through from Tekla Structural Designer to FBEAM and any loads in the global axis system that are applied to a rotated beam are passed through to FBEAM as if they are applied through the vertical web of the beam.
11. When several beams on the same floor are exported from Tekla Structural Designer there can be several instance of 'Floor' in FBEAM. Each of these could be assigned a different size of 'Mesh'. However, only one of these will be imported into Tekla Structural Designer. It is recommended that either only one value of mesh is set for these beams or that any required changes to the slab mesh are made in Tekla Structural Designer and not FBEAM.
12. In composite design, transverse reinforcement whether in the slab or as part of the beam data is exported and imported as an area only. In Tekla Structural Designer the area is derived from the bar size and spacing (or mesh size). In the case of bars, the export transfers the area and in FBEAM the default bar size will be shown with the exported area. For import the area in FBEAM is processed by Tekla Structural Designer to find a bar size and spacing that has an area greater than (or equal to) the area held in FBEAM. Furthermore when 'loose bars' are used in the slab (unusual) these are added to any from the beam data and transferred as one value to FBEAM. Clearly on import these cannot be separated. This approach has an impact on round tripping that the user must be aware of and make appropriate adjustments post import or export.
13. Partial length shear connector layout with empty segments cannot be created in Tekla Structural Designer. If cases like these are detected when importing from FBEAM the partial shear connector layout will be extended for the full length of the beam in Tekla Structural Designer.

14. Web opening stiffeners applied only to one side of a beam are always set to the right hand side when importing into FBEAM. When importing from FBEAM to Tekla Structural Designer, the existing stiffeners in Tekla Structural Designer keep their side regardless of how they were modeled in FBEAM.

Export to FBEAM

Before exporting your FABSEC® beams, you should ensure the analysis has been performed.

1. On the **BIM Integration** tab, click **FBEAM Export**.

   The **FBEAM Export** dialog box opens. Any FABSEC® beams in the model are listed in the dialog.
2. Select the beams that you want to transfer to FBEAM, and click **Next**.
3. Select the combinations you want the beams designed for, and click **Next**.
4. Select the location to save to, name the file, and click **Next**.

   The export status of the selected beams is displayed.
5. Click **Finish**.

You can now go in to FBEAM and import the file that you have just created into an existing or new project, so that the beams can be designed.

Import from FBEAM

NOTE In order to import FABSEC® beams, you must have previously created them in your Tekla Structural Designer model, and have exported them to FBEAM. When you do this, each beam is given a unique identifier. After the beams have been designed in FBEAM you must then choose the Tekla Structural Designer XML option to export them to a file. This enables Tekla Structural Designer to know to which beam in your model the imported details apply.

1. On the **BIM Integration** tab, click **FBEAM Import**.

   The **FBEAM Import** dialog box opens. Any FABSEC® beams in the model are listed in the dialog.
2. Select the file that you want to import.
3. Click **Next**

   A list of the FABSEC® beams in the file is displayed from which you can choose the beams to be imported.

4. Select the beams that you want to import.

   You can only select beams to import that correspond to existing beams in the Tekla Structural Designer model.

5. Click **Next**

   The import status of the selected beams is displayed. If any errors have occurred, hovering the cursor over the error displays the reason in a tooltip.

6. Click **Finish**.

   Provided that Tekla Structural Designer recognizes the beams, the details in the file are imported to the associated beam properties, (including in particular the beam UDA properties) in your model.

Review the imported beams

When FABSEC® beams are imported, User Defined Attributes are added to the beam properties to record import status and fire design details. These can be reviewed graphically.

To review the FABSEC® beam UDAs graphically:

1. If necessary, change the view regime (page 89) to a **Review View**.
2. On the **Review** tab, click **UDA**.
3. In the **Properties** window, with **[M]ode** set as **Review**, change the **Attribute** to **Import Status**.

   Each FABSEC® beam is color-coded to represent the import status that applies.

Export a model to ADAPT

Before exporting to ADAPT, read the following limitations. Then, you can proceed to follow the detailed instructions to perform the export.

Limitations

The scope of the ADAPT Export is constrained by the fact that some geometries, configurations and properties possible in Tekla Structural Designer are beyond the scope of ADAPT. This imposes some unavoidable limitations on the Export at this time, for example:

- Only concrete and steel materials are exported.
- Only horizontal beams and slabs are exported.
- Roof and wall panels are not supported.

- For concrete walls, the following are not supported (all of which may be in a Tekla Structural Designer); wall openings, non-rectangular walls, wall extension, walls with top ends not at a level.

NOTE Both loads as well as reactions are linked to their reference member/object in Tekla Structural Designer. This means that any loads or reactions applied to a member that are beyond scope of the export will not be exported either.

Details of what gets exported and what is beyond scope are as follows:

Units	• Units are SI in the export file
Levels	• Number of levels Included • Level height Included
Gridlines	• Beyond scope (Adapt import)
Materials	• Concrete: Normal weight Included • Structural Steel Included • Concrete: Lightweight Beyond scope (Adapt import) • Mild Steel (Rebar) Beyond Scope (Tekla Structural Designer export) • Timber Beyond Scope (Tekla Structural Designer export) • Prestressing Beyond Scope (Tekla Structural Designer export) • Other / generic Beyond Scope (Tekla Structural Designer export)
Load combinations	• All active combinations are included apart from Seismic Inertia which is beyond scope
Loadcases	• All loadcases are included apart from EHF/NHF which is beyond scope
Beams	• Regular and irregular sections are Included • Beam at level Included • Export by span Included • Horizontal offsets Included • Vertical offsets Included • Tekla Structural Designer Labelling Included

	• Beam not at level Beyond Scope (Tekla Structural Designer export) • End trimming at column interface Beyond Scope (Tekla Structural Designer export) • Sloping beams Beyond scope (Adapt import) • Axially rotated beams Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Columns	• Regular and irregular sections are Included • Export by stack Included • Axially rotated columns Included • Tekla Structural Designer labeling Included • Tilted columns Included • Multi-stack columns Included • Columns with holes Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Walls	• Concrete meshed walls Included • Concrete mid-pier walls Included • Export by stack Included • Tekla Structural Designer labeling Included • No internal wall members Included • Multi-stack walls Included • Bearing walls Included (exc. reactions) • Wall alignments Included • Slopping walls Beyond scope (Adapt import) • Wall openings Beyond scope (Adapt import)

	• Wall extensions Beyond Scope (Tekla Structural Designer export) • Non-rectangular walls Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Slabs	• Concrete slab on beams Included • Concrete flat slabs Included • Export by panel Included • Tekla Structural Designer labeling Included • Vertical offsets Included • Slab overhangs Included • Non-concrete slabs Beyond Scope (Tekla Structural Designer export) • Slopping slabs Beyond scope (Adapt import) • Reinforcement Beyond scope (Adapt import)
Drop panels	• Concrete slab drops Included • Trimming at slab edges Included • Reinforcement Beyond scope (Adapt import)
Openings	• Slab rectangular openings Included • Slab circular openings Included
Braces	• Beyond Scope (Tekla Structural Designer Export)
Loading	• Self weight Included • User applied Force Included • User applied Moment Included • Level Point loads Included • Horizontal Line loads Included • Level area loads Included • Offsets from reference level Beyond scope (Adapt import)

	• Line and Area Loads applied to horizontal plane Beyond scope (Adapt import) • Vertical line loads applied as point loads at levels
Releases	• Translational releases Beyond Scope (Tekla Structural Designer Export) • Torsional releases Beyond Scope (Tekla Structural Designer Export)
Stiffness modifiers	• Beyond Scope (Tekla Structural Designer Export)

Instructions

1. On the **BIM Integration** toolbar, click **ADAPT Export**.

   This shows a dialog which gives the name of the ADAPT import file which Tekla Structural Designer will create. You can change the name and location of the file if necessary.
2. Once the file details are correct click **Save** to create the ADAPT import file.
3. Launch ADAPT and import the file to see your project.

Export a model to STAAD

To export a model to STAAD, see the following limitations and instructions.

Limitations

The scope of the STAAD Export is constrained by the limitations of the STAAD Text File (.std) format.

Note that:

- Elastic extensions cannot be defined in STAAD. Therefore, additional 1D elements are created to preserve model connectivity.
- The exported file will not group elements in any way. 1D elements will not be grouped as members (or columns, beams, etc). In addition, there will not be any grouping of faceted elements from a curved member. 2D elements will not be grouped into panels.
- Section data is exported as analysis properties only. This means that no attempt is made to reference STAAD library sections.
- Material data is exported as analysis properties only. This means that no attempt is made to reference STAAD library materials.

- Objects such as Shear Only Walls have no equivalent native object in STAAD and some of the internal data such as panel geometry and spring properties end up lost during conversion.

Instructions

1. Create the model as usual.
2. On the **BIM Integration** tab, click **STAAD Export**.

   The **Save As** dialog opens.
3. If necessary, change the name and location of the file.
4. Click **Save**.
5. Start STAAD and open the file.

Export a model to Autodesk Robot Structural Analysis

Before you export a Tekla Structural Designer model to Autodesk Robot Structural Analysis, read the limitations of Robot Export in the following paragraphs. Once you have read the limitations, you can proceed to the detailed instructions on how to export models from Tekla Structural Designer to Autodesk Robot Structural Analysis.

Limitations

The scope of the Robot Export is constrained by the limitations of the Robot Text File (STR) Format.

Note that:

- The model exported is the one used for 1st order linear static analysis (see <Linear analysis of structures containing material non-linearity> to remove non-linearity from the exported model.)
- 1D element springs are totally omitted from the export.
- Elastic extensions cannot be defined in Robot. Therefore, additional 1D elements are created to preserve model connectivity.
- The exported file will not group elements in any way. 1D elements will not be grouped as members (or columns, beams, etc). In addition, there will not be any grouping of faceted elements from a curved member. 2D elements will not be grouped into panels.
- Section data is exported as analysis properties only. This means that no attempt is made to reference Robot library sections.
- Material data is exported as analysis properties only. This means that no attempt is made to reference Robot library materials.
- Loads in projection are converted to equivalent loads in Robot.

- Robot will not import the material properties for timber from the STR file. When the file is opened in Robot, you can delete the G value and adjust the values manually after import.
- It is not possible to define part-length distributed torsional loads in the STR file. Therefore, they are converted to equivalent full-length distributed torsional loads.

Instructions

1. Create the model as usual.
2. On the **BIM Integration** tab, click **Robot Export**.

   The **Save As** dialog opens.
3. If necessary, change the name and location of the file.
4. Click **Save**.
5. Start Robot and open the file.

Export a model to the cloud

To export a Tekla Structural Designer model to the cloud, see the following instructions.

1. Create the model as usual.
2. On the **BIM Integration** tab, click **Cloud Export**.

   The **Save As** dialog opens.
3. If necessary, change the file name and location.
4. Click **Save**.

Export to One Click LCA

One Click LCA is third party software for performing a life cycle assessment of your structure and optimizing its embodied carbon.

From Tekla Structural Designer you can either compile an *offline* report, which is a compact summary of materials in the structure that can be edited if required before being manually imported into One Click LCA, or you can upload the report data directly to One Click LCA *online*.

Overview

Scope

Tekla Structural Designer includes material information in tabular data tables, in general the scope of the exported data matches this. This gives a lot of the most commonly requested material quantity information, but is not considered to be exhaustive:

- Data available in Tekla Structural Designer but not exported
  - Shear studs on composite beams
- Data NOT available in Tekla Structural Designer (and therefore not exported)
  - Reinforcement in any slab other than cast-in-place reinforced concrete slabs
  - Bearing Walls
  - Shear Only Walls
  - Transverse shear reinforcement required by composite beams
  - surface area of timber members, cold rolled members, general material members
  - DELTABEAM® mass - a zero value is exported to help ensure allowance is added manually
  - analysis elements
- Exported Data not otherwise available in Tekla Structural Designer
  - Precast slab screed/topping area
  - Composite slab decking type / area
  - material in gable and parapet posts
  - surface area of defined “roof panels” and “wall panels”

The above summary is not guaranteed to be exhaustive, it is important to consider “what is missing”? Are all the significant physical items you defined in Tekla Structural Designer being covered by the export?

Additional information

The following notes may assist with initial use of the export:

- Grouping in One Click LCA

  On import there is a phase where it is suggested that lots of the exported data items can be grouped together on the basis of a shared “IFCMATERIAL”. The sections below give examples where you may consider the value of doing this:

    - composite / non-composite beams

      the mass of these beam types are exported separately.

      you may wish to apply higher carbon factors to composite beams to allow for shear studs and transverse shear reinforcement.

    - steel beams of all different fabrication types

      the mass of steel members of different fabrication types are exported separately.

      e.g. rolled, plated, Westok, FABSEC®, DELTABEAM®

      you may wish to apply different carbon factors to these different types of member

    - surface areas

      the surface areas of slabs and many beams are all exported separately.

      if you have no requirement to use any surface area information then it will be easier to leave these grouped for simpler deletion.

- Slabs
    - The data for each slab is exported separately (a slab may contain many slab-items).

      In simple situations this will mean one group of data for each level, but if multiple slabs are defined in a level you will get separate data for each.

    - Data for different types of slab is also kept separate

      e.g. Cast-in-place reinforced slabs, post-tensioned slabs, precast slabs, composite slabs, various other general material slabs

      For reinforced concrete slabs this allows a more detailed understanding of reinforcement per slab.

    - Limitation - as noted above reinforcement data is not available/ exported for other slab types, this will need to be allowed for in one of two ways:
        - Use the surface area information to add an allowance per m2
        - Adjust the carbon factor or concrete/topping in the slabs to allow for reinforcement content

      In either case this will mean avoiding grouping of the relevant data items.

- “roof panels” and “wall panels” surface areas
    - Roof and wall surface area data is included in case it assists with the inclusion of cladding materials.

Feedback

- Please send us feedback if you feel that:
  - Important data is missing
  - Included data should be totalled/presented differently - perhaps there should be options to do things in different ways (possibly at different stages of design?)
  - There should be options to filter out certain data types because you don't always want to use them (for example surface area info)

Show report

1. On the **BIM Integration** tab, click **One Click LCA**.

   The **One Click LCAReport** dialog opens.
2. Click **Show report** The report opens in Excel, (or whatever software is associated with .xlsx files).

The resulting materials file can then be edited if required before being manually imported into **One Click LCA**, or it can be passed on to a third party and imported into **One Click LCA**.

Show online results

NOTE	Online export requires a One Click LCA license.

1. On the **BIM Integration** tab, click **One Click LCA**.

   The **One Click LCAReport** dialog opens.
2. Enter your One Click LCA user name and password, then click **Log in**.
3. Click **Show online results**. The material data from the Tekla Structural Designer model is uploaded to One Click LCA.
4. Proceed through the One Click LCA wizard:
   a. Create/Choose project
   b. Consider groupings
   c. Review / apply mapping

   On completion the amount of embodied carbon in the model is determined.

Export to IDEA StatiCa Connection Design

Before exporting connections to IDEA StatiCa, you are advised to be aware of the limitations. Then, you can proceed to follow the detailed instructions to perform the export.

NOTE Only valid connections listed in the Project Workspace Connections (page 83) tree can be exported.

Limitations

NOTE The link is compatible with the latest versions of IDEA StatiCa 9.1 and 10.0; it is also compatible with IDEA StatiCa 10.1 (build 10.1.113), but may not work with earlier builds of 10.1.

The following geometrical limitations should be noted.

- Connection checks to BS 5950 are not supported.
- For connections identified as 'Moment Connections' using Update connections in Tekla Structural Designer any out of plane members are not exported.
- For any connections identified in Tekla Structural Designer the brace members are not passed through to IDEA.
- For any connection type, haunches are not exported.
- Tekla Structural Designer uses ToS and the wire model is at this level too. The export to IDEA depicts the ToS so that the 3D graphic looks correct. However, the wire model typically connects at offsets of half the beam depth. This introduces additional forces/moments due to eccentricity of line of action etc. This line of action of the force set can be adjusted in IDEA.

Instructions for the export to IDEA StatiCa

NOTE Export requires an IDEA StatiCa licence.

1. Right-click the connection in a Scene View.
2. Choose Export Connection to IDEA StatiCa from the right-click menu, selecting the analysis results to use at the same time.
3. Specify the file name and location for this connection.

   The IDEA StatiCa application opens to allow the connection to be designed, the following data having been transferred:

   - Connection geometry
   - Section profile & material grade

- Connection forces for each active solved Tekla Structural Designer combination.

NOTE The Tekla Structural Designer model remains frozen until you close the IDEA StatiCa connection.

4. Add bolts, stiffeners etc as required and design the connection.

When you close the connection Tekla Structural Designer becomes active once more; the connection file is automatically embedded in the model.

Review of IDEA connections designed in Tekla Structural Designer

1. Within Tekla Structural Designer click Review>UDA
2. In the Properties Window set Mode to Review and Attribute to IDEA StatiCa file

   Joints associated with IDEA StatiCa are shown
3. Use the right-click context menu to open previously exported connections back to IDEA StatiCa.

Related video

Integration with IDEA StatiCa Connection design

4 Create models

To get started with modeling in Tekla Structural Designer see:

- Get to know Tekla Structural Designer basic working methods (page 155), for a few first principles
- Create the model (page 174), to learn how to create and modify model objects

Once you are comfortable creating objects, see:

- Edit the model (page 321), to get familiar with the model editing commands
- Check the model (page 338), for validation and measuring commands
- BIM integration (page 108), to learn how to exchange model data between applications

See also

Create and design foundations (page 688)

BIM integration (page 108)

4.1 Get to know Tekla Structural Designer basic working methods

Before creating real models, get to know some basic methods and techniques that will help you to work efficiently.

We recommend you familiarize yourself with how to:

- work with grids and construction lines (page 188)
- define construction levels (page 175)
- zoom and rotate (page 156) the model
- create (page 174), select (page 158) and edit the properties of (page 168) entities

- re-position entities (page 169) by moving nodes or edges
- copy, move and mirror (page 321) part of, or all of the model

Zoom, pan, rotate and walk through the model

You can use the mouse to manually zoom, pan, or rotate the model. Additionally, you can rotate the model by using the **ViewCube**, and walk through your model in 3D views.

Zoom in and out, or zoom extents

- According to your needs, do one of the following:

To	Do this
Zoom in or out	• Scroll in or out with the middle mouse button.
Zoom to the center of the visible objects	• Right-click anywhere within the view. • In the context menu, select **Zoom Out**. **NOTE** You can also use the keyboard shorcut **ZA** to zoom extents.

Pan the view

- Hold down the middle mouse button and drag the view according to your needs.

Rotate the view manually

You can use the **ViewCube** to adjust the view. However, if none of the standard **ViewCube** are appropriate, you can also rotate the model manually.

- Hold down the right mouse button and drag the view according to your needs.

Adjust the view with the ViewCube

1. Move the mouse pointer over the **ViewCube**.
2. According to your needs, do one of the following:

To	Do this
Display one of the eight isometric views	• Click the required vertex on the **ViewCube**.

	 **TIP** If the required vertex is not visible, click one of the other vertices to spin the **ViewCube** until you can see the required vertex.
Display one of the twelve edge views	• Click the required edge on the **ViewCube**.  **TIP** If the required edge is not visible, spin the **ViewCube** by clicking the vertex that is adjacent to the required edge.
Display one of the six face views	• Click the required face on the **ViewCube**.  **TIP** If the required face is not visible, spin the **ViewCube** by clicking the vertex adjacent to the required face. **NOTE** When a face view is displayed, the additional **ViewCube** controls appear:  • If the required face is not visible in a face view, roll the **ViewCube** by

	clicking one of the triangular controls.

Walk through the model in a 3D view

- On the **Home** tab, click **Walk**.

NOTE In the **Walk** mode:

- To move forward, backward, left or right, use the arrow keys.
- To move up and down, use the **Q** and **Z** keys.
- To rotate the view, hold down the right mouse button and drag the view.
- To exit **Walk** mode, press **Esc**.

Display a 2D view in 3D

1. In a 2D view, click the **2D/3D** toggle button, located on the bottom right corner of the window.

Select entities

Many Tekla Structural Designer commands require you to make selections. You can make single and area selections from any 2D or 3D view. You can also make selections from the **Project Workspace**.

TIP Use (page 64) to display everything *not* in the selection as ghosted, this makes it easier to focus on the selection.

TIP If planar objects, such as slabs, roofs and area loads, obscure what you want to select, you can hide them using the (page 68) window.

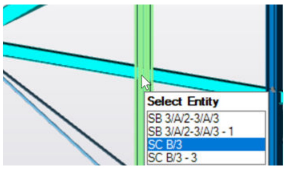
Use the different methods listed below to select the entities.

Select single entities

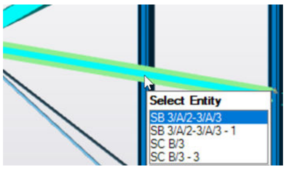
1. In a 2D or 3D view, move the mouse pointer over the desired entity.

   If the entity is the only one at the location, it becomes highlighted and its name is shown in the **Select Entity** tooltip.

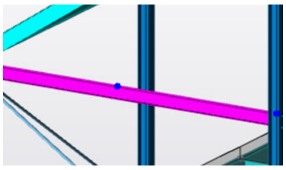
   If several entities are located at the same location, their names are all listed in the **Select Entity** tooltip.



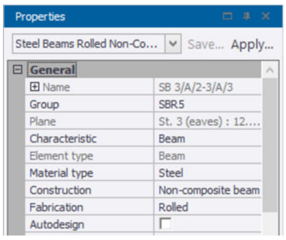
Use the up and down keys to scroll through the list to reach the desired entity.



2. When the desired entity is highlighted in the **Select Entity** tooltip, press **Enter** to select it.



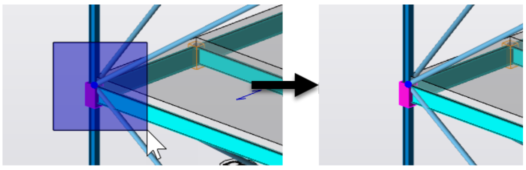
The selected entity is then displayed in the *active selection color*, the nodes for positioning it are exposed, and the entity properties are displayed in the **Properties** window.



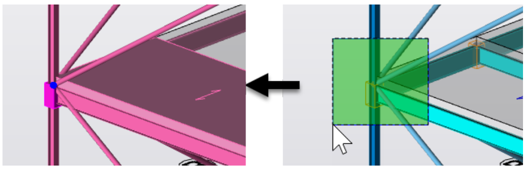
Select multiple entities using area selection

1. You can select multiple objects using area selection. The dragging direction affects the selection of objects.

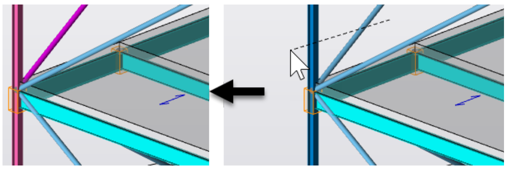
    a. To select all entities that are completely within a rectangular area, hold down the left mouse button and drag the mouse from **left to right**.



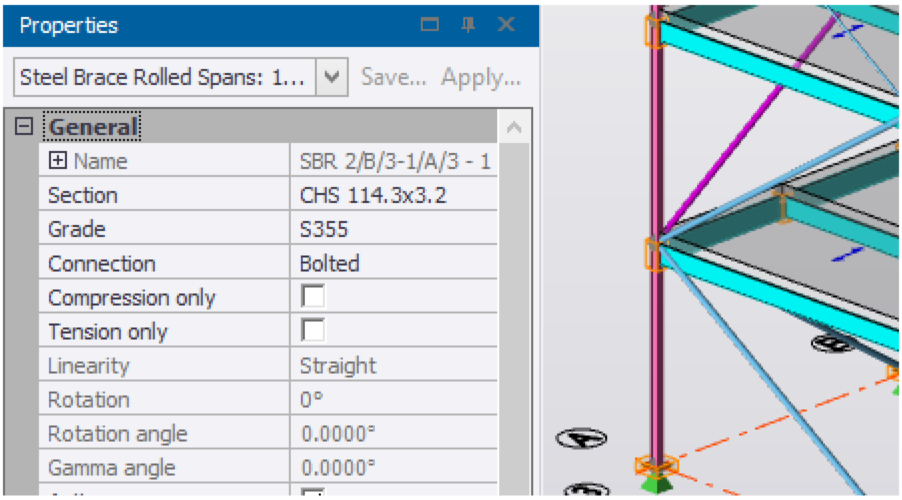
b. To select all entities that are at least partly within a rectangular area, hold down the left mouse button and drag the mouse from **right to left**.



c. To select all objects that are at least partly crossed by a straight line, hold down the **shift key**, then hold down the left mouse button and drag the mouse to draw the line.



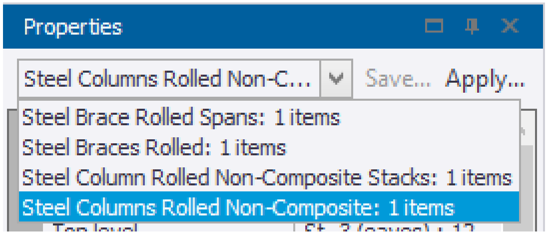
After selection, one entity type is displayed in the *active selection color* and has its properties shown in the **Properties** window.

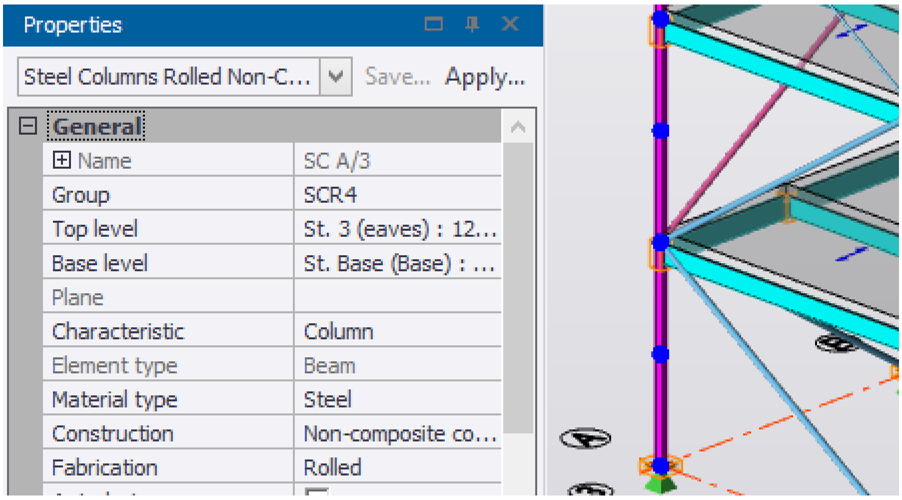


Any other entity types in the selection are displayed in the *local selection color*.

NOTE You can change the *active* and *local* selection colors by clicking **Home > Settings > Scene > Colors > Selection**.

2. To display the properties of another entity type in the selection, choose it from the droplist at the top of the **Properties** window.





NOTE If there is only a single item in the active selection, all its properties are shown; if there are multiple items in the active selection, only common properties are shown.

Select using Find

1. Click **Find** on either the **Home** tab, or the **Quick Access** toolbar.

   The **Find** dialog opens.

2. In the **Find** box type any of the following, in whole, or in part:

   a. An object 'Type'

   b. An object 'Name'

   c. An object 'Index'

   As you type, any objects that match the criteria are listed in the table below.

3. When the object(s) that you require are shown, click to highlight them as follows:
   a. Single click to highlight a single object
   b. Ctrl+click to select multiple non-sequential rows
   c. Shift+click to select multiple sequential rows Once the object(s) are highlighted, **Select** becomes available.
4. Click **Select** to locate the object in the active view, zoom in, and display its properties in the **Properties** window.

NOTE If the object doesn't exist in the active view a new view containing the object will become active.

TIP Sometimes when clicking **Select** the object being located is obscured from view by other objects. By leaving (page 64) checked this is avoided, as all objects other than those being found are made semi-transparent.

NOTE If not using Ghost Unselected, you can change the color used to highlight the found entity by clicking **Home > Settings > Scene > Colors > Selection** and changing the **Global - User** color.

Select from the Project Workspace

1. Do one of the following:

To	Do this
Select individual members from the **Structure** tree	a. Expand the **Members** branch and any appropriate sub branches until you can click the required member. b. Right-click the member name. c. In the context menu, select **Select in visible views**. The member's properties are viewed in the **Properties** window.
Select member groups or individual members from the **Groups** tree	a. Expand the appropriate branches until you can click the required group or member name. b. Right-click the member or group name. c. In the context menu, select **Select in visible views**. The member's or group's properties are viewed in the **Properties** window.

Select nodes

Sometimes you need to select only the nodes of an entity, for example when moving a beam.

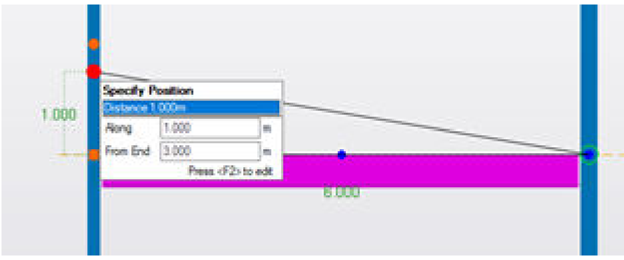
1. In a 2D or 3D view, select the entity.

   The entity is selected and any nodes associated with the entity are displayed.

2. Move the mouse pointer over the desired node.

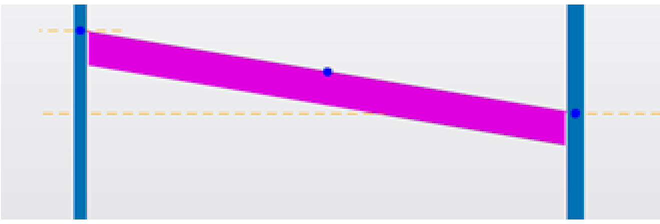
   The node becomes highlighted and appears in the **Select Entity** tooltip.

3. Press **Enter**.



   You will now be prompted to pick a point to move the node to.

4. Click at a new location to reposition the node.



The entity is updated accordingly.

See also: Re-position entities by moving nodes or edges (page 169)

Modify the selection

You can add or remove entities from the current selection.

1. Do one of the following:

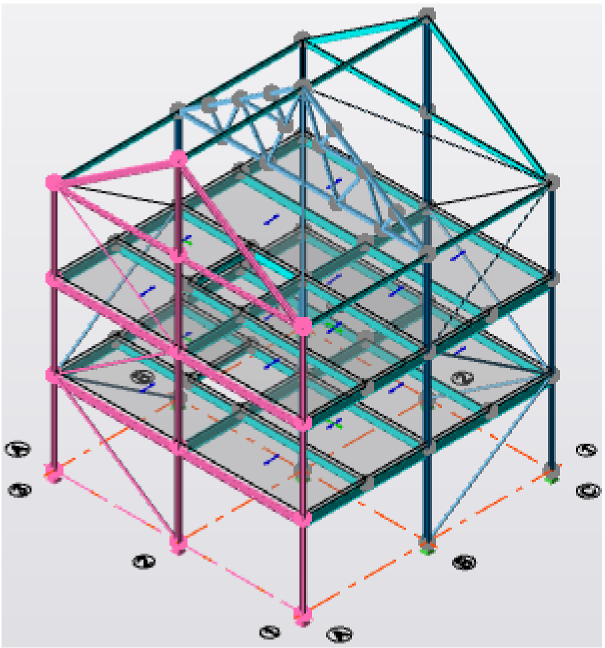
To	Do this
Add an entity to the current selection	a. Hold down the **Ctrl** key. b. Click the entity that you want to add to the selection. The properties of the newly selected entity are added to the **Properties** window.
Remove an entity from the current selection	• Click the entity you want to de-select. The entity's properties are removed from the **Properties** window.
Remove multiple entities	a. Do one of the following: • To de-select only the entities that are encompassed by the box, move the mouse pointer to the left corner of the imaginary box that will encompass the entities. • To de-select the entities that are encompassed by the box and that it crosses, move the mouse pointer to the right corner of the imaginary box that will encompass the entities. b. Hold down the **Ctrl** key and drag the mouse pointer to the opposite corner of the box. A rectangle on the screen shows the area that you are selecting. c. Release the mouse button and the **Ctrl** key. The entities are removed from the **Properties** window.
Remove a beam/column/wall and simultaneously remove its constituent spans/stacks/panels	If a continuous beam and its constituent spans are selected, they can all be deselected with a

	single mouse-click while holding down the **Ctrl** key. The same method can be applied to a continuous column and its constituent stacks, or a wall and its constituent panels.
Remove all entities	• Press **Esc**. The information displayed in the **Properties** window is cleared.

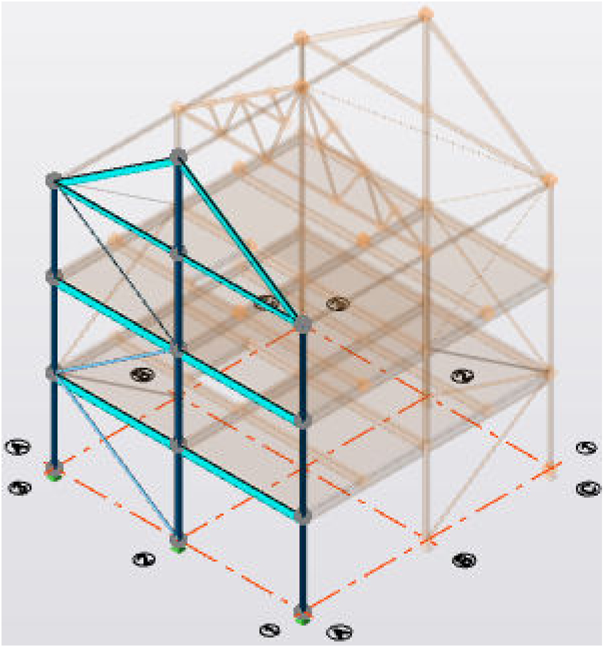
Use Ghost Unselected to focus on the selection

You can toggle **Ghost Unselected** on and off using the  button in the bottom corner of the view.

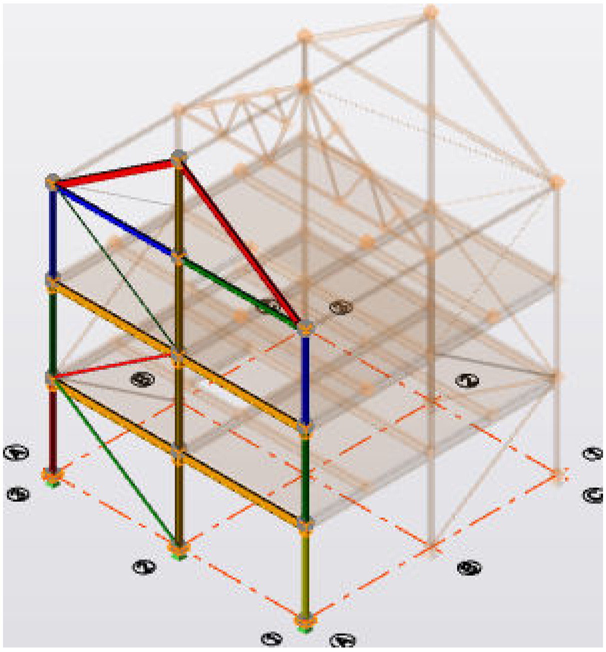
- **Ghost Unselected off**: selected entities are displayed in the *active selection color*, this can make it hard to distinguish between individual entities in the selection.



- **Ghost Unselected on**: selected entities are displayed in their nomal colors (appropriate to the Scene View that is active), unselected objects are semi-transparent. If nothing is selected, everything is displayed in its normal color.
    - In a Structure View, the selection would be displayed as:



- In a Review View showing Design Status, the selection would be displayed as:



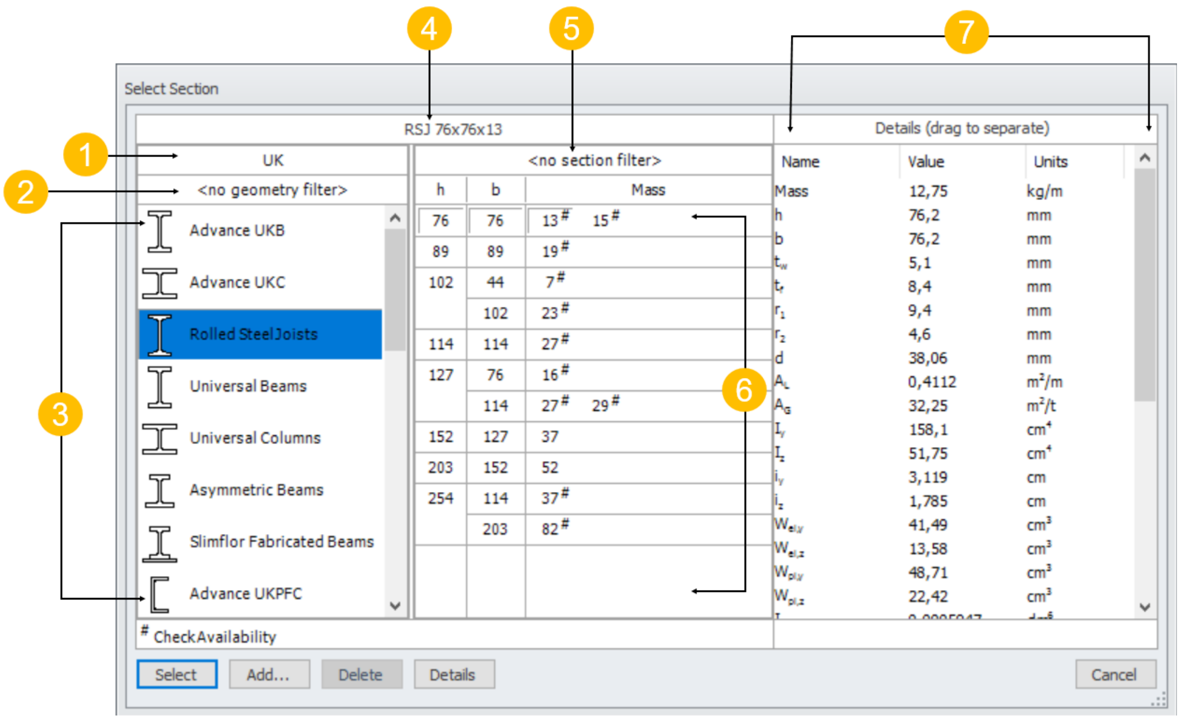
Select a section in the Select Section dialog box

Select Section dialog box allows you to select sections to use them in your model. You can also add and delete user-defined sections in the database by using the dialog, if necessary.

To access the **Select Section** dialog box:

1. On the **Model** tab, select the member you want to design.
2. In the , click the arrow on the right side of the **Section** field.
3. Click **<New\Edit...>**.

   The **Select Section** dialog box opens.



1. **Country** list
2. **Geometry filter**
3. **Page** pane
4. Selected section
5. **Section filter**
6. **Item** pane
7. **Details** pane

1. In the **Country** list, select the desired country.
2. In the **Page** pane on the left, select the desired geometry.

NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.

3. In the **Item** pane on the right, select the desired section size.
4. Click **Select**.

Edit entity properties

Any selection of one or more entities can be edited using the **Properties** window. Individual **model objects** can also have their properties edited using the **Properties** dialog box.

Edit properties using the Properties window

All entities can have their properties edited using the **Properties** window

1. Select the entity either in the graphical display or in the **Structure** tree.

   For more information, see Selection methods (page 158).
2. In the **Properties** window, modify the properties according to your needs.

Edit properties using the Properties dialog box

Individual **model objects** can have their properties edited using the **Properties** dialog box

1. Hover the mouse pointer over the object that you want to edit.
2. In the **Select Entity** tooltip, select the object.

   For more information, see Selection methods (page 158).
3. Right-click the object.
4. In the context menu, select **Edit**.

   The **Properties** dialog box opens.
5. Modify the properties according to your needs.
6. Click **OK**.

Edit properties of multiple entities

NOTE You can view and modify the common properties of multiple entities in the **Properties** window.

Note that:

- If the value of a certain item is not identical for all the selected entities, the cell appears blank in the **Properties** window.
- If you modify the blank cell, Tekla Structural Designer applies the new setting to all the selected entities.
- If you have selected multiple entities of different types, the property information is displayed separately for each type. Use the list at the top of the **Properties** window for moving between types.

1. Select the entities in the graphical display.

   For more information, see Selection methods (page 158).
2. If you have selected entities of different types, select the desired type in the list on top of the **Properties** window.
3. In the **Properties** window, modify the properties according to your needs.
4. Repeat steps 2–3 for entities of different types.

Re-position entities by moving nodes or edges

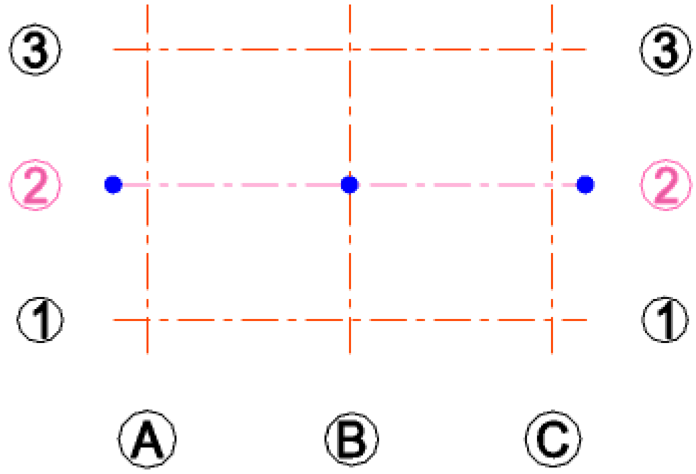
After selecting an entity you can re-position it by moving nodes or edges.

Modify one end of a grid or construction line

Grid and construction lines can only be modified in 2D views.

1. Select the grid or construction line.

The nodes at each end and at the mid-point become visible.



2. Select the end node that you want to move.

NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click where you want to move the end node.

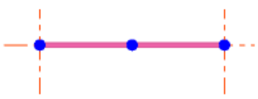
   Tekla Structural Designer redraws the line, moving the end node to the selected point.

Move a grid or construction line

Grid and construction lines can only be moved in 2D views.

1. Select the grid or construction line.

   The nodes at each end and at the mid-point become visible.



2. Select the end node that you want to move.

NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click where you want to move the end node.

   Tekla Structural Designer redraws the line, moving the end node to the selected point.

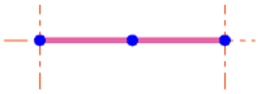
Modify one end of a member

To move an entire member, see: Move objects (page 322).

To move just one end of a member, proceed as follows:

1. Select the member in a 2D or 3D view.

   The nodes at each end and at the mid-point become visible.

2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click a grid or construction point where you want to move the end node.

   Tekla Structural Designer redraws the member, moving the end node to the selected point.

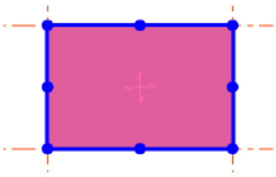
Modify slab items and panels by moving a node

To move entire slab items and panels, see: Move objects (page 322).

To move a slab/panel node, proceed as follows:

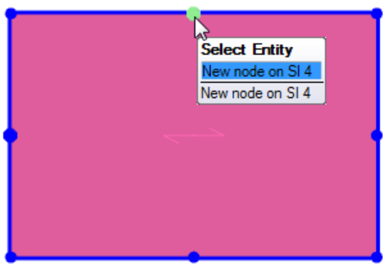
1. Select the slab item or panel that you want to modify in a 2D or 3D view.

   The nodes at each vertex and at the mid-point of each edge become visible.

2. Click a node to select it.

   You can select either corner nodes, or nodes at the mid-points of the edge.

3. Move the mouse pointer over an existing construction point or intersection.
4. Click the point to reposition the node.

NOTE When you reposition nodes, note that:

- The new node position must be in the same plane as the slab item or panel.
- When you move a node at the mid-point of an edge, it becomes a new corner node. Tekla Structural Designer automatically creates two new mid-point nodes.
- When you move a corner node directly over an adjacent corner node, Tekla Structural Designer deletes the original corner node.

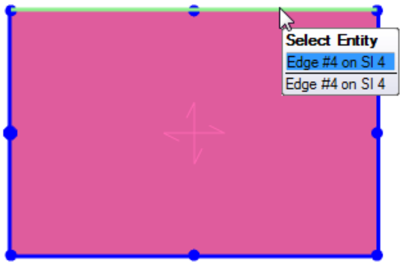
Modify slab items by moving an edge

To move entire slab items and panels, see: Move objects (page 322).

To move a slab/panel edge, proceed as follows:

1. Select the slab item that you want to modify in a 2D or 3D view.
2. Click an edge to select it.

   Ensure that you have selected an edge, and not a node.



3. Move the mouse pointer over an existing construction point or intersection.
4. Click to reposition the edge so that it passes through the selected point.

NOTE The new edge position must be in the same plane as the slab item.

Modify walls by moving a node

To move an entire wall, see: Move objects (page 322).

To move a wall node, proceed as follows:

1. Select the wall that you want to modify in a 2D or 3D view.

   The nodes at each vertex become visible.

2. Click a node to select it. If you want the result of moving the node to be a rectangular wall, the **Rectangular** property in **Properties** window should be selected, otherwise it should be unselected.
3. Move the mouse pointer over an existing construction point or intersection.
4. Click the point to reposition the node.

NOTE The new node position must be in the same plane as the wall.

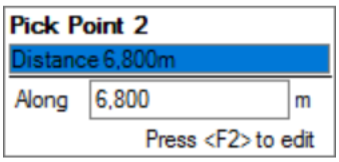
If the **Rectangular** property was checked, both the selected node and one other will be moved as required to form a rectangular wall.

Tips for basic tasks

Here we provide useful hints and tips that help you use the Tekla Structural Designer user interface and its basic features more efficiently.

Use the tooltip for input in a command

When you need to specify a distance or pick a point, a tooltip appears, displaying the current value.



1. Do one the following:
   - Move the mouse pointer to the required point and click the point to use it.
   - Press **F2** to type an exact number, and press **Enter**.

Undo a command

1. In the **Quick Access** toolbar, click **Undo**.

Cancel a command or go back to the previous prompt

Yellow command prompts are viewed on the top of the active view to guide you through each command. To react to command prompts, see the following instructions.

- Do one of the following:

To	Do this
Cancel a command	• Press **Esc** at the first command prompt.
Go back to the previous command prompt	• Press **Esc** at a subsequent command prompt.

TIP Selecting another command also cancels the current command.

Example

The parallel grid line command consists of four steps. Hence, four consecutive command prompts are displayed.

1. Add Parallel Grid/Construction Line: Select Reference Line <press ESC to cancel>
2. Add Parallel Grid/Construction Line: Specify Distance **<press ESC to cancel>**
3. Add Parallel Grid/Construction Line: Pick Point 1 **<press ESC to cancel>**
4. Add Parallel Grid/Construction Line: Pick Point 2 **<press ESC to cancel>**

The grid line is created after the fourth prompt. After that, prompt 2 is automatically redisplayed.

At this point, you can either continue to create additional lines parallel to the original, or go back to the first prompt by pressing **Esc**.

Then, you can either select a new reference line and continue again, or cancel the command by pressing **Esc** again.

4.2 Create the model

When you model in Tekla Structural Designer, you create and work with different types of model object. In most cases, a model object represents a member type that will exist in the real building or structure, or that will be closely related to it. You will also work with modeling aids such as grids and construction lines, that represent information that is only relevant when you are creating the model.

You can use the following modeling aids:

- Construction levels (page 175)

- Grids (page 178), construction lines (page 188) and free points
- Frames and slopes
- Dimensions (page 196)
- DXF reference drawings (page 186)

You can create the following physical member/object types:

- Beams, columns and braces (page 196)
- Walls and cores (page 234)
- Slabs and decks (page 258)
- Trusses and steel joists
- Portal frames
- Cold rolled members

The following panels/objects are useful for applying loads to the model:

- Wall panels and roof panels
- Ancillaries (page 285)
- Equipment (page 295)
- Inactive members (page 306)

You can also use the following general members/objects when required:

- Supports (page 314)
- Analysis elements (page 317)

Create and manage construction levels

In the **Construction Levels** dialog box, you can define and modify the levels that you need to construct your model. The levels can be floors, roofs, or intermediate levels required to define specific items.

Open the Construction Levels dialog

- Do one of the following:
    - On the **Model** tab, click **Construction Levels**.
    - In the **Structure** tree, double-click **Levels**.

Insert a single construction level

1. In the **Construction Levels** dialog box, select an existing level.

2. According to your needs, click either **Insert Above** or **Insert Below**.
3. If necessary, change the level type:
   - If you are creating a steel structure, set each new level as **T.O.S.** (top of steel).
   - If you are creating either a concrete or mixed material structure, set each new level as **S.S.L.** (structural slab level).
4. If necessary, change the level name.
5. For the new level, do one of the following:
   - Specify the height above the base in the **Level** field.

     The inter-story spacing is calculated automatically.
   - Specify the inter-story spacing in the **Spacing** field.

     The level is calculated automatically.

   NOTE A default height is calculated for the new level based on the spacings of any existing levels immediately above or below it. Either accept the height, or adjust it as required.
6. If you want Tekla Structural Designer to treat the level as a floor, select the **Floor** option.

Insert multiple construction levels

Multiple levels can be inserted, either above the current top level, or below the current bottom level.

1. In the **Construction Levels** dialog box, select an existing level.
2. According to your needs, click **New on Top** or **New at the Bottom**
3. For each construction level, type the level spacing and click **OK**.

   TIP If you have new levels at 12' 6" , 18' 6" , 27' 6" , 36' 6" and 27" above the current level, you can specify the levels as 12' 6" , 18' 6" , 3*9'.
4. If necessary, change the level names.
5. If necessary, change the level type:
   - If you are creating a steel structure, set each new level as **T.O.S.** (top of steel).
   - If you are creating either a concrete or mixed material structure, set each new level as **S.S.L.** (structural slab level).

Make a level an identical copy of another level

If needed, you can determine a level as an identical copy of another level. In this case, all changes made to either the source level or the identical level are automatically applied to both levels. To make a level an identical copy of another level, see the following instructions.

1. Open the **Construction Levels** dialog box.
2. At the level you want to be a copy, click the arrow in the **Source** column.
3. Select the level that you want the current level to be identical to.
4. Click **OK**.

Make a level an independent copy of another level

If you determine a level as an independent copy of another level, any changes made to each level only apply to that level. To make a level an independent copy of another level, see the following instructions.

1. Open the **Construction Levels** dialog box.
2. At the level you want to be a copy, click the arrow in the **Source** column.
3. Select the level that you want the current level to be identical to.
4. Click **OK**.
5. Reopen the **Construction Levels** dialog box.
6. At the same level, click the arrow in the **Source** column again.
7. Set the level as **-unique-**.
8. Click **OK**.

Modify the properties of a construction level

You can modify some properties of construction levels in the **Construction Levels** dialog box dialogs, but other parameters can only be edited in the **Properties** window. For more information, see the following instructions.

1. In the **Structure** tree, click the **+** sign next to  **Levels**.
2. Click the name of the construction level that you want to modify.

   The properties of the level are viewed in the **Properties** window.
3. In the **Properties** window, modify the level properties according to your needs.

Delete construction levels

To delete construction levels, see the following instructions.

WARNING Deleting a construction level completely deletes the entire level and all its associated details, such as beams, members, or slab. You cannot recover the level once you have deleted it.

1. In the **Construction Levels** dialog box, select the level that you want to remove.
2. Click **Delete**.
3. Click **OK**.

> **NOTE** You can also delete construction levels using the **Structure** tree. Note that :
>
> a. Click the **+** sign next to **Levels**.
> b. Right-click the level that you want to remove.
> c. In the context menu, select **Delete**.

Create and manage architectural grids and grid lines

Architectural grids

An architectural grid is a collection of grid lines that lie in a horizontal plane. Each architectural grid is only displayed in the 3D View and at the lowest level in the structure. Vertical lines can be projected from each grid intersection of the architectural grid. The vertical lines extend to the topmost level at which the architectural grid has been applied.

You can assign a color for each architectural grid, and set to display grid line names and vertical lines in the architectural grid properties.

NOTE Even when architectural grid properties are checked, they are only displayed provided that the **Architectural Grids** --> **Text2D** and **Architectural Grids** --> **VerticalLines** options are selected in **Scene Content**.

Grid lines

Each grid line is associated with an architectural grid, and is only created at a given level, provided the architectural grid has been applied at that level.

Grid lines that have been created at a given level can optionally be shown or not shown in both the 2D and 3D views using the level properties.

NOTE Even when the show/hide options are checked the lines are only displayed provided that the relevant **Grid & Construction Lines** options are also checked in **Scene Content**.

TIP Even when you have activated the display of grid & construction lines in **Scene Content** for a particular 2D or 3D view, you may still find that the grid lines are not displayed on particular levels.

In this situation, do the following:

1. In the **Structure** tree, open the **Levels** branch.
2. Click the level at which you want to view the grids.

   The level properties are viewed in the **Properties** window .
3. In the **Properties** window:
   - To make grids visible in a 2D view, select the **Show grids in plane view** option.
   - To make grids visible in a 3D view, select the **Show grids in the 3D view** option.

If the grids are still not visible in the 3D view at certain levels, ensure that grids are applied at the levels in question. For further information, see Apply an architectural grid to a specific level (page 185).

Create grid lines

Tekla Structural Designer allows you to create various types of grid lines according to your needs. For detailed instructions, see the following paragraphs.

Create a single grid line

NOTE If you want to create a series of grid lines which form a regular or irregular, rectangular or radial grid line system, the **Rectangular Wizard** and **Sector Wizard** are the fastest options.

1. Ensure that the 2D view containing the construction level on which you want to create your grid line is active.
2. Go to the **Model** tab.
3. In the **Grid Line** list, select **Grid Line**.
4. In the **Properties** window, review the default next vertical/horizontal grid name and edit if required.
5. Select the point where you want the grid line to start.

TIP The tooltip views the mouse pointer's exact coordinates. If the mouse pointer has not snapped to the required point, press **F2** to type the exact coordinates of the required point.

6. Select the point where you want the grid line to end.

TIP The tooltip views either the absolute, relative, or polar coordinates of the end point, depending on whether the **ABS**, **REL**, or **POL** button is selected in the (page 67).

- To switch the display, click one of the other buttons.

NOTE The grid line does not extend to infinity. Do ensure that the grid line is sufficiently long to meet your needs.

Tekla Structural Designer creates the grid line between the selected points.

Create parallel grid lines

This option creates a grid line parallel to an existing one, but of a different length. To use this option, you must have at least one existing grid line in the current 2D view.

1. Go to the **Model** tab.
2. In the **Grid Line** list, select **Parallel**.
3. In the **Properties** window, type a name for the grid line.
4. Select the grid line to which your new grid lines will be parallel.
5. Move the mouse pointer to the distance where you want to place the parallel grid line.

TIP The tooltip views the distance of the dotted line from the initial grid line you selected in step 3. Press **F2** to type the exact distance.

For more accuracy, you can zoom in.

6. Click to locate the new grid line.
7. Click the points where you want the new grid line to start and end.

   The construction line is now created.
8. Move the mouse pointer and place other grid lines with respect to the line you selected in step 3, or press **Esc** to end grid placement.

TIP To create parallel grid lines of the same length as the selected gridline, select the **Parallel (quick)** command.

When you create quick parallel grid lines, you can press **F2** to define the distance between successive pairs of grid lines, separating the numbers with commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

For example, you can specify bay centers of 18', 27' , 18' 6" , 18' 6" , 18' 6" and 27" as `18' , 27' , 3x18' 6" , 27"`.

Create perpendicular grid lines

To use this option, you must have at least one existing grid line in the current 2D view.

1. Go to the **Model** tab.
2. In the **Grid Line** list, select **Perpendicular**.
3. In the **Properties** window, type a name for the grid line.
4. Select the grid line to which your new grid line will be perpendicular.
5. Move the new grid line to the desired position and click to locate it.

TIP The tooltip views the perpendicular distance of the dotted line from the middle of the grid line you selected. Press **F2** to type the exact distance.

6. Click the points where you want the new grid line to start and end.
7. Move the mouse pointer and place other grid lines with respect to the line you selected in step 4, or press **Esc** to end grid placement.

Create a rectangular grid line system

1. Ensure that the 2D view representing the construction level on which you want to create your rectangular grid line system is active.
2. On the **Model** tab, in the **Grid Line** list, select **Rectangular Wizard**.

   The **Rectangular Grid Wizard** opens.
3. Type a name for the grid and select a color for the grid lines.
4. Ensure that each construction level on which you want to create the grid layout is selected.
5. Click **Next**.

6. Define the origin of the grid line system either by clicking it in the 2D view, or by entering its coordinates in the **Rectangular Grid Wizard**.
7. Click **Next**.
8. Select if you want to create grid lines in the X direction only, in the Y direction only, or in both directions.
9. Select the style that you want to use for the grid lines.
10. Click **Next**.
11. On the following pages, define the layout of grids for the bays in the X and Y directions.

    NOTE You can define either a regular or an irregular grid layout:

    - For a regular grid, define the number of bays you want to create and the bay centers.
    - For an irregular grid, define the distance between successive pairs of grid lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x18'6"`).

    You can also specify the reference of the first grid line, and by how much you want to increment that reference to give the references of the other lines that you create.

    - For alphanumeric grid lines, if you specify that the first grid line reference is to be D, and specify an increment of 3, your grid lines will be referenced D, G, J, M, and so on.

    ---

12. Click **Next**.
13. Define the rotation of the grid either by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**.

    TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the grid line system's local x or local y direction. This is helpful if you are creating a grid line system that is not orthogonal.

    ---

14. Click **Next**.
15. Specify the angle between the axes with respect to either the X or Y axis system.
16. Click **Finish**.

Create a radial grid line system

1. Ensure that the 2D view representing the construction level on which you want to create your radial grid line system is active.

2. On the **Model** tab, in the **Grid Line** list, select **Radial Grid Wizard**.

   The **Radial Grid Wizard** opens.

3. Type a name for the grid and select a color for the grid lines.
4. Ensure that each construction level on which you want to create the grid layout is selected.
5. Click **Next**.
6. Define the origin of the grid line system either by clicking it in the 2D view, or by entering its coordinates in the **Radial Grid Wizard**.
7. Click **Next**.
8. Select whether you want to create the radial lines only, the arcs only, or both of these.
9. Select the style that you want to use for the grid lines.
10. Click **Next**.
11. Define the layout of the arcs that you will create.

    NOTE You can define either a regular or an irregular grid layout:

    - For a regular grid, define the number of bays you want to create and the bay centers.
    - For an irregular grid, define the distance between successive pairs of grid lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x10'6"`).

    You can also specify the reference of the first grid line, and by how much you want to increment that reference to give the references of the other lines that you create.

    - For alphanumeric grid lines, if you specify that the first grid line reference is to be D, and specify an increment of 3, your grid lines will be referenced D, G, J, M, and so on.

    You can also select whether the arc grid lines are represented as curves, or as a series of straight lines between the points where the arc intersects the other grid lines created as part of this process.

    ---

12. Click **Next**.
13. Define the layout of radial grid segments that you want to achieve.

    Again, you can specify the reference of the first grid line, and how you want to increment the reference to give the references of the other radial lines that you create.

14. Click **Next**.
15. Define the rotation of the grid either graphically by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**

    TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the grid line system's local x or local y direction.

16. Click **Finish**.

Create a grid arc

1. Go to the **Model** tab.
2. In the **Grid Line** list, select **Arc**.
3. In the **Properties** window, type a name for the grid line.
4. Select the point which lies at the center of the grid arc which you want to create.

    The tooltip views the mouse pointer's exact coordinates.
5. Click the points where you want the grid arc to start and end.
6. Move the mouse pointer and place other grid arcs with respect to the line you selected in step 4, or press **Esc** to end grid placement.

Number and renumber grids

Each grid that you create is automatically numbered. To set the initial number or letter for naming grids, or to renumber all grids, see the following instructions.

Set the initial number or letter used for grids

1. On the **Home** tab, click **Model Settings**.
2. Go to **References** --> **General**.
3. In the **Grid Line Naming** section, set the initial values and the naming style.
4. Click **OK**.

Renumber all grids

1. In the **Structure** tree, right-click **Architectural Grids**.
2. In the context menu, select **Renumber**.

A dialog box is displayed which allows you to renumber the grid line based on

- the type from direction (default ON) or
- choose grid name type and direction

3. Select the parameters as required, then click OK.

   All grids in the model are renumbered in the chosen sequence.

See also

Change the name of a grid line or grid arc (page 185)

Change the name of a grid line or grid arc

To change the name of a single grid line or grid arc, see the following instructions.

1. Select the grid line or grid arc that you want to name.
2. In the **Properties** window, modify the name according to your needs.

See also

Number and renumber grids (page 184)

Change the name or color of an architectural grid (page 185)

Apply an architectural grid to a specific level

To view an existing architectural grid at a specific level of your model, you must apply the grid to the level. See detailed instructions in the following paragraphs.

1. In the **Structure** tree, click **Architectural Grids**.
2. Right-click the architectural grid name.
3. In the context menu, select **Edit...**
4. Ensure that the levels where you want the architectural grid to be applied are selected.

See also

Create and manage architectural grids and grid lines (page 178)

Change the name or color of an architectural grid

To modify the properties of an existing architectural grid, see the following instructions.

1. In the **Structure** tree, click the **+** sign next to **Architectural Grids**.
2. Select the architectural grid whose name you want to change.
3. In the **Properties** window, modify the architectural grid's properties according to your needs.

See also

Change the name of a grid line or grid arc (page 185)

Import grids from a DXF file or a shadow of the DXF file

If necessary, you can import grids from an existing DXF file to your Tekla Structural Designer document. In addition, you can use a DXF file as a shadow, or a base that helps you in creating objects in Tekla Structural Designer.

NOTE The DXF file that you use must be available before you start the import process, either sent to you, or created by you.

---

1. Open a 2D view of a construction level, and go to the **Model** tab
2. In the **Grid Line** list, select **Import DXF**.

   An **Open** dialog opens.
3. Browse to the file that you want to import, select the file, and click **Open**.

   The **DXF Import Wizard** opens.
4. On the first page, manage the layers and colors that you want to import.
5. Select if you want to import the architectural grids from the file, or import the file as a shadow.
   - If you import the architectural grids, all elements in the selected layers of the DXF file are mapped to the Tekla Structural Designer grid lines. You must therefore ensure that you switch off all the layers in the DXF file, apart from the layers in which the grids have been defined.
   - • If you import the DXF file as a shadow, Tekla Structural Designer imports the .dxf file but does not create any Tekla Structural Designer objects. You can use the intersection points and other elements as the source on which to add the Tekla Structural Designer objects you require.
6. If necessary, adjust the scale and offsets for the DXF file.
7. Click **Next**.
8. Select how the grids are created:
   - To create separate named grids for each layer, select **By layer**.

- To create separate named grids for each color in the DXF file, select **By color**.
- To create a single merged grid containing all layers and colors, select **Merged**.

9. Ensure that each level to which you want to import the grid layout is selected.
10. Click **Finish**.

See also

Create and manage architectural grids and grid lines (page 178)

Change the name of a grid line or grid arc (page 185)

Change the name or color of an existing architectural grid (page 185)

Extend, move, or rotate grid lines and arcs

You can modify grid lines and arcs in different ways according to your needs. You can stretch, shorten, or rotate grid lines, or move them in a perpendicular direction. As for grid arcs, you can stretch and shorten them, adjust their radius, or move them in any direction. For detailed instructions, see the following paragraphs.

RESTRICTION	You can only move grid lines and arcs in 2D views.

Stretch, shorten, or rotate a grid line

1. In a 2D view, select the grid line that you want to modify.

   The end nodes and the center node of the grid line become visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The node moves to its new position.

Move a grid line to a perpendicular direction

1. In a 2D view, select the grid line that you want to modify.

   The end nodes and the center node of the grid line become visible.
2. Click the center node.
3. Click the location where you want to move the node.

   The grid line moves to its new position.

Stretch or shorten a grid arc

1. In a 2D view, select the grid arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The arc stretches or shortens accordingly.

Adjust the radius of a grid arc

1. In a 2D view, select the grid arc that you want to modify.

   The end and center nodes of the arc perimeter and the center node of the arc visible.
2. Click the center node of the arc perimeter.
3. Click the location where you want to move the node.

   The arc radius adjust accordingly.

Move a grid arc

1. In a 2D view, select the grid arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.
2. Click the center node of the arc.
3. Click the location where you want to move the node.

   The arc moves to its new location.

Create and manage construction lines

Construction lines serve the same purpose as architectural and construction lines. The only difference is that they do not display a construction bubble. To create construction lines, see the following instructions.

Create a single construction line

NOTE If you want to create a series of construction lines which form a regular or irregular, rectangular or radial construction line system, the **Rectangular Wizard** and **Sector Wizard** are the fastest options.

---

1. Ensure that the 2D view representing the construction level on which you want to create your construction line is active.
2. Go to the **Model** tab.

3. In the Construction Line list, select Construction Line.
4. Select the point where you want the construction line to start.

TIP The tooltip views the mouse pointer's exact coordinates. If the mouse pointer has not snapped to the required point, press **F2** to type the exact coordinates of the required point.

5. Select the point where you want the construction line to end.

TIP The tooltip views either the absolute, relative, or polar coordinates of the end point, depending on whether the **ABS**, **REL**, or **POL** button is selected in the (page 67).

- To switch the display, click one of the other buttons.

NOTE The construction line does not extend to infinity. Do ensure that the construction line is sufficiently long to meet your needs.

Tekla Structural Designer creates the construction line between the selected points.

Create parallel construction lines

This option creates a construction line parallel to an existing one, but of a different length. To use this option, you must have at least one existing construction line in the current 2D view.

1. Go to the **Model** tab.
2. In the **Construction Line** list, select **Parallel**.
3. Select the construction line to which your new construction lines will be parallel.
4. Move the mouse pointer to the distance where you want to place the parallel construction line.

TIP The tooltip views the distance of the dotted line from the initial construction line you selected in step 3. Press **F2** to type the exact distance.

For more accuracy, you can zoom in.

5. Click to locate the new construction line.

6. Click the points where you want the new construction line to start and end.

   The construction line is now created.

7. Move the mouse pointer and place other construction lines with respect to the line you selected in step 3, or press **Esc** to end construction placement.

TIP To create parallel construction lines of the same length as the selected construction line, select the **Parallel (quick)** command.

When you create quick parallel construction lines, you can press **F2** to define the distance between successive pairs of construction lines, separating the numbers with commas. If you have a number of bays that are at the same centers, then you can specify these as a single entry.

For example, you can specify bay centers of 18', 27' , 18' 6" , 18' 6" , 18' 6" and 27" as `18' , 27' , 3x18' 6" , 27"`.

Create perpendicular construction lines

To use this option, you must have at least one existing grid or construction line in the current 2D view.

1. Go to the **Model** tab.
2. In the **Construction Line** list, select **Perpendicular**.
3. Select the construction line to which your new construction line will be perpendicular.
4. Move the new construction line to the desired position and click to locate it.

TIP The tooltip views the perpendicular distance of the dotted line from the middle of the construction line you selected. Press **F2** to type the exact distance.

---

5. Click the points where you want the new construction line to start and end.
6. Move the mouse pointer and place other construction lines with respect to the line you selected in step 4, or press **Esc** to end construction placement.

Create a rectangular construction line system

1. Ensure that the 2D view representing the construction level on which you want to create your rectangular construction line system is active.
2. On the **Model** tab, in the **Construction Line** list, select **Rectangular Wizard**.

   The **Rectangular Grid Wizard** opens.
3. Define the origin of the construction line system either by clicking it in the 2D view, or by entering its coordinates in the **Rectangular Grid Wizard**.
4. Click **Next**.
5. Select if you want to create construction lines in the X direction only, in the Y direction only, or in both directions.
6. Select the style that you want to use for the construction lines.
7. Click **Next**.
8. On the following pages, define the layout of constructions for the bays in the X and Y directions.

   NOTE You can define either a regular or an irregular construction layout:

   - For a regular construction, define the number of bays you want to create and the bay centers.
   - For an irregular construction, define the distance between successive pairs of construction lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x18'6"`).

   ---

9. Click **Next**.
10. Define the rotation of the construction either by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**.

    TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the construction line system's local x or local y direction. This is helpful if you are creating a construction line system that is not orthogonal.

    ---

11. Click **Next**.
12. Specify the angle between the axes with respect to either the X or Y axis system.
13. Click **Finish**.

Create a radial construction line system

1. Ensure that the 2D view representing the construction level on which you want to create your radial construction line system is active.
2. On the **Model** tab, in the **Construction Line** list, select **Radial Grid Wizard**.

   The **Radial Grid Wizard** opens.
3. Define the origin of the construction line system either by clicking it in the 2D view, or by entering its coordinates in the **Radial Grid Wizard**.
4. Click **Next**.
5. Select whether you want to create the radial lines only, the arcs only, or both of these.
6. Select the style that you want to use for the construction lines.
7. Click **Next**.
8. Define the layout of the arcs that you will create.

   NOTE You can define either a regular or an irregular construction layout:

   - For a regular construction, define the number of bays you want to create and the bay centers.
   - For an irregular construction, define the distance between successive pairs of construction lines, separating the numbers by commas. If you have a number of bays that are at the same centers, you can specify them as a single entry (for example, `3x10'6"`).

   You can also select whether the arc construction lines are represented as curves, or as a series of straight lines between the points where the arc intersects the other construction lines created as part of this process.

   ---

9. Click **Next**.
10. Define the layout of radial construction segments that you want to achieve.

    Again, you can specify the reference of the first construction line, and how you want to increment the reference to give the references of the other radial lines that you create.
11. Click **Next**.

12. Define the rotation of the construction either graphically by moving the mouse over the 2D view and clicking, or by typing values into the **Rectangular Grid Wizard**

    TIP In the **Rectangular Grid Wizard**, you can specify the rotation with respect to either the construction line system's local x or local y direction.

13. Click **Finish**.

Create construction arcs

1. Go to the **Model** tab.
2. In the **Construction Line** list, select **Arc**.
3. Select the point which lies at the center of the construction arc which you want to create.

   The tooltip views the mouse pointer's exact coordinates.
4. Click the points where you want the construction arc to start and end.
5. Move the mouse pointer and place other construction arcs with respect to the line you selected in step 3, or press **Esc** to end construction placement.

Extend, move, or rotate construction lines and arcs

You can modify construction lines and arcs in different ways according to your needs. You can stretch, shorten, or rotate construction lines, or move them in a perpendicular direction. As for construction arcs, you can stretch and shorten them, adjust their radius, or move them in any direction. For detailed instructions, see the following paragraphs.

RESTRICTION You can only move construction lines and arcs in 2D views.

Stretch, shorten, or rotate a construction line

1. In a 2D view, select the construction line that you want to modify.

   The end nodes and the center node of the construction line become visible.
2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The node moves to its new position.

Move a construction line to a perpendicular direction

1. In a 2D view, select the construction line that you want to modify.

   The end nodes and the center node of the construction line become visible.

2. Click the center node.
3. Click the location where you want to move the node.

   The construction line moves to its new position.

Stretch or shorten a construction arc

1. In a 2D view, select the construction arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.

2. Click one of the end nodes.
3. Click the location where you want to move the node.

   The arc stretches or shortens accordingly.

Adjust the radius of a construction arc

1. In a 2D view, select the construction arc that you want to modify.

   The end and center nodes of the arc perimeter and the center node of the arc visible.

2. Click the center node of the arc perimeter.
3. Click the location where you want to move the node.

   The arc radius adjust accordingly.

Move a construction arc

1. In a 2D view, select the construction arc that you want to modify.

   The end nodes of the arc perimeter and the center node of the arc visible.

2. Click the center node of the arc.
3. Click the location where you want to move the node.

   The arc moves to its new location.

Create frames and slopes

Frames and slopes are different 2D views of the model that can help you in creating your model. To create a frame or a slope, see the following instructions.

Create a frame

A frame is a 2D View of the model, created in a vertical plane defined by an existing grid line. Since only the members that lie within the plane of the frame are displayed, a frame view can be particularly useful for defining bracing.

1. Obtain a 3D view of your model where you can see the base grid line associated with the frame that you want to create.
2. On the **Model** tab, click **Frame**.
3. Position the mouse pointer over the grid line for the frame that you want to create.
4. Click to create the frame.

   Tekla Structural Designer creates a frame view for the selected grid line.

   TIP To open a frame view:

   a. Click the **+** sign on the left side of **Frames** in the **Structure** tree.

   b. Double-click the name of the frame whose view you want to open.

Create a slope

A slope is a 2D View of the model, created in a sloped plane. You can define a slope by selecting 3 existing grid points. Since only those members that lie within the plane of the slope are displayed, this type of view can be particularly useful for defining inclined roofs and ramps.

NOTE Before creating a slope, note that:

- In order to create a slope, you need to be able to click three existing (not co-linear) grid points that lie in the plane of the slope.
- Grid points are formed at grid line intersections on construction levels. Therefore, if the points required to define the slope do not currently exist, you may need to insert new construction levels and grid lines to form them.
- A sloped plane must be entirely contained within a single sub model because the sub model determines the mesh parameters to be applied.
- In a sloped plane, positive Y is always aligned to the up-slope direction, so that positive X is always perpendicular to the slope.

---

1. Obtain a 3D view of your model where you can see three grip points that define the sloped plane.

TIP If you cannot see the grid line intersections on a particular construction level in the 3D view:

a. In the **Structure** tree, select the level.

b. In the , select the **Show grids in the 3D view** option.

If you wish, you can clear the **Show grids in the 3D view** option for other levels to simplify the display.

2. On the **Model** tab, click **Sloped Plane**.
3. Click the three points which define the sloped plane.

   Tekla Structural Designer creates sloped plane view.

TIP To open a slope view:

a. Click the **+** sign on the left side of **Slopes** in the **Structure** tree.

b. Double-click the name of the slope whose view you want to open.

Create dimensions

Dimensions allow you to show distances between the appropriate points in your structure. The dimension lines are included on any drawings you create. To create dimensions, see the following instructions.

Create a single dimension

1. On the **Model** tab, click **Dimension**.
2. Click the grid point at the start of the dimension.
3. Click the grid point at the end of the dimension.

   Tekla Structural Designer shows a line between the selected points.
4. Move the line to the point where you want the dimension line to lie.

   Choose a point that is easily visible and does not conflict with the rest of your model's details.
5. Click to create the dimension.

Create beams, columns and braces

These topics introduce you to the methods of creating beams columns and braces (in any material).

We recommend you familiarize yourself with how to:

- Create beams (page 210)
- Create columns (page 197)
- Create braces (page 227)

Create columns

This section focuses on the operations required to create columns (in any material).

- Specify the column type and section size (page 197)
- Create a single column or series of columns (page 200)
- Create inclined columns and cranked columns (page 202)
- Create gable posts or parapet posts (page 202)
- Align a column to a specific angle or an angled... (page 203)
- Modify the position of columns and column stacks (page 204)

The following topics are relevant to steel columns only:

- Setting out steel and cold formed columns (page 204)
- Create plated or compound section steel columns (page 206)
- Specify a column splice (page 207)
- Add a base plate to a steel column (page 207)
- Create web openings (page 223)

Concrete columns specifically have an automatic alignment facility:

- Specify concrete column alignment relative to the grid (page 208)
- Modify the column alignment or specify offsets (page 209)

See also

Column properties (page 956)

Specify the column type and section size

Before you can place a column you must first specify the column type and an initial section size.

Specify the type of column

- On the **Model** tab, do one of the following:

To	Do this
Specify a steel column	1. Click the arrow under **Steel Column**. 2. In the list, select the **Steel Column** type.
Specify a plated or compound steel column	1. Click the arrow under **Steel Column**. 2. In the list, select the **Plated** column type.
Specify a concrete filled steel column	1. Click the arrow under **Steel Column**. 2. In the list, select the **Concrete Filled** column type.
Specify a concrete encased steel column	1. Click the arrow under **Steel Column**. 2. In the list, select the **Concrete Encased** column type. 3. In the**Properties** window, click **Encasing section**. 4. Click the arrow on the right side of **Encasing section**. 5. In the list that appears, select **<New\Edit...>** 6. Enter the size of encasing concrete section.
Specify a cast-in-place, or precast concrete column	• 1. Click the arrow under **Concrete Column**. 2. In the list that appears, select **Concrete Column** for cast-in-place, or **Precast**.
Specify a timber column	Click **Timber Column**.
Specify a cold formed column	In the **Cold Formed** group, click **Column**.

Specify the size of steel, cold formed and timber columns

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The (page 166) opens.
4. Select the desired section size, and click **Select**.

   TIP To define a custom section, click **Add...**

5. Before proceeding to create the column, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of plated or compound steel columns

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The (page 166) opens.
4. In the (page 166), choose the required compound section type from the left hand pane.
5. Select the desired section size, and click **Select**.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**

   b. Type the sections, plate dimensions, and gaps according to your needs.

   c. To add the section to the list, **OK**.

      The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

   d. Select the section in the **Select Section** dialog box.

   ---

6. Before proceeding to create the column, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of cast-in-place, or precast concrete columns

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The **Section** dialog box opens.
4. Select the column shape.
5. Define the dimensions of the column.

   NOTE Only click **Add** if you want to create a hollow column.

   Then, to define the void:

   a. In the tabular part of the dialog, select the shape and dimensions of the void.

   b. Leave the minor and major offsets as 0.0 to position the void centrally in the column, or adjust as necessary to create an offset.

   ---

6. Click **OK**
7. Before proceeding to create the column, adjust the remaining properties in the **Properties** window according to your needs.

Create a single column or series of columns

With Tekla Structural Designer, you can create various kinds of column. You can model columns in 2D views, frame views, and structure views.

Create columns in a level view

1. Ensure that you have defined the construction levels between which the column will run and the grid points between which it will lie.
2. Select the column type and size (page 197).
3. In the **Properties** window , adjust the base level and top level of the column, and other column properties, if necessary.
4. Do one of the following:

To	Do this
Create a single column	a. Click the point where you want to place the column. b. Press **Esc** to finish creating columns.
Create a series of columns	a. Move the mouse pointer to one corner of an imaginary box that will encompass the grid intersection points where you want to create columns. b. Hold down the left mouse button. c. Drag the mouse pointer to the diametrically opposite corner of the box. d. Release the mouse button.

Create a single column in a frame or structure view

NOTE In order to define a column in a frame view or a structure view, you must have already created the construction levels between which the column will run, and the grid points between which it will lie.

1. Select the column type and size (page 197).
2. Click the start point of the column.
3. Click the end point of the column.

   Tekla Structural Designer creates the column between the selected points.

Create plated or compound section steel columns

1. On the **Model** tab, click the arrow under  **Steel Column**.

2. In the list that appears, select **Plated**.
3. In the **Properties** window, click the arrow next to the **Section** property.
4. In the list that appears, select **<New\Edit...>**

   The **Select Section** dialog box opens.
5. Select the desired section type and section size.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**

   b. Type the sections, plate dimensions, and gaps according to your needs.

   c. To add the section to the list, **OK**.

      The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

   d. Select the section in the **Select Section** dialog box.

   ---

6. In the **Properties** window, adjust the remaining properties according to your needs.
7. To proceed, see **Create columns in a 2D view** and **Create a single column in a frame view or a structure view**.

   TIP In order to apply a plated or compound section an existing steel column, do the following:

   a. In the **Properties** window, set **Fabrication** to **Plated**.

   b. Click the arrow next to the **Section** property.

   c. In the list, either select a compound section that has already been applied in the model, or select **<New\Edit...>** to add a new one.

Create concrete filled or encased concrete columns

1. On the **Model** tab, click the arrow under **Steel Column**.
2. In the list that appears, select either **Concrete Filled** or **Concrete Encased**.

   The column adopts the properties that are currently displayed in the **Properties** window for creating steel columns.
3. In the **Properties** window, adjust the properties as necessary.

4. To proceed, see **Create columns in a 2D view** and **Create a single column in a frame view or a structure view**.

> **NOTE** You can now modify the **Section** property and specify a different section size above the space position.

See also

Create gable posts or parapet posts (page 202)

Create inclined columns and cranked columns

Tekla Structural Designer allows you to create inclined and cranked steel and concrete columns.

Create an inclined column

An inclined column is any column that is not truly vertical. In order to define an inclined column, you must have defined the construction levels between which the column will run, and the grid points between which it will lie.

> **NOTE** An inclined column can only be created in a frame view or a structure view.

1. Select the column type and size. (page 197)
2. In a frame or structure view, Click the start point of the column.
3. Click the end point of the column.

   Tekla Structural Designer creates the column between the selected points.

Create a cranked column

In order to define a cranked column, you must have created the construction levels between which the column will run and the grid points between which it will lie.

> **NOTE** An inclined column can only be created in a frame view or a structure view.

1. Select the column type and size. (page 197)
2. Click the start point of the column.
3. Hold down the **Ctrl** key and click the next node of the column.
4. Repeat step 3 until you have selected the second to last node of the column.
5. Release the **Ctrl** key and click the end point of the column.

   Tekla Structural Designer creates the column.

Create gable posts or parapet posts

Steel parapet posts are single span members with fixed end connections. Their specific purpose is to act as a means to transfer load from wind wall panels into columns: the decomposed load from the panel is applied as a point load, and the moment at the node connects the parapet post to the column. To create parapet or gable posts posts in Tekla Structural Designer, see the following instructions.

NOTE Parapet posts are not designed in Tekla Structural Designer.

---

1. On the **Model** tab, click any element type (for example, **Steel Column**).
2. In the **Properties** window, set the **Characteristic** to **Gable post** or **Parapet post**.

   The properties in the **Properties** window are updated to a type that is appropriate for the chosen post.
3. In the **Properties** window, adjust the properties according to your needs.

   NOTE For gable posts:

   - If an axial load release is required at the top, consider carefully which wind load deflection parameters are required.

   ---
4. Click the start point of the post.

   NOTE If you are using a point along a member, do the following:

   a. Click the member to see its points.

   b. Either click the point that you want to use, or type the distance to the point from the start of the member.

   ---
5. Click the end point of the post.

   Tekla Structural Designer creates the post.

Align a column to a specific angle or an angled grid line

You can align a column to a specific angle or a grid line before creating the column. Once created an existing column can be realigned by editing the angle in the **Properties** window.

1. Do one of the following:
   - To create a new column, on the **Model** tab, select the desired column type.
   - To align an existing column to a specific angle, in the model, click the desired column.

2. In the **Properties** window, click the **Rotation** property.
3. In the list, select one of the following options:
   - The **0°**, **90°**, **180°**, and **-90°** options align the column to the global axes.
   - The **Angle** option aligns the column to the exact rotation angle you specify.
   - The **Define** option aligns the column to the angle of any grid line you select. (Applies to new columns only).

Related information

Related video

Align columns to grids

Modify the position of columns and column stacks

You may sometimes need to modify the position of columns and column stacks in your model. To do so, see the following instructions.

Move a column

See: Move objects (page 322).

Modify the position of a single column stack

You can change the position of a column stack in either a frame view or a structure view.

1. In (page 55), ensure that the **Grid & Construction Lines** options is selected.
2. Select the column stack that you want to move.
3. Click one end node for the column stack to be moved.

   Ensure the node is highlighted in the **Select Entity** tooltip when selecting.
4. Click a grid or construction point where you want to move the selected end node.

   The entire column is redrawn, with the selected node moved to the new position.
5. Select the next end node.
6. Click a grid or construction point where you want to move the selected end node.

   The column is redrawn once more, with the selected node moved to the new position.

Setting out steel and cold formed columns

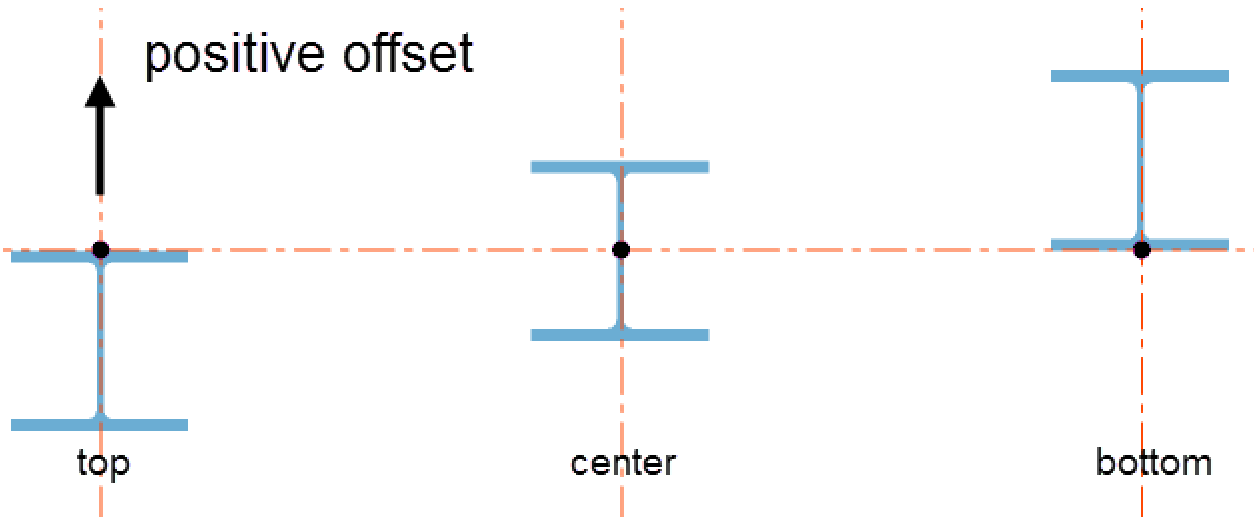
With Tekla Structural Designer, you can create various different types of steel and cold formed columns, including series of columns, inclined and cranked columns, gable or parapet posts, and plated, concrete filled or concrete encased columns. In addition, Tekla Structural Designer allows you to specify a column splice, align a column to a specific angle, or modify the position of existing columns.

Steel column alignment

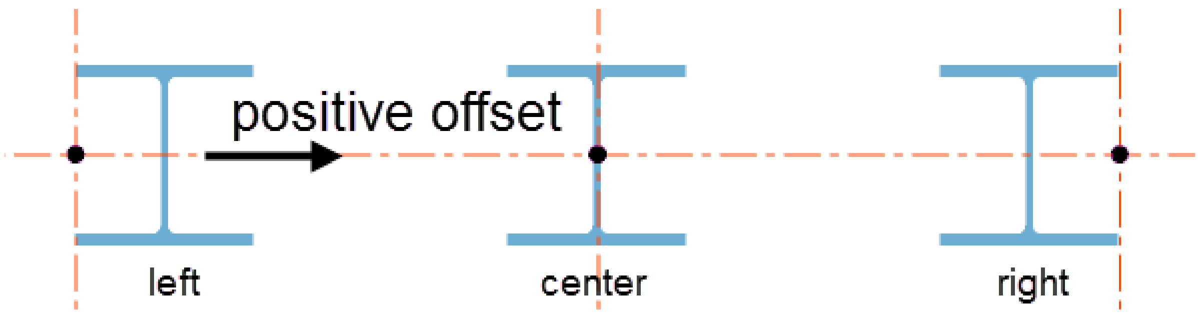
You can create a single steel column over several story heights. These kind of columns are referred to as column stacks. The columns can start and finish at any level. Different column sections can be defined in each stack, provided that a splice is defined at the change point.

Each steel column stack is placed on an insertion line between points, but its geometry is drawn to reflect the major and minor snap points and any offsets specified in the column properties.

Major offset:



Minor offset:



NOTE Steel column alignment snap points and offsets have no effect on the positioning of 1D solver elements in the solver models. The solver elements are always inserted along the steel column insertion lines.

This is different to the approach adopted for concrete column solver elements, in which the alignment snap points and offsets are structurally significant.

Create plated or compound section steel columns

Along with many other types of steel columns, Tekla Structural Designer allows you to create plated or compound section steel columns in your models.

Create plated or compound section columns

1. On the **Model** tab, click the arrow under **Steel Column**.
2. In the list that appears, select **Plated**.
3. Click the arrow on the right side of the **Section** property.
4. In the list that appears, select **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the section type and size.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**
   b. Type the sections, plate dimensions, and gaps according to your needs.
   c. To add the section to the list, **OK**.

      The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.
   d. Select the section in the **Select Section** dialog box.
6. In the **Properties** window, adjust the remaining properties according to your needs.
7. Click the start point of the column.
8. Click the end point of the column.

   Tekla Structural Designer creates the column.
9. Do one of the following:
   - To create more similar column, click the start point of the new column.
   - To stop creating columns, press **Esc**.

Apply a plated or compound section to an existing steel column

1. In the **Properties** window, go to **Fabrication**.
2. Set **Fabrication** to **Plated**.
3. Go to **Section** and do one of the following:
   - Select an existing compound section.
   - Click **<New\Edit...>** to select a new section.

See also

Add, modify and delete user-defined sections (page 890)

Specify a column splice

You can add splices at the base of each column stack (except for stack 1) as required. The splice offset is used to locate each splice at a practical distance above the floor level.

1. Select the desired columns.

   The properties of the columns are displayed in the **Properties** window.
2. Expand the properties of the column stack within which you want to create the splice.
3. Under **Release**, select the **Splice** option.
4. If necessary, modify the splice offset.

   NOTE The **Section** property is now editable, so that you can specify a different section size above the splice position, if necessary.

See also

Specify the column type and section size (page 197)

Add a base plate to a steel column

While base plates are automatically added to **steel** columns when they are created, you might also be required to manually add a base plate to an existing column in some situations (for example if the old base plate has been deleted, or a supporting member has been removed, or the column material has been changed).

To add a base plate:

1. On the **Design** toolbar, click  **Base Plate**.
2. In the **Properties** window:

a. Leave **Autosize** selected, or unselect it to manually define the plate size.

b. Edit the bolt properties as required.

c. Edit the anchor plate properties as required.

d. Edit the weld properties as required.

e. Edit the concrete base properties as required.

3. Click on a steel column, or drag a box around multiple steel columns to apply. The base plate is applied to all steel columns in the selection that didn't previously have a base plate attached.

See also

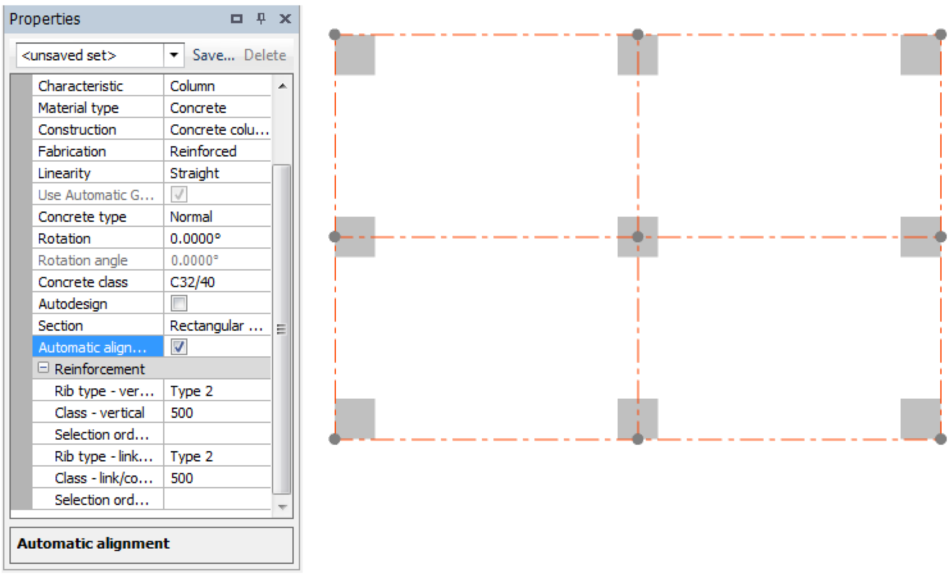
Create a single column or series of columns (page 200)

Specify concrete column alignment relative to the grid

The initial placement of each column relative to the grid depends on whether the **Automatic alignment** option is selected in the concrete column properties.

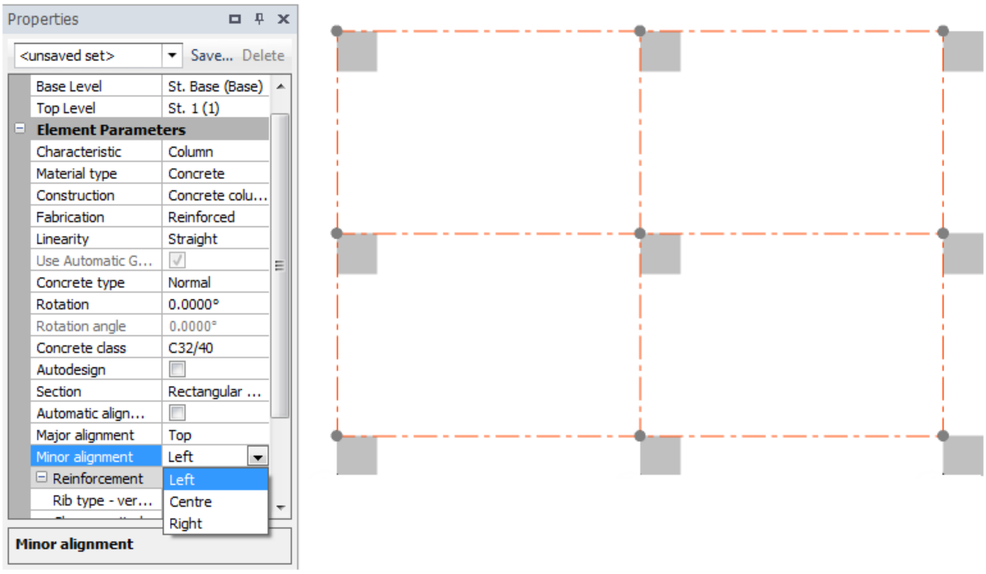
If **Automatic alignment** is selected:

Tekla Structural Designer aligns the columns on the perimeter of the grid with their faces flush to the perimeter, and the internal columns centrally on the grid.



If **Automatic alignment** is not selected:

Tekla Structural Designer aligns the columns according to the **Major alignment** and **Minor alignment** options in the concrete column properties.



- To switch the alignment option, in the **Properties** window, select or clear **Automatic alignment**.

See also

Align a column to a specific angle or an angled grid line (page 203)

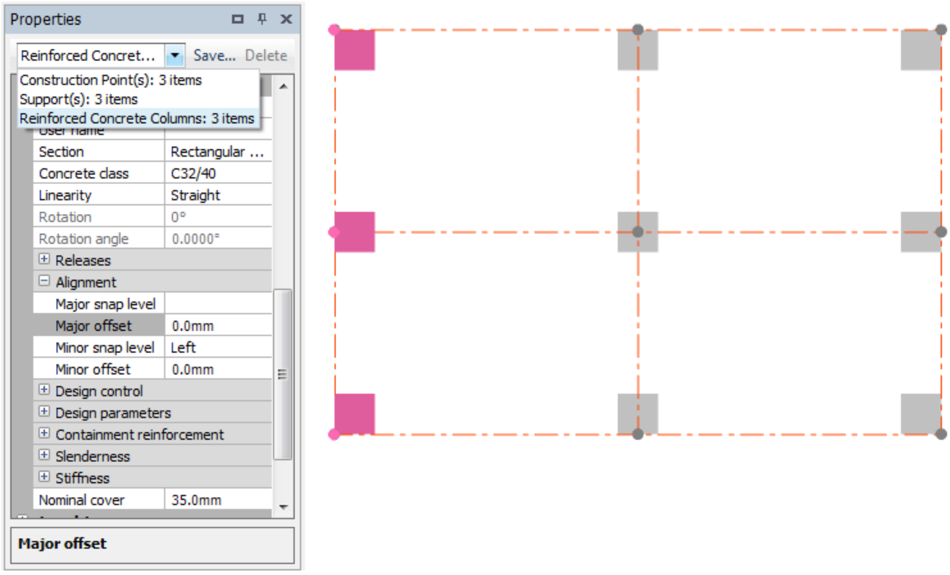
Modify concrete column alignment or specify offsets (page 209)

Modify concrete column alignment or specify offsets

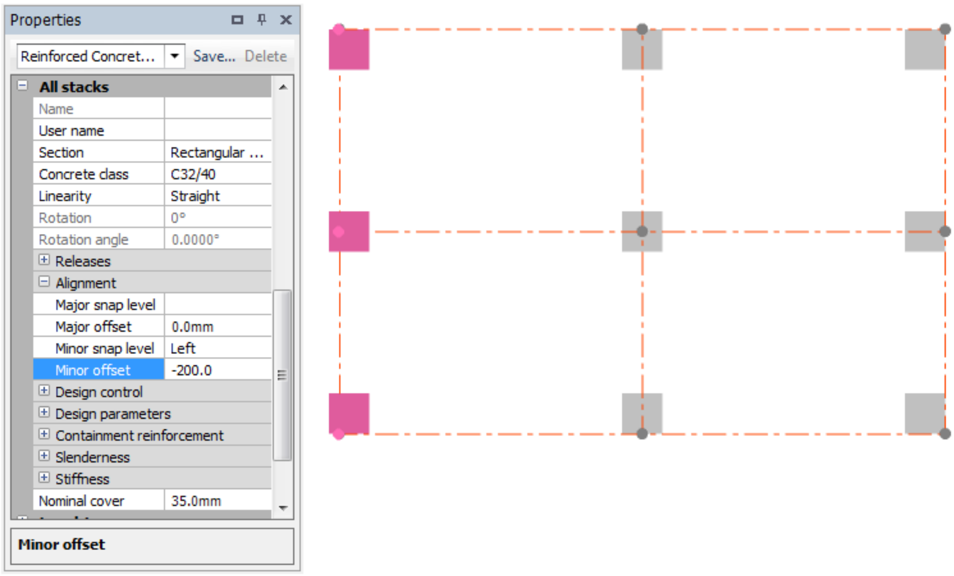
Once you have created columns, you can adjust their alignments and specify their offsets, if necessary. You can adjust the alignment of a single column in the **Properties** window or the **Properties** dialog box, whereas multiple columns can only be realigned in the **Properties** window.

The following example illustrates editing the alignment in the **Properties** window.

1. Select the columns that you want to offset.
2. Ensure that the column properties are viewed in the **Properties** window.



3. Modify the **Major offset** and **Minor offset** values according to your needs.



Tekla Structural Designer moves the columns relative to the major and minor snap levels by the value that you specified.

See also

Specify concrete column alignment relative to the grid (page 208)

Create beams

This section focuses on the operations required to create beams (in any material).

- Specify the beam type and section size (page 211)

- Create single-span beams (page 213)
- Create continuous beams (page 214)
- Create curved beams (page 215)
- Modify the position of beams (page 216)

The following topics are relevant to steel beams only:

- Setting out steel and cold formed beams (page 216)
- Create plated or compound section steel beams (page 218)
- Create Westok cellular, Westok plated or FABSEC® beams (page 219)
- DELTABEAM® (page 220)
- Create web openings (page 223)
- Add haunches to steel beams (page 226)

See also

Beam properties (page 938)

Member global offsets (page 231)

Specify the beam type and section size

Before you can place a beam you must first specify the beam type and an initial section size.

Specify the type of beam

- On the **Model** tab, do one of the following:

To	Do this
Specify a steel beam	1. Click the arrow under **Steel Beam**. 2. In the list that appears, select the desired beam type.
Specify a plated, Westok, FABSEC®, or DELTABEAM®	1. Click the arrow under **Steel Beam**. 2. In the list, select the required beam type.
Specify a cast-in-place, or precast concrete beam	1. Click the arrow under **Concrete Beam**. 2. In the list that appears, select **Beam** for cast-in-place, or **Precast**.
Specify a timber beam	Click **Timber Beam**.
Specify a cold formed beam	In the **Cold Formed** group, click **Beam**.

Specify the size of steel, cold formed and timber beams

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The (page 166) opens.
4. Select the desired section size, and click **Select**.

   TIP To define a custom section, click **Add...**

5. Before proceeding to create the beam, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of plated or compound steel beams

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.
3. In the list that appears, select **<New\Edit...>**

   The (page 166) opens.
4. In the (page 166), choose the required compound section type from the left hand pane.
5. Select the desired section size, and click **Select**.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**
   b. Type the sections, plate dimensions, and gaps according to your needs.
   c. To add the section to the list, **OK**.

      The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.
   d. Select the section in the **Select Section** dialog box.

6. Before proceeding to create the beam, adjust the remaining properties in the **Properties** window according to your needs.

Specify the size of cast-in-place, or precast concrete beams

1. In the **Properties** window, click **Section**.
2. Click the arrow on the right side of **Section**.

3. In the list that appears, select **<New\Edit...>**

   The **Section** dialog box opens.
4. Select the beam shape.
5. Define the dimensions of the beam.
6. Click **OK**
7. Before proceeding to create the beam, adjust the remaining properties in the **Properties** window according to your needs.

Create single-span beams

Tekla Structural Designer allows you to create concrete, steel, and timber single-span beams. For detailed instructions, see the following paragraphs.

Create a single-span beam

1. Specify the beam type and section size (page 211)
2. In the **Properties** window, adjust the beam properties according to your needs.
3. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.
4. Click the end point of the beam.

   NOTE If Tekla Structural Designer prompts you to pick another point, and you do not want to create a continuous beam, do one of the following:

   - Click again the end point again.
   - Press **Enter**.

Create a series of single-span beams

RESTRICTION In order to a series of single-span beams using this method, the floor or construction level must already contain the columns between which the beams will run. You must also use a 2D view of the floor or construction level.

1. Select the beam type and size (page 211)
2. In the **Properties** window, adjust the beam properties according to your needs.

3. Move the mouse pointer to one corner of an imaginary box that will encompass the columns between which you want to create beams.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the diametrically opposite corner of the box.

   Ensure that the box encompasses all of the columns between which you want to create beams.
6. Release the mouse button.

   Tekla Structural Designer creates the beams between each adjacent pair of columns within the area you selected.

See also

Create continuous beams (page 214)

Create curved beams (page 215)

Create continuous beams

You can create continuous beams in your model by selecting the **Continuous** option before creating the beam. For more information, see the following instructions.

RESTRICTION You cannot create continuous beams that are curved either horizontally or vertically.

---

1. Specify the beam type and section size (page 211)

   You can create steel, concrete, timber, or cold rolled beams.
2. In the **Properties** window, ensure that the **Continuous** option is selected.
3. In the **Properties** window, adjust the remaining properties according to your needs.
4. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

   ---
5. Click the next node of the beam.
6. Repeat step 5 until you have clicked the second to last node of the beam.
7. Click the end point of the beam.

   Tekla Structural Designer prompts you to pick another point.
8. To create the beam, do one of the following:

- Click the end point again.
- Press **Enter**.

Tekla Structural Designer creates the continuous beam.

See also

Create single span beams (page 213)

Create curved beams (page 215)

Create curved beams

Tekla Structural Designer allows you to create curved steel, concrete, and cold formed beams in your models, if necessary. To model curved beams, see the following instructions.

1. Select the beam type and size. (page 211)
2. In the **Properties** window, ensure that **Linearity** is set to one of the following:
   - **Curved major**, if you want the beam to curve vertically.
   - **Curved minor**, if you want the beam to curve horizontally.

   TIP You can control the direction in which horizontally curved beams curve.

   When you place the beam, you select its start point and its end point. The curve of the beam always lies on the left side of the line from the start point to the end point of the beam.

3. Still in the **Properties** window, specify an appropriate chord height value to define the curve.

   TIP You can reverse the curve direction of a vertically curved beam by using a negative chord height value.

   NOTE Horizontally curved beams always use the chord height defined in the property set. Therefore, they do not curve automatically to fit on any curved grid line that you may have defined.

4. In the **Properties** window, adjust the remaining properties according to your needs.
5. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

6. Click the end point of the beam.

   Tekla Structural Designer creates the beam.

   NOTE If you are using a 2D view when defining beams that curve vertically, you cannot see the beam on the graphical display.

   Change to a 3D view to see the beams.

See also

Create single span beams (page 213)

Create continuous beams (page 214)

Modify the position of beams

If necessary, you can modify the position of an individual beam in your model by moving one of its end nodes. You can modify the position of a beam in both 2D and 3D views.

To move an entire beam, see: Move objects (page 322).

1. Select the desired beam.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click the grid or construction point where you want to move the end node.

   Tekla Structural Designer moves the end node to the selected point.

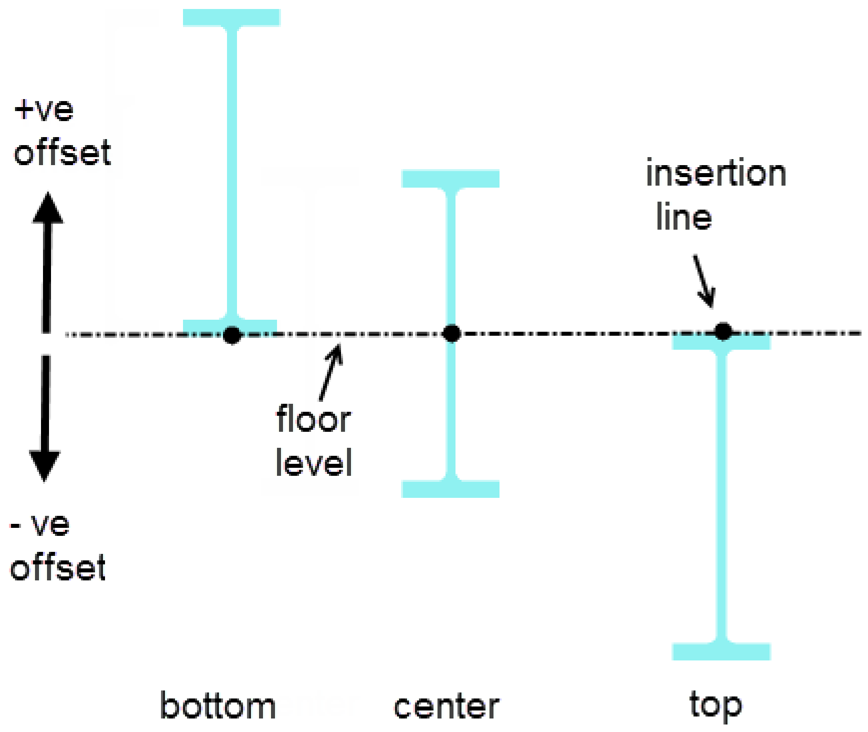
Setting out steel and cold formed beams

You can define steel beams as single span, or continuous over multiple spans. Even if you create a continuous beam, you can still define different beam sections in each span, if necessary.

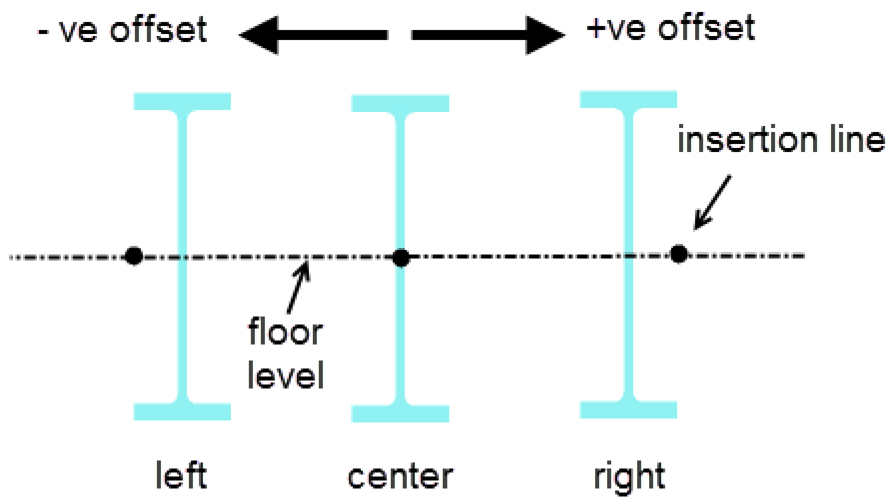
Each steel beam span is placed on an insertion line between points, but its geometry is drawn to reflect:

- the major and minor snap points and any offsets specified in the beam properties

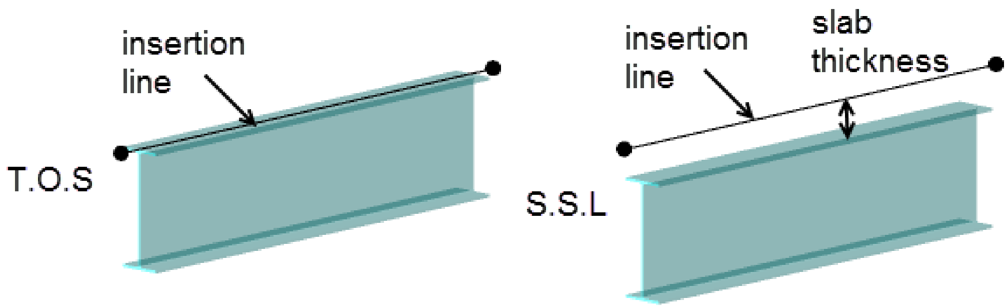
  Major snap and offset:



Minor snap and offset:



• the level type specified in the **Construction Levels** dialog box dialog:



- When the type is set to **T.O.S.** (top of steel), each beam is displayed according to the alignment snap points and offsets specified.
- When the level type is set to **S.S.L.** (structural slab level), each beam is also lowered by the slab thickness specified in the **Construction Levels** dialog box.

NOTE Steel beam alignment snap points and offsets and the construction level type have no effect on the positioning of 1D solver elements in solver models. This is a different approach to that adopted for concrete beam solver elements.

Create plated or compound section steel beams

Along with many other types of steel beams, Tekla Structural Designer allows you to create plated or compound section steel beams in your models.

Create plated or compound section beams

1. On the **Model** tab, click the arrow under **Steel Beam**.
2. In the list that appears, select **Plated**.
3. Click the arrow on the right side of the **Section** property.
4. In the list that appears, select **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the section type and size.

   TIP If the desired section is not listed, you can add it as follows:

   a. Click **Add...**

   b. Type the sections, plate dimensions, and gaps according to your needs.

c. To add the section to the list, **OK**.

   The section is automatically added to the materials database. Therefore, you do not need to redefine it in other models.

d. Select the section in the **Select Section** dialog box.

---

6. In the **Properties** window, adjust the remaining properties according to your needs.
7. Click the start point of the beam.
8. Click the end point of the beam.

   Tekla Structural Designer creates the beam.
9. Do one of the following:
   - To create more similar beams, click the start point of the new beam.
   - To stop creating beams, press **Esc**.

Apply a plated or compound section to an existing steel beam

1. In the **Properties** window, go to **Fabrication**.
2. Set **Fabrication** to **Plated**.
3. Go to **Section** and do one of the following:
   - Select an existing compound section.
   - Click **<New\Edit...>** to select a new section.

See also

Add, modify and delete user-defined sections (page 890)

Create Westok cellular, Westok plated or FABSEC® beams

Tekla Structural Designer also allows you to create Westok Cellular, Westok Plated or FABSEC® beams, if that is necessary in your model. For more information, see the following instructions.

1. On the **Model** tab, click the arrow next to **Steel Beam**.
2. In the list, select the desired beam type.

   The beam adopts the properties that are currently displayed in the **Properties** window.
3. In the **Properties** window adjust the properties according to your needs.
4. Click the start point of the beam.

TIP If you are using a point along a member, do the following:

a. Click the member to view its points.

b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

5. Click the end point of the beam.

TIP If Tekla Structural Designer prompts you to pick another point, do one of the following:

- Click the end point again.
- Press **Esc**.

Tekla Structural Designer creates the beam.

See also

Create single span beams (page 213)

Create plated or compound section steel beams (page 218)

Create continuous beams (page 214)

Create curved beams (page 215)

DELTABEAM®

DELTABEAM® is a proprietary slim floor system. While DELTABEAM® sections can be analysed in Tekla Structural Designer they are not designed.

NOTE DELTABEAMS® are manufactured by Peikko in Finland: www.peikko.com.

The Peikko Designer download, (English language version) is available from:

https://www.peikko.com/design-tools/

Workflow in Tekla Structural Designer

- Initial DELTABEAM® section sizes are selected from the database and applied in the model. Peikko Designer can be used if required to better determine suitable initial sizes.
- The model is analysed, and a member forces report is created. This report is then sent to Peiko.
- Peiko provide the detailed design and final analysis properties.
- The Tekla Structural Designer model is updated as required.
- If the properties have been revised, the self weight could be affected and the model should be reanalysed.
- A further iteration may or may not be required.

Create DELTABEAMS®

1. On the **Model** tab, click the arrow next to **Steel Beam**.
2. In the list, select DELTABEAM®

   The beam will adopt the properties displayed in the **Properties** window.
3. In the **Properties** window select the section required.

   NOTE D Series internal beams and DR Series edge beams can be selected provided the database country is set to Finland, Norway, Sweden, or Europe. You can add user defined sections to the database for the same countries if required.

4. Select the construction as either composite, or non-composite, as required.
5. Adjust the remaining properties according to your needs.
6. Click the start point of the beam.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

7. Click the end point of the beam.

   TIP If Tekla Structural Designer prompts you to pick another point, do one of the following:

   - Click the end point again.
   - Press **Esc**.

   Tekla Structural Designer creates the beam.

   NOTE If you are placing a DR series edge beam, the flange will be drawn to the *left* of the line from start point to end point (easily verified in a 3D scene view). Once the beam has been placed, if you want to switch it to the other side simply select **Reverse** on the **Edit** tab and then click on the beam.

See also:

- Create single-span beams (page 213)
- Create continuous beams (page 214)

Create a member forces report

If you haven't already got a suitable report style configured you will need to create one as follows:

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.

2. To add a report, click **Add** and type a name for the report in the **Active Style** field.

3. To add the member forces chapter to the report, expand the Analysis headung in 'Chapters and Options' and drag the Member Forces chapter to the Report Structure pane on the right.

4. Review the report structure and modify it (page 831) according to your needs.

5. To limit the output to selected levels, frames, planes, or sub structures, apply a model filter (page 832).

   TIP You can further limit the output of **Loadcases** and **Combinations** sub chapters by applying a loading filter (page 832).

6. Click **OK**.

7. Click **Show Report** to review the report and check that it is displaying the required level of data.

Once the report is configured to your needs, it can then be exported to Excel, PDF, Word, or Tedds.

See also:

• Create general arrangement drawings (page 855)

Add DELTABEAM® properties to the database

Having submitted your member forces report to Peiko, you might be provided with a new set of analysis properties that are not in the Tekla Structural Designer database. In this situation you would need to add them as follows:

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.

2. Go to **Sections**, and click **Manage Sections**.

   The **Sections** dialog box opens.

3. In the country list, select the database country.

4. In the **Page** pane on the left, select the appropriate DELTABEAM Series.

5. Click **Add...**

6. Enter values for each of the requested variables as supplied by Peiko.

7. Click **OK**.

The new section size is now displayed in the **Item** pane.

Related video

[Easy modeling of Peikko Deltabeams]

Create web openings in steel members

You can add square, rectangular, and circular web openings to beams and columns either by specified them manually, or by using a **Quick layout** option. FABSEC® beams can also have elongated openings specified.

RESTRICTION

1. Although web openings can be added to steel beams and columns, they are only considered in the design of:
   - Non-composite beams designed to US codes, BS codes, or Eurocodes
   - Composite beams designed to US codes, BS codes, or Eurocodes
2. Web openings are only valid in check design. If the auto design flag is switched on the openings will be removed.

Web opening creation methods

You add web openings via the **Web openings** page of the **Properties** dialog box.

The **Quick layout** method adds web openings to meet the geometric and proximity recommendations published by the SCI. With this method, Tekla Structural Designer creates web openings at the maximum depth and spaced at the minimum centers recommended for the section size.

With the **Quick layout** option cleared you can also define web openings manually using one of the following two methods:

- Clicking the **Add** button adds a new line to the web openings grid and allows you to define the geometric properties of the opening.
- Clicking the **Add...** button opens the **Web opening** dialog box where you can get more help and guidance when defining the opening.

Both manual methods use and signs to indicate faulty data when defining the opening parameters.

For more assistance, hover the mouse pointer over a or sign.

The Web opening dialog box

In the **Web opening** dialog box, you can use the following buttons to automatically define the position of the web opening:

- The **Center** button positions the opening on the center of the member.
- The **Auto** button positions the opening to meet the spacing recommendations by the SCI.

As web openings are defined, they are immediately visible in the diagram in the **Web opening** dialog box.

Tekla Structural Designer also performs design checks when you are defining web openings in the **Web opening** dialog box. The checked areas are end posts, web posts, web opening dimensions and tee dimensions.

The and signs in the **Web opening** dialog box help you to decide whether to make any adjustments to the opening parameters before Tekla Structural Designer checks the design.

NOTE The design checks carried out at the current stage are geometric checks only. Compliance with recommended limits is no guarantee that the opening will pass the subsequent engineering design checks.

Add web openings using the Quick layout option

Using the **Quick layout** method allows you to create maximum depth openings spaced at the minimum centers, appropriate to the section size.

1. Click the member to which you want to add web openings.
2. In the **Properties** window, ensure that the **Autodesign** option is cleared.
3. Right-click the member, and in the context menu, select **Edit [element name]**.

   The **Properties** dialog box opens.
4. Go to **Web openings**.
5. Select the **Quick layout** option.
6. In the **Label openings from** list, select where you want to start the setting out from.
7. In the **Type** column, select the web opening type.

   Data for the first web opening is automatically created as follows:

   - **l_O**: the length of the opening (applies to rectangular openings only)
   - **d_O**: the depth of the opening
   - **L_{CR}**: the distance from the setting out point to the center of the opening

- **L_C**: the distance from the end 1 on the member to the center of the opening
- **d_C**: the distance from the top of the member to the center of the opening
- **L_{CR} relative to**: indicates the setting out point from which L_{CR} is measured.
- **Nr. rel. to**: specifies an existing opening number that you want to use as the setting out point for the new opening.

  Nr. rel. to only applies if you selected one of the **Opening ->** options in the **L_{CR} relative to** column.

8. If necessary, in the **Stiffening** column, select the location of stiffening.
   a. Type the details of the stiffeners manually as follows:
      - **d_S**: depth of the stiffener
      - **t_S**: thickness of the stiffener
      - **L_S**: length of the stiffener
      - **e_S**: the distance from the edge of the opening to the center of the stiffener
9. To create further openings from the selected setting out points, click **Add** and select the opening type.

   TIP You can use the **Label openings from** list to switch to a new setting out point for the next opening at any point.

10. When you have created the desired openings, click **OK**.

Add web openings manually

1. Click the member to which you want to add web openings.
2. In the **Properties** window, ensure that the **Autodesign** option is cleared.
3. Right-click the member, and in the context menu, select **Edit [element name]**.

   The **Properties** dialog box opens.
4. Go to **Web openings**.
5. Ensure that the **Quick layout** option is not selected.
6. In the **Label openings from** list, select where you want to start the setting out from.
7. Click **Add...**
8. In the **Type** column, select the web opening type.

9. Specify the following details for the opening:
   - **l_O**: the length of the opening (applies to rectangular openings only)
   - **d_O**: the depth of the opening
   - **L_{CR}**: the distance from the setting out point to the center of the opening
   - **L_C**: the distance from the end 1 on the member to the center of the opening
   - **d_C**: the distance from the top of the member to the center of the opening

     ---

     TIP Click Center for dC to be automatically calculated to position the opening centrally in the section depth.

   - **L_{CR} relative to**: indicates the setting out point from which L_{CR} is measured.
   - **Nr. rel. to**: specifies an existing opening number that you want to use as the setting out point for the new opening.

     Nr. rel. to only applies if you selected one of the **Opening ->** options in the **L_{CR} relative to** column.
10. If necessary, in the **Stiffening** column, select the location of stiffening.
    a. Type the details of the stiffeners manually as follows:
       - **d_S**: depth of the stiffener
       - **t_S**: thickness of the stiffener
       - **L_S**: length of the stiffener
       - **e_S**: the distance from the edge of the opening to the center of the stiffener
11. To create the opening, click **OK**.
12. To create further openings, do one of the following:
    - To create multiple copies of a selected opening, click **Copy...**
    - To create a single opening of a different size or spacing, click **Add...**

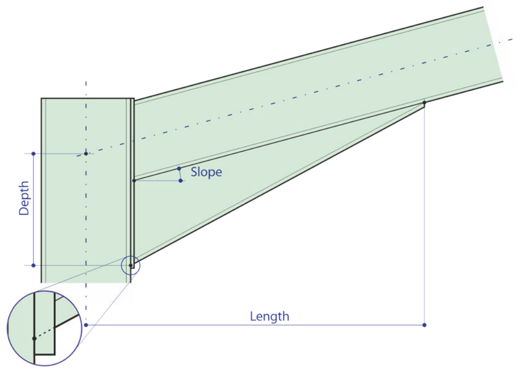
Add haunches to steel beams

1. Right-click the steel beam to which you want to add a haunch, and in the context menu, select **Edit [element name]**.

   The **Properties** dialog box opens.
2. Go to **Haunches**.
3. Select the appropriate **Create** option for the required haunch location.

4. In the **Haunch** dialog, define the haunch parameters as required.

   The haunch depth is measured from the beam centerline and length is measured horizontally from the sharp end of the haunch to the beam insertion point. In the case of an inclined beam the haunch length and depth would therefore be measured as shown below.



5. When you have defined the haunch parameters, click **OK**.

See also

Create single span beams (page 213)

Create braces

Tekla Structural Designer allows you to create single braces, or pairs of braces (in an X, K, V, or A configuration).

Braces are single span members with pinned end connections that are only able to resist axial compression and tension. Their purpose is to provide lateral stability to your structure.

TIP You can specify rigid frames to achieve the same effect as by using braces. If necessary, you can use both rigid frames and braces within a single structure.

Element loads cannot be applied directly to a brace itself and moments due to self weight loading are ignored.

NOTE Although it is possible to model A or V brace configurations using individual brace members instead of a brace pair, Tekla Structural Designer is then not able to calculate the notional loads or EHFs (Equivalent Horizontal Forces) correctly. In this case, elements of the vertical loads that are supported by the bracing system are lost, and are not included in the notional load or EHF calculations.

Setting out braces

Each steel brace is placed on an insertion line between points, with its geometry being drawn to reflect the major and minor snap points and any offsets specified in the brace properties.

NOTE The alignment snap points and offsets have no effect on the positioning of the solver elements created in the solver models, as these will be created directly between the insertion points.

1. Do one of the following:

To	Do this
Create a steel brace	a. On the **Model** tab, click the arrow under **Steel Brace**. b. In the list, select the desired brace type.
Create a cold formed brace	a. On the **Model** tab, click **Cold Formed**. b. In the list, select **Brace**. c. Select the desired brace type.
Create a timber brace	a. On the **Model** tab, click the arrow on the right side of **Timber Brace**. b. In the list, select the desired brace type.

2. In the **Properties** window, go to the **Section** property.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, click **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the desired section, and click **Select**.
6. In the **Properties** window, adjust the remaining properties according to your needs.

Specify the brace type and section size

Before creating a brace, you need to specify its type and section size. For more information, see the following instructions.

1. Do one of the following:

To	Do this
Create a steel brace	a. On the **Model** tab, click the arrow under **Steel Brace**. b. In the list, select the desired brace type.

Create a cold formed brace	a. On the **Model** tab, click **Cold Formed**. b. In the list, select **Brace**. c. Select the desired brace type.
Create a timber brace	a. On the **Model** tab, click the arrow on the right side of **Timber Brace**. b. In the list, select the desired brace type.

2. In the **Properties** window, go to the **Section** property.
3. Click the arrow on the right side of **Section**.
4. In the list that appears, click **<New\Edit...>**.

   The **Select Section** dialog box opens.
5. Select the desired section, and click **Select**.
6. In the **Properties** window, adjust the remaining properties according to your needs.

Create a single brace

NOTE A and V Braces should be modeled using special tools. Although you can model the exact same A or V brace arrangement using individual brace members, Tekla Structural Designer cannot calculate the notional loads or EHFs (Equivalent Horizontal Forces) of the braces correctly. In this case, elements of the vertical loads that are supported by the bracing system are lost, and are not included in the notional load or EHF calculations.

1. Select the brace type and size.
2. Click the start point of the brace.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.
3. Click the end point of the brace.

   Tekla Structural Designer creates the brace.

Create an X, K, V or A brace

1. Do one of the following:

To	Do this
Create a steel brace	• On the **Model** tab, click the arrow under **Steel Brace**.
Create a cold formed brace	a. On the **Model** tab, click **Cold Formed**. b. In the list, select **Brace**.
Create a timber brace	• On the **Model** tab, click the arrow on the right side of **Timber Brace**.

2. In the list, select the desired brace pattern.

   The braces adopt the properties that are currently displayed in the **Properties** window.

3. In the **Properties** window, adjust the properties according to your needs.
4. Click to identify a bottom corner of the bay that you want to brace.
5. Click to identify the opposite bottom corner of the bay.
6. Click to identify a top corner of the bay.
7. Click to identify the opposite top corner of the bay.

   Tekla Structural Designer creates the brace pattern within the selected area.

Specify a brace as tension only or compression only

You can move the end nodes of individual steel, concrete, cold formed, or timber braces in both 2D and 3D views. For more information, see the following instructions.

Once a brace has been created, you can specify it as Tension only or Compression only in the **Properties** window.

NOTE Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed, they will be treated as linear elements.

1. Select the brace.
2. In the **Properties** window select either **Compression only**, or **Tension only** as required.

Modify the position of a brace

You can move the end nodes of individual steel, concrete, cold formed, or timber braces in both 2D and 3D views. For more information, see the following instructions.

To move an entire brace, see: Move objects (page 322).

1. Select the brace.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click a grid or construction point where you want to move the end node.

   Tekla Structural Designer redraws the brace, moving the end node to the selected point.

See also

Brace properties (page 951)

Member global offsets (page 231)

Member global offsets

In certain situations you may have a requirement to model different global (X,Y,Z) physical offsets at each end of a member.

In Tekla Structural Designer global offsets can be applied to steel, cold formed, cold rolled and timber member types, but with the following exceptions:

- columns (all materials),
- concrete beams
- analysis elements
- members with haunches
- curved members

NOTE Global offsets have no affect on the positioning of the 1D solver elements in the solver models; they have no impact on loading, analysis or design. Global offsets only affect the graphics and BIM integration.

Global offsets are applied to the member ends **before** any major/minor local offsets, the local offsets thus being applied to the whole length of the member in relation to the new line.

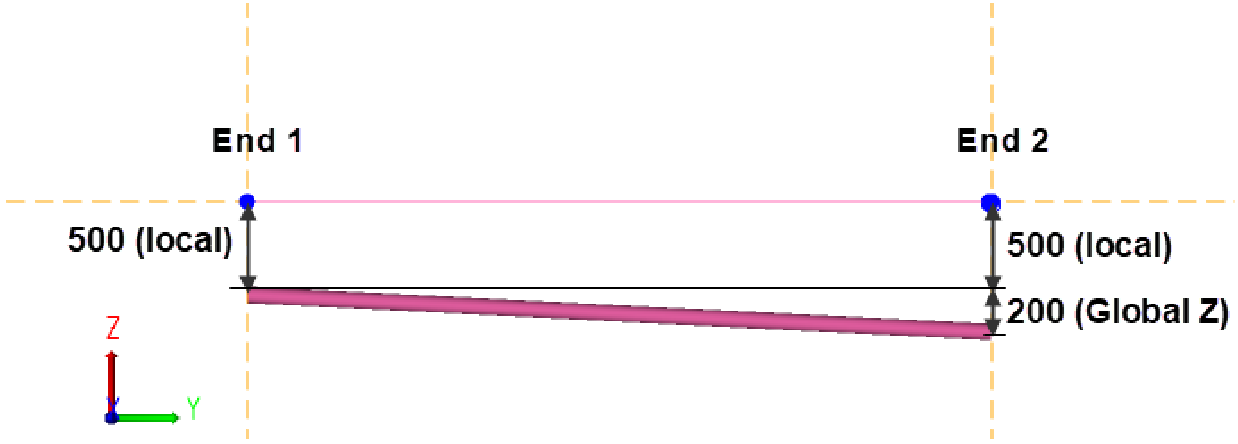
NOTE Global offsets are included in the BIM import/export to/from Tekla Structures

Global offset combined with major offset example

A horizontal brace member is specified with a Global End 2 offset of -200mm and a major (local) offset of -500mm, so that its alignment properties are as follows:

Alignment	
Global offset end 1	[0.0, 0.0, 0.0] mm
Global offset end 2	[0.0, 0.0, -200.0] mm
X	0.0mm
Y	0.0mm
Z	-200.0mm
Major snap level	Centre
Major offset	-500.0mm
Minor snap level	Centre
Minor offset	0.0mm

While the offsets do not affect the insertion line (denoting the 1D solver element in the analysis model), the brace itself is offset in the graphics as shown below:

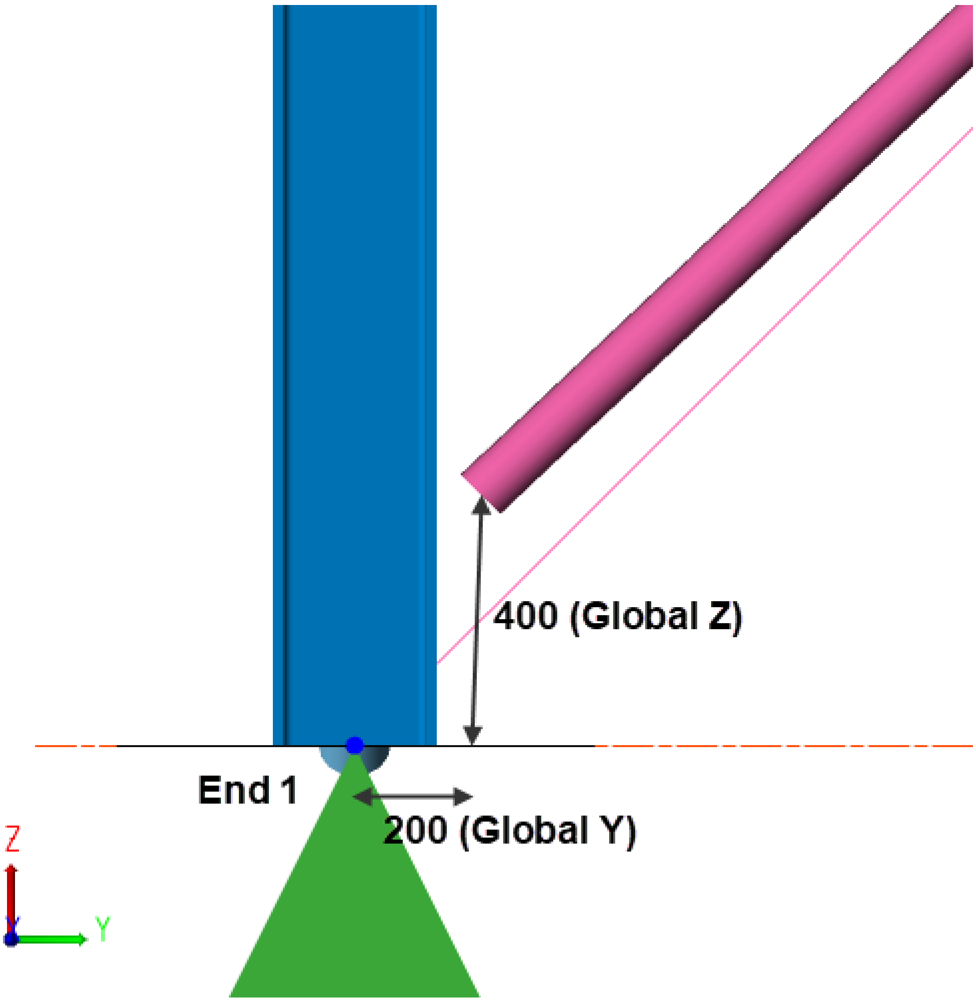


Global offset only example

A diagonal brace member is specified with a Global End 1 offsets of 200mm in Z and 400mm in Y, the Major and Minor (local) offsets are both 0mm, so that its alignment properties are as follows:

⊟ Alignment	
⊟ Global offset end 1	[0.0, 200.0, 400.0] mm
X	0.0mm
Y	200.0mm
Z	400.0mm
⊞ Global offset end 2	[0.0, 0.0, 0.0] mm
Major snap level	Centre
Major offset	0.0mm
Minor snap level	Centre
Minor offset	0.0mm

While the offsets do not affect the insertion line (denoting the 1D solver element in the analysis model), the brace itself is offset in the graphics as shown below:



Global offsets

Create walls, cores and bearing walls

These topics introduce you to the methods of creating walls, cores, and bearing walls.

We recommend you familiarize yourself with how to:

- Create concrete walls (page 235)
- Create concrete cores (page 245) (from existing concrete walls, columns and beams)
- Create bearing walls (page 248)
- Create shear only walls (page 252)
- Create general walls (page 254)

Create concrete walls

This section focuses on the operations required to create concrete walls.

Overview of the concrete wall model

Tekla Structural Designer allows you to create both meshed and mid-pier concrete walls.

Geometric rules for meshed concrete walls

- Meshed walls are defined as quadrilaterals in a single plane that can be vertical or sloping, (unlike mid-pier concrete walls which must be rectangular in a vertical plane).
- Openings (page 241) are permitted in meshed walls.
- The alignment and offsets in the wall properties are not structurally significant as they have no affect on the solver elements that are formed in the solver model.
- By default, meshed walls use the mesh parameters that are defined in **Structure Properties**. However, you can override these for an individual wall by checking **Override model's** in the Meshing section of the wall's properties. You are then able to apply a user-defined mesh to the wall.
- Concrete meshed walls can be included in cores, General meshed walls cannot.

Sub-division of meshed walls

Each wall object is split into separate panels only at those construction levels where an element or slab attaches to the wall. At the levels where the wall has been split, wall beam elements are also inserted.

Geometric rules for mid-pier concrete walls

- Mid-pier walls must be rectangular in a vertical plane.
- Wall openings are ignored.
- The alignment and offsets specified in the wall properties are not structurally significant as they have no affect on the solver elements that are formed in the solver model.
- Mid-pier walls can be included in cores.

Sub-division of mid-pier walls

Each wall object is split into separate panels only at those construction levels where an element or slab attaches to the wall. At the levels where the wall has been split, wall beam elements are also inserted.

Continuous concrete walls

Tekla Structural Designer allows you to create walls that are several stories high and that can start and finish at any level.

Although Tekla Structural Designer only creates one wall in this case, you can define different thicknesses for each wall panel. The panels are then set back on one or both faces, depending on the alignment that you have specified.

Creating a continuous wall this way, instead of defining a new wall on each floor, does not have any significance for analysis or design purposes. However, it is ultimately important for detailing purposes.

Analysis of concrete walls

The points that you use to place a concrete wall define the exact size and position of the wall’s analysis model. The alignment and extension properties of the wall have no effect on the analysis model.

Tekla Structural Designer can adopt either an FE meshed or mid-pier wall analysis on a wall by wall basis.

Support

Provided that the **AutomaticGenerateSupport** option is selected, if no slab or other member exists beneath the wall when it is first created, Tekla Structural Designer automatically places a support underneath the wall.

Material Type

For walls with the **Material type** set to concrete (as opposed to general), there are then two types of **Fabrication** available:

- Cast-in-place
- Precast

For each fabrication type there are two **Concrete types** available:

- Normal
- Lightweight

The **Grade** lists all the available grades in the Materials database under the current head code for the selected Concrete type.

Releases

You can apply minor axis releases at the top and the bottom of each panel in order to model pinned connections to incoming slabs and members.

Door and window openings

You can only define door and window openings in meshed walls. Tekla Structural Designer does not allow you to create openings in mid-pier walls. For more information openings, and the alternative ways to model them, see: Concrete wall openings analysis model (page 242)

Purpose of concrete walls

Both meshed and mid-pier concrete walls introduce structural strength and stiffness to your model. However, they do not perform the same function as wall panels. This means that wind loads calculated by the Wind Wizard cannot be applied to your structure if the model does not contain wall panels.

In order to apply wind loads, you must create additional wall panels in the same physical locations as the concrete walls.

Create meshed or midpier concrete walls

You can create meshed or mid-pier concrete walls both in 2D views and frame or structure views. Mid-pier walls are always rectangular; meshed walls can either be rectangular or quadrilateral.

For detailed information on creating concrete walls, see the following instructions.

Create rectangular walls in a level view

1. On the **Model** tab, click the arrow below  **Concrete Wall**.
2. In the list that appears, select the desired wall type.
   - Mid-pier Wall
   - Meshed Wall
3. Go to the **Properties** window.
4. Adjust the top and base level for the wall, if necessary.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. In the model, click the start point of the wall.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.
   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

7. Click the end point of the wall.

   Tekla Structural Designer creates the wall.

Create rectangular walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run, and the grid points between which it will lie.

1. On the **Model** tab, click the arrow below **Concrete Wall**.
2. In the list that appears, select the desired wall type.
   - Mid-pier Wall
   - Meshed Wall
3. Go to the **Properties** window.
4. Ensure the **Rectangular** property is selected.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. Click a point to define a corner of the wall.
7. Click a point to define the opposite corner. Tekla Structural Designer creates the wall between the selected two points.

Create quadrilateral walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run, and the grid points between which it will lie.

1. On the **Model** tab, click the arrow below **Concrete Wall**.
2. In the list that appears, select the **Meshed Wall**.
3. Go to the **Properties** window.
4. Ensure the **Rectangular** property is unselected.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. Click the point where the base of the wall should start.
7. Click the point where the base of the wall should end.
8. Click the point where the top of the wall should start.
9. Click the point where the top of the wall should end. Tekla Structural Designer creates the wall between the selected four points.

See also

Concrete meshed and mid-pier wall properties (page 965)

How meshed walls are represented in solver models (page 577)

How mid-pier walls are represented in solver models (page 582)

Create door or window openings (page 241)

Specify extensions and releases
Tekla Structural Designer allows you to trim new walls automatically, and trim or extend existing walls manually. In addition, you can specify minor axis releases to the top or bottom part of panels. For more information, see the following instructions.

See also

Create concrete walls (page 237)

Create door and window openings (page 241)

Concrete wall extensions

Specify extensions

- Do one of the following:

To	Do this
Automatically trim a new wall back to the face of existing columns and walls	1. In the **Properties** window, select the **AutomaticExtension** option. 2. Create the wall.
Manually trim or extend existing walls	1. Select the wall that you want to trim or extend. 2. Go to the **Properties** window. 3. In the **End 1 extension** or **End 2 extension** field, define the desired extension. **NOTE** A positive extension extends the wall length beyond its insertion point. A negative extension trims the wall back from its insertion point.

Specify releases

1. Select the wall to be released.
2. In the **Properties** window, go to **Releases**.
3. In the **Minor Top** or **Minor Bottom** list, select the desired release:
   - **Fixed**
   - **Pinned**

- **Continuous (incoming members pinned)**: only available for FE meshed walls

TIP To specify a pinned connection to a supported slab, use an FE meshed wall and select the **Continuous (incoming members pinned)** option.

We recommend this option because the **Pinned** option also releases the wall panel above from the wall panel below. This may result in a mechanism during the analysis.

Create and modify wall supports

If needed, you can set Tekla Structural Designer to automatically generate support for walls. In addition, you can modify the degree of freedom of the wall support, and remove an unnecessary wall support. For more information, see the following paragraphs.

Automatic support generation

The **AutomaticGenerateSupport** option in wall properties controls whether a support is automatically created at the wall base level.

When the **AutomaticGenerateSupport** option is selected:

- If there are members or slabs underneath the wall capable of providing support, no support is generated.
- If there are no members or slabs underneath the wall capable of providing support, a support is generated.

When the **AutomaticGenerateSupport** option is cleared:

- If the **Generate support** option is selected, a support is generated.
- If the **Generate support** option is not selected, no support is generated.

When a support is required, Tekla Structural Designer forms it as follows:

- Under a meshed wall, Tekla Structural Designer creates a line support.
- Under a mid-pier wall, Tekla Structural Designer creates a point support.
- Under a bearing wall, Tekla Structural Designer creates a series of point supports.

When a support is required, its degrees of freedom are as specified in wall properties, under **Wall support**.

See also

Create concrete walls (page 237)

Modify wall support fixity

NOTE Supports can only be edited or deleted for both mid-pier and meshed concrete walls via the wall properties.

1. In the **Properties** window, expand **Wall support**.
2. Specify the degrees of freedom according to your needs.

   NOTE The discrete supports at each node are always angled in the global axis system, and not aligned with the wall major/minor axes.

   That is why you need to set both **Mx** and **My** to **Free**. This way, you can ensure that angled walls are pinned out of plane. This is not strictly necessary if the wall is aligned in global X or Y. In this case, you can set only **Mx** or **My** free, as appropriate.

   Similarly, set both **Mx** and **My** to **Fixed** in order to ensure that angled walls are fixed out of plane.

Remove a wall support

NOTE Supports can only be edited or deleted for both mid-pier and meshed concrete walls via the wall properties.

1. In the **Properties** window, clear the **Generate support** option.

   NOTE The **Generate support** option is only available for editing if the **AutomaticGenerateSupport** option is inactive. When **AutomaticGenerateSupport** is selected, **Generate support** is automatically cleared if members are created underneath a wall to support it. Similarly, **Generate support** is automatically reselected, if the supporting members are deleted.

Create door or window openings

Tekla Structural Designer allows you to create openings for both doors and windows in existing walls. For detailed instructions, see the following paragraphs.

RESTRICTION Openings are only active in meshed walls.

1. Open a frame view of the frame that contains the desired meshed wall.
2. On the **Model** tab, click **Wall Opening**.
3. Do one of the following:

- To define the opening relative to the bottom left corner of the wall, click the outline of an existing wall panel.
- To define the opening relative to the selected node, click a specific node in the model.

4. Click the first corner of the opening or press **F2** to define its exact position.
5. Drag the mouse pointer to the opposite corner of the opening.
6. Click the opposite corner of the opening or press **F2** to define its exact position.

   Tekla Structural Designer creates the opening.

See also

Create meshed or midpier concrete walls (page 237)

Meshed wall openings analysis model (page 242)

Meshed wall openings analysis model

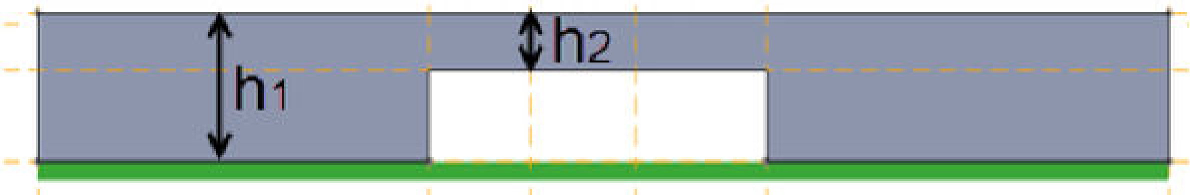
Limitations of wall openings

1. If you have specified a door or window opening in a wall panel you must model the wall using FE elements, otherwise a "Walls with openings have a mid-pier" validation error is displayed and the analysis will not proceed.
2. A "Panel contains openings - these are ignored in design" warning will always be issued when a wall containing openings is designed. When you encounter this warning, as well as taking stock of the design implications; you need also to consider if the analysis model is appropriate, as potentially it may not reflect your original intention. In certain situations the **Alternative model for wall openings** (described below) may prove to be a better solution.

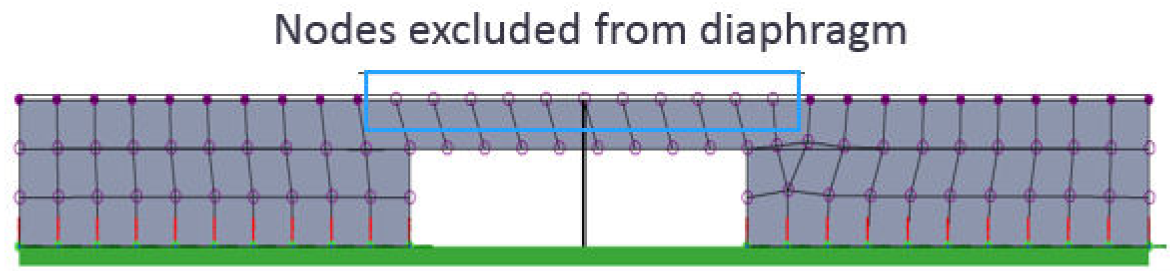
Analysis model applied to meshed wall panels with openings

If an opening is introduced in a meshed wall, the properties of the "lintel" wall beam directly above the opening are automatically adjusted in order to prevent the panel being unrealistically stiff. The adjustments that are applied are as follows:

• wall beam properties in the lintel use the lintel depth (h_2), rather than the panel depth (h_1)



• wall beam nodes in the lintel are removed from the slab diaphragm



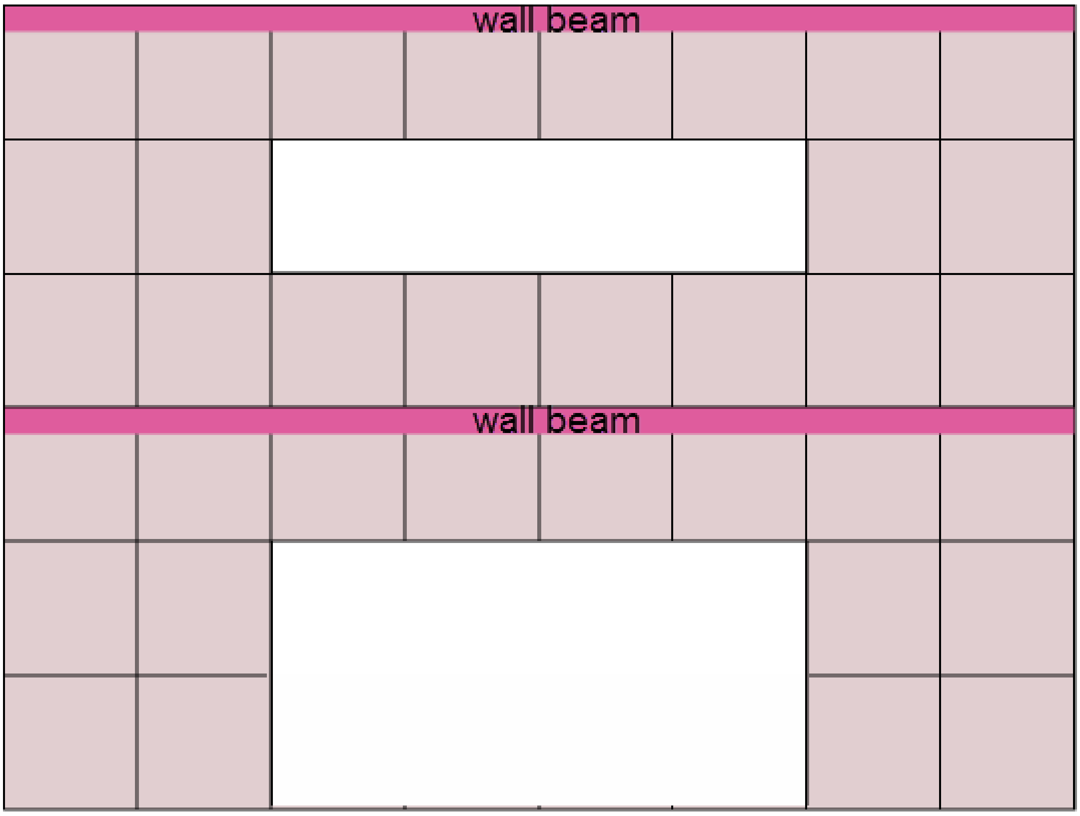
Modeling in this way the lintel becomes less stiff resulting in improved wall results when subject to gravity and lateral loading.

Alternative model for wall openings

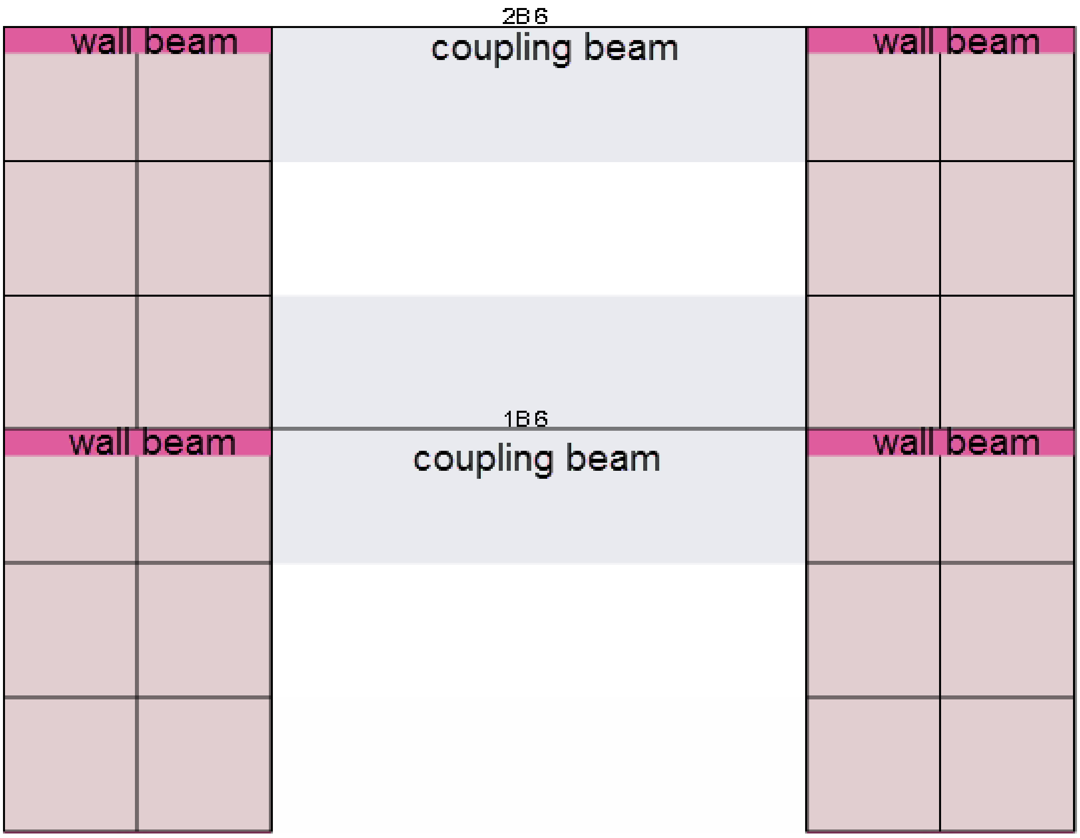
If the presence of an opening would form a beam like strip above or below the opening, you are advised to create separate wall panels to each side of the opening and then model the strip between the panels with a connecting beam ('coupling beam').

This method can be demonstrated by considering the below example, consisting of a two story wall with a large opening at each level.

If the openings were to be created as a window and door the resulting model would be as shown:



However, by separating the wall into discrete panels and inserting coupling beams you obtain an alternative model as below:



Such an idealisation enables the panels either side of the openings to be designed for their respective forces and enables the strips between the openings to be designed as beams.

Of course, this approach will require some additional detailing, but that would have been the case anyway had the openings been added and subsequently ignored by the design.

Create concrete cores

Tekla Structural Designer allows you to combine concrete wall panels, columns, and beams to form cores.

Overview of concrete cores

Concrete wall panels, columns, and beams can be combined to form cores for which overall analysis results are automatically calculated and available for review and output.

Such cores might then be used in a number of ways, for example:

- The engineer may wish to see/ output core forces to; better understand the distribution of lateral forces within the structure; for additional checks they wish to undertake; for overall core foundation forces or exclusive core reactions for cores on foundation mats.

- Cores provide a way of checking lateral forces per object type or SFRS type. This might be used when assessing buildings for some codes which require frames in dual systems to resist a minimum amount of seismic load.
- Adding all elements of the lateral system to a single core will give overall building vertical and lateral load, overturning moments or accumulated seismic torsion, in any direction.

Any number of walls (meshed and mid-pier), columns, and beams can be added to a core.

- Constituent objects of a core do not need to be physically attached.
- Once defined the core is listed in the structure tree and can be graphically selected as a single entity. When selected, its properties are displayed in the Property Window and Delete, Move, Copy and Mirror operations can be applied to it.

The directions of each core can be defined as any of; Dir 1/2 - Main Building Directions; Principal 1/2 - Major and minor local axis; Angle - (w.r.t.) Global Coordinate System.

The following core properties are automatically calculated/ displayed; Core section for each level; Core centroid and its coordinates above and below each level; a Core line with Local coordinate system (LCS) assigned and displayed at the centroid location; a Core support (for reporting purposes not analysis).

After analysis the following core results are available:

- **Core Line Forces** - 2D Integrated Results termed “Core Line” results for cores giving the following overall core forces; Axial Force, Major and Minor axis Moment and Shear, Torsion.
  - In the Results View via 2D Integrated Results > Core Lines, force diagrams of these results are displayed along the core line w.r.t. its LCS, with numerical values displayed at the top and bottom of each core stack and in the Tooltip when the result diagram is cursored over.
  - Results are available for all of; All Static Analyses including chase-down; RSA Seismic both for individual modes and Modal combination; Imposed/Live load reductions.
  - Tabular Results - Core results are also available in Tabular form via Analyze > Tabular Data and the associated Report option Analysis > “Core Line Forces”.

    NOTE Settings controls in the Member Forces and Wall Line Forces Report items allow you to optionally exclude columns/walls assigned to cores.

- **Core Reactions** - overall Reaction results are calculated for the core and can be viewed in the Results view via Reactions > Cores and output.

Create concrete core (assisted mode)

Cores are either created in **Assisted** or **Unassisted** mode. In assisted mode (which is the default), all potential members of a core are automatically highlighted when the cursor is moved over any of the constituent members, giving a preview of the core that can be created - a single click then adds all the highlighted walls to a new core.

1. On the **Model** tab, click **Cores**.
1. **Assisted Mode**
2. In the **Properties** window, select **Assisted**.
3. Select an appropriate wall panel or column and all highlighted members are added to the core.

   The mode switches to **Modify** to enable you to edit the core if required.
4. If no modifications are required, press Esc to finish.

NOTE Assisted Mode detects overlapping concrete walls, columns and coupling beams* that may potentially form a core.

*Beams for which the Construction property is set = "Coupling Beam". Note that no additional design checks are performed for this construction type so currently it serves only to mark beams; for consideration by the assisted mode core function; for which additional design checks beyond those currently made in the program may be required.

Create concrete core (unassisted mode)

1. On the **Model** tab, click **Cores**.
2. In the **Properties** window, unselect **Assisted**.
3. Select either a wall panel or column as the first member of the core.
4. Select the next wall panel, column, or beam to be part of the core.

NOTE You can drag the mouse pointer to select multiple members.

5. Continue selecting until all the members of the core have been chosen, and then press Esc to finish.

   Tekla Structural Designer combines the selected members into a single core.

Set the core axis direction

Analysis results are displayed with respect to the axis diection of each core. This can be aligned to the main building directions, the major and minor local axis, or to a specified angle.

1. Select the core.
2. In the **Properties** window, choose the rotation angle as required.

Modify concrete core

1. On the **Model** tab, click  **Cores**.
2. In the **Properties** window,
   a. Set **Mode** to **Modify**
   b. Select the **Active Core** as required.
   c. Set **Area Selection Mode** to **Set On** to add members to the core, or **Set Off** to remove members from the core.
3. Select a member to add or remove from the core.
4. Continue selecting or press Esc to finish.

   Tekla Structural Designer combines the selected members into a single core.

Dissociate concrete core

Existing cores can be disconnected if required.

1. Hover the mouse pointer over the core.
2. If a different entity is highlighted, press the <down arrow> cursor key until the required core reference is shown in the Select Entity tooltip.
3. Right click and select Dissociate Core.

Related video

Core modeling

Create bearing walls

Bearing walls provide resistance to vertical compressive loads (but not lateral loads) and support certain other member types. Unreinforced masonry walls, for example, can be modeled as bearing walls.

You can model bearing walls over several story heights. In these cases, Tekla Structural Designer creates a single wall with a uniform thickness between the base and top level.

Tekla Structural Designer determines the location of the wall from the alignment specified in the bearing wall properties, and the selected insertion points.

NOTE Bearing walls do not perform the same function as wall panels. In other words, bearing walls do not allow you to apply loads calculated by the **Wind Wizard** to your structure.

Therefore, in order to apply wind loads, you must create additional wall panels in the same locations as the bearing walls.

Material type

Three **Material types** are available for bearing walls:

- Concrete
- Timber
- General

Concrete Bearing Walls

There are two **Concrete types** available:

- Normal
- Lightweight

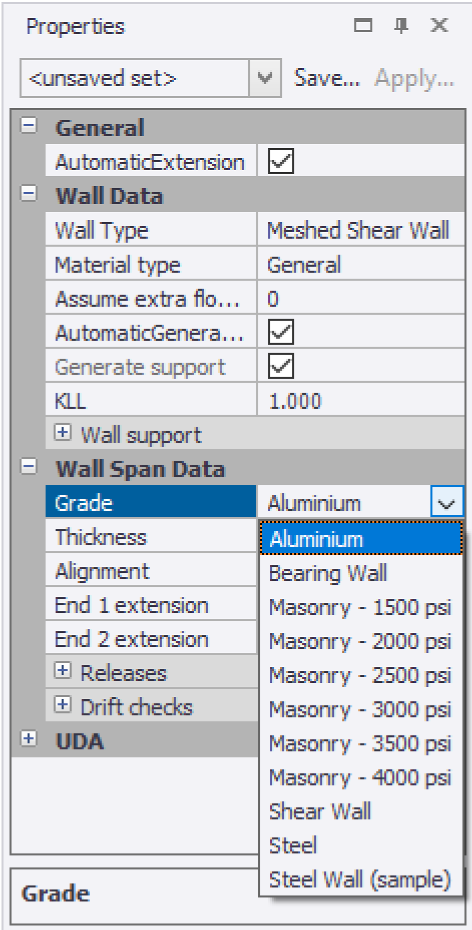
The **Grade** lists all the available grades in the Materials database under the current head code for the selected Concrete type.

Timber Bearing Walls

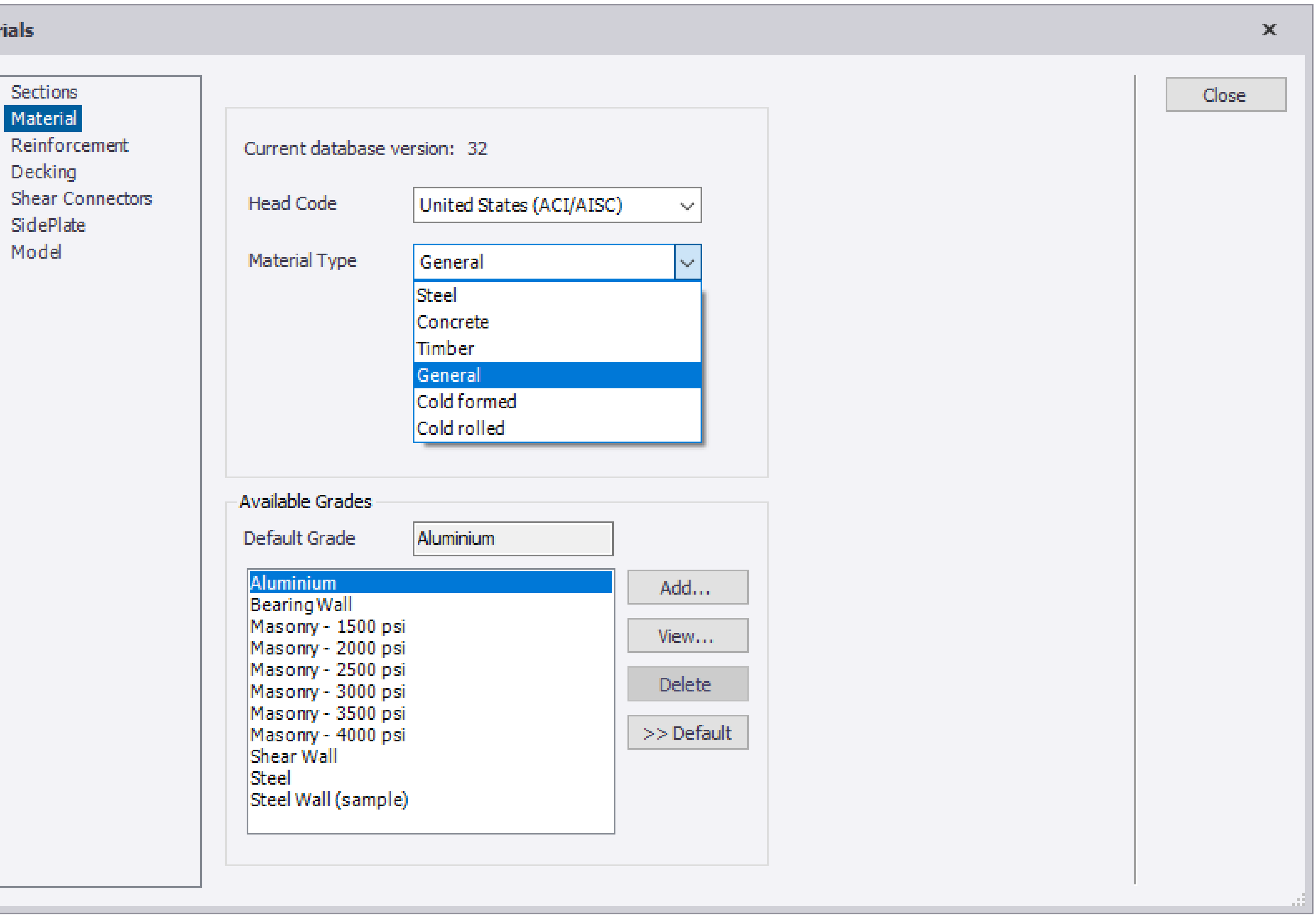
The **Grade** lists all the available grades in the Materials database under the current head code for Timber.

General Bearing Walls

The **Grade** lists all the available **General** materials in the Materials database under the current head code.



If the grade you want to use is not listed, you can open the Materials dialog and add the grade to the database (page 902), taking care to first select the General material type as shown below.



Geometric rules

- Bearing walls can only be created as rectangular in a vertical plane.
- Wall openings are ignored.
- Bearing walls cannot be included in cores.

Create bearing walls in a level view

NOTE Ensure that you have defined the construction levels between which the wall will run, and the grid points between which the wall will lie.

1. On the **Model** tab, click  **Bearing Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. Go to the **Properties** window.
3. If necessary, adjust that the base level and top level of the wall.
4. Adjust thickness and other wall properties according to your needs.
5. In the model, click the start point of the wall.
6. Click the end point of the wall.

   Tekla Structural Designer creates the wall between the selected start and end points.

Create bearing walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run and the grid points between which it will lie.

---

1. On the **Model** tab, click  **Bearing Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. In the **Properties** window, adjust the thickness and other wall properties according to your needs.
3. In the model, click the first corner of the wall.
4. Click the opposite corner of the wall.

   Tekla Structural Designer creates the wall between the selected points.

See also:

Bearing wall properties (page 1008)

How bearing walls are represented in solver models (page 588)

Create shear only walls

Shear only walls provide resistance to in-plane lateral loads only.

You can model shear only walls over several story heights. In these cases, Tekla Structural Designer creates a single wall with a uniform thickness between the base and top level.

Tekla Structural Designer determines the location of the wall from the alignment specified in the shear only wall properties, and the selected insertion points.

NOTE Requirements for these walls are that they must be; within a single bay (i.e. do not overlap one or more columns); strictly rectangular; vertical and surrounded by columns and beams (other than at the bottom edge).

NOTE Shear only walls do not perform the same function as wall panels. In other words, shear only walls do not allow you to apply loads calculated by the **Wind Wizard** to your structure.

Therefore, in order to apply wind loads, you must create additional wall panels in the same locations as the shear only walls.

Material type

The chosen material will only affect the weight of the wall. Whilst Shear only walls are most suited to masonry, three **Material types** are available:

- Concrete
- Timber
- General

Geometric rules

- Shear only walls can only be created as rectangular in a vertical plane.
- They must be totally enclosed by members (concrete or steel), except at the base.
- Wall openings are ignored.
- Shear only walls cannot be included in cores.

Create shear only walls in a level view

NOTE Ensure that you have defined the construction levels between which the wall will run, and the grid points between which the wall will lie.

1. On the **Model** tab, click **Shear Only Wall**.

   The wall will adopt the properties displayed in the **Properties** window.
2. Go to the **Properties** window.
3. If necessary, adjust the base level and top level of the wall.
4. Adjust stiffness, thickness, and other wall properties according to your needs.
5. In the model, click the start point of the wall.
6. Click the end point of the wall.

   Tekla Structural Designer creates the wall between the selected start and end points.

Create shear only walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run and the grid points between which it will lie.

1. On the **Model** tab, click **Shear Only Wall**.

   The wall will adopt the properties displayed in the **Properties** window.

2. In the **Properties** window, adjust the stiffness, thickness, and other wall properties according to your needs.
3. In the model, click the first corner of the wall.
4. Click the opposite corner of the wall.

   Tekla Structural Designer creates the wall between the selected points.

See also:

Shear only wall properties (page 1011)

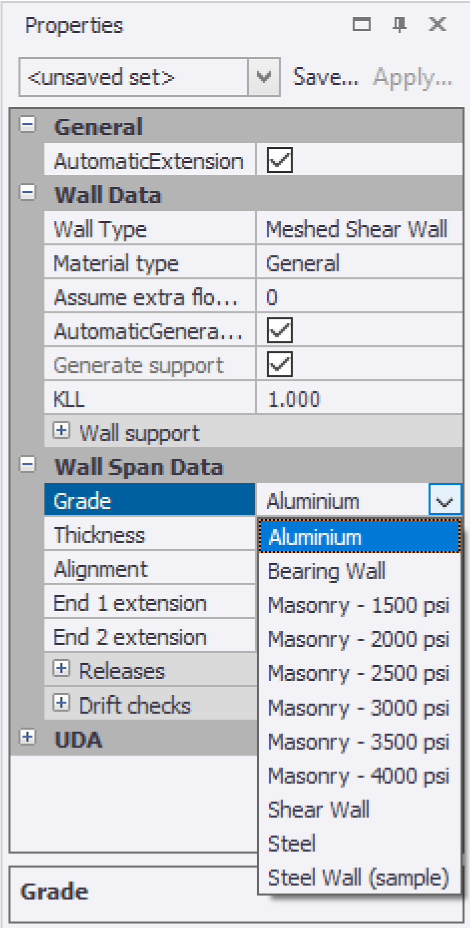
How shear only walls are represented in solver models (page 584)

Create general walls

You can create meshed general walls in level, frame, or structure views. For detailed information on creating general walls, see the following instructions.

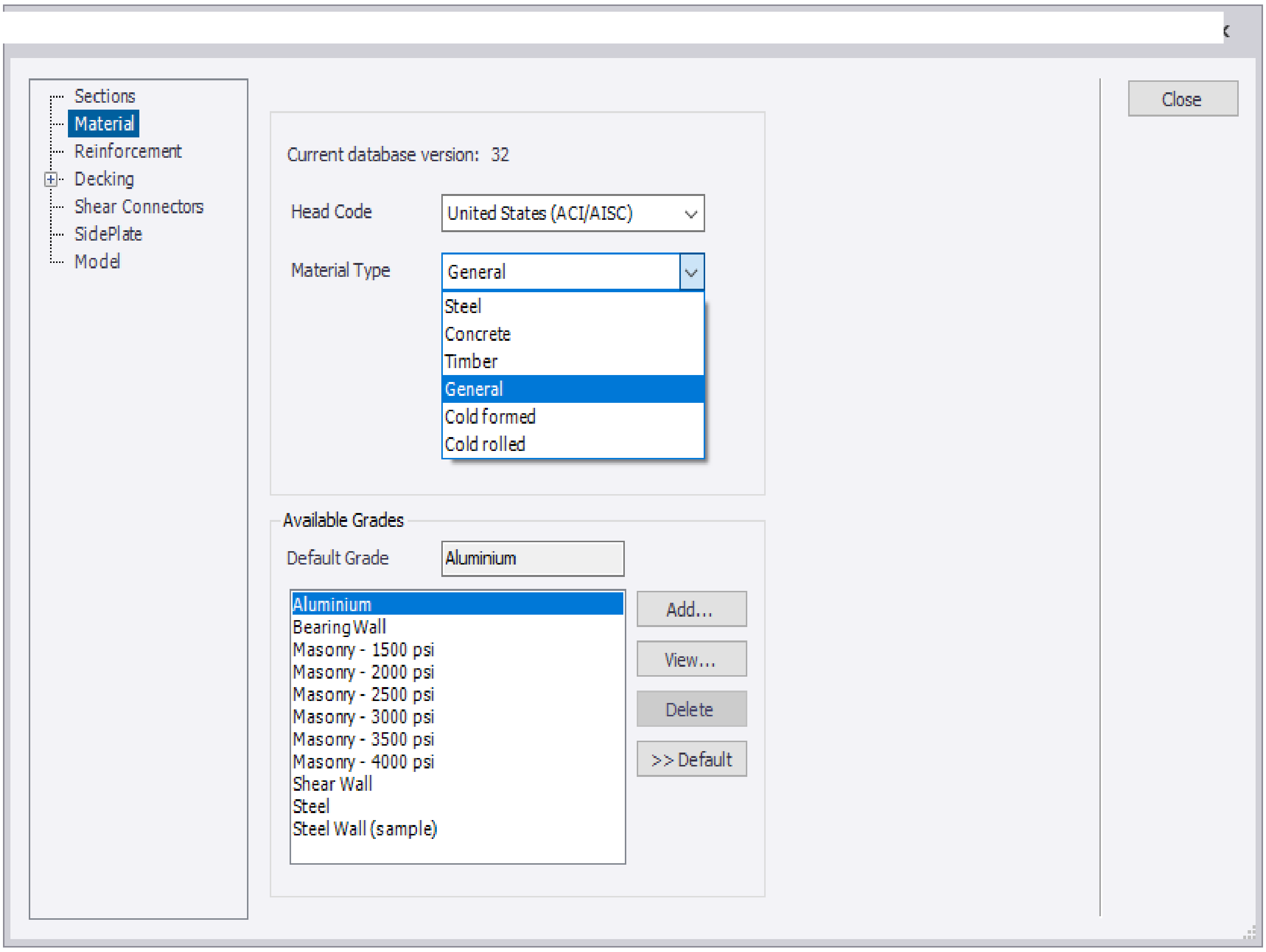
Material type

The **Grade** lists all the available **General** materials in the Materials database under the current head code.



NOTE Care should be taken when choosing the grade, as some general materials in the database may not be appropriate for meshed walls. Meshed walls resist loads in all directions and isotropic stiffness properties are assumed.

If the grade you want to use is not listed, you can open the Materials dialog and add the grade to the database (page 902), taking care to first select the General material type as shown below.



Geometric rules

- Meshed walls are defined as quadrilaterals in a single plane that can be vertical or sloping, (unlike mid-pier concrete walls which must be rectangular in a vertical plane).
- Openings (page 241) are permitted in meshed walls.
- The alignment and offsets in the wall properties are not structurally significant as they have no affect on the solver elements that are formed in the solver model.

- Concrete meshed walls can be included in cores, General meshed walls cannot.

Sub-division of meshed walls

Each wall object is split into separate panels only at those construction levels where an element or slab attaches to the wall. At the levels where the wall has been split, wall beam elements are also inserted.

Create rectangular walls in a level view

1. On the **Model** tab, click the arrow below **General Wall**.
2. Go to the **Properties** window.
3. Ensure the Material Type is set to General, then use the Grade property to select the general material type.
4. Adjust the top and base level for the wall, if necessary.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. In the model, click the start point of the wall.

    TIP If you are using a point along a member, do the following:

    a. Click the member to view its points.

    b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

7. Click the end point of the wall.

    Tekla Structural Designer creates the wall.

Create rectangular walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run, and the grid points between which it will lie.

1. On the **Model** tab, click the arrow below **General Wall**.
2. Go to the **Properties** window.
3. Ensure the Material Type is set to General, then use the Grade property to select the general material type.
4. Ensure the **Rectangular** property is selected.
5. Adjust the remaining properties, such as the wall thickness, if necessary.
6. Click a point to define a corner of the wall.

7. Click a point to define the opposite corner. Tekla Structural Designer creates the wall between the selected two points.

Create quadrilateral walls in a frame or structure view

NOTE In order to define a wall in a frame or structure view, you must have already defined the construction levels between which the wall will run, and the grid points between which it will lie.

1. On the **Model** tab, click the arrow below **General Wall**.
2. Go to the **Properties** window.
3. Ensure the **Rectangular** property is unselected.
4. Ensure the Material Type is set to General, then use the Grade property to select the general material type.
5. Adjust the top and base level for the wall, if necessary.
6. Adjust the remaining properties, such as the wall thickness, if necessary.
7. Click the point where the base of the wall should start.
8. Click the point where the base of the wall should end.
9. Click the point where the top of the wall should start.
10. Click the point where the top of the wall should end. Tekla Structural Designer creates the wall between the selected four points.

See also

General wall properties (page 974)

How meshed walls are represented in solver models (page 577)

Specify extensions and releases (page 239)

Create and modify wall supports (page 240)

Create door or window openings (page 241)

Create slabs and decks

This section focuses on the operations required to create slabs and decks.

- Overview of the slab model (page 259)
- Create slab items (page 263)
- Create slab or mat openings (page 265)
- Add overhangs to existing slab or mat edges (page 267)
- Apply curved edges to existing slab items (page 269)

- Create column drops (page 270)
- Specify the material for general slab types (page 270)
- Split and join slabs and mats (page 273)
- Modify slab/panel span direction (page 274)

See also

Modify slab items and panels by moving a node (page 171)

Modify slab items by moving an edge (page 172)

Overview of the slab model

Slabs allow you to decompose loads placed on a floor back to the supporting structure. You can create slabs in either levels or inclined planes. For more information, see the following paragraphs.

Slabs and slab items

Tekla Structural Designer stores slab data in the form of parent slabs, each consisting of one or more individual panels, called slab items. Slab items can be either connected or separated from each other. However, they must be on the same level.

Every parent slab has a unique name. A slab name used at one level can not be re-used at a different level.

Every parent slab has general properties of slab type (and associated deck type) as follows:

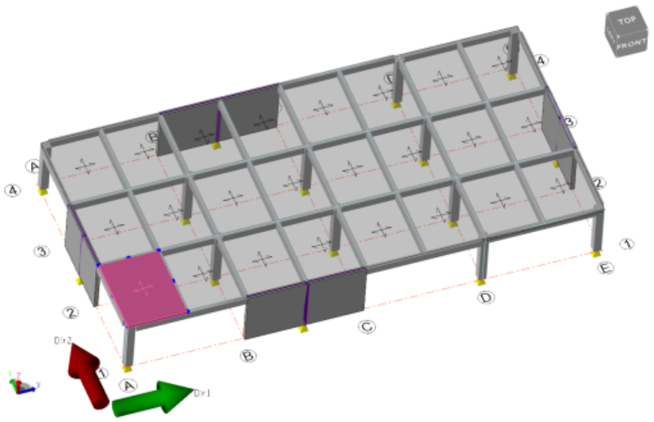
- Slab on beams
    - Reinforced concrete
    - Post tension
- Flat slab
    - Reinforced concrete
    - Post tension
- Precast
    - Precast concrete planks
- Composite slab
    - Profiled metal decking
- General
    - Steel plate
    - Timber
    - General

Every parent slab also has a load decomposition property:

- One-way
- Two-way

Slab items initially inherit their properties from the parent slab. However, once the slab items have been created, you can modify them to amend certain properties: you can change the rotation angle, override the slab depth, or exclude a panel from the diaphragm. Once you have overridden the slab depth, you can also apply a vertical offset in order to model a slab step.

In the view below, there are 24 slab items grouped together in one slab.



Tekla Structural Designer sets some data on the slab level and is common to all slab items, while other data is set at the slab item level. In simple overview terms, the data breakdown is as follows:

Slab data:

- Slab type
- Deck type
- Decomposition
- Thickness
- Vertical offset
- Material properties
- Analysis settings
- General design settings

Slab item data:

- Cover
- Reinforcement information
- Specific design settings

Therefore, you can create slabs over a wide area. While creating slabs, you do not have to consider sub-sections of the slab - you can simply create one big expanse of slab.

When it comes to design, you need to conceptualize the slab as a series of design panels, or slab items. Each slab item will have its own design settings and its own design results. You can select different reinforcement in different panels. You also have to consider pattern loading, where some panels are loaded, and others not.

When results are later presented in calculations and drawings, you can specifically reference the design panels.

Comparison of slab types

The different slab types available in Tekla Structural Designer are compared in the below table.

	Slab on beams	Flat slab	Precast	Composite	General
Deck Type	• Reinforced concrete	• Reinforced concrete	• Precast concrete planks	• Profiled metal decking	• steel plate • timber • general
	See: Create slab items (page 263)				
Decomposition	• 1-way • 2-way	• 2-way	• 1-way	• 1-way	• 1-way • 2-way
	See: Overview of one-way and two-way load decomposition (page 385)				
Diaphragm action	• rigid • semi-rigid • none	• rigid • semi-rigid • none	• rigid • semi-rigid • none	• rigid • semi-rigid • none	• rigid • semi-rigid • none
	See: Overview of diaphragm action in roof panels and slabs (page 482)				
Load	• all panel load types	• all panel load types	• all panel load types	• all panel load types	• all panel load types
	See: Apply panel loads (page 364)				
Modification factors for 2-way spanning slabs	• yes	• yes	• n/a	• n/a	• yes
	See: Analysis Settings>Modification factors (page 1061)				

<table>
<tr><th></th><th>Slab on beams</th><th>Flat slab</th><th>Precast</th><th>Composite</th><th>General</th></tr>
<tr><td rowspan="2">Meshed in 3D analysis and grillage chasedown models</td><td>• 1-way: no
• 2-way: optional</td><td>• optional</td><td>• no</td><td>• no</td><td>• 1-way: no
• 2-way: optional</td></tr>
<tr><td colspan="5">See: Define whether slabs are meshed for 3D building analysis and grillage chasedown analysis (page 464)</td></tr>
<tr><td rowspan="2">Meshed in FE Chasedown analysis</td><td>• 1-way: no
• 2-way: yes</td><td>• yes</td><td>• no</td><td>• no</td><td>• 1-way: no
• 2-way: yes</td></tr>
<tr><td colspan="5">See: Manage FE meshed slabs (page 464)</td></tr>
<tr><td rowspan="2">Designed in Tekla Structural Designer</td><td>• 1-way - beyond scope
• 2-way- yes</td><td>• yes</td><td>• beyond scope</td><td>• beyond scope</td><td>• beyond scope</td></tr>
<tr><td colspan="5">See: Design slabs and run punching shear checks (page 616)</td></tr>
</table>

Vertical alignment

The vertical alignment of all slabs in a given level is determined by the construction level type specified in the Construction Levels dialog. The possible types are:

- **T.O.S.** (Top Of Steel): the base of each slab item aligns with the level.
- **S.S.L.** (Structural Slab Level): the top of each slab item aligns with the level.
- **T.O.F.** (Top Of Foundation): the base of each slab item aligns with the level.

Vertical offsets

A vertical offset can be specified after checking ‘Override slab depth’ in the slab item properties.

These offsets are not structurally significant in slabs as they have no effect on the analysis mesh position relative to the top of the slab in the solver model. However, the mesh properties would reflect any change in the slab thickness.

Slab edges

Slab edges can be either straight or curved.

See: Apply curved edges to existing slab items (page 269)

Slab openings

You can create openings of any shape within slabs. These are only considered when the slab is meshed for load decomposition or analysis.

See: Create slab or mat openings (page 265)

Column drops

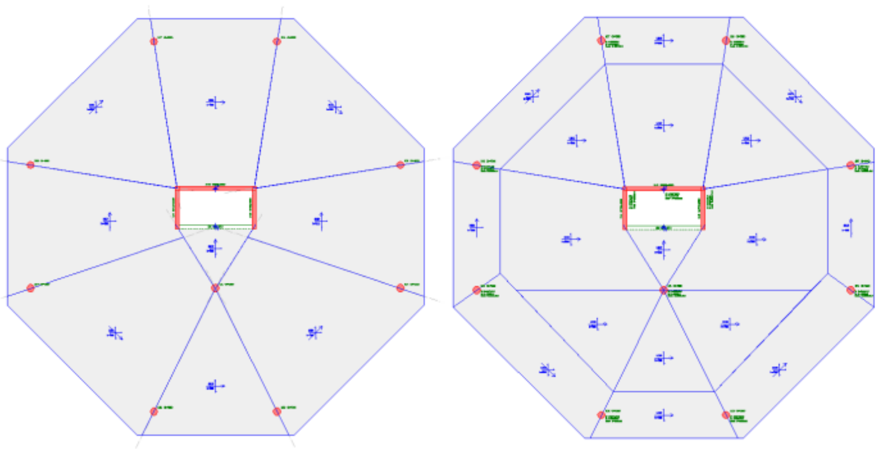
Column drop panels are slab items with an increased thickness. In flat slabs, at points where the slab is supported by columns, you can use column drops to thicken the slab.

See: Create column drops (page 270)

Panel sub-division

Regardless of how the slabs and slab items are initially created, you can further divide or re-form them with the **Slab Split** and **Slab Join** commands. There are several reasons why you may choose to use the previously mentioned commands, related to adding steps, loading patterns, and designing panels.

For flat slabs in particular, the way that slabs are split for the purposes of pattern loading is a matter of engineering judgement. The views below view two options that two different engineers might both justifiably choose for the same slab perimeter.



See: Split and join slabs and mats (page 273)

Create slab items

Slab items are individual slab panels that form parent slabs. You can create slab items either by bay, or by points. For more information, see the following instructions.

Select the slab type and specify slab properties

1. On the **Model** tab, click the arrow under  **Slabs**.
2. In the list that appears, select the desired slab type.
3. In the **Properties** window, define the slab item parameters for the selected slab type.
4. Ensure that the **Slab** and **Select bays** properties are set according to your needs.

Specify the parent slab to which the slab item belongs

1. In the **Properties** window, go to the **Slab** property.
2. Click the arrow on the right side of **Slab**.
3. In the list that appears, select whether you want to:
   - Create a new slab
   - Add to existing slab

     NOTE When you select this option, at the point of creating the slab item, Tekla Structural Designer checks if there is a existing slab in the plane with same properties.

     If Tekla Structural Designer finds an identical slab, the new slab item will be added to it.

     If Tekla Structural Designer finds two identical slabs, the closest one is used.

     If Tekla Structural Designer does not find an identical slab, a new parent slab is created.

   - Manually select the parent slab

     NOTE This option is only available in 2D views after one or more slab items have been placed in that view.

4. When the slab item properties are defined, you can place the item either by bays or by picking points.

Create slab items by bay

You can only create slab items by bay in 2D Views.

1. In the **Properties** window, ensure that the **Select bays** option is selected.
2. Do one of the following:

To	Do this

To add an individual slab item	a. Click the outline of a bay bounded by beams or walls.
To add slab items into all bounded bays	• Drag a box that encompasses the bays. **NOTE** Dragging the box from left to right places items in those bays totally enclosed by the box. Dragging the box from right to left places items in all bays that are either enclosed by the box, or cross it. Holding down **Shift** while dragging creates a line instead of a box. Tekla Structural Designer places slab items in all bays that cross the line.

Create slab items by points

You can create slabs in both 2D and 3D Views.

1. In the **Properties** window, ensure that the **Select bays** option is cleared.
2. Click the start point of the slab item.
3. Click other points that define the slab item outline.
4. Double-click the final point to create the slab item.

Create slab or mat openings

When you have created slabs or mats, you can create different kinds of openings to them according to your needs. You can define rectangular, circular, and irregular openings. For more information on how to create and delete slab and mat openings, see the following instructions.

Simple openings

You can quickly define simple openings within existing slabs. Simple openings are rectangular or circular in plan.

The **Slab Opening** command is located on the list in the **Slabs** group. The command is only active in 2D views.

Slab openings can:

- Cross more than one slab item or slab
- Be overlaid or joined to create openings which together have shapes other than rectangular
- Cut across a stepped edge
- Be applied to level an sloping slabs

Openings cannot:

- Be applied to one-way spanning slabs

- Reside within or cut a column drop

Irregular Openings

Alternatively, you can create more complex openings by using construction lines and constructing slab items around an irregular shape.

Create rectangular openings

1. Open a 2D view of the level containing the slab item or mat panel within which you want to create an opening.
2. Do one of the following:

To	Do this
Create slab openings	a. On the **Model** tab, click the arrow on the top right corner of the **Slabs** group. b. In the list that appears, select **Slab Opening**.
Create mat openings	• On the **Foundations** tab, click **Mat Opening**.

The opening properties are viewed in the **Properties** window.

3. In the **Properties** window, set the **Opening Type** to **Rectangular**.
4. If necessary, specify a rotation angle to rotate the opening on plan.
5. Click within the outline of an existing slab item or mat panel to define the first corner of the opening, or press **F2** to define its exact position.
6. Drag the mouse pointer to the opposite corner of the opening.
7. Click the opposite corner of the opening, or press **F2** to define its exact position.

Tekla Structural Designer creates the opening.

Create circular openings

1. Open a 2D view of the level containing the slab item or mat panel within which you want to create an opening.
2. Do one of the following:

To	Do this
Create slab openings	a. On the **Model** tab, click the arrow on the top right corner of the **Slabs** group. b. In the list that appears, select **Slab Opening**.

Create mat openings	• On the **Foundations** tab, click **Mat Opening**.

The opening properties are viewed in the **Properties** window.

3. In the **Properties** window, set the **Opening Type** to **Circular**.
4. Click within the outline of an existing slab item or mat panel to define the center of the opening, or press **F2** to define its exact position.
5. Drag the mouse pointer to define the radius of the opening, or press **F2** to define the exact radius.

   Tekla Structural Designer creates the opening.

Delete slab or mat openings

1. In the **Structure** tree, open the **Slab Openings** branch.
2. Right-click the name of the slab or mat opening that you want to delete.
3. In the context menu, select **Delete**.

   Tekla Structural Designer deletes the opening.

Add overhangs to existing slab or mat edges

At times, the edge of a slab or mat may extend beyond a grid line, either to the edge of the beam which supports the edge of the slab, or around the perimeter of the building to meet the inside face of the cladding. In order to take these cases in to account, Tekla Structural Designer allows you to define overhangs to the edges of a slab. An overhang may extend across many slab items in one slab and can be curved, or tapered if required.

Any loads that you define over an overhang are included in the total loading on your building.

You can only create overhangs in 2D views.

Add an overhang along the full length of a supporting beam

NOTE This method can only be used if a beam exists along the entire slab/mat edge to which the overhang is to be added.

1. Open a 2D view of the level containing the slab/mat for which you want to create the overhang.
2. Do one of the following:

To	Do this
Create an overhang over a supporting beam to a slab edge	a. On the **Model** tab, click the arrow in the top right corner of the **Slabs** group.

	b. In the list that appears, select **Slab Overhang**. c. In the **Properties** window, check **LengthOfBeam**. d. If necessary, in the **Properties** window, modify the width and other properties of the overhang. e. Click on the beam over which the overhang is to extend.
Create an overhang over a supporting beam to a mat edge	• On the **Foundations** tab, the arrow under **Mat Opening**. • In the list that appears, select **Mat Overhang**. • In the **Properties** window, check **LengthOfBeam**. • If necessary, in the **Properties** window, modify the width and other properties of the overhang. • Click on the beam over which the overhang is to extend.

Tekla Structural Designer creates the overhang along the length of the beam.

Add an overhang to a slab or mat edge between two points

NOTE In order to define a slab overhang between two points, you must have already defined the slab to which it applies.

1. Open a 2D view of the level containing the slab/mat for which you want to create an overhang.
2. Do one of the following:

To	Do this
Create an overhang to a slab edge	a. On the **Model** tab, click the arrow in the top right corner of the **Slabs** group. b. In the list that appears, select **Slab Overhang**.
Create an overhang to a mat edge	• On the **Foundations** tab, the arrow under **Mat Opening**. • In the list that appears, select **Mat Overhang**.

The slab or mat overhang properties are viewed in the **Properties** window.

3. Ensure the **LengthOfBeam** property is unchecked.

4. If necessary, in the **Properties** window, modify the width and other properties of the overhang.
5. Click along the edge of an existing slab item or mat panel to define the start point of the overhang, or press **F2** to define its exact position.
6. Click along the same edge to define the end point of the overhang, or press **F2** to define its exact position.

   Tekla Structural Designer creates the overhang between the two points.

Add a curved overhang to a slab or mat edge

While following one of the above procedures to add an overhang along the length of a supporting beam, or between two points:

1. In the **Properties** window, uncheck the **Edge parallel** property.
2. In the **Properties** window, define the **Curvature** of the overhang.
3. Continue to create the overhang in the normal way.

Add a tapered overhang to a slab or mat edge

While following one of the above procedures to add an overhang along the length of a supporting beam, or between two points:

1. In the **Properties** window, check the **Tapered** property.
2. Use the **Width1** property to define the length of overhang at end 1.
3. Use the **Width2** property to define the length of overhang at end 2.
4. Continue to create the overhang in the normal way.

Apply curved edges to existing slab items

You may sometimes need to create slab items that have curved edges instead of linear ones. For detailed instructions to do so, see the following paragraphs.

1. Select the slab items that you want to modify.
2. According to you needs, do one of the following:

To	Do this
Apply the same degree of curvature to all selected slab edges	a. In the **Properties** window, go to **All edges**. b. Clear the **Linear** option. c. Define the required curvature. A positive value creates an inward curve, whereas a negative value creates an outward curve.
Apply curvature to a specific edge	a. In the **Properties** window, go to the properties of the required edge. The edges are numbered. b. Clear the **Linear** option.

	c. Define the required curvature. A positive value creates an inward curve, whereas a negative value creates an outward curve.

Tekla Structural Designer redraws the slab item with the specified curvature.

Create column drops

In order to increase punching resistance, you can insert drop panels, or column drops, within concrete slabs at points where they are supported by columns. Column drops are slab thickenings that can be positioned above the slab, below the slab, or both. Column drops are rectangular in plan, and aligned to the column axes. Column drops are always positioned centrally over the supporting column.

The extent of the column drop is limited by the extent of the slab boundary. The cut back drop shape may not be rectangular.

NOTE In order to define a slab drop you must have already defined the concrete slab to which it applies.

1. On the **Model** tab, click the arrow on the top right corner of the **Slabs** group.
2. In the list that appears, select **Column Drop**.

   The column drop properties are viewed in the **Properties** window.
3. If necessary, in the **Properties** window, modify the properties of the column drop.
4. Click an existing column connected to a concrete slab panel to create a single column drop, or drag a box around multiple columns to create a series of column drops.

   NOTE To insert the column drop correctly, ensure that the type of the construction level containing the slab is **S.S.L.**, not **T.O.S.**

Specify the material for general slab types

General slab types can have steel plate, timber, or general deck types. The material grade properties for each deck type can either be selected from the material database or they can be user-defined.

Choosing the grade from the material database

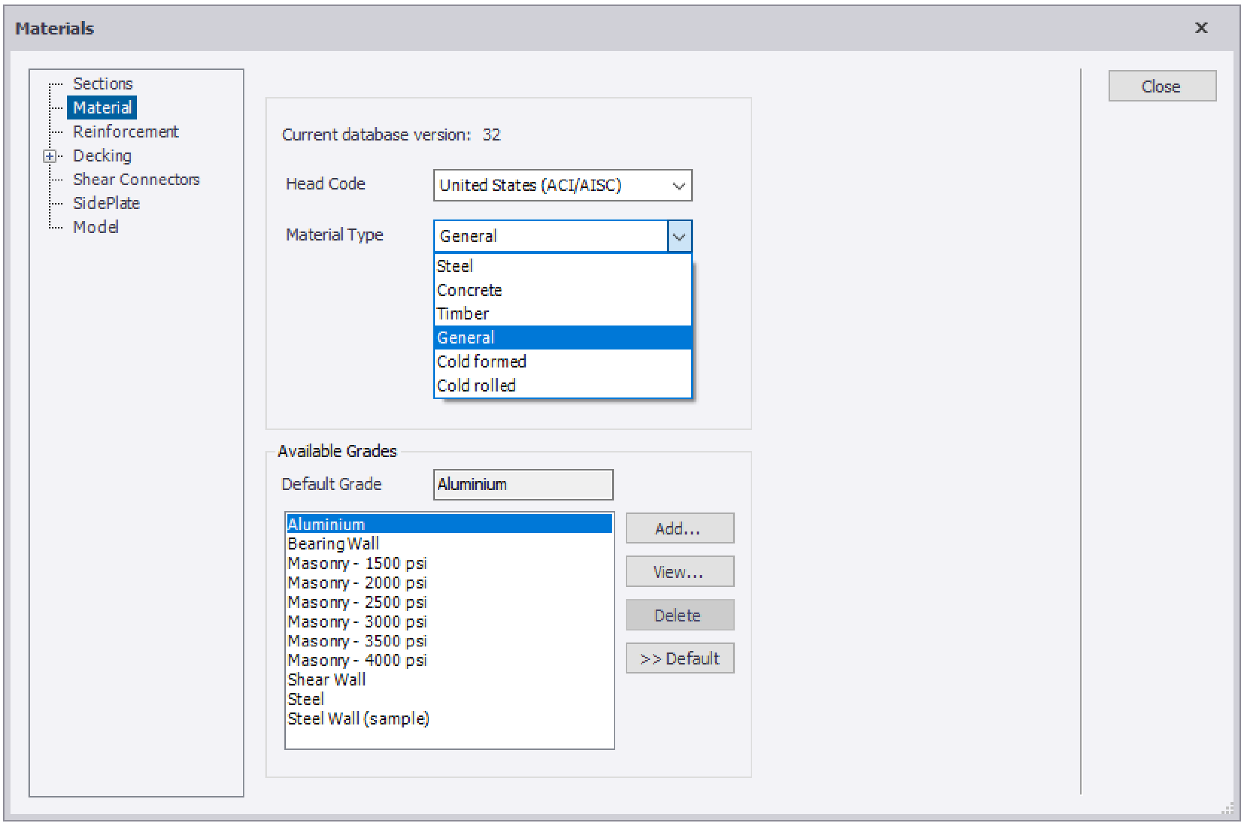
If the **Deck type** is set to:

- **Steel plate** - the **Grade** lists all the available **Steel** materials in the Materials database under the current head code.
- **Timber** - the **Grade** lists all the available **Timber** materials in the Materials database under the current head code.

> **NOTE** When using a timber deck type, the program will initially default to a material override option rather than displaying a Grade. This is because the Poisson’s ratio in the timber materials database is generally invalid for isotropic elastic analysis (of 2D elements), for which it must be > 0 and < 0.5. You can disable the override to select a timber grade, but you must then enable it again to enter a valid Poisson’s ratio (other grade properties such as density and modulus of elasticity will be retained).

- **General** - the **Grade** lists all the available **General** materials in the Materials database under the current head code.

If the general material grade you want to use is not listed, you can open the Materials dialog and add the grade to the database (page 902), taking care to first select the General material type as shown below.



Override material properties option

Unlike other slab types, general slab types each have an option to override material properties. This option is provided primarily to allow the existing properties of steel or timber deck slabs created in older versions of Tekla Structural Designer (pre 2020 SP5) to be retained, but it can also be used to enter the material properties directly if required.

NOTE When creating new timber decks, the program will default to the override option. This is because the Poisson’s ratio in the timber materials database is generally invalid for isotropic elastic analysis (of 2D elements), for which it must be > 0 and < 0.5. You can disable the override to select a timber grade, but you must then enable it again to enter a valid Poisson’s ratio (other grade properties such as density and modulus of elasticity will be retained).

Split and join slabs and mats

Tekla Structural Designer allows you to both sub-divide existing slab items into smaller items, and merge slab items into larger items.

NOTE The commands for splitting and joining slab items are only accessible in 2D views.

Split slab and mat items

1. Do one of the following:

To	Do this
Split slabs	• On the **Model** tab, click **Slab Split**.
Split mats	• On the **Foundations** tab, click **Mat Split**.

2. Hover the mouse pointer over the slab or mat edge, grid point, or other point where you want to start the split.

   NOTE The points used to define the cut line can be outside the boundary of the slabs being split. Thus, they do not need to be on the slab edges.

3. Click the start point of the split.
4. Click the second point of the split on either the same or other item or panel.
5. Do one of the following:
   - To continue the split, click subsequent split points.
   - To end the split, click the second point of the split again.

   Tekla Structural Designer splits any slab items entirely crossed by the split along the cut line.

Join slab and mat items

RESTRICTION You can only join slab items that share a common edge.

1. Do one of the following:

To	Do this
Split slabs	• On the **Model** tab, click **Slab Join**.

Split mats	• On the **Foundations** tab, click **Mat Join**.

2. Select the first of the slab items (the master item) that you wish to join.

   NOTE The slab items selected later will adopt the properties of the master item.

3. Select the second slab item.

   Tekla Structural Designer joins the selected slab items to create a new one.

4. Select additional slab items as required, or press **Esc**.

Modify slab/panel span direction

The span direction of slab items or wall/roof panels can be edited via the **Properties** window. The span direction of individual slab items and wall/roof panels can also be edited graphically.

NOTE Provided SpanDirection is switched on in Scene Content, the rotation angle/X direction is represented by a blue arrow.

Modify a single span direction graphically

The span direction can be edited graphically , in both 2D and 3D views.

1. Select the slab/panel item.
2. Select the span direction symbol. A yellow command prompt is displayed and the **Properties** window lists the X direction mode options of **Align with edge** and **Define by 2 points**
3. Select the desired mode.
4. Select either a slab edge (of any slab item) or two points, depending on the selected mode.

   The slab direction is automatically set to the identified angle.

Modify span direction via the Properties window

1. Select the slab/ panel item(s).
2. In the **Properties** window enter the required rotation angle/X direction. The slab span direction is updated to the new angle.

Create trusses and joists

These topics introduce you to the methods of creating trusses and joists.

We recommend you familiarize yourself with how to:

- Create trusses (page 275)
- Create steel joists (page 277)

Create trusses

Trusses are particular arrangements of members that Tekla Structural Designer calculates automatically for you. Once you have created a truss, you can copy the truss throughout your model as necessary. In addition, you can pick a truss in your model, and move it to a desired location.

Create a truss

The **Truss Wizard** helps you to create trusses in your models. For detailed instructions on how to use the **Truss Wizard** to create trusses, see the following instructions.

1. On the **Model** tab, click **Steel Truss**, or **Timber Truss** according to your needs.

   The **Truss Wizard** opens.
2. Select the truss shape.
3. In the model, click the start point of the truss.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

   ---
4. In the model, click the end point of the truss.
5. In the **Truss Wizard**, click **Next**.
6. Specify the truss alignment parameters.
7. Click **Next**.
8. Specify the truss details.
9. Click **Finish**.

Create a space truss

Tekla Structural Designer allows you to create linear and planar space trusses. You can create space trusses in a 2D or 3D view. For more information, see the following instructions.

1. On the **Model** tab, click the arrow next to **Steel Truss**, or **Timber Truss**, according to your needs.

2. In the list that appears, select **Space**.

   The **Space Truss Wizard** opens.
3. Define the truss type, the alignment, and the number or bays.
4. Click **Next**.
5. In the model, click the truss location points:
   - For a linear truss, click the start and end points.
   - For a planar truss, click the four corners of the truss.
6. In the **Space Truss Wizard**, specify the truss width and height.

   If you are creating a planar truss, Tekla Structural Designer calculates the width automatically.
7. Click **Next**.
8. Define whether the truss should be straight or curved.
9. Click **Finish**.

Create a free form truss

You can create a free form truss by placing a series of truss members in the required shape. For more information, see the following instructions.

RESTRICTION	Free form trusses cannot be created in a 3D view, or a 2D level view.

1. Open the 2D frame view in which you want to create the truss.
2. On the **Model** tab, click the arrow next to **Steel Truss**, or **Timber Truss**, according to your needs.
3. In the list that appears, select **Free Form**.
4. Click the start point of the first truss member.
5. Click the end point of the first truss member.
6. Repeat steps 4 and 5 to place each truss member as required.
7. When the truss geometry is complete, press **Esc** to finish.

Modify the geometry of existing steel and space trusses

After creating a steel, timber, or space truss, you can modify their geometry by using the **Edit** command. For more information, see the following instructions.

1. Hover the mouse pointer over the desired truss so that it becomes highlighted.
2. Right-click the truss.

3. In the context menu, select **Edit [element name]**.

   Depending on the type of the truss, either the **Truss Wizard** or the **Space Truss Wizard** opens.
4. Modify the truss properties according to your needs.
5. To save the changes, click **Finish**.

Modify the properties of existing trusses

You can modify the section sizes, material grades, section orientations, and other truss properties in the **Properties** window. For more information, see the following instructions.

1. Move the mouse pointer over the desired truss, so that it becomes highlighted.
2. Click the truss.
3. In the **Properties** window, modify the truss properties according to your needs.

Create steel joists

Steel joists, or bar joists, are a specific type of members used in the United States. They are simply supported secondary members that, in turn, support slab and roof loads. Steel joists are constrained to standard types specified by the US Steel Joist Institute, and standardized in terms of span, depth and load carrying capacity. To create steel joists in your model, see the following paragraphs.

Specify the section for Standard and Special joist types

1. On the **Model** tab, click **Steel Joist**.
2. In the **Properties** window, specify the **Steel joist type**
3. Click the arrow on the right side of **Steel joist type**.
4. In the list that appears, select either **Standard** or **Special** as required.
5. Select the **Section** parameter.
6. Click the arrow on the right side of **Section**.
7. In the list that appears, select **<New\Edit...>** The **Select Section** dialog box opens.
8. Select the desired joist type and size.
9. Click **Select**.
10. In the **Properties** window, adjust the remaining parameters according to your needs.

Specify the section for the Girder joist type

1. On the **Model** tab, click **Steel Joist**.
2. In the **Properties** window, select the **Steel joist type** parameter.
3. Click the arrow on the right side of **Steel joist type**.
4. In the list that appears, select **Girder**.
5. Select the **Section** parameter.
6. Click the arrow on the right side of **Section**.
7. In the list that appears, select **<New\Edit...>** The **Girder Section** dialog opens.
8. Either:
   - Enter the girder properties and then click **OK**.
   - Click **Add** if you want to add another girder section.
9. In the **Properties** window, adjust the remaining parameters according to your needs.

Create a steel joist

1. Select the joist type and size.
2. Click the start point of the joist.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.
   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

3. Click the end point of the joist.

NOTE The ends of steel joists can be on different levels.

Move a steel joist

You can modify the joist position in both 2D and 3D views.

1. Select the joist.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

3. Click the grid or construction point where you want to move the end node.

   Tekla Structural Designer moves the end node to the selected point.

Create portal frames

Tekla Structural Designer allows you to create both single-span and multi-span portal frames. After creating the portal frame, you can modify the properties of either the entire portal frame, or a portal frame member. For more information, see the following paragraphs.

Create a single or multi-span portal frame

1. On the **Model** tab, click **Portal Frame**.
2. In the **Properties** window use **Span Count** to specify the number of spans.
3. Click the start point of the first span of the portal frame.

   The start point sets the local X and Y coordinate origin for the portal frame.
4. Click the end point of the last span of the portal frame.

   The start point defines the positive local X axis direction for the portal frame.

   NOTE The end point must lie in the same construction level as the start point.

   The **Portal Frame** dialog box opens.
5. On the **Spans** page, select the first span and click **Edit...**
6. Define the eaves level, select the frame type, and complete the frame geometry for the span.
7. Click **OK**.
8. According to your needs, do one or all of the following:
   - On the **Rafters** page, review the rafter section sizes.
   - On the **Columns** page, review the column section sizes.
   - On the **Haunches** page, specify eaves and apex haunches. See:Portal frame haunch geometry (page 281)
   - On the **Bases** page, specify the base fixity, and adjust the base levels according to your needs.

NOTE You can also specify bases as *nominally pinned* or *nominally fixed* if required, as follows:

a. close the **Portal Frame** dialog box,

b. open the appropriate **Support** in the Portal Frame **Properties** window,

c. set the Rotational stiffness x **Type** as *nominally pinned* or *nominally fixed*,

d. enter the stiffness %

For more details, see: Partial fixity of column bases (page 316).

- Use the **Valleys**, **Ties**, **Tie Members**, **Parapets**, and **Parapet Members** pages to specify any additional elements.

9. To create the portal frame, click **OK**.

Modify the properties of an existing portal frame

- According to your needs, do one of the following:

To	Do this
Modify the overall frame properties	1. Hover the mouse pointer over the desired portal frame, so that it becomes highlighted. 2. Right-click the portal frame. 3. In the context menu, select **Edit [element name]**. The **Portal Frame** dialog box opens. 4. Modify the properties according to your needs. 5. Click **OK**.
Modify the properties of an individual portal frame member	1. Hover the mouse pointer over the desired portal frame member, so that it becomes highlighted. 2. In the **Select Entity** tooltip, scroll until the desired member is highlighted. 3. In the model, right-click the member. 4. In the context menu, select **Edit [element name]**. The **Properties** dialog box opens. 5. Modify the properties according to your needs. 6. Click **OK**.

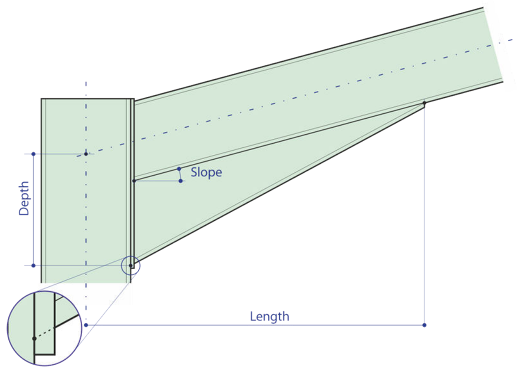
Add copy or mirror spans in an existing portal frame

1. Open the **Portal Frame** dialog box (as described in the above topic).

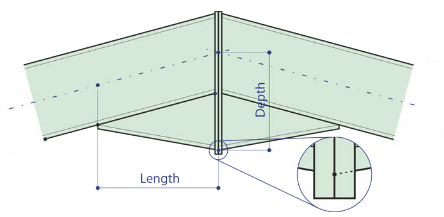
2. Go to the **Spans** page of the **Portal Frame** dialog box.
3. Use the available buttons to introduce additional spans:
   - To introduce a new span at the end of the frame, click **Add**.
   - To insert a new span below the currently highlighted span, click **Insert**.
   - To copy the currently highlighted span to a pre-existing target span, click **Copy...**
   - To mirror the currently highlighted span to a pre-existing target span, click **Mirror...**
4. Click **OK**.

Portal frame haunch geometry

Eaves haunch dimensions



Apex haunch dimensions



Create cold-rolled sections

Tekla Structural Designer allows you to create multiple characteristic types of cold-rolled sections. The types are track, stud, and joist (US), or eaves beam, purlin, and rail (UK), depending on the language that you are using. For more information, see the following paragraphs.

RESTRICTION You can model and analyze cold-rolled in Tekla Structural Designer, but cold-rolled sections are not designed.

Create cold-rolled sections

1. On the **Model** tab, click any steel member type, for example, **Steel Beam**.
2. In the **Properties** window, set the **Characteristic** property to the desired cold-rolled section type.

   The properties in the **Properties** window are updated to match the selected cold-rolled section type.
3. In the **Properties** window, adjust the properties of the cold-rolled section according to your needs.
4. Click the start point of the member.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.
5. Click the end point of the member.

   Tekla Structural Designer creates the member.

Modify the position of a cold-rolled section

You can modify a cold-rolled section in both 2D and 3D views.

To move an entire cold-rolled section, see: Move and rotate objects (page 322).

1. Click the section that you want to modify.
2. Click the end node that you want to move.
3. Click the new position of the selected end node.

   The end node moves to the selected position.

Create wall and roof panels

Wall panels, also referred to as wind walls, allow you to apply loads calculated by the **Simple Wind Loading Generator** and **Wind Wizard** to your structure. Wall panels do not introduce any structural strength or stiffness to your structure. If you wish to introduce walls that resist gravity, or lateral loads, you must define them as concrete walls.

Roof panels allow loads placed on a sloping plane to be decomposed back to the supporting structure. Area loads on roofs can act either vertically, or normal to the roof plane. To create and modify roof panels, see the following instructions.

Create wall panels

RESTRICTION Note that:

- In order to define a wall panel, you must have already defined the grid points that define the panel vertices.
- Wall panels must lie in a single plane. Otherwise, Tekla Structural Designer will fail the panel during validation.
- The wall panel must consist of at least 3 points.

1. On the **Model** tab, click **Wall Panel**.
2. Click the start point of the panel.
3. Click the remaining points of the panel.
4. To define the end point of the panel, do one of the following:
    - Double-click the end point.
    - Click the end point, and click the start point again.

    Tekla Structural Designer creates a wall panel between the selected points.

Create wall panels with parapets

NOTE To ensure the wind analysis accounts for the parapet correctly, a wall panel with a parapet should be modeled in two parts:

1. Create an ordinary wall panel up to the roof level
2. Create a second wall panel above the roof level and marking the panel as a parapet

1. Open a frame view in which you want to create the wall.
2. Create the wall panel below the roof level normally.
3. Create the wall panel above the roof level normally.

    NOTE You may need to create a new construction level to define the top level of the parapet.

4. Press the **Esc** key.
5. Hover the mouse pointer over the second wall panel, so that it becomes highlighted.
6. Click the second wall panel.
7. In the **Properties** window, select the **Is a parapet wall** option.

See also: Parapet wall panel load decomposition (page 1014)

Modify the properties of a wall panel

You can modify the properties of a panel wall in both 2D and 3D Views.

1. Hover the mouse pointer over the panel, so that i becomes highlighted.
2. Click the panel to select it.

> **TIP** If necessary, add further roof panels to the selection by holding down the **Ctrl** key and clicking the panels.

3. Go to the **Properties** window.
4. Modify the properties according to your needs.

   Tekla Structural Designer automatically applies the changes to all the selected panels.

Create roof panels

RESTRICTION Note that:

- In order to define a roof panel, you must have already defined the grid points that define its outline.
- Roof panels must lie in a single plane. Otherwise, Tekla Structural Designer will fail the panel during validation.

1. On the **Model** tab, click **Roof Panel**.
2. Click the start point of the panel.
3. Click the remaining points of the panel.
4. To define the end point of the panel, do one of the following:
   - Double-click the end point.
   - Click the end point, and click the start point again.

   Tekla Structural Designer creates a roof panel between the selected points.

Modify the properties of roof panels

You can modify the properties of roof panels in both 2D and 3D Views.

1. Hover the mouse pointer over the panel, so that it becomes highlighted.
2. Click the panel to select it.

   TIP If necessary, add further roof panels to the selection by holding down the **Ctrl** key and clicking the panels.

3. Go to the **Properties** window.
4. Modify the properties according to your needs.

   Tekla Structural Designer automatically applies the changes to all the selected panels.

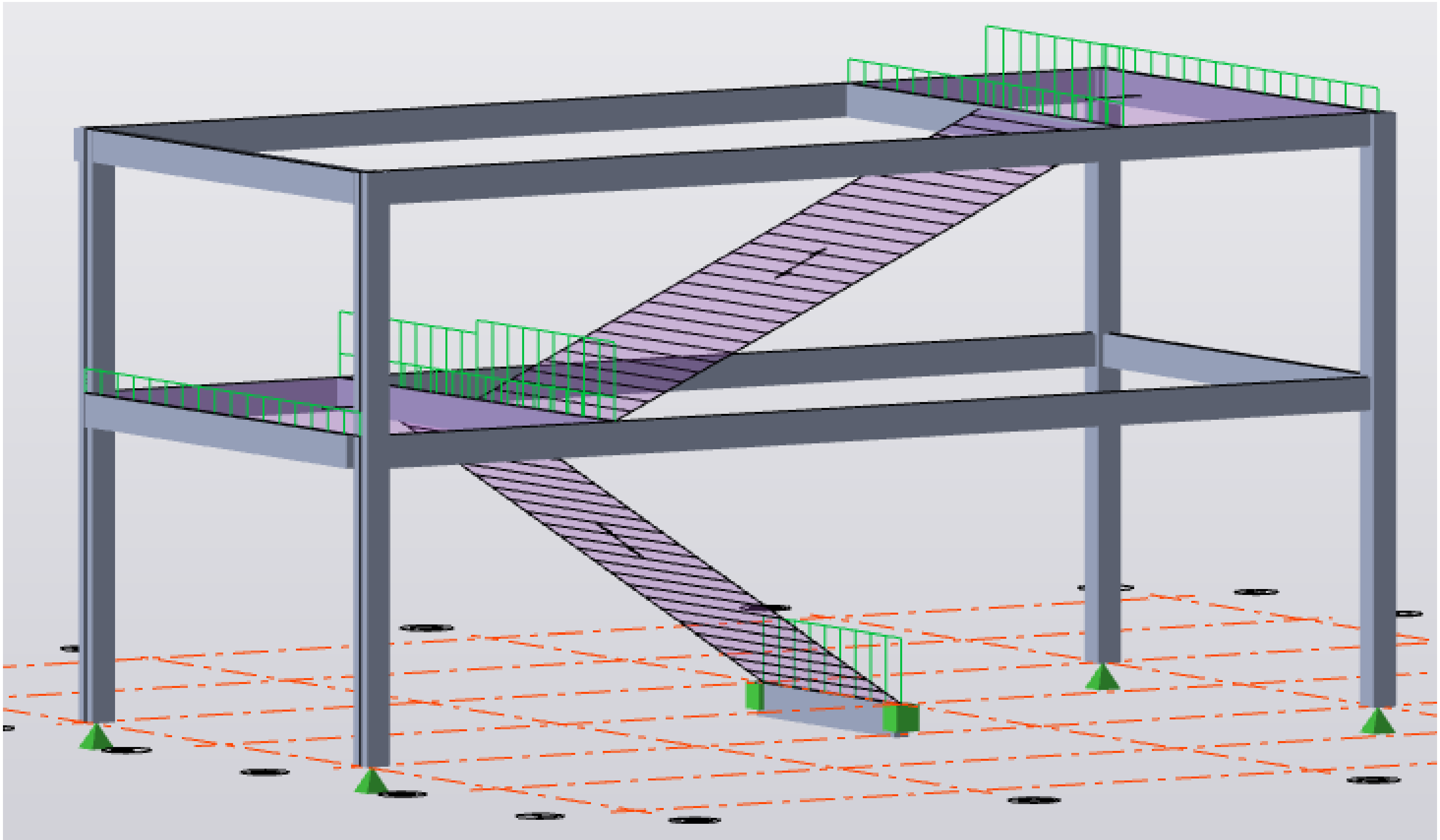
Ancillaries

Ancillaries enable the quick and efficient application of loading from ancillary items such as ladders, stairs and pipework etc - that are not part of the main structural frame.

What are ancillaries used for?

While primarily intended for use in industrial structures, some features associated with ancillary loads are also beneficial to the AEC market - eg:

- Modeling of stairs in steel, concrete, and timber structures,
- The use of stairs in combination with inactive members.



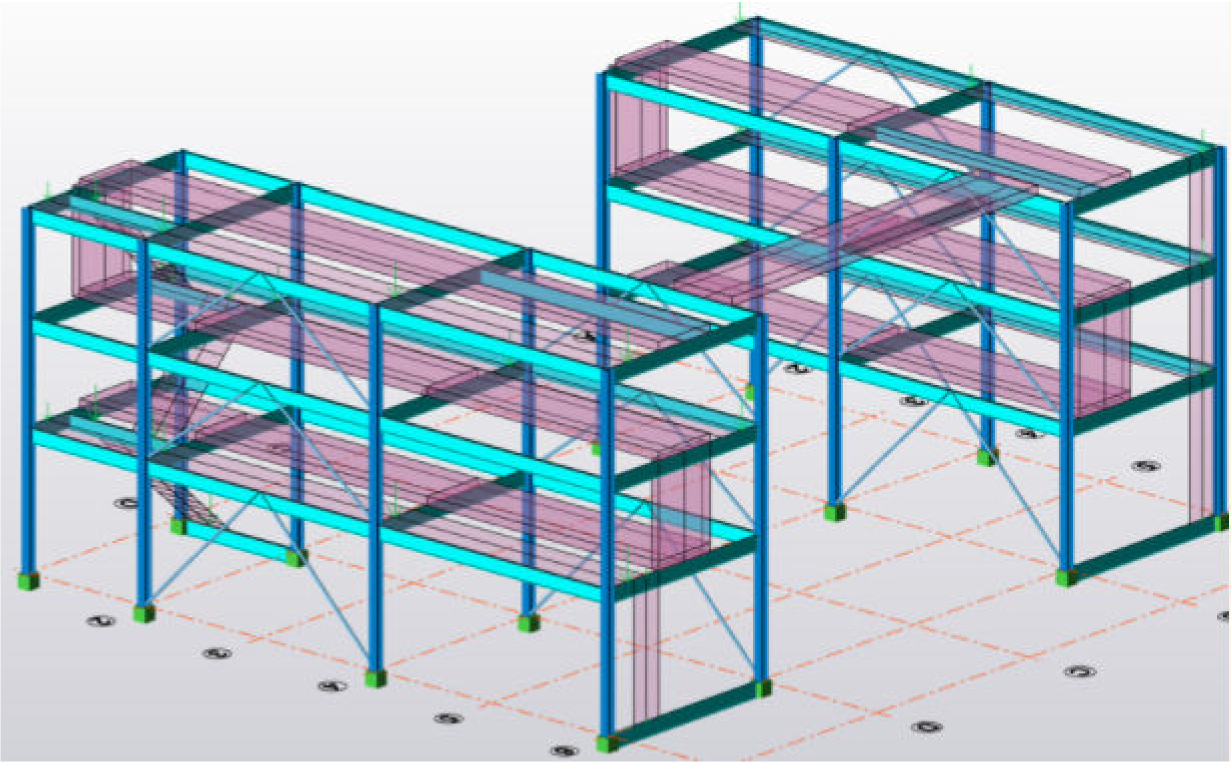
Ancillaries can also be used to model loads from:

- Walkways/Catwalks
- Ladders (with/without cages)
- Access Platforms
- Operating Platforms (Storage/Standard)
- Lines of Pipework
- Lines of Cable Tray

In Tekla Structural Designer each of the above types can be created in the form of either a *line*, or *area* ancillary.

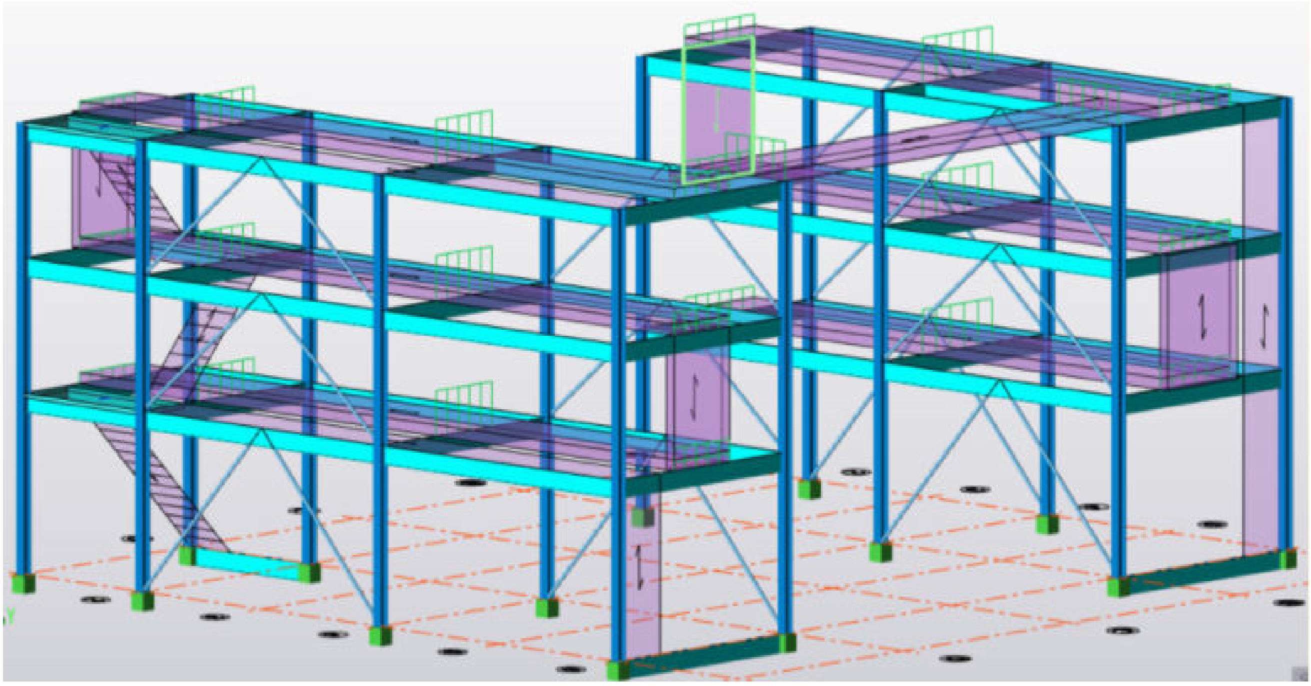
Line ancillaries

Line ancillaries can be defined horizontally, vertically or sloped. They are created by selecting a series of support points in order to create a "run". Each line has a defined width in a user defined plane (perpendicular to length).



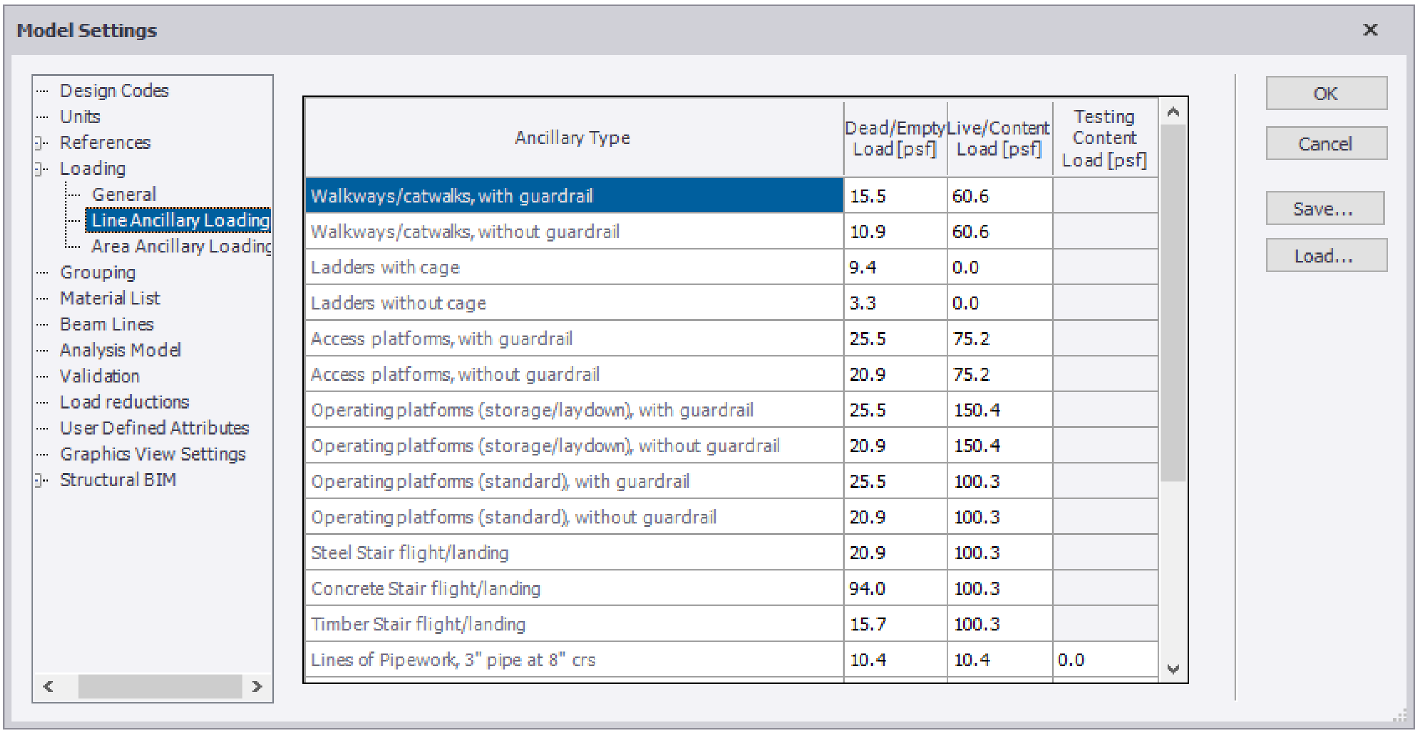
Area ancillaries

Area ancillaries can be defined horizontally, vertically or sloped. They are created in the same way as a panel, by selecting the vertices of the area.



Ancillary load default values

Each Line or Area Ancillary has default Dead Loads and Imposed/Live Loads which can be preset to be project specific from the sub-pages under **Loading** in **Model Settings**.



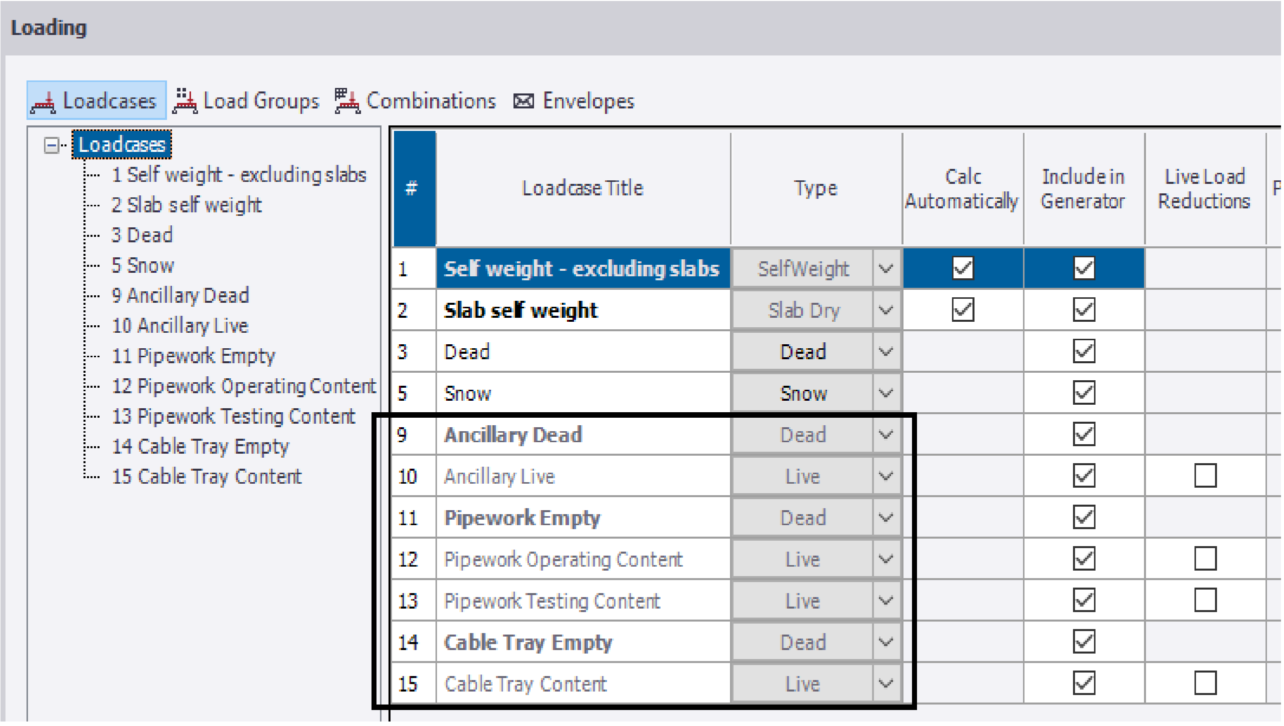
Ancillary Type	Dead/Empty Load [psf]	Live/Content Load [psf]	Testing Content Load [psf]
Walkways/catwalks, with guardrail	15.5	60.6	
Walkways/catwalks, without guardrail	10.9	60.6	
Ladders with cage	9.4	0.0	
Ladders without cage	3.3	0.0	
Access platforms, with guardrail	25.5	75.2	
Access platforms, without guardrail	20.9	75.2	
Operating platforms (storage/laydown), with guardrail	25.5	150.4	
Operating platforms (storage/laydown), without guardrail	20.9	150.4	
Operating platforms (standard), with guardrail	25.5	100.3	
Operating platforms (standard), without guardrail	20.9	100.3	
Steel Stair flight/landing	20.9	100.3	
Concrete Stair flight/landing	94.0	100.3	
Timber Stair flight/landing	15.7	100.3	
Lines of Pipework, 3" pipe at 8" crs	10.4	10.4	0.0

The default values can be overridden when individual ancillaries are created.

NOTE For Pipework Operating & Testing Content Loadcase Types, you can select whether these are considered as Dead or Imposed (Live) loads from **Model Settings > Loading > General**.

Ancillary loadcases

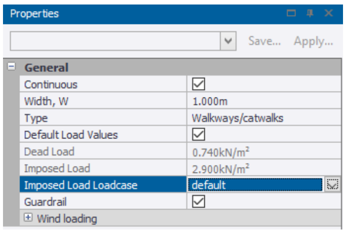
Ancillary loads are created in dedicated loadcases which are automatically added and removed as the loads are added/deleted. These dedicated loadcases specifically aid combination building for Industrial design.



Most types of ancillary load are created in the **Ancillary Dead** & **Ancillary Live (Imposed)** loadcases. The two exceptions are lines of pipework which are created in the **Pipework Empty**, **Pipework Operating Content** & **Pipework Testing Content** loadcases, and lines of cable tray which are created in the **Cable Tray Empty** & **Cable Tray Content** loadcases.

NOTE

If working to Eurocodes, the ancillary imposed case Ψ and ϕ factors default to $\Psi_1 = 1.0$, $\Psi_2 = 0.9$, $\Psi_3 = 0.8$, $\phi = 1.0$.



Some ancillaries may require different values in which case a new imposed loadcase should be manually added with the desired Ψ and ϕ factors. Then when the ancillary load is being created the loads can be put into the new loadcase instead of the dedicated loadcase by changing the **Imposed Load Loadcase** in the **Properties** window from 'default' to that required.

Ancillary load decomposition

All decomposed loads from ancillaries are present in the analysis and design.

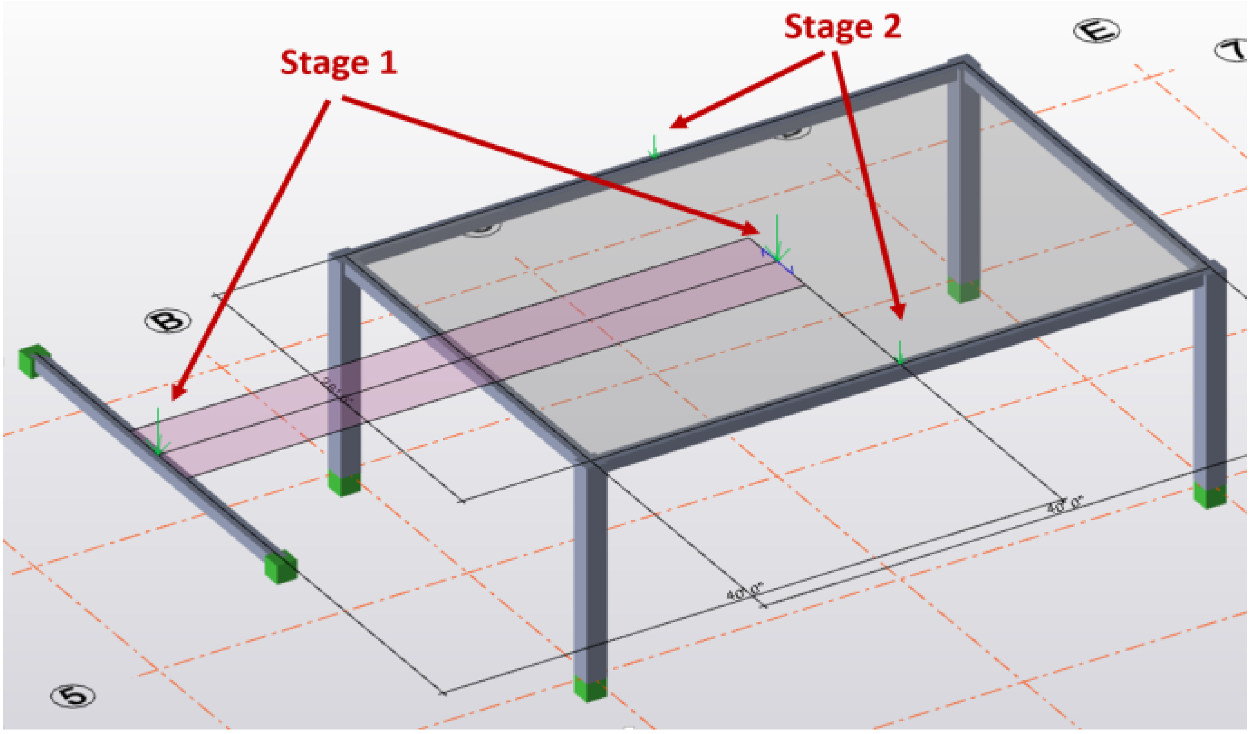
Once loads are decomposed the ancillaries themselves play no further part in analysis and design.

Line ancillary decomposition

Line ancillaries are treated as simply supported beams, spanning onto supporting members/slabs.

Decomposition is in two stages from the line ancillary to its supports then onwards.

In the below example the line ancillary is supported by a beam at one end and a slab spanning perpendicular to the ancillary at the other. At the slab end, stage 2 decomposition occurs to distribute the load on to beams supporting the slab.



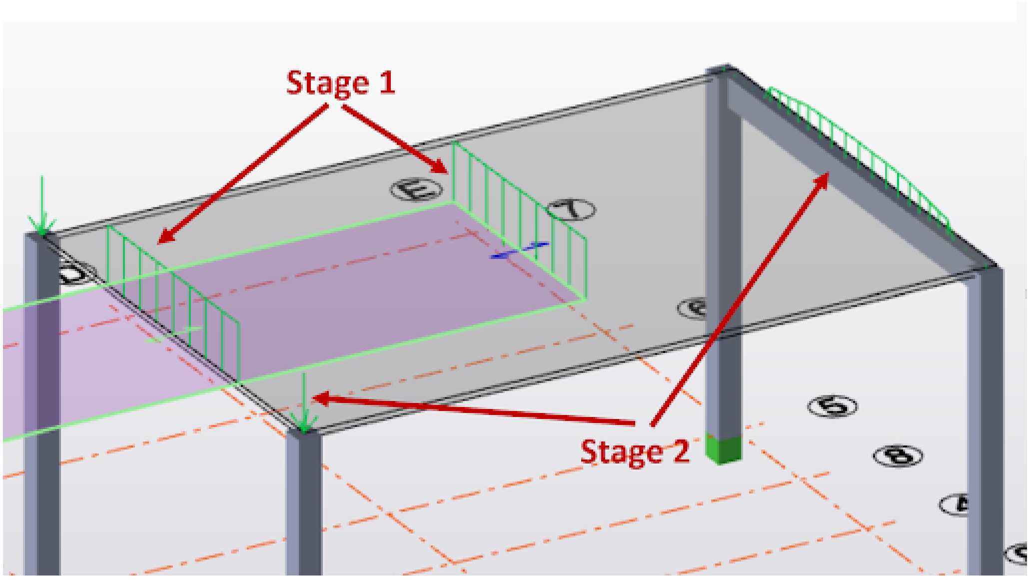
Area ancillary decomposition

In the same way as a roof panel, an area ancillary has a span direction which determines the load decomposition onto surrounding supporting members/ slabs.

Decomposition is in two stages from the area ancillary to its supports then onwards.

In the below example the area ancillary is supported by a slab which spans in the same direction as the ancillary.

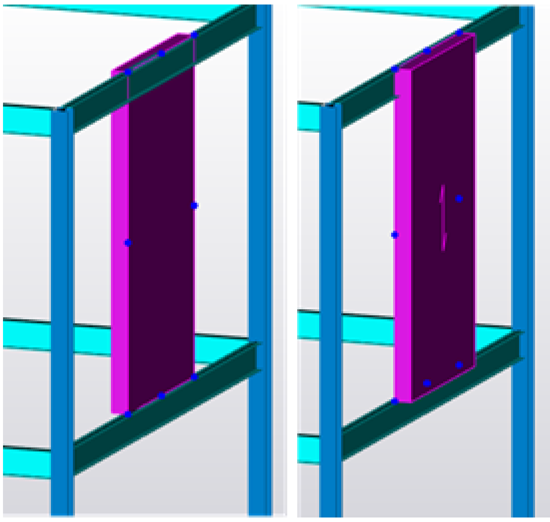
The UDLs generated in the stage 1 decomposition are then decomposed on to the members supporting the slab in stage 2.



Create line ancillary loads

1. On the **Model** tab, click **Line Ancillary**.
2. In the **Properties** window,
   a. If the load is to only span between two points, uncheck the **Continuous** setting, otherwise leave it checked.
   b. Enter the load width
   c. Select the required **Type**.
   d. Accept the default load values, or define your own.
   e. Specify the remaining properties as appropriate to selected Type.

NOTE
1. For walkways and platforms the presence of a guardrail and channel height alters dead loading.
2. For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking the Reversed property.



Reversed line of pipework or cable tray

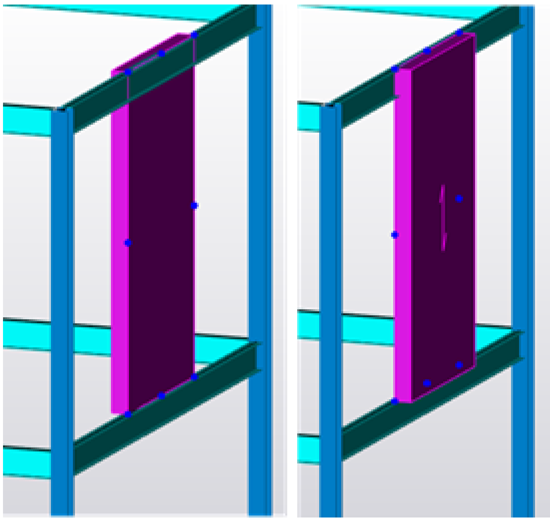
3. Click the start point of the line.
4. Click the remaining points of the line.
5. Double-click to define the end point of the line.
6. Pick a reference point to define the ancillary plane.

   Tekla Structural Designer creates a line ancillary between the selected points.

Create area ancillary loads

1. On the **Model** tab, click **Area Ancillary**.
2. In the **Properties** window,
   a. The rotation angle defines the span direction measured relative to the first two points clicked to identify the area.
   b. Select the required **Type**.
   c. Accept the default load values, or define your own.
   d. Specify the remaining properties as appropriate to selected Type.

NOTE
1. For walkways and platforms the presence of a guardrail and channel height alters dead loading.
2. For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking the Reversed property.



Reversed line of pipework or cable tray

3. Click the start point of the area.
4. Click the remaining points of the area.
5. To define the end point of the area, do one of the following:
   - Double-click the end point.
   - Click the end point, and click the start point again.

   Tekla Structural Designer creates an area ancillary load between the selected points.

Create an ancillary loads report

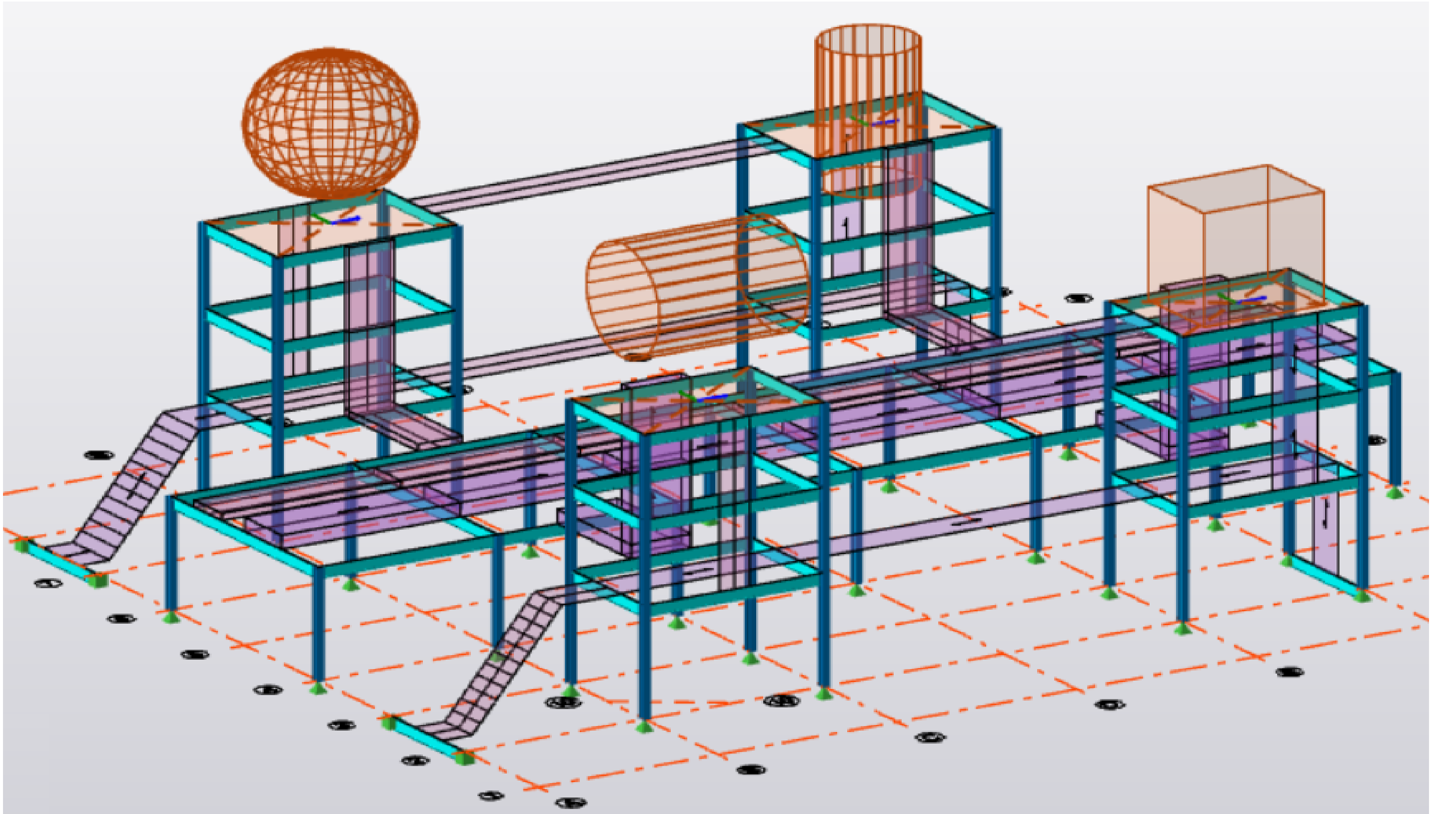
A table of the ancillary loads applied to the model is available in the **Industrial Structure Loading** report, which is created as follows:

1. In the list on the left side of the **Report** toolbar, select **Industrial Structure Loading**.
2. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

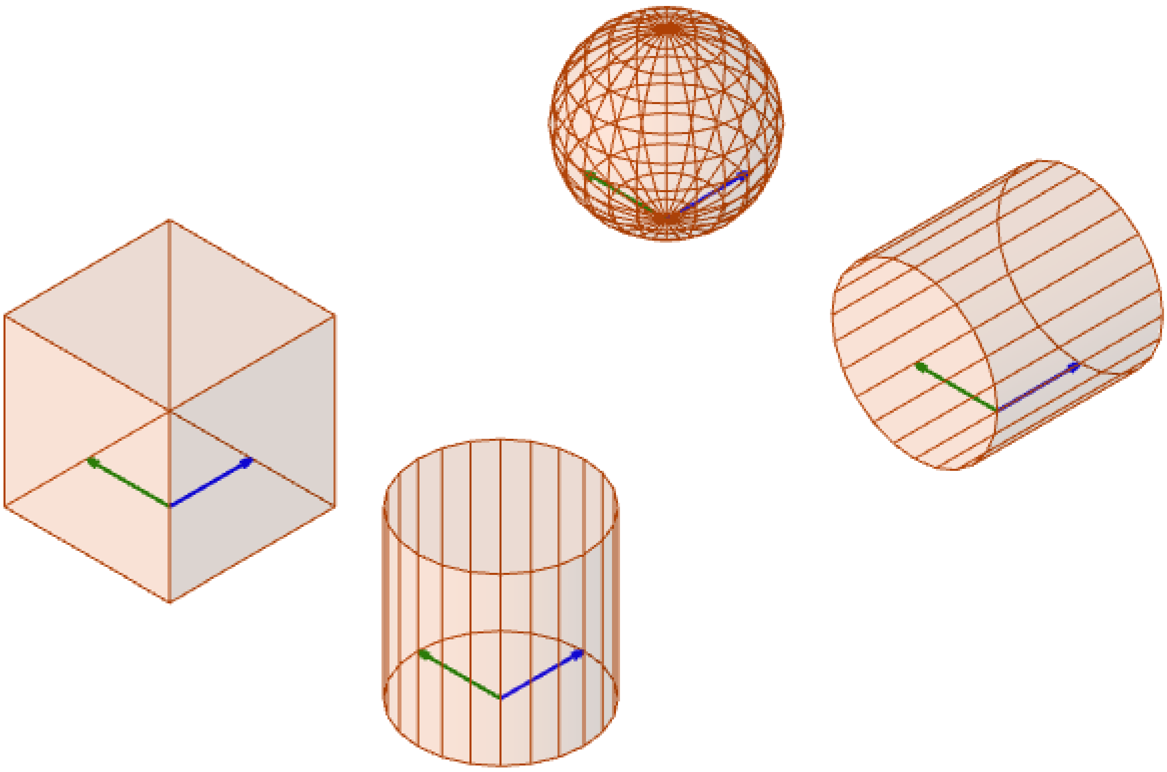
Equipment

Equipment is primarily intended for use in industrial structures, enabling the quick and efficient application of loading from plant items, tanks & cylinders etc. - that are not part of the main structural frame.



Overview

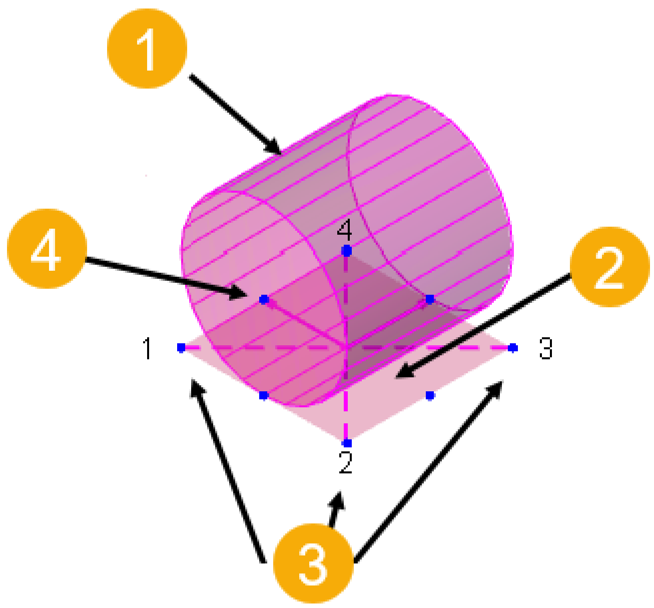
Equipment can be displayed in Tekla Structural Designer in the form of a sphere, vertical cylinder, horizontal cylinder or cuboid.



Equipment has an associated Center of Gravity (CoG) which is indicated by the user co-ordinate system (UCS) as shown above.

Equipment also has a number of support points through which load is transferred to the structure. At least three support points are required and there is no limit on the maximum. The area bounded by these support points is referred to as the 'Loading Area'.

The support points are numbered and become active when the equipment is selected. Unnumbered nodes at the mid-points of loading area edges also become active.

Equipment support points and loading area	
	
1	Equipment
2	Loading area
3	Support points
4	Unnumbered node **NOTE** **Unnumbered** nodes are provided to facilitate editing of the Loading Area - they are **not** support points.

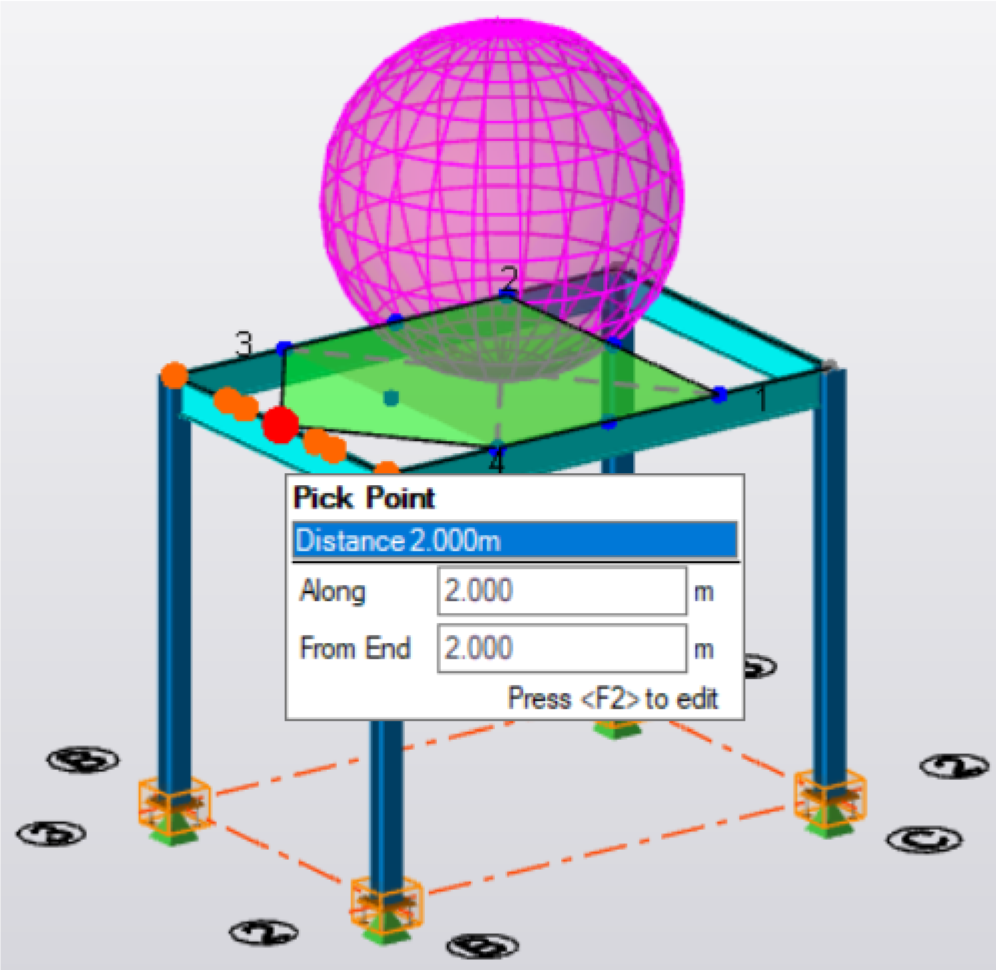
The 3D geometry, UCS and loading area can be switched on or off for each equipment shape within Scene Content (page 68).

Once placed, equipment is listed in the Project Workspace Structure Tree (page 58) under the Loading Elements group.

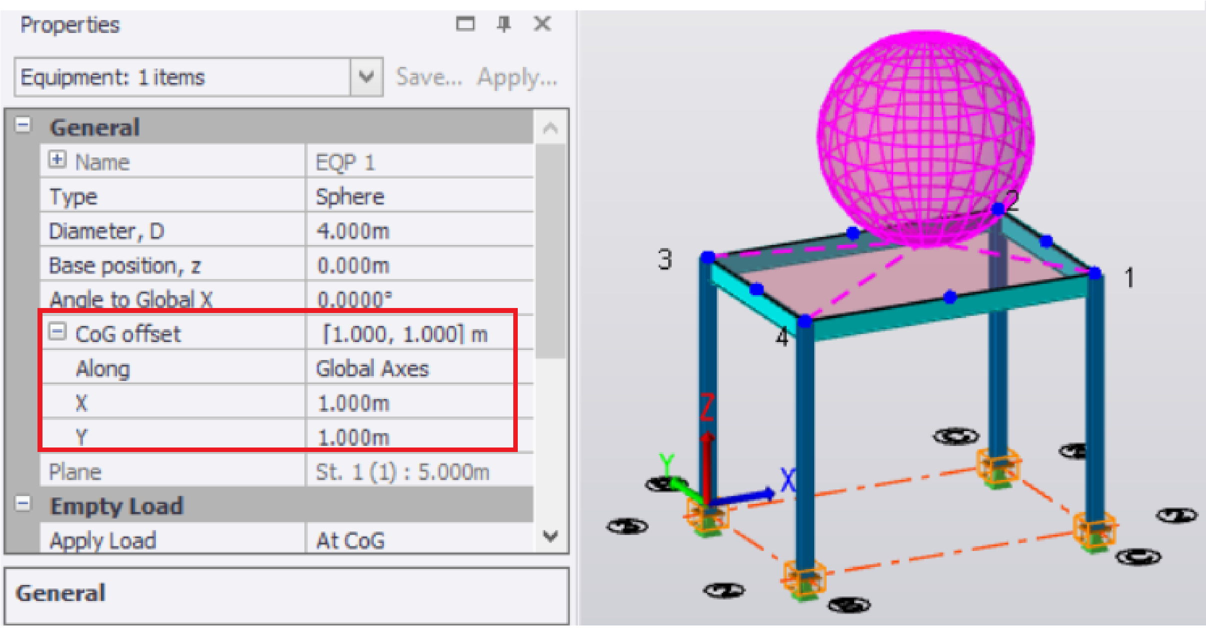
Modeling

Equipment must be defined in the horizontal plane.

Once it has been defined, you can alter the shape of the loading area by dragging any of its nodes. Dragging a numbered support point moves it, dragging an unnumbered node (as shown below) introduces a new support point.



By default the CoG of the equipment and the CoG of the support points coincide. You can specify an offset to the equipment CoG to facilitate the adjustment of equipment loads on the structure.



Equipment must be placed in such a way that each support point is itself supported by a structural item or slab item.

No member, ancillary, slab, wall or other equipment may be attached to an equipment entity.

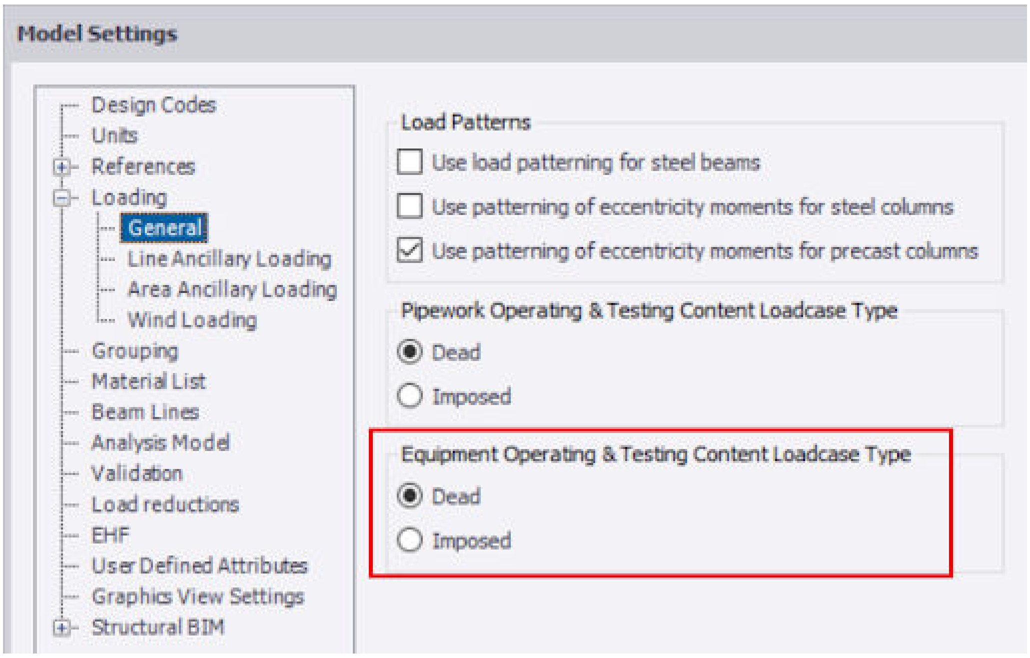
Equipment loadcases

Equipment loads have three dedicated gravity loadcases which are automatically added and removed as the loads are added/deleted.

- Equipment Empty
- Equipment Operating Content
- Equipment Testing Content

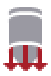
These dedicated loadcases specifically aid combination building for Industrial design.

NOTE For Equipment Operating & Testing Content Loadcase Types, you can select whether these are considered as Dead or Imposed (Live) loads from **Model Settings > Loading > General**.



When equipment is added, initially the loads in the dedicated loadcases default to zero. When the load is specified it can either be applied at the CoG of the equipment, or at the equipment support points. This is important when the CoG is different between the loaded and unloaded conditions.

Once equipment has been defined, additional horizontal or vertical point loads can be applied to it in other dead and imposed/live loadcases.



The **Equipment** load command for doing this is located in the Structure Loads group on the **Load** toolbar.

Equipment load decomposition

All decomposed loads from equipment are present in the analysis and design.

Once loads are decomposed the equipment itself plays no further part in analysis and design.

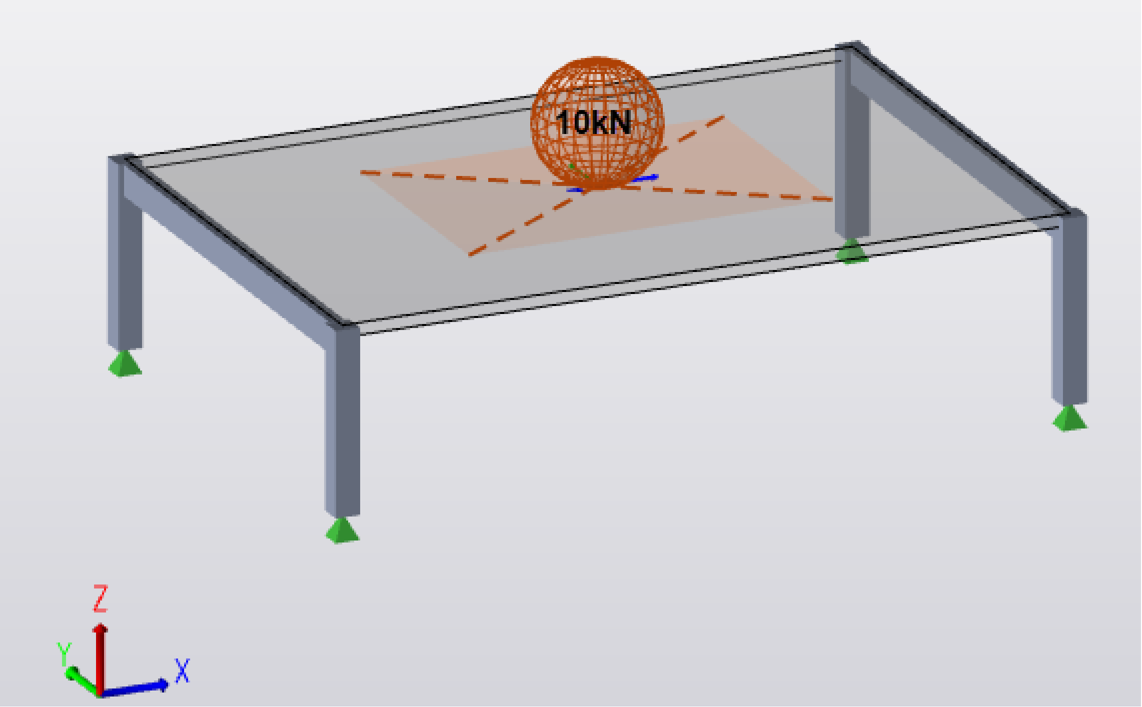
Equipment decomposition for loads applied at CoG

Decomposition is in two stages: from the equipment to the equipment support points, then into the structure.

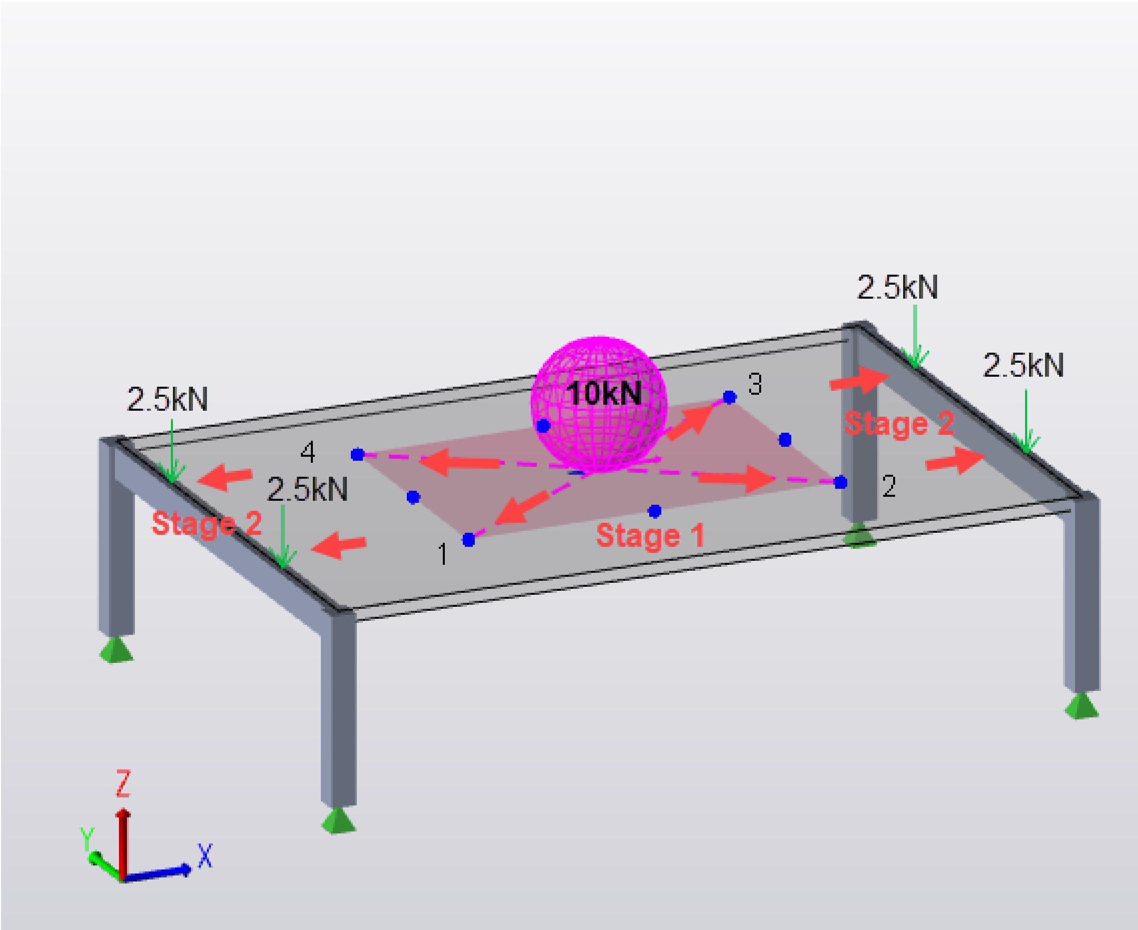
For each equipment, the load distribution distributes the total load proportionately to the equipment support points based on an iterative decomposition approach in which the loading area acts as rigid.

NOTE The distribution of a point load applied to a rigid plate is a very complex problem to solve, if done by FE then results are determined by stiffness of the plate. The iterative approach is approximate but believed to be sufficiently accurate for the purpose. If you do not want to use this approach, the loads should be applied at supports instead.

In the following example equipment is placed on a one-way slab which spans in the global X direction. The equipment has 4 support points positioned symmetrically about the slab center-line. A load of 10kN is applied at the equipment CoG.

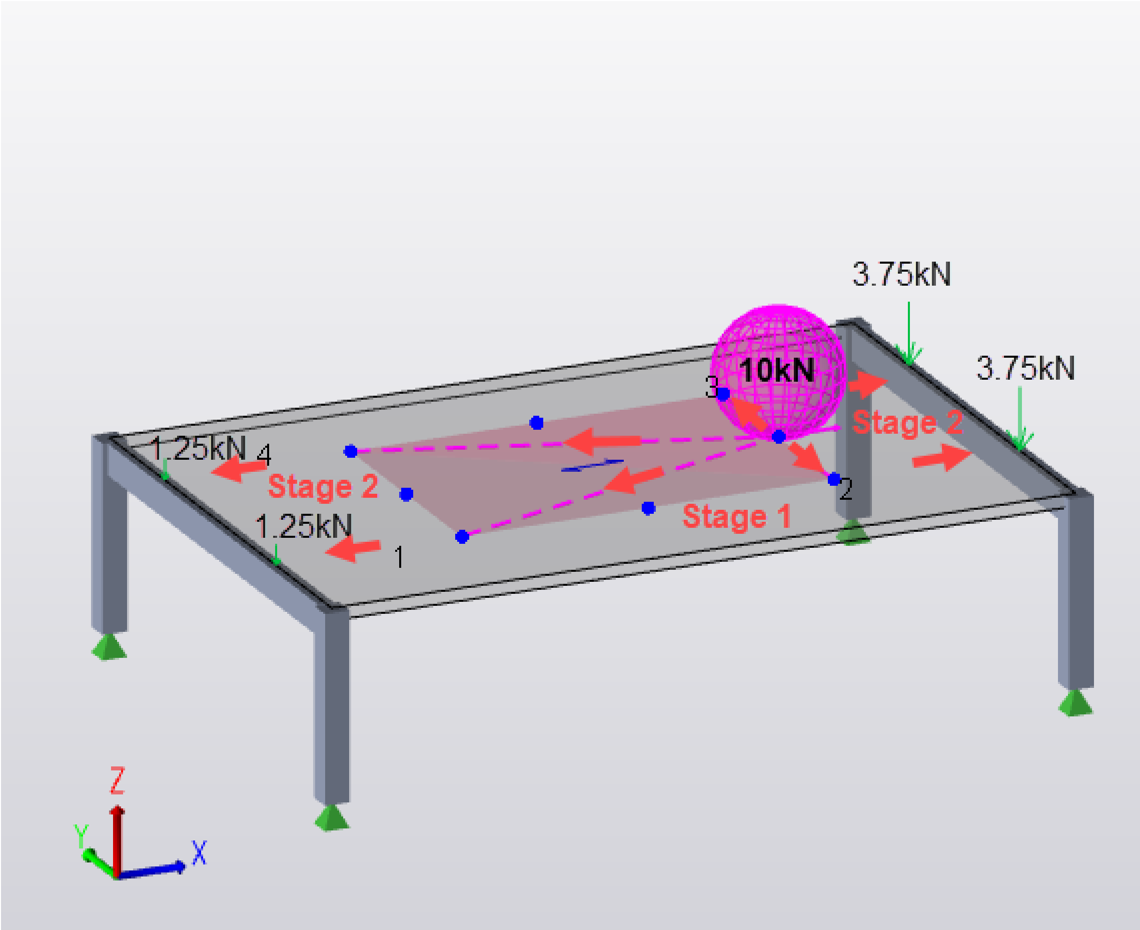


When the model is analyzed the equipment load is decomposed to the equipment support points, then on to the supporting members as shown below.



- **Stage 1** - the 10kN load is decomposed to the 4 numbered support points, due to symmetry these each receive 2.5kN.
- **Stage 2** - support point loads are decomposed on to the members supporting the slab. Again due to symmetry, the resulting member point loads are each 2.5kN

If the CoG of the equipment is offset along the global X axis so that it now lies at the edge of the loaded area, when the model is re-analyzed the decomposed loads are updated as shown below:



- **Stage 1** - because the CoG has been offset, support points 2 and 3 each receive 5kN whereas support points 1 and 4 remain unloaded.
- **Stage 2** - support point loads are decomposed on to the members supporting the slab, resulting in 2 point loads of 1.25kN on the left, and 2 point loads of 3.75kN on the right.

Equipment decomposition for loads applied at supports

In this case, only the second stage decomposition is required on to the supporting members.

Create equipment and equipment loads

1. Open a 3D View, or 2D Level View in which to create the equipment.

   NOTE Equipment must be defined in a horizontal plane, and so cannot be created in a 2D Inclined Plane View.

2. On the **Model** tab, click **Equipment**.
3. In the **Properties** window, define the equipment shape and size:
    a. Select the required **Type**.
    b. Enter the geometric properties (Length, Breadth, Height, or Diameter) required to define the size.
    c. Leave **Base position, z** as 0 to place the equipment directly on it's loaded plane, or enter a non-zero value to raise/lower it.
    d. Leave **Angle to Global X** as 0 , or enter an angle to rotate it.
4. In the **Properties** window, select whether the empty, operating content, and testing loads are to be applied at the center of gravity of the equipment, or through the supports.
5. Define the equipment load values:
    - If applied **At CoG**, enter each value.
    - If applied **At Supports**, click the ellipsis button then enter the required value at each support.
6. In the 3D or 2D View, click the start point of the loaded area.
7. Click the remaining points of the loaded area.
8. To define the end point of the loaded area, do one of the following:
    - Double-click the end point.
    - Click the end point, and click the start point again.

    NOTE At least three support points are required, no limit on maximum.

    Tekla Structural Designer creates the equipment positioned at the center of gravity of the loaded area.

Create additional equipment loads in other loadcases

Once equipment has been defined you can apply additional loads to it in loadcases other than the dedicated 'equipment' ones. These loads can be applied in any global or local direction.

1. In the **Loading** list, select an appropriate loadcase.

    NOTE This cannot be one of the existing dedicated 'equipment' loadcases.

2. On the **Load** toolbar, click **Equipment**.
3. In the **Properties** window, select the load direction.

4. In the **Properties** window, select whether the load is to be applied at the center of gravity of the equipment, or through the supports.
5. Define the load value:
   - If applied **At CoG**, enter the load value and the **Offset from mid height** above or below the loaded area at which the load is to be applied.
   - If applied **At Supports**, click the ellipsis button then enter the required value at each support.
6. In a 2D or 3D View, pick the equipment to apply the load to.

Create an equipment loads report

A table of the equipment loads applied to the model is available in the **Industrial Structure Loading** report, which is created as follows:

1. In the list on the left side of the **Report** toolbar, select **Industrial Structure Loading**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Inactive members

Single span members can be made inactive for analysis and design while still being kept in the model for load distribution and for determination of effective lengths.

This feature is particularly useful for industrial structures, as frequently not all the secondary beams are included in the analysis and design model, but they are still required to distribute load in the structure and act as restraints to supporting members.

An inactive member is totally ignored in the solver model as is any load carried by it. Therefore, to allow the load to be distributed it is instead decomposed to nodal forces in a pre-analysis decomposition stage - this happens for all applied loads on inactive members on the basis of the members being simply supported.

Which members can be made inactive?

Only single span beams, braces, analysis elements, purlins and rails have the potential to be inactive. They each have an **Active** setting in the properties which defaults to 'on' but can be unchecked.

NOTE Members cannot be made inactive if:

- they are multi-span, or,
- they support another member (active or inactive)

To make a member inactive

By selecting the Show/Alter State Active (page 727) attribute in a **Review View**, inactive members are color coded allowing the active/inactive setting to be toggled graphically.

Alternatively, you can change the setting manually in the **Properties** window as follows:

1. Select the member in a 2D or 3D view.
2. In the **Properties** window deselect **Active**.

Inactive member load decomposition

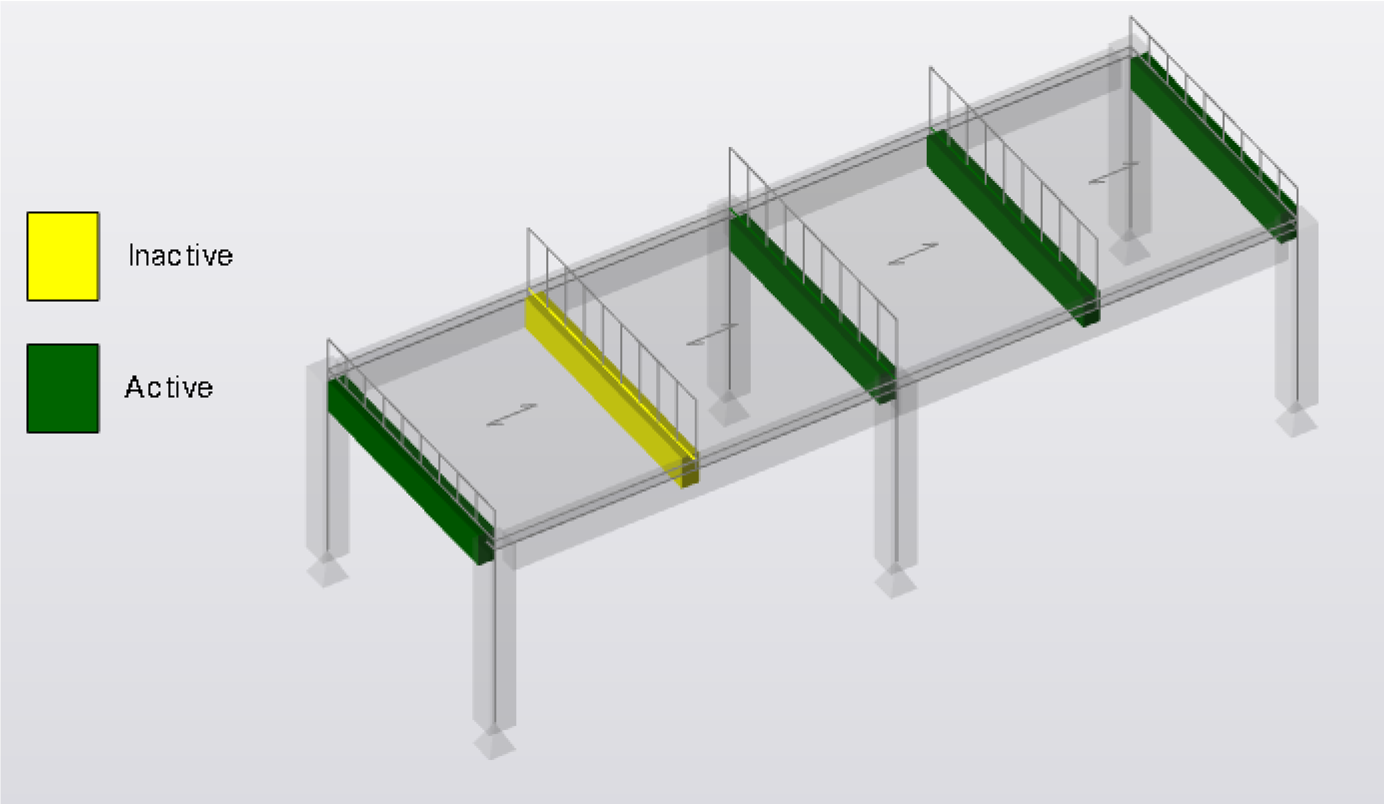
The first stage of load decomposition treats active and inactive members in exactly the same way.

This is then followed by a second stage of load decomposition for the inactive members only, as follows:

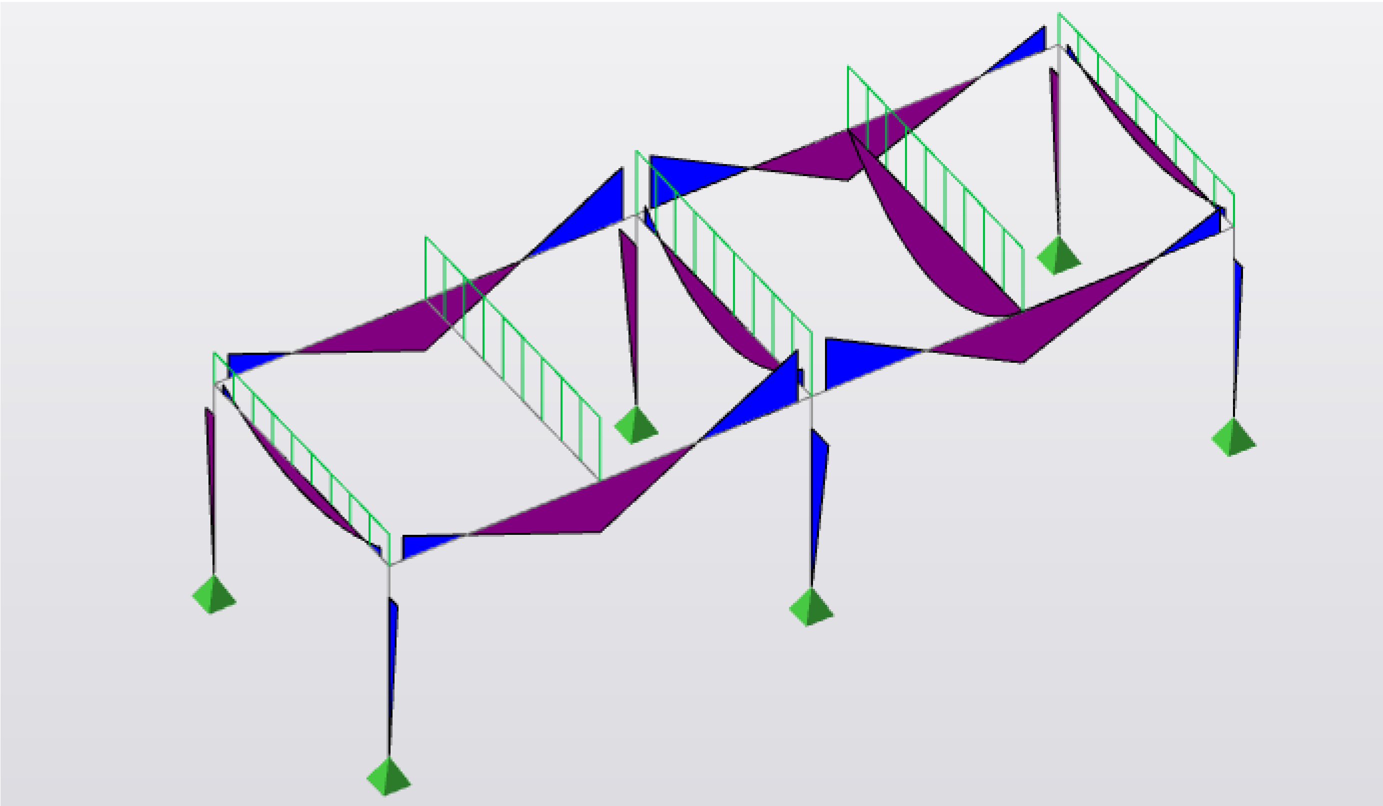
- End reactions are calculated for the inactive members.
- These are then applied as nodal forces to the supporting members.

After decomposition has taken place analysis can proceed with the inactive members removed from the solver model.

In the below example, by using the Show/Alter State Active (page 727) attribute one beam has been made inactive. The view shows decomposed loads - note that the inactive beam still has slab load decomposed to it in the same way as the active beams (as a result of the first decomposition stage).



The effects of the second decomposition stage can be seen by switching to a Results View showing major moments.



In the second stage the end reactions of the inactive beam have been applied as nodal forces to the supporting edge beams in the first bay causing moments to be generated in them, (without this second stage the two supporting beams would have otherwise been completely unloaded.)

No moments are generated in the inactive beam itself because after the decomposition has taken place it does not exist in the solver model.

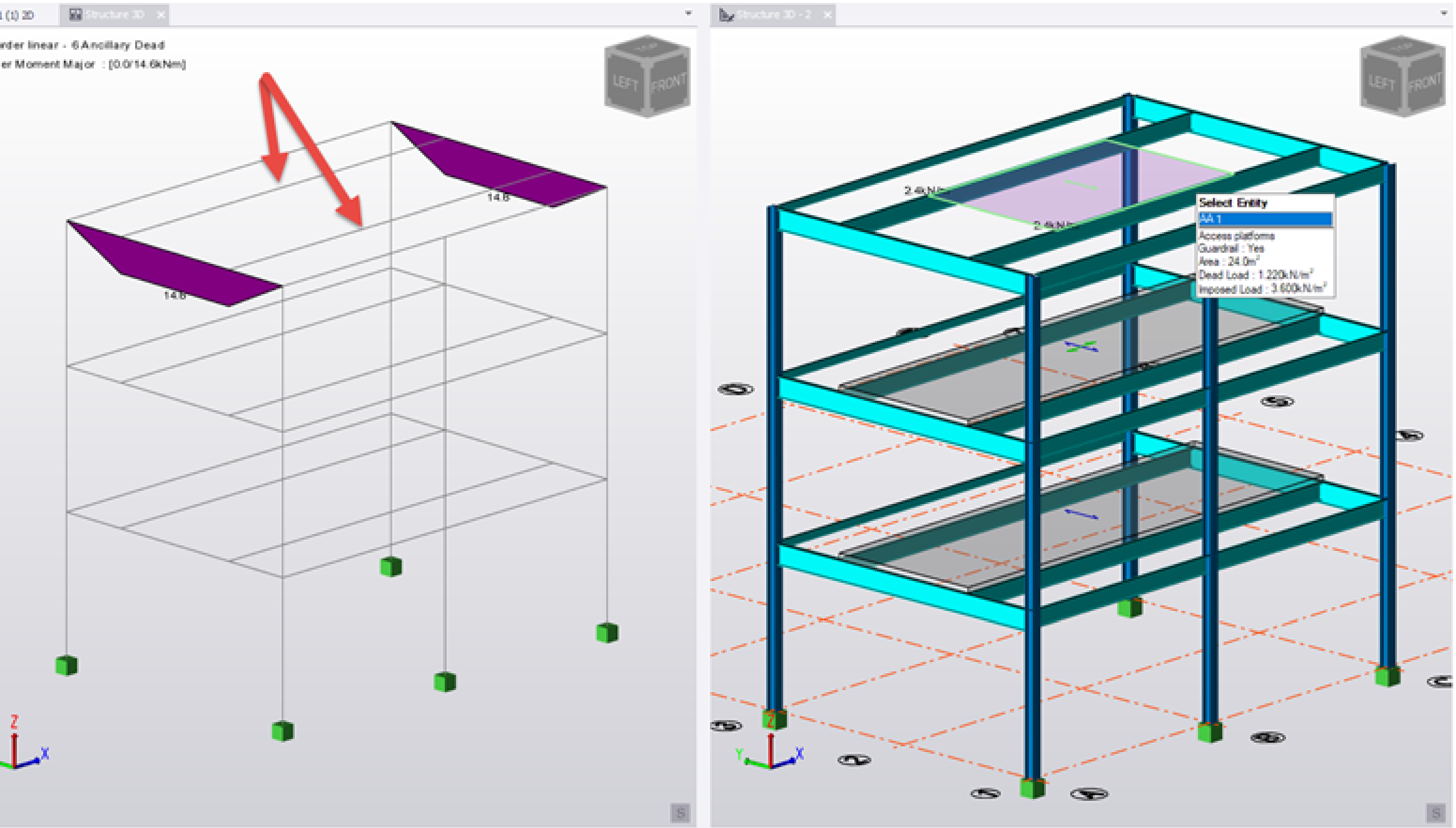
Typical usage cases for inactive members

There are potentially many uses for inactive members, the following being some examples.

Industrial Structures

Inactive members can be used in conjunction with ancillaries in order to apply loads to the structure while not themselves participating in the analysis.

In the below example an area ancillary decomposes load on to two inactive beams, which is then applied as nodal forces on to the supporting members.

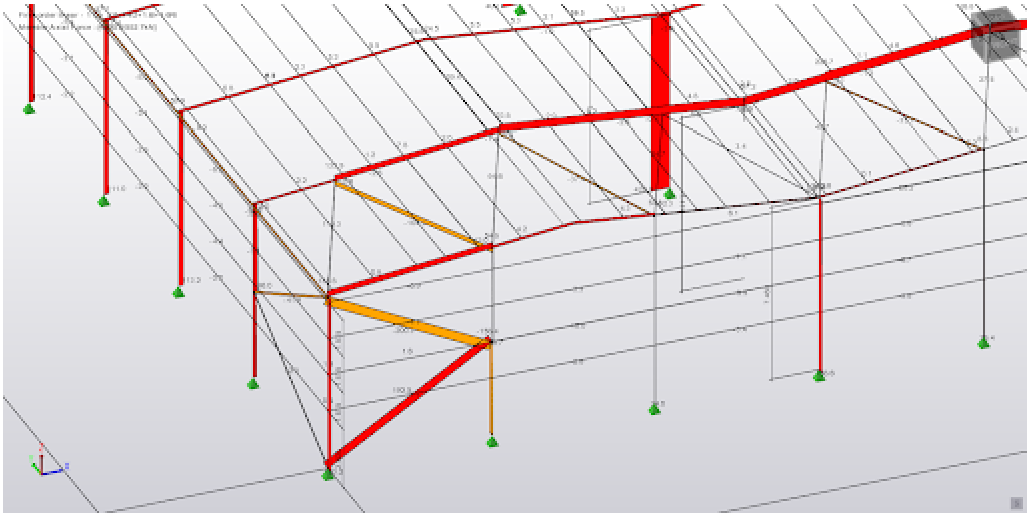


Purlins and Rails

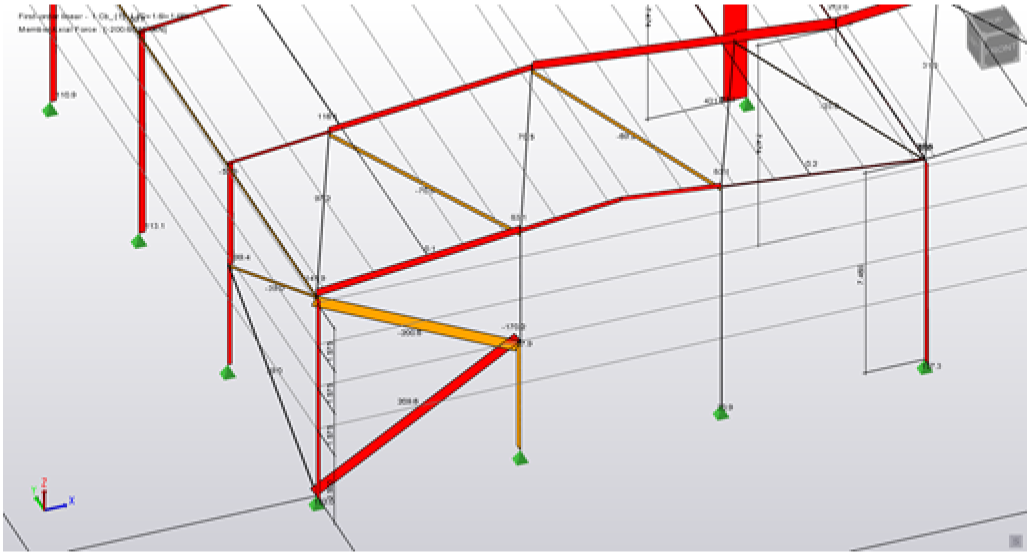
In large portal sheds, purlins and rails may be included in the model to act as a means of load distribution; the engineer would not want to see the forces in them when reviewing forces in the main structural members, or consider the small axial loads that develop in purlins and rails when gravity loadcases are analysed.

By setting the purlins and rails to be inactive these goals are achieved.

This is demonstrated in the below example - in the top view the purlins are rails are active, consequently small forces develop in them in gravity combinations and the results views are quite clutterred with values that are of no interest to the engineer.



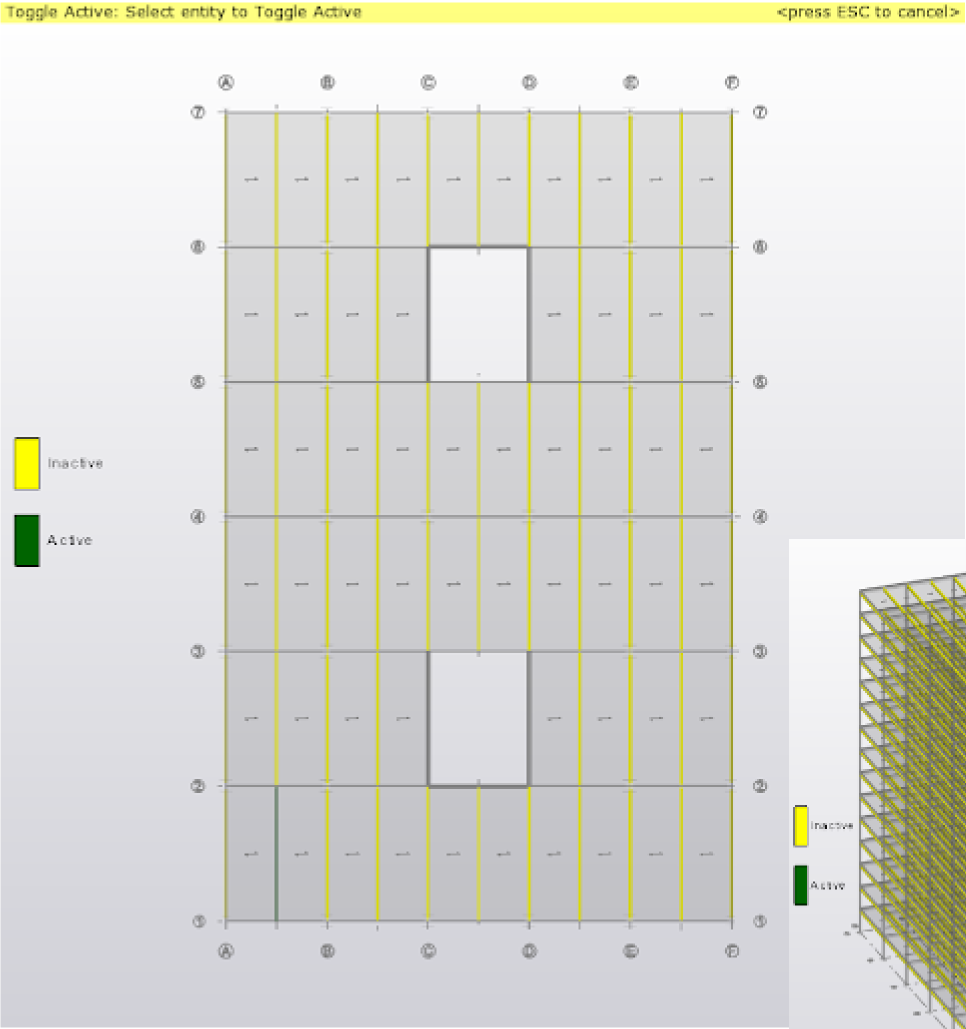
In the second view the purlins and rails have been set inactive, providing a clearer display and showing the values the engineer would expect/assume from such a combination.

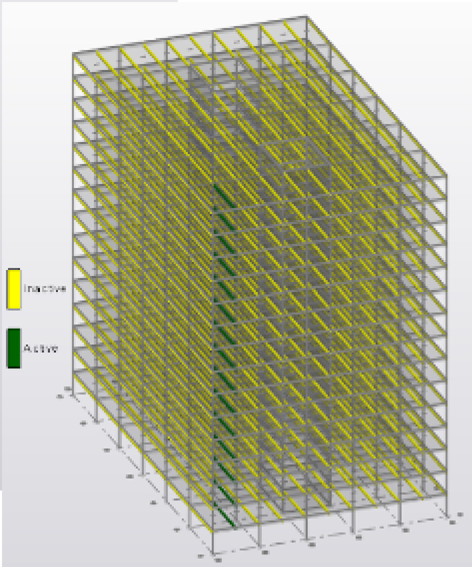


Grouped Design

Design can be speeded up by enabling group design and limiting the number of active members in a group.

This is demonstrated in the below example of a regular floor, all beams in the group apart from one have been made inactive which significantly reduces the design time.



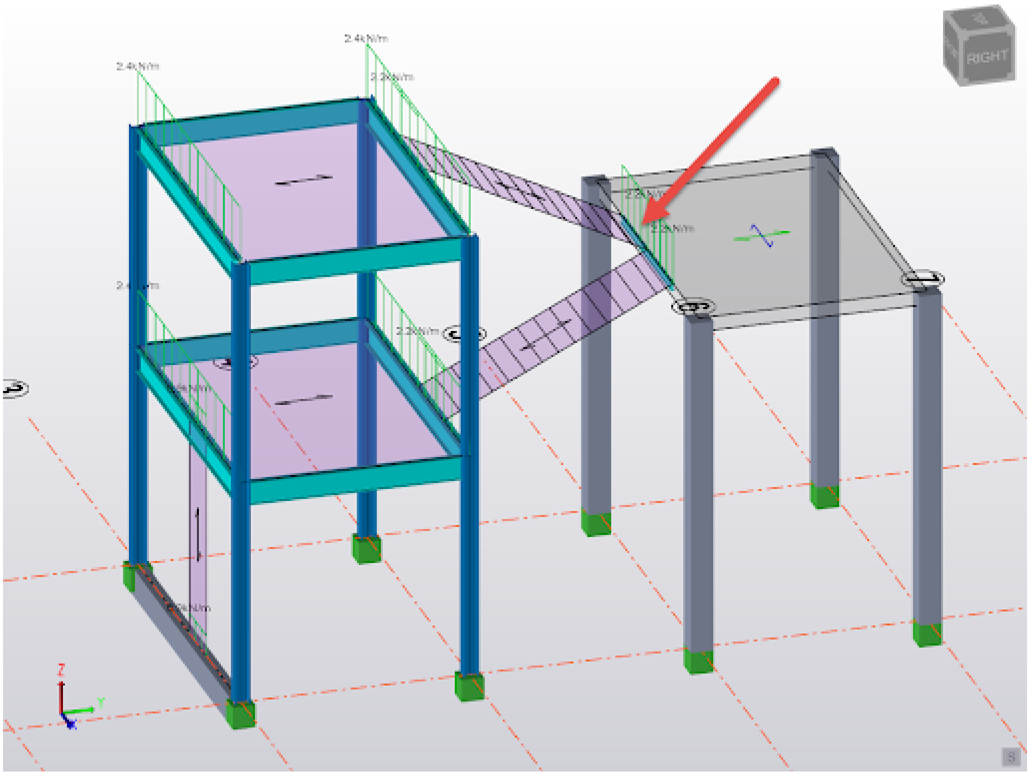


NOTE When making the non-critical beams inactive to speed up design:

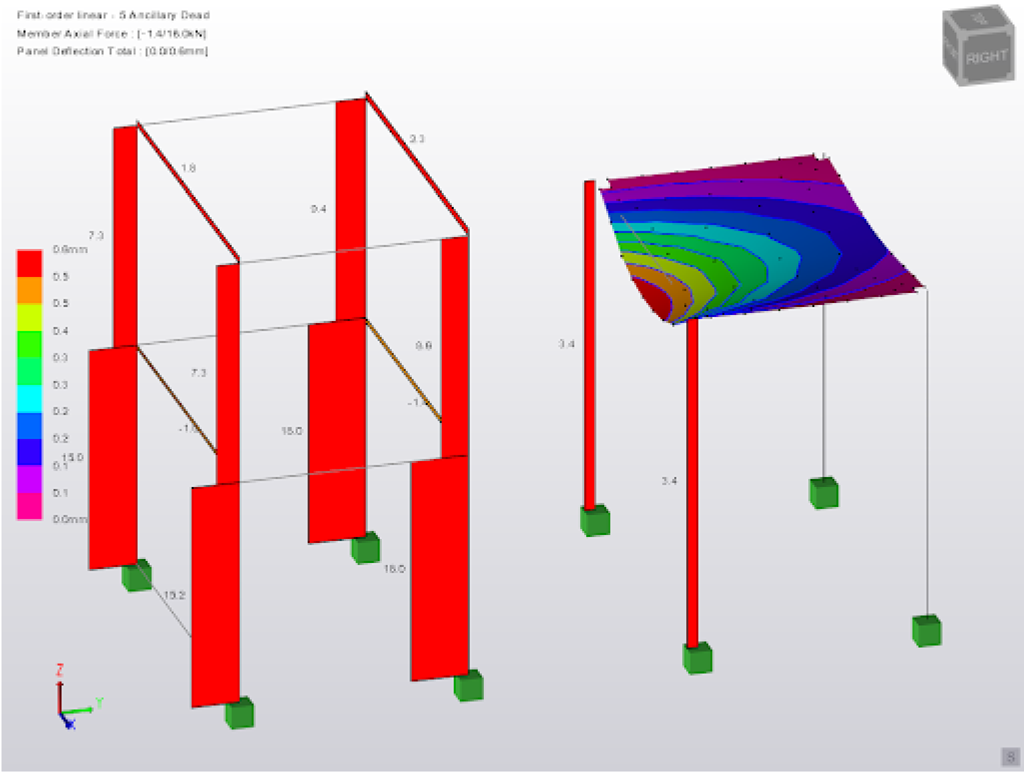
- You are responsible for choosing critical beam(s)
- Primary beams cannot be set to inactive

To support stairs at a slab edge

If an area ancillary stair spans onto a slab edge, in order to decompose the stair load on to the slab you can add an inactive member along just that part of the slab edge to which the stair attaches.



Provided that the slab is meshed for 3D analysis, the inactive member provides a load path - the load becomes point loads on slab at each end inactive member (i.e. at the edges of the stair).



Create supports

Supports allow you to constrain points in your structure vertically and rotationally. You can use supports to model connections to existing structures, so that you do not need to incorporate them in your current model. In addition, you can change the fixity provided at an existing support by modifying the support properties.

Create a single support

RESTRICTION	You can only place supports at existing grid points.

1. On the **Model** tab, click **Support**.

   The support will adopt the properties currently displayed in the **Properties** window.

2. If necessary, in the **Properties** window, adjust the support properties.
3. Click the grid point where you want to create the support.

Create a rotated support using 3 grid points

You can apply a local coordinate system to your supports, allowing the X and Y axes to be rotated about global the Z axis.

NOTE	The z direction of the support cannot be changed.

1. On the **Model** tab, click **Support**.
2. Go to the **Properties** window.
3. Ensure that the **3 Grid Points** option is selected.
4. Click the grid point where you want to create the support.
5. To define the support direction along the x axis, click a second grid point.
6. To define the support direction along the y axis, click a third grid point.

Create spring supports

You can create linear and non linear spring supports in a similar fashion.

1. On the **Model** tab, click **Support**.
2. Go to the **Properties** window.
3. Ensure that in the direction where you want the spring to act, the degree of freedom is set to **Free**.
4. In the appropriate stiffness properties, do one of the following:
    - Select the **Spring Linear** option, and define a single stiffness value that acts in both the positive and negative directions.
    - Select the **Spring Non-linear** option, and define two stiffness values, one to act in the positive direction and another to act in the negative direction.
5. Click the grid point where you want to create the support.

Create nominally pinned or nominally fixed supports

A support placed under a column is a special case which can be specified as nominally pinned, or nominally fixed if required.

NOTE	A validation error is produced if a nominally pinned or nominally fixed support is placed at any other location.

1. On the **Model** tab, click **Support**.
2. Go to the **Properties** window.

3. Ensure that the Mx and/ or My directions about which you require partial fixity are set to **Free**.
4. In the appropriate rotational stiffness properties, do one of the following:
    - Select the **Nominally pinned** option, and define the required stiffness value as a percentage of the column stiffness 4EI/L.
    - Select the **Nominally fixed** option, and leave the stiffness value as 100% of the column stiffness 4EI/L.
5. Click the grid point where you want to create the support.

For more information, see: Partial fixity of column bases (page 316)

Modify support properties

You can modify support properties in both 2D and 3D views.

1. Hover the mouse pointer over the desired support, so that it becomes highlighted.
2. Click the support.

    TIP You can add further supports to the current selection by holding down the **Ctrl** key and clicking the additional supports.

3. Go to the **Properties** window.
4. Modify the support properties according to your needs.

    Tekla Structural Designer automatically applies any changes to the selected supports.

Partial fixity of column bases

Two additional types of rotational linear spring are provided to allow partial fixity to be modelled, these are:

- Nominally pinned
- Nominally fixed

These are specifically provided for supports under columns (of any material), but will result in a validation error if placed under walls, or if they are used for any other supports.

The support stiffness is based on the column properties (E*I/L)

- E = Young's Modulus of the column
- I = relevant bending stiffness (Ixx or Iyy) of the column
- L = distance from the support to the first column point (stack) that is on a Construction Level checked as a Floor in the Levels dialog, i.e. combined length of all the stacks until a floor is found.

NOTE Where no Floor has been defined in the column above the support, then L is taken as total length of column.

Partial fixity spring stiffness is calculated as follows for each of the two bending releases Mx and My:

- Nominally pinned (spring stiffness) - x% * 4 * E*I/L (default x% = 10%)
- Nominally fixed (spring stiffness) - x% * 4 * E*I/L (default x% = 100%)

Since the spring stiffness is dependent upon stack height and column stiffness (E and I), the spring stiffness will change if any changes are made to column stack height, column E or I values.

In addition, since for steel, Auto Design can change the column size (and hence I value) the spring stiffness will change with any change in column size.

Create analysis elements

Linear and non-linear elements and springs can be modeled in Tekla Structural Designer as analysis elements.

See also

Analysis Element properties (page 1020)

Element types (page 318)

Create analysis elements

1. On the **Model** tab, click **Element**.
2. In the **Properties** window, select the Element type (page 318) that you require.
3. In the **Properties** window, adjust the material and/or properties of the element according to your needs.
4. Click the start point of the element.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

5. Click the end point of the element.

   Tekla Structural Designer creates the element.

Create analysis element springs

1. On the **Model** tab, click **Element**.

   The element will adopt the properties currently displayed in the **Properties** window.

2. In the **Properties** window, select the spring Element type (page 318) that you require.
3. Enter the the material and properties of the element according to your needs.
4. Click the start point of the element.

   TIP If you are using a point along a member, do the following:

   a. Click the member to view its points.

   b. Click the point that you want to use, or type the distance from the start of the member to the desired point.

   ---

5. Click the end point of the element.

   Tekla Structural Designer creates the element.

6. Select the newly created element.
7. In the **Properties** window, enter the spring stiffness properties that you require.

Modify the position of analysis elements

You can modify the position of an individual analysis element in both 2D and 3D views.

1. Select the element that you want to move.
2. Select the end node that you want to move.

   NOTE Ensure that only the desired end node is highlighted in the **Select Entity** tooltip.

   ---

3. Click the grid or construction point where you want to move the end node.

   Tekla Structural Designer moves the end node to the selected point.

Element types

This topic introduces the different element types available when creating Analysis Elements

Beam

The **Beam** element type is defined as a 2 node element with 6 potential releases; 3 translational (Fx, Fy, Fz) and 3 rotational (Mx, My, Mz); at each end.

Material properties and the following section properties are required:

- Ax
- A parallel to minor
- A parallel to minor
- Ix
- I major
- I minor

Truss

The **Truss** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

Material properties and the area of the section are required.

This element type acts in both tension and compression.

Tension only

The **Tension only** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

Material properties and the area of the section are required.

This element type acts in tension only.

NOTE Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements

---

Compression only

The **Compression only** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

Material properties and the area of the section are required.

This element type acts in compression only.

NOTE Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements

---

Linear axial spring

The **Linear axial spring** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

An axial **Spring Stiffness** property is required.

This element type acts in both tension and compression.

Linear torsional spring

The **Linear torsional spring** element type is defined as a 2 node element which has torsional (Mx) fixity at each end. The remaining 5 degrees of freedom; 3 translational (Fx, Fy, Fz) and 2 rotational (My, Mz); are released at each end.

A rotational **Spring Stiffness** property is required.

Non-linear axial spring

The **Non-linear axial spring** element type is defined as a 2 node element which has axial (Fx) fixity at each end. The remaining 5 degrees of freedom; 2 translational (Fy, Fz) and 3 rotational (Mx, My, Mz); are released at each end.

Seperate tension and compression **Spring Stiffness** properties are required and also the maximum tension and compression values.

NOTE Non-linear elements require non-linear analysis. If linear analysis is performed they will be treated as linear elements.

Non-linear torsional spring

The **Linear torsional spring** element type is defined as a 2 node element which has torsional (Mx) fixity at each end. The remaining 5 degrees of freedom; 3 translational (Fx, Fy, Fz) and 2 rotational (My, Mz); are released at each end.

Seperate tension and compression torsional **Spring Stiffness** properties are required and also the maximum torsional tension and compression values.

NOTE Non-linear elements require non-linear analysis. If linear analysis is performed they will be treated as linear elements.

Link

A basic link element is implemented in Tekla Structural Designer, defined as a 2 node element with 6 springs; 3 translational (Fx, Fy, Fz) and 3 rotational (Mx, My, Mz); at end 2.

This element type is used internally to effect partial fixity in beams.

4.3 Edit the model

After creating the model and the necessary members within it, you may need to make some modifications.

For example you may want to:

- Edit the properties of (page 168) entities
- Re-position entities (page 169) by moving nodes or edges

Additional model editing commands are located on the **Edit** tab, these allow you to:

- copy objects (page 321) and loads
- move objects (page 322) or move the model
- mirror objects (page 322)
- delete objects (page 329)
- join and split members
- Automatically join all concrete beams (page 331)
- Reverse member axes and panel faces (page 332)
- use cutting planes to hide a part of your model
- remove any unused unused slopes, frames, construction and grid lines
- merge planes (page 337)
- Rationalize the model (page 335)
- create infill members
- edit or add free points

Copy and rotate objects

To copy, or copy and rotate objects, see the following instructions.

1. Select the objects that you want to copy.
2. On the **Edit** toolbar, click **Copy**.
3. If you want to rotate the object when you copy it, in the **Properties** window, type the rotation about the Z axis.
4. Click the reference node.
5. Click the point where you want to copy the reference node.

   Tekla Structural Designer moves the selected member to the new location, and rotates them according to the specified rotation value.
6. Place more copies in your model, or press **Esc** to finish.

Move and rotate objects

To move, or move and rotate objects, see the following instructions.

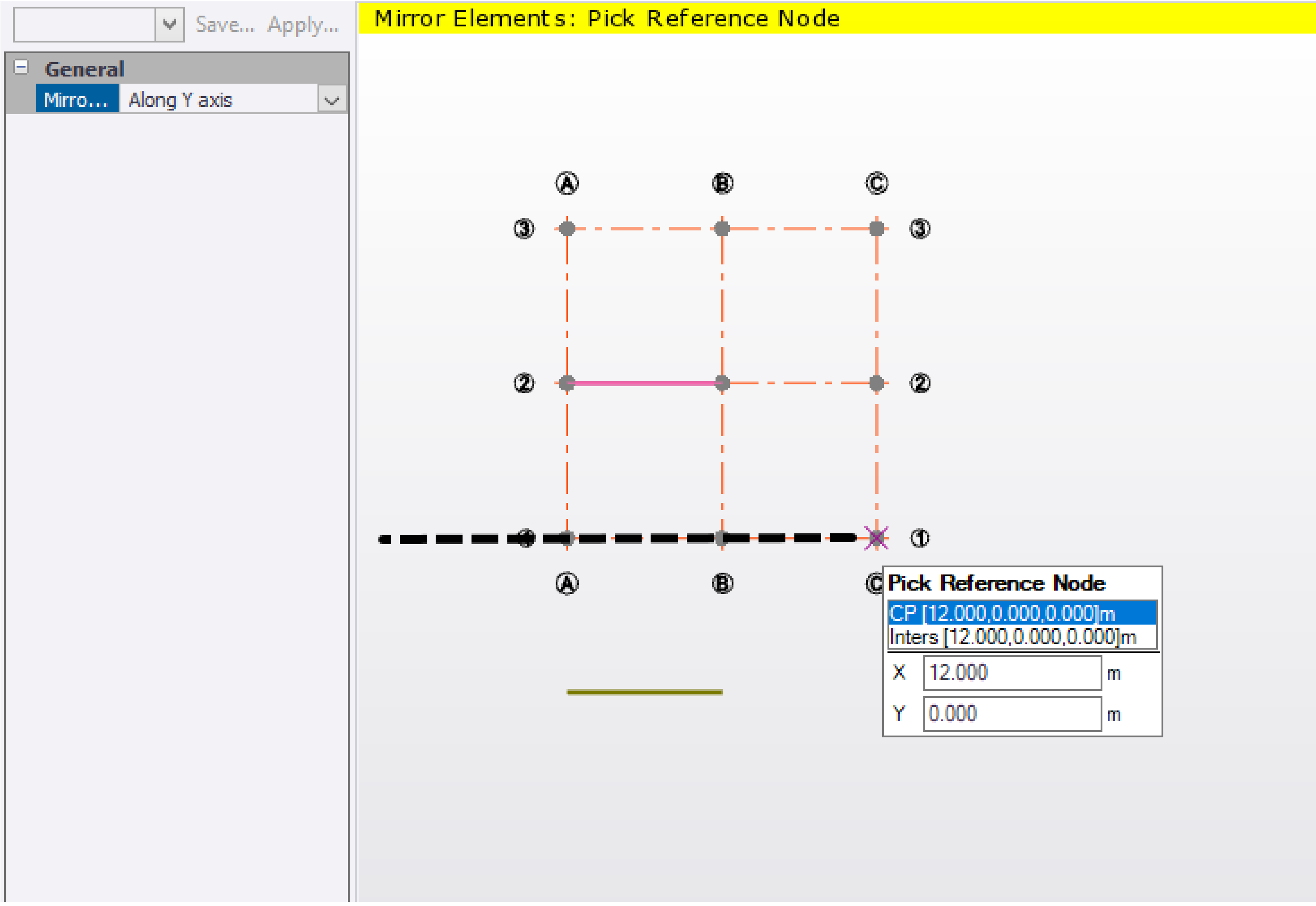
1. Select the objects that you want to move.
2. On the **Edit** toolbar, click **Move**.
3. If you want to rotate the object while moving it, in the **Properties** window, type the rotation about the Z axis.
4. Click the reference node.
5. Click the point where you want to move the reference node.

   Tekla Structural Designer moves the selected member to the new location, and rotates them according to the specified rotation value.

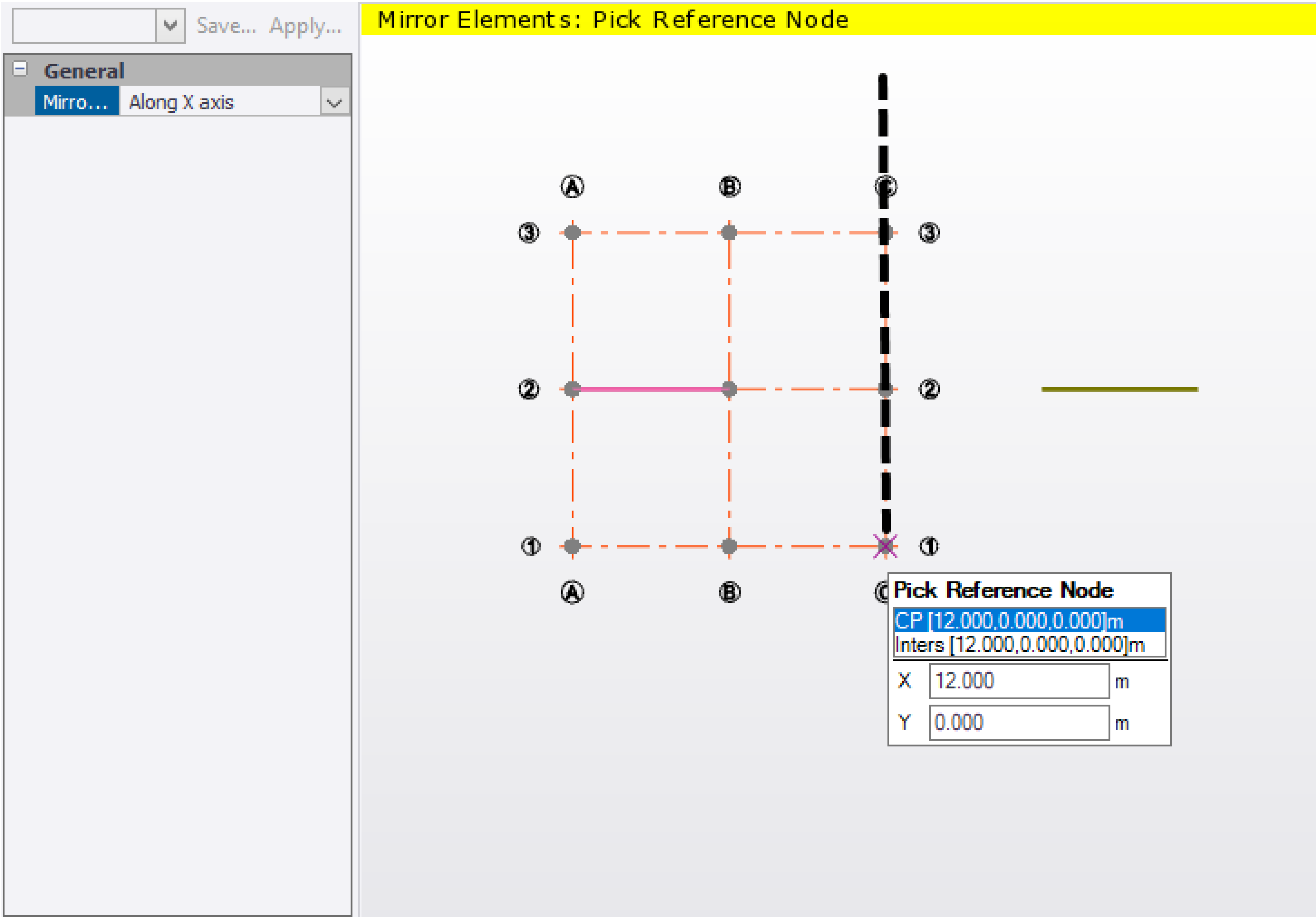
Mirror objects to new locations

To copy existing elements by mirroring them, see the following instructions.

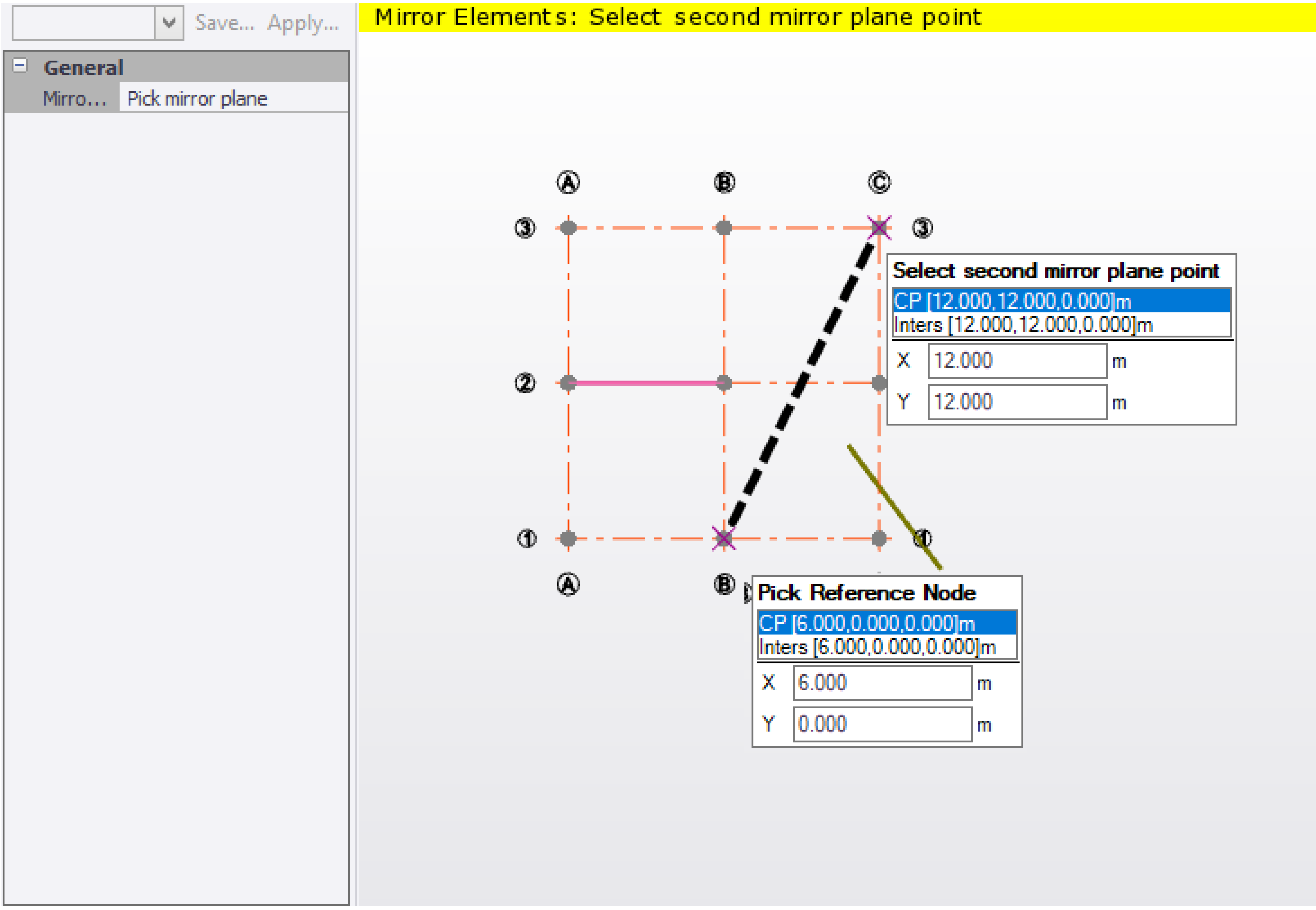
1. Select the objects that you want to mirror.
2. On the **Edit** toolbar, click **Mirror**.
3. In the **Properties** window, select the desired **Mirror mode**:
   - **Along Y axis** mirrors the objects about a global XZ plane defined by a single reference point.



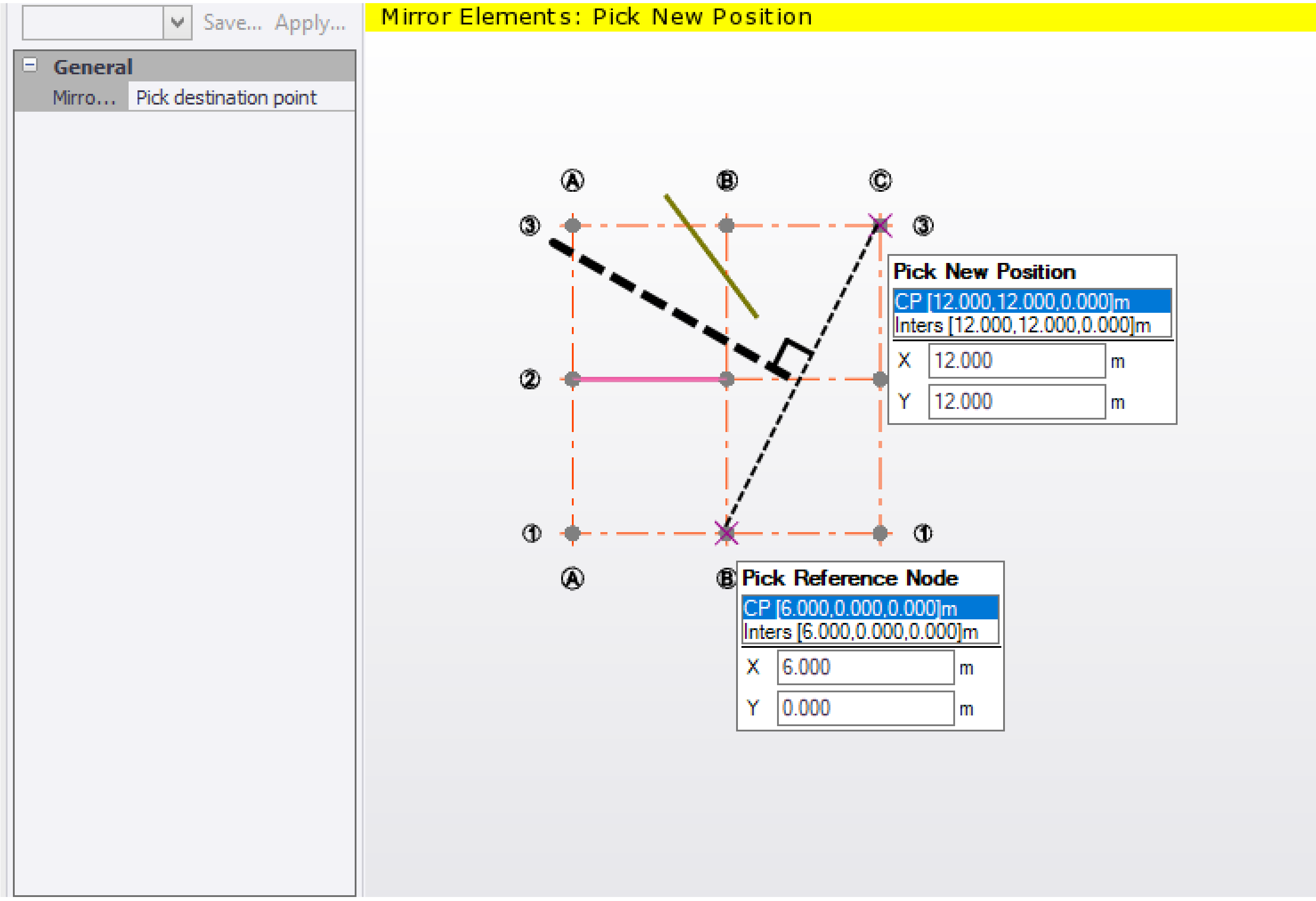
- **Along X axis** mirrors the objects about a global YZ plane defined by a single reference point.



- **Pick mirror plane** mirrors the objects about a plane defined between two points.



- **Pick destination point** mirrors the objects about a plane which perpendicularly bisects a line defined between two points.



4. In the model, click the reference points.
5. If you are using the **Pick mirror plane** or **Pick destination point** mode, click the second point to define the mirror plane.

See also

Copy and rotate objects (page 321)

Copy loads

Tekla Structural Designer allows you to quickly apply the same loads to multiple members by using the **Copy Loads** command.

Copy all member loads from one span to another

1. On the **Edit** toolbar, click **Copy Loads**.

2. Go to the **Properties** window.
3. Set **Mode** to **Copy Span Loading**.
4. Select whether you want to copy loads in the current loadcase or in all loadcases.

NOTE If you select the current loadcase, remember to select the required loadcase in the (page 55).

5. In the model, click the span that contains the member loads that you want to copy.
6. Click the span that you want to apply the loads to. The source span and the destination span do not need to be the same length.

   Tekla Structural Designer copies the member loads to the selected span.
7. Do one of the following:
   - Click additional spans to continue applying the loads.
   - Press **Esc** to select a different loaded element span to copy loads from.
   - Press **Esc** twice to finish.

Only copy one member load to another span

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the **Properties** window, set **Mode** to **Copy Member/Area Load**.
3. In the model, click the member load that you want to copy.
4. Click the span that you want to apply the load to. The source span and the destination span do not need to be the same length.

   Tekla Structural Designer copies the member load to the selected span.
5. Do one of the following:
   - Click additional spans to continue applying the loads.
   - Press **Esc** to select a different load to copy.
   - Press **Esc** twice to finish.

Copy panel area, level, and slab loads

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the (page 55), select the desired loadcase.
3. In the **Properties** window, set **Mode** to **Copy Member/Area Load**.
4. Click the area, slab, or level load that you want to copy.

5. Click the panel that you want to apply the load to.

   Tekla Structural Designer copies the load to the selected panel.
6. Do one of the following:
   - Click additional panels to continue applying the loads.
   - Press **Esc** to select a different load to copy.
   - Press **Esc** twice to finish.

Copy panel point, line, and patch loads

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the (page 55), select the desired loadcase.
3. In the **Properties** window, set **Mode** to **Copy Plane Loads**.
4. Click the panel point, line, or patch load that you want to copy.

   TIP If necessary, you can click to add other loads to the selection, or remove them from the selection by clicking them once more.

   A red circle indicates the original reference point for the selected loads.
5. To apply the loads, do one of the following:
   - Click a panel node to define a new reference point at that node.

     The loads are applied at the same offset from the new reference point.
   - Click anywhere within a panel boundary at a different level to define a new reference point directly above or below the original reference point.

     The loads are applied at the same offset from the new reference point.

     WARNING If you choose a new reference point that results in the loads being applied outside the panel area, Tekla Structural Designer does not apply the loads to the model. In this situation, a warning appears during validation.
6. Do one of the following:
   - Click to define further reference points to continue applying the loads. .
   - Press **Esc** to select a different loads to copy.
   - Press **Esc** twice to finish.

Copy structure loads

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the **Properties** window, set **Mode** to **Copy Member/Area Load**.
3. In the model, click the structure load to be copied.
4. Click the position you want to apply the load to.

   Tekla Structural Designer copies the load to the new location.
5. Do one of the following:
   - Click additional locations to continue applying the load.
   - Press **Esc** to select a different load to copy.
   - Press **Esc** twice to finish.

Copy loads to another loadcase

1. On the **Edit** toolbar, click **Copy Loads**.
2. In the (page 55), select the loadcase from which you want to copy the loads.
3. Go to the **Properties** window.
4. Set **Mode** to **Copy Loads to another Loadcase**.
5. Select the loadcase that you want to copy the loads to.
6. In the model, click the load that you want to copy.

   TIP If necessary, you can click to add other loads to the selection, or remove them from the selection by clicking them once more.

7. To copy the loads to the loadcase displayed in the **Properties** window, press **Enter**.
8. Do one of the following:
   - Press **Esc** to select additional loads, and then press **Enter** to copy them.
   - Press **Esc** twice to finish.

Delete entities

1. Do one of the following:

To	Do this
Delete using the keyboard	• Select the entity. • Press **Delete**
Delete using the Quick Access toolbar	• Select the entity.

	• From the Quick Access toolbar choose **Delete**
Delete using the context menu	• Right click the entity you want to delete. • From the context menu choose **Delete Element**
Delete from the Edit ribbon	• Select the entity. • From the Edit ribbon choose **Delete**

Join and split members

You can use the **Join** command for joining discontinuous members, thus creating a continuous member. The **Split** command, on the other hand, allows you to split continuous members of any material.

See also

Automatically join all concrete beams (page 331)

Join members

You can use the **Join** command to manually join concrete beams, even when the **Allow automatic join end** options are not selected. This is because the **Allow automatic join end** only applies to the automatic joining that occurs during design process or when you run the **Beam Lines** command.

RESTRICTION You can only join:

- Two similar members if that the angle between the members is less than 45 degrees in both plan and elevation.
- Beams that have an end point in common.

  If Tekla Structural Designer fails to join two beams, their end points have probably been defined using different construction or grid lines.

1. On the **Edit** toolbar, click **Join**.
2. Hover the mouse pointer over the member that you want to join to another member.

   Both the original member and the member to which it will be joined become highlighted. The point where they will be joined is indicated by a red dot.

   NOTE If Tekla Structural Designer tries to join the wrong end, move the mouse pointer towards the other end of the member until Tekla Structural Designer indicates the correct end.

3. Click the highlighted members to join them.
4. Click additional members to join them, or press **Esc** to finish.

Split members

RESTRICTION You can only split members that have previously been joined.

1. On the **Edit** toolbar, click **Split**.
2. Hover the mouse pointer over previously joined members that you want to split.

   The member is highlighted, and Tekla Structural Designer uses a red dot to indicate the point where the member will split.
3. According to your needs, do one of the following:
   - Click the member to split it at the indicated point.
   - Move the mouse pointer further along the member to identify other points where the member could be split.

   NOTE If the member being split is a concrete beam, Tekla Structural Designer splits the member into two separate beams. The first beam has the **Allow automatic join end 2** option cleared, whereas the second one has the **Allow automatic join end 2** option cleared. This prevents Tekla Structural Designer from automatically making the two beams continuous again when the model is designed.

Automatically join all concrete beams

Tekla Structural Designer automatically forms concrete beam lines as part of the combined analysis and design process. However, if you would prefer to have greater control yourself, you can run the **Beam Lines** command manually. This way, you can verify that continuous beam lines are formed as you intend before proceeding with the design.

When run manually, the **Beam Lines** command applies to all concrete beam members in the model, irrespective of whether they are selected or not.

NOTE Discontinuous concrete beam ends can only be joined if the **Allow automatic join end** option is selected for the appropriate beam ends at the join. Then, Tekla Structural Designer only joins the ends if all the following criteria are met:

- The angle in plan at which the two beams meet is less than the **Limiting join angle in plan** specified in **Model Settings** --> **Beam Lines**

- The angle in elevation at which the two beams meet is less than the **Limiting join angle in elevation** specified in **Model Settings** --> **Beam Lines**
- The amount by which the cross sectional areas of the two beams overlap is greater than the **Minimum section overlap** percentage specified in **Model Settings** --> **Beam Lines**

In addition, if either of the two beam ends being joined is pinned, Tekla Structural Designer does not join them unless the **Join pinned beam end** option is selected in **Model Settings** --> **Beam Lines**.

To join the concrete beams in the model automatically, do the following:

- On the **Edit** tab, click **Beam Lines**.

See also

Join and split members (page 330)

Reverse member axes and panel faces

Sometimes when creating a model, you may end up creating beams or wall panels that run or face in the wrong direction. This may lead to problems in the analysis phase. In order to fix these kinds of inconsistencies, you can use the **Reverse** command.

Reverse the local axis of a beam

You can easily end up with beams running forwards and backwards if a consistent approach has not been adopted when selecting start and end nodes. Beams that run in different directions can result in confusing force diagrams. By manually reversing the axes of beams, you can make all beams run left to right and bottom to top in a plan view.

To reverse a beam, do the following:

1. On the **Edit** toolbar, click **Reverse**.
2. Click a beam to reverse its direction.

Reverse the outward face of a wind panel

The front of each wall should be facing outwards in order to correctly determine the wind direction relative to the wall. Ensure that all the outward faces are displayed in the color assigned to the front of the wind wall.

If a wall is facing in the wrong direction, reverse it by doing the following:

1. On the **Edit** toolbar, click **Reverse**.
2. Click a wind panel to reverse its direction.

Manage cutting planes

Initially, the six cutting planes form a cube around the extents of each model. By activating a cutting plane, you can move it inwards so that it slices through the model. Everything on the positive side of the plane is temporarily hidden from the view, making it easier to work on different areas inside the model.

Activate or deactivate a cutting plane

The active cutting plane faces are shown in a different colour. (By default blue indicates the positive side of the plane and red the reverse side).

1. On the **Edit** toolbar, click **Cutting Planes**.

   Tekla Structural Designer displays the six existing cutting planes.

2. Click a cutting plane to activate it.

   Tekla Structural Designer displays the positive side of the plane in blue, and the reverse side in red.

   TIP To deactivate a cutting plane, click an active plane again.

Move a cutting plane to hide a part of the model

When a cutting plane is active, an arrow projects from its center. You can use the arrow to reposition the plane.

1. Click the arrow at the center of the cutting plane.

   Tekla Structural Designer views a line that indicates the direction in which you can move the plane.

2. Click the desired position of the plane, or press **F2** to type the exact distance.

   Tekla Structural Designer redraws the cutting plane at its new position. The cutting plane slices to the model, and everything to the positive side of the plane is hidden.

Re-display a hidden part of the model

- Click the cutting plane to deactivate it.

  Tekla Structural Designer displays any previously hidden parts of the model.

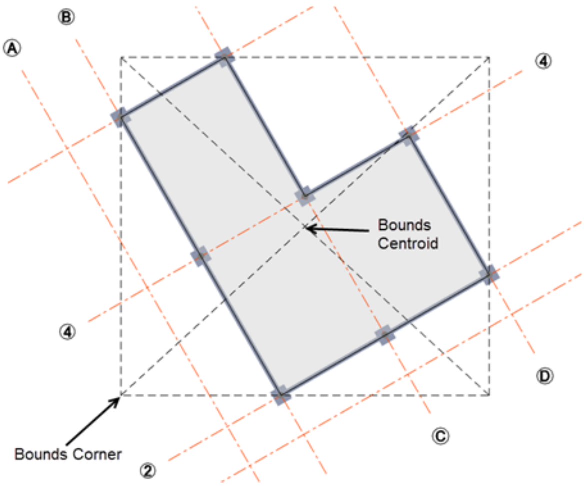
Move the model or the DXF shadow

If necessary, you can relocate the entire model to a new coordinate in the XY plane by using the **Move Model**. You can relocate the DXF shadow of the model similarly with the **Move DXF Shadow** command.

Move the model

In order to move the model, you first have to decide which reference point to use.

You can determine the reference points by drawing an imaginary box aligned to X and Y around the extents of the model, as shown in the following image:



Bounds Corner equals the lower left hand corner of the imaginary box, whereas **Bounds Centroid** equals the centroid of the imaginary box.

1. On the **Edit** toolbar, click **Move Model**.
2. Set the reference point to either **Bounds Corner** or **Bounds Centroid**.
3. Enter the required target coordinate of the reference point.
4. Click **Move**.

Move the DXF shadow

The **Move DXF Shadow** command is only available if you have previously imported a DXF shadow (page 116).

1. On the **Edit** toolbar, click **Move DXF Shadow**.
2. Select an existing point in the DXF shadow as the reference point.
3. Click the new position of the reference point.

Rationalize the model

In order to remove all unused sloped planes, frames, grids, and construction lines in your model, you can use the **Rationalize** command. If necessary, the **Rationalize** command also allows you to update all grid and construction lines to extend only a short length beyond the point where they are required. After running the command, you can see a summary of the changes in the **Process Window**.

Delete unused sloped planes, frames, grids, and construction lines

1. On the **Edit** toolbar, click **Rationalize**.

   The **Rationalize** dialog box opens.
2. Select the unused items that you want to remove.
3. Click **OK**.

   Tekla Structural Designer deletes the unused items of the selected types, and their associated items.

Update grid and construction line length

1. On the **Edit** toolbar, click **Rationalize**.

   The **Rationalize** dialog box opens.
2. Select **Shrink/Extend Grid & Construction lines**.
3. Click **OK**.

   Tekla Structural Designer updates all grid and construction lines in the model to extend a fixed length (500 mm / 1'8") beyond the point where they are required.

Create infill members

In order to quickly place a pattern of infill members into selected plays in a level, sloped plane, or frame, you can use the **Create Infills** command. Note that only the bays with members attached to all sides can be selected.

RESTRICTION	You cannot create infills in 3D views.

Define the infill properties and pattern

1. On the **Edit** toolbar, click **Create Infills**.
2. Go to the **Properties** window.
3. In **Define Beams**, select one of the following:
    - **By number** allows you to specify the number of equally spaced members in the pattern.
    - **By spacing** allows you to specify the exact positions of the beams, separating them by a comma.
    - **By max spacing** allows you to create as many equally spaced beams in the pattern as possible without exceeding the max spacing.
4. In **Direction**, select one of the following:
    - **Perpendicular** allows you to have the members drawn perpendicular to the highlighted edge member.
    - **Parallel with left** allows you to have the members drawn parallel to the edge member that connects to the end 1 of the highlighted edge member.
    - **Parallel with right** allows you to have the members drawn parallel to the edge member that connect to the end 2 of the highlighted edge member.
5. Define the element parameters according to your needs.

    TIP If you have saved infill properties to a named property set, you can recall them by selecting the set in the drop list at the top of the **Properties** window.

Place the pattern in a single bay

1. Hover the mouse pointer over the required bay, adjacent to the required edge member.
2. If necessary, change the orientation by moving the cursor to a different edge member, or by adjusting the general parameters in the **Properties** window.

3. Click the bay to create the pattern.

Place the pattern in multiple bays

1. Hover the mouse pointer over one of the bays, adjacent to the required edge member.
2. If necessary, change the orientation by moving the cursor to a different edge member, or by adjusting the general parameters in the **Properties** window.
3. Click the bays in which you want to create the pattern.

Merge planes

Where a model contains very close planes, as can often occur in imported models, for example, the Merge Planes command allows the you to merge two planes by selecting a destination plane and then a source plane.

1. On the **Edit** toolbar, click **Merge Planes**.

   Existing Level, Frame, and Slope planes are displayed in the active view.

   NOTE If the view is cluttered by having all the plane types displayed simultaneously, and you only want to merge Levels, consider switching off Frames and Slopes in Scene Content.

2. Pick the destination plane.
3. Pick the source plane.

   If the planes can be merged, everything on the source plane will be moved to the destination plane, and the source plane will be deleted. If the merge is not possible, a message is given in the Process window and the model will be unchanged.

Create and manage free points

Free points can be used to connect beams, columns, braces etc. without having to define an intersection between grid lines or construction lines.

NOTE Once a free point has an item connected to it, if the item is moved a new point is created and the existing free point remains.

Create a free point

A frame is a 2D View of the model, created in a vertical plane defined by an existing grid line. Since only the members that lie within the plane of the frame are displayed, a frame view can be particularly useful for defining bracing.

1. Make sure that Free Points are visible in Scene Content.
2. Right click in the view and pick Add free point... from the context menu.
3. Enter the point co-ordinates and click OK.

Adding, moving or deleting free points from the Edit tab

1. On the **Edit** tab, click **Free Points...**

   A Free Points dialog is displayed.
2. Click **Add** to add a new point, **Delete** to remove an existing point, or click in the table to edit the co-ordinates of existing points.

Related video

Free point defined by XYZ coordinates

4.4 Validate the model

You can validate the model at any time in order to trap errors that will cause the solver to fail before the model is submitted for analysis. Validation is also automatically performed during design.

NOTE You can also measure distances and angles (page 339) to check your modeling.

Run model validation

- On the **Model** or **Load** tab, click **Validate**.

  Tekla Structural Designer performs the validation checks. If your model contains any issues, warning messages appear.

Adjust the conditions considered in model validation

1. On the **Home** tab, click **Model Settings**.

2. Go to the **Validation** page.
3. Select the conditions that you want Tekla Structural Designer to consider during validation checks.
4. Click **OK**.

Measure distances and angles

To know the exact distances or angles between different points in the model, you can use the Tekla Structural Designer measuring commands. You can find the measuring commands on the **Model** tab, in the **Miscellaneous** group.

Measure distances

To measure the distance between any two points in the model, see the following instructions.

1. On the **Model** tab, click **Measure**.
2. Click a node to define the start position.
3. Click a second node to define the end position.

   The distance between the nodes is displayed in the current view. To clear the measurement, press **Esc**.

Measure angles

RESTRICTION	You can only measure angles in 2D Views.

1. On the **Model** tab, click **Measure Angle**.
2. Click a node to define the arc center.
3. Click a second node to define the start position.
4. Click a third node to define the end position.

   The clockwise angle between the start and end position is displayed in the current view. To clear the measurement, press **Esc**.

5 Apply loading

The **Load** tab allows you to:

- Manage loadcases, groups, combinations, envelopes and patterns (page 340)
- Apply panel, member, and structure loads (page 364)
- Apply wind, snow, and seismic loads (page 387)

5.1 Manage loadcases, groups, combinations, envelopes and patterns

Loadcases, groups, combinations and envelopes are all managed in the Loading dialog (page 357). Load patterns are reviewed via **Update Patterns** on the **Load** tab.

- Manage loadcases (page 340)
- Manage load groups (page 342)
- Manage load combinations (page 344)
- Manage load patterns (page 353)

Manage loadcases

Before applying loads to your model, you must first define the loadcases within which the loads will be contained.

To define loadcases, see:

- Create loadcases (page 341)
- Activate reductions in live or imposed loadcases (page 341)
- Renumber all loadcases (page 342)

Create loadcases

When you create a new model, Tekla Structural Designer automatically creates a loadcase whose type is **Self weight - excluding slabs**. You cannot access the loadcase because Tekla Structural Designer automatically calculates the loads within it using the objects in your structure. Tekla Structural Designer also creates three other loadcases that are initially empty. However, you will almost certainly need to create other loadcases that contain the loads that your building must withstand.

1. On the **Load** tab, click **Loadcases**.

   The Loading dialog (page 357) opens on the **Loadcases** page. On this page, you can see all currently existing loadcases.
2. Click **Add**.
3. In **Loadcase Title**, name the loadcase.
4. In **Type**, select the desired loadcase type.
5. Select whether the loadcase is included when you automatically generate load combinations.
6. Click **OK**.

   Tekla Structural Designer adds the new loadcase to the list of loadcases in the **Loading** list.

Activate reductions in live or imposed loadcases

When you create a loadcase whose type is **Live** (US) or **Imposed** (other head codes), you can allow Tekla Structural Designer to automatically calculate load reductions in accordance with the percentages specified on the Load reductions page in **Model Settings**. To do so, see the following instructions.

Activate reductions in live and roof live loadcases (US)

1. On the **Load** tab, click **Loadcases**.

   The Loading dialog (page 357) opens on the **Loadcases** page. On this page, you can see all currently existing loadcases.
2. Select the **Live** or **Roof Live** loadcase to which you want to apply the reductions.
3. Select the **Live Load Reductions** option.
4. Click **OK**.

Activate reductions in imposed loadcases (other head codes)

NOTE You cannot activate reductions in loadcases whose type is set to **Roof Imposed**.

1. On the **Load** tab, click **Loadcases**.

   The Loading dialog (page 357) opens on the **Loadcases** page. On this page, you can see all currently existing loadcases.
2. Select the **Imposed** loadcase to which you want to apply the reductions.
3. Select the **Imposed Load Reductions** option.
4. Click **OK**.

Renumber all loadcases

When you delete loadcases from the Loading dialog (page 357), the remaining loadcases retain their original loadcase number. If necessary, you can renumber the remaining loadcases in sequence.

For more information, see the following instructions.

1. On the **Loading** tab of the **Project Workspace**, right-click **Loadcases**.
2. In the context menu, select **Renumber**.

Manage load groups

Load Groups facilitate the generation of combinations (primarily for design of industrial structures where many loading scenarios must be considered).

To manage load groups, see:

- Overview of load groups (page 342)
- Create load groups (page 343)
- Inclusive and exclusive load groups example (page 344)

Overview of load groups

Features of load groups

- Load Groups are an aid to building Combinations - both manually and using the Generator - and their use is entirely optional.
- Once a Combination is built using Load Groups, the link to the Load Group is lost – the Combination is made up only of Loadcases. Only Loadcases and Combinations are analysed – Load Groups are not.

- Each Load Group holds items of a single load type - e.g. “Dead” or “Live”...etc. - and can contain both multiple loadcases and other Load Groups
- The fundamental Load Groups setting is the “Class” which is either “Inclusive” or “Exclusive”
  - When “Inclusive” – all loadcases are added at once into a combination
  - When “Exclusive” – loadcases are used one at a time in combinations. Thus for example, where an Exclusive Load Group contains four loadcases, the Generator will produce four combinations for all required combinations which include the group’s load Type (e.g. “Imposed”), each containing only one of the group’s four cases.

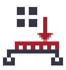
The load groups process

To make use of Load Groups you would proceed as follows:

1. Define Loadcases as normal
2. If required, create Load Groups from Loadcases and/or other Load Groups
3. Create Combinations by combining Loadcases and/or Load Groups

The end result is Loadcases and Combinations (built up of factored loadcases). These are then run through analysis and design.

Create load groups

1. On the **Load** tab, click  **Load Groups**.

   The Loading dialog (page 357) opens on the **Load Groups** page. On this page, you can see all currently existing load groups.
2. Click **Add**.
3. Enter the **Load Group Title**.
4. In **Class**, select the desired load group class.
   - Inclusive - all loadcases are added at once into a combination
   - Exclusive – loadcases are used one at a time in combinations
5. Click the load group name in the left hand panel of the dialog to display the available loadcases.
6. Select each loadcase in turn to be included and click the right arrow button to copy it into the load group.

   NOTE A load group can only contain items of one load type.

7. Click **OK**.

   Tekla Structural Designer adds the new load group to the list of load groups.

Inclusive and exclusive load groups example

If you were to manually create a combination to include:

- Loadcase 1
- Load Group 1 ("Inclusive" which contains Loadcases 2 & 3)
- Load Group 2 ("Inclusive" which contains Loadcases 4 & 5)
- Load Group 3 ("Exclusive" which contains Loadcases 6 & 7)
- Load Group 4 ("Exclusive" which contains Loadcases 8 & 9)

When you click OK to close the Loading dialog, the following combinations are created with the relevant factors according to Load Type

Combination	Contains Loadcases
Combination 1	1 + 2 + 3 + 4 + 5 + 6 + 8
Combination 2	1 + 2 + 3 + 4 + 5 + 6 + 9
Combination 3	1 + 2 + 3 + 4 + 5 + 7 + 8
Combination 4	1 + 2 + 3 + 4 + 5 + 7 + 9

See also

Create load groups (page 343)

Manage load combinations

Load combinations allow you to assemble sets of loadcases, applying the appropriate factors for the strength and service condition. These factors are specific to the design code that you are using.

To manage load combinations, see:

- Load combination classes (page 344)
- Generate load combinations automatically (page 345)
- Create load combinations manually (page 346)
- Create modal mass combinations (page 347)
- Import loadcases and combinations from a spreadsheet (page 347)
- Renumber all load combinations (page 352)

Load combination classes

Combinations fall into five different classes, with a number of options available for each of the classes:

Combination class	Description	Active	Strength	Service
Construction Stage	Only required for design of composite beams	Not applicable	Not applicable	Not applicable
Gravity	Consists of gravity loads only (self weight, dead, slab dry, slab wet, imposed, roof imposed, snow)	On/Off	On/Off	On/Off
Lateral	In addition to gravity loads, contains lateral loads due to notional loads or wind	On/Off	On/Off	On/Off
Seismic	Consists of gravity and/or lateral loads as well as seismic loadcases	On/Off	On	Not applicable
Modal Mass	Only required if you perform a modal analysis	On/Off	Not applicable	Not applicable

- **Active**: Selecting and clearing the option switches the combination on and off for analysis and design.
- **Strength**: If the **Strength** option is not selected and the combination is active, the combination is not assessed for design.
- **Service**: If the **Service** options is not selected and the combination is active, the combination is not assessed for deflection.

Generate load combinations automatically

The easiest way to create load combinations is to generate the combinations automatically. In order to do so, see the following instructions.

NOTE **Construction Stage** combinations must be created manually.

1. On the **Load** tab, click **Combination**.

   The Loading dialog (page 357) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.

2. Click **Generate...**

   The **Combination Generator** dialog box opens.

3. On the first page of the dialog, specify the initial parameters according to your needs.

   Depending on the number of combination types that you selected on the first page, Tekla Structural Designer creates one or more pages of combinations.

4. Click **Next**.
5. Review each page of combinations, and adjust them as necessary.
6. Click **Next** to move on to the next page.
7. To save the load combination, click **Finish**.

   Tekla Structural Designer adds the load combination to the list of combinations available in the **Loading** list.

   TIP To review the factors and options that have been applied to a combination, click the combination name in the list on the left side of the **Loading** dialog box.

Create load combinations manually

If necessary, you can create load combinations manually. In this case, Tekla Structural Designer uses a default factor for each loadcase when you add it to the combination.

1. On the **Load** tab, click **Combination**.

   The Loading dialog (page 357) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.

2. Click **Add**.
3. In **Design Combination Title**, name the combination.
4. In **Class**, select the combination class.

5. Depending on the combination class that you selected, do some of the following:
   - Select **Active** in order to include the combination in analysis or design.
   - Select **Strength** in order to assess the combination for design.
   - Select **Service** in order to assess the combination for deflection.
6. In the left side pane of the **Loading** dialog box, click the combination name.
7. Select the loadcases that you want to add in the combination, and click **>>**.
8. Click **OK** to save the load combination.

   Tekla Structural Designer adds the new load combination to the list of combinations in the **Loading** list.

TIP To review the factors and options that have been applied to a combination, click the combination name on the left side pane of the **Loading** dialog box.

Create modal mass combinations

1. On the **Load** tab, click **Combination**.

   The Loading dialog (page 357) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.
2. Click **Add**.
3. In **Design Combination Title**, name the combination.
4. In **Class**, set the combination class to **Modal Mass**.
5. In the left side pane of the **Loading** dialog box, click the combination name.
6. Select the loadcases that you want to add in the combination, and click **>>**.
7. On the **Applied mass** tab, set the directions to be considered, and if necessary, specify the level below which mass can be ignored.
8. On the **Second order effects** tab, define the amplifier that you want to apply to the combination.
9. Click **OK** to save the load combination.

   Tekla Structural Designer adds the new load combination to the list of combinations in the **Loading** list.

Import loadcases and combinations from a spreadsheet

Loadcases and combinations defined in an Excel Spreadsheet can be imported via the ‘Import...’ button on the Loading dialog.

The data must be in the form of a matrix of combination factors in which loadcases form the row headers and combinations the column headers or vice-versa. Where a matrix value is blank the associated case is omitted from the combination. The load type can be set by including in the loadcase name “DL” for dead load, “LL” for Imposed and “Wind” for Wind (without quotes).

The selected spreadsheet’s Worksheets are automatically listed and one is selected for import.

Flexible controls allow definition of whether Loadcases are listed in a column and combinations in a row or vice-versa and selection of the case and combination column/row sources. Identified cases and combinations can be reviewed prior to import and the case Type can be edited as necessary.

NOTE The following default/ automatically determined loadcases are not supported by this feature and would need to be manually added to combinations after import; Self weight - excluding slabs, Slab self weight, Notional Loads (EHF, NHF and NL).

Example

We want to import the following loadcases and combinations:

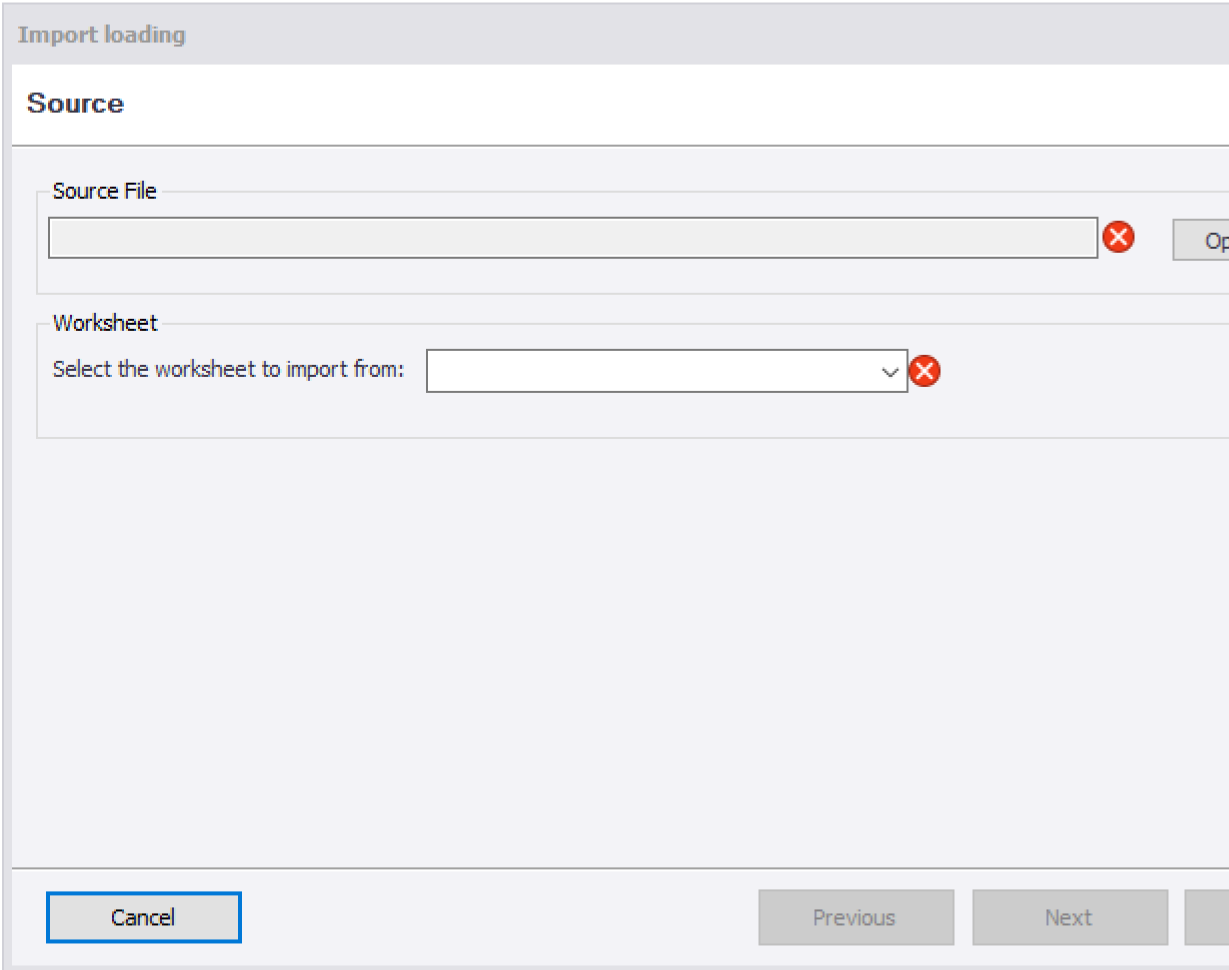
Combination Name	Loadcase	Strength Factor
dead + imposed	DL Dead	1.35
	LL Imposed	1.5
Dead+Imp+Wind Accomp	DL Dead	1.35
	LL Imposed	1.5
	Wind	0.75
Dead+Imp Accomp +Wind	DL Dead	1.35
	LL Imposed	1.05
	Wind	1.5
Dead+Wind	DL Dead	1.35
	Wind	1.5

1. In an Excel spreadsheet a matrix of loadcases and combinations is set out like this:

	A	B	C	D
1			Combinations	
2	Cases	Dead+Imposed	Dead+Imp+Wind Accomp	Dead+Imp Accomp+Wind
3	DL Dead	1.35	1.35	1.35
4	LL Imposed	1.50	1.50	1.05
5	Wind		0.75	1.50
6				

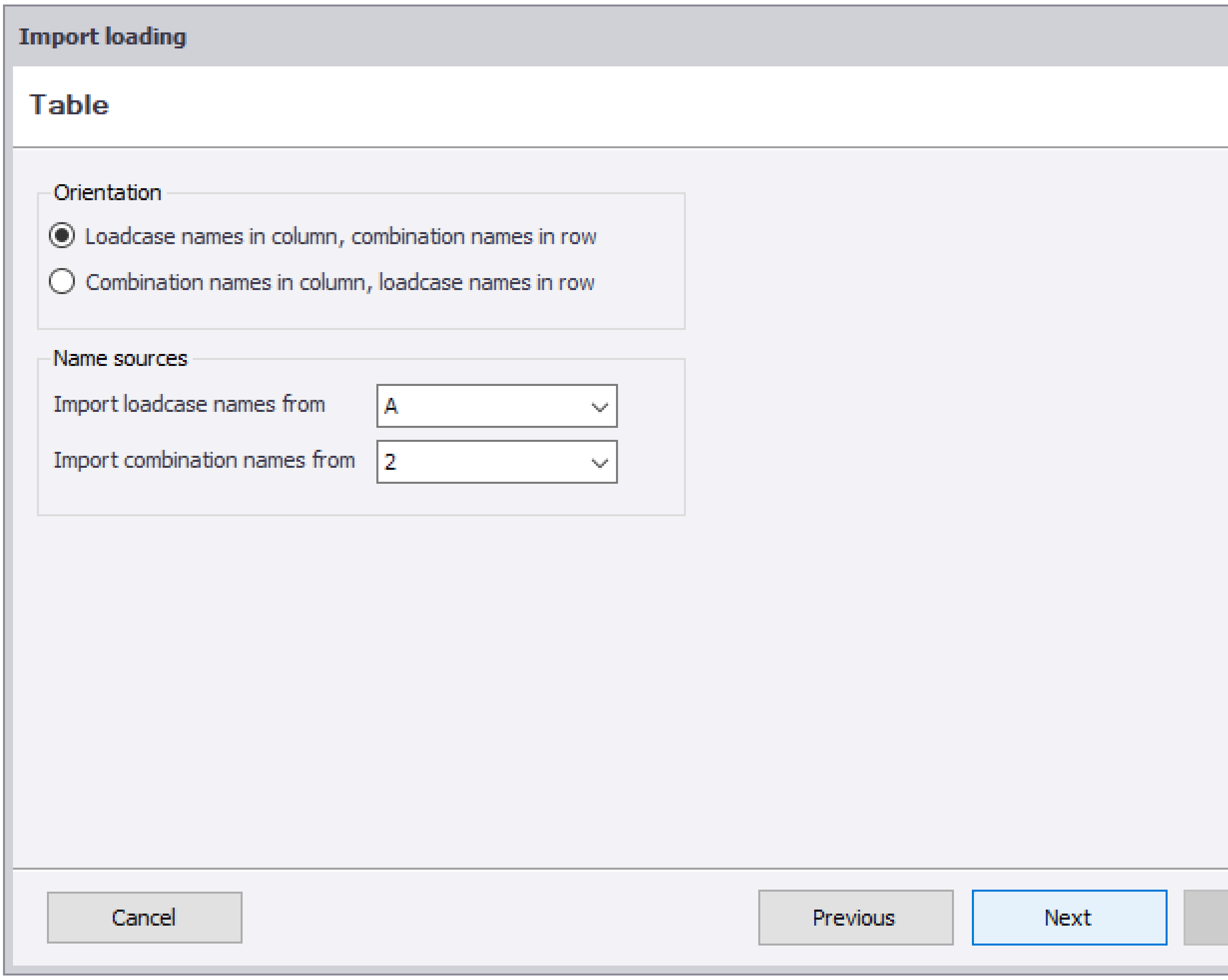
NOTE An empty cell is used to indicate a Loadcase is not to be included in particular Combination.

2. Starting from either the Loadcases or Combinations dialog in Tekla Structural Designer, click Import to start the Import loading wizard, the first page of which is:



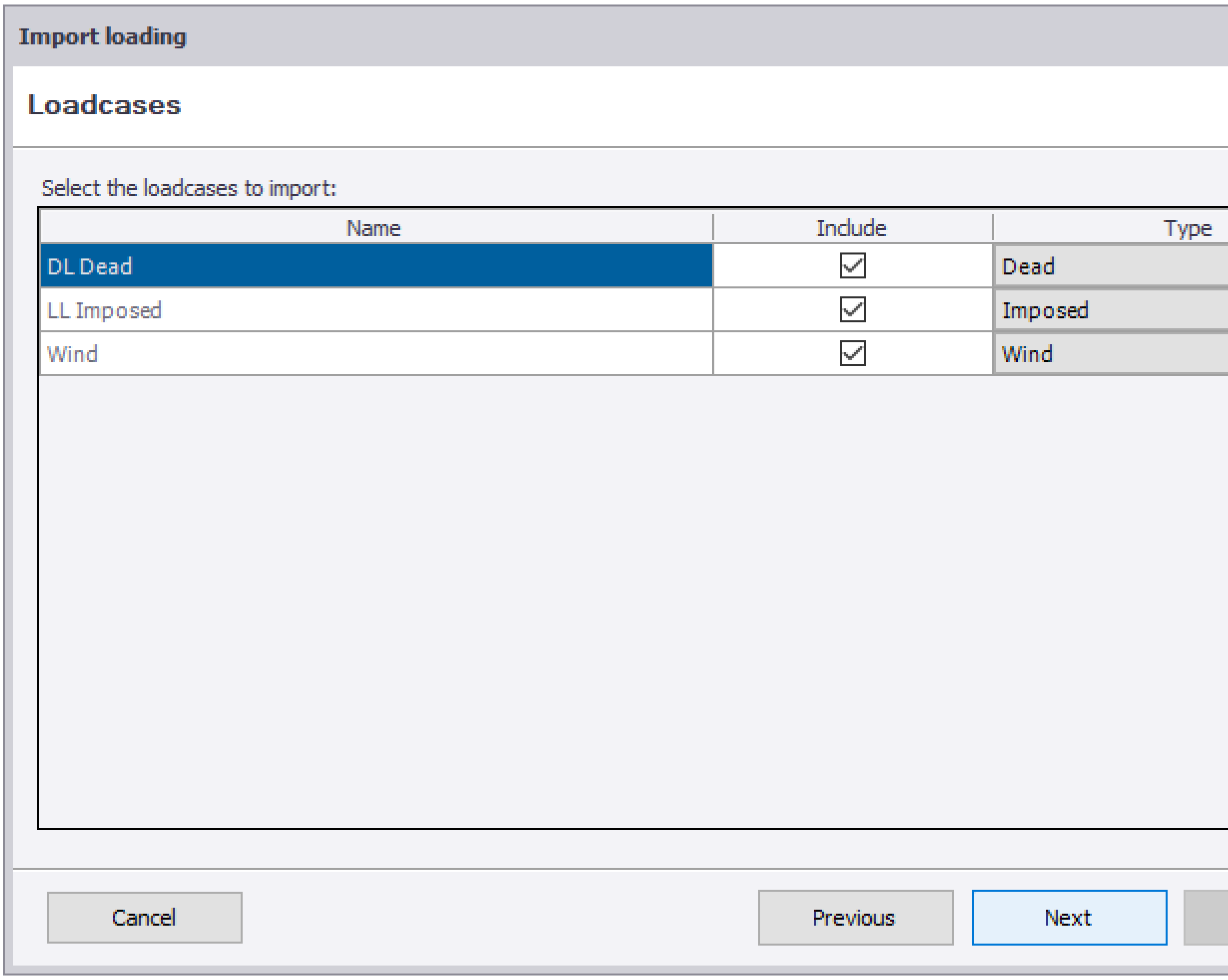
3. Click **Open ...** to navigate to the Excel file and the Sheet within the Excel file which contains the matrix and then click **Next**.

4. In the next page of the dialog you then choose the orientation of your matrix and confirm the Excel column & row which contain the Loadcase and Combination names (i.e. the matrix column and row headers). For this example the correct orientation and name sources are:



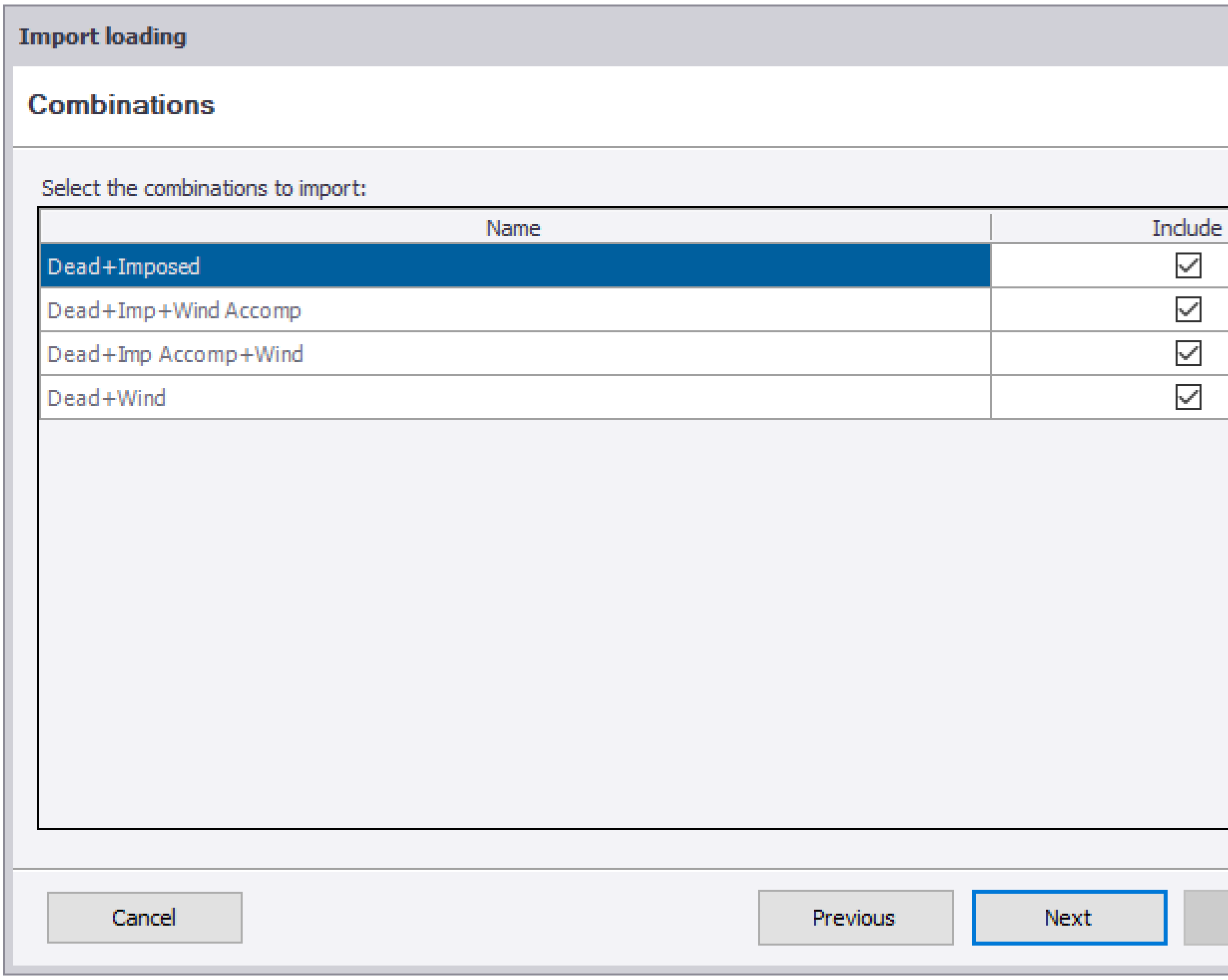
5. The next page of the dialog allows you to confirm/amend Loadcase options (a) **Include** (in the imported loadcases) - default is Include checked on, and (b) **Type** (the Loadcase type from Dead, Slab Dry, Slab Wet, Live, etc) - default is Dead for those names that can't be identified

automatically:



6. The next page of the dialog allows you to confirm/amend the Combination option **Include** (in the imported combinations) - default is

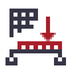
checked on:



7. The final page of the dialog summarizes the number of loadcases and combinations that will be imported, after which click **Finish** to complete the import process.
8. In the Loading dialog review the combinations, amend the combination classes where required, and if necessary include additional loadcases (e.g. automatically determined self weight).

Renumber all load combinations

When you delete load combinations from the **Loading** dialog box, the remaining load combinations retain their original loadcase number. If necessary, you can renumber the remaining load combinations in sequence. For more information, see the following instructions.

1. On the **Loading** tab of the **Project Workspace**, right-click  **Combinations**.
2. In the context menu, select **Renumber**.

Manage envelopes

You can use envelopes to view analysis results for multiple combinations simultaneously. When you do so, Tekla Structural Designer displays the maximum positive and negative values along each member from any combination included in the envelope.

NOTE When using envelopes, note the following:

- If you have defined patterned load combinations, you only need to include the base case pattern combination in the envelope. This way, Tekla Structural Designer automatically includes all pattern combinations derived from the base case in the envelope.
- You can include gravity, lateral, and seismic combinations in the same envelope.
- You should not include seismic RSA combinations in envelopes, as Tekla Structural Designer currently cannot display the results.

Create envelopes

1. On the **Load** tab, click **Envelope**.

   The Loading dialog (page 357) opens on the **Envelopes** page. This page displays all currently existing envelopes.
2. Click **Add**.
3. Name the envelope.
4. On the left side pane of the **Loading** dialog box, click the name of the envelope.
5. Select the load combinations that you want to add in the envelope, and click **>>**.
6. Click **OK** to save the envelope.

Manage load patterns

Having live/imposed loads applied only to a portion of the structure can produce a more unfavorable loading than the "fully loaded" condition. More unfavorable loading conditions can be considered in Tekla Structural Designer by the use of load patterns.

To manage load patterns, see:

- Overview of load patterns (page 354)
- Apply patterning to live loadcases (page 356)

- Apply patterning to load combinations (page 356)
- Update load patterns (page 356)

Overview of load patterns

NOTE When you apply pattern loading to imposed loads, the factors of the loaded and unloaded spans are specific to the design code that you are using.

The basic steps of applying pattern loads

1. Set individual imposed loadcases to be patterned (page 356) according to your needs.

   These loadcases are referred to as fully loaded pattern loadcases.

2. Set the gravity combinations that contain imposed loadcases to be patterned (page 356) according to your needs.

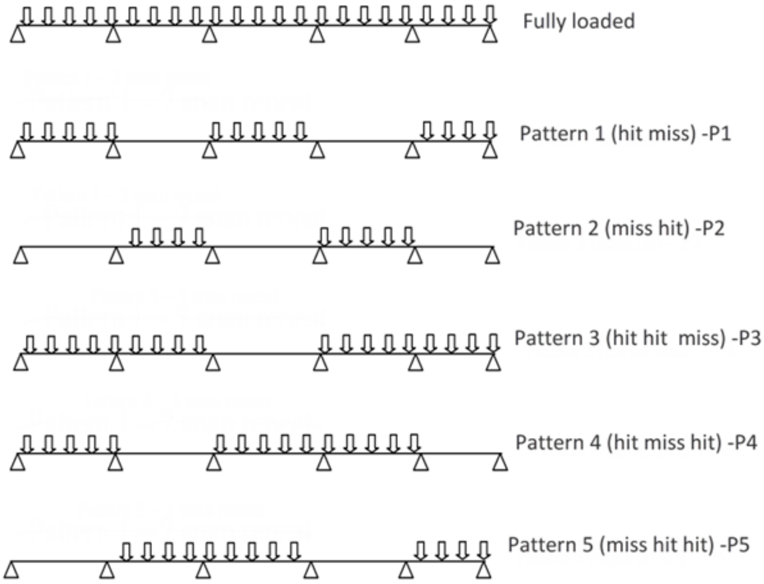
   These gravity combinations are referred to as base case pattern combinations.

After load decomposition, the building analysis automatically sets up the pattern cases for concrete beams in Dir1 and Dir2 directions as follows:

- 10 pattern cases for each pattern gravity combination
- 5 pattern cases for beams along Dir1 and 5 for beams along Dir2

NOTE By default, patterns are only applied to beam loads, and slab loads that have been decomposed on to beams. Loads applied to meshed slabs should be manually patterned using engineering judgement. For more information, see Update load patterns (page 356).

Pattern loadcases



A pattern combination containing patterned imposed loadcases results in 11 combinations: the base case combination, and 10 pattern combinations derived from the base combination.

If you later modify the building geometry, update load patterns (page 356) to ensure the load patterns reflect the changes.

NOTE Tekla Structural Designer contains a set of 10 pattern combinations associated with each fully loaded pattern combination. The pattern combinations are the same for beams and slabs.

The rules of applying load patterns to slabs and beams

Slab load patterning only applies where two-way slabs have been meshed in the solver model:

- in FE chasedown analysis
- in 3D analysis where two-way slabs are set as meshed

Therefore, the slab load pattern setting has no affect on:

- one-way spanning slabs
- two-way slabs not meshed in 3D analysis, as the slab loads are being decomposed to beams and walls prior to creation of the solver model
- two-way slab in Grillage chasedown analysis, as the slab loads are being decomposed to beams and walls prior to creation of the solver model

Therefore, in 3D analysis and Grillage chasedown analysis:

- When a beam is set to **Full Load**, it receives the full decomposed load from adjacent unmeshed two-way slabs, irrespective of whether the slabs themselves are set to **Full Load** or **Min Load**.
- When a beam is set to **Min Load**, it receives the min decomposed load from adjacent unmeshed two-way slabs, irrespective of whether the slabs themselves are set to **Full Load** or **Min Load**.

Apply patterning to live loadcases

If necessary, you can apply load patterning to live/imposed loadcases in a combination. In order to do so, see the following instructions.

1. On the **Load** tab, click **Loadcases**.

   The Loading dialog (page 357) opens on the **Loadcases** page. On this page, you can see all currently existing loadcases.
2. Click the live/imposed loadcase to which you want to apply patterning.
3. Select the **Pattern Load** option.
4. Click **OK**.

Apply patterning to load combinations

You can only apply load combinations to gravity combinations, as lateral and seismic combinations do not consider pattern loading. For detailed information on how to apply pattern loading to load combinations, see the following instructions.

1. On the **Load** tab, click **Combination**.

   The Loading dialog (page 357) opens on the **Combinations** page. On this page, you can see all currently existing load combinations.
2. On the left side pane of the **Loading** dialog box, select the name of the load combination.
3. In **Parameters**, ensure that the **Pattern** option is selected.
4. Click **OK**.

NOTE If you wish, you can use pattern loading for every gravity combination in your model. However, doing so may create many additional combinations.

Update load patterns

Tekla Structural Designer applies patterning to beam loads automatically. However, if you need to apply patterning to slab loads for slab design or other purposes, you must update the load patterns manually.

1. On the **Load** tab, click **Update Patterns**.
2. In the **Properties** window, select each pattern and adjust the loading status of the existing beams and slabs.
3. To switch the loading status of a slab, click the slab in the model.

Loading dialog

Summary

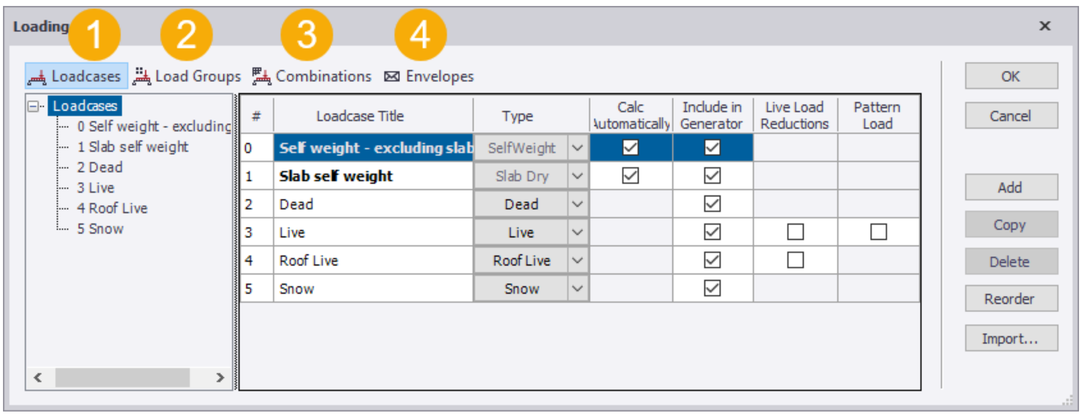
The **Loading** dialog is used to define loadcases, load groups, combinations and envelopes.

Location

On the **Load** tab, click:

- **Loadcases** - to open the dialog on the **Loadcases** page.
- **Load Groups** to open the dialog on the **Load Groups** page.
- **Combination**- to open the dialog on the **Combinations** page.
- **Envelope**- to open the dialog on the **Envelopes** page.

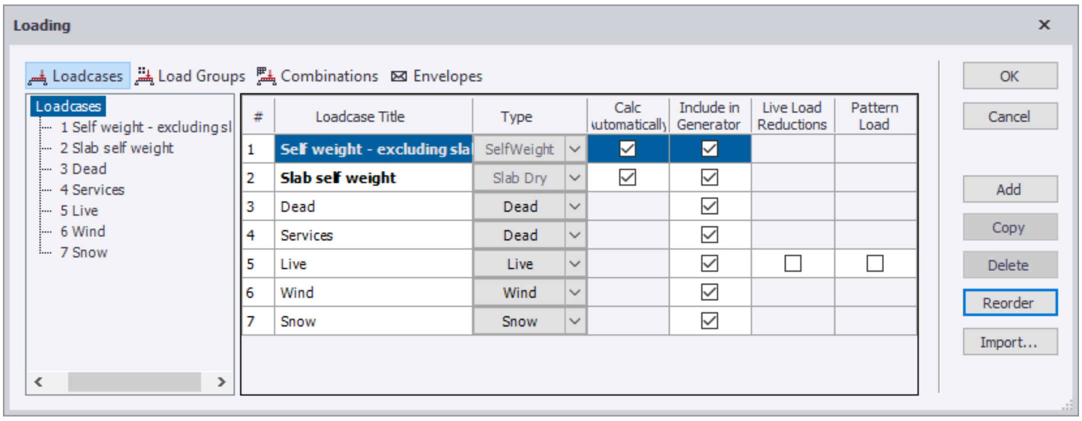
Content



1. Loadcases

Loadcases table

Select **Loadcases** in the left hand pane to show a table of loadcases that have been defined.



You can click within the table to:

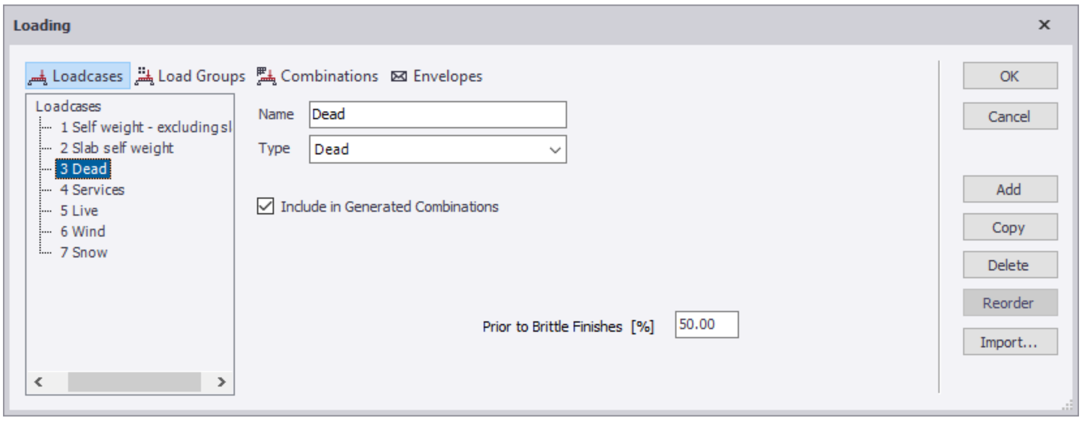
- Rename a Loadcase Title.
- Change the type of a loadcase.
- Choose whether the "Self weight excluding slabs" and Slab self weight" cases are to be calculated automatically, or defined manually.
- Choose which cases are to be included when generating the load combinations.
- Apply reductions to live/imposed cases.
- Apply pattern loading to live/imposed cases.

Using the buttons you can:

- **Add** a new loadcase to the table.
- **Copy** an existing loadcase.
- **Delete** an existing loadcase.
- **Reorder** to move loadcases up or down the table.
- **Import...** loadcases from a spreadsheet.

Loadcase parameters

Select an individual loadcase in the left hand pane to show parameters specifically applicable to that loadcase.



Certain values can only be specified by selecting individual loadcases, for example:

- For dead loadcases: **Prior to Brittle Finishes %** .
- For live/imposed loadcases: **Long Term %** .

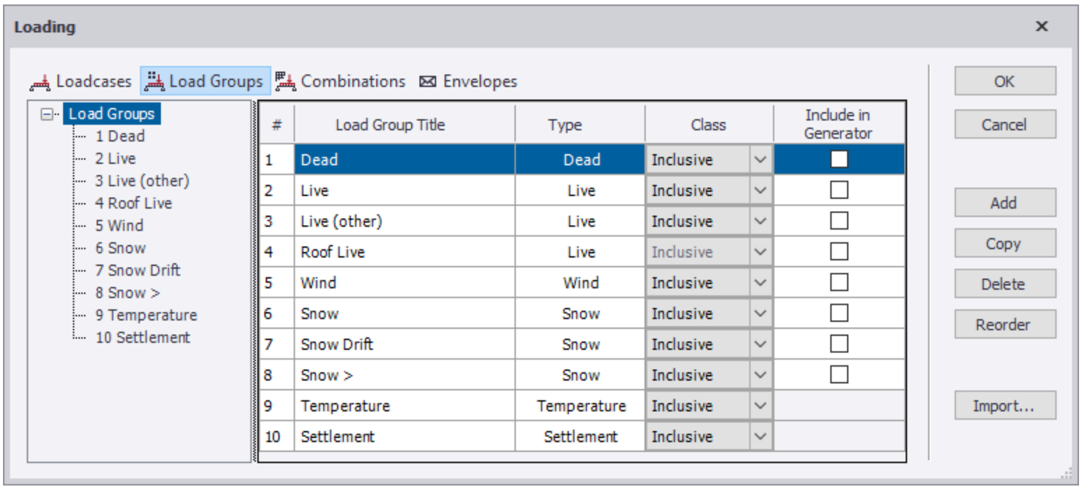
See also

Manage loadcases (page 340)

2. Load Groups

Load Groups table

Select **Load Groups** in the left hand pane to show a table of any load groups that have been defined.



Click within the table to:

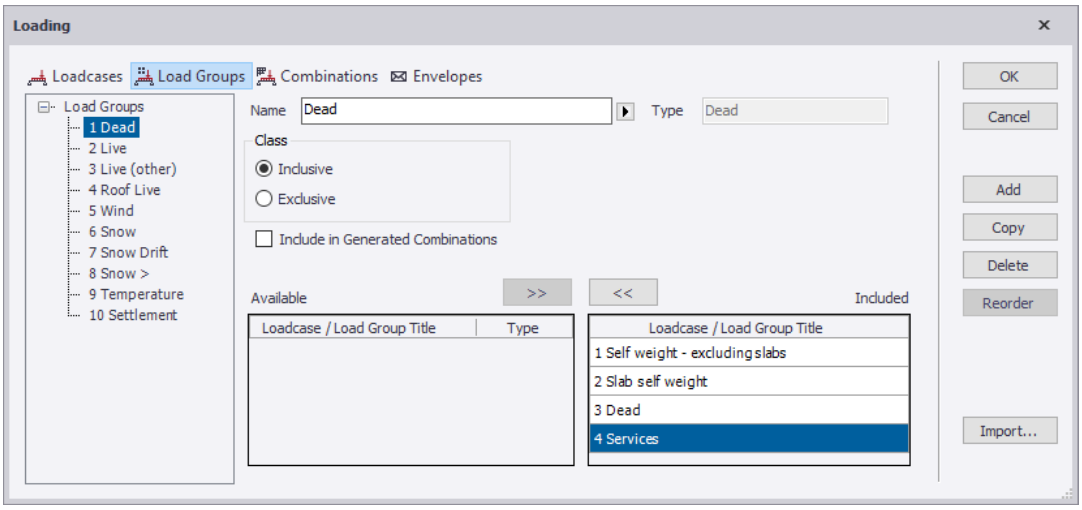
- Rename a load group title.
- Set the load group class as Inclusive, or Exclusive.
- Choose whether the load group is to be included in the generator.

Using the buttons you can:

- **Add** a new load group in the table.
- **Copy** an existing load group.
- **Delete** an existing load group.
- **Reorder** to move load groups up or down the table.
- **Import...** load groups from a spreadsheet.

Load group parameters

Select an individual load group in the left hand pane to set up the load group content.



The load group class is set as Inclusive, or Exclusive.

The loadcases/load groups currently in the load group are shown in the Included list.

Use the [>>] and [<<] buttons to add or remove loadcases/load groups from the Available list to the Included list.

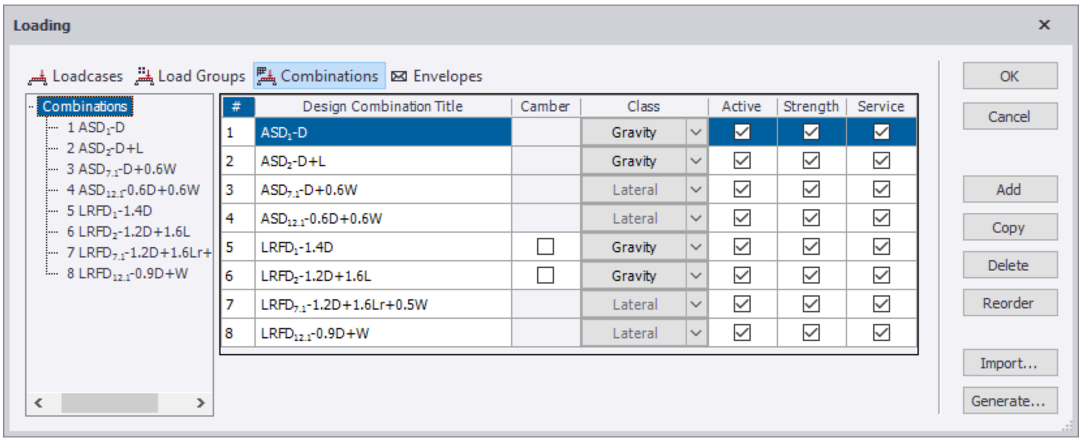
See also

Manage load groups (page 342)

3. Combinations

Combinations table

Select **Combinations** in the left hand pane to show a table of combinations that have already been defined or generated.



You can click within the table to:

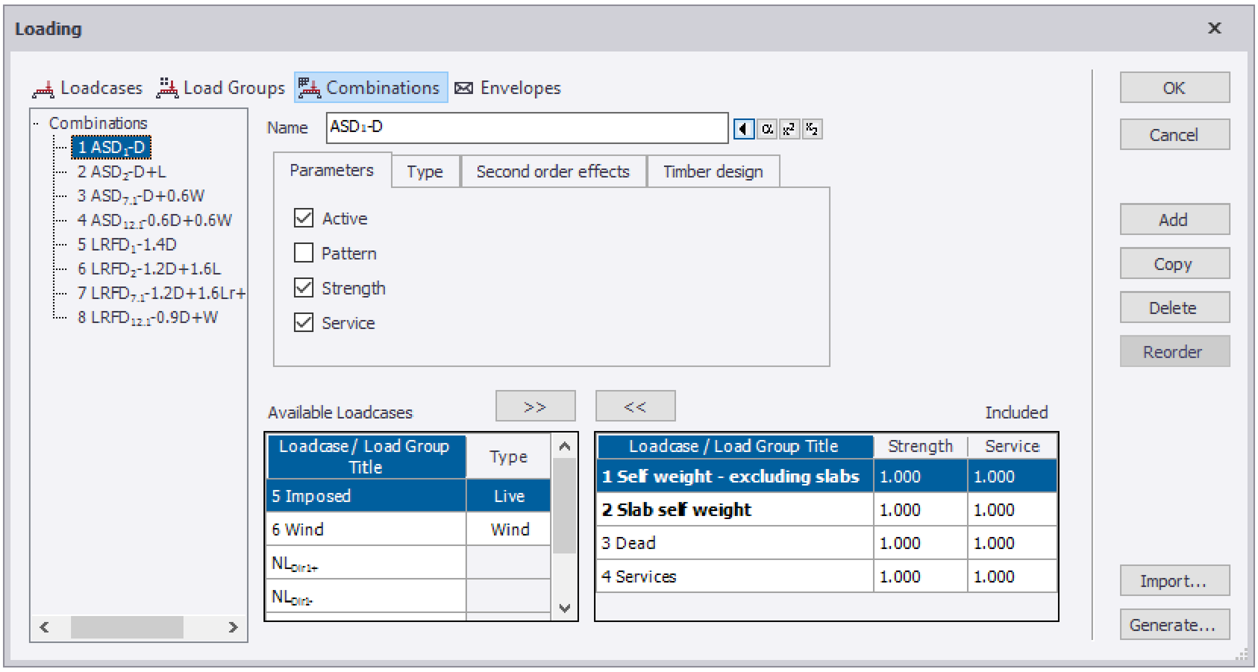
- Rename a combination title.
- Change the class of a combination.
- Choose the gravity combination to be used for any steel beam camber calculations.
- Choose which combinations are active for analysis and design.
- Choose which active combinations are to be assessed for strength.
- Choose which active combinations are to be assessed for service.

Using the buttons you can:

- **Add** a new combination in the table.
- **Copy** an existing combination.
- **Delete** an existing combination.
- **Reorder** to move combinations up or down the table.
- **Import...** combinations from a spreadsheet.

Combination parameters

Select an individual combination in the left hand pane to show the parameters specifically applicable to that combination only.



The tabbed boxes are used to set:

- **Parameters**: (Active, Pattern, Strength and Service).
- **Type**: Head code dependent type settings:
  - US head code - ACI/LRFD or ASD.
  - Eurocode - Formula type (STR, EQU, GEO)
  - India - Limit State or Working Stress
  - BS & Australia - not applicable
- **Second order effects**: the amplification factor, its direction, and whether it is applied to all loads or lateral loads only.
- **Timber design**: (US and Eurocodes only) the load duration/time effect factor.

The loadcases currently in the combination and the strength and serviceability factors that apply are shown in the Included list.

- Use the [>>] and [<<] buttons to add or remove loadcases from the Available Loadcases list to the Included list.
- In the **Strength** column specify the strength factors.
- In the **Service** column specify the service factors.

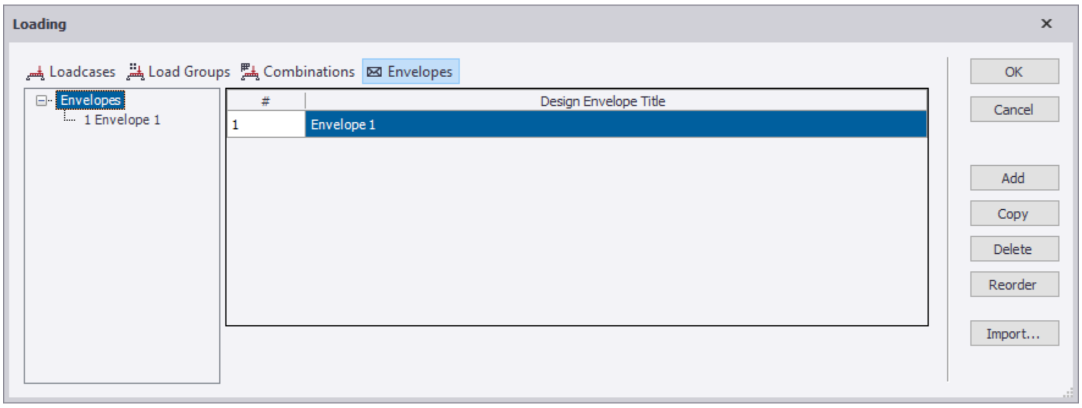
See also

Manage load combinations (page 344)

6. Envelopes

Envelopes table

Select **Envelopes** in the left hand pane to show a table of any envelopes that have been defined.

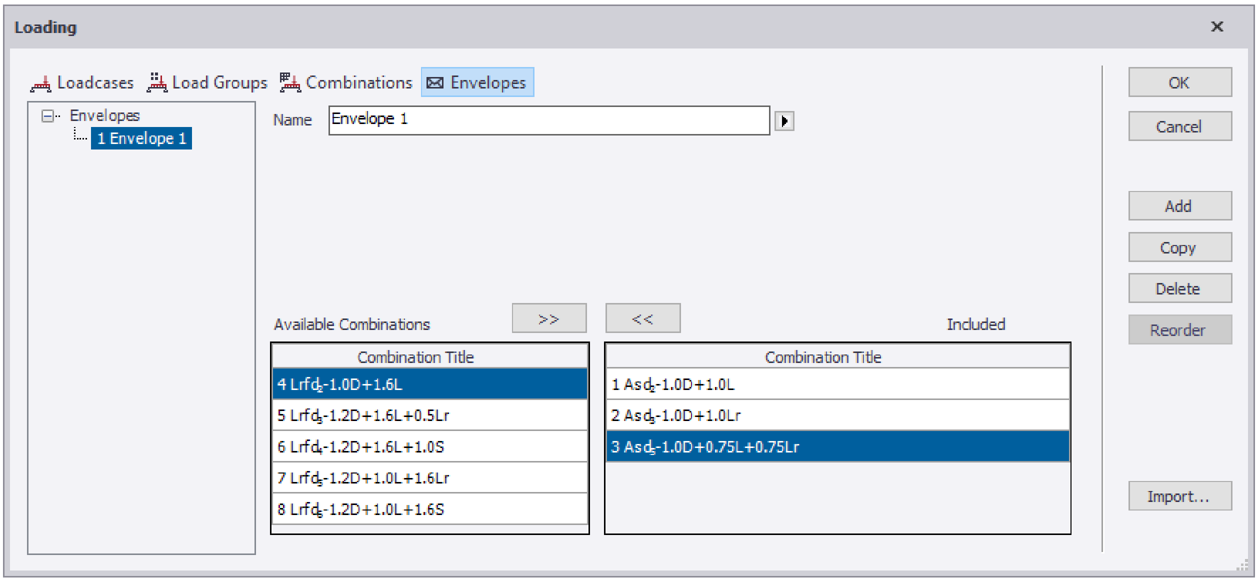


Using the buttons you can:

- **Add** a new envelope in the table.
- **Copy** an existing highlighted envelope.
- **Delete** an existing highlighted envelope.
- **Reorder** to move envelopes up or down the table.
- **Import...** to import envelopes from a spreadsheet.

Envelope parameters

Select an individual envelope in the left hand pane to set up the envelope content.



The combinations currently in the envelope are shown in the Included list.

Use the [>>] and [<<] buttons to add or remove combinations from the Available Combinations list to the Included list.

5.2 Apply panel, member, and structure loads

- Apply panel loads (page 364)
- Apply member loads (page 370)
- Apply structure loads (page 372)
- Modify panel, member, and structure loads (page 383)
- Delete panel, member, and structure loads (page 383)
- Decompose panel loads (page 383)

Apply panel loads

You can use panel loads to apply loads to slab items, roof panels, and wall panels. Tekla Structural Designer contains several types of panel loads, including point loads, line loads, patch loads, polygonal loads, area loads, slab loads, and level loads. In order to apply the different panel loads to your model, see the following instructions.

You can use the different panel loads in the following ways:

Panel load type	Use	Notes
Point load	Can be applied anywhere within an individual or across multiple slab items, roof panels, or wall panels.	Can only be applied in 2D views.
Patch load		
Polygonal load		
Area load	Entirely covers a slab item, a roof panel, or a wall panel.	Not applicable
Slab load	Entirely covers all slap items in a parent slab.	A parent slab can consist of slab items that are physically separate from each other, but are on the same level.
Level load	Entirely covers all parent slabs in a level.	

Create point loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate loadcase.
3. On the **Load** tab, click **Point**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

6. Click the load position, or press **F2** to type the exact position.

   NOTE If you move the slab item or panel by manually selecting & re-positioning its nodes, the point load does not move with the slab or panel.

   However, if you move any of the grid lines defining the reference node, the load will move as well.

Create line loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate loadcase.

3. On the **Load** tab, click **Line**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

   ---

6. Click the start point of the load, or press **F2** to type the exact position.

   Note that the start point is an offset (X, Y) from the selected reference node.
7. Click the end point of the load, or press **F2** to type the exact position.

   Note that when you type the position using the keyboard, the end point is an offset (X, Y) from the load start position.

Create patch loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate loadcase.
3. On the **Load** tab, click **Patch**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

   ---

6. Click to specify a corner position of the load, or press **F2** to type the exact corner position.
7. Click to specify the size of the load, or press **F2** to type the size.

   The size is specified as the offset dimensions from the selected corner position.
8. Click to specify the rotation angle of the load, or press **F2** to type the rotation angle.

Create polygonal loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate loadcase.
3. On the **Load** tab, click **Polygon**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

6. Click the start point of the load, or press **F2** to type the exact position.

   Note that the start point is an offset (X, Y) from the selected reference node.

7. Click a corner point of the load, or press **F2** to type the exact position.
8. Repeat step 7 according to your needs.
9. When the polygon is complete, do one of the following:
   - Press **Esc**.
   - Click the first corner that you defined.

   TIP To modify the shape of an existing polygonal load the following:

   a. In the model, select the load.
   b. Click one vertex of the load.
   c. Click the point where you want to move the vertex.

Create perimeter loads

1. Open a 2D or 3D view showing the level at which you want to apply the load.
2. In the **Loading** list, select an appropriate loadcase.
3. On the **Load** tab, click **Perimeter**.
4. In the **Properties** window, adjust the load details according to your needs.

   TIP If necessary, you can select one or both of the following options:

- **Create as line loads**: creates the load as a series of separate line loads along each external edge.
- **Ignore openings**: only creates the load around external perimeters, and not around internal opening perimeters.

5. Click any slab or mat item.

   Tekla Structural Designer creates the load around the external perimeter of every continuous area of slabs at the selected level.

Create variable patch loads

NOTE Variable patch loads do not have to be quadrilaterals. Instead, they can be any polygonal shape.

1. Open a 2D or 3D view showing the level at which you want to apply the load.
2. In the **Loading** list, select an appropriate loadcase.
3. On the **Load** tab, click **Var. Patch**.
4. In the **Properties** window, specify three load values to describe the variable loads.
5. Adjust the remaining load details according to your needs.
6. In the model, click a reference node from which the load position can be offset.

   The load position can be the start or end point of any member at the level.

   TIP To be able to select a reference node, go to **Scene Content**, and select **Points**.

7. Click the start point of the load, or press **F2** to type the exact position.

   Note that the start point is an offset (X, Y) from the selected reference node.

8. Click a corner point of the load, or press **F2** to type the exact corner position.
9. Repeat step 8 according to your needs.
10. When the polygon is complete, do one of the following:
    - Press **Esc**.
    - Click the first corner that you defined.
11. Click one of the corners of the polygon to specify the position of **Load 1**.
12. Click another corner to specify the position of **Load 2**.

13. Click a third corner to specify the position of **Load 3**.

TIP To modify the shape of an existing patch load the following:

a. In the model, select the load.

b. Click one vertex of the load.

c. Click the point where you want to move the vertex.

Create area loads

1. Open a 2D view of the level within which you want to apply the load.
2. In the **Loading** list, select an appropriate loadcase.
3. On the **Load** tab, click **Area**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Click the panel to which you want to apply the load.

Create variable area loads

Variable area loads can only be applied to non-horizontal slab items and roof panels, as well as wall panels and concrete walls. Therefore, they cannot be applied in a 2D view.

1. Open the 3D view, frame, or sloped plane view in which you want to apply the load.
2. In the **Loading** list, select an appropriate loadcase.
3. On the **Load** tab, click **Var. Area**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Click the slab item or panel to which you want to apply the load.

Create slab loads

1. In the **Loading** list, select an appropriate loadcase.
2. On the **Load** tab, click **Slab**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load to all the slab items within a parent slab, in the model, click any slab item within the parent slab.

Create level loads

Click any slab panel in order to apply the load to all slabs within the level.

1. In the **Loading** list, select an appropriate loadcase.

2. On the **Load** tab, click **Level**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load to all slabs within the level, in the model, click any slab item.

Apply member loads

By using member loads, you can apply loads to one-dimensional members, such as beams, columns, and braces.

Tekla Structural Designer allows you to create and apply the following types of member loads:

- Full-length uniformly distributed loads (UDLs)
- Partial-length uniformly distributed loads (UDLs)
- Partial-length variable deck loads (VDLs)
- Trapezoidal loads
- Point loads
- Moment loads
- Full-length torsional uniformly distributed loads (UDLs)
- Partial-length torsional uniformly distributed loads (UDLs)
- Partial-length torsional variable deck loads (VDLs)

Create full-length UDLs

In order to create a full-length uniformly distributed load (UDL) and apply it to a member, see the following instructions.

1. In the **Loading** list, select an appropriate loadcase.
2. On the **Load** tab, click **Full UDL**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load, click anywhere along the member.

Create partial-length UDLs or VDLs

In order to create UDLs (uniformly distributed loads) or VDLs (variable deck loads) that only apply to a selected part of the member, see the following instructions.

1. In the **Loading** list, select an appropriate loadcase.
2. On the **Load** tab, click **UDL** or **VDL**.

3. In the **Properties** window, adjust the load details according to your needs.
4. Hover the mouse pointer over the member to which you want to apply the load.
5. Click the start point of the load along the member, or press **F2** to type the exact position.
6. Click the end point of the load along the member, or press **F2** to type the exact position.

Create trapezoidal loads

In order to apply trapezoidal loads to the members in your model, see the following instructions.

1. In the **Loading** list, select an appropriate loadcase.
2. On the **Load** tab, click **Trapezoidal**.
3. In the **Properties** window, adjust the load details according to your needs.
4. Hover the mouse pointer over the member that you want to apply the load to.
5. Click the load position along the member, or press **F2** to type the exact position.

   The load position defines the point at which the symmetrical trapezoidal load reaches its maximum intensity.

Create point loads and moment loads

In order to create and apply point loads or moment loads to the members in your model, see the following instructions.

1. In the **Loading** list, select an appropriate loadcase.
2. On the **Load** tab, in the **Member Loads** group click **Point** or **Moment**.
3. In the **Properties** window, adjust the load details according to your needs.
4. Hover the mouse pointer over the member to which you want to apply the load.
5. Click the load position along the member, or press **F2** to type the exact position.

Create full-length torsional UDLs

In order to apply full-length torsional uniformly distributed loads (UDLs) to the members in your model, see the following instructions.

1. In the **Loading** list, select an appropriate loadcase.
2. On the **Load** tab, click **Torsion Full UDL**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load, click anywhere along the member.

Create partial-length torsional UDLs and VDLS

In order to create and apply torsional uniformly distributed loads (UDLs) or variable deck loads (VDLs) that only apply to a selected part of the member, see the following instructions.

1. In the **Loading** list, select an appropriate loadcase.
2. On the **Load** tab, click **Torsion UDL** or **Torsion VDL**.
3. In the **Properties** window, adjust the load details according to your needs.
4. Hover the mouse pointer over the member to which you want to apply the load.
5. Click the start point of the load along the member, or press **F2** to type the exact position.
6. Click the end point of the load along the member, or press **F2** to type the exact position.

Apply structure loads

The five types of structure loads that you can apply are:

- Diaphragm loads: applied at any point within a rigid or semi-rigid diaphragm.
- Equipment loads (page 305): applied to equipment, (in loadcases other than the dedicated 'equipment' ones).
- Nodal loads (page 382): applied at solver node locations.
- Temperature loads (page 382): global rises in temperature applied to elements or panels.
- Settlement loads (page 382): translations or rotations applied to supports in the support UCS system.

Diaphragm loads and diaphragm load tables

Diaphragm loads are Building Direction 1, Building Direction 2, and Mz torsion loads applied at any point within a rigid or semi-rigid diaphragm.

Typically these might be used for the application of externally determined level loads (for example from Wind Tunnel testing of large/ unusual structures).

The **Diaphragm** load command is available when a 2D plan view is active - it can be used to apply a load to any diaphragm within the current view.

If you have multiple loads to apply at different levels, the **Diaphragm Table** of loads provides a quicker means of doing this - either by typing the loads and their positions directly into the table, or by pasting the data into the table from a spreadsheet.

Add a diaphragm load in a 2D view

To apply a load to any diaphragm within the current 2D view:

Related video

Diaphragm loading

1. Open a 2D view of the level containing the diaphragm to which you want to apply the load.
2. Click the **Load** tab on the ribbon.
3. In the **Loading** list, select an appropriate loadcase.

   Diaphragm on the **Load** tab should now be active.

4. Click **Diaphragm**
5. In the **Properties** window, adjust the load details according to your needs.
6. Click the load position, or press **F2** to type the exact position.

Add a diaphragm load in a 3D view

To apply a load to any diaphragm within the current 3D view:

Related video

Diaphragm loading

1. Open a 3D view containing the diaphragm to which you want to apply the load.
2. Click the **Load** tab on the ribbon.
3. In the **Loading** list, select an appropriate loadcase.

   Diaphragm on the **Load** tab should now be active.

4. Click **Diaphragm**
5. In the **Properties** window, adjust the load details according to your needs.
6. At the level of the diaphragm, click an existing construction point to apply the load at that position, or hover over an existing member to apply the load at a distance along the member (press **F2** to type the exact position).

Add a diaphragm load in a 3D view by selecting the diaphragm level

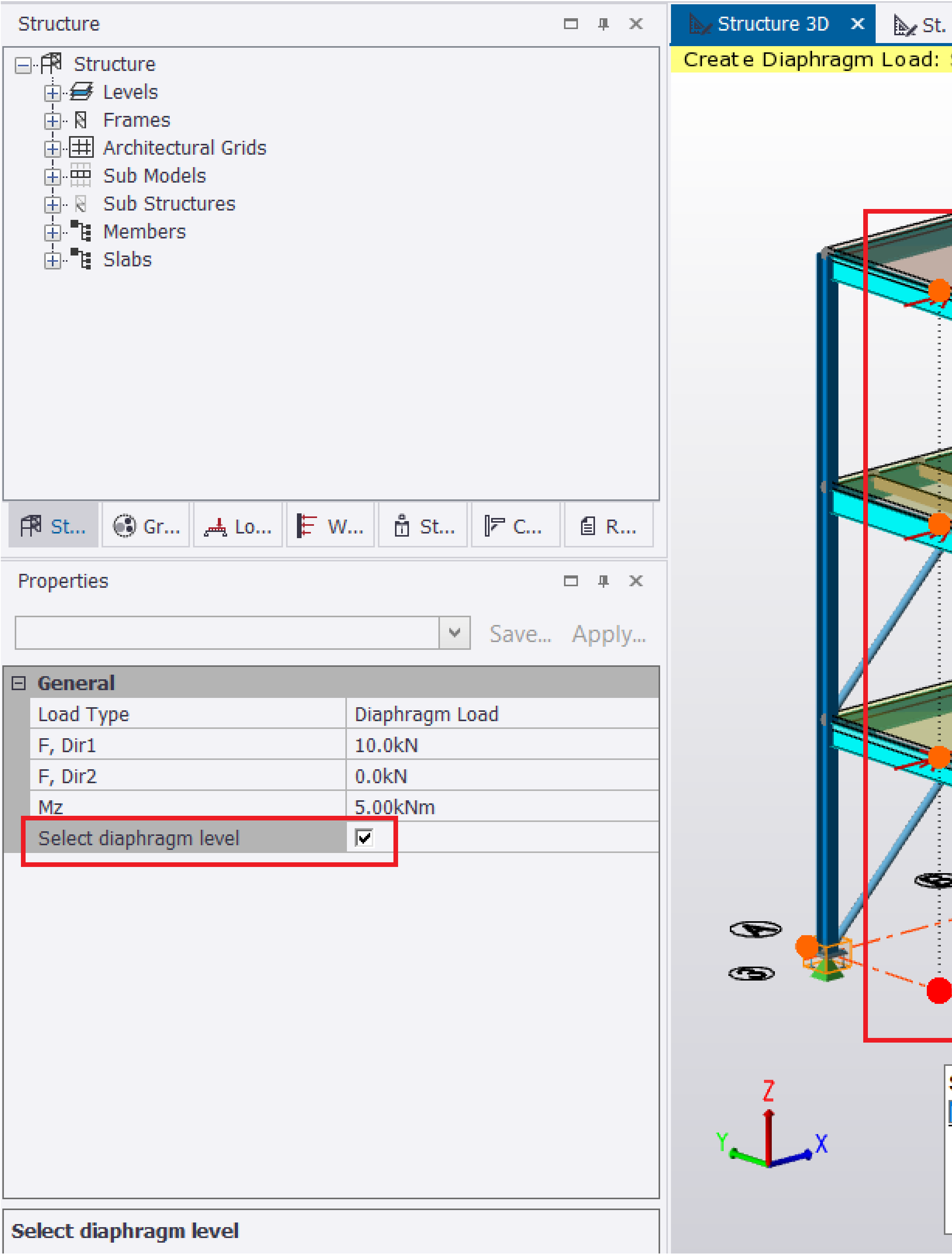
In this method, the load can be applied to a diaphragm at any level by selecting an existing point at any level:

1. Open a 3D view containing the diaphragm to which you want to apply the load.
2. Click the **Load** tab on the ribbon.
3. In the **Loading** list, select an appropriate loadcase.

   Diaphragm on the **Load** tab should now be active.

4. Click **Diaphragm**
5. In the **Properties** window, adjust the load details according to your needs.
6. In the **Properties** window, check **Select diaphragm level**
7. At any level, click an existing construction point, or hover over an existing member to specify a distance along the member (press **F2** to type the exact position). A vertical line is displayed with nodes along it at each

diaphragm position above or below the identified position.

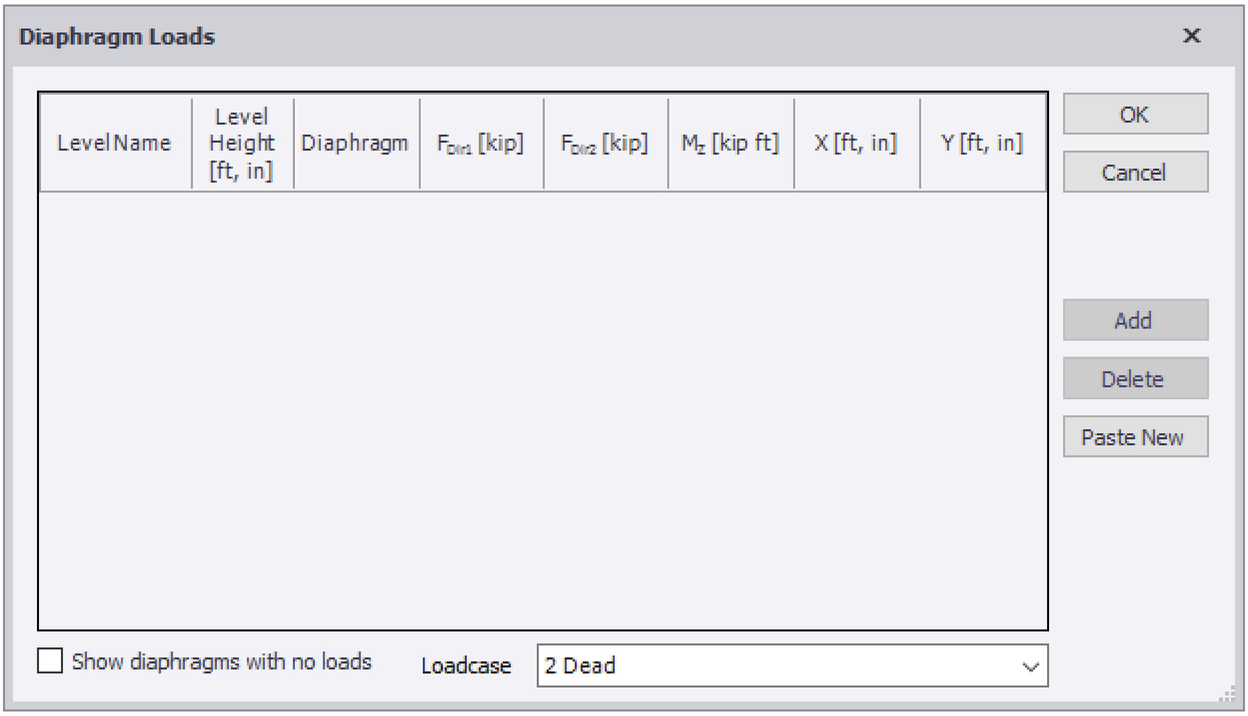


8. Click on a node to apply the load to the diaphragm at that level.

Add multiple loads using the diaphragm load table

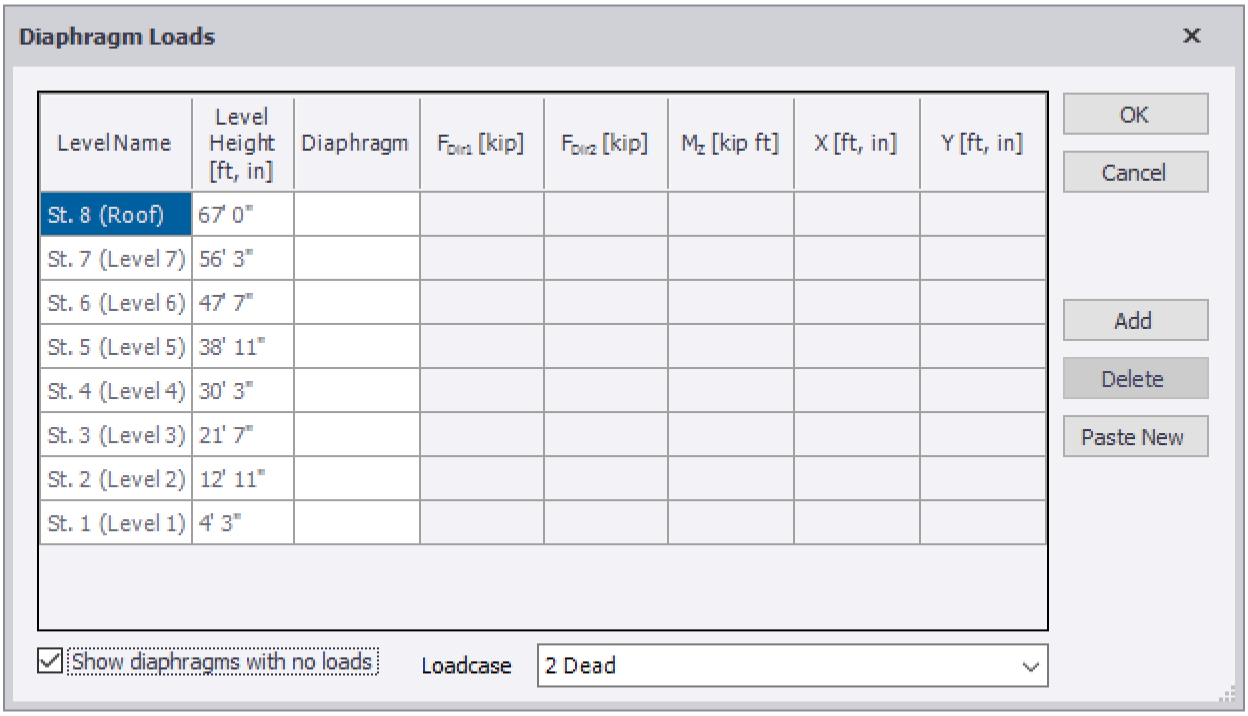
The diaphragm loads table provides a way to quickly apply multiple diaphragm loads at different levels, either by typing the loads and their positions directly into the table, or by pasting the data into the table from a spreadsheet.

1. On the **Load** tab, click **Diaphragm Table**.



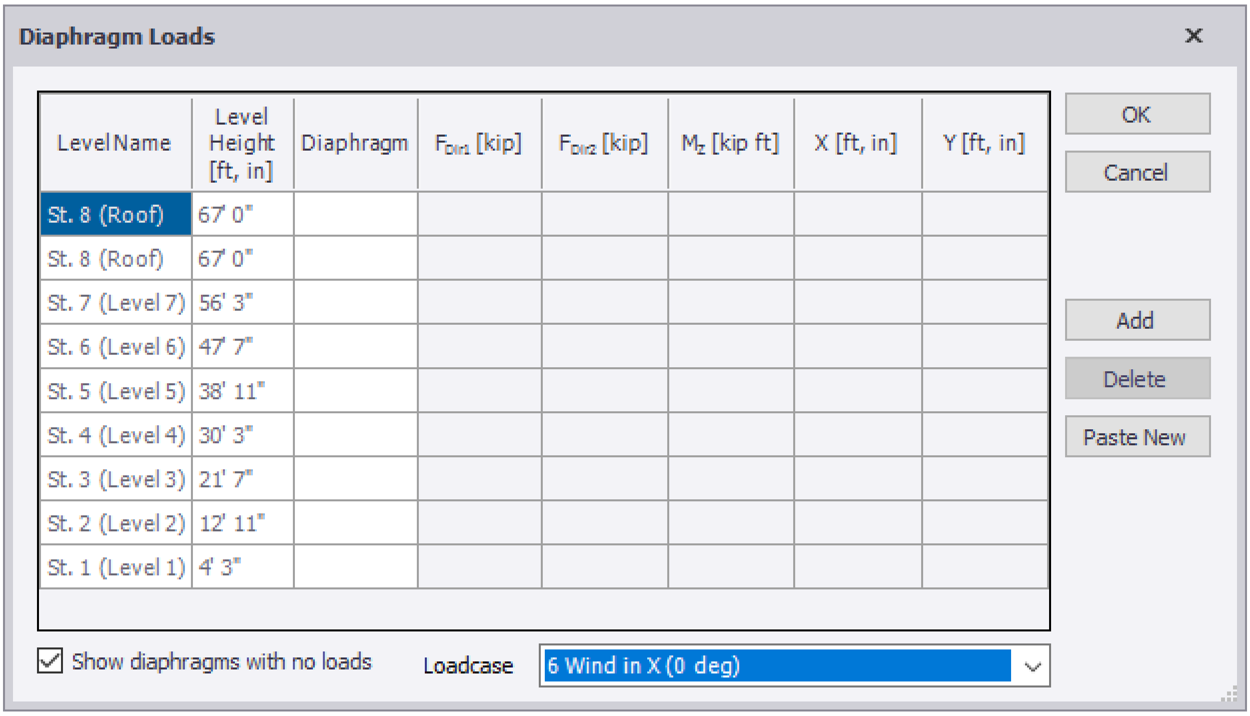
The Diaphragm Loads table opens in a dialog. If any loads had already been added (via the Diaphragm Load command, or the Diaphragm Loads table) these would be listed.

2. In order to see all the separate diaphragms at each level to which loads could be applied, click the **Show diaphragms with no loads** box.

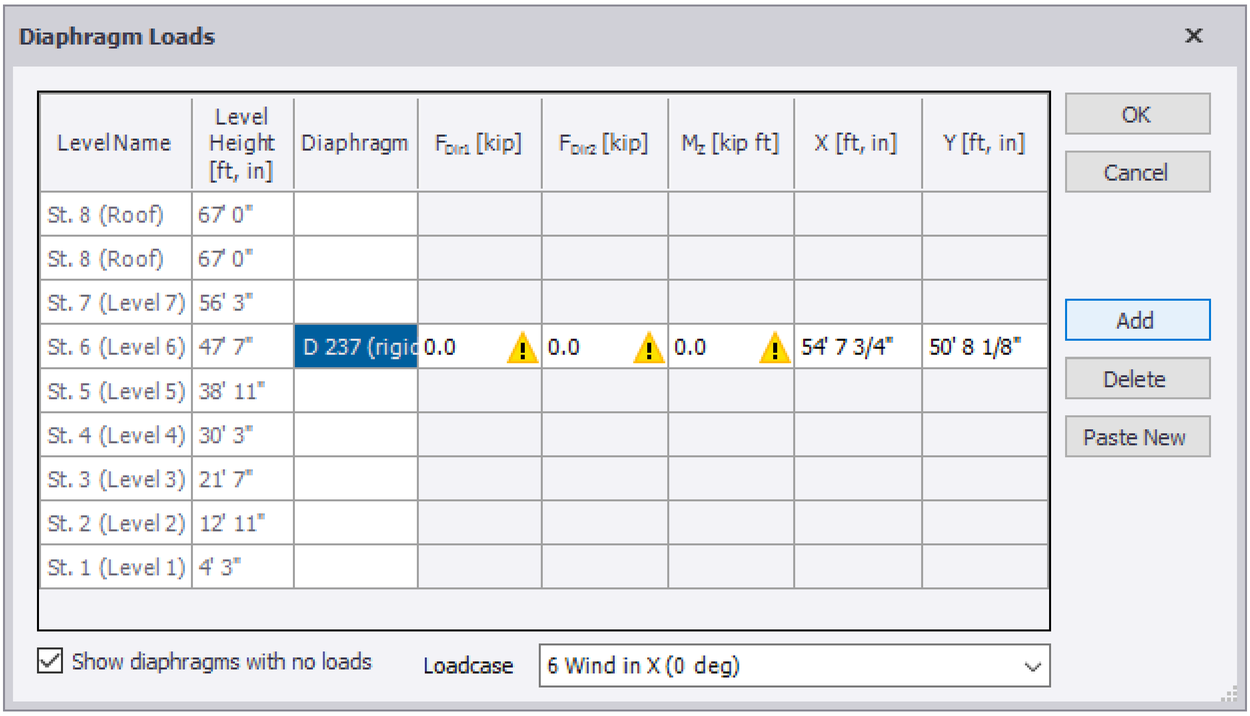


Each level containing a diaphragm is listed, (if more than one diaphragm exists at the same level a separate row is created for each).

3. In the **Loading** list at the bottom of the dialog, select the loadcase within which you want to apply the load.



4. In the table, click on the diaphragm level to which you want to apply the load, then click **Add**.



A row is displayed for entering the load. The initial default for the load position is simply the middle point in X and Y of the diaphragm extremities.

5. Enter the load values and the actual X and Y coordinates.
6. Add further rows of loading data as required, either at the same level or other levels.

NOTE Multiple loads can be applied to a single diaphragm if required.

7. Click **OK** when done.

The loads are displayed on the model iat their defined locations.

NOTE To edit existing loads in the table, simply open it once more. To delete loads from the table, select the diaphragm in the table and click **Delete**.

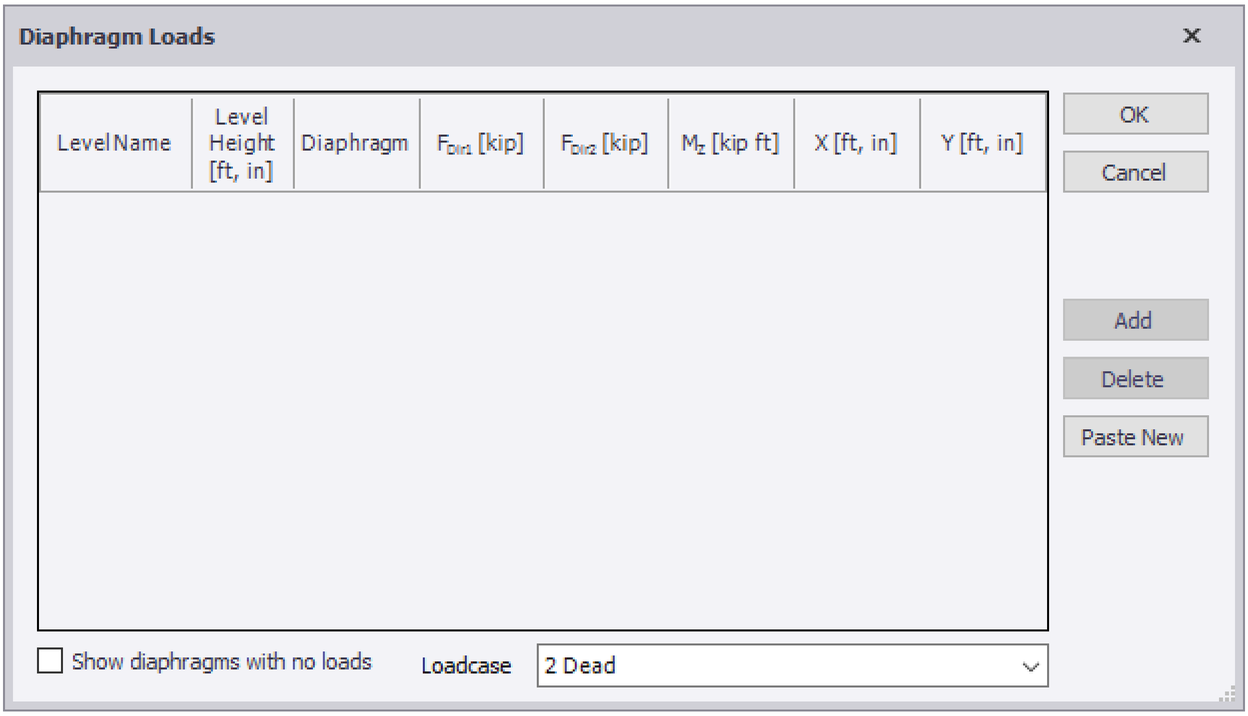
Paste multiple loads into the diaphragm load table

If you have been provided with the loading data in the form of a spreadsheet you can simply paste the data directly into the table.

NOTE The data must includes a Z (Level height) column and you must ensure that it is in the following order (which is not the same order as in the diaphragm loads table):

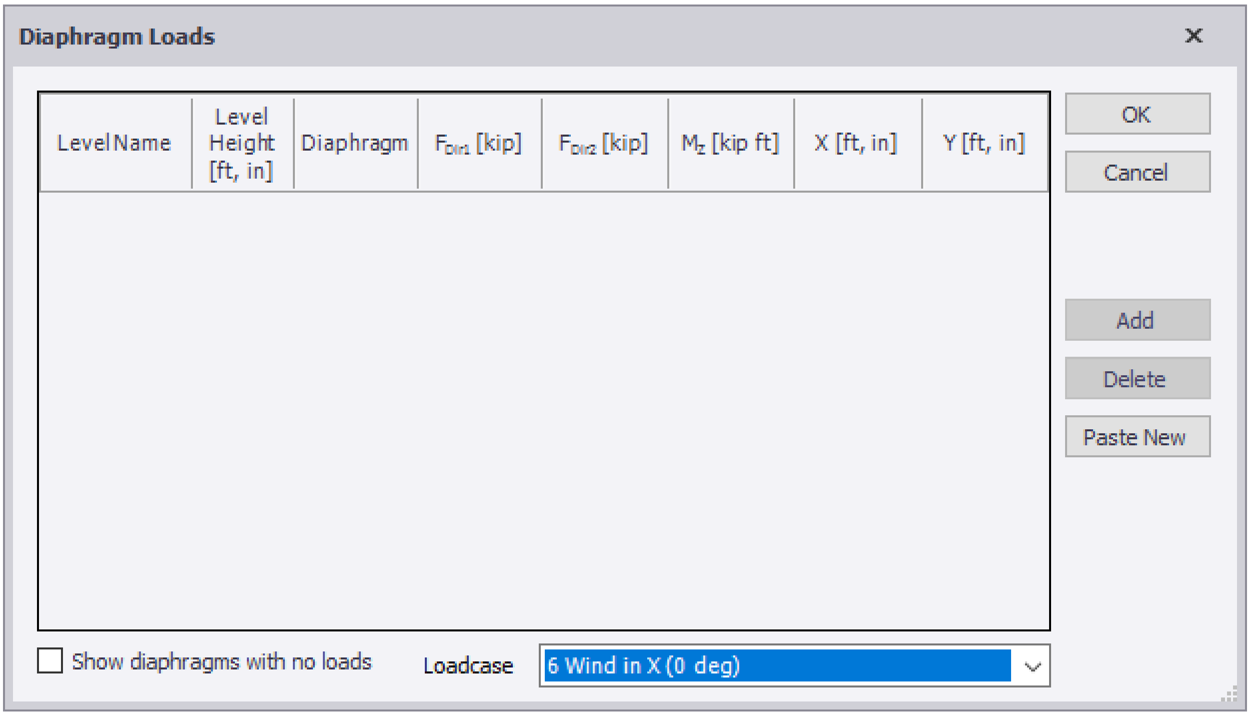
1. Force in direction 1
2. Force in direction 2
3. Torsion moment
4. Position X
5. Position Y
6. Position Z

1. Open the spreadsheet containing the data to be pasted and copy the contents to the clipboard.
2. In Tekla Structural Designer, on the **Load** tab, click **Diaphragm Table**.

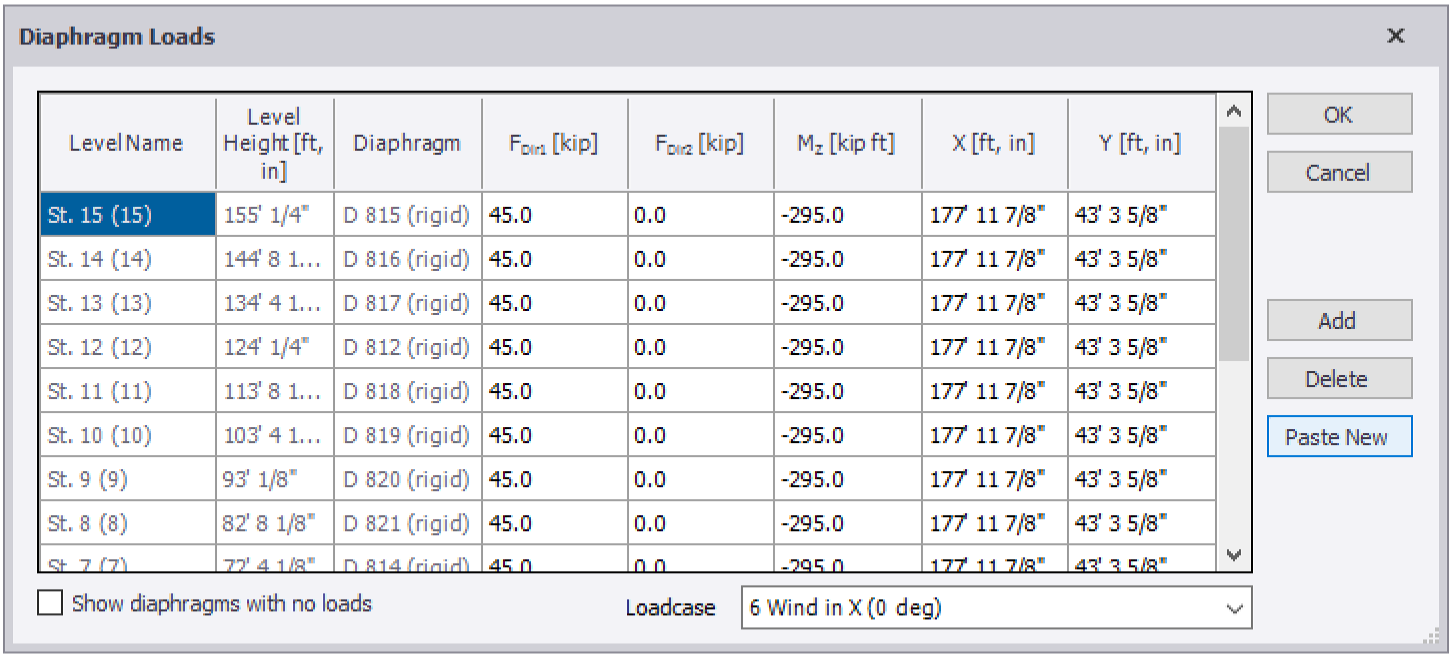


The Diaphragm Loads table opens in a dialog.

3. In the **Loading** list at the bottom of the dialog, select the loadcase within which you want to apply the load.



4. Click **Paste New**.



Provided the spreadsheet contents were in the expected format, the loads and their positions are pasted into the table.

NOTE Existing diaphragm loads will be deleted / replaced by the pasted data.

5. Click **OK** when done.

Create nodal loads

To create nodal loads at the solver nodes in your model, see the following instructions.

1. Click the **Load** tab on the ribbon.
2. In the **Loading** list, select an appropriate loadcase.

   Nodal on the **Load** tab should now be active.

3. Click **Nodal**.
4. In the **Properties** window, adjust the load details according to your needs.
5. To define the load position, click the desired node.

Create temperature loads

Temperature loads are global rises in temperature that you can apply to individual elements or panels in the model. For more information on how to create temperature loads, see the following instructions.

1. Click the **Load** tab on the ribbon.
2. In the **Loading** list, select an appropriate loadcase.

   Temperature on the **Load** tab should now be active.

3. Click **Temperature**.
4. In the **Properties** window, adjust the load details according to your needs.
5. Click anywhere along the element to apply the load.

Create settlement loads

Settlement loads are translations or rotations that you can apply to a support. To create settlement loads, see the following instructions.

1. Click the **Load** tab on the ribbon.

2. In the **Loading** list, select an appropriate loadcase.

   Settlement on the **Load** tab should now be active.

3. Click **Settlement**.
4. In the **Properties** window, adjust the load details according to your needs.
5. In the model, click a supported node to place the load.

Modify panel, member, and structure loads

To modify the properties of existing loads, see the following instructions.

1. In the model, select the load that you want to modify.
2. In the **Properties** window, adjust the load details according to your needs.

Delete panel, member, and structure loads

If necessary, you can delete existing loads in your model. In order to do so, see the following instructions.

1. Ensure that the loadcase containing the load is displayed in the **Loading** list.
2. On the **Quick Access** toolbar at the top of the window, click **Delete**.
3. Click the load that you want to delete.

Decompose panel loads

TIP Although Tekla Structural Designer carries out load decomposition automatically when you click to analyze the structure, you have the option to decompose loads manually by using the **Decomposition** command.

In the context of big or complex models, decomposing manually can potentially save time, as it allows you to check that panel loads have been decomposed as you intend before running the analysis.

Decompose panel loads for an individual construction level

1. Open a 2D view of the desired level. (page 86)
2. Click the '2D' toggle button at the bottom right corner of the view to display it in 3D.

3. On the **Load** tab, click **Decomposition**.

   Tekla Structural Designer generates an FE mesh within the two-way slab panels, and applies the resulting decomposed loads to the supporting members.

   TIP In case you cannot see the changes in the model at this point, ensure that you have selected the correct settings in **Scene Content** - See **View decomposed loads** below.

Decompose panel loads to all required levels

1. Open the **Structure 3D** view.

2. On the **Load** tab, click **Decomposition**.

   Tekla Structural Designer generates an FE mesh within the two-way slab panels, and applies the resulting decomposed loads to the supporting members.

   TIP In case you cannot see the changes in the model at this point, ensure that you have selected the correct settings in **Scene Content** - See **View decomposed loads** below.

View decomposed loads graphically

1. Open a 3D view of the model, or a 2D view displayed in 3D.
2. In **Scene Content**, go to **Loading**.
3. Select the **Decomposed** option.
4. According to your needs, do one of the following:

To	Do this
View the decomposed load values	• In the cell on the right side of **Decomposed**, select both **Geometry** and **Text**.
View the decomposed loads without load values	• In the cell on the right side of **Decomposed**, only select **Geometry**.

5. In the **Loading** list, select the loadcase.

NOTE Decomposed loads do not exist for 2-way slab items at levels where the **Mesh 2-way Slabs in 3D Analysis** option has been selected.

At levels where this is not the case, you can view the decomposed loads. However, you cannot see any shell results from the FE load decomposition.

View applied and decomposed member loads in a table

1. Open a 3D view of the model, or a 2D view displayed in 3D.
2. Right-click on the member, and from the context menu select **Show Member Loading**. Loads that have been applied directly to the member, and loads that have been decomposed to the member are displayed in a table. The 'In Proj.' checkbox is used to indicate applied loads applied in projection as opposed to along the element.
3. To filter the data by Loadcase, Source, Direction, or Type:
    a. Click the appropriate column header to filter by (Loadcase..., Source..., Direction..., or Type...)
    b. From the drop list that appears clear the categories that you don’t want to be displayed.
    c. Click the Close button under the drop list.

Overview of one-way and two-way load decomposition

The way in which Tekla Structural Designer decomposes panel loads depends on how the slabs/panels are modeled and how they are spanning.

Slab/Panel type	Decomposition
One-way spanning slab item	Tekla Structural Designer 1-way decomposes panel loads applied to the slab directly on to those supporting members/two-way spanning slab edges before performing 3D analysis. The rotation angle of the slab item determines the decomposition direction. **NOTE** Any openings in 1 way slabs are ignored and thus have no impact on load decomposition.

<table>
<tr><th>Slab/Panel type</th><th>Decomposition</th></tr>
<tr><td>Roof panel</td><td>Tekla Structural Designer 1-way decomposes panel loads directly on to supporting members/two-way spanning slab edges before performing 3D analysis.

The rotation angle of the roof panel determines the decomposition direction.</td></tr>
<tr><td>Two-way spanning slab item</td><td>• At those levels where the Mesh 2-way Slabs in 3D Analysis option is selected, load decomposition is not required.

• At other levels, any loads applied to two-way slabs are automatically 2-way decomposed back to supporting members during the 3D pre-analysis stage, immediately prior to the 3D analysis stage.

NOTE Where openings have been defined, any portion of a panel load that lies within the opening will not be decomposed.</td></tr>
<tr><td>Roof panel overlapping a one-way spanning slab item</td><td>Tekla Structural Designer 1-way decomposes panel loads applied to the slab directly on to supporting members/two-way spanning slab edges before performing 3D analysis.

The rotation angle of the slab item (and not the roof panel) determines the decomposition direction.

NOTE Any openings in 1 way slabs are ignored and thus have no impact on load decomposition.</td></tr>
<tr><td>Roof panel overlapping a two-way spanning slab item</td><td>• If there are no openings in the slab, decomposition is as per Two-way spanning slab item type described above.

• Where openings exist, any panel load applied inside the opening is</td></tr>
</table>

Slab/Panel type	Decomposition
	first one-way decomposed in the direction defined by the rotation angle of the roof panel on to the two-way slab at the edge of the opening. It is then further decomposed back to supporting members as per **Two-way spanning slab item** type described above. **NOTE** This does not apply when the panel load is an area load that has been applied to the slab item (as opposed to the roof panel), or a level load, or a slab load. In each of these instances the load will automatically be boxed out around the opening.

5.3 Apply wind, snow, and seismic loads

Wind loads can be applied using the **Wind Wizard**, or they can be applied manually - when applied manually they can take the form of panel, member, or structure loads, or simple wind loads.

Open structure wind loads are determined by running the **Wind Wizard**, and applied to members, ancilaries and equipment via their properties

Snow loads can either be applied using the **Snow Wizard** or they can be applied manually.

Seismic loads can only be applied using the **Seismic Wizard**.

- Apply wind loads using the wind wizard (page 387)
- Apply wind loads manually (page 393)
- Apply open structure wind loads (page 394)
- Apply snow loads using the snow wizard (page 396)
- Apply snow loading manually (page 406)
- Apply seismic loads (page 406)

Apply wind loads using the wind wizard

In Tekla Structural Designer, you can apply wind loads to your structure by using the **Wind Wizard** to generate a wind model.

- Create a wind model and wind loads (page 388)
- Modify wind zones and wind zone loads (page 390)
- Create and manage wind loadcases (page 392)

Create a wind model and wind loads

You can use the **Wind Wizard** to automate the wind modeling process. Where appropriate, the **Wind Wizard** uses databases to determine the appropriate wind details for your structure location, and then calculates the appropriate wind loading details according to the selected wind loading code.

RESTRICTION The **Wind Wizard** is not currently available for the Australian **AS:1170.2** wind loading code variant.

Once you have defined the wind directions in which you are interested, Tekla Structural Designer automatically calculates the appropriate wind zones on the roofs and walls of your structure. You can set the type of each roof to achieve the correct zoning, and then tailor the zoning to account for particular features in more detail, if you so require.

The wind modeling process can automatically define standard wind loadcases for you based on the usual internal pressure coefficients, or you can define the loadcase information yourself. In both cases, the appropriate wind pressures are calculated on each zone. You can then combine the wind loadcases into design combinations as usual.

NOTE The determination of the wind speeds, the pressures, and the zones is rigorous. However, do remember that the final wind loads that are adopted are your responsibility.

Run the Wind Wizard

NOTE You must define at least one wall or roof panel before running the **Wind Wizard**.

1. On the **Load** tab, click  **Wind Wizard...**

   The **Wind Wizard** opens.

   NOTE The **Wind Wizard** varies slightly according to the head code that you are using.

2. Define the necessary information for the wind model. To go to the next page in the wizard, click **Next**.
3. Once you have defined all the necessary information, click **Finish**.

NOTE After running the **Wind Wizard**, you can review the roof and wall zones for each wind direction.

After you have created the wind model with the **Wind Wizard**, you can open a wind view to graphically display the wind zones and loading that apply for a particular wind direction.

Add wind directions

1. On the **Load** tab, click **Wind Wizard...**

   The **Wind Wizard** opens.
2. Click **Next** until you are on the **Results** page.
3. Click **Add Dir.**
4. Specify the properties of the new wind direction.
5. Click **Finish**.

NOTE Remember to add new wind loadcases and design combinations to incorporate the wind loading for the new direction into your calculations.

Delete wind directions

1. On the **Load** tab, click **Wind Wizard...**

   The **Wind Wizard** opens.
2. Click **Next** until you are on the **Results** page.
3. Click the wind direction that you want to delete.
4. Click **Del Dir.**
5. Click **Finish**.

NOTE Remember to update the existing wind loadcases and design combinations to remove the details for the wind direction you have deleted from your calculations.

Delete the entire wind model

To delete the entire wind model and start the wind modeling process from scratch, do the following:

1. On the **Load** tab, click **Delete Wind**.

   Tekla Structural Designer deletes all the previously defined wind directions and wind loadcases.

Modify wind zones and wind zone loads

In order to view and modify the wind zones and loading that apply to a particular wind direction, you have to open the appropriate wind view. Then, you can modify the wind zones and wind zone loads according to your needs.

Open a wind view

NOTE Wind views are only available if you have already created the wind model with the **Wind Wizard**.

---

1. In the **Project Workspace**, go to the **Wind** tab.
2. Right-click the wind direction that you want to view.
3. In the context menu, click **Open View**.

   Tekla Structural Designer opens the selected wind view, and the **Zone Loads** tab, which allows you to review wind zone data graphically.

View wind zones

1. Open a wind view that displays the details of the desired wind direction.
2. On the **Zone Loads** tab, click **Wind Zones**.

   The wind view displays the zones that are applied to the structure for the selected wind direction.

Modify wind zones

1. Open a wind view that displays the details of the desired wind direction.
2. In the **Project Workspace**, go to the **Wind** tab.
3. For the wind direction in question, expand the **Roof Zones** or **Wall Zones** branch.
4. In the desired branch, right-click the panel label containing the zone that you want to edit.

5. In the context menu, select **Edit Zones...**

   The **Zone Properties** dialog box opens.
6. Modify the wind zones according to your needs.
7. Click **OK**.

View the wind zone loads

NOTE Remember to update the zone loads when you make changes to your model.

1. Open a wind view that displays the details of the desired wind direction.
2. On the **Zone Loads** tab, click **Zone Loads**.
3. In the **Loading** list, select the desired wind loadcase.

   Tekla Structural Designer displays the loads that are applied to the structure for this loadcase in this wind direction.

Modify wind zone loads

1. Open a wind view that displays the details of the wind direction whose zone loads you want to modify.
2. On the **Zone Loads** tab, click **Zone Loads**.
3. In the **Loading** list, select the desired wind loadcase.

   Tekla Structural Designer displays the loads that are applied to the structure for this loadcase in this wind direction.
4. In the model, click the wind zone that you want to change.
5. In the dialog that opens, clear the **Use Default Values** option.
6. Modify the values that determine the loads.

   TIP If you want to reduce the net pressure for benecifial loads to zero, select the **Beneficial Load** option.
7. To update the zones, **OK**.

Update wind zone loads

When you change the roofs or walls of your structure, Tekla Structural Designer does not automatically update the changes to existing wind zoning. Updating the zoning is manual because you may wish to make more alterations before you recalculate the zoning.

Once you have completed your changes, incorporating them and recalculating the zoning details is simple. To reinstate the zoning after making changes, do the following:

1. On the **Load** tab, click **Update Zones**.

   The wind zoning calculations run in the background. Once the calculations are complete, Tekla Structural Designer views the new zoning layout for your structure.

   NOTE If you have defined your own zone layout for any roof or wall, the existing zones will be maintained when you make changes. Make sure to update these zones appropriately.

Create and manage wind loadcases

Once you have defined the basic wind data for your model, and calculated wind zoning in the **Wind Wizard**, you can define the required wind loadcases manually. For detailed instructions, see the following paragraphs.

Define wind loadcases

1. On the **Load** tab, click **Wind Loadcases**.

   The **Wind Loadcases** dialog box opens.

2. Do one of the following:

   - Click **Add** to add the details of each wind loadcase individually.
   - Click **Auto** to generate standard wind loadcases for the wind directions that you have defined in the **Wind Wizard**.

     NOTE You cannot use the **Auto** option once you have created other wind loadcases.

Add wind loadcases

1. On the **Load** tab, click **Wind Loadcases**.

   The **Wind Loadcases** dialog box opens.

2. Click **Add**.
3. Define the details of the new loadcase according to your needs.
4. Repeat steps 2 and 3 for each loadcase that you want to create.
5. Click **OK** to close the **Wind Loadcases** dialog box.

   NOTE Remember to update or add design combinations, so that they take the new wind loadcases into account.

Delete wind loadcases

1. On the **Load** tab, click **Wind Loadcases**.
2. Click the wind loadcase that you want to delete.
3. Click **Delete**.
4. Repeat steps 2 and 3 to delete further loadcases.
5. Click **OK** to close the **Wind Loadcases** dialog box.

   NOTE Remember to update your design combinations, so that they take the deleted wind loadcases into account.

Apply wind loads manually

In case you do not want to construct an entire wind model, you can elect instead to apply the wind loads manually. These can be applied as panel, member, or structure loads in the usual way, or as simple wind loads.

Create loadcases for manual wind loads

1. On the **Load** tab, click **Loadcases**.

   The **Loading** dialog box opens.
2. On the **Loadcases** page, click **Add**.
3. Name the new loadcase.
4. Set the type to **Wind**.
5. Click **OK**.

Create simple wind loads

RESTRICTION You can only create simple wind loads in the **Structure 3D** view.

NOTE In order access the **Simple Wind Loading** dialog box, you must first create and select the loadcase where you want to add the simple wind loads.

1. In the **Loading** list, select a manually created wind loadcase.
2. On the **Load** tab, click **Simple Wind**.
3. In the model, click to define the start point of the load width.

4. Click to define the end point of the load width.

   The **Simple Wind Loading** dialog box opens. The dialog box allows you to specify a single area wind load from the lowest level of the building up to the highest level that contains a rigid diaphragm.

   TIP The wind load does not have to begin at the lowest level of the building.

   If you have one or more levels below ground, you can adjust the value of the lowest level in the **Simple Wind Loading** dialog box.

   ---

5. Specify the required area load.
6. To create a stepped profile up the height of the building, use the **Insert Above**, **Insert Below**, **Quick Above...** and **Quick Below...** buttons.
7. Specify the load for each level.
8. When you load is complete, click **OK**.

Modify simple wind load vertical properties

1. Double-click anywhere in an existing simple wind load.

   The **Simple Wind Loading** dialog box opens.
2. Modify the load properties according to your needs.
3. Click **OK**.

Modify the simple wind load width

1. Select the simple wind load that you want to modify.
2. Select one of the two nodes that define the load width.
3. Click where you want to move the selected node.

   Tekla Structural Designer moves the selected node to the new location, and regenerates the load.

Apply open structure wind loads

Apply open structure wind load to selected entities

Open structure wind loads can only be applied to structural members, ancillaries, and equipment.

1. Select the members to which the load is to be applied.

2. In the **Properties** window, under **Wind loading**, select **Apply open structure wind load**

   Additional properties then become available, allowing the shape factor and effective area factors used in the wind load calculation to be customized.

3. Edit these additional properties, if required.

   NOTE The **Apply open structure wind load** property is already selected by default for ancillaries and equipment, but can be deselected if required.

Run the wind wizard

NOTE You must apply an open structure wind load to at least one entity before running the **Wind Wizard**.

1. On the **Load** tab, click **Wind Wizard...**

   The **Wind Wizard** opens.

   NOTE The **Wind Wizard** varies slightly according to the head code that you are using.

2. On the first page of the wizard select the **Apply Open Structure Wind Load** option.
3. Step-through the remaining pages of the wizard to define the necessary information for the wind model.
4. Once you have defined all the necessary information, click **Finish**.

   NOTE After you have created the wind model with the **Wind Wizard**, you can open a wind view to graphically review the open structure wind loads for each wind direction.

Define wind loadcases

1. On the **Load** tab, click **Wind Loadcases**.

   The **Wind Loadcases** dialog box opens.

2. Do one of the following:
   - Click **Add** to add the details of each wind loadcase individually.

- Click **Auto** to generate standard wind loadcases for the wind directions that you have defined in the **Wind Wizard**.

NOTE You cannot use the **Auto** option once you have created other wind loadcases.

Apply snow loads using the snow wizard

In Tekla Structural Designer, you apply snow loads to your structure by running the **Snow Wizard** to input basic snow data and set up the required snow loadcases. Provided that Roof Panels have been modeled, some of these loadcases will be populated with Uniform Snow automatically; the remaining loadcases then have Uniform Snow, Valley Snow and Local Drift Snow applied manually as required. As an alternative, you can also choose to apply the snow loads manually.

NOTE The **Snow Wizard** is not currently available for the Indian, Australian, or British Standard snow code variants.

- Overview of snow loading using the snow wizard (page 396)
- Roof panel types (page 397)
- Run the snow load wizard (page 398)
- Snow loadcases (ASCE7) (page 398)
- Snow loadcases (Eurocode) (page 401)
- Apply drift loads to loadcases on completion of the snow wizard (page 403)
- Update snow loads (page 405)
- Delete the snow model (page 406)

Apply snow loading manually (page 406)

How do I define snow loading? (Playlist)

Overview of snow loading using the snow wizard

This section provides a brief overview of snow loading in Tekla Structural Designer.

The intensity of snow load is based upon geographic location, building/roof geometry, environmental factors and local roof factors.

All snow loading falls into three categories:

- Uniform Snow load (the first fall of snow)
- Drifted uniform snow (the first fall of snow blown into uneven uniform loading)

- Drift loading (local build-up of snow load behind steps, objects, parapets)

Prior to running the **Snow wizard...** you should ensure that roofed areas of the model are 'clad' with roof panels.

The **Snow wizard...** can then be run in order to define the basic snow load factors and in the case of the Eurocode, the snow loadcases and types relevant to the particular National Annex.

From this information all the required snow loadcases are automatically set up and the loads in the undrifted (or balanced) snow loadcase are created.

Following the wizard, you then manually define the drift cases. To do this, select the relevant snow loadcase, and then define the key attributes for the drift load prior to placing the load in the relevant position on the roof of the building.

The end result is a series of snow loadcases ready to be combined in the Combination Generator with other loadcases.

The basic steps are summarized as follow:

1. Apply **Roof Panel**s to the model
2. Run the **Snow wizard...**
3. Apply snow loads to the drift loadcases set up by the wizard
4. Combine snow loadcases into design combinations
5. Perform the static design

How do I define snow loading? (Playlist)

Roof panel types

For the snow loading calculations, Tekla Structural Designer has to distinguish between monopitch (monoslope) and pitched roofs. This is determined from the RoofType property that has been assigned to the roof panel.

The RoofType property is mapped for the determination of snow loading as follows:

RoofType	For snow loading this is considered as:
Default	Monopitch (Monoslope)
Flat	Monopitch (Monoslope)
Monopitch (Monoslope)	Monopitch (Monoslope)
Hip Gable	Pitched
Hip Main	Pitched
Mansard	Pitched

Run the snow load wizard

With the **Snow Wizard**, you can automatically generate snow loadcases in both **ASCE7** and **EN 1991-1-3** snow loading codes. Using the wizard, you can set up basic snow data, snow loadcases, and snow loads within the balanced, or undrifted, snow loadcases.

NOTE Snow loads in drift snow loadcases are not set up automatically, and must therefore be manually applied. For more information, see Manually apply snow loads to snow loadcases.

---

1. On the **Load** tab, click **Snow Load** --> **Snow wizard...**

   The **Snow Wizard** opens.

2. On the first page of the **Snow Wizard**, define the basic data.
3. Click **Next**.
4. On the second page, specify the loadcases that you want to generate, and the number of drift directions to be considered.
5. Click **Finish**.

   If necessary, you can now review the snow loadcases that you created.

See also

Snow wizard (ASCE7) (page 1220)

Snow wizard (Eurocode) (page 1211)

How do I define snow loading? (Playlist)

Snow loadcases (ASCE7)

The snow loadcases that are set up will depend on the head code that is being worked to. This section will look at the snow loadcases when the headcode is set to United States.

NOTE All loads and lengths of loads are assumed to act on a horizontal projection of the roof surface.

Once set up these loadcases are standard loadcases so you can include them in combinations in the normal manner.

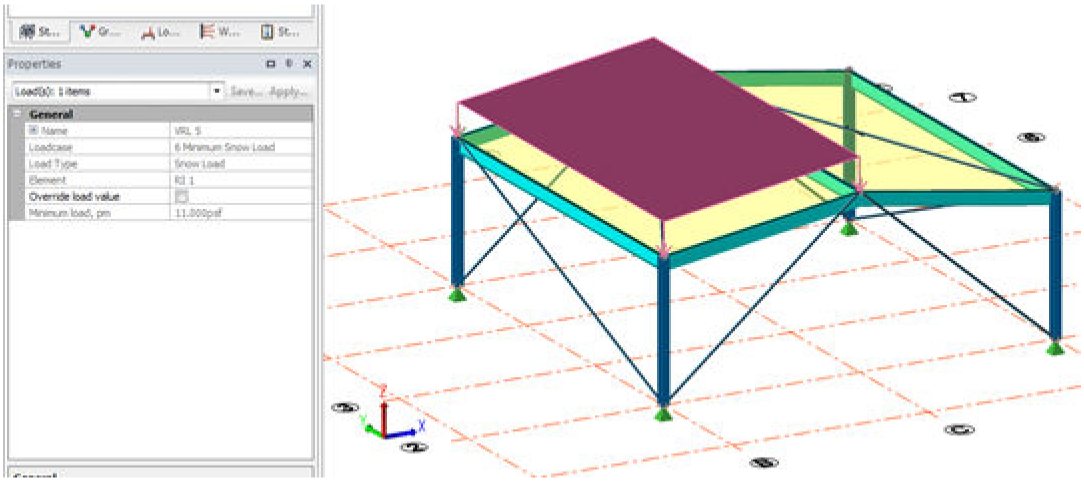
In the **Snow wizard...** you specify the Snow Loadcases to be set up from the following list:

- Minimum Snow Load - for low sloped roofs
- Balanced Snow load - uniform snow load and rain on snow load
- Unbalanced Snow Load - the number of cases required being chosen to reflect the number of wind directions considered

- Drift Snow Load
- Rain on Snow Surcharge

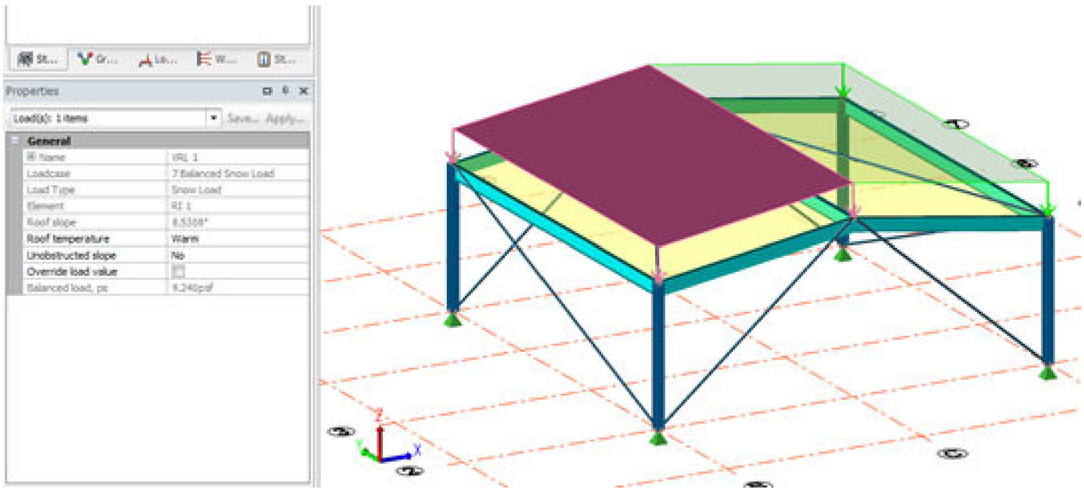
Minimum snow load

Loads would need to be manually applied to this loadcase on completion of the **Snow wizard...** - the only snow load applicable being Uniform Snow.



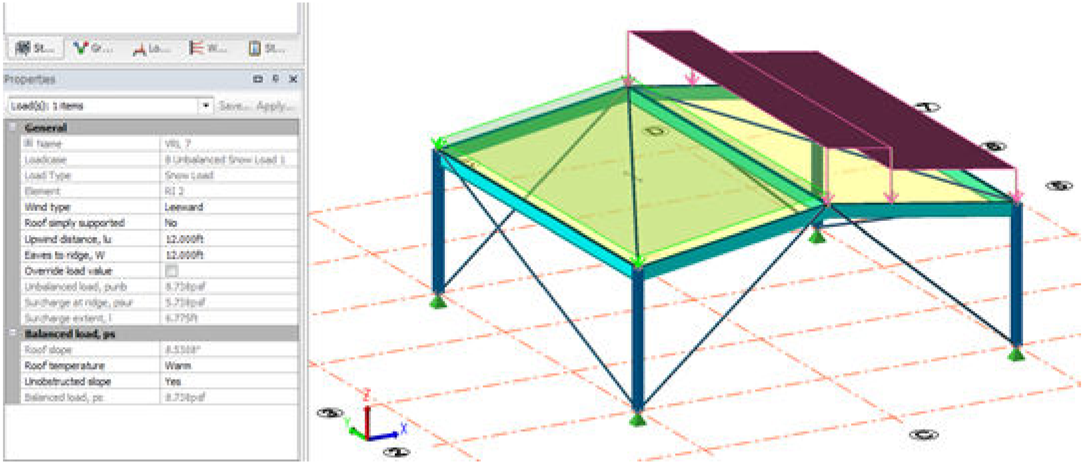
Balanced snow load

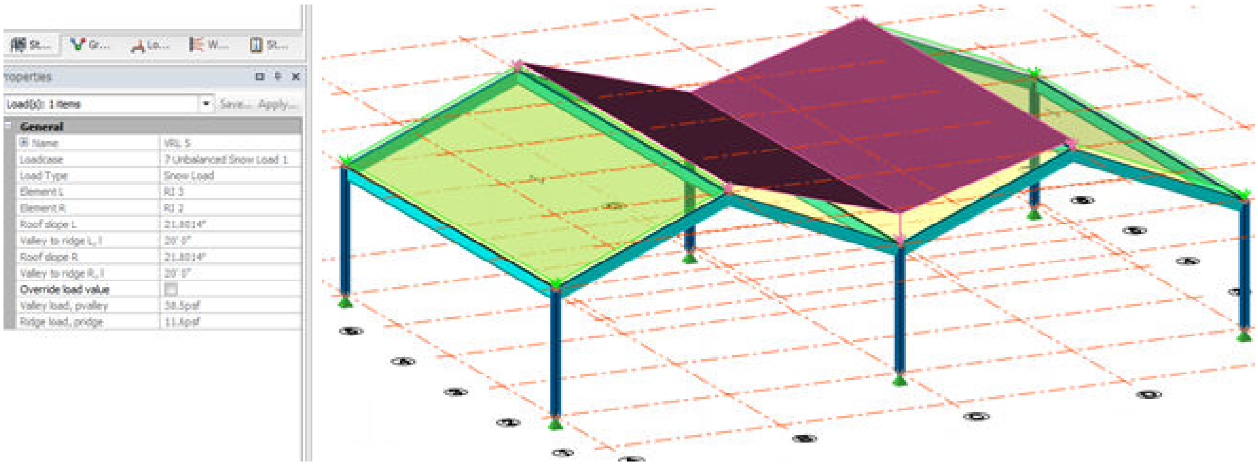
This loadcase is automatically populated on completion of the **Snow wizard...**



Unbalanced snow load

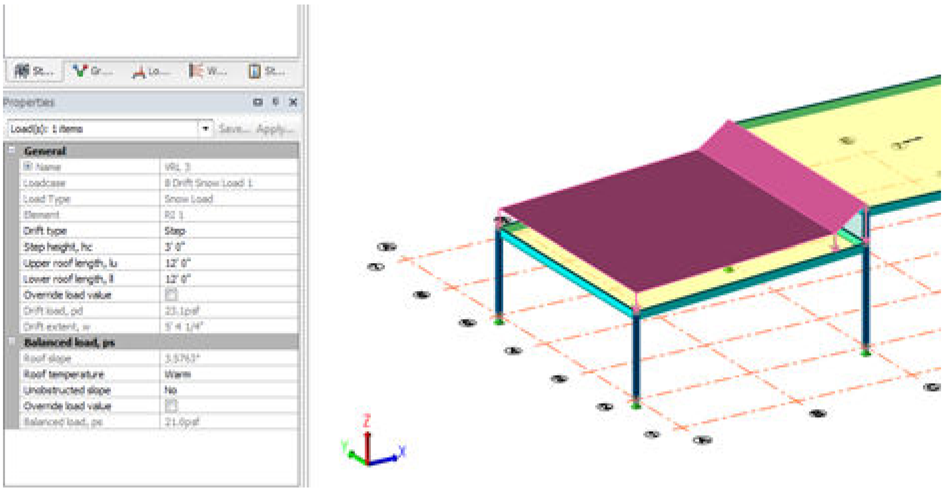
Loads would need to be manually applied to these loadcases on completion of the **Snow wizard...** - both Uniform Snow and Valley Snow loads may be applicable, (depending on the model geometry).





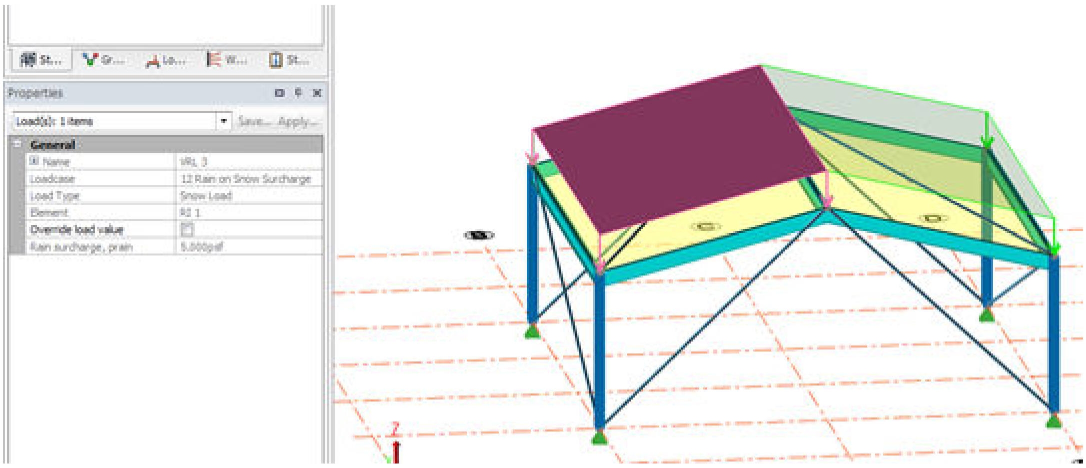
Drift snow loads

Loads would need to be manually applied to these loadcases on completion of the **Snow wizard...** - the only snow load applicable being Local Drift Snow.



Rain on Snow Load Surcharge

This loadcase is automatically populated on completion of the **Snow wizard...**



Snow loadcases (Eurocode)

The snow loadcases that are set up will depend on the head code that is being worked to. This section will look at the snow loadcases when the headcode is set to Eurocode.

NOTE All loads and lengths of loads are assumed to act on a horizontal projection of the roof surface.

Once set up these loadcases are standard loadcases so you can include them in combinations in the normal manner.

In the **Snow wizard...** you choose the Snow Loadcases to be set up from the following list:

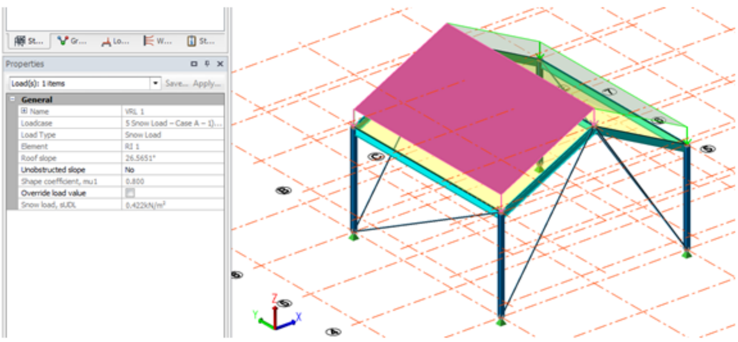
- Snow Load - Case A - 1) Undrifted
- Snow Load - Case A - 2) Drifted *
- Snow Load - Case B1 - 1) Undrifted Snow Load - Case B1 - 2) Drifted *
- Snow Load - Case B1 - 3) Undrifted (Acc)
- Snow Load - Case B1 - 4) Drifted (Acc) *
- Snow Load - Case B2 - 1) Undrifted
- Snow Load - Case B2 - 2) Drifted *
- If any drifts from Annex B are selected
  - Snow Load - Case B2 - 3) Drifted (Annex B) (Acc) *
- Snow Load - Case B3 - 1) Undrifted
- Snow Load - Case B3 - 2) Drifted *
- Snow Load - Case B3 - 3) Undrifted (Acc)
- If any drifts from Annex B are selected
  - • Snow Load - Case B3 - 4) Drifted (Annex B) (Acc) *

NOTE For cases marked * - the number of cases actually set up will depend on the number of wind directions that are asked for.

NOTE The Eurocode / National Annex recommends which loadcases to generate.

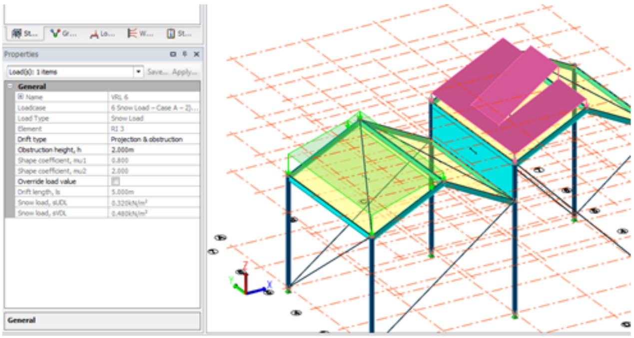
Undrifted loadcases

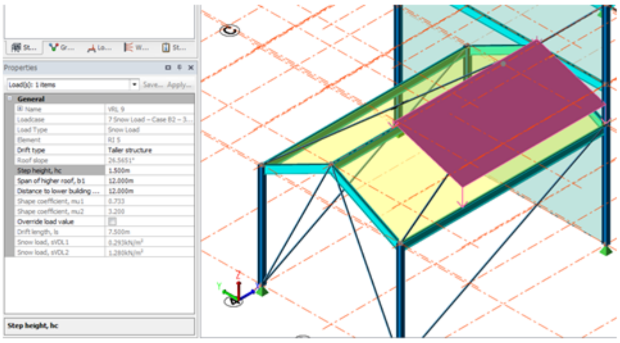
Any undrifted loadcases that have been set up in the **Snow wizard...** are automatically populated with uniform loading on completion of the wizard



Drifted loadcases

All drifted loadcases that have been set up in the **Snow wizard...** would need to be have their snow loads manually applied on completion of the wizard.





Apply drift loads to loadcases on completion of the snow wizard

The loads in the drifted snow loadcases have to be applied manually as uniform snow, valley snow, or local drift snow loads. Any snow loads that are not appropriate for the selected loadcase cannot be selected.

Apply uniform snow loads

1. In the **Loading** list, select a snow loadcase generated by the wizard appropriate for uniform snow.
2. On the **Load** tab, click **Snow Load** --> **Uniform Snow**.
3. In the **Properties** window, adjust the load details according to your needs.
4. To apply the load, click the relevant roof panel.

NOTE In the unbalanced cases, setting **Roof type** option to **Pitched** allows you to select whether the wind type of the load should be **Windward** or **Leeward**.

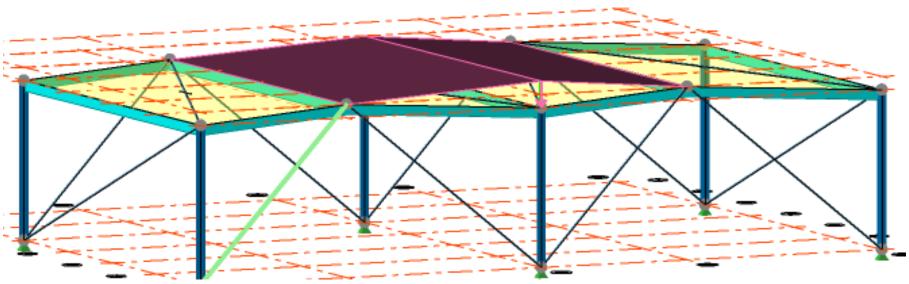
Apply a valley snow load

1. In the **Loading** list, select a snow loadcase generated by the wizard appropriate for valley snow.
2. On the **Load** tab, click **Snow Load** --> **Valley Snow**.
3. Select the left roof panel of the valley in which the load is applied.

   Tekla Structural Designer automatically determines the right panel, and calculates the valley load.

   TIP If necessary, you can use apply a user override to the calculated load. Even if you rerun the **Snow Wizard**, Tekla Structural Designer maintains any user overrides that you have applied.

   NOTE Graphic displays depict the loading magnitude of a snow load, and not depth of loading. Remember this when reviewing the load that has been calculated on structures like the low pitched valley model shown below:

   Although the above image appears to indicate snow load rising above the ridges, the display actually views the loading magnitude.

Apply a local drift snow load

1. In the **Loading** list, select a snow loadcase generated by the wizard appropriate for local drift snow.
2. On the **Load** tab, click **Snow Load** --> **Local Drift Snow**.
3. Go to the **Properties** window.
4. Specify the drift type, and then enter the height, and any other parameters required to define the drift.
5. In the model, select the roof panel edge where the drift is to form, taking care to click close to the required edge.

TIP The edge closest to the cursor is highlighted, if this isn't where you want the drift to form move the cursor toward the edge required before clicking to select.

6. Click the start point of the drift on the highlighted edge.
7. Click the end point of the drift on the highlighted edge.

TIP If necessary, you can use apply a user override to the calculated load. Even if you rerun the **Snow Wizard**, Tekla Structural Designer maintains any user overrides that you have applied.

8. If you have another drift load to apply to the same panel in the same loadcase, repeat the above steps starting from step 3.

NOTE If working to the US headcode, when applying the second and subsequent drift loads to the same roof panel, a **Do not apply balanced load** setting is automatically enabled in the load properties. (This setting was disabled for the first drift load to ensure the balanced load gets applied.) The second and subsequent drift loads use the balanced load of the first drift to determine their drift load and extent. Working in this way, it is not necessary to override the load value in order to apply the second and subsequent drift loads.

Override snow loads

You can apply user overrides for snow load values and drift lengths, if necessary.

1. Select the snow load.
2. In the **Properties** window, select the **Override load value** option.
3. Specify the override value.

   Tekla Structural Designer maintains the user overrides, even if you rerun the **Snow Wizard**.

Update snow loads

If you have created a snow model using the **Snow Wizard**, and later decide to modify the roof geometry, the snow loads need to be recalculated. To do so, see the following instructions.

- On the **Load** tab, click **Snow Load** --> **Update Snow Loads**.

See also

Delete the snow model (page 406)

Delete the snow model

Occasionally, you may need to delete the entire snow model, and start the process from scratch. In order to do so, see the following instructions.

- On the **Load** tab, click **Snow Load** --> **Delete Snow**.

  Tekla Structural Designer resets the basic data that was previously defined in the **Snow Wizard**, and deletes the snow loadcases.

Apply snow loading manually

This approach provides an alternative method to apply snow loading to the structure, without having to run the **Snow Wizard**

In order to manually apply snow loads, you must first create a loadcase (page 341) and set its load type to Snow.

You can then manually apply loads (page 364) to this loadcase in the normal way.

NOTE Snow loadcases created manually in this way can have panel, member, and structure loads applied but cannot have snow (i.e. Uniform Snow, Valley Snow and Local Drift Snow) loads applied.

Apply seismic loads

Use the **Seismic Wizard** to define all the parameters required to set up the seismic loadcases and combinations.

RESTRICTION The **Seismic Wizard** is not currently available for the Australian **AS:1170.4** loading code variant.

Create seismic loads in the Seismic Wizard

1. On the **Load** tab, click **Seismic Load** --> **Seismic Wizard...**

   The (page 407) opens.

   NOTE The parameters in the **Seismic Wizard** vary according to the loading code that you have selected in **Model Settings**.

   ---

2. In the **Seismic Wizard**, define the necessary parameters for the seismic loading and loadcases. To go to the next page, click **Next**.

3. Once you have defined the parameters, click **Finish**.

   The **Combination Generator** dialog box opens.

4. In the **Combination Generator** dialog box, define the seismic load combinations.

5. Click **Finish**.

Display the horizontal design spectrum

After running the **Seismic Wizard**, you can view the horizontal design spectrum.

1. On the **Load** tab, click **Seismic Load** --> **Horizontal Spectrum**.

2. If necessary, in the **Properties** window, you can switch **Direction** between **Dir 1** and **Dir 2**.

Delete seismic loads

- On the **Load** tab, click **Seismic Load** --> **Delete Seismic**.

  Tekla Structural Designer deletes all the previously defined seismic loads in your model.

See also

Seismic wizard in detail (page 407)

Seismic wizard in detail

The **Seismic Wizard** is run to specify the seismic analysis method (RSA or ELF) and to set up the seismic loadcases and combinations.

The information that is entered in the seismic wizard will vary according to the seismic loading code that is being worked to:

- Using the ASCE7 seismic wizard (page 407)
- Using the UBC 1997 seismic wizard (page 415)
- Using the Eurocode EN1998-1:2004 seismic wizard (page 423)
- Using the IS1893 seismic wizard (page 430)
- Code spectra and site specific spectra (page 436)
- Seismic loadcases (page 448)

Using the ASCE7 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

- Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the ASCE Horizontal Design Spectrum (page 439).
Site Specific Spectra (user defined - based on S_d and T)	User defined spectra are appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 436).
Site Specific Spectra (user defined - generic curve)	User defined spectra are appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve manually, or by pasting data from a spreadsheet. **NOTE** The spreadsheet data must be in the form of period vs acceleration values and in ascending order of period (the input dialog features an automatic Sort command if the latter is not the case). See: Code spectra and site specific spectra (page 436).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

The Seismic Design Category (SDC) is determined on this page.

NOTE In ASCE7-10 (not ASCE7-05) there is an additional value of C_S calculated for a building < 5 stories and where T <0.5s. In this instance S_S is taken as 1.5 and the additional value of C_S should be used to give a final value of C_S. This clause has not been implemented in Tekla Structural Designer.

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic weight combination is determined.
Number of stories	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Site Occupancy	
Site class	This field allows you to set the appropriate soil conditions, A-E - (A - Hard rock, B - Rock, C - Very dense soil/soft rock, D - Stiff soil, E - Soft soil. as defined in ASCE7-05 & ASCE-10 Table 20.3-1.
Occupancy/ Risk Category	This field allows you to set the appropriate occupancy category, I-IV, as defined in ASCE 7-05 &-10 Table 1-1.
Importance Factor, I_e	This is automatically derived from the occupancy class.
Override check box	Checking this box allows you to override the importance factor, (some countries require different Importance Factors to those given in the US codes).
Seismic Design Category (SDC)	
Seismic Design Category (SDC)	User editable (range A-F)
Alternative seismic design category determination	Select this check box in order to use the alternative determination from IBC2009 1613.5.6.1 / IBC2012 1613.5.6.

Property	Description
User Defined SDC	Select this check box in order to specify your own SDC Category
Max earthquake spectral response acceleration	
S_S – short period (0.2s)	This figure can be determined from the maps found in ASCE 7-05 &-10 (units % of g, range 1 – 500)
S_1 – 1.0s period	This figure can be determined from the maps found in ASCE 7-05 &-10 (units % of g, range 1 – 500)
Design spectral response spectral acceleration	
S_{DS} – short period	This is automatically determined from SS and the site class, according to ASCE 7-05&-10
S_{D1} – 1s period	This is automatically determined from S1 and the site class, according to ASCE 7-05&-10
Next	Unless the SDC is type A, clicking **Next** takes you to the **Structure Irregularities page** described below; for SDC type A only, clicking **Next** takes you to the **Seismic Loading page**.

Structure Irregularities page

For SDC types B to F this page is used to indicate any irregularities in plan or vertically.

Property	Description
Structure Plan Irregularities	Check the appropriate boxes to define any plan irregularities.
Structure Vertical Irregularities	Check the appropriate boxes to define any vertical irregularities.
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Equivalent Lateral Force Procedure

Property	Description
	• Use Modal Response Spectrum Analysis **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

For SDC types B to F this page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$.

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_a	This is automatically derived from ASCE7-05&-10 clause 12.8.2.1.
User defined fundamental period T	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period	
Long period transition period, T_L	This figure can be determined from figs 22-15, 16, 17, 18, 19 and 20 found in ASCE 7-05 or figs 22-12, 13, 14, 15 and 16 found in ASCE 7--10.
T_S	This value is derived ($T_s = S_{D1}/S_{DS}$)
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select from Steel moment resisting frames, Concrete moment resisting frames, Eccentrically braced steel frames or All other structural systems (ref ASCE7-05 / ASCE7-10 Table 12.8-2
Approx fundamental period, T_a	This is automatically derived from ASCE7-05&-10 clause 12.8.2.1.

Property	Description
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_a, or it can be a user defined value, or it will be calculated from the modal analysis.
Next	Clicking **Next** takes you to the **Seismic Force Resisting System page** described below.

Seismic Force Resisting System page

For SDC types B to F this page is used to determine the response modification coefficient, R, and other factors in the X and Y directions.

Property	Description
Seismic force resisting system Dir 1 and Dir 2	
System	This field allows you to set the appropriate system, (eg. bearing wall, building frame, moment resisting frame etc.) from ASCE7-05 &-10 Table 12.2-1.
Type	This field allows you to set the appropriate type for the chosen system from ASCE7-05 &-10 Table 12.2-1.
Coefficients & Factors Dir 1 and Dir 2	
Response modification coefficient, R	This is automatically derived, but you are given the facility to edit the calculated value.
System over-strength factor	This is automatically derived, but you are given the facility to edit the calculated value.
Deflection amplification factor, C_d	This is automatically derived, but you are given the facility to edit the calculated value.
Redundancy factor, ρ	This is automatically derived from the SDC, (SDC B-C ρ = 1.0, SDC D-F ρ = 1.3) but you are given the facility to edit the calculated value.
Next	Clicking **Next** takes you to the **Effective Seismic Weight page** described below.

Effective Seismic Weight page

This page is used to determine the Structure Seismic Dead Weight.

You should include those loadcases that you want to contribute to the Structure Seismic Dead Weight.

NOTE This "base shear combination" is used to develop the seismic design loading and is not used in any analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

On this page you can:

- choose whether to perform seismic drift checks, and set the allowable story drift factor to be used
- define appropriate localization parameters for using the ASCE7 code outside of the US.

Property	Description
Skip seismic drift checks - Check this box if you do not require seismic drift checks to be performed.	
Structure Type	Select the structure type for determining the allowable story drift factor.
Risk Category	Displays the risk category that was selected on an earlier page of the wizard.
Allowable Story Drift Factor	The factor to be used when calculating the allowable story drift is determined from the above two parameters. Click 'override' if you want to enter your own value.
Include localization - Check this box in order to display the below 'ELF & RSA' and 'RSA only' localization settings.	
ELF & RSA	
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor, (ASCE7 code value = 1/(R/I)).
User Defined Static Base Shear (Dir1 and Dir2)	This field allows you to override the code value for the static base shear, (ASCE7 code value V = Cs x W and Cs = SDS / (R /Ie)).
% eccentricity for accidental torsion	This field allows you to override the code eccentricity for accidental torsion, (ASCE7 code value = 5%). **NOTE** The same value is applied to all floors in the structure.

Property	Description
(Dir1 and Dir2)	
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination),(ASCE7 factor = 0.2 x S_{DS} x Dead loads).
RSA only	(only available if RSA has been selected)
% Mass participation for RSA	This field allows you to set your own required % mass participation, (ASCE7 code value = 90%). **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (ASCE7 code value: 85% x V / V_t).
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (ASCE7 code value: 85% x V / V_t). **NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 448) and open the **ASCE7 Seismic Combination Generator** described below.

ASCE7 Seismic Combination Generator

The Combination Generator sets up seismic combinations from the following ASCE7-5 and ASCE7-10 "core" combinations that include seismic loads.

ASD - Seismic Combinations

5) 1.0D + 0.714E

6) 1.0D + 0.75L + 0.536E + 0.75S

8) 0.6D + 0.714E

NOTE In ASCE, in the above 0.714 equates to 0.7 and 0.536 equates to 0.525. Tekla Structural Designer utilizes the ASCE factors - you can however change them if required.

LRFD - Seismic Combinations

5) 1.2D + 1.0E + 1.0L + 0.2S

7) 0.9D + 1.0E

(Where E is the seismic component of the loading in the combination.)

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this check box to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service factors.
Finish	Click **Finish** to generate the Seismic combinations.

Using the UBC 1997 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure in accordance with the Uniform Building Code.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

1. Click Home > Model Settings > Design Codes
   a. Set the Design Code for Seismic Loading to UBC
   b. Click OK
2. Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the UBC Horizontal Design Spectrum.
Site Specific Spectra	This option is appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 436).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

The Seismic Design Category (SDC) is determined on this page.

NOTE In ASCE7-10 (not ASCE7-05) there is an additional value of C_S calculated for a building < 5 stories and where T <0.5s. In this instance S_S is taken as 1.5 and the additional value of C_S should be used to give a final value of C_S. This clause has not been implemented in Tekla Structural Designer.

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).

Property	Description
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic weight combination is determined.
Number of stories	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Soil Profile Occupancy	
S - Soil Profile Type	This field allows you to set the appropriate soil conditions, A-E - (A - Hard rock, B - Rock, C - Very dense soil/soft rock, D - Stiff soil, E - Soft soil. As defined in UBC97 Table 16-J.
Occupancy Category	This field allows you to set the appropriate occupancy category, I-IV. As defined in UBC97 Table 16-K.
Importance Factor, I	This is automatically derived from the occupancy category.
Override check box	Checking this box allows you to override the importance factor, (some countries require different Importance Factors to those given in the US codes).
Seismic Zone	
Seismic Zone	This field allows you to set the appropriate zone, I-4. As defined in UBC97 Figure 16-2
Z - Seismic Zone Factor	This is automatically derived for the seismic zone.
Near Source (for Seismic Zone 4 only)	
Seismic Source Type	This field allows you to set the appropriate source type, A-C.
Distance to Seismic Source	The distance to the seismic source.
Next	Clicking **Next** takes you to the **Structure Irregularities page** described below.

Structure Irregularities page

For SDC types B to F this page is used to indicate any irregularities in plan or vertically.

Property	Description
Structure Plan Irregularities	Check the appropriate boxes to define any plan irregularities.

Property	Description
Structure Vertical Irregularities	Check the appropriate boxes to define any vertical irregularities.
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Static Force Procedure • Use Modal Response Spectrum Analysis **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

For SDC types B to F this page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$.

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_a	This is automatically derived. If RSA was selected on the previous page this option is dimmed.
User defined fundamental period	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select from Steel moment resisting frames, Concrete moment resisting frames, Eccentrically braced steel frames or All other structural systems
Approx fundamental period, T_a	This is automatically derived.

Property	Description
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_a, or it can be a user defined value, or it will be calculated from the modal analysis.
Next	Clicking **Next** takes you to the **Seismic Force Resisting System page** described below.

Seismic Force Resisting System page

This page is used to determine the response modification coefficient, R, and other factors in the directions 1 and 2.

Property	Description
Seismic force resisting system Dir 1 and Dir 2	
System	This field allows you to set the appropriate system, (eg. bearing wall, building frame, moment resisting frame etc.) from UBC97 Table 16-N.
Type	This field allows you to set the appropriate type for the chosen system from UBC97 Table 16-N.
Coefficients & Factors Dir 1 and Dir 2	
Response modification coefficient, R	This is automatically derived, but you are given the facility to edit the calculated value.
System over-strength factor	This is automatically derived, but you are given the facility to edit the calculated value.
Redundancy factor drift, ρ_{drift}	This factor has to be entered manually.
Redundancy factor, ρ	This factor has to be entered manually.
Next	Clicking **Next** takes you to the **Effective Seismic Weight page** described below.

Effective Seismic Weight page

This page is used to determine the Structure Seismic Dead Weight.

You should include those loadcases that you want to contribute to the Structure Seismic Dead Weight.

NOTE This “base shear combination” is used to develop the seismic design loading and is not used in any analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

On this page you can:

- choose whether to perform seismic drift checks, and set the allowable story drift factor to be used
- define appropriate localization parameters for using the UBC code outside of the US

Property	Description
Skip seismic drift checks - Check this box if you do not require seismic drift checks to be performed.	
Structure Type	Select the structure type for determining the allowable story drift factor.
Allowable Story Drift Factor	The factor to be used when calculating the allowable story drift is varies according to the Structure Type. Click 'override' if you want to enter your own value.
Include localization - Check this box in order to display the below 'ELF & RSA' and 'RSA only' localization settings.	
SFP & RSA	
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor, (UBC code value = 1/R).
User Defined Static Base Shear (Dir1 and Dir2)	This field allows you to override the code value for the static base shear, (UBC code value V = C_V x I x W / (R x T)).
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion, (UBC code value = 5%). **NOTE** The same value is applied to all floors in the structure.

Property	Description
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination), (UBC factor = 0.5 x Ca x I x Dead loads for strength design and 0.0 for allowable stress design).
RSA only	(only available if RSA has been selected)
% Mass participation for RSA	This field allows you to set your own required % mass participation, (UBC code value = 90%). **NOTE** This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, UBC code values: Code ground motion = 90% x V / V_{Design} Site specific ground motion = 80% x V / V_{Design} Irregular structures = 100% x V / V_{Design}
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, UBC code values: Code ground motion = 90% x V / V_{Design} Site specific ground motion = 80% x V / V_{Design} Irregular structures = 100% x V / V_{Design} **NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 448) and open the **UBC Seismic Combination Generator** described below.

UBC Seismic Combination Generator

The Combination Generator sets up seismic combinations from the following UBC 1997 "core" combinations that include seismic loads.

ASD - Seismic Combinations

5) 1.0D + 0.714E

6) 1.0D + 0.75L + 0.536E + 0.75S

8) 0.6D + 0.714E

NOTE In ASCE, in the above 0.714 equates to 0.7 and 0.536 equates to 0.525. Tekla Structural Designer utilizes the ASCE factors - you can however change them if required.

LRFD - Seismic Combinations

5) 1.2D + 1.0E + 1.0L + 0.2S

7) 0.9D + 1.0E

(Where E is the seismic component of the loading in the combination.)

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this check box to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service factors.
Finish	Click **Finish** to generate the Seismic combinations.

Using the Eurocode EN1998-1:2004 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

- Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the EN 1998-1 Horizontal Design Spectrum (page 441).
Site Specific Spectra	This option is appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 436).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic inertia combination is determined.

Property	Description
Number of storeys	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Ground acceleration	
Region	Only applies for the Malaysia NA: • Peninsular Malaysia, • Sarawak, • Sabah
Reference Peak Ground Acc, a_{gR}	Refer to PD6698:2009
Design Ground Acc, a_g	a_{gR} x γI.
Importance & Ground	
Importance class	For Base Eurocode and UK NA: I- IV as defined in BS EN 1998-1:2004 - Table 4.3. For Singapore NA: Ordinary or Special
Ground type	For Base Eurocode and UK NA: A-E For Malaysia NA: Rock, Stiff Soil, Flexible Soil For Singapore NA: C, D, S_1 (A - Rock, B - Very dense soil/gravel/clay, C - Deposits of dense/ medium dense soil, D - Loose to medium soil, E - Surface alluvium. Ref BS EN 1998-1:2004 - Table 3.1.)
Importance Factor, γ_I	This is automatically derived from the occupancy class.
Spectrum type	For Base Eurocode and UK NA: Spectrum Type - 1 or 2 BS EN 1998-1:2004 - Cl 3.2.2.2.1(P) For Malaysia NA: Spectrum Type - 1 For Singapore and Norway NA: (No choice)
Lower bound factor, β	This is automatically derived.
Upper limit of the period of the constant spectral	This is automatically derived.

Property	Description
acceleration branch, T_c	
Structural Ductility Class	Low, Medium or High - for whole building (not directional) If ag ≤ 0.78 m/s2 or if ag x S ≤ 0.98 m/s2 - Structure suitable for Low Seismicity (where S is the soil factor) If not - Structure suitable for Medium or High Seismicity
Site natural period, T_S (Malaysia NA only)	This is only required if the ground type is Flexible Soil.
Elastic response spectral displacement, S_{DR} (Malaysia NA only)	This is only required if the ground type is Flexible Soil.
Next	Clicking **Next** takes you to the **Structure regularity page** described below.

NOTE The Norway NA for EC1998 has the following differences:-

Soil factors specific to Norway

- φ factor for Imposed load types A, B and C = 1.0 (not 0.8)

- γI importance factors specific to Norway- Spectrum specific to Norway (TB, TC and TD factors)

- q values for Ductility Class Medium use for both Medium and High

Structure regularity page

For Medium and High structural ductility this page is used to indicate any irregularities in plan or elevation. (For Low structural ductility only the irregularities in elevation are displayed.)

Property	Description
Structure Plan Regularity - Cl 4.2.3.2	Check the appropriate boxes to define any plan irregularities.
Structure Elevation	Check the appropriate boxes to define any elevation irregularities.

Property	Description
Regularity - Cl 4.2.3.3	
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Equivalent Lateral Force Procedure • Use Modal Response Spectrum Analysi **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

This page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_A	If RSA was selected on the previous page this option is dimmed.
User defined fundamental period	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select from Moment resistant space steel frames, Moment resistant space concrete frames, Eccentrically braced steel frames, All other structures (ref BS EN 1998-1 Cl 4.3.3.2.2)
Approx fundamental period, T_A	This is automatically derived.
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_A, or it can be a user defined value, or it will be calculated from the modal analysis.

Property	Description
Next	Clicking **Next** takes you to the **Behaviour Factor page** described below.

Behaviour Factor page

For Low Ductility Class, q = 1.5, (but the value can be changed by the user) - all other fields on the page are dimmed and cannot be changed.

For Medium or High Ductility Classes, this page is used to determine the Behaviour Factor in direction 1 and 2.

Property	Description
Ductility Class	Medium or High.
Structure Type	Moment resistant space steel frames, Moment resistant space concrete frames, Eccentrically braced steel frames, All other structures
Frame Type	This options displayed here depend on the structure type selected above.
α_u / α_l	This is a user defined multiplication factor - for the structure
User defined q	For Medium or High Ductility Classes, select this check box in order to edit the calculated q value.
Behaviour Factor, q	This is automatically derived, but you are given the facility to edit the calculated value.
Next	Clicking **Next** takes you to the **Seismic Inertia Combination page** described below.

Seismic Inertia Combination page

This page is used to set up the loadcases for the Seismic Inertia Combination.

You should include those loadcases that you want to contribute to the effective seismic weight of the structure.

NOTE This “Seismic Inertia Combination” is used to develop the seismic design loading and is classed as a modal mass combination for the modal analysis. It is not used in any other analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

On this page you can:

- choose whether to perform seismic drift checks, and set the allowable story drift factor to be used

- define appropriate localization parameters

Property	Description
Skip seismic drift checks - Check this box if you do not require seismic drift checks to be performed.	
Structure Type	Select the structure type for determining the allowable story drift factor.
Importance Class	Displays the risk category that was selected on an earlier page of the wizard.
Allowable Story Drift Factor	The factor to be used when calculating the allowable story drift is determined from the above two parameters. Click 'override' if you want to enter your own value.
-	
ELF & RSA	
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor, (Eurocode value = 1/q).
Design Static Base Shear, V (Dir1 and Dir2)	This field allows you to override the code value for the static base shear, (Eurocode value $F_B = S_d (T_I) \times m \times \lambda$).
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion, (Eurocode value = 5%). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination),((Eurocode - No vertical loads considered - factor = 0.0).
RSA only	(only available if RSA has been selected)
% Mass participation for RSA	This field allows you to set your own required % mass participation, (Eurocode value = 90%).

Property	Description
	NOTE This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (Eurocode - No base shear adjustment).
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors, (Eurocode - No adjustment). **NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 448) and open the **EC8 Seismic Combination Generator** described below.

EC8 Seismic Combination Generator

The Combination Generator uses the two “core” combinations from EC8 that include seismic loads as below:

Equation 6.12a

$G_{kj} + A_{Ed} + \psi_{2,i} \times Q_{k,i}$

Equation 6.12b

$G_{kj} + A_{Ed}$

Provided the generate option is selected these will then be used, with factors as specified, to generate an appropriate number of combinations to cover for direction +/- Dir1 and Dir2 with +/- eccentricities.

Property	Description
Initial Parameters page	
Delete / Replace	Select whether to: • Delete all previously generated combinations

Property	Description
combinations	• Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this check box to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service combinations.
Finish	Click **Finish** to generate the Seismic combinations.

Using the IS1893 seismic wizard

The Seismic Wizard guides you through the process of defining the information that is required in order to calculate the seismic loading on the structure in accordance with the Indian IS1893 code.

This section runs through each page of the wizard and discusses the various options.

Starting the Wizard

To initiate the wizard:

1. Click Home > Model Settings > Design Codes
    a. Set the Design Code for Seismic Loading to IS1893 (Part 1) and select the year required
    b. Click OK
2. Click Load > Seismic Wizard...

The Seismic Wizard will start, and you can use its pages to define the necessary information. The first page displayed is **Site Specific Spectra**.

Site Specific Spectra page

You can choose the analysis procedure to be used, either:

Property	Description
Code Spectra	Choose this option to use the IS893 (Part 1) Horizontal Design Spectrum.
Site Specific Spectra	This option is appropriate for locations which use another country's loading and design codes where the code spectrum is not relevant. For this option you create a user defined spectrum by specifying the required curve parameters and slope options. See: Code spectra and site specific spectra (page 436).
Next	Clicking Next takes you to the **Basic information page** described below.

NOTE No vertical spectrum is considered in the current release.

Basic information page

The SZone Factor is determined on this page.

Property	Description
Structure details	
Height to the highest level	This field defaults to the structure height (calculated from the base to the highest point on the structure).
Ignore seismic in floor (and below)	Only floors above this level are considered when the seismic weight combination is determined.
Number of storeys	This field defaults to the number of ticked floors in the Construction levels dialog less the ones ignored in field above.
Zone & Site	
Seismic Zone	This field allows you to set the appropriate zone, II-V.
Site class	This field allows you to set the appropriate soil conditions, I-III.
Importance Factor, I	This is automatically derived from the occupancy class.
Next	Clicking **Next** takes you to the **Structure Irregularities page** described below.

Structure Irregularities page

This page is used to indicate any irregularities in plan or vertically.

Property	Description
Structure Plan Irregularities	Check the appropriate boxes to define any plan irregularities.
Structure Vertical Irregularities	Check the appropriate boxes to define any vertical irregularities.
Analysis procedure to be used	Having specified any irregularities, you then choose the analysis procedure. • Use Equivalent Lateral Force Procedure • Use Modal Response Spectrum Analysis **NOTE** Based on the irregularities you have defined, one or both of the above methods may be unavailable. Any additional information, assumptions or warnings applicable to the selected a method are displayed for information.
Next	Clicking **Next** takes you to the **Fundamental Period page** described below.

Fundamental Period page

This page is used to determine the fundamental period, $T_{Dir\ 1}$ and $T_{Dir\ 2}$.

Property	Description
Fundamental Period Definition	
Use approx fundamental period T_a	This is automatically derived. If RSA was selected on the previous page this option is dimmed.
User defined fundamental period	If RSA was selected on the previous page this option is dimmed.
Use modal analysis	Modal analysis is run to determine fundamental periods of the structure in Dir1 and Dir2.
Fundamental Period Dir 1 and Dir 2	
Structure Type	Select the structural type.

Property	Description
Approx fundamental period, T_a	This is automatically derived.
Fundamental period, $T_{Dir\ 1}$, $T_{Dir\ 2}$	Depending on your choice of definition this will either be taken as T_a, or it can be a user defined value, or it will be calculated from the modal analysis.
Next	Clicking **Next** takes you to the **Seismic Force Resisting System page** described below.

Seismic Force Resisting System page

This page is used to determine the response modification coefficient, R, and other factors in the directions 1 and 2.

Property	Description
Seismic force resisting system Dir 1 and Dir 2	
Type	This field allows you to set the appropriate type.
System	This field allows you to set the appropriate system for the chosen type.
Coefficients & Factors Dir 1 and Dir 2	
Response modification coefficient, R	This is automatically derived, but you are given the facility to edit the calculated value.
Next	Clicking **Next** takes you to the **Effective Seismic Weight page** described below.

Effective Seismic Weight page

This page is used to determine the Structure Seismic Dead Weight.

You should include those loadcases that you want to contribute to the Structure Seismic Dead Weight.

NOTE This “base shear combination” is used to develop the seismic design loading and is not used in any analysis of the structure.

Clicking **Next** takes you to the **Localization page** described below.

Localization page

On this page you can:

- choose whether to perform seismic drift checks, and set the allowable story drift factor to be used
- define appropriate localization parameters

Property	Description
Skip seismic drift checks - Check this box if you do not require seismic drift checks to be performed.	
Allowable Story Drift Factor	The factor to be used when calculating the allowable story drift. Click 'override' if you want to enter your own value.
Include localization - Check this box in order to display the below 'ELF & RSA' and 'RSA only' localization settings.	
ELF & RSA	
Scale Factor between elastic and design spectra (Dir1 and Dir2)	This field allows you to override the code value for the scale factor.
User Defined Static Base Shear (Dir1 and Dir2)	This field allows you to override the code value for the static base shear.
% eccentricity for accidental torsion (Dir1 and Dir2)	This field allows you to override the code eccentricity for accidental torsion). **NOTE** The same value is applied to all floors in the structure.
Vertical seismic load effect combination factor	This field allows you to override the combination factor for vertical seismic loading (usually a small additional factor for the dead loads in the combination).
RSA only	(only available if RSA has been selected)
% Mass participation for RSA	This field allows you to set your own required % mass participation.

Property	Description
	NOTE This overrides the setting in Analysis Options/1st order seismic when running the Seismic Wizard. **NOTE** It is not usually possible to achieve 100% mass participation in a modal analysis so max value is 99%. **NOTE** This setting can benefit some users wanting nearer 100% to be conservative.
Scaling of forces (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors
Scaling of Deflections for Drift Checks (% of Base Shear) (Dir1 and Dir2)	This field allows you to set your required scaling factors. **NOTE** A zero factor means no scaling will be performed on deflections.
Finish	Click **Finish** to automatically generate the Seismic loadcases (page 448) and open the **Seismic Combination Generator** described below.

Seismic Combination Generator

The Combination Generator sets up seismic combinations from the "core" combinations that include seismic loads.

Property	Description
Initial Parameters page	
Delete / Replace combinations	Select whether to: • Delete all previously generated combinations • Replace only combinations generated by this run
Scenario	If you need to generate combinations for a range of different scenarios, ensure you type in (or choose from the droplist if appropriate) a scenario name. This scenario name is then added as a prefix to all the combinations generated in the this run of the generator. You can then rerun the generator to create additional scenarios.

Property	Description
	Clicking **Next** takes you to the **Combinations page** described below.
Combinations page	
Generate	Select the seismic combinations to be generated with factors as specified.
Keep existing Seismic Combinations	If you want to retain previously set up Seismic Combinations you can select this check box to do so. **NOTE** Warning - some factors may no longer be correct for these combinations after rerunning the Wizard.
Next	Click **Next** to specify the service factors.
Finish	Click **Finish** to generate the Seismic combinations.

Code spectra and site specific spectra

Code Spectra

Different analysis procedures are provided for determining the elastic design response spectrum, the first of which is simply to adopt the code spectra.

- Default spectra (page 437)

Site Specific Spectra (user defined)

In addition to the code spectra, Tekla Structural Designer also allows you to define your own site specific spectra by specifying significant periods and acceleration. This procedure is applicable to both ELF and RSA analysis.

The curve equations used for the different head codes are as follows:

- ASCE Horizontal Design Spectrum (page 439)
- EN 1998-1 Horizontal Design Spectrum (Europe, UK, Singapore NA) (page 441)
- EN 1998-1 Horizontal Design Spectrum (Malaysia NA) (page 442)
- IS893 (Part 1) Horizontal Design Spectrum (page 444)
- ASCE7/UBC Horizontal Design Spectrum - Taiwan (page 446)

This procedure can be used for locations which use another country's loading and design codes where the code spectra are not relevant and so the local site spectra need to be defined.

Site Specific Spectra (user defined - generic curve)

A third route is to set a site specific generic curve, either manually, or by pasting data from a spreadsheet. This procedure requires an RSA to be run (not ELF). This has been added primarily for remote markets using the US codes It is available for the US Headcode with loading codes:

- ASCE7-05
- ASCE7-10
- ASCE7-16

It is not available for UBC, EC or IS codes as these codes do not define the process to be used with a generic spectrum.

User defined RSA curve

Default spectra

The following details the initial values of the parameters for spectra for each of the seismic head codes:

ASCE - Horizontal Design Spectrum

Limits	Equation	Segment	Ref
$0 \le T \le T_0$	S_{DS} x (0.4 + 0.6 x T / T_0) x (1/(R/I_e)	Start point/ linear	11.4.5 (1)
$T_0 \le T \le T_s$	S_{DS} x (0.4 + 0.6 x T / T0) x (1/(R/Ie)	Constant	11.4.5 (2)
$T_s \le T \le T_L$	(S_{D1}/T)/(R/I)	Constant/T	11.4.5 (3)
$T_L \le T \le 4s$	(S_{D1} x T_L / T^2)/(R/I)	Constant/T^2	11.4.5 (4)

UBC - Horizontal Design Spectrum

Limits	Equation	Segment
$0 \le T \le T_0$	C_a + 1.5 x T / T_0 x (2.5 / R-1)	Start point/linear
$T_0 \le T \le T_s$	2.5 x C_a /R	Constant
$T_s \le T$	C_v / T/R	Constant/T

EC - Horizontal Design Spectrum (UK, Singapore and Europe)

Limits	Equation	Segment	Ref
$0 \le T \le T_B$	a_g x S x [2/3 + T/T_B x (2.5 / q - 2/3)]	Start point/ linear	3.13
$T_B \le T \le T_C$	a_g x S x 2.5 / q	Constant	3.14
$T_C \le T \le T_D$	a_g x S x T_C/T x 2.5 / q	Constant/T	3.15
$T_D \le T \le 4s$	a_g x S x 2.5 / q x (T_C x T_D) / T^2	Constant/T^2	3.16

EC - Horizontal Design Spectrum (Malaysia)

For **Rock soil** sites

Limits	Equation	Segment
$0 \leq T \leq T_C$	$((2 \times \pi)^2 \times \gamma I \times S_{DR}(1.25) / (T_C \times T_D)) / q$	Constant
$T_C \leq T \leq T_D$	$((2 \times \pi)^2 \times \gamma I \times S_{DR}(1.25) / (T \times TD)) / q$	Constant/T
$T_D \leq T \leq 4s$	$((2 \times \pi)^2 / T2 \times [\lambda I \times S_{DR}(1.25) + \gamma I \times m_r \times (T - T_D))] / q$	Constant/T^2

For **Stiff soil** sites

Limits	Equation	Segment
$0 \leq T \leq T_C$	$((2 \times \pi)^2 \times \gamma I \times S_{DR}(1.25) \times 1.5 / (T_C \times T_D)) / q$	Constant
$T_C \leq T \leq T_D$	$((2 \times \pi)^2 \times \gamma I \times S_{DR}(1.25) \times 1.5 / (T \times TD)) / q$	Constant/T
$T_D \leq T \leq 4s$	$((2 \times \pi)^2 / T2 \times [\lambda I \times S_{DR}(1.25) \times 1.5 + \gamma I \times m_r \times (T - T_D))] / q$	Constant/T^2

For **Flexible soil** sites

Limits	Equation	Segment
$0 \leq T \leq T_C$	$((2 \times \pi)^2 \times \gamma I \times S_{DR} (1.5T_S) \times 3.6 / (T_C \times T_D)) / q$	Constant
$T_C \leq T \leq T_D$	$((2 \times \pi)^2 \times \gamma I \times S_{DR}(1.5T_S) \times 3.6 / (T \times TD)) / q$	Constant/T
$T_D \leq T \leq 4s$	$((2 \times \pi)^2 / T2 \times [\lambda I \times S_{DR}(1.5T_S) \times 3.6 + \gamma I \times m_r \times (T - T_D))] / q$	Constant/T^2

IS - Horizontal Design Spectrum

For **rocky, or hard soil** sites

Limits	Damping factor	Segment
$0 \leq T \leq 0.10$	(1 + 15 T)	Start point/linear
$0.10 \leq T \leq 0.40$	(2.50)	Constant/T
$0.40 \leq T \leq 4.00$	(1.00/T)	Constant/T^2

For **medium soil** sites

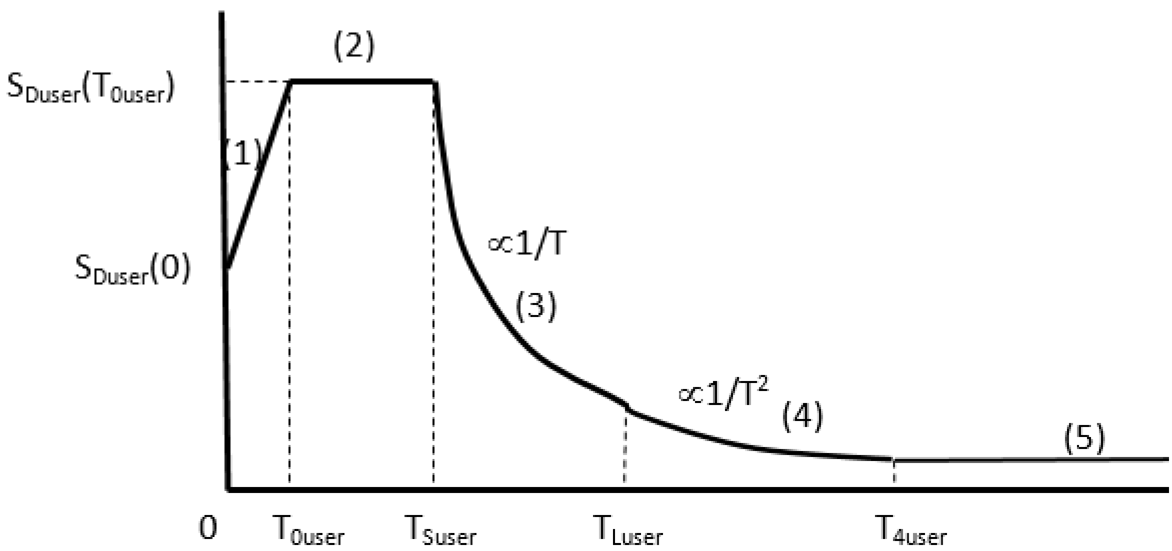
Limits	Damping factor	Segment
$0 \leq T \leq 0.10$	(1 + 15 T)	Start point/linear
$0.10 \leq T \leq 0.55$	(2.50)	Constant/T

Limits	Damping factor	Segment
0.55 ≤ T ≤ 4.00	(1.36/T)	Constant/T^2

For **soft soil** sites

Limits	Damping factor	Segment
0 ≤ T ≤ 0.10	(1 + 15 T)	Start point/linear
0.10 ≤ T ≤ 0.67	(2.50)	Constant/T
0.67 ≤ T ≤ 4.00	(1.67/T)	Constant/T^2

ASCE Horizontal Design Spectrum



NOTE ASCE7 has either sloped second segment or horizontal 4th segment - not both.

Parameters

- $S_{Duser}(0)$ – units g
- $S_{Duser}(T_{0user})$ – units g
- T_{0user} – units sec
- T_{Suser} – units sec
- T_{Luser} – units sec
- T_{4user} – units sec, default = 4s

Input limits

- $S_{Duser}(0) > 0$
- $S_{Duser}(T_{0user}) > 0$ if $T_{0user} > 0$
- $0s \leq T_{0user} < T_{Suser} < T_{Luser} \leq T_{4user}$

Curve Equations

Design Response Spectrum curves for $S_a(g)$ are defined by

NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{0user}$	$S_a(g)(T) = S_{Duser}(0) + ((S_{Duser}(T_{0user}) – S_{Duser}(0)) \times T / T_{0user})$
(2) – straight line	$T_{0user} \leq T \leq T_{Suser}$	$S_a(g)(T) = S_{Duser}(T_{0user})$
(3) – curve	$T_{Suser} \leq T \leq T_{Luser}$	$S_a(g)(T) = S_{Duser}(T_{0user}) \times T_{Suser}/ T$
(4) – curve	$T_{Luser} \leq T \leq T_{4user}$	$S_a(g)(T) = S_{Duser} (T_{0user}) \times T_{Suser} \times T_{Luser}/ T^2$
(5) – straight line continued from (4)		

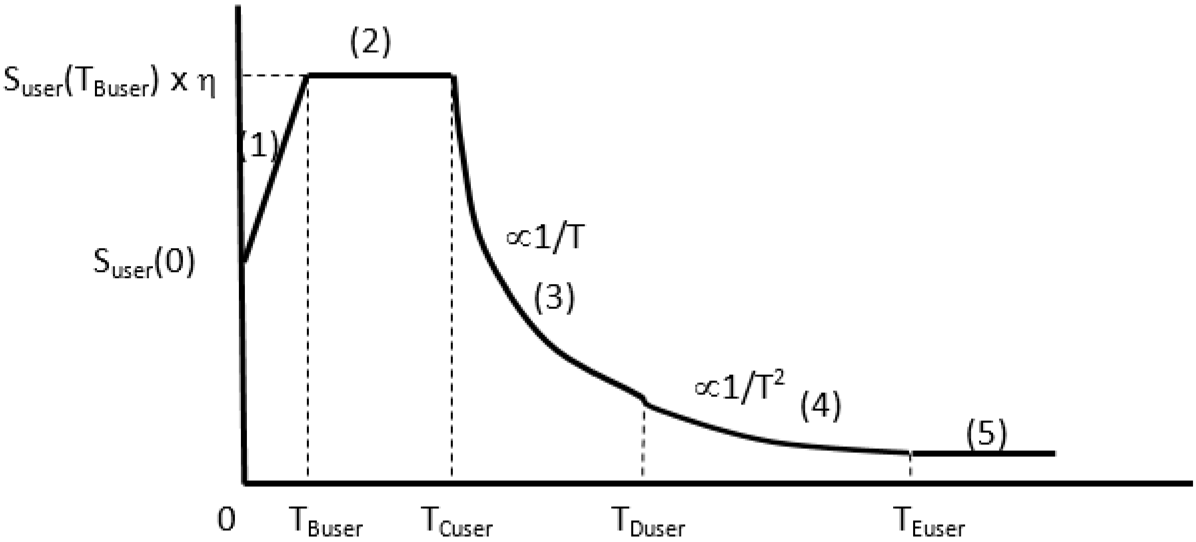
Adjusted Design Response Spectrum curves for $S_a(g)/(R/I_e)$ are defined by

NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{0user}$	$S_D(T) = [S_{Duser}(0) + ((S_{Duser}(T_{0user}) - S_{Duser}(0)) \times T / T_{0user})] /(R/I_e)$
(2) – straight line	$T_{0user} \leq T \leq T_{Suser}$	$S_D(T) = S_{Duser}(T_{0user}) / (R/I_e)$
(3) – curve	$T_{Suser} \leq T \leq T_{Luser}$	$S_D(T) = S_{Duser}(T_{0user}) \times T_{Suser} / T / (R/I_e)$
(4) – curve	$T_{Luser} \leq T \leq T_{4user}$	$S_D(T) = S_{Duser}(T_{0user}) \times T_{Suser} \times T_{Luser} / T^2 / (R/I_e)$
(5) – straight line continued from (4)		

NOTE	If T_{0user} = 0, then no line (1) exists If T_{Luser} = T_{4user} then no line (4) exists

EN 1998-1 Horizontal Design Spectrum (Europe, UK, Singapore NA)



Parameters

- $S_{user}(0)$ – units g
- $S_{user}(T_{Buser})$ – units g
- T_{Buser} – units sec
- T_{Cuser} – units sec
- T_{Duser} – units sec
- T_{Euser} – units sec, default = 4s

Input limits

- $S_{user}(0) > 0$
- $S_{user}(T_{Buser}) > 0$
- $0s \le T_{Buser} < T_{Cuser} < T_{Duser} \le T_{Euser}$

Curve Equations

Elastic Response Spectrum curves for $S_e/a_g$are defined by

NOTE	same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{Buser}$	$S_e/a_g(T) = S_{user}(0) + ((S_{user}(T_{Buser}) \times \eta - S_{user}(0)) \times T / T_{Buser})$
(2) – straight line	$T_{Buser} \leq T \leq T_{Cuser}$	$S_e/a_g(T) = S_{user}(T_{Buser}) \times \eta$
(3) – curve	$T_{Cuser} \leq T \leq T_{Duser}$	$S_e/a_g(T) = S_{user}(T_{Buser}) \times \eta \times T_{Cuser}/ T$
(4) – curve	$T_{Duser} \leq T \leq T_{Euser}$	$S_e/a_g(T) = S_{user} (T_{Buser}) \times \eta \times T_{Cuser} \times T_{Duser}/ T^2$
(5) – straight line continued from (4)		

Design Spectrum for Elastic Analysis curves for S_e/a_g are defined by

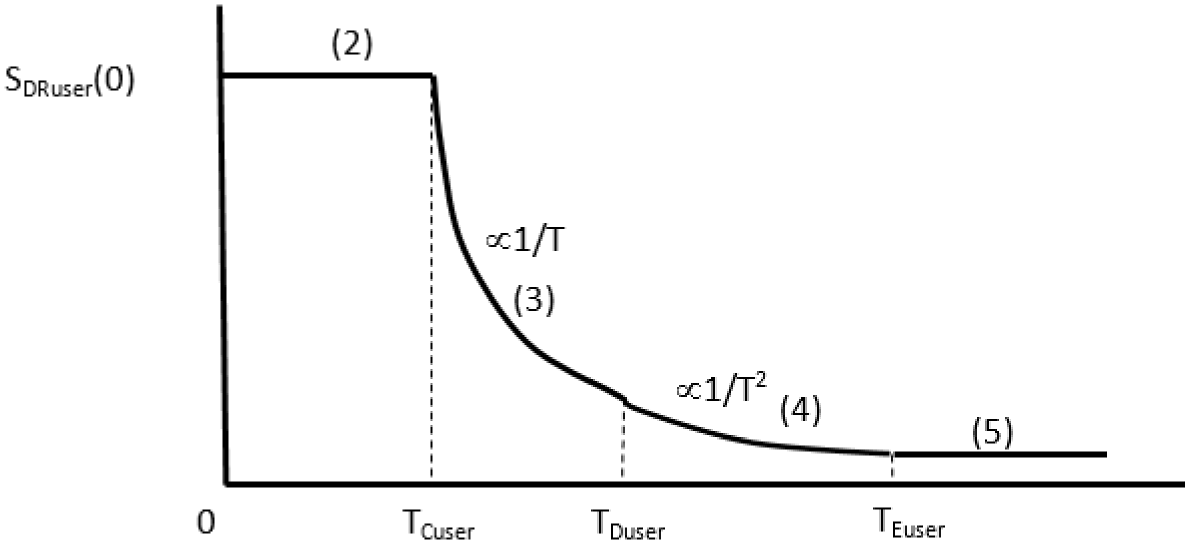
NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{Buser}$	$S_d/a_g(T) = [\ 2/3 \times S_{user}(0) + ((S_{user}(T_{Buser}) / q - 2/3 \times S_{user}(0)) \times T / T_{Buser})]$
(2) – straight line	$T_{Buser} \leq T \leq T_{Cuser}$	$S_d/a_g(T) = S_{user}(T_{Buser}) / q$
(3) – curve	$T_{Cuser} \leq T \leq T_{Duser}$	$S_d/a_g(T) = S_{user}(T_{Buser}) \times T_{Cuser}/ T / q$
(4) – curve	$T_{Duser} \leq T \leq T_{Euser}$	$S_d/a_g(T) = S_{user} (T_{Buser}) \times T_{Cuser} \times T_{Duser}/ T^2/ q$
(5) – straight line continued from (4)		

NOTE If $T_{Buser} = 0$, then no line (1) exists

If $T_{Duser} = T_{Euser}$ then no line (4) exists

EN 1998-1 Horizontal Design Spectrum (Malaysia NA)



Parameters

- $S_{DRuser}(0)$ – units g
- $S_{DRuser}(T_{Buser})$ – units g
- T_{Buser} – units sec
- T_{Cuser} – units sec
- T_{Duser} – units sec
- T_{Euser} – units sec, default = 4s

Input limits

- $S_{DRuser}(0) > 0$
- $S_{DRuser}(T_{Buser}) > 0$
- $0s \le T_{Buser} < T_{Cuser} < T_{Duser} \le T_{Euser}$

Curve Equations

Elastic Response Spectrum curves for $S_e/a_g$are defined by

NOTE	same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{Buser}$	$S_e/a_g(T) = S_{DRuser}(0) + ((S_{DRuser}(T_{Buser}) - S_{DRuser}(0)) \times T / T_{Buser})$

Line	Limits	Equation
(2) – straight line	$T_{Buser} \leq T \leq T_{Cuser}$	$S_e/a_g(T) = S_{DRuser}(T_{Buser})$
(3) – curve	$T_{Cuser} \leq T \leq T_{Duser}$	$S_e/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} / T$
(4) – curve	$T_{Duser} \leq T \leq T_{Euser}$	$S_e/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} \times T_{Duser} / T^2$
(5) – straight line continued from (4)		

Design Spectrum for Elastic Analysis curves for S_d/a_g are defined by

NOTE same curve for both Dir1 and Dir2

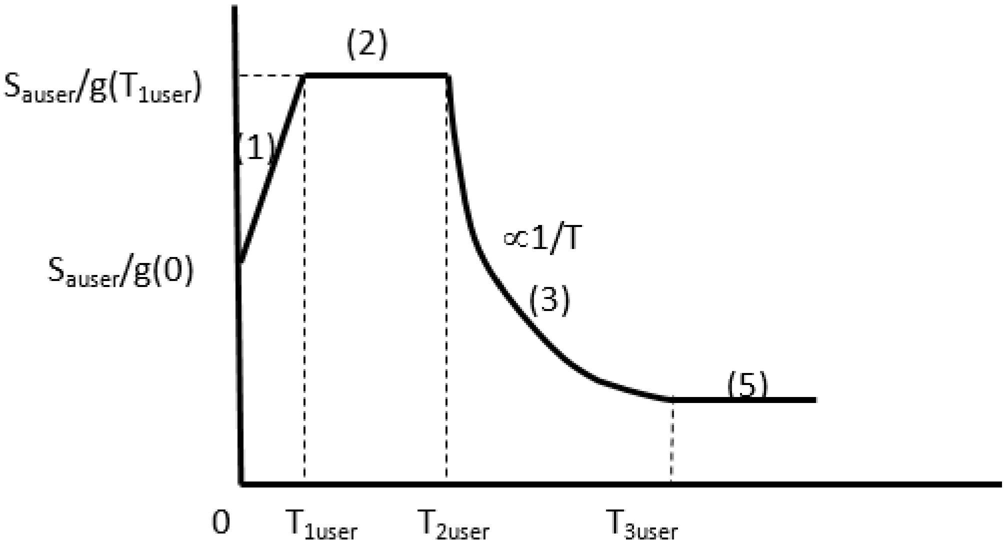
Line	Limits	Equation
(1) – straight line	$0 \leq T \leq T_{Buser}$	$S_d/a_g(T) = [S_{DRuser}(0) + ((S_{DRuser}(T_{Buser}) - S_{DRuser}(0)) \times T / T_{Buser})] / q$
(2) – straight line	$T_{Buser} \leq T \leq T_{Cuser}$	$S_d/a_g(T) = S_{DRuser}(T_{Buser}) / q$
(3) – curve	$T_{Cuser} \leq T \leq T_{Duser}$	$S_d/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} / T / q$
(4) – curve	$T_{Duser} \leq T \leq T_{Euser}$	$S_d/a_g(T) = S_{DRuser}(T_{Buser}) \times T_{Cuser} \times T_{Duser} / T^2 / q$
(5) – straight line continued from (4)		

NOTE If $T_{Buser} = 0$, then no line (1) exists(as in the example above)

If $T_{Duser} = T_{Euser}$ then no line (4) exists

NOTE The above by default does not quite equate to Line (4) in the Malaysian NA because there is an extra term $(2 \times \pi)^2 / T^2 \times \gamma I \times m_r \times (T - T_D)$ that tweaks the $1/T^2$ curve slightly. In Sarawak, this term is non-existent, in Peninsular Malaysia it does not exist for Flexible soils. In the Tekla Structural Designer User defined spectrum it is also non-existent.

IS893 (Part 1) Horizontal Design Spectrum



NOTE	In this example T3user = T4user so there is no line (4)

Parameters

- Damping $factor_{user}$
- $S_{auser}/g(0)$ – units g
- $S_{auser}/g(T_{1user})$ – units g
- T_{1user} – units sec
- T_{2user} – units sec
- T_{3user} – units sec, default = 4s
- T_{4user} – units sec, default = 4s

Input limits

- $S_{auser}/g(0) > 0$
- $S_{auser}/g(T_{1user}) > 0$
- $0s \leq T_{1user} < T_{2user} < T_{3user} \leq T_{4user}$

Curve Equations

Average Response Spectrum curves for S_a/g are defined by

NOTE	same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{1user}$	$S_a(g)(T)$ = Damping factor$_{user}$ x $[S_{auser}/g(0) + ((S_{auser}/g(T_{1user}) - S_{auser}/g(0)) \times T / T_{1user})]$
(2) – straight line	$T_{1user} \le T \le T_{2user}$	$S_a(g)(T)$ = Damping factor$_{user}$ x $S_{auser}/g(T_{1user})$
(3) – curve	$T_{2user} \le T \le T_{3user}$	$S_a(g)(T)$ = Damping factor$_{user}$ x $S_{auser}/g(T_{1user}) \times T_{2user}/ T$
(4) – curve	$T_{3user} \le T \le T_{4user}$	$S_a(g)(T)$ = Damping factor$_{user}$ x $S_{auser}/g(T_{1user}) \times T_{2user} \times T_{3user}/ T^2$
(5) – straight line continued from (4)		

Design Response Spectrum curves for $A_h(g)$ ($=S_a/g/((2xR)/(ZxI))$) are defined by

NOTE same curve for both Dir1 and Dir2

Line	Limits	Equation
(1) – straight line	$0 \le T \le T_{1user}$	$A_h(g)(T)$ = Damping factor$_{user}$ x $[S_{auser}/g(0) + ((S_{auser}/g(T_0) - S_{auser}/g(0)) \times T / T_{1user})]$ But not less than Z/2
(2) – straight line	$T_{1user} \le T \le T_{2user}$	$A_h(g)(T)$ = Damping factor$_{user}$ x $S_{auser}/g(T_{1user}) / ((2xR)/(ZxI))$
(3) – curve	$T_{2user} \le T \le T_{3user}$	$A_h(g)(T)$ = Damping factor$_{user}$ x $S_{auser}/g(T_{1user}) \times T_{2user}/ T / ((2xR)/(ZxI))$
(4) – curve	$T_{3user} \le T \le T_{4user}$	$A_h(g)(T)$ = Damping factor$_{user}$ x $S_{auser}/g(T_{1user}) \times T_{2user} \times T_{3user}/ T^2 / ((2xR)/(ZxI))$
(5) – straight line continued from (4)		

NOTE If $T_{1user} = 0$, then no line (1) exists

If $T_{3user} = T_{4user}$ then no line (4) exists

ASCE7/UBC Horizontal Design Spectrum - Taiwan

Taiwan code Figure 2

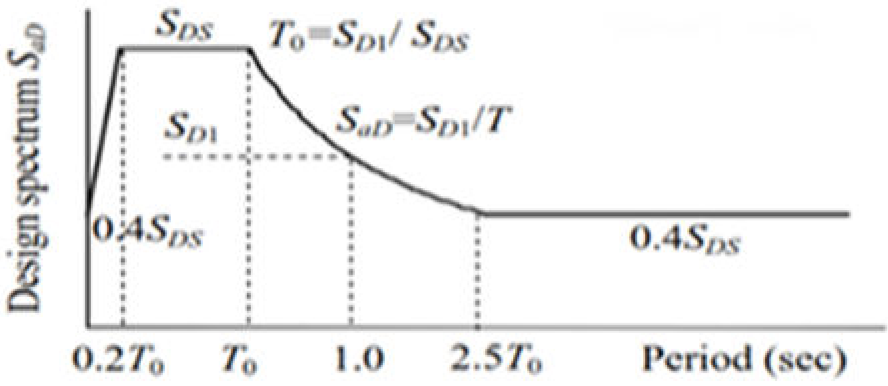


Figure 2. Design response spectrum developed by site-adjusted parameters S_{DS} and S_{D1}

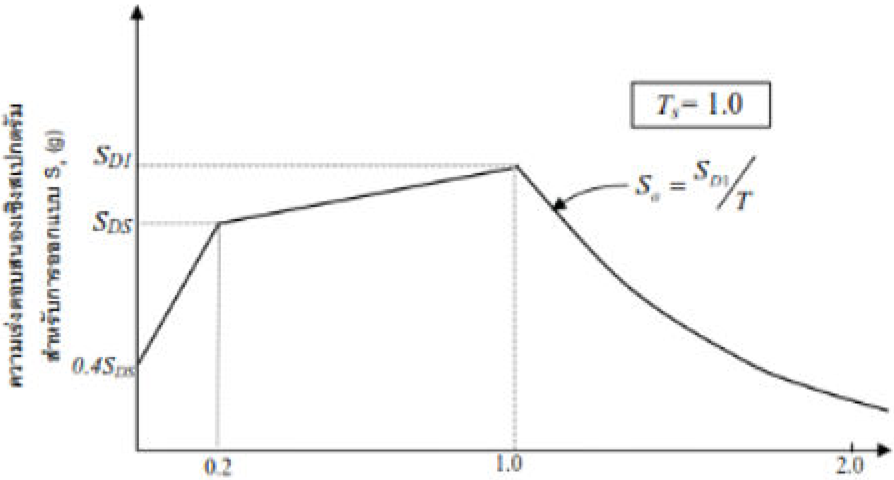
Specifically, the last section of curve (Segment 4) extends horizontally from $2.5T_0$ with a value of 0.4 x S_{DS}

NOTE The above curve can be obtained by selecting Horizontal 4th Segment on the Site Specific Spectra page of the Seismic Wizard.

ASCE7/UBC Horizontal Design Spectrum - Thailand

When $S_{d1} > S_{ds}$

Thailand code Figure 1-4-3



Specifically, the previously flat section of curve (Segment 2) rises from S_{DS} at T = 0.2 to S_{D1} At T = 1.0

NOTE The above curve can be obtained by selecting Sloped 2nd Segment on the Site Specific Spectra page of the Seismic Wizard.

Seismic loadcases

Once the Seismic Wizard has been run the loadcases required to be applied to the structure can be generated.

Equivalent Lateral Force (ELF) Loadcases

If an Equivalent Lateral Force (ELF) analysis procedure was selected in the Wizard, the load is applied to the structure at each level as determined for the relevant direction - loadcases being created as follows:

For structures with no torsion

- Seismic Dir1 - loadcase type Seismic
- Seismic Dir2 - loadcase type Seismic

For all other structures:

- Seismic Dir1 - loadcase type Seismic
- Seismic Dir2 - loadcase type Seismic
- Seismic Torsion - loadcase type Seismic

NOTE + and - factors are applied in the combinations to achieve the possible options - +Dir, -Dir, +torsion and -torsion.

Response Spectrum Analysis (RSA) Loadcases

If a response spectrum analysis procedure was selected in the Wizard, the loadcases are created as follows:

For structures with no torsion

- Seismic Dir1 - loadcase type Seismic RSA
- Seismic Dir2 - loadcase type Seismic RSA

For all other structures:

- Seismic Dir1 - loadcase type Seismic RSA
- Seismic Dir2 - loadcase type Seismic RSA
- Seismic Torsion - loadcase type Seismic RSA

NOTE Enveloping is used in the combinations to achieve the possible options - +Dir, -Dir, +torsion and -torsion.

---

6 Analyze models

With Tekla Structural Designer you can perform an extensive range of different analyses.

- Get started with analysis (page 450)
- To run a specific analysis for selected loadcases/combinations, or to run all the analyses necessary for design, see Run analyses (page 491)
- To review the results once the analyses have completed, see Display analysis results (page 498)
- To review the solver model used for a particular analysis type, see View and manage solver models (page 551)

6.1 Get started with analysis

This section explains some basic concepts and procedures you need to know to get started with analysis in Tekla Structural Designer.

Click the links below to find out more:

- Analysis types in
- Adjust and apply analysis settings (page 460)
- What is a solver model (page 460)
- FE meshing, sub models and diaphragms (page 463)

Analysis types in Tekla Structural Designer

The following analysis types can be run in Tekla Structural Designer.

1st order linear

1st order linear static analysis is suitable for structures where secondary effects are negligible. Any nonlinear springs or nonlinear elements present are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: No
- Material: No

Related task: Run 1st order linear analysis (page 491)

1st order non-linear

This is a nonlinear analysis with loading applied in a single step.

It is suitable for structures where secondary effects are negligible and nonlinear springs/elements are present.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: No
- Material: Yes

Related task: Run a 1st order non-linear analysis (page 492)

1st order modal

This is an unstressed modal analysis which can be used to determine the structure's natural frequencies.

The structure is assumed to be in an unstressed state and nonlinear elements are constrained to act linearly.

Nonlinearity Included:

- Geometric: No
- Material: No

Related task: Run a 1st order modal analysis (page 492)

2nd order linear

This is a 2-stage P-Delta analysis which is suitable for structures where secondary effects are of comparable magnitude to primary effects. Any nonlinear springs or nonlinear elements present are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: Yes
- Material: No

Related task: Run a 2nd order linear analysis (page 493)

2nd order non-linear

This is a nonlinear analysis with loading applied in a single step.

It is suitable for structures where secondary effects are of comparable magnitude to primary effects and nonlinear springs/elements are present.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: Yes
- Material: Yes

Related task: Run a 2nd order non-linear analysis (page 493)

2nd order buckling

This is a linear buckling analysis which can be used to determine a structure's susceptibility to buckling.

The stressed state of the structure is determined from linear analysis; therefore nonlinear elements are constrained to act linearly.

Loadcases and Combinations to be considered in the analysis can be pre-selected.

Nonlinearity Included:

- Geometric: Yes
- Material: No

Related task: Run a 2nd order buckling analysis (page 493)

FE chasedown

This analysis type cannot be run in isolation, it is only performed when it is required as part of another process, such as **Analyze All (Static)**, or **Design All**.

Separate analyses are performed for a series of 3D sub models each containing the members between two horizontal planes with fully meshed floors. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below. The results are always used for slab design and optionally used for beam, column and wall design.

Grillage chasedown

This analysis type cannot be run in isolation, it is only performed when it is required as part of another process, such as **Analyze All (Static)**, or **Design All**.

Separate analyses are performed for a series of 3D sub models each containing the members between two horizontal planes, floors only being meshed at those levels where they have also been meshed in 3D Analysis. The complete series of models is chased down from top to bottom so loads are carried from the level above to the level below.

Analyze All (Static)

This is a full series of analyses that would be carried out as part of **Design All (Static)** but with no design. All the analyses required to enable a design to be performed are included:

- 3D analysis
    - 1st order linear/non-linear
    - 2nd order linear/non-linear (only if this has been specified by the user in **Design > Design Settings > Analysis**)
- FE chasedown analysis (if required)
- Grillage chasedown analysis (if required)

Related task: Run Analyze All (Static) (page 496)

3D only (Static)

This is a similar to **Analyze All (Static)** but excludes chasedowns to save time during scheme design, (for example while addressing overall stability, sway, drift, wind drift, etc.)

Analyses performed:

- 3D analysis
    - 1st order linear/non-linear

- 2nd order linear/non-linear (only if this has been specified by the user in **Design > Design Settings > Analysis**)

Related task: Run 3D only (Static) (page 496)

1st order RSA seismic

This is a Modal Response Spectrum Analysis used to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 1st Order Linear Analysis

Related task: Run a 1st order RSA seismic analysis (page 494)

2nd order RSA seismic

This is a Modal Response Spectrum Analysis used to determine the peak response of the structure to earthquakes. Any nonlinear springs or nonlinear elements present are constrained to act linearly. For RSA Seismic Combinations, the peak responses are enveloped around the static results for 2nd Order Linear Analysis

Related task: Run a 2nd order RSA seismic analysis (page 494)

Analysis limitations and assumptions

Certain specific limitations and assumptions relating to the various analysis types are expanded upon below:

Linear analysis of structures containing material nonlinearity

If a structure containing nonlinear springs or nonlinear elements is subjected to a linear (i.e. 1st or 2nd order linear, 1st order modal, or 2nd order buckling) analysis, then the nonlinear springs/elements are constrained to act linearly as described below:

Nonlinear spring supports

In each direction in which a nonlinear spring has been specified, a single value of stiffness is applied which is taken as the greater of the specified -ve or +ve stiffness.

Any specified maximum capacities of the spring are ignored.

Tension only, or compression only elements

If either tension only or compression only element types have been specified, they are constrained to act as truss element types instead.

Nonlinear axial spring, or nonlinear torsional spring elements

If either of these element types have been specified, they are constrained to act as linear axial spring, or linear torsional spring element types instead.

A single value of stiffness is applied which is taken as the greater of the specified -ve or +ve stiffness.

Any specified maximum capacities of these spring elements are ignored.

Tension only X braces

It is essential that the "X Brace" pattern is used to input cross braces as brace pairs rather than creating them individually. For linear analysis one of the braces in a brace pair is automatically inactivated, ensuring that the model's lateral stiffness, and hence lateral drift, is reasonably correct. If braces are input individually this will not be the case.

To determine which brace in each pair is inactivated the program pushes the structure simultaneously in the positive direction 1 and positive direction 2. The brace that goes into tension retains its full stiffness, while the compression brace becomes inactive.

If the above process fails to determine which of the pair goes into tension and which is inactivated then a shear is applied to the structure and the braces are re-assessed.

Analysis of structures containing geometric nonlinearity

It is assumed that where secondary effects are significant (for example the structure is close to buckling), the engineer will elect to undertake a 2nd order analysis. If a 1st order analysis is performed any secondary effects will be ignored.

Analysis of structures containing curved beams

The member analysis for curved members in the plane of the curve is approximated by joining the values at the nodes, which are correct. For detailed analysis of curved members it is your responsibility to ensure sufficient discretization. More refined models can be achieved, if required, by decreasing the maximum facet error.

Analysis of compound (plated) steel beams and columns

Compound (plated) sections are 2 chords or more connected by battens, lattice or welded.

- Static calculations for these section types are the same as for solid sections. The section characteristics are calculated in the basis of the actual section.
- The torsional constant of a compound section is calculated as a total of torsional constants of the chords in a compound section.
- Plane section remain plane.
- The material is homogeneous, isotropic and linearly elastic.
- Saint Venant's principle applies.

Story shears

The story shears that are output are obtained by resolving the loads at column nodes horizontally into Direction 1 and Direction 2. Any loads associated with V & A braces are not included because these occur at mid-beam position and not at column nodes.

Member Deflections

There is a known issue when calculating member deflection profiles in combinations which can affect the following analysis types:

- 2nd Order Linear
- 1st Order Nonlinear
- 2nd Order Nonlinear

This occurs when the structures behavior is significantly nonlinear because the deflection profile is currently based on linear superposition of the loadcases within it. Clearly as structural response becomes more nonlinear the assumption that deflections can be superposed becomes less valid. This can cause a deflected profile to be calculated which deviates from the correct profile. The deviation can become significant if loadcases fail to solve, but the combination succeeds in solving, as components of the deflected shape are missing entirely. It is suggested that for the three analysis types listed member deflections in combinations be used with caution and engineering judgment.

It should be noted that this limitation only affects member deflection profiles between solver nodes. All other results, including member force profiles and deflection at the solver nodes are correct.

Torsion load analysis - relative angle of twist

Any section that is subject to torsional moment will rotate through an angle, θ. If the cross-section is non-circular this will also be accompanied by warping.

To be able to determine stresses on members subject to torsional moments it is necessary to determine θ (and for "Open" sections its derivatives also).

For single span pinned steel beams only: a torsion load analysis is performed which enables θ (and its derivatives) to be calculated and made available in the Load Analysis View.

For open sections: a more accurate approach is used to determine θ and its derivatives. The 'cases' in DG9, SCI P057 & SCI P385 being used which depend on the end conditions and loading conditions on the beam.

This more accurate analysis is carried out for the following open sections:

i. I Symmetric

ii. I Asymmetric

iii. I Plated (including Westok Plated)

iv. Channel

v. Westok cellular beyond scope

For all other sections not mentioned above (including compound sections): a "Standard" analysis is carried out to determine θ only using the following equation:

$1 / GI_T * \int T(x)$

Where:

I_T = torsion constant

G = shear modulus of steel

T(x) = function of torsion moment

- 2nd Order Linear
- 1st Order Nonlinear
- 2nd Order Nonlinear

Modal analysis - active mass

Modal analysis - active mass

In a 1st order modal analysis mass is assigned to nodes of the analysis model. In simple terms (neglecting rotation terms for the consistent mass matrix) half of each element mass is assigned to each node it is attached to.

Mass that is assigned to a translational support cannot go anywhere - i.e. it is not "active".

Summed Active Mass

Reported in the Dynamic Masses table, this is the actual total active mass for each direction, but expressed in terms of force units rather than mass.

Summed Total Translational Mass

Reported in the **Dynamic Masses** table, this is the total system mass for each direction, again expressed in terms of force units rather than mass.

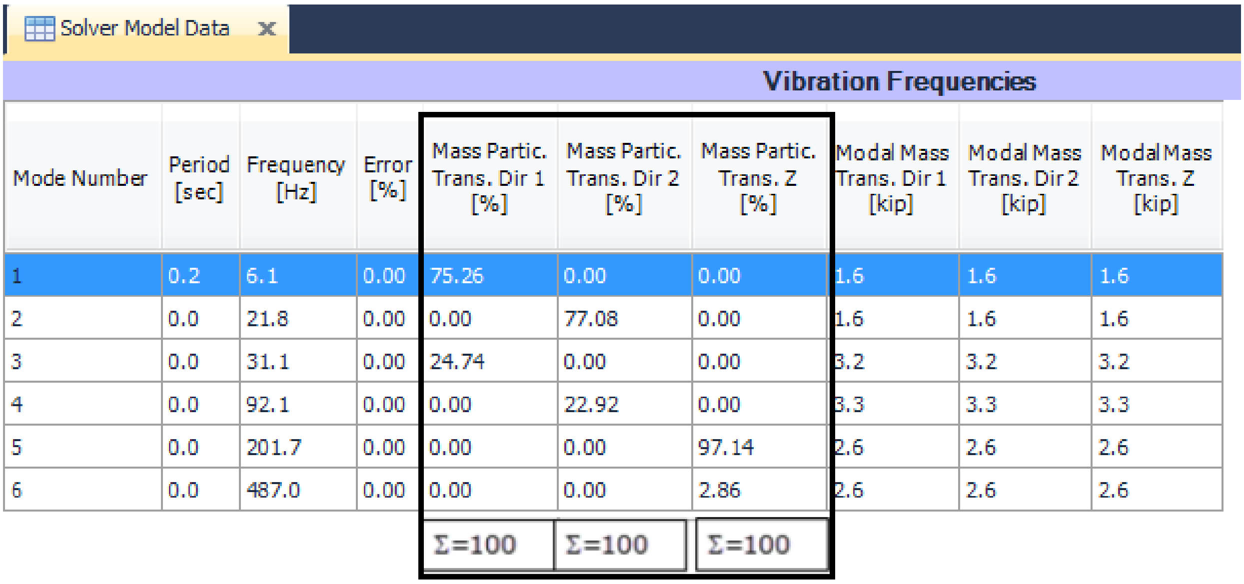
Translation %

Reported in the **Summed Mass** table, this is the proportion of mass that is active for each direction. For a building this will usually be close to but not quite 100% as some mass always goes to the supports.

Translation % = (Summed Active Mass / Summed Total Translational Mass) x 100

Participation Translation %

Reported in the **Summed Mass** table, this is the sum of Mass Participation (reported in the Modal frequencies table) for all modes for each direction. Design codes stipulate this should be >= 90% for seismic analysis usually for two orthogonal lateral directions.



Solver Model Data

Vibration Frequencies

Mode Number	Period [sec]	Frequency [Hz]	Error [%]	Mass Partic. Trans. Dir 1 [%]	Mass Partic. Trans. Dir 2 [%]	Mass Partic. Trans. Z [%]	Modal Mass Trans. Dir 1 [kip]	Modal Mass Trans. Dir 2 [kip]	Modal Mass Trans. Z [kip]
1	0.2	6.1	0.00	75.26	0.00	0.00	1.6	1.6	1.6
2	0.0	21.8	0.00	0.00	77.08	0.00	1.6	1.6	1.6
3	0.0	31.1	0.00	24.74	0.00	0.00	3.2	3.2	3.2
4	0.0	92.1	0.00	0.00	22.92	0.00	3.3	3.3	3.3
5	0.0	201.7	0.00	0.00	0.00	97.14	2.6	2.6	2.6
6	0.0	487.0	0.00	0.00	0.00	2.86	2.6	2.6	2.6
				Σ=100	Σ=100	Σ=100			

Modal analysis - modal mass

After running a 1st order modal analysis, modal masses for each mode are available in the Modal Frequencies tabular display.

Modal Mass

In Tekla Structural Designer the modal mass, M_i is given by the following matrix equation:

$$M_i = \{\psi\}_i^T [M] \{\psi\}_i$$

Where $\{\Psi\}_i$ is the unity-scaled mode shape (often termed mode vector) of the i^{th} mode (i.e. any single mode) and [M] is the mass matrix. The meaning of the unity-scaled mode shape is that the (numerically) largest modal displacement is set to unity and all other displacements are scaled accordingly. The term $\{\Psi\}_i$ is used to differentiate this mode shape from the mass-normalized shape $\{\Phi\}_i$ which is the mode shape actually reported by Tekla Structural Designer .

This equation comes from modal analysis theory. It may also be termed "generalized mass".

Another way to state this equation, which is found in some design guides, is a summation equation for point masses and their associated modal displacements for a system of discretized mass distribution:

We have from CCIP-016:

$$\hat{m}_j = \sum_{i=1}^{N} \mu^2_{j,i}\, m_i$$

where i is each on N points on the structure, having mass m_i at which the mode shape $\mu_{j,i}$ is the j^{th} mode is known.

According to this reference - "Conceptually, the modal mass can be thought of as the mass of an equivalent single degree of freedom system... which represents the j^{th} mode."

It can be seen how this is equivalent to the matrix equation given above. Actually Tekla Structural Designer makes use of a shortcut calculation since it already has mode shapes which are normalized to mass.

The mass-normalized mode shape $\{\Phi\}_i$ and the unity-normalized mode shape $\{\Psi\}_i$ are related as follows:

$$\{\Phi\}_i = \frac{1}{\sqrt{M_i}} \{\psi\}_i$$

From this we can state the following, where Φ_2 is the largest modal displacement from the mass-normalized mode shape (which we already have from the Tekla Structural Designer modal analysis):

$$\Phi_2 = \frac{1}{\sqrt{M_i}} \times 1$$

Hence:

$$\text{Modal mass} = M_i = \frac{1}{\Phi_2^2}$$

Unstable Structures

Flat Slab Structures

If a concrete structure exists with only flat slabs and columns (i.e. no beams and no shear walls), and the slab is modelled with a diaphragm this is an unstable structure, assuming that the concrete columns are pinned at the foundation level (current default).

To prevent the instability you should mesh the slabs, as the resulting model does then consider the framing action that results from the interaction of the slabs and columns.

Adjust and apply analysis settings

To adjust the analysis settings of the current project or set them as default settings for future projects, see the following instructions.

Adjust analysis settings in the current project

1. On the **Analyze** toolbar, click **Settings**

   The **Model Settings** dialog box opens.
2. Review and modify the analysis settings according to your needs.
3. Do one of the following:
   - To apply the changes to the current project, click **OK**.
   - To save the changes to the selected settings set, click **Save...**
   - To revert to the analysis settings specified in the selected settings set, click **Load...**

Adjust analysis settings in future projects

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to the **Analysis Settings** page.
3. In the list at the top of the page, select the settings set that you want to modify.
4. Review and modify the analysis settings according to your needs.
5. To save the settings as defaults for future projects that use the selected settings set, click **OK**.

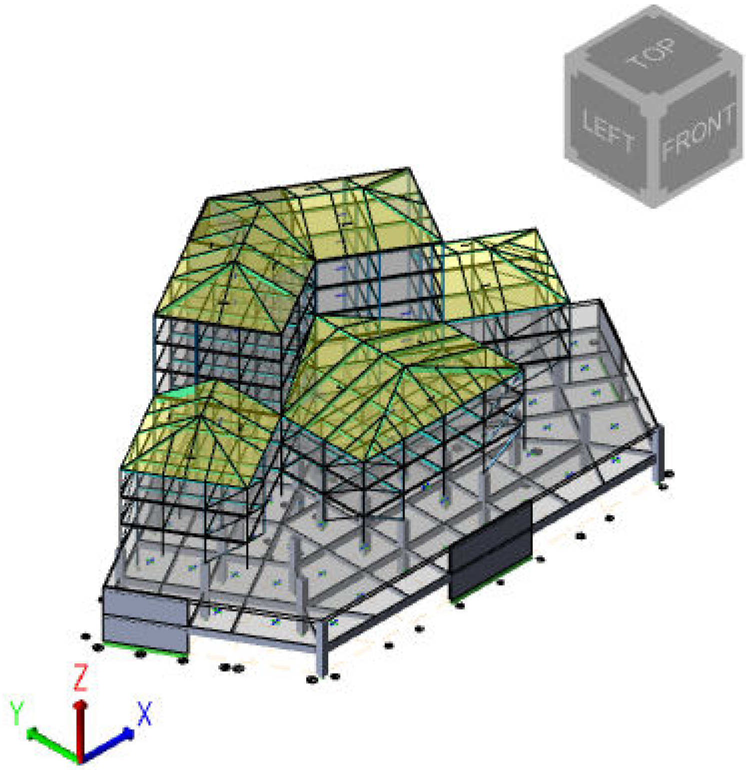
See also

Analysis Settings (page 1050)

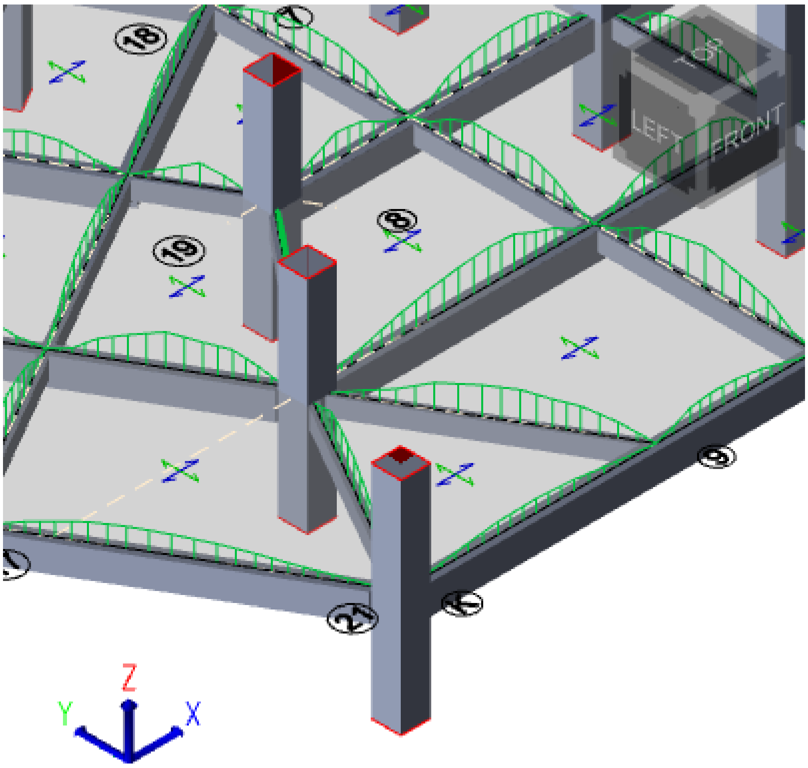
What is a solver model

When you use Tekla Structural Designer to model, analyze, and design structures, you will become familiar with the following concepts:

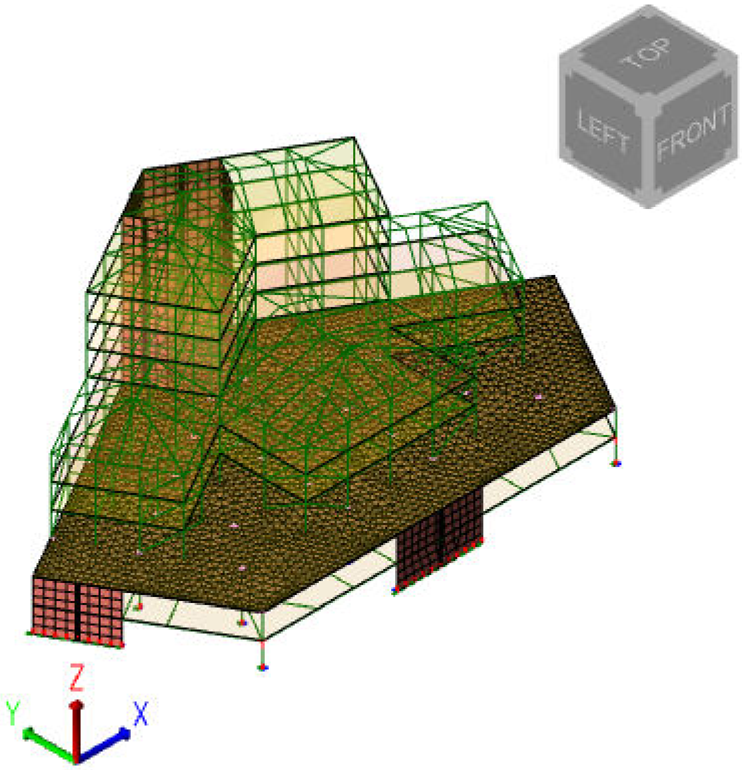
A *structural model* is a 3D model that includes the physical members/objects you create using Tekla Structural Designer, and information related to them. Each member/object in the physical model will exist in the completed structure.



The structural model also contains information about the loadcases and combinations that act on the physical members/objects.



A *solver model* is a 3D model that is created from the structural model. It is used for analyzing structural behavior, and for design.



When you create a solver model, Tekla Structural Designer generates the following analysis objects and includes them in the solver model:

- Solver nodes
- Support conditions for nodes
- 1D solver elements
- 2D solver elements

FE meshing, sub models and diaphragms

Click the links below to find out more about FE meshing, sub models and diaphragms.

To	Click the link below:
Choose whether to mesh slabs for 3D analysis; specify slab mesh	Manage FE meshed slabs (page 464)
Specify wall mesh	Manage FE meshed walls (page 480)
Manage the sub models used in chasedown analyses	Manage sub models (page 488)

To	Click the link below:
Activate and manage rigid and/or semi-rigid diaphragms	Diaphragm action in roof panels and slabs (page 482)

Manage FE meshed slabs

At the levels where two-way spanning slabs exist, Tekla Structural Designer applies FE meshing as follows:

- Two-way spanning slabs are **always** meshed in the FE chasedown analysis that occurs as part of the static design process.
- Two-way slabs are by default **not** meshed for 3D building analysis or grillage chasedown analysis. This ensures the quickest solution time. A 3D pre-analysis process (in which the slabs are meshed) is used to decompose loads from the slabs on to supporting members.
- Alternatively, you can choose to mesh two-way slabs for 3D building analysis and grillage chasedown analysis, either at all, or selected construction levels. In this case Tekla Structural Designer does not perform the load decomposition during 3D pre-analysis for the two-way slabs at those levels.

In all of the above mentioned situations, the slab mesh density is set globally, but can be overridden at individual levels if required.

Click the links below to find out more:

- Define whether slabs are meshed for 3D building analysis and grillage chasedown analysis (page 464)
- Adjust global slab mesh properties (page 465)
- Apply different mesh properties at different levels (page 466)
- Review the slab mesh before the analysis (page 466)
- How slab properties and features impact on meshing (page 467)
- Slab meshing controls (page 471)

Define whether slabs are meshed for 3D building analysis and grillage chasedown analysis

To ensure the quickest solution time, two-way slabs are by default **not** meshed for 3D building analysis or grillage chasedown analysis. In this case, the diaphragm option (none, semi-rigid, or rigid) in the slab properties determines their axial rigidity and a 3D pre-analysis process is used to FE decompose loads from the slabs on to supporting members.

If you prefer, you can instead choose to mesh the slabs, either at all, or selected construction levels.

NOTE Two-way spanning slabs are **always** meshed in the FE chasedown analysis that occurs as part of the static design process.

Use meshed two-way slabs in at all construction levels

1. Go to the **Project Workspace**.
2. In the **Structure** tree, select **Levels**.
3. In the **Properties** window, select the **Mesh 2-way Slabs in 3D Analysis** option.

Use FE decomposed slab loads at all construction levels

1. Go to the **Project Workspace**.
2. In the **Structure** tree, select **Levels**.
3. In the **Properties** window, clear the **Mesh 2-way Slabs in 3D Analysis** option.

Use meshed two-way slabs at selected construction levels

1. In the **Structure** tree, expand the **Levels** branch.
2. Click one of the desired construction levels.
3. According to your needs, either select or clear the **Mesh 2-way Slabs in 3D Analysis** option.
4. Repeat step 3 for all the desired construction levels.

Adjust global slab mesh properties

Initially, the same mesh properties are applied globally to all meshed slabs. In order to adjust the global slab mesh properties, see the following instructions.

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click **Structure**.
3. Go to the **Properties** window.
4. Accept or adjust the **Slab Mesh Size**.

5. Accept or adjust the **Slab Uniformity Factor**. At 100% uniformity all slabs in the mesh will be of approximately equal area.
6. Accept or adjust the **Slab Mesh Type**:
   - **QuadDominant**: mainly consists of quadrilateral 2D elements, but may use occasional triangular elements to create a better mesh.
   - **QuadOnly**: only consists of quadrilateral 2D elements.
   - **Triangular**: only consists of triangular 2D elements.

See also

Apply different mesh properties at different levels (page 466)

Apply different mesh properties at different levels

If you need to apply different mesh parameters at a specific level, you can do so by creating sub models. For more information, see the following instructions.

1. Create a sub model. (page 489)
2. In the **Structure** tree, expand the **Sub Models** branch.
3. Select the sub model that you created.
4. In the **Properties** window, select the **Override model's** option.
5. Adjust the mesh size and uniformity factor according to your needs.

Review the slab mesh before the analysis

In order to review the existing slab mesh in your model before running a particular analysis, see the following instructions.

1. In the **Status bar** at the bottom of the window, click **Solver View**.

   The solver view opens.
2. Right-click anywhere in the solver view.
3. In the context menu, go to **Solver models**.
4. In the list, select the solver model appropriate to the analysis that you want to run.

   If the slab mesh is applicable to the selected solver model, it is displayed.

   NOTE The slab mesh is not displayed in the working solver model. This is because the working solver model displays the model in its form before any analysis, and 2D elements are only formed during the analysis.

See also

View and manage solver models (page 551)

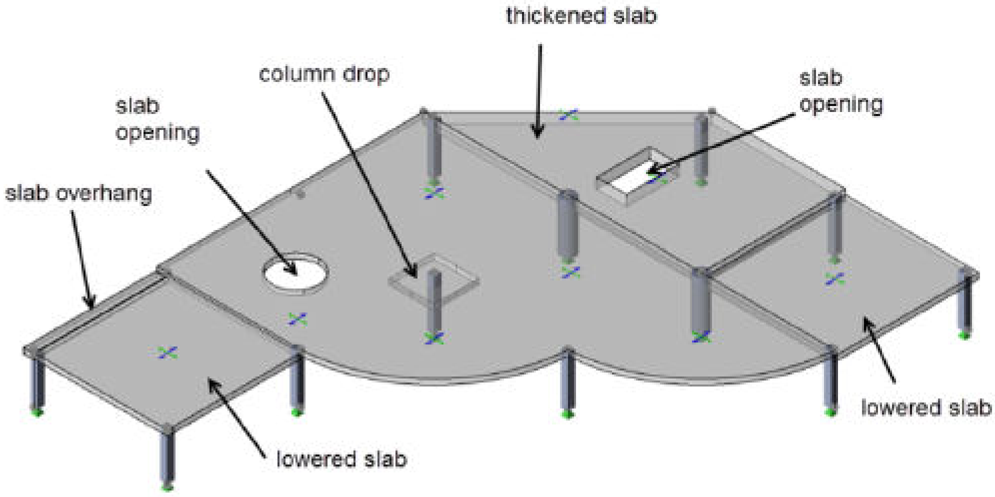
How slab properties and features impact on meshing

The slab mesh is always created in a single plane, but the resulting mesh is can be affected by the slab properties and features as described below.

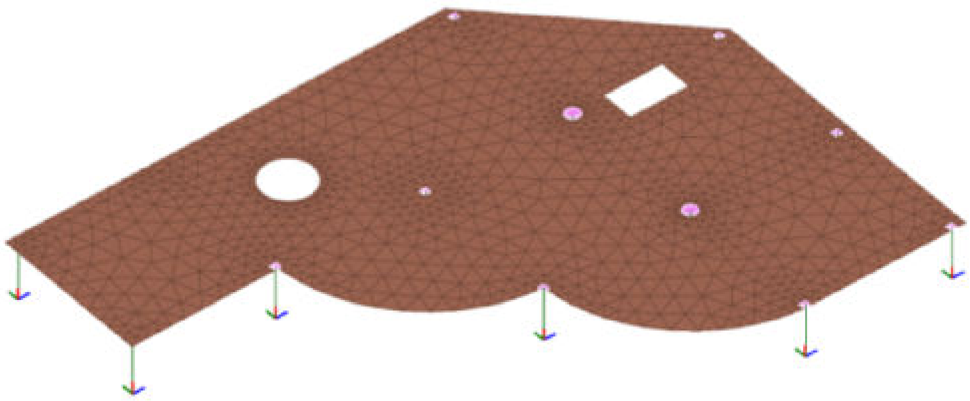
Slab thickness, vertical offsets, column drops and openings

The mesh of shell 2D solver elements is always created in the same plane, irrespective of whether slabs have different thicknesses, slabs items have been raised/lowered via vertical offsets, or column drops have been applied. Shells are not created inside slab openings and any loads placed within openings are not applied to the model.

Consider the example shown below. This features curved slab boundaries, circular and rectangular openings, thickened slab panels, lowered slab panels and a slab overhang. A column drop panel has also been inserted at one of the locations where the slab is supported by a column.



In the resulting FE solver model, since vertical offsets are not structurally significant the analysis mesh is formed at the same level relative to the top of the slab. The mesh properties do however reflect the change to the slab thicknesses in the different slab areas.



NOTE Beam solver elements and slab meshes can only be offset vertically from one another by being defined in different construction levels.

Other slab properties

Rotation Angle

Different slab items in the same slab can have different rotation angles.

This property is used for the following where appropriate:

- Span direction for 1-way load decomposition
- To determine the 2D solver element local axes in the solver model
- Bar direction for Slab on Beam and Flat Slabs.

Include in Diaphragm

This property is only available when the Diaphragm option is Semi-Rigid or Rigid. Individual slab items in the same slab can be included or excluded as required.

The effect of excluding a slab item depends on the Diaphragm option as follows:

- Semi-Rigid - excluded slab items are not meshed with semi-rigid 2D solver elements
- Rigid - internal nodes not considered in the nodal constraints

NOTE Where 2 items share a boundary and one item is included and one excluded, then the nodes along the shared boundary are included in the diaphragm.

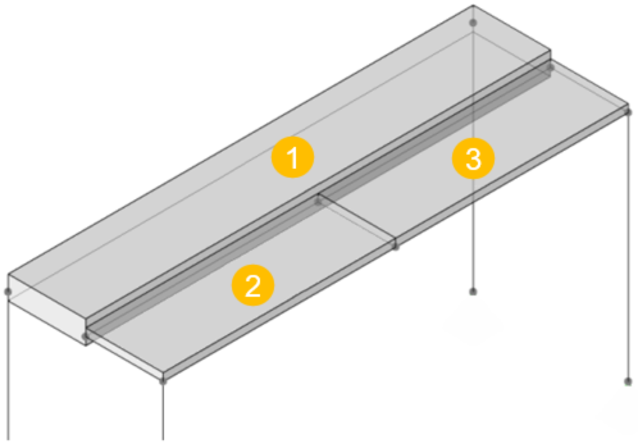
NOTE When a slab item is excluded from the diaphragm this has no effect on the mesh of shell 2D solver elements used in some of the solver models for 2-way spanning slabs.

Mesh groups

To facilitate meshing, Tekla Structural Designer automatically gathers slab items and features together into mesh groups, and meshes them as a single entity. A mesh group contains one or more slab items with identical analysis attributes. Since slab depth is a key analysis attribute, by definition a slab step, or a column drop, will produce an additional mesh group.

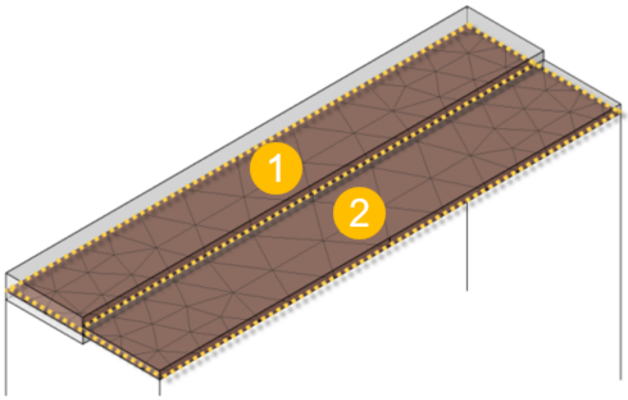
Example: Mesh groups at a slab step

In the following image, you can see three separate slab items:



1. d = 300 mm
2. d = 100 mm
3. d = 100 mm

Although there are three slab items in the previous image, there are only two different slab depths. That is why Tekla Structural Designer only creates two mesh groups, as shown in the following image:



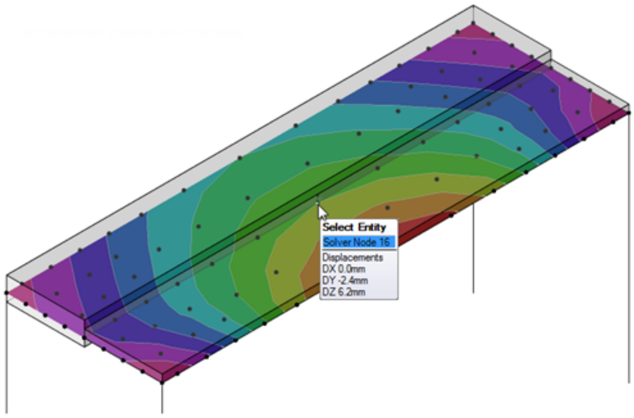
1. mesh group 1
2. mesh group 2

Discontinuity of force contours at slab steps and column drops

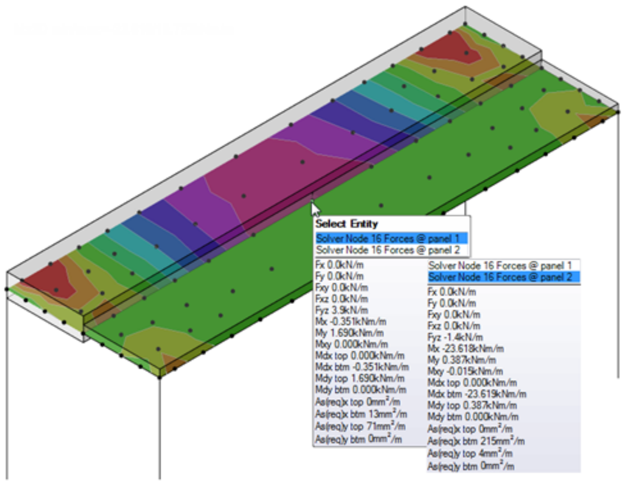
When slab items on either side of a slab step are placed into different mesh groups, the solver nodes along the boundary are shared by both groups. Each node on the boundary reports a single value of deflection, but two values of force, one for each group. That is why there will be a discontinuity of force contours along the boundary.

See the following examples:

Example: Deflection contours (no discontinuity)



Example: Moment contours, discontinuity along boundary



The force discontinuity is a genuine result. The slab items share the same curvature at the step and have the same elastic modulus, so the moment must be directly proportional to the inertia of each slab item.

NOTE Other programs may average the value across the boundary when generating the contours. However, we prefer the approach of Tekla Structural Designer because averaging would result in an unrealistically high design of the thinner slab.

Mesh group boundary warnings

Meshing may fail or produce undesirable results when there is challenging mesh group boundary geometry. In this case, Tekla Structural Designer warns you about the source points source of meshing issues.

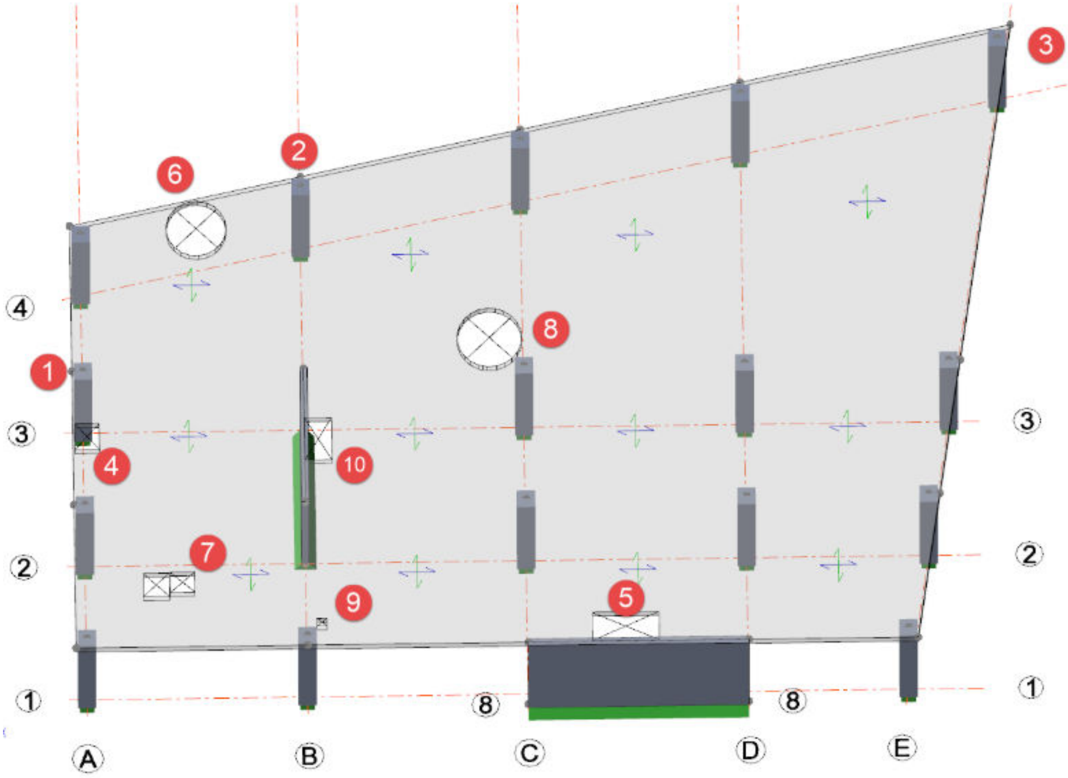
Examples of possible warning triggers are:

- short edges
- distance between a hole and an edge
- small area enclosed by a mesh group

Slab meshing controls

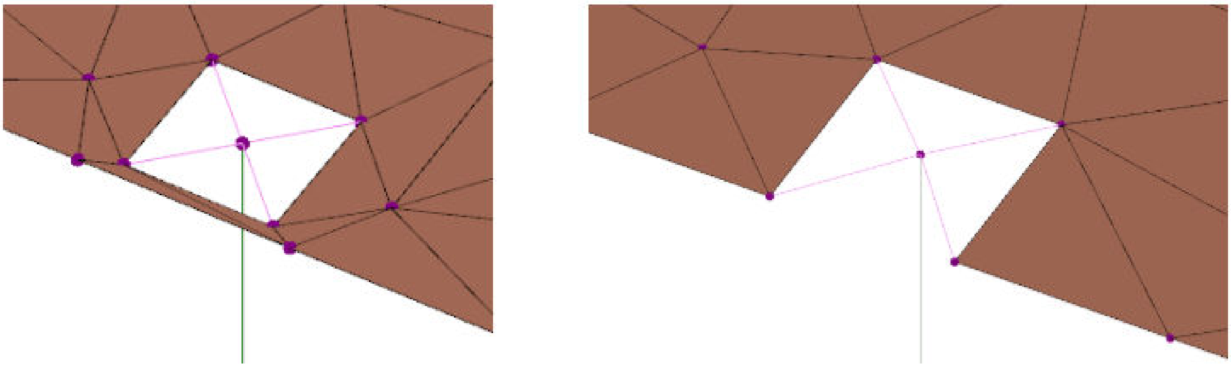
Meshing issues tend to be caused when the area being meshed ends up with really narrow strips of slab, for example when the slab edge ends up just outside the edge of a column.

On the **Meshing** page of Analysis options (page 1050) certain meshing tolerance controls can be adjusted when required to solve these type of issue.



The model shown above has various challenging situations which are discussed below along with an indication of whether changing the automatic merging distances in Analysis Options can have an impact:

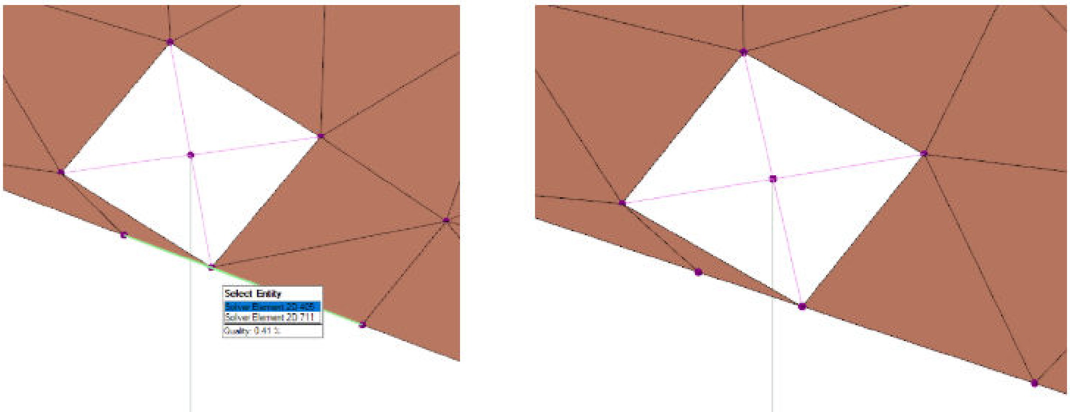
1. Column face close to parallel slab edge - (yes)



In this example the slab edge is positioned 50mm outside the column face. The view above left shows the analysis model you would get when the Column boundary to slab edge merging distance is < 50mm. The view above right shows the model you get if the merging distance is set to 50mm or above. The column corner nodes merge (move to) the slab edge eliminating the narrow strip.

The simplified analysis model avoids mesh quality warnings and gives almost identical results in terms of column transfer forces and slab design.

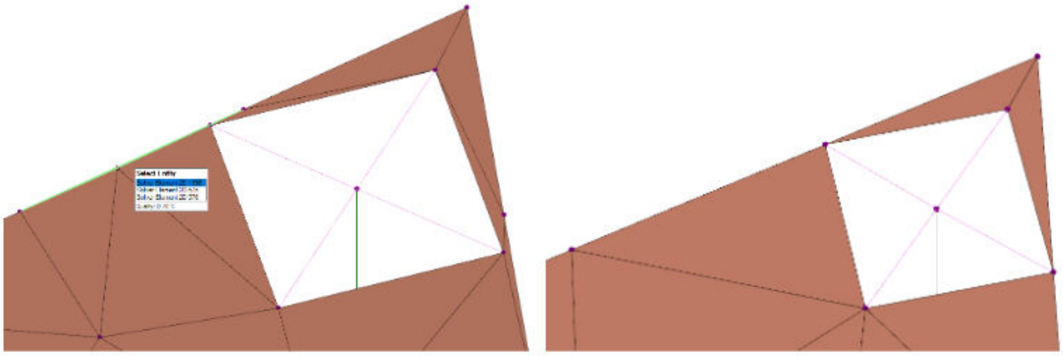
2. Column face close to inclined slab edge - (yes)



In this example the slab edge sits just a few mm outside the column corner. The view above left shows analysis model where a very poor quality element runs past the corner, (you would only end up with this model if the Column boundary to slab edge merging distance were to be set less than the default 5mm. The view above right shows the model you get if the merging distance is set to the default 5mm or above. One column corner node merges (moves to) the slab edge eliminating the narrowest part of strip.

The simplified analysis model avoids mesh quality warnings and gives almost identical results in terms of column transfer forces and slab design.

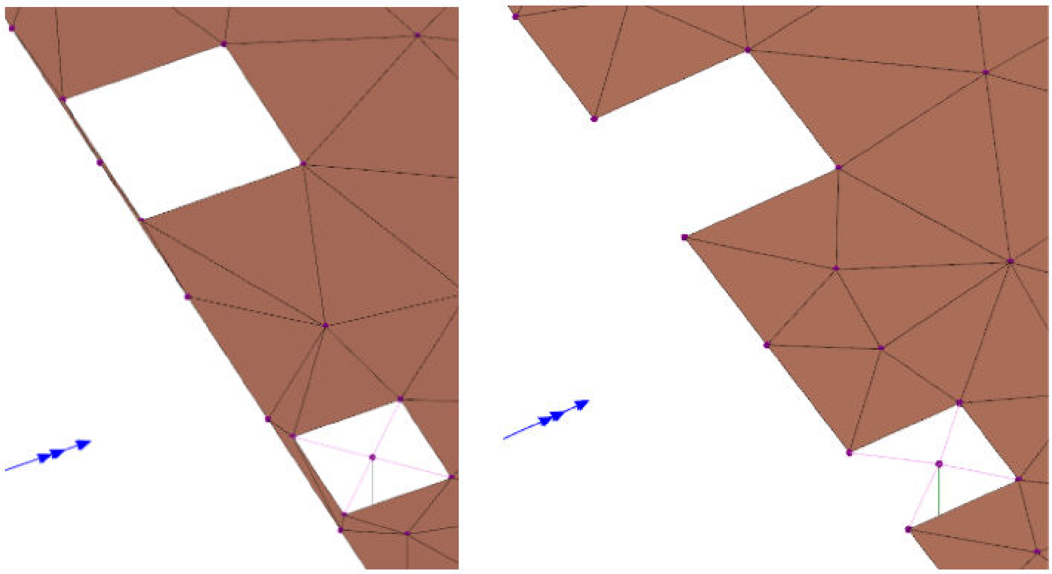
3. Column close to multiple non-parallel slab edges - (yes)



This is just another variation on positions 1 and 2. The view above left shows analysis model you would end up with if the Column boundary to slab edge merging distance were to be set less than the default 5mm. The view above right shows the model you get provided the default merging distance (5mm) is retained - meshing is simplified avoiding mesh quality warnings and gives almost identical results in terms of column transfer forces and slab design.

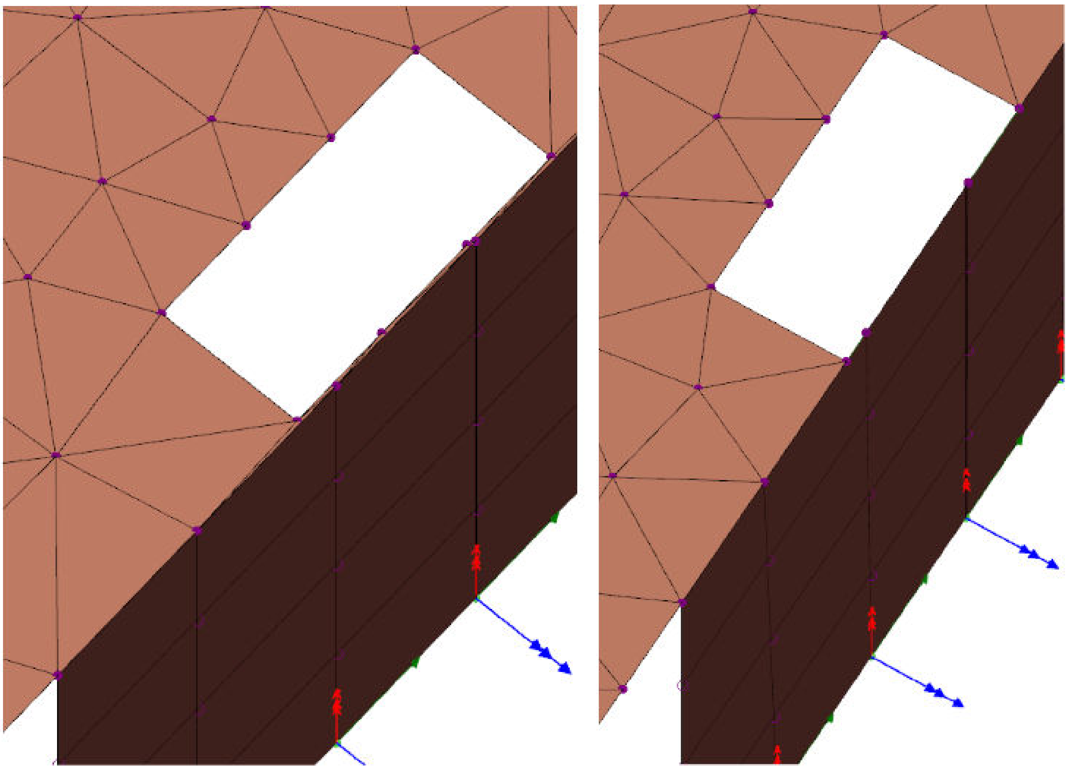
4. Opening near slab edge - (yes)

In the following examples merging will make things better, but the reality is that a slab will never be cast with thin isolated strips at the edge - you cannot effectively reinforce a 50 to 100mm wide strip of slab. In all these the ideal solution is that the modelling should reflect the reality of realistic minimum edge distances.



A hole is defined approximately 20mm from the slab edge. The effect is the same as at the column cut-out boundary - above on the right shows how with the Opening to slab edge merge distance set to 50mm the opening corners move to the slab edge. This creates a better analysis model.

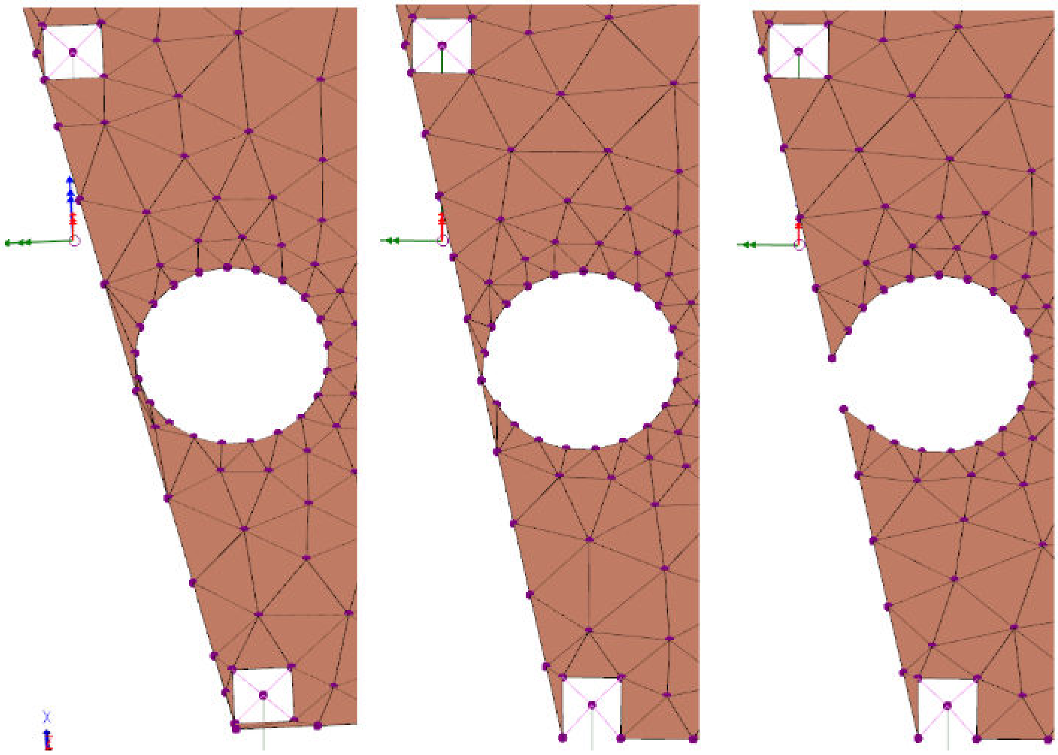
5. Opening near wall inserted along slab edge - (yes)



This is basically the same as the previous situation except that a wall happens to be defined along the edge. In such cases the user is often tempted to extend the opening into the physical width of the wall but does not extend it right to the wall insertion line.

There are pros and cons to both models. On the left the narrow meshed strip of slab can give mesh quality warnings and errors. On the right the slab mesh is better, but the model relies a bit more heavily on the wall beam to transfer moments. However, the overall minor axis moment transferred to the wall only drops from 75.7 to 72.5 kNm.

6. Circular opening near slab edge - (yes)

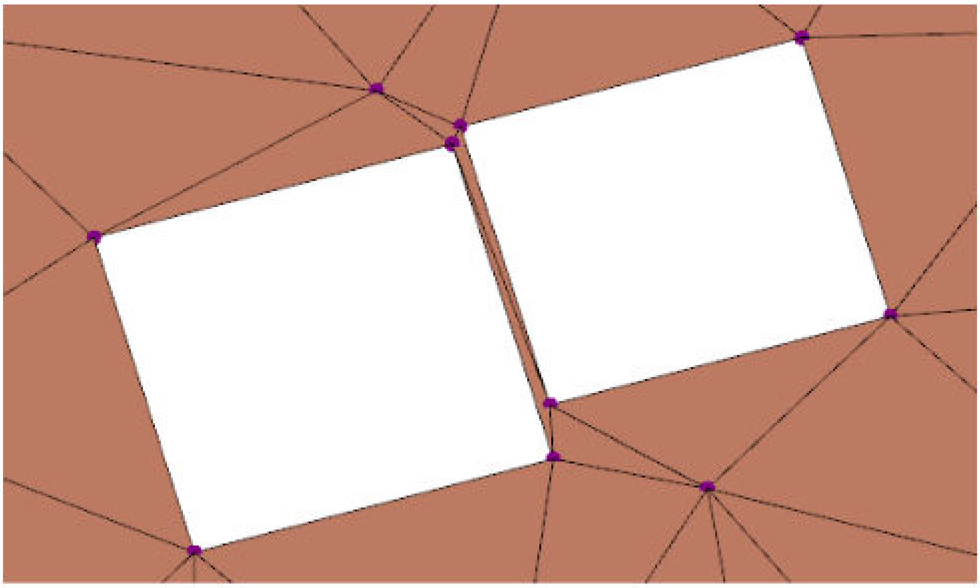


Above left shows the sort of meshing that will occur when a circular opening is created near to the slab edge. Above centre shows an example where closest point on the circle is merged to edge. The meshing looks a bit better but this is still not a great model because you will get strange local results at the connecting node. On the right where theOpening to slab edge merge distance is set to 100mm the edge gets completely broken - this is probably the best model.

7. Closely spaced openings - (no)

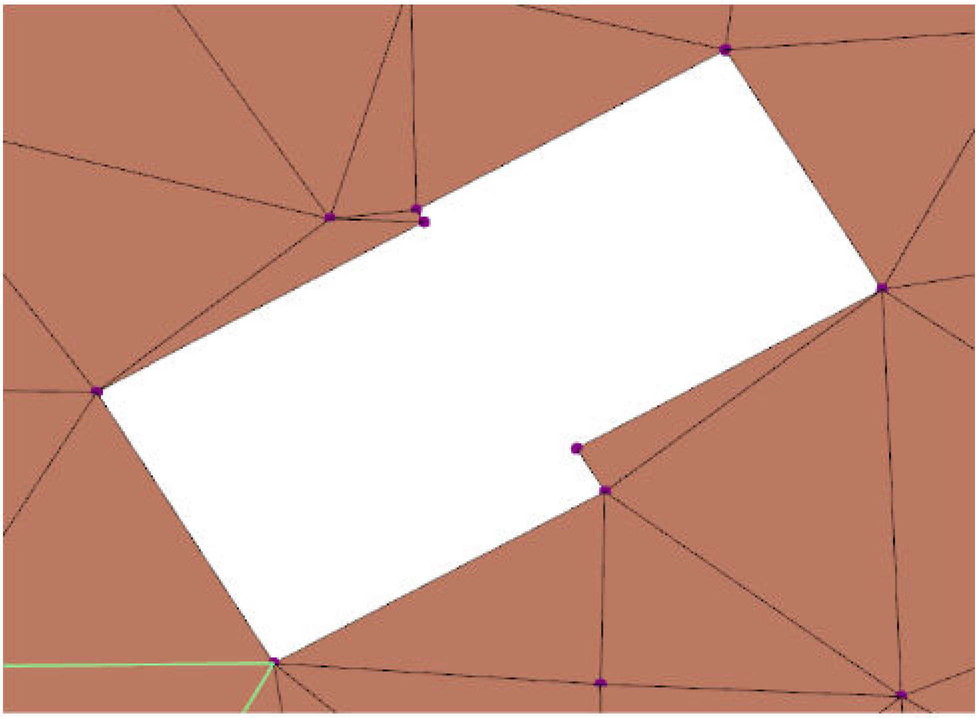
These are all examples where the merging settings have no effect.

In each case the problems can generally be resolved by adjusting the openings so that they overlap slightly.

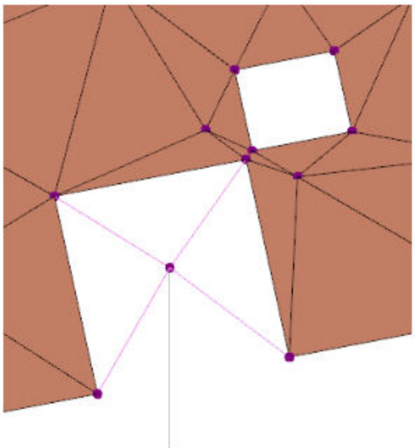


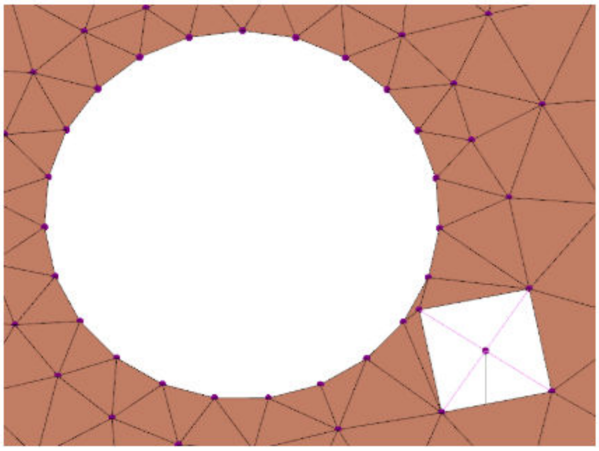
Above - shows meshing when a narrow (unbuildable?) strip of slab is left between openings.

Below shows the model when the openings are manually adjusted to eliminate this strip.

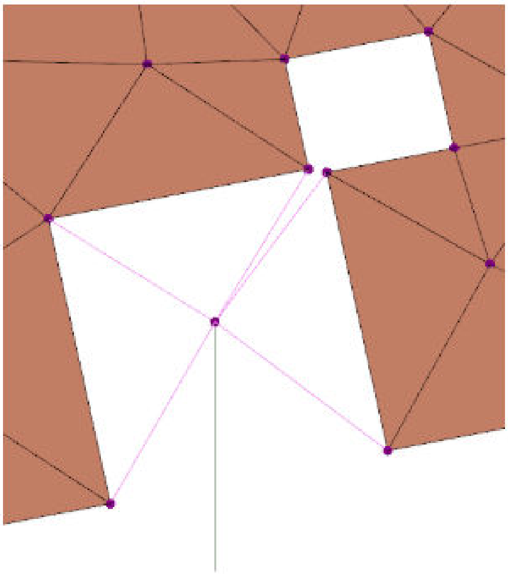


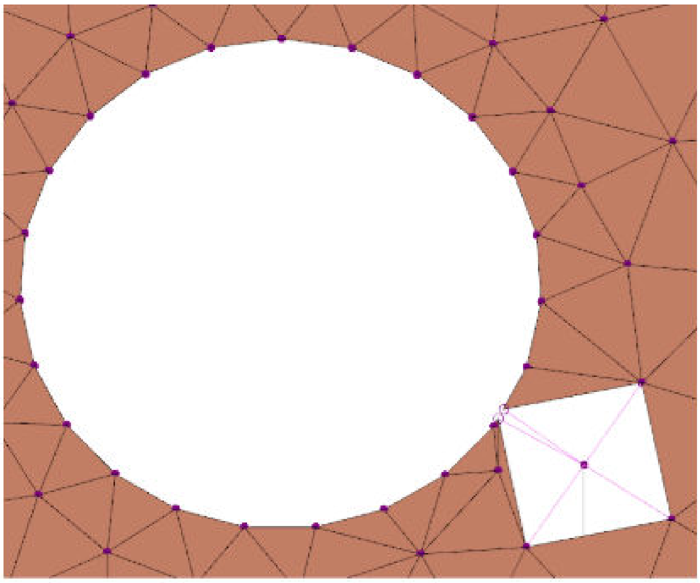
8 and 9. Opening near column - (no)



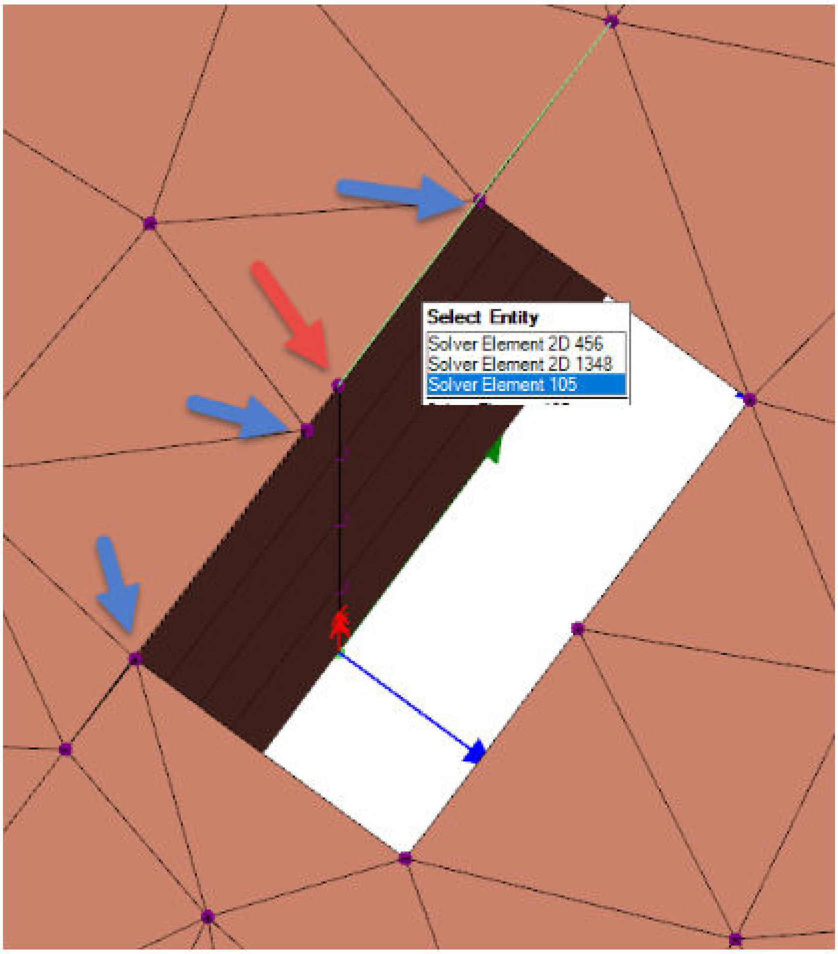


In both these cases it would again be better to adjust the openings so that they overlap the column boundaries as suggested below.



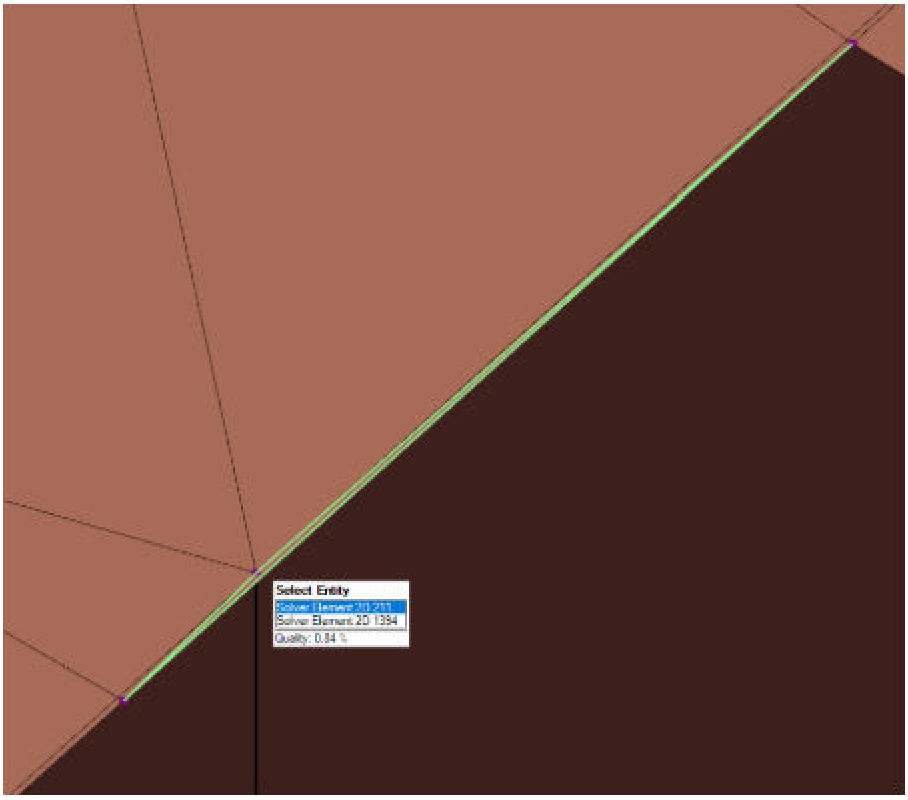


10. Opening near wall with slab on other side - (no, unless certain conditions apply)

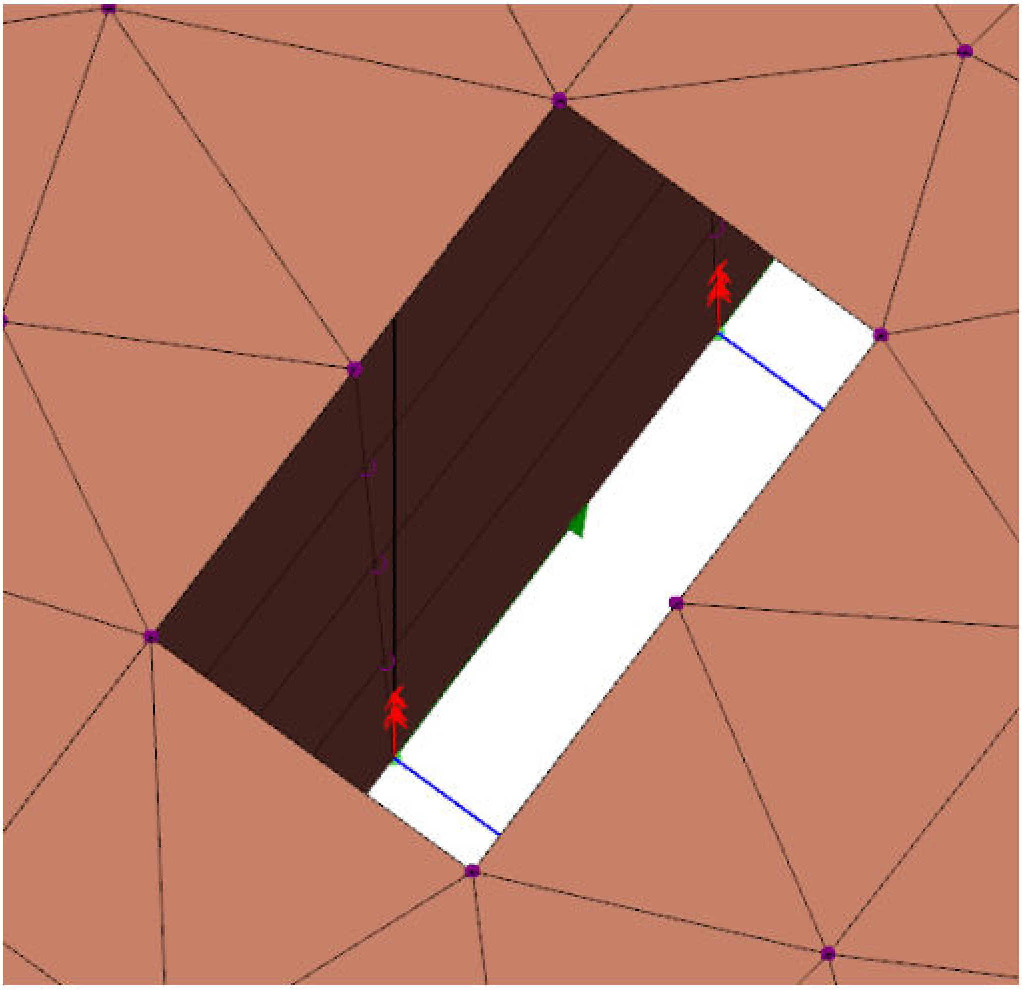


In this case the opening is defined with an edge that is approximately 4mm from the insertion line of the wall. With the Hardpoint to edge distance tolerance at the default of 5mm this creates a model that is poorly connected along the edge of the opening. The node highlighted in red is not included in the slab meshing (because it is just 4mm away from the edge). The 3 slab nodes highlighted in blue are not directly connected to the wall.

If the hard point distance is reduced to 1mm then the model changes as shown below, the node marked in red above is now included in the slab mesh so the adjacent nodes have a better connection to the wall.



However, the best model would be obtained if the opening edge got exactly aligned with the wall, this can actually be quite hard to achieve in the manual positioning of openings. A workaround at present is to force a “slab mesh boundary” along the line of the wall by making the slab on one side slightly different to the slab on the other - then the automatic mergings will create a model as shown below.



Manage FE meshed walls

By default, concrete walls automatically adopt an FE meshed model when the structure is analyzed. Meshed walls automatically use the mesh parameters of the model. However, if necessary, you can override the mesh parameters and apply a user defined mesh to an individual wall.

NOTE If necessary, you can modify concrete walls so that they use a mid-pier model instead of an FE meshed model.

Adjust global wall mesh properties

Initially, the same mesh properties are applied globally to all meshed walls. In order to adjust the global wall mesh properties, see the following instructions.

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click **Structure**.
3. Go to the **Properties** window.

4. If necessary, in **Wall Mesh Type**, change the shape of the wall mesh:
    - **QuadDominant**: mainly consists of quadrilateral 2D elements, but may use occasional triangular elements to create a better mesh.
    - **QuadOnly**: only consists of quadrilateral 2D elements.
    - **Triangular**: only consists of triangular 2D elements.
5. Adjust the horizontal and vertical sizes of the wall mesh according to your needs.

See also

Apply different mesh properties to an individual wall (page 481)

Apply different mesh properties to individual walls

If you need to apply different wall mesh properties to specific walls, you can override the model mesh properties, and modify the properties of the individual wall. For more information, see the following instructions.

1. In the model, select the walls that you want to modify.
2. Go to the **Properties** window.
3. If necessary, in **Wall Mesh Type**, change the shape of the wall mesh:
    - **QuadDominant**
    - **QuadOnly**
    - **Triangular**
4. Adjust the horizontal and vertical sizes of the wall mesh according to your needs.

Review the wall mesh before the analysis

In order to review the existing slab mesh in your model before running a particular analysis, see the following instructions.

1. In the **Status bar** at the bottom of the window, click **Solver View**.

    The solver view opens.
2. Right-click anywhere in the solver view.
3. In the context menu, go to **Solver models**.
4. In the list, select the solver model appropriate to the analysis that you want to run.

    If the wall mesh is applicable to the selected solver model, it is displayed.

    NOTE The wall mesh is not displayed in the working solver model. This is because the working solver model displays the model in its form before any analysis, and 2D elements are only formed during the analysis.

Diaphragm action in roof panels and slabs

Roof panels and slabs both have the potential to act as a diaphragms.

Both rigid and semi-rigid diaphragms can be modelled.

- Overview of diaphragm action in roof panels and slabs (page 482)
- Managing diaphragm action in roof panels and slabs (page 485)

Overview of diaphragm action in roof panels and slabs

Roof panels and slabs will both act as a diaphragms provided they have **Include in diaphragm** checked in their properties.

While **Include in diaphragm** is the only property used to determine diaphragm action in roof panels, for slab items the diaphragm properties are also determined by the **solver model** under consideration and the following properties:

- **Diaphragm option** slab item property
- **Decomposition** slab item property
- **Mesh 2-way Slabs in 3D analysis** level or slope property

How these and other choices affect the diaphragm model are described in the topics below.

Include in Diaphragm

Roof panels:

The **Include in diaphragm** choice works as follows:

- On - a diaphragm is created within the roof panel using a mesh of semi-rigid 2D solver elements
- Off - no diaphragm is created

Slab items:

For slab items the **Include in diaphragm** choice is only available when the **Diaphragm option** is Semi-Rigid or Rigid. Individual slab items can then be included or excluded as follows:

- On - a diaphragm is created within the slab item which may take the form of a shell mesh, a semi-rigid mesh, or nodal constraints
- Off - no diaphragm is created

NOTE Where 2 slab items share a boundary and one item is included and one excluded, then the nodes along the shared boundary are included in the diaphragm.

NOTE When a slab item is excluded from the diaphragm this has no effect on the mesh of shell 2D solver elements used in some of the solver models for 2-way spanning slabs.

Diaphragm option

The choice of **Diaphragm option** is set in the slab item properties. The chosen option is applied to **all** slab items in the same slab.

The available options are:

- Rigid
- Semi-rigid
- None

How your choice affects the 2D solver element types used depends on the method of decomposition selected.

NOTE For roof panels there is no choice of diaphragm option - it is always treated as semi-rigid.

Decomposition

While roof panels are always one-way spanning, for slab panels you can choose the **Decomposition** method in the slab item properties. The chosen option is applied to **all** slab items in the same slab.

Two options are available: Two-Way Spanning or One-Way Spanning - although for certain slab types the value is fixed as follows:

- Composite Slab - One-Way only
- Precast Slab - One-Way only
- Slab on Beams - Two-Way or One-Way
- Flat Slab - Two-Way only
- Steel Deck - One-Way or Two-Way
- Timber Deck - One-Way only

One-way spanning slabs are unmeshed unless the **Diaphragm option** is set to Semi-rigid, in which case they will be meshed with semi-rigid 2D solver elements.

NOTE Semi-rigid 1-way spanning slabs cannot be designed.

Two-way spanning slabs always adopt a mesh of **shell** 2D solver elements for the FE chasedown and FE Load Decomposition solver models. However for the 3D Analysis and Grillage chasedown solver models they are:

- **unmeshed** - if the Diaphragm option is set to None,
- meshed with **semi-rigid** 2D solver elements - if the Diaphragm option is set to Semi-rigid, or,
- meshed with **shell** 2D solver elements - if the Diaphragm option is set to Rigid.

NOTE Semi-rigid 2-way spanning slabs can be designed, but only using the results from the FE Chasedown, not the 3D Analysis or Grillage Chasedown.

Mesh 2-way Slabs in 3D Analysis

Two-way spanning slabs are always meshed with **shell** 2D solver elements in FE Chasedown and Load decomposition solver models. When the **Mesh 2-way Slabs in 3D Analysis** property (specified in the Level, or Slope properties) is checked the same meshing parameters are then extended to the Grillage chasedown and 3D Analysis solver models.

Summary of diaphragm constraint and mesh type configurations

The configurations of mesh and nodal constraints applied to each solver model resulting from the different permutations of the **Decomposition**, **Diaphragm option**, and **Mesh 2-way slabs in 3D analysis** properties are recapped in the table below.

Decomposition	Diaphragm Option	Mesh 2-way slabs in 3D Analysis	FE Load Decomposition & FE Chasedown Models	Grillage Chasedown & 3D Analysis Models
1-way	None	Not Applicable	No mesh; no nodal constraints	No mesh; no nodal constraints
	Semi-Rigid	Not Applicable	Semi-Rigid mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints
	Rigid	Not Applicable	No mesh; Nodal constraints	No mesh; Nodal constraints
2-way	None	Yes	Shell Mesh; no nodal constraints	Shell Mesh; no nodal constraints
		No	Shell Mesh; no nodal constraints	No mesh; no nodal constraints
	Semi-Rigid	Yes	Shell mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints
		No	Shell Mesh; no nodal constraints	Semi-Rigid mesh; no nodal constraints

	Rigid	Yes	Shell Mesh; Nodal constraints	Shell Mesh; Nodal constraints
		No	Shell Mesh; Nodal constraints	No Mesh; Nodal constraints

Managing diaphragm action in roof panels and slabs

Tekla Structural Designer forms diaphragms in every slab item of a parent slab when **Diaphragm option** is set to **Rigid** or **Semi-rigid**.

When a rigid diaphragm is specified, Tekla Structural Designer automatically applies nodal constraints to the associated slab nodes.

In turn, when a semi-rigid diaphragm is specified, Tekla Structural Designer creates a mesh of semi-rigid 2D elements within the slab.

Activate rigid diaphragm option within a slab

By default, Tekla Structural Designer automatically creates rigid diaphragm within slabs. If necessary, you can review and modify the setting for specific slabs as follows:

1. Go to the **Project Workspace**.
2. In the **Structure** tree, expand the **Slabs** branch.
3. Select the slab that contains the diaphragm.
4. In the **Properties** window, set **Diaphragm option** as required.

NOTE You can set the **Diaphragm option** as required in both slab item properties and parent slab properties. Either way, all slab items within the parent slab adopt the new setting.

Activate semi-rigid diaphragm option within a slab

You can form semi-rigid diaphragms in both one-way and two-way spanning slabs. In addition, you can review and modify the diaphragm properties by selecting the slab within which they are located.

1. Go to the **Project Workspace**.
2. On the **Structure** tree, expand the **Slabs** branch.
3. Select the desired slab.
4. In the **Properties** window, set **Diaphragm option** to **Semi-rigid**.
5. If necessary, to adjust the flexibility of the diaphragm, type a value in the **Divide stiffness by** option.

TIP You can set the **Diaphragm option** as required in both slab item properties and parent slab properties. Either way, all slab items within the parent slab adopt the new setting.

Activate semi-rigid diaphragm option within a roof panel

You can form semi-rigid diaphragms in roof panels as follows:

1. Go to the **Project Workspace**.
2. On the **Structure** tree, expand the **Roofs** branch.
3. Select the desired roof panel.
4. In the **Properties** window, select **Include in diaphragm**.

NOTE You cannot form a *rigid* diaphragm within a roof panel.

Adjust semi-rigid diaphragm flexibility

When the Diaphragm option is Semi-Rigid a **Divide stiffness by** value can be set in the slab item properties. The same value is applied to **all** slab items in the same slab.

The stiffness determined from the material properties and slab thickness is divided by this value in order to adjust semi-rigid diaphragm flexibility.

Adjust global semi-rigid mesh properties

When a semi-rigid mesh is created the 2D solver element shape (triangular or quad), the degree of mesh uniformity, and mesh type adopted are obtained from the **Structure Properties**. These parameters can be overridden for individual sub-models by setting different values in the respective **Sub Model Properties**.

1. Go to the **Project Workspace**.
2. In the **Structure** tree, click **Structure**.
3. Go to the **Properties** window.
4. Accept or adjust the **Semi-Rigid Mesh Size**.
5. Accept or adjust the **Semi-Rigid Uniformity Factor**.
6. Accept or adjust the **Semi-Rigid Mesh Type**:
   - **QuadDominant**: mainly consists of quadrilateral 2D elements, but may use occasional triangular elements to create a better mesh.
   - **QuadOnly**: only consists of quadrilateral 2D elements.
   - **Triangular**: only consists of triangular 2D elements.

NOTE Beam elements are not split by semi-rigid 2D element nodes.

Apply different semi-rigid mesh properties at different levels
If you need to apply different semi-rigid mesh parameters at a specific level, you can do so by creating sub models.

1. Create a sub model. (page 489)
2. In the **Structure** tree, expand the **Sub Models** branch.
3. Select the sub model that you created.
4. In the **Properties** window, select the **Override model's** option.
5. Adjust the semi-rigid mesh size, uniformity factor and type according to your needs.

Identify the nodes constrained by rigid diaphragms
In order to see which nodes are constrained by diaphragms, see the following instructions.

1. In the **Status bar** at the bottom of the window, click **Solver View**.

   A solver view opens. The rigid and semi-rigid diaphragms are represented as different colored shaded planes.

   Nodes that are constrained by a rigid diaphragm:

   - Must lie within, or be on the edge of, the shaded rigid diaphragm plane.
   - Must be solid rather than hollow, or excluded.

   Therefore, the following nodes are not constrained by a rigid diaphragm:

   - Solid nodes that lie outside the shaded rigid diaphragm plane.
   - Hollow, or excluded, nodes.

Exclude individual nodes from a rigid diaphragm
If necessary, you can exclude specific nodes from a rigid diaphragm. In order to do so, see the following instructions.

1. In the status bar at the bottom of the window, click **Review View**. A review view opens.
2. On the **Review** tab, click **Diaphragm On/Off**.

The nodes are displayed as follows:

- Included nodes are only constrained if they lie within or on the edge of a rigid diaphragm. If they lie outside the rigid diaphragm, they are not constrained by it.
- Excluded nodes are always unconstrained.

3. Click a node to switch whether it is included in the diaphragm or not.

Exclude slab items from a diaphragm
By default, Tekla Structural Designer forms a rigid diaphragm in all the individual slab items within a slab. However, you can decide to exclude specific slab items. In order to do so, see the following instructions.

1. In the structure view, select the slab item that you want to exclude.
2. In the **Properties** window, clear the **Include in diaphragm** option.

TIP You can also exclude individual slab items graphically in the Review View. On the **Review** tab, click **Diaphragm On/Off** and then click the slab item to include/exclude.

Manage sub models

Tekla Structural Designer initially treats each structure as a single sub model. In practice, this means that the Tekla Structural Designer applies the same mesh parameters to all meshed slabs. However, if needed, you can create sub-models in your model by creating horizontal planes between levels. Each individual sub model controls the slab mesh parameters at the levels within it.

Tekla Structural Designer creates additional sub models automatically for every level specified as a **Floor** in the **Construction Levels** dialog box when Tekla Structural Designer performs one of the following:

- Grillage chasedown analysis
- FE chasedown analysis

You can also define sub models manually in the **Sub Models** dialog box.

Tekla Structural Designer uses the he slab mesh parameters specified for a sub model in any analysis that requires the slabs to be meshed, such as load decomposition, building analysis with meshed floors, or FE chasedown analysis.

In both grillage and FE chasedown, Tekla Structural Designer performs the analyses one sub model at a time. Tekla Structural Designer first analyzes the topmost sub model, and then applies its support reactions as loads for the analysis of the sub model below. The sequence continues until Tekla Structural Designer has analyzed all sub models down to the foundation level.

For both grillage and FE chasedown analysis, you can modify the default support conditions applied to the sub-models, if necessary.

Definitions of sub model characteristics

- **Sub model**: a part of the 3D model between two horizontal sub model divide planes.

  Each sub model contains all members entirely between the two horizontal planes. For the columns, wall and braces split by a divide plane, the stacks and brace length above the top plane are included in the sub model, as are the stacks and brace length below the lower plane.
- **Sub model divide planes**: horizontal planes that you can add, delete or move in the 3D structure. Sub model divide planes are notional and infinite. Tekla Structural Designer only allows the planes to cut through the structure where they only split the following members:
  - Column stacks
  - Wall stacks
  - Steel braces
- **Sub model supports**: the artificial supports that Tekla Structural Designer defines for the column and wall stack ends and braces that pass through the divide planes
- **Structure supports**: the supports that the user has defined in the 3D structure
- **Column and wall stacks**: the span length of a column or wall.
- **Volume of the sub model**: the 3D space that exists between any two adjacent sub model divide planes.

Basic rules of sub models

When all sub models are considered together, they form the complete structure. Only column stacks, wall stacks, and braces that are split by sub model divide planes can be in several sub models.

This means that:

- Each member in the 3D model is in at least one sub model.
- A sub model cannot contain a member that is already in another sub model, unless that member is a column, wall, or brace divided by a sub model divide plane.
- A sub model must contain at least one beam member, one truss member, or one slab item.

Create sub models

Sub models allow you to apply different slab mesh parameters within your structure. To create sub models, see the following instructions.

1. In the **Structure** tree, double-click **Sub Models**.

   The **Sub Models** dialog box opens.

2. According to your needs, do one of the following:

To	Do this
Automatically create sub models	• Click **Generate**.
Manually create sub models	• According to your needs, click **Insert Above**, or **Insert Below**.

3. If necessary, modify the height above the base of each level in the **Level** field.

4. Click **OK**.

See also

Sub Models dialog (page 1223)

Sub Model Properties (page 937)

Open a 3D view of a sub model

To display a sub model in its own 3D view, see the following instructions.

1. In the **Project Workspace**, open the **Structure** tree.

2. Expand the **Sub Models** branch.

3. Double-click the sub model.

   Tekla Structural Designer opens a 3D view of the selected sub model.

> **TIP** To open a solver view of the sub model, right-click the sub model, and select **Open solver view**.

Delete sub models

If necessary, you can delete existing sub models in the **Sub Models** dialog box. For detailed information on deleting sub models, see the following instructions.

1. In the **Structure** tree, double-click **Sub Models**.

   The **Sub Models** dialog box opens.

2. Select the sub model that you want to delete.

3. Click **Delete**.

   Tekla Structural Designer deletes the sub model, and the floors are transferred to the sub model immediately above the deleted one.

6.2 Run analyses

Various analyses can be run from the **Analyze** toolbar.

Click the links below to find out how to:

- Run a 1st order linear or non-linear analysis (page 491)
- Run a 1st order vibration analysis (page 492)
- Run a 2nd order linear or non-linear analysis (page 493)
- Run a 2nd order buckling analysis (page 493)
- Run a seismic analysis (page 494)
- Run FE chasedown or grillage chasedown analysis (page 495)
- Run Analyze All (Static) (page 496)
- Run 3D only (Static) (page 496)

Once the analysis has completed, the Project Workspace can be used to verify the results in a couple of ways:

- Check sum of reactions against load input (page 497)
- Check stability and overall displacement (page 498)

Run a 1st order linear or non-linear analysis

To run either a linear or non-linear 1st order analysis on your model, see the following instructions.

See also

The Results View (page 498)

View tabular results for support reactions (page 592)

View tabular results for nodal deflections (page 592)

View tabular results for solver element end forces (page 593)

Run 1st order linear analysis

1. On the **Analyze** tab, click **1st Order Linear**.

   The **Select loading** dialog box opens.

2. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.
3. Click **OK**.

Tekla Structural Designer analyzes the model.

At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 1st order non-linear analysis

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the **1st Order Non-Linear** page.
3. If necessary, adjust the convergence criteria and relaxation factors.
4. Click **OK**.
5. On the **Analyze** tab, click **1st Order Non-linear**.

   The **Select loading** dialog box opens.

6. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.
7. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 1st order modal analysis

In order to run a modal analysis on your model, see the following instructions.

NOTE To run a modal analysis, your model must contain an active modal mass combination.

For more information, see Create modal mass combinations (page 347).

---

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the page (page 1052).
3. If necessary, adjust the analysis options.
4. On the **Analyze** tab, click **1st Order Modal**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** opens to allow you to display the mode shapes graphically. Tabular results can also be viewed in a **Solver Model Data View**.

Run a 2nd order linear or non-linear analysis

In order to run either a linear or a non-linera 2nd order analysis on your model, see the following instructions.

Run a 2nd order linear analysis

1. On the **Analyze** tab, click **2nd Order Linear**.

   The **Select loading** dialog box opens.

2. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.

3. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 2nd order non-linear analysis

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the **2nd Order Non-linear** page.

3. If necessary, adjust the convergence criteria and relaxation factors.

4. Click **OK**.

5. On the **Analyze** tab, click **2nd Order Non-linear**.

   The **Select loading** dialog box opens.

6. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.

7. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 2nd order buckling analysis

In order to run a buckling analysis on your model, see the following instructions.

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.

2. In the dialog box, go to the **2nd Order Buckling** page.
3. If necessary, adjust the buckling options.
4. Click **OK**.
5. On the **Analyze** tab, click **2nd Order Buckling**.

   The **Select loading** dialog box opens.
6. In the **Select loading** dialog box, select the combinations and loadcases that you want to analyze.
7. Click **OK**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

See also

The Results View (page 498)

View buckling factors (page 597)

Run a seismic analysis

In order to run either a 1st or 2nd order seismic analysis on your model, see the following instructions.

Run a 1st order RSA seismic analysis

1. On the **Analyze** toolbar, click **Settings**.

   The **Analysis Settings** dialog box opens.
2. In the dialog box, go to the **1st Order Seismic** page.
3. If necessary, adjust the analysis options.
4. Click **OK**.
5. On the **Analyze** tab, click **1st Order RSA Seismic**.

   Tekla Structural Designer analyzes the model.

   At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

Run a 2nd order RSA seismic analysis

- On the **Analyze** tab, click **2nd Order RSA Seismic**.

Tekla Structural Designer analyzes the model.

At the end of the analysis process, the **Results View** and the **Results** toolbar open, and allow you review the analysis results graphically.

See also

RSA seismic results (page 518)

The Results View (page 498)

View tabular results for support reactions (page 592)

View tabular results for nodal deflections (page 592)

View tabular results for solver element end forces (page 593)

Run FE chasedown or grillage chasedown analysis

You can run FE chasedown and grillage chasedown analyses by using the **Analyze All (Static)** command. The analyses are also run when they are required as a part of the combined analysis and design process.

RESTRICTION FE chasedown and grillage chasedown analyses are run for loadcases only, and not for load combinations.

In a combined analysis and design process, Tekla Structural Designer performs the previously mentioned analyses as follows:

- If the model contains two-way slabs or the user has selected the appropriate option in the **Design Settings** dialog box, Tekla Structural Designer performs FE chasedown analysis.

  TIP To control whether Tekla Structural Designer performs FE chasedown analysis, do the following:

  1. On the **Design** tab, click **Settings**.
  2. Under concrete beams, columns, or walls, go to **General Parameters**.
  3. Select whether you want to design the members for FE chasewdown analysis results.
  4. Click **OK**.

- If the model contains one or more concrete members, Tekla Structural Designer performs the grillage chasedown analysis.

See also

Analyze all combinations and loadcases (page 496)

Run Analyze All (Static)

Static designs can only be performed provided suitable analysis results exist - while these results are created when you run a combined analysis and member design, in some situations you may prefer to run the analysis separately and then selectively design parts of the model as required.

Analyze All (Static) facilitates this as it performs all the analyses for static loadcases and active static load combinations that are required to enable the selective designs to be carried out.

To run a separate analysis in this way proceed as follows:

1. On the **Analyze** toolbar, click **Analyze All (Static)**.

Tekla Structural Designer performs the following analyses:

- 1st order linear/non-linear
- 2nd order linear/non-linear (only if this has been specified by the user in **Design > Design Settings > Analysis**))
- Grillage chasedown (if one or more concrete members exist)
- FE chasedown (if two-way spanning slabs exist)

At the end of the analyses if a 2D or 3D view is active it is switched to a **Review View**.

See also

The Results View (page 498)

View tabular results for support reactions (page 592)

View tabular results for nodal deflections (page 592)

View tabular results for solver element end forces (page 593)

Run 3D only (Static)

3D only (Static) analysis can be used to save time during scheme design, for example while addressing overall stability, sway, drift, wind drift, etc.

To run a 3D only analysis proceed as follows:

1. On the **Analyze** toolbar, click **3D only (Static)**.

Tekla Structural Designer performs the following 3D analyses:

- 1st order linear/non-linear

- 2nd order linear/non-linear (only if this has been specified by the user in **Design > Design Settings > Analysis**)

At the end of the analysis if a 2D or 3D view is active it is switched to a **Review View**.

See also

The Results View (page 498)

View tabular solver model data and results (page 591)

Review tabular data (page 762)

Check sum of reactions against load input

Once you have performed an analysis in your model, you can use the **Loading** tree to quickly check that the total reaction from each analysis equates to the total load on structure. This way, you can quickly verify that none of the applied loads is missing.

1. In the **Project Workspace**, go to the **Loading** tab.
2. Review the status of each loadcase and combination.

   The status options are the following:

   - : total reaction equates to the total load on structure
   - : total reaction does not equate to the total load on structure
   - : total reaction is not available

Example cross-checks

The **General** section of the properties for each loadcase provides summations (in global X, Y & Z) of the different load types applied to the structure, from these the total applied load is determined. The total reaction from the 3D Analysis result is also reported in this section. Assuming a 3D analysis, FE chasedown analysis and grillage chasedown analysis have all been performed:

- The **Total User Applied Load** should equate to the **Total Load on Structure**.
- The **Total Load on Structure** should equate to the **Total Reaction** for 3D analysis reported in the **General** section.
- The **Total Load on Structure** should equate to the **Total Reaction** for FE chasedown analysis reported in the **FE ChaseDown** section.
- For each sub-model in the **FE ChaseDown** section the **Load Applied** when added to the **Load from above** should equate to the **Reaction**
- The **Total Load on Structure** should equate to the **Total Reaction** for Grillage chasedown analysis reported in the **Grillage ChaseDown** section.

- For each sub-model in the **Grillage ChaseDown** section the **Load Applied** when added to the **Load from above** should equate to the **Reaction**

Check stability and overall displacement

You can use the **Status** tree in the **Project Workspace** to review the stability checks and overall displacement of the model.

Review the stability checks and overall displacement in the Status tree

1. In the Project Workspace click the **Status** tab
2. In the**Status** tree , expand the **Design** heading.
3. Expand the required check or overall displacement to review as required.

> **NOTE** Further details of the checks performed are available by reviewing the tabular design data.

See also:

Filter tabular data (page 811)

6.3 Display analysis results

Once analysis has been run, you can switch 2D and 3D scene views to the **Results View** regime to display graphical analysis results. You can also display analysis results for individual members and walls in a **Load Analysis View**

Click the links below to find out more:

- The Results View (page 498)
- The Load Analysis View (page 542)

Tabular results are also available, either by creating reports, or by viewing the tabular solver model data.

See also

View tabular solver model data (page 591)

The Results View

To change to a **Results View** click  in the **Status bar** at the bottom of the main window.

After setting the analysis type and loading (page 500), most of the results can then be displayed simply by selecting from the appropriate toolbar group:

Reactions

- Display reactions (page 500)

1D Results

- Display 1D results (page 502)
- Display 1D deflections (page 502)
- Animate 1D and 2D deflections (page 502)

Sway Drift...

- Display sway drift and story shear (page 503)

Notional Loads

- Display notional forces and seismic equivalent lateral forces (page 504)

2D Results

- Display 2D results (page 504)
- Display 2D deflections (page 510)
- Animate 1D and 2D deflections (page 502)
- Display AsReq contours (page 510)

2D Integrated Results

- Display wall lines (page 511)
- Display core lines (page 511)
- Manage, display and design result lines (page 515)
- Manage and display result strips (page 512)
- **NOTE** Result lines and result strips must be created before they can be displayed.

Mode Shapes

- Display mode shapes (page 518)

RSA Results

- RSA seismic results (page 518)

If required, adjustments can be applied to the diagrams as follows:

- To change contour intervals and colors, see Customize the display of 2D contours (page 522)
- To adjust the amplitude of the diagrams, see Change result diagram scale settings (page 523)

- In 2D views, it is sometimes necessary to switch to an isometric projection, see Display 2D view in isometric projection (page 524)

Set the analysis type and loading for viewing analysis results

When Tekla Structural Designer has performed several analysis types, the results of each analysis are held separately. Therefore, there is no need to re-perform a particular analysis to recall its results.

1. On the **Results** toolbar, in the **Result Type** group, select the required analysis method.

   NOTE If you select: 1st order modal, 2nd order buckling, 1st order RSA seismic or 2nd order RSA seismic analyses, you should also choose the **Mode** to be displayed.

   ---

2. In the **Result Type** group, if required, click **Reduce Axial Force** if you want to take into account reductions when viewing axial forces in a loadcase or combination in which reductions have been applied.
3. In the **Result Type** group, click **Strength Factors**, or **Service Factors**, as required, to control which factors are used when displaying results for combinations.
4. In the **Loading** list, click either the **Select Loadcase**, **Select Combination**, or **Select Envelope** button.
5. In the **Loading** list, select the desired loadcase, combination, or envelope.

You can now proceed to select the diagram to be displayed.

See also

The Results View (page 498)

Display reactions

Support reactions

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **Reactions** group, select the reactions to be displayed:

1D	Reactions at supports under 1D elements (e.g. column support reactions),
2D	Reactions at supported nodes of 2D elements (individual meshed wall supports and mat foundation supports),

	Distributed wall reactions (Fz) - available for self-supported meshed, mid-pier, bearing and shear-only wall panels with horizontal bottom levels. When active the nodal support reactions exclusive to the wall are hidden. Summary of assumptions, • Wall panel has enough height to ensure that all loading that contributes to the reaction is far enough from the support such that the principle of St. Venant is considered valid, • Material is perfectly homogeneous and isotropic, • Elastic limit is nowhere exceeded and 'E' is same in tension and compression.
	Integrated reactions at core supports. When active support reactions exclusively from 1D and 2D core members are hidden.

4. For **Distributed wall reactions** the droplist beneath should be set to **Total**, for other support reactions you can select the desired reaction from the droplist.

Fx	support local x axis reaction (corresponds to **Fminor** in the Foundation Reactions Report),
Fy	support local y axis reaction ((corresponds to **Fmajor** in the Foundation Reactions Report),
Fz	support local z axis reaction (corresponds to **Fvert** in the Foundation Reactions Report),
Mx	support local x axis moment (corresponds to **Mmajor** in the Foundation Reactions Report).
My	support local y axis moment (corresponds to **Mminor** in the Foundation Reactions Report)
Mz	support local z axis moment (corresponds to **Mtor** in the Foundation Reactions Report)
Fxyz	support local reactions in **Fx** and **Fy** and **Fz**
Mxyz	support local reactions in **Mx** and **My** and **Mz**
Total	all support local reactions (**Fx**, **Fy**, **Fz**, **Mx**, **My** and **Mz**.

5. From the **Text** group select **Reaction** to display values on the diagram.

Beam end reactions

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **Reactions** group, select **Beam Ends**, then from the droplist beneath select the desired reaction.
4. From the **Text** group select **Reaction** to display values on the diagram.

Display 1D results

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **1D Results** group, select **Results**, then from the droplist underneath it, select the desired result.
4. From the **Text** group select **Forces** to display values on the diagram.

See also

Change result diagram scale settings (page 523)

Display 2D view in isometric projection (page 524)

Display 1D deflections

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **1D Results** group, select **Deflections**, then from the droplist underneath it, select the desired deflection type.
4. From the **Text** group select **Deflection** to display values on the diagram.

See also

Change result diagram scale settings (page 523)

Animate 1D and 2D deflections (page 502)

Animate 1D and 2D deflections

Once the required deflections are displayed, they can be animated as follows:

1. Right click in the view and from the context menu select **Animate**.

   The animation commences.
2. If required you can change the number of frames per second, the cycle duration and the amplitude by adjusting the sliders and clicking **Apply** in the **Animation** dialog.
3. To end the animation, simply close the **Animation** dialog.

   NOTE Animations are available for static displacements and those for modal and buckling analysis.

Display sway drift and story shear

Once you have selected the analysis type, and the loadcase, combination, or envelope, you can simply view the results by selecting the desired reaction on the **Results** tab. The commands in the **1D Results** group and the **Deflections** group display the results for 1D elements, such as beams, columns, and trusses, and walls modelled using the mid-pier option. Conversely, the commands in the **2D Results** group and the **2D Deflections** group display the results for 2D elements, such as FE slabs and FE walls. For more information, see the following instructions.

Sway

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the droplist beneath select **Sway Dir 1** or **Dir 2** as required.
4. From the **Text** group select **Deflection** to display values on the diagram.

Relative sway

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the droplist beneath select **Relative Sway Dir 1** or **Dir 2** as required.
4. From the **Text** group select **Deflection** to display values on the diagram.

Wind drift

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the droplist beneath select **Wind Drift Dir 1** or **Dir 2** as required.
4. From the **Text** group select **Deflection** to display values on the diagram.

Shear

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **Sway Drift and Story Shear** group, select **Drift**, then from the droplist beneath select **Dir 1 Shear** or **Dir 2 Shear** as required.

See also

Change result diagram scale settings (page 523)

Display notional forces and seismic equivalent lateral forces

To view the notional forces or seismic equivalent lateral forces that apply to a particular load combination or loadcase, see the following instructions.

View the magnitude of EHF in a combination

RESTRICTION The command is only applicable to the Eurocode head code.

1. In the **Loading** list, select the required combination.
2. On the **Results** tab, click **EHF**.

View notional loads in a combination

RESTRICTION The command is only applicable to the ACI/AISC head code.

1. In the **Loading** list, select the required combination.
2. On the **Results** tab, click **NF**.

View the magnitude of NHF in a combination

RESTRICTION The command is only applicable to the BS, IS and Australian head codes.

1. In the **Loading** list, select the required combination.
2. On the **Results** tab, click **NHF**.

View the magnitude of equivalent lateral forces in a seismic loadcase or combination

1. In the **Loading** list, select the required seismic loadcase or combination.
2. On the **Results** tab, click **Seismic**.

Display 2D results

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **2D Results** group, select **Results**, then from the droplist beneath select the result required.

Mdx top	Wood Armer bending along panel x axis, top surface

Mdx bottom	Wood Armer bending along panel x axis, bottom surface
Mdy top	Wood Armer bending along panel y axis, top surface
Mdy bottom	Wood Armer bending along panel y axis, bottom surface
Bearing pressure	Bearing pressure
Fx	Axial force in panel x axis
Fy	Axial force in panel y axis
Fxy	Complimentary in-plane shear
Fxz	Shear in panel z axis in the panel xz plane
Fyz	Shear in panel z axis in the panel yz plane
Mx	Bending along panel x axis
My	Bending along panel y axis
Mxy	Plate torsional moment
σ_x top	In-plane axial stress in the x direction, top surface
σ_y top	In-plane axial stress in the y direction, top surface
τ_{xy} top	In-plane shear stress, top surface
σ_x bottom	In-plane axial stress in the x direction, bottom surface
σ_y bottom	In-plane axial stress in the y direction, bottom surface
τ_{xy} bottom	In-plane shear stress, bottom surface
σ_x max tension	Maximum tension stress in the x direction for both surfaces $= \max(\sigma_x\text{top}, \sigma_x\text{bottom}, 0) \geq 0$
σ_y max tension	Maximum tension stress in the y direction for both surfaces $= \max(\sigma_y\text{top}, \sigma_y\text{bottom}, 0) \geq 0$
σ_x max compression	Maximum compression stress in the x direction for both surfaces $= \min(\sigma_x\text{top}, \sigma_x\text{bottom}, 0) \leq 0$
σ_y max compression	Maximum compression stress in the y direction for both surfaces $= \min(\sigma_y\text{top}, \sigma_y\text{bottom}, 0) \leq 0$
σ_x in-plane	In-plane center stress in the x direction
σ_y in-plane	In-plane center stress in the y direction
σ_x in-plane tension	In-plane center tension stress in the x direction $= \max(\sigma_x\text{in-plane}, 0) \geq 0$
σ_y in-plane tension	In-plane center tension stress in the y direction $= \max(\sigma_y\text{in-plane}, 0) \geq 0$
σ_x in-plane compression	In-plane center compression stress in the x direction $= \min(\sigma_x\text{in-plane}, 0) \leq 0$

σ_y in-plane compression	In-plane center compression stress in the y direction = min ($\sigma_y$in-plane, 0) ≤ 0

NOTE Tooltips are available for the contour plots, by hovering over nodes. These display all the relevant results at the node for the selected loading.

See also

Stresses in 2D elements (page 506)

Stresses in 2D elements

Which stresses can be displayed?

You can view the stresses on the outer faces of 2D elements for both slabs and walls by selecting the required result from droplist beneath **Results** in the **2D Results** group.

The first 6 values are calculated directly from the forces and moments:

1. σ_x top - in-plane axial stress in the x direction, top surface
2. σ_y top - in-plane axial stress in the y direction, top surface
3. τ_{xy} top - in-plane shear stress in xy direction, top surface
4. σ_x bottom - in-plane axial stress in the x direction, bottom surface
5. σ_y bottom - in-plane axial stress in the y direction, bottom surface
6. τ_{xy} bottom - in-plane shear stress in xy direction, bottom surface

NOTE The above 6 values are available for loadcases & combinations, but not envelopes

The next 4 values are determined from the first 6 values:

1. σ_x max tension - maximum tension stress in the x direction for both surfaces
2. σ_y max tension - maximum tension stress in the y direction for both surfaces
3. σ_x max compression - maximum compression stress in the x direction for both surfaces
4. σ_y max compression - maximum compression stress in the y direction for both surfaces

NOTE The above 4 values are available for loadcases, combination & envelopes.

The center stress values are calculated directly from the forces and moments:

1. σ_x in-plane - in-plane axial stress in the x direction, center
2. σ_y in-plane - in-plane axial stress in the y direction, center

The last 4 values are determined from the above 2 values:

1. σ_x in-plane tension - maximum tension stress in the x direction center
2. σ_y in-plane tension - maximum tension stress in the y direction center
3. σ_x in-plane compression - maximum compression stress in the x direction center
4. σ_y in-plane compression - maximum compression stress in the y direction center

How might these results be used?

Users performing the design of structures with concrete core walls are interested to know which panels within the walls are cracked. Which panels are cracked can be determined by comparing the maximum tensile stress in each panel to the concrete tensile strength.

Tekla Structural Designer calculates stress values from the gross section properties (ignoring the reinforcement). To determine cracked panels, you can see the maximum tension (and compression) stress in each direction for each panel, across loadcases, combinations and envelopes.

Calculation of in-plane axial and shear stress

For loadcases and combinations, by using the 2D element thickness, stresses (based on the gross section properties) can be calculated from the forces at the nodes:

$\sigma_x top = F_x / t + 6M_x / t^2$

$\sigma_y top = F_y / t + 6M_y / t^2$

$\tau_{xy} top = F_{xy} / t + 6M_{xy} / t^2$

$\sigma_x bottom = F_x / t - 6M_x / t^2$

$\sigma_y bottom = F_y / t - 6M_y / t^2$

$\tau_{xy} bottom = F_{xy} / t - 6M_{xy} / t^2$

$\sigma_x in\text{-}plane = F_x / t$

$\sigma_y in\text{-}plane = F_y / t$

NOTE • Tension stresses are positive

- Compression stresses are negative

The process for enveloping the above values is the same as that used for other envelopes. For each of the items, a pair of values is found, these are the minimum & maximum values across all loadcases and combinations.

Calculation of maximum tension and compression stress for loadcases and combinations

For loadcases and combinations, the maximum tension and compression values are determined for a specific direction by finding the maximum or minimum of the top and bottom stresses in that direction:

σ_x max tension = Max ($\sigma_x$top, $\sigma_x$bottom, 0.0)

σ_y max tension = Max ($\sigma_y$top, $\sigma_y$bottom, 0.0)

σ_x max compression = Min ($\sigma_x$top, $\sigma_x$bottom, 0.0)

σ_y max compression = Min ($\sigma_y$top, $\sigma_y$bottom, 0.0)

σ_x in-plane tension = Max ($\sigma_x$in-plane, 0.0)

σ_y in-plane tension = Max ($\sigma_y$in-plane, 0.0)

σ_x in-plane compression = Min ($\sigma_x$in-plane, 0.0)

σ_y in-plane compression = Min ($\sigma_y$in-plane, 0.0)

For envelopes, the maximum tension and compression values are determined by applying the above equations to the enveloped values. Envelopes yield two values for each of the 4 entries in the dropdown.

Calculation of maximum tension and compression stress for envelopes

For envelopes, the maximum tension and compression values are determined by applying the above equations for loadcases and combinations to the enveloped values.

Envelopes yield two values for each of the 4 entries in the droplist.

For **tension stresses** (x or y - only x shown for brevity) the values are returned are:

σ_x max tension = m_1 / m_2 , where:

m_1 = Min ($\sigma_x$top $_{\text{max across all cases \& combs}}$, $\sigma_x$bottom $_{\text{max across all cases \& combs}}$, 0.0)

m_2 = Max ($\sigma_x$top $_{\text{max across all cases \& combs}}$, $\sigma_x$bottom $_{\text{max across all cases \& combs}}$, 0.0)

For **compression stresses** (x or y - only x shown for brevity) the values returned are:

σ_x max compression = m_1 / m_2 , where:

m_1 = Min ($\sigma_x$top $_{\text{min across all cases \& combs}}$, $\sigma_x$bottom $_{\text{min across all cases \& combs}}$, 0.0)

m_2 = Max ($_x top_{\text{ min across all cases \& combs}}$, $\sigma_x bottom_{\text{ min across all cases \& combs}}$, 0.0)

In summary the values visible in the tooltip are:

σ_x max tension	=	Min ($\sigma_x top_{\text{max across all cases \& combs}}$, $\sigma_x bottom_{\text{max across all cases \& combs}}$, 0.0) / Max ($\sigma_x top_{\text{max across all cases \& combs}}$, $\sigma_x bottom_{\text{max across all cases \& combs}}$, 0.0)
σ_y max tension	=	Min ($\sigma_y top_{\text{max across all cases \& combs}}$, $\sigma_y bottom_{\text{max across all cases \& combs}}$, 0.0) / Max ($\sigma_y top_{\text{max across all cases \& combs}}$, $\sigma_y bottom_{\text{max across all cases \& combs}}$, 0.0)
σ_x max compression	=	Min ($\sigma_x top_{\text{min across all cases \& combs}}$, $\sigma_x bottom_{\text{min across all cases \& combs}}$, 0.0) / Max ($\sigma_x top_{\text{min across all cases \& combs}}$, $\sigma_x bottom_{\text{min across all cases \& combs}}$, 0.0)
σ_y max compression	=	Min ($\sigma_y top_{\text{min across all cases \& combs}}$, $\sigma_y bottom_{\text{min across all cases \& combs}}$, 0.0) / Max ($\sigma_y top_{\text{min across all cases \& combs}}$, $\sigma_y bottom_{\text{min across all cases \& combs}}$, 0.0)

Key points when using stress values

1. In Walls X direction is horizontal in plane of wall and Y is vertical
2. “Top”/”Bottom” is dependent on shell local axis system, but if you are only concerned about max values you don’t need to worry about this - use the “max” options which consider both faces.
3. For engineers wanting to consider tensile stresses in walls the “σ_y in-plane tension” option will be of greatest interest.

- This is based purely on the *membrane* tension stress (i.e. ignoring out of plane bending effects).
- This can be viewed for enveloped results
- It is very easy to see walls/panels in which no tension stress is developing.
- In a full 3D view it may be difficult to assess whether a particular cracking stress level is exceeded. Viewing results in 2D views or sub structures may be helpful here.
- It should be clear that this is based on the concrete section only - reinforcement content is not considered.

See also

Display 2D results (page 504)

Display 2D deflections

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **2D Results** group, select **Deflections**, then from the droplist beneath select the desired deflection type.
4. From the **Text** group select **Deflection** to display values on the diagram.

See also

Customize the display of 2D contours (page 522)

Change result diagram scale settings (page 523)

Animate 1D and 2D deflections (page 502)

Display AsReq contours

AsReq contours can either be displayed as values, or as a pass/fail threshold. Displaying as a pass/fail threshold helps to visualize minimum patch sizes when optimizing panel and patch reinforcement as it highlights the specific areas in which the existing reinforcement is not sufficient.

1. Go to the **Results** toolbar.
2. Set the analysis type and loading for viewing analysis results (page 500).

   NOTE Typically the analysis method will be FE chase-down

3. In the **2D Results** group , select **As Req**.
4. From the first droplist, select the desired direction and slab face.

5. From the second droplist, select **Value**, or **Pass/Fail** as required.

See also

Customize the display of 2D contours (page 522)

Change result diagram scale settings (page 523)

Display wall lines

Tekla Structural Designer automatically creates a wall line at the centroid of every meshed shear wall to facilitate the display of wall forces.

1. In the **Status bar**, click **Results View**.

   The **Results** toolbar opens.

2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **2D Integrated Results** group, select **Wall Lines**, then from the droplist in the same group, select the desired result.
4. From the **Text** group select **Forces** to display values on the diagram.

   Tekla Structural Designer displays the selected result on the wall line.

   TIP If you are working in a 2D view and you can't see any results, try displaying the view in isometric projection (page 524).

Display core lines

Tekla Structural Designer automatically creates a core line at the centroid of each concrete core to facilitate the display of core forces.

1. In the **Status bar**, click **Results View**.

   The **Results** toolbar opens.

2. Set the analysis type and loading for viewing analysis results (page 500).
3. In the **2D Integrated Results** group, select **Core Lines**, then from the droplist in the same group, select the desired result.
4. From the **Text** group select **Forces** to display values on the diagram.

   Tekla Structural Designer displays the selected result on the core line.

   NOTE All forces in the results are rotated to be in the axis system of the core line.

TIP If you are working in a 2D view and you can't see any results, try displaying the view in isometric projection (page 524).

See also

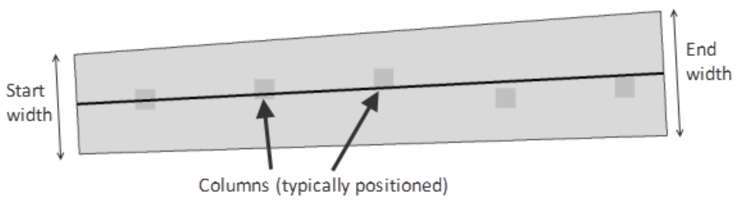
Create concrete cores (page 245)

Manage and display result strips

You can place user-defined result strips across 2D element meshes. For these strips, Tekla Structural Designer determines the force and moment results from the shell/plate/membrane nodal analysis results. These results can then be used for design purposes, typically for slab design.

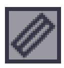
Engineering judgement is required when positioning the strips to ensure suitable design forces are obtained.

By default, strips have parallel edges, but tapering strips can also be defined as shown in the following image:



Create result strips

1. Open a 2D view of the FE mesh where you want to place the strip.
2. In the **Status bar**, click  **Results View**.

   The **Results** toolbar opens.
3. In the **2D Integrated Results** group, click  **Result Strip**.
4. In the **Properties** window, adjust the result strip properties according to your needs.
5. Click the start point of the strip.
6. Click the end point of the strip.

   TIP Neither the start or the end point have to match nodes in the mesh.

   Tekla Structural Designer creates a strip between the selected points.
7. Do one of the following:

- Continue placing further strips.
- Press **Esc** to exit the command.

View the results for result strips

Once you have created a result strip in the model, you can obtain results for it without re-running the analysis. In order to view the results, do the following:

1. Open a 3D view containing the strips whose results you want to view.
2. Go to the **Results** toolbar.
3. In the first list of the **Result Type** group, select the desired result type.
4. In the **Loading** list, select the loadcase or combination that you want to display.
5. In the **2D Integrated Results** group, click the effect that you want to view.

   Tekla Structural Designer displays the selected effect on the strip, and its maximum positive and negative values, calculated according to the result type specified in the strip properties.

NOTE Tekla Structural Designer contains the following three ways to calculate the results:

Method name	Details
Normal	• The results on the center of the result strip are calculated at each station*. • Tekla Structural Designer considers the shell elements local to each station, and calculates a weighted average force based on the distance of the element nodes from the station. The process is repeated for all stations along the center line of the strip to give the results.
Maximum	• The results on the transverse line across the strip are calculated for each station* along the strip. • Tekla Structural Designer considers the shell elements local to each point, and calculates a weighted average

Method name	Details
	force based on the distance of the element nodes from the point. The maximum result across the strip from all points is taken as the result for the station on the strip center line. The process is repeated for all stations along the center line of the strip to give the results. • The values calculated at points are always weighted averages of results at adjacent nodes. Therefore, they are always less than the peak nodal values. • Maximum values include nodes within the strip.
Average	• Average over strip width. • The results are obtained in the same way as for the maximum option, but in this case, they are averaged to give the results for each station*. The process is repeated for all stations along the center line of the strip to give the results. •

* Along the strip center line, there is a user-defined number of stations. At each station, there is a transverse line with a user defined number of points along it. Final results are always given by station, and obtaining them may or may not use points.

All forces in the results are rotated to be in the axis system of the result strip.

Delete result strips

TIP To delete a strip, ensure that Result Strips are switched on in **Scene Content**.

1. Open a view containing the strip that you want to delete.

2. In the **Quick Access** toolbar, click **Delete**.
3. In the model, click the strip that you want to delete.

   Tekla Structural Designer deletes the selected strip.

See also

Change result diagram scale settings (page 523)

Display 2D view in isometric projection (page 524)

Manage, display and design result lines

You can place user-defined result lines across 2D element meshes. For these lines, Tekla Structural Designer determines the force and moment results from the shell/plate/membrane nodal analysis results. These results can then be used to assess design solutions, typically for the design of wall panels with openings.

Engineering judgement is required when positioning the lines to ensure suitable design forces are obtained.

Create result lines

1. Open a 2D view containing the wall or slab within which you want to place the strip.
2. In the **Status bar**, click **Results View**. The **Results** toolbar opens.
3. In the **2D Integrated Results** group, click **Result Line**.
4. Click the start point of the line.
5. Click the end point of the line.

   TIP Neither the start or the end point have to match nodes in the mesh.

   Tekla Structural Designer creates a result line between the selected points.
6. Do one of the following:
   - Continue placing further result lines.
   - Press **Esc** to exit the command.

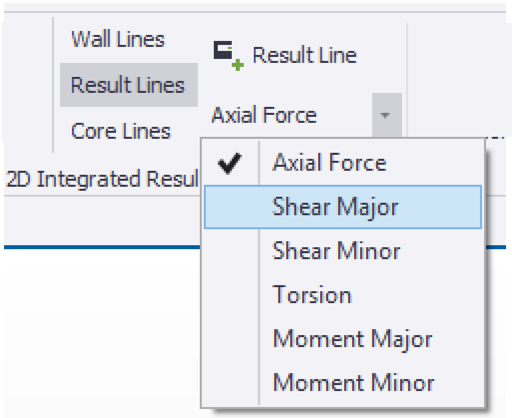
View the analysis results for result lines

NOTE An analysis must be performed after adding or editing result lines in order to obtain up to date results.

To view the results, do the following:

1. Open a view containing the result lines whose results you want to view.
2. In the **Status bar**, click  **Results View**.

   The **Results** toolbar opens.
3. Set the analysis type and loading for viewing analysis results (page 500).
4. In the **2D Integrated Results** group, click **Result Lines**, then from the droplist in the same group, select the desired result.
5. From the droplist in the **2D Integrated Results** group, select the effect that you want to view.



6. From the **Text** group select **Forces** to display values on the diagram.

   Tekla Structural Designer displays the selected result on the result line.

NOTE All forces in the results are rotated to be in the axis system of the result line.

TIP If you are working in a 2D view and you can't see any results, try displaying the view in isometric projection (page 524).

See also:

View tabular results for result lines (page 594)

Interactively design result lines

1. Open a view containing the result line to be designed.
2. Hover the mouse pointer over the result line that you want to design.

   The **Select Entity** tooltip appears.
3. In the **Select Entity** tooltip, navigate to the result line (RL) name by using the arrow keys.

4. Right-click the result line.
5. In the context menu, select **Interactive Design...** (as a column section, or as a wall section as required).

   The selected (column or wall) interactive design dialog is displayed with the Additional Design Cases page populated with the result line results.
6. Modify the reinforcement parameters as required to achieve a satisfactory design, and click the **Check** button to examine the detailed design results.
7. Click **OK** to save the designed reinforcement to the result line.

   NOTE For help on using the interactive dialogs, see and .

Features of result line design

- The interactive design dialog features full interactive manual selection of both lateral and vertical reinforcement, interaction diagrams and auto-design and check options, just as for a regular wall/ column section interactive design.
- The wall length/ column depth considered is the Result Line length.
- Result lines are not constrained to cross sections in the horizontal plane - vertical sections can be used above/between openings to investigate forces and reinforcement requirements in “coupling beams”.
  - The section design always considers the main bars (running perpendicular to the cross section) as being on the inner layer, from a design perspective this will tend to be conservative but the engineer should give this some consideration when working with non-horizontal sections.
  - The design does not consider the existing reinforcement specified in wall properties - only that which is defined in the Interactive design dialog.
  - Currently this feature is not linked with Reports and so it is envisaged output will be via screenshots of the interactive design and check results dialogs.
- The same result line can be used to interactively design as a column, and as a wall. Both designs are saved to the result line independently.

Delete result lines

TIP To delete a result line, ensure that Result Lines are switched on in **Scene Content**.

1. Open a view containing the result line that you want to delete.
2. In the **Quick Access** toolbar, click **Delete**.

3. In the model, click the result line that you want to delete.

   Tekla Structural Designer deletes the selected result line.

Related video

Interactive design using Result lines

Display mode shapes

When Tekla Structural Designer has performed several analysis types, the results of each analysis are held separately. Therefore, there is no need to re-perform a particular analysis to recall its results.

1. Open a suitable 2D or 3D View and change the view regime (page 89) to a **Results View**.
2. On the **Results** tab, in the **Result Type** group, select the required analysis type.
3. In the **Loading** list, select the required loadcase or combination.

   > **NOTE** For RSA seismic analysis, only select load cases, as mode shapes are not applicable for combinations, and therefore cannot be viewed.

4. In the second list of the **Result Type** group, select the desired mode.
5. Select the diagram that you want to display.

See also

The Results View (page 498)

RSA seismic results

For information on how RSA seismic results are displayed for different loadcases and combinations, see the following paragraphs.

Result Type

When the result type is 1st or 2nd order RSA seismic, the results that can be displayed depend on the type of the currently selected loadcase or combination.

Mode Shapes

Mode shapes can be displayed for:

- RSA Seismic loadcases:
  - Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**

- All modes that are relevant for the selected loadcase
- Effective seismic weight combination:
  - List of all modes returned by the modal analysis

Mode shapes are not displayed for:

- RSA torsion loadcases
- Static loadcases included in the RSA seismic combination
- RSA seismic combinations

See also

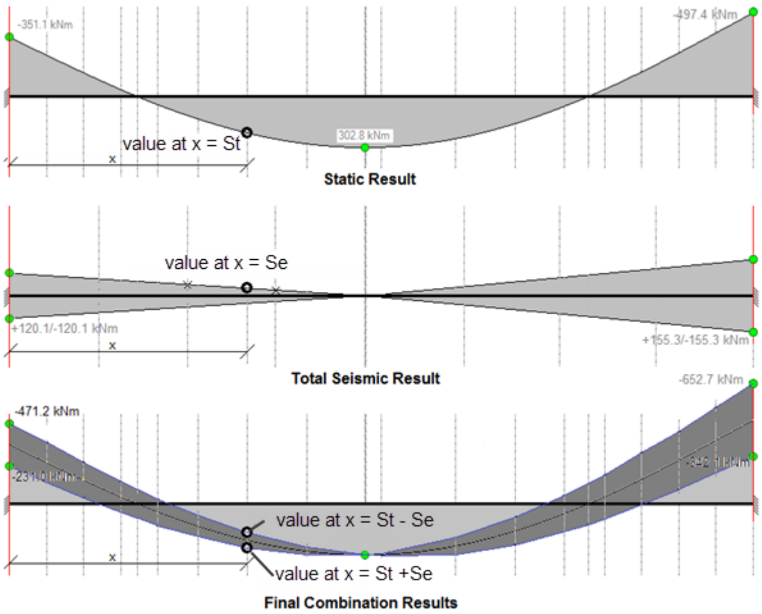
- Display mode shapes (page 518)

1D Element Results

1D element results (and Load Analysis View results) are displayed as follows:

Loadcase or combination	Display method
RSA seismic loadcases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at various points along each member and displayed on both the positive and negative side of the diagram. Therefore, the diagrams are always symmetrical about the base line. All relevant modes: A standard enveloped diagram is displayed.
RSA torsion loadcases	Displayed as per 1st order linear analysis.
Static loadcases included in the RSA seismic combination	Displayed as per 1st order linear analysis.
Effective seismic weight combination	Not available
RSA seismic combinations	An envelope is drawn displaying the seismic results above and below the static result: • Baseline goes through the static values • Top line = static value + seismic value

Loadcase or combination	Display method
	• Bottom line = static value - seismic value



See also

- Display 1D results (page 502)

Story Shear

Story shears are displayed as follows:

Loadcase or combination	Display method
RSA seismic loadcases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at each position of interest and the result is displayed as both positive and negative. All relevant modes: A standard diagram with a single value at each point of interest is displayed.
RSA torsion loadcases	Displayed as per 1st order linear analysis.

Loadcase or combination	Display method
Static loadcases included in the RSA seismic combination	Displayed as per 1st order linear analysis.
Effective seismic weight combination	Not available
RSA seismic combinations	The diagram displays two values at each point of interest: • Static value + seismic value • Static value - seismic value

See also

- Display sway drift and story shear (page 503)

Support Reactions

Support reactions are displayed as follows:

Loadcase or combination	Display method
RSA seismic loadcases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at each support and the result is displayed as both positive and negative. All relevant modes: A standard diagram is displayed.
RSA torsion loadcases	A standard diagram is displayed.
Static loadcases included in the RSA seismic combination	A standard diagram is displayed.
Effective seismic weight combinations	Not available
RSA seismic combinations	The diagram displays two values at each support: • Static value + seismic value • Static value - seismic value

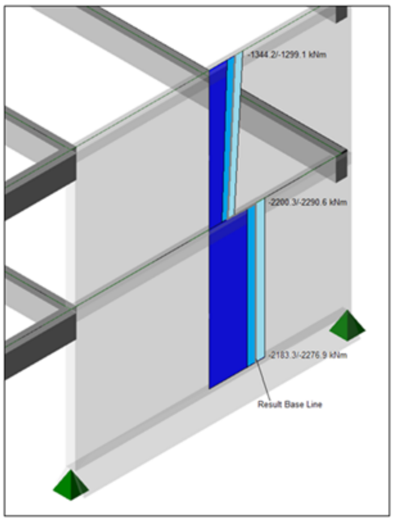
See also

- Display reactions (page 500)

Concrete Wall Results

Concrete wall results are displayed as follows:

Loadcase or combination	Display method
RSA seismic loadcases	Combined (CQC) or combined (SRSS), depending on your choice in **Analysis Settings**: Absolute values are determined at points along the wall line and displayed on both the positive and negative side of the diagram. Therefore, the diagrams are always symmetrical about the wall line. All relevant modes: A standard diagram is displayed.
RSA torsion loadcases	A standard diagram is displayed.
Static loadcases included in the RSA seismic combination	A standard diagram is displayed.
Effective seismic weight combinations	Not available
RSA seismic combinations	An envelope is drawn displaying the seismic results above and below the static result: • Baseline goes through the static values • Top line = static value + seismic value • Bottom line = static value - seismic value



Customize the display of 2D contours

By default, all contour diagrams consist of 10 evenly sized contours, each accounting for 10% of the total range. If necessary, you can increase or decrease the number of contours, and also change the size and the color of individual contours.

1. On the **Home** tab, click **Settings**.

   The **Settings** dialog box opens.

   NOTE If you intend to make the changes to the contours in the current model, ensure that you are making changes to the active settings set.

2. Go to **Scene** --> **Contours** .
3. According to your needs, do one or more of the following:

To	Do this
Add new contours	• Click **Split**.
Delete existing contours	• Click **Delete**.
Modify the size of contours	• In the **Size [%]** column, type new values in the appropriate cells.
Change the color of contours	a. In the **Color** column, click the color that you want to change. b. Select a new color. c. Click **OK**.
Revert to the default contour settings	• Click **Reset**.

4. Click **OK**.

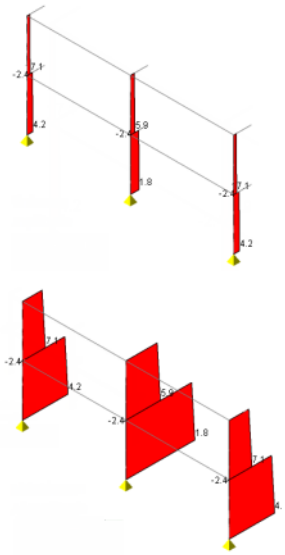
See also

The Results View (page 498)

Change result diagram scale settings

- Move the sliders in the **Scale Settings** group according to adjust the diagram amplitude according to your needs.

The below example shows the effect on an axial force diagram when the 1D Results slider is increased from minimum to maximum.



See also

The Results View (page 498)

Display 2D view in isometric projection

Because the diagrams are plotted on each element in the planes in which they act when you are working in a 2D view, you will need switch on an isometric display to see the out of plane forces.

If the 2D view is currently displayed in plan, the **2D**/**3D** toggle button at the bottom right corner of the screen is labeled **3D**.

- To change the current view, click the **2D**/**3D** toggle button.

See also

The Results View (page 498)

Sign conventions and coordinate systems

Tekla Structural Designer adopts the standard convention that lower case x, y, and z represent local coordinate systems, whereas upper case X, Y, and Z represent the global coordinate system. For more information on analysis result sign conventions, see the following paragraphs.

Axis systems

The following table presents the axis systems that can be used in Tekla Structural Designer:

Axis system name	Description
Global coordinate system	The global XYZ axis system within which all other systems exist.
Building directions 1 and 2	The principle axes of the building, where dir 1 is rotated at an angle to global X in the horizontal plane.
User coordinate system	A local coordinate system defined by the system or the user.
1D member local coordinate system	The local coordinate system that is applicable to all 1D members, such as beams, columns, and braces.
Mid-pier wall coordinate system	The local coordinate system that is applicable to walls modeled using the mid-pier option.
2D member local coordinate system	The local coordinate system that is applicable to all 2D members, walls, and slabs.
Result line coordinate system	The local coordinate system that is applicable to result lines.
Result strip coordinate system	The local coordinate system that is applicable to result strips.
Foundation reaction coordinate system	The local coordinate system that is applicable to foundations.

General information

All global (XYZ) and local (xyz) axis systems follow the right-hand rule, where:

- x axis is the pointing index finger.
- y axis is the crooked middle finger.
- z axis is the extended thumb.

In the directions of positive rotation:

- About x: the y axis moves toward the z axis.
- About y: the z axis moves toward the x axis.
- About z: the x axis moves toward the y axis.

Object orientation

Tekla Structural Designer considers the orientation of the object when displaying the analysis results. Therefore, to apply the sign convention correctly, you need to know which is the end 1 and which is the end 2 of beams or walls, and which is the face A of columns.

If you select the **Direction** option for an element in **Scene Content**, Tekla Structural Designer displays an arrow on all beams, walls and columns. This arrow points from the start to the end of beams and walls, and from the bottom to the top of columns along the face A. Looking down from the top of a column, Face B, C, and D then follow in the clockwise direction.

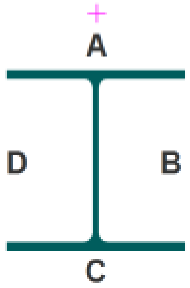
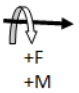
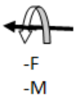


Diagram conventions

All arrows should point in the direction of the force or moment, as the following image illustrates:

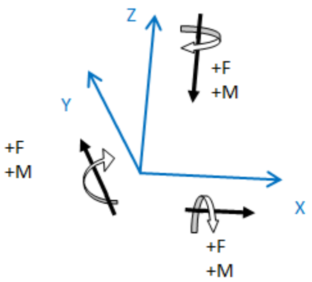


If the arrows are reversed, they become negative forces and moments, as the following image illustrates:

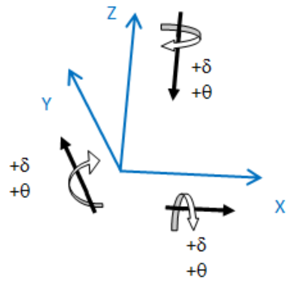


Global coordinate system

The following image illustrates the global axis system and applied load directions.



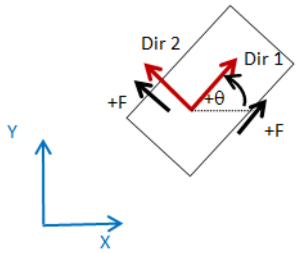
The resulting deflection directions appears as follows.



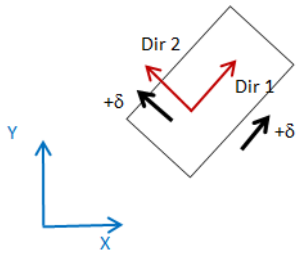
Building directions 1 and 2

Global axes (positive Z vertically up) and angle between X and direction 1 is θ, where θ is positive in right-hand rule about Z.

The following image illustrates the building directions and applied load directions.



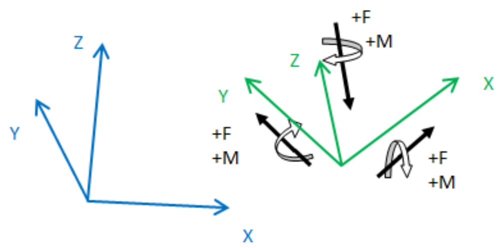
The resulting deflection directions appear as follows.



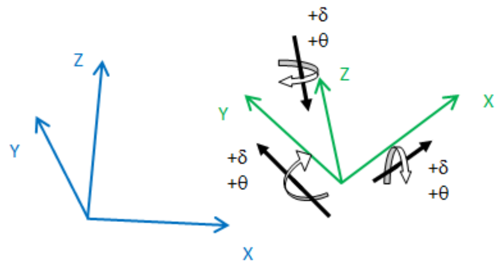
User coordinate system

A user coordinate system can be at any angle to the global coordinate system.

The following image illustrates the axis system of a user coordinate system axis and applied load directions.



The resulting deflection directions appear as follows.



NOTE Every support is given a user coordinate system. Automatically created supports under certain objects default to the following method:

- Support under a single column or wall rotates the foundation forces to align with the y/z-axes of the column or wall
- Support under a mat foundation - uses the global coordinate system.

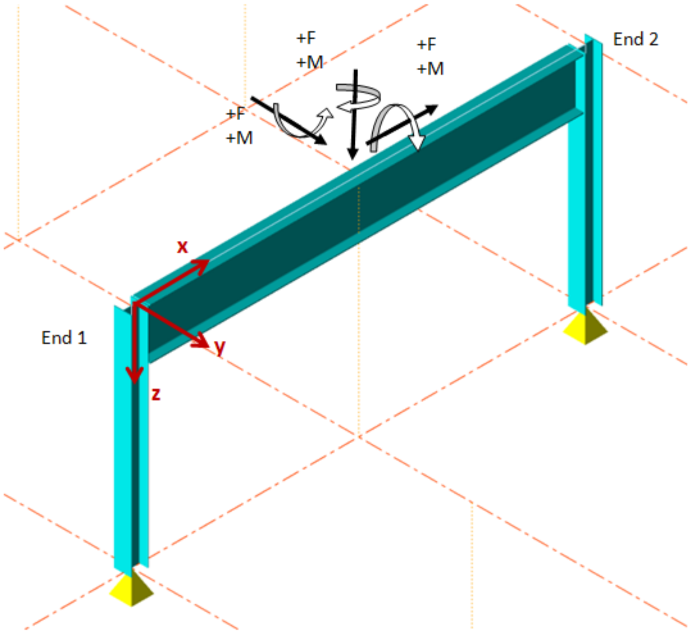
All other supports default to the global coordinate system.

1D member local coordinate system (general case)

Local axis system and applied load directions:

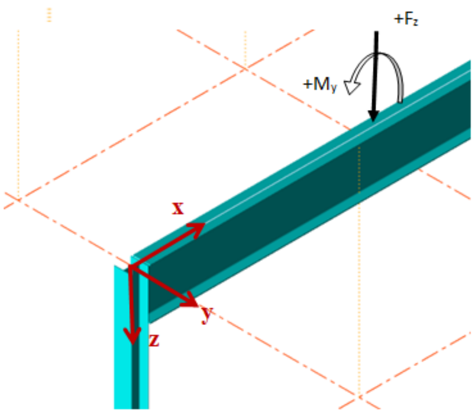
- The local x axis along member starts from end 1 and ends in end 2
- When y = 0:
  - The local z axis lies in the plane created by the local x axis and the global Z axis.
  - The global Z component of the local z axis is always negative.
  - The local y axis follows the right-hand rule.

  y = positive clockwise rotation of y and z axes about the x axis looking towards positive x.

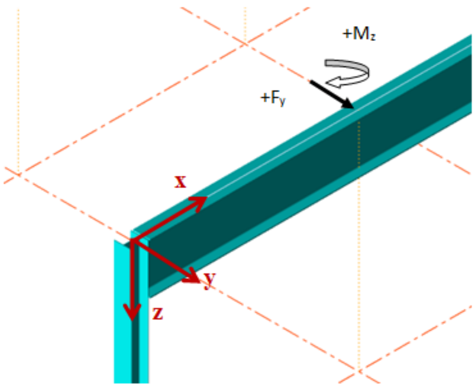


Applied force directions:

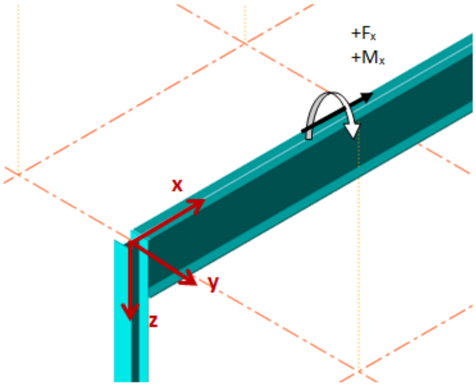
- z = Major (Fz and My):



- y = Minor (Fy and My):



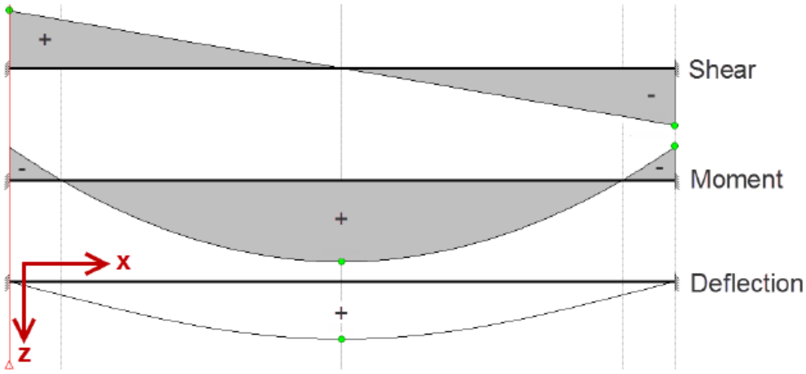
- x = Axial:



Result axis system and directions

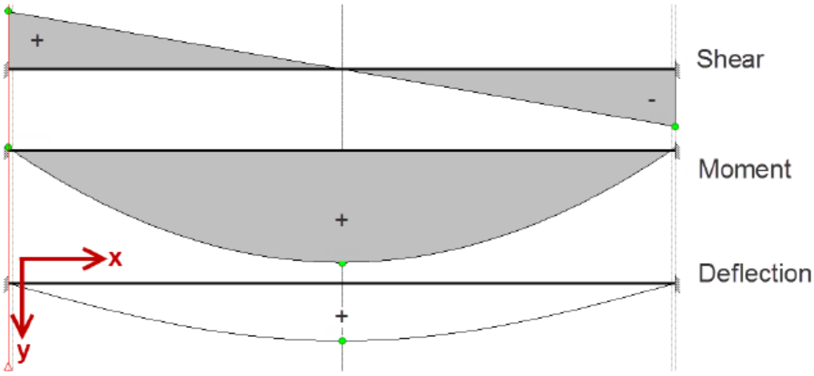
In the major axis:

- Moment major = bending about the y axis
- Shear major = shear along the z axis

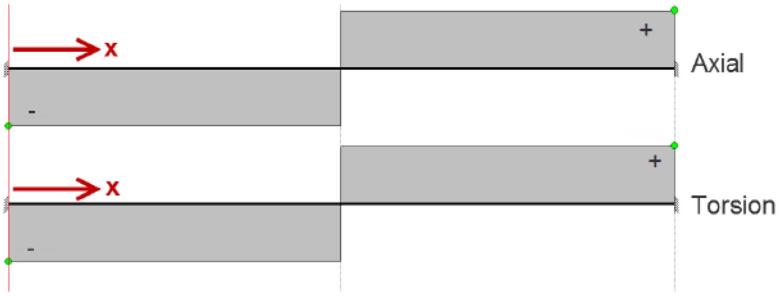


In the minor axis:

- Moment minor = bending about the z axis
- Shear minor = shear along the y axis

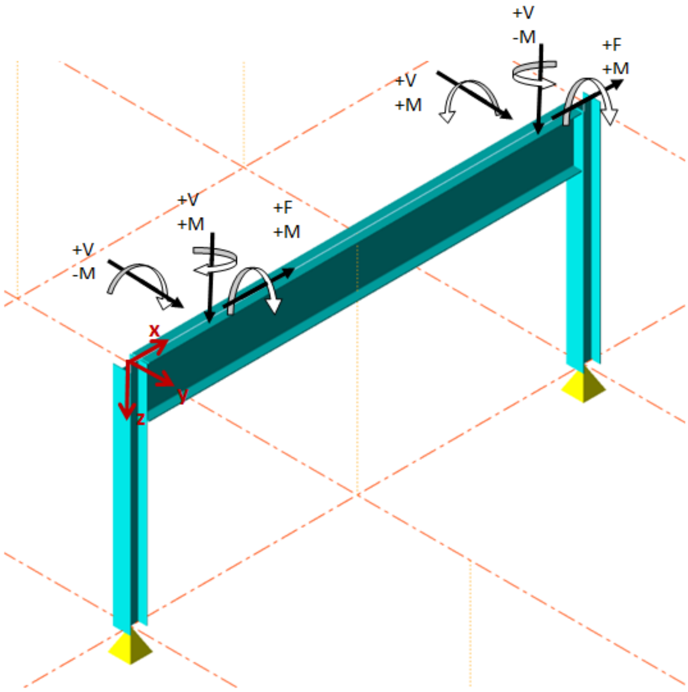


In the axial direction:



Resulting member end forces and directions

Member end forces are the forces applied to the rest of the structure by the member. Based on loading applied above, the forces would be applied as follows:



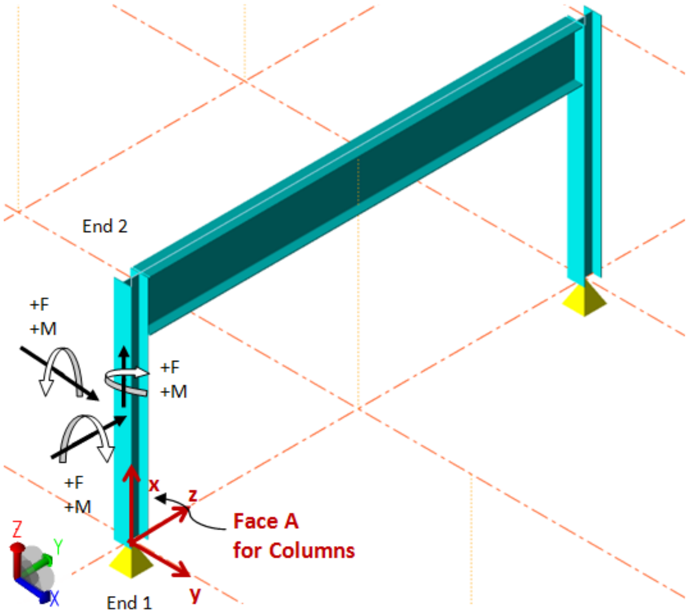
1D member local coordinate system (vertical members)

Local axis system and applied load directions

Local x aligns with global Z (vertical):

- When γ = 0:
    - The local y axis aligns with global X.
    - The local z axis according to the right-hand rule.

  γ = positive clockwise rotation of y and z about the x axis towards positive x.



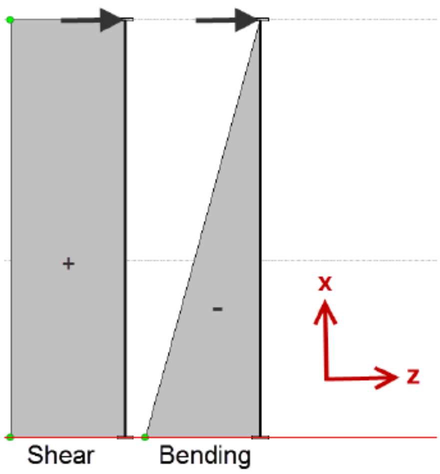
Applied force directions as displayed in the previous image:

- z = Major
- y = Minor
- x = Axial

Result axis system and directions

In the major axis:

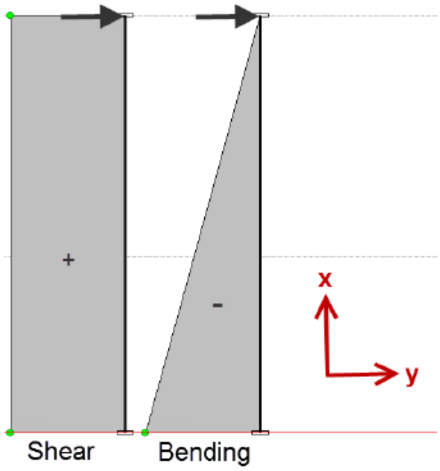
- Moment major = bending about the y axis
- Shear major = shear along the z axis



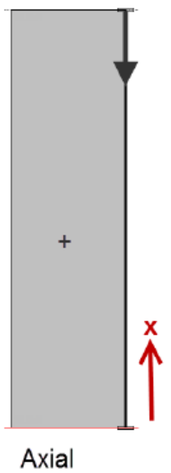
In the minor axis:

- Moment minor = bending about the z axis

- Shear minor = shear along the y axis



In the axial direction:

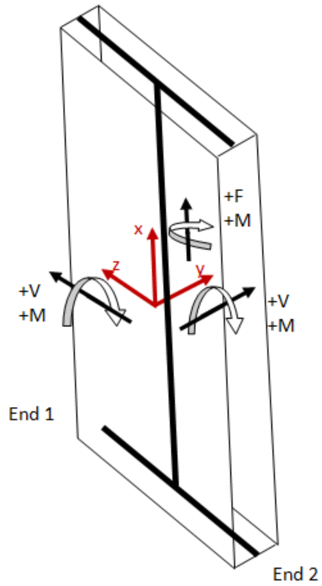


Mid-pier wall coordinate system

Wall axis system and applied load directions

As the following image illustrates, centered on the centroid of the cut section:

- the x axis lies along the stem mid-pier element (positive lowest to highest)
- the z axis lies along the plane of the wall (positive end 2 to end 1)
- the y axis follows the right-hand rule and is normal to the wall.

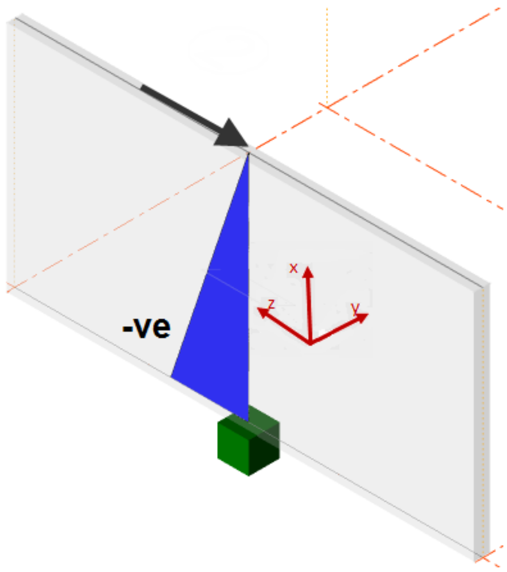


The results from a mid-pier model are in the same axis system as the result line in a meshed wall.

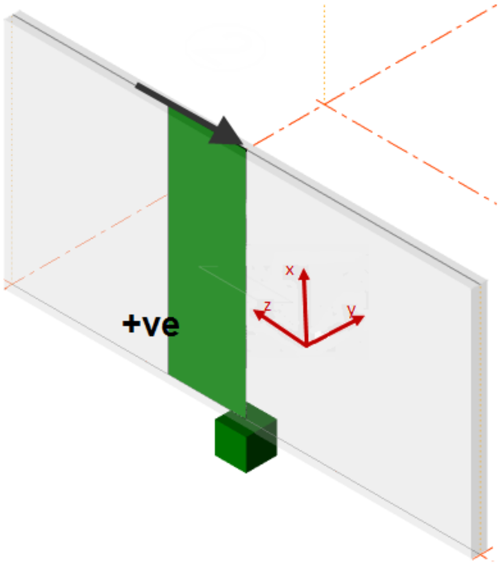
Result axis system and direction

In the major axis:

- Moment major = bending about the y axis:

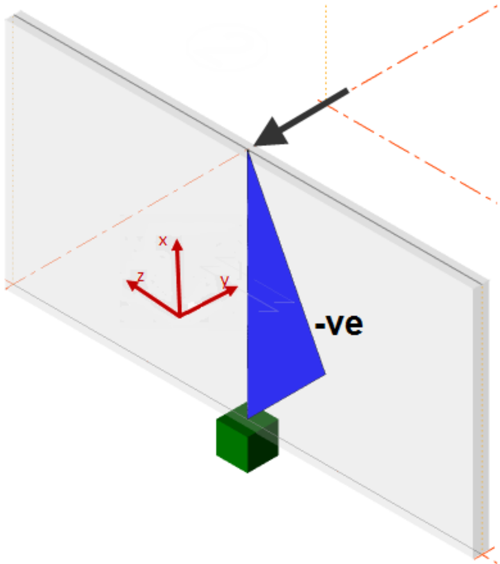


- Shear major = shear along the z axis:

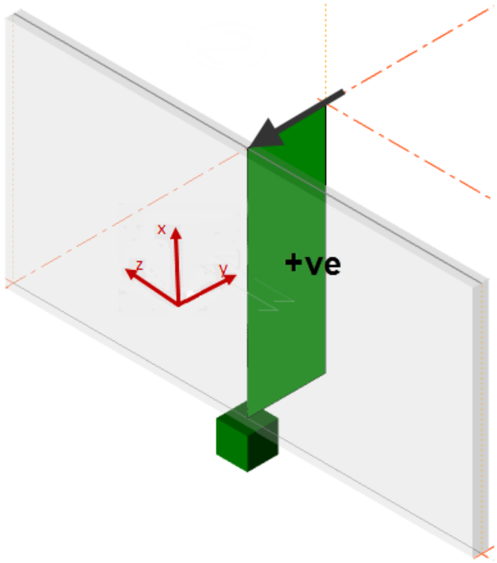


In the minor axis:

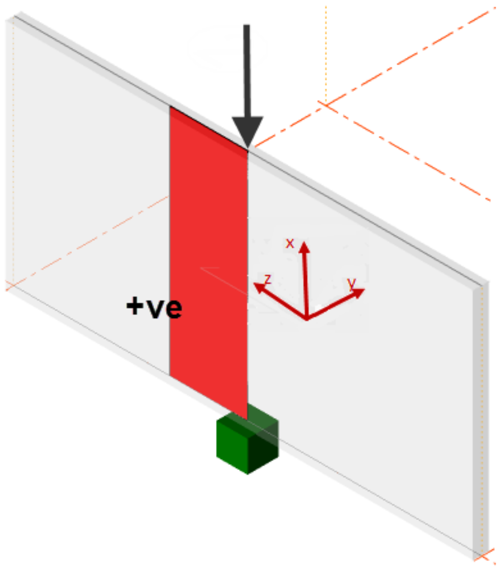
- Moment minor = bending about the z axis:



- Shear minor = shear along the y axis:



In axial and torsion, force is in the x axis and torsion about the x axis:



2D member local coordinate system

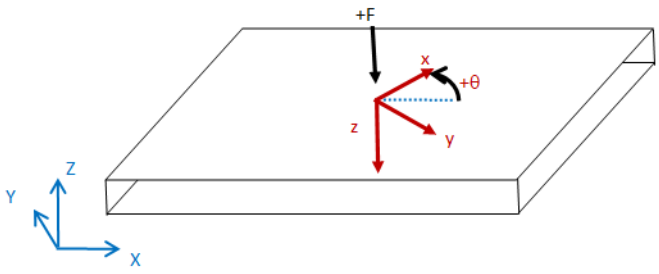
Horizontal panel local axis system and applied load directions

Horizontal panel local axes are the following:

- The local z axis is normal to the plane of the panel
- When θ = 0:
    - The local x axis plane is in the plane of the panel, aligned with the global X axis and positive in the positive global X direction.

- The local y axis is in the plane of the panel and follows the right-hand rule.

θ = positive clockwise rotation of the x and y axis about the z axis looking towards positive z.



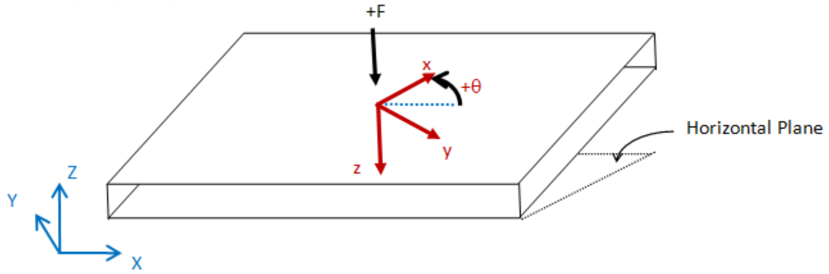
Vertical and sloped panel local axis system and applied load directions

Vertical and sloped panel local axes are the following:

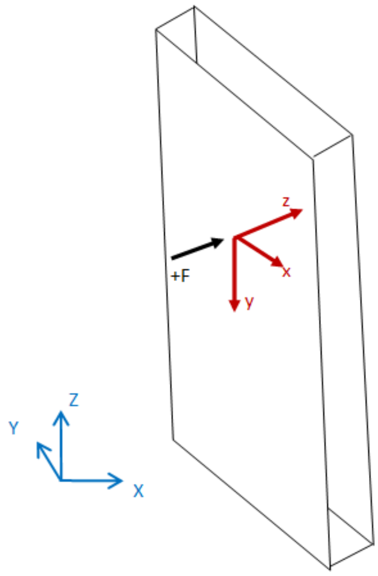
- The local z axis is normal to the plane of the panel.
- When θ = 0:
  - The local x axis plane is in the plane of the panel and in a horizontal plane.
  - The local y axis is in the plane of the panel and follows the line of greatest slope of the plane (positive in the direction of positive global).

  θ = positive clockwise rotation of x and y a bout z looking towards positive x.

Sloped panel (axes at θ):

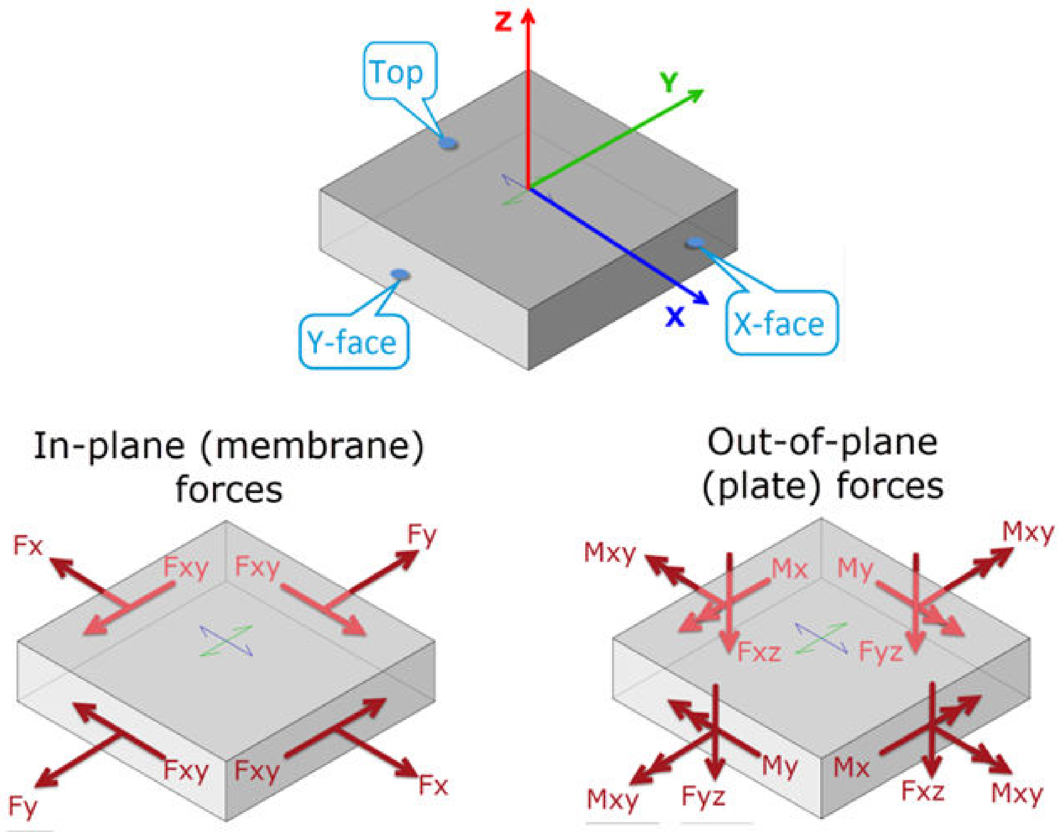


Vertical panel (axes when q = 0):



2D member forces sign convention

The sign convention for 2D member forces is not the same as that of 1D elements. The following diagram illustrates the forces and the panel and 2D element axis system (for results):



- The arrows in the diagram show positive force directions.
  - The double-arrow convention is used for moments: the moment is around the double-arrow, positive being clockwise when looking in the arrow direction.
  - The forces act on a member face cut anywhere in the FE mesh, perpendicular to the force direction.

  Thus, for example, Mx acts on the X face that is perpendicular to the X axis and is moment resulting from spanning in the X direction, Fxz is the out-of-plane shear force acting on the X face, and so on.
- The wood armer design moments (denoted by the *d* suffix) act in the same manner as the unprocessed moments without the *d* suffix. Thus, Mdx acts in the same manner as Mx, and so on.
- The design moments are further classified into top and bottom components for the slab design process.
- The positive Z axis direction (up) follows the right-hand rule and, therefore, is not the same as that for the 2D member local coordinate system. This is because the 2D member local coordinate system for the applied load directions displays the positive applied load direction convention that, for Z only, is opposite to the convention of the global and 2D element axes.
- A positive moment creates tension in the top surface of the shell. Therefore, the moment over a supporting column is positive, whereas the span moment is negative.
- The conventions for wall results are exactly the same as conventions for columns, so they can be interpreted in the same way.
- The compression of axial loads (Fx and Fy) is negative.
- Out-of-plane shear (Fxz and Fyz) is positive when shear is such that moment is increasing in the positive X or Y direction.

Result line coordinate system

Centered on the centroid of the cut section:

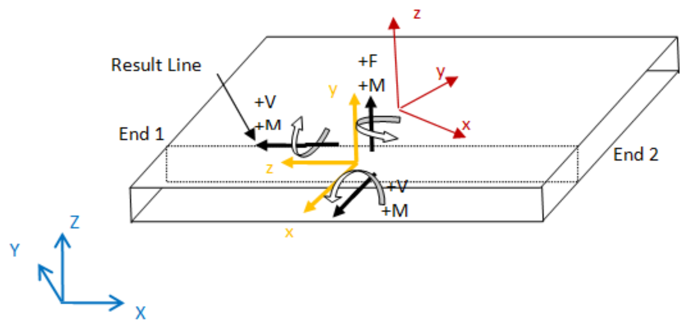
- The z axis lies along the result line (positive end 2 to end 1).
- The y axis normal to plane of mesh (generally positive in the positive Z direction, in special cases positive x towards positive Z).
- The x axis follows the right-hand rule and lies in the mesh, so x is perpendicular to the cut line.

NOTE	The results from a result line are exactly like those for a mid-pier model when the cut is horizontal and the cut direction matches the direction required.

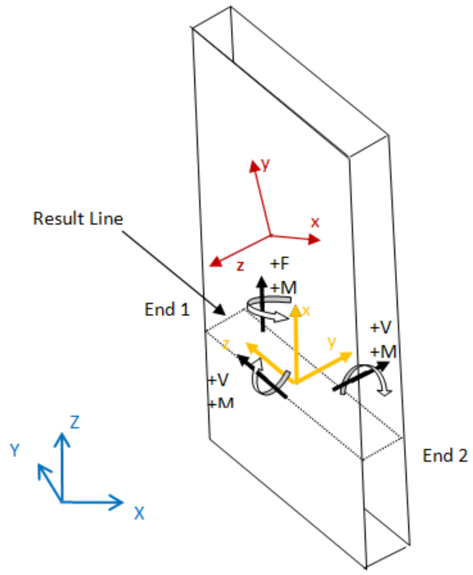
Result Axis System

- In the major axis: bending about the y axis and shear along the z axis
- In the minor axis: bending about the z axis and shear along the y axis
- Axial and torsion: force in x and bending about the x axis

General case:



Special case:



Result strip coordinate system

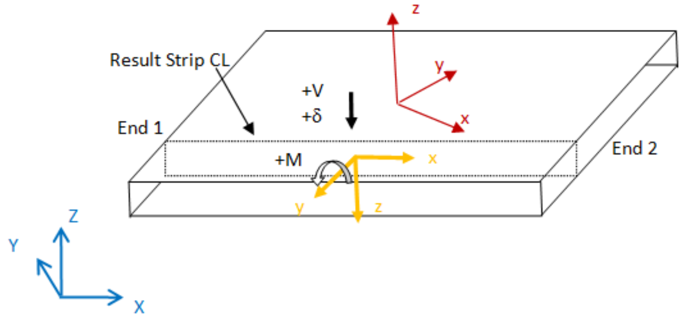
Centered at each station along the strip center line, whether there is a single or several continuous strips:

- The z axis is normal to plane of mesh (generally positive in the negative Z direction, in special cases positive x towards positive Z).
- The x axis lies along the result strip (positive end 1 to end 2)
- The y axis lies along the transverse line to the result strips and follows the right-hand rule, so the y axis is perpendicular to the strip line.

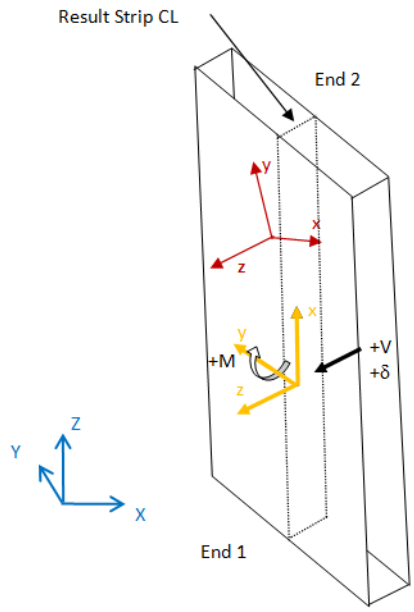
Result axis system

- Deflection in the z direction
- Out of plane moment about the y axis
- Shear in the z direction

General case:

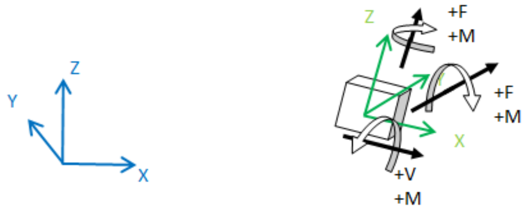


Special case:

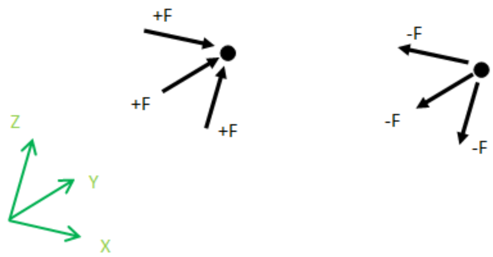


Foundation reaction coordinate system

As the following image illustrates, the foundation reaction coordinate system is aligned with the coordinate system for the support node, whether that is the global coordinate system or a user coordinate system.



Reactions are the forces applied to the structure by the foundation. They appear as follows.



The Load Analysis View

Analysis results for individual members and walls can be displayed in a Load Analysis View.

Open a Load Analysis View

1. Right-click the member whose loading and analysis results you want to view.
2. In the context menu, select **Open Load Analysis View**.

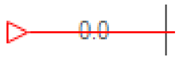
   Tekla Structural Designer opens the load analysis view and the **Loading Analysis** tab.
3. In the **Loading** list, select the loadcase or combination whose results you want to view.
4. Select the analysis type using the list in the **Result Type** group.
5. If you are viewing the results for a load combination, in the **Result Type** group, select whether you want to view the results based on strength or service factors.

6. In the **Direction** group, select the axis type (**Axial**, **Major**, or **Minor**).

   Tekla Structural Designer displays the results according to the selected options. If necessary, you can control the results in the **Properties** window.

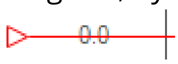
Load Analysis View properties for columns

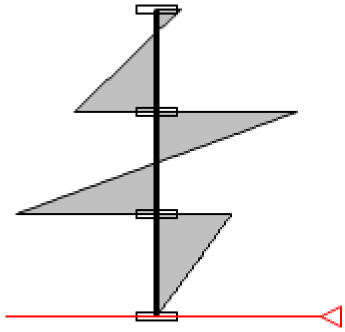
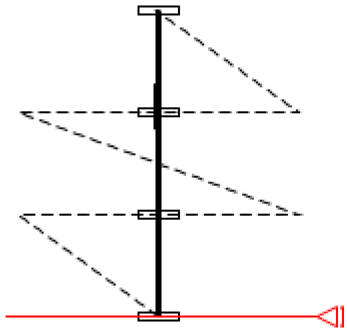
Result Type: Axial

Property	Description
Distance	The results can be reported at any position along the column, either by typing the distance directly into the **Properties** window, or, on the diagram, by dragging this slider  to the position required. **NOTE** For concrete columns only: If rigid zones have been applied, only the non-rigid length of the column is displayed in the loading analysis view.
Stack	Specifies the stack for which results are displayed.
Axial force above	The axial force in the column immediately above the cross section at the distance specified.
Axial force below	The axial force in the column immediately below the cross section at the distance specified.
Axial force reduced above	The axial force in the column immediately above the cross section at the distance specified, taking into account imposed load reductions.
Axial force reduced below	The axial force in the column immediately below the cross section at the distance specified, taking into account imposed load reductions.
Torsion moment above	The torsion in the column immediately above the cross section at the distance specified.

Property	Description
Torsion moment below	The torsion in the column immediately below the cross section at the distance specified.
Show axial force	If cleared, the axial force diagram is removed from the view.
Show axial force reduced	If cleared, the axial force diagram is removed from the view.
Show torsion moment	If cleared, the torsion diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Result Type: Major, Minor

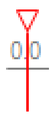
Property	Description
Distance	The results can be reported at any position along the column, either by typing the distance directly into the **Properties** window, or, on the diagram, by dragging this slider  to the position required. **NOTE** For concrete columns only: If rigid zones have been applied, only the non-rigid length of the column is displayed in the loading analysis view.
Stack	Specifies the stack for which results are displayed.
Shear above	The major or minor shear force immediately above the cross section at the distance specified.
Shear below	The major or minor shear force immediately below the cross section at the distance specified.

Property	Description
Moment above	The major or minor moment immediately above the cross section at the distance specified. The Moments diagram is solid and shaded. 
Moment below	The major or minor moment immediately below the cross section at the distance specified.
Ecc. Moment above	The major or minor moment due to eccentricity immediately above the cross section at the distance specified. The eccentricity moments diagram is dashed and unshaded. 
Ecc. Moment below	The major or minor moment due to eccentricity immediately below the cross section at the distance specified.
Relative deflection	The relative deflection in the major or minor direction at the distance specified.
Applied load above	The applied distributed load in the major or minor direction immediately

Property	Description
	above the cross section at the distance specified.
Applied load below	The applied distributed load in the major or minor direction immediately below the cross section at the distance specified.
Applied force	The applied point load in the major or minor direction at the distance specified.
Applied moment	The applied moment in the major or minor direction at the distance specified.
Show loading	If cleared, the loading diagram is removed from the view.
Show shear	If cleared, the shear diagram is removed from the view.
Show moment	If cleared, the moment diagram is removed from the view.
Show relative deflection	If cleared, the relative deflections diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Load Analysis View properties for beams

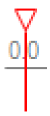
Result Type: Axial

Property	Description
Distance	The results can be reported at any position along the beam, either by typing the distance directly into the **Properties** window, or, on the diagram, by dragging this slider shown below to the position required.  **NOTE** For concrete beams only: If rigid zones have been

Property	Description
	applied, only the non-rigid length of the beam is displayed in the loading analysis view.
Span	Specifies the span for which results are displayed.
Axial force left	The axial force in the beam immediately to the left of the cross section at the distance specified.
Axial force right	The axial force in the beam immediately to the right of the cross section at the distance specified.
Torsion moment left	The torsion in the beam immediately to the left of the cross section at the distance specified.
Torsion moment right	The torsion in the beam immediately to the right of the cross section at the distance specified.
Relative angle of twist	The relative angle of twist (due to torsion) in the beam cross section, at the distance specified.
Angle of twist derivative	A droplist allowing the selection of the derivative of angle of twist: • First • Second • Third (This property is displayed for “Open” sections only).
Angle of twist derivative left	The relative angle of twist (due to torsion) in the beam cross section, immediately to the left of the distance specified. (This property is displayed for “Open” sections only).
Angle of twist derivative right	The relative angle of twist (due to torsion) in the beam cross section, immediately to the right of the distance specified. (This property is displayed for “Open” sections only).

Property	Description
Show axial force	If cleared, the axial force diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show torsion moment	If cleared, the torsion diagram is removed from the view. (This property is only displayed if the Axial direction is selected.)
Show relative angle of twist	If cleared, the relative angle of twist diagram is removed from the view.
Show angle of twist derivative	If cleared, the angle of twist derivative diagram is removed from the view. (This property is displayed for “Open” sections only).
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

Result Type: Major, Minor, (plus Major Principal and Minor Principal for angle sections)

Property	Description
Distance	The results can be reported at any position along the beam, either by typing the distance directly into the **Properties** window, or, on the diagram, by dragging this slider shown below to the position required.  **NOTE** For concrete beams only: If rigid zones have been applied, only the non-rigid length of the beam is displayed in the loading analysis view.
Span	Specifies the span for which results are displayed.

Property	Description
Shear left	The major or minor shear force immediately to the left of the cross section at the distance specified.
Shear right	The major or minor shear force immediately to the right of the cross section at the distance specified.
Moment left	The major or minor moment immediately to the left of the cross section at the distance specified.
Moment right	The major or minor moment immediately to the right of the cross section at the distance specified.
Relative deflection	The relative deflection in the major or minor direction at the distance specified.
Applied load left	The applied distributed load in the major or minor direction immediately to the left of the cross section at the distance specified.
Applied load right	The applied distributed load in the major or minor direction immediately to the right of the cross section at the distance specified.
Applied force	The applied point load in the major or minor direction at the distance specified.
Applied moment	The applied moment in the major or minor direction at the distance specified.
Show loading	If cleared, the loading diagram is removed from the view.
Show shear	If cleared, the shear diagram is removed from the view.
Show moment	If cleared, the moment diagram is removed from the view.
Show relative deflection	If cleared, the relative deflection diagram is removed from the view.
Show dimensions	If cleared, the dimensions are removed from the view.
Show extremes	If cleared, the max and min values are removed from the view.

RSA Seismic Results in a Load Analysis View

Loading Analysis Views for 1st or 2nd Order RSA Seismic result types use the same rules as those applied to multi-member Results Views for the same result types, i.e. as follows:

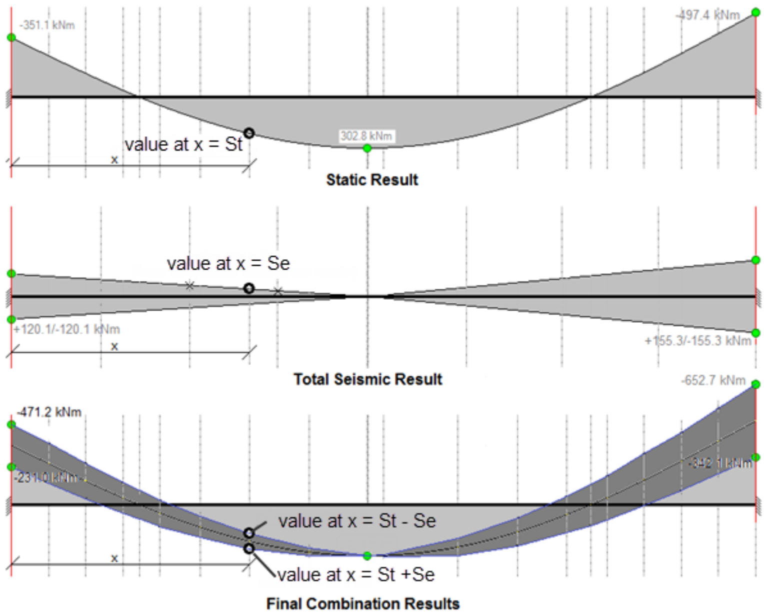
RSA Seismic Loadcases

- Combined (CQC) or Combined (SRSS) - depending on your choice in Analysis Options
    - Absolute values are determined at various points along each member and then plotted on both the positive and negative side of the diagram, (so that the diagrams are always symmetrical about the base line).
- All relevant modes - a standard enveloped diagram is displayed
- RSA Torsion Loadcases - displayed as per 1st order linear analysis
- Static Loadcases included in the RSA Seismic Combination - displayed as per 1st order linear analysis
- Effective Seismic Weight Combination - not available

RSA Seismic Combinations

A droplist is provided to allow you to view:

- Design Profile - derived from the Static+Seismic result, the Design Profile is always plotted on the same side of the base line as the Static Only result
- Static Only - displayed as per 1st order linear analysis
- Seismic Only - absolute values are determined at various points along each member and then plotted on both the positive and negative side of the diagram
- Static+Seismic - an envelope is drawn showing the seismic results above and below the static result.
    - Base line is through the static values
    - Top line is static value + seismic value
    - Bottom line is static value - seismic value



6.4 Solver models

You can display the solver model used for each analysis type in 2D or 3D by opening an appropriate solver view.

If you have performed more than one analysis type, then (providing the geometry and loading have not changed between runs), each solver model is retained.

NOTE Changes to mesh size or uniformity do not constitute a change in the geometry. Hence, if different meshes have been applied for each analysis type, the different solver models are retained.

To	Click the link below:
Learn about the different solver model types and what is displayed within them	Solver model types (page 552)
Open a solver view to display a solver model	Open a solver view (page 559)
Change the solver model type displayed in a solver view	View the solver model used for a particular analysis (page 560)
View properties of solver model objects	View solver model object properties (page 561)
Learn about rigid offsets and rigid zones used in concrete beams and columns	How concrete beams and columns are represented in solver models (page 565)

To	Click the link below:
Learn about the analytical model used for meshed walls	How meshed walls are represented in solver models (page 577)
Learn about the analytical model used for mid-pier concrete walls	How mid-pier walls are represented in solver models (page 582)
Learn about the analytical model used for shear only walls	How shear only walls are represented in solver models (page 584)
Learn about the analytical model used for bearing walls	How bearing walls are represented in solver models (page 588)
View tabular solver model data and solver model results	View tabular solver model data (page 591)

Solver model types

You can review the different solver models by opening a Solver View and then choosing the solver model type required from the right-click menu.

Working Solver Model

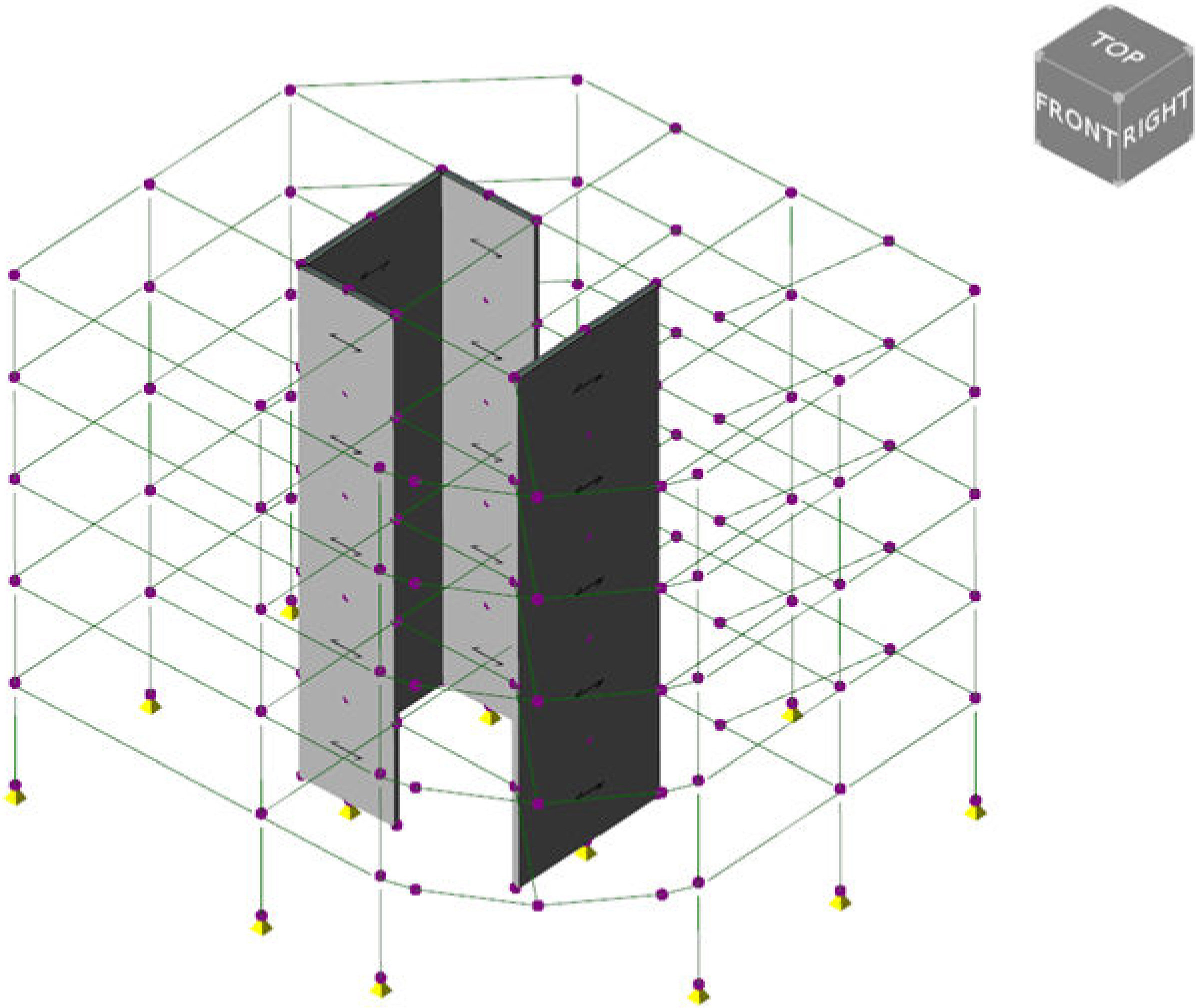
The **Working Solver Model** shows the model in its form prior to any analysis.

Although 1D elements, solver nodes, and diaphragms are displayed, 2D elements are not (even when you have chosen to mesh slabs/walls). This is because 2D elements are only formed at the point of analysis.

Solver Model used for 1st Order Linear and 2nd Order Linear

The solver model used for 1st order and 2nd order linear analysis potentially features a mix of 1D analysis elements, FE meshes and diaphragms as follows:

- beams and columns are modelled as 1D analysis elements
- walls are either mid-pier analysis elements, or FE meshes
- slabs (optionally) form rigid, or semi rigid, diaphragms in floors
- 1-way slabs have their loads decomposed on to supporting members at a preliminary stage of the analysis.
- 2-way slabs are (typically) not meshed, in which case they will also have their loads decomposed on to supporting members at a preliminary stage of the analysis.
- 2-way slabs (optionally) can be meshed, this is only recommended for special cases, typically where slabs participate in the lateral load stability system, e.g. transfer slabs
- supports are user defined



Any FE meshes in the solver model are formed using the mesh parameters in place for the most recent run of the chosen analysis (i.e. 1st order linear or 2nd order linear).

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

2-way slabs meshed

Optionally you can choose to mesh all 2-way slabs - making a fully meshed model (both walls and floors) possible.

This is generally not recommended as it will inevitably increase the model size, (and potentially the time to solve for large models), although it might be considered that a fully meshed model behaves more "correctly" where slabs are considered to be part of the lateral load resisting system of the structure.

It is more likely that you will choose to mesh specific floor levels only (e.g. transfer levels), keeping other levels unmeshed.

Solver Model used for 1st Order Non Linear and 2nd Order Non Linear

These solver models are basically the same as those used for 1st order analysis with the exception that they will also feature non linear elements.

Any FE meshes in these solver models are formed using the mesh parameters in place for the most recent run of the chosen analysis (i.e. 1st order non-linear or 2nd order non-linear).

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis types after changes to either geometry or loading will prevent you from displaying results for these models.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for 1st Order Modal

Any FE meshes in this 3D solver model are formed using the mesh parameters in place for the most recent run of 1st order modal analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for 2nd Order Buckling

Any FE meshes in this 3D solver model are formed using the mesh parameters in place for the most recent run of 2nd order buckling analysis.

If the analysis has yet to be run, the current mesh parameters are applied.

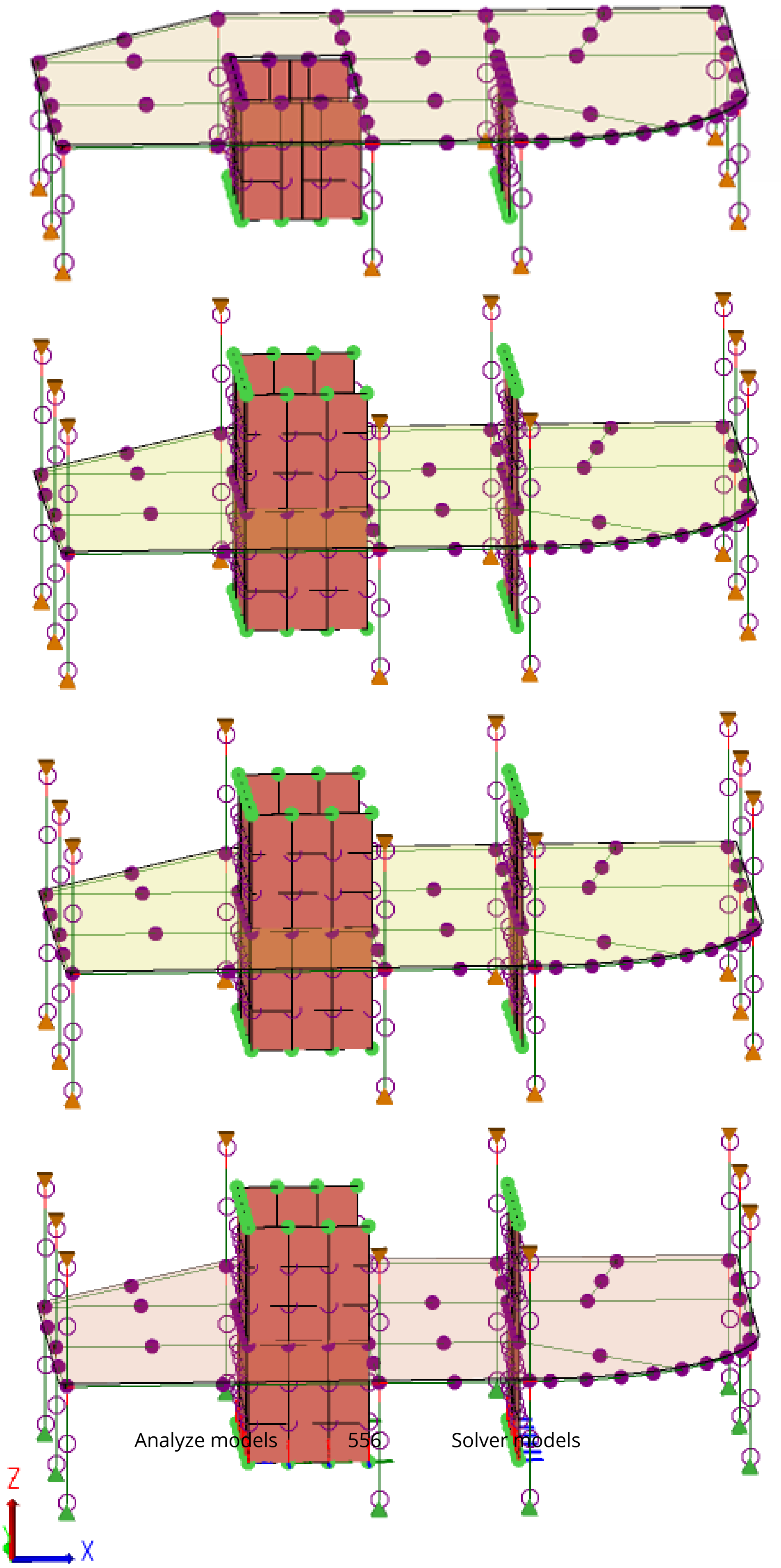
Running any other analysis type after changes to either geometry or loading will prevent you from displaying results for this model.

NOTE Results are still displayed for the "old" solver model until the working solver model is updated to reflect the changes (by running an analysis). Changes to mesh size or uniformity do not cause the working solver model to be updated: consequently if you run an analysis with certain mesh parameters, then a second analysis type with different mesh parameters, both sets of results can be displayed.

Solver Model used for Grillage Chasedown

In grillage chasedown a 3D sub model is formed for each floor including those columns and walls that connected to the floor.

The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.

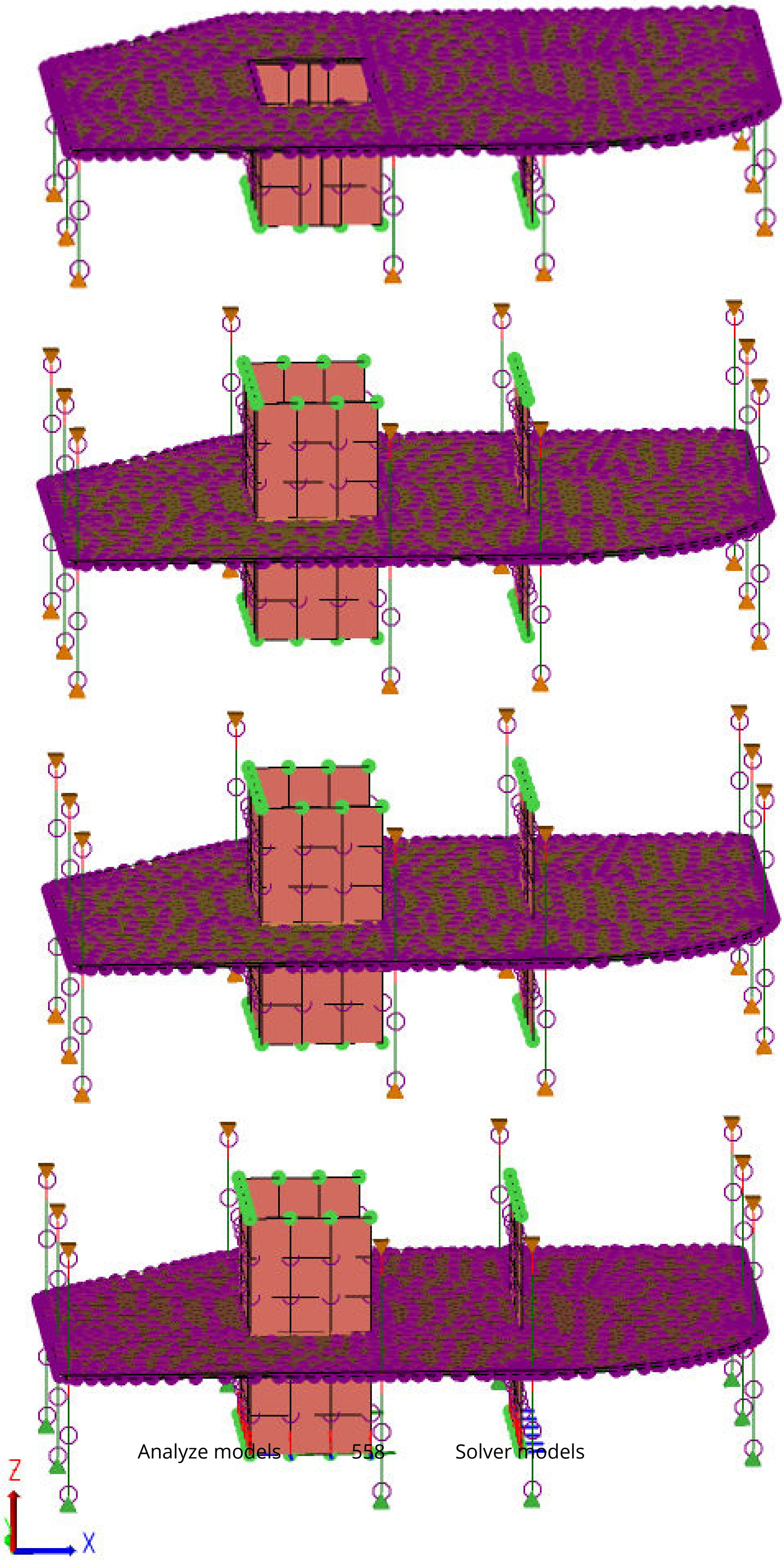

Z
X

they have been set as meshed for 3D Analysis. For all other slabs **load decomposition** is carried out prior to the analysis.

Solver Model used for FE Chasedown

FE chasedown is similar to grillage chasedown, with 3D sub models being formed at each level; the one difference being that in the FE chasedown the two-way slabs are always meshed.

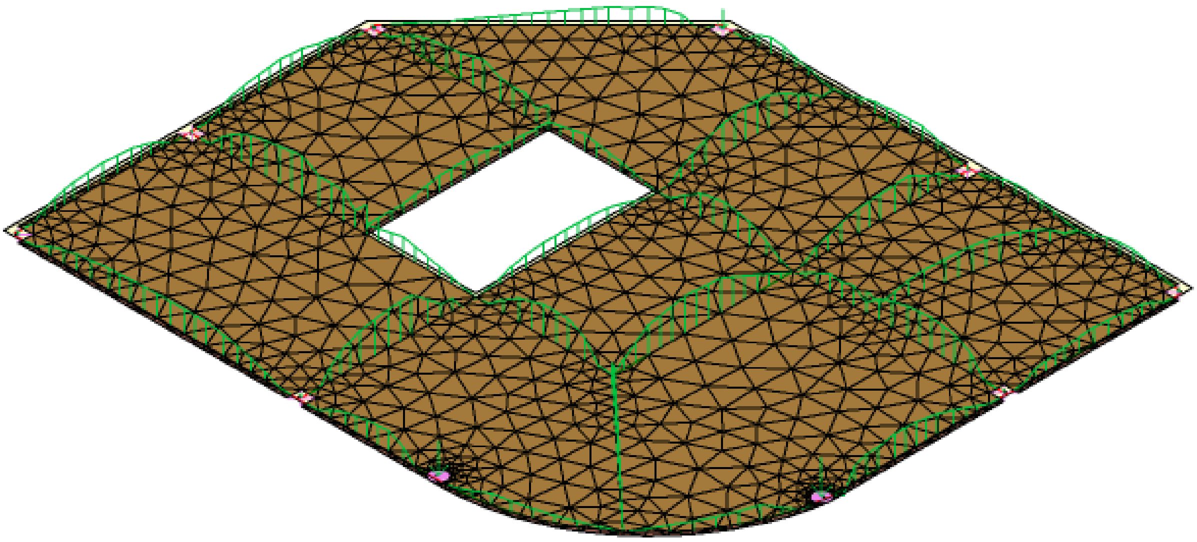
The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.


Z
X

Solver Model used for Load Decomposition

At each level, (provided you have not checked the Mesh 2-way Slabs in 3D Analysis option), a solver model is created solely for the purpose of decomposing slab and panel loads back on to the supporting members. As these load decomposition models are only used during the pre-analysis stage, by default they are not retained.

However, if you want to examine the load decomposition model used at a given level this is possible by editing the level properties prior to analysis and selecting **keep solver model**.



Solver Model used for Load Decomposition

The sub models are analysed sequentially for gravity loads, starting at the top level and working down. Support reactions from each level are transferred to the level below.

Refresh Solver Model

The solver model currently displayed is updated to reflect changes that have occurred in the working solver model since the last analysis. The 2D element mesh is also updated to reflect the current mesh parameters. Previous analysis results are also erased for this solver model

Open a solver view

You can either open a new view that displays the solver model, or change another 2D or 3D view to a solver view. In order to open a solver view, see the following instructions.

Open a solver view as a new view

1. To duplicate the existing 2D or 3D view, right-click the tab.
2. In the context menu, select **Duplicate view**.

   A duplicate of the current view opens.
3. To change the view type of the duplicate, do one of the following:

   - Right-click the duplicate tab, and in the context menu, select **Solver view**.
   - In the **Status bar** at the bottom of the window, click **Solver View**.

Change the existing view to a solver view

1. Open an appropriate 2D or 3D view.
2. To change the view type of the view, do one of the following:

   - Right-click the view tab and in the context menu, select **Solver view**.
   - In the **Status bar** at the bottom of the window, click **Solver View**.

See also

View the solver model used for a particular analysis (page 560)

View solver model object properties (page 561)

View the solver model used for a particular analysis

In case you need to view the solver model that Tekla Structural Designer uses for a particular analysis type, see the following instructions.

1. Open a solver view. (page 559)
2. Right-click anywhere in the solver view.
3. In the context menu, select **Solver models**.
4. In the submenu, select the desired solver model.

   Tekla Structural Designer opens the selected solver model.

See also

View solver model object properties (page 561)

View solver model object properties

After opening a solver view (page 559), you can select solver nodes, solver elements, and supports in order to see their properties in the **Properties** window.

- Solver node properties (page 561)
- Solver element properties (page 561)
- Solver element (1D) types (page 562)
- Solver element 2D properties (page 564)
- Support properties (page 1017)

Solver node properties

When in a Solver View, solver node properties are displayed in the Properties Window as shown below. Only certain of these can be edited; properties that are dimmed are derived and cannot be changed directly.

Property	Description
General	
Fx, Fy, Fz	The translational degrees of freedom at the node.
Fx, Fy, Fz	The rotational degrees of freedom at the node.
Coordinate	The node location.
P-Delta	This property cannot be edited.
Exclude from Diaphragm	Check to remove the node from the diaphragm.
Diaphragm #	Specifies the diaphragm number to which the node is connected.

Solver element properties

When in a Solver View, solver element properties are displayed in the Properties Window as shown below. Only certain of these can be edited; properties that are dimmed are derived and cannot be changed directly.

Property	Description
General	
Active	When this is set to False the solver element is inactive in the analysis. Only certain member types (braces,

Property	Description
	analysis elements) can be made inactive.
Type	The type of the solver element
Fabrication	The fabrication type of the solver element.
Construction	The construction type of the solver element.
Material	The solver element material.
Gamma angle	Defines the element orientation about its local x axis. When gamma = 0, the local z lies in the plane created by the local x axis and the global z axis.
Length	The solver element length.
Start Releases	
Fx, Fy, Fz	These define the translational end releases at end 1.
Mx, My, Mz	These define the rotational end releases at end 1.
End Releases	
Fx, Fy, Fz	These define the translational end releases at end 2.
Mx, My, Mz	These define the rotational end releases at end 2.

Solver element (1D) types

Eight different 1D solver element Types are available in Tekla Structural Designer as follows:

Beam

An element in any material, with user defined area and inertia properties, and user-definable end releases - used in **all solver models** for:

- Columns (any material)
- Beams (any material)
- Truss top, bottom and side members (any material)
- Mid-pier concrete wall: wall-beam, and wall-column elements
- Bearing wall: wall-beam elements
- Analysis Elements (any material) with element type: Beam

Truss

An element in any material, with user-defined cross sectional area, and pinned ends (releases not being editable) - used in **all solver models** for:

- Braces (any material) that have not been set as tension or compression only
- Truss internal members (any material) that have not been set as tension or compression only
- Bearing wall: wall-column elements
- Analysis Element (any material) with element type: Truss

Truss 1D solver elements are also used in **linear solver models** only for:

- Braces (any material) that have been set as tension or compression only
- Truss internal members (any material) that have been set as tension or compression only

Tension only

A pin ended member in any material, with user-defined cross sectional area, that can only transfer tension. This is a non-linear element which requires non-linear analysis - hence used in **non-linear solver models only** for:.

- Braces (any material) that have been set as tension only
- Truss internal members (any material) that have been set as tension only
- Analysis Element (any material) with element type: Tension only

Compression only

A pin ended member in any material, with user-defined cross sectional area, that can only transfer compression. This is a non-linear element which requires non-linear analysis -hence used in **non-linear solver models only** for:

- Braces (any material) that have been set as compression only
- Truss internal members (any material) that have been set as compression only
- Analysis Element (any material) with element type: Compression only

Linear axial spring

An element that deflects linearly when an axial force is applied, in accordance with a user-defined axial spring stiffness. These are specified as:

- Analysis Element (any material) with element type: Linear axial spring

Linear torsional spring

An element that rotates linearly when a torsion force is applied, in accordance with a user-defined rotational spring stiffness. These are specified as:

- Analysis Element (any material) with element type: Linear torsional spring

Non-linear axial spring

An element that deflects non-linearly when an axial force is applied, in accordance with a user-defined axial spring stiffness. These are used in **non-linear solver models only** for:

- Analysis Element (any material) with element type: Non-linear axial spring

Non-linear torsional spring

An element that rotates non-linearly when a torsion force is applied, in accordance with a user-defined rotational spring stiffness. These are used in **non-linear solver models only** for:

- Analysis Element (any material) with element type: Non-linear torsional spring

Solver element 2D properties

When in a Solver View, solver element 2D properties are displayed in the Properties Window as shown below. None of these properties can be directly edited in the Solver View.

Property	Description
Panel	
Type	The 2D element type will be Shell or Semi-rigid depending on the Slab Type, Decomposition and Diaphragm Options that have been set.
Thickness	The 2D element thickness is derived from a different property depending on the slab type: • Composite Slab; Precast Slab; Slab on Beams; Flat Slab - Overall Depth • Steel Deck; Timber Deck - Thickness
Orientation	The 2D element orientation in the solver model follows the rotation angle defined for the slab item to which it belongs.

Property	Description
DivideStiffnessBy	This property applies to semi-rigid elements only. It adjusts the stiffness determined from the material properties in order to control semi-rigid diaphragm flexibility.
CrackedOption	yes/no
Nodes	
Node 1, Node 2, Node 3	The node numbers associated with this element.

How concrete beams and columns are represented in solver models

Solver elements for most members are created directly between the member insertion points - they do not take into account major and minor snap points, or any offsets that might have been specified in the member properties. The exception to this rule is that solver elements for concrete columns and concrete beams do take into account snap points and offsets - **Rigid offsets** are then automatically introduced where necessary to connect the solver elements.

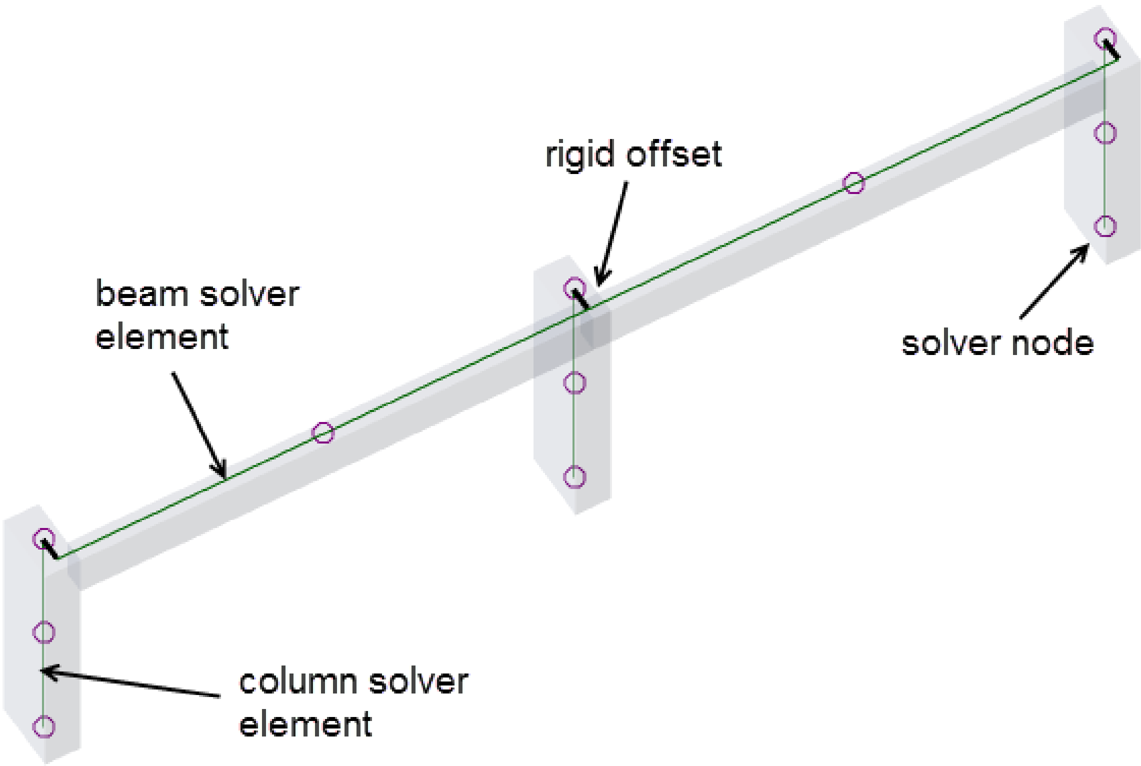
NOTE The rules applied to insertion of solver elements for concrete columns are different to those that are applied to concrete beams. For concrete structures this enables you to simplify the grid layout but then employ offsets to position the members exactly.

Design codes also allow engineers to assume parts of concrete beams and columns are rigid, leading to more efficient designs. Tekla Structural Designer uses **Rigid zones** to cater for this where columns and beams are connected and where beams are connected to other beams. Columns can have rigid zones when they are the supporting or supported member, but beams will only have rigid zones when they are the supported member.

Rigid offsets

For concrete beams and columns rigid offsets are automatically applied to the start and end of solver elements as required to ensure that the solver model is properly connected.

This will be necessary whenever the 1D solver elements are not co-linear. A typical example of this occurs when concrete edge beams are aligned to be flush with the face of the supporting columns, as shown below:



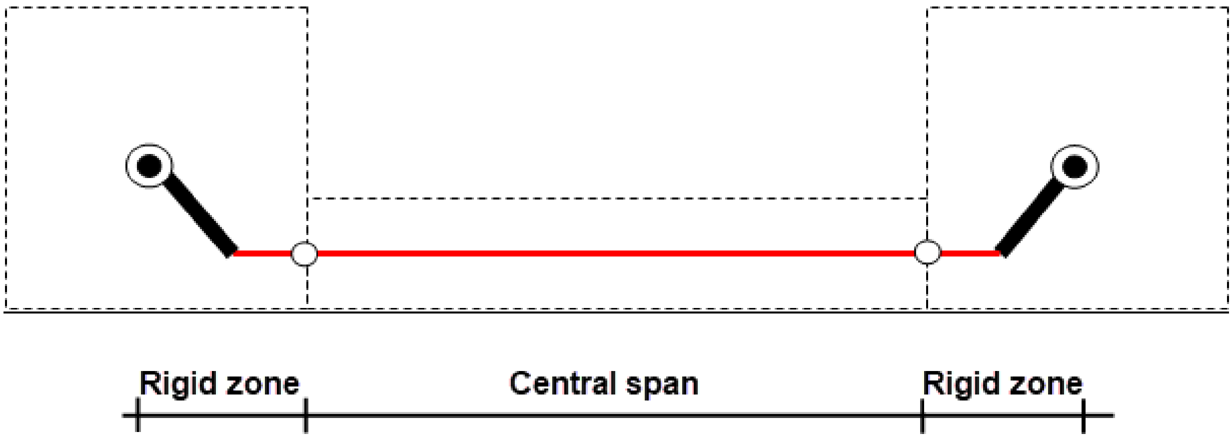
TIP To see rigid offsets: open a Solver View, and then in Scene Content ensure that 1D Elements> RigidOffsets is selected.

Click the below link to view a couple of examples illustrating the use of rigid offsets in concrete columns and beams:

- Rigid offsets examples (page 567)

Rigid zones

Unless you have chosen not to apply them, rigid zones are created at concrete column/beam connections. The proportion of the zone which is modeled as rigid (the thick black line shown below) is specified as a percentage, the remaining portion of the rigid zone (the red line inside the rigid zone) remains elastic. The proportion of the rigid zone that is rigid is specified in Model Settings and can vary between 0 - 100%



As shown above, the elastic portion of the rigid zone is aligned with the central span solver element.

In most situations in order to get an efficient design you would want rigid zones to be applied. You can however choose not to consider them by checking the **Rigid zones not applied** option that is provided in Model Settings, this will deactivate them throughout the model. You can also selectively deactivate rigid zones at specific column/beam connections by unchecking the **Apply rigid zones** option that is provided in the column properties under the **Design control** heading.

- For example, you might choose not to apply them if you encounter problems with short members and rigid zones which cannot be overcome by modifying the physical model.
- When rigid zones are not applied, the position of releases in analysis model is affected, and member start and end points for design are also adjusted.

There is a significant difference between Rigid Zones Not Applied and Rigid Zones Applied with 0% rigidity. The total elastic length of a member is the same in the two models, but the position of releases and start/end of design members will be different.

Rigid zones should not be confused with rigid offsets which are used to ensure that the analysis model is properly connected, i.e. it is possible to have rigid offsets in the model even if rigid zones are turned off.

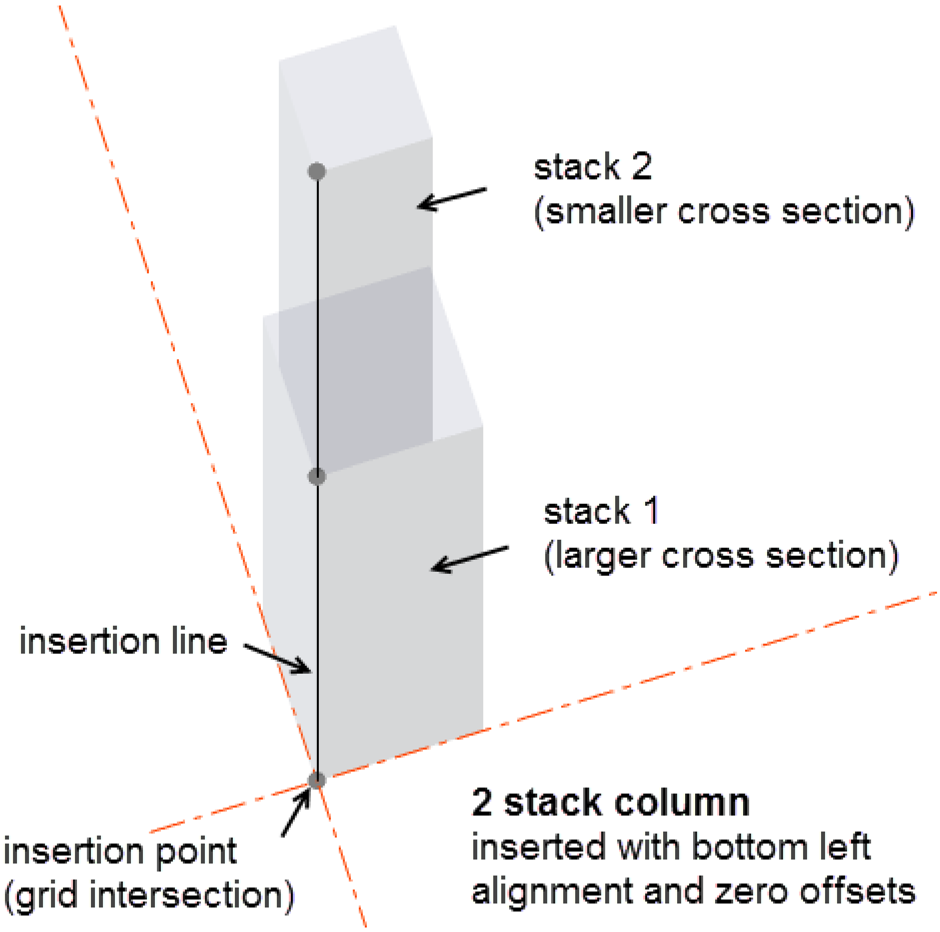
Click the below link to view a couple of examples illustrating the use of rigid zones in fixed and pin ended beams:

- Rigid zones examples (page 571)

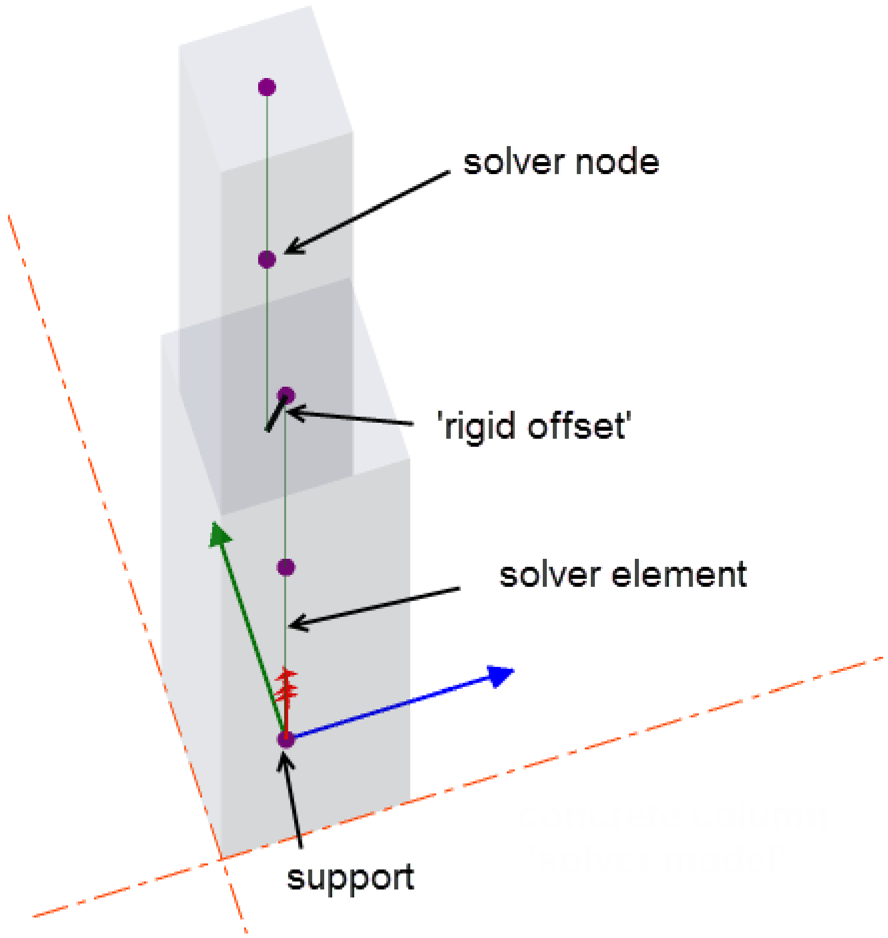
Rigid offsets examples

Rigid offsets example 1 - concrete column

Consider the two stack concrete column shown below - this has been inserted with its alignment properties set to bottom left so that the outer column faces remain flush despite a smaller section being introduced in stack 2.



Since solver elements for concrete columns always take into account any snap points or offsets, they will always be located at the centroid of each stack - thus they do not necessarily coincide with the insertion line used to position the column originally. In this example the centroid position shifts from one stack to the next which causes a “rigid offset” to be created automatically to connect the solver elements. Similar rigid offsets would also be created as required to connect incoming beams into the column centroids.

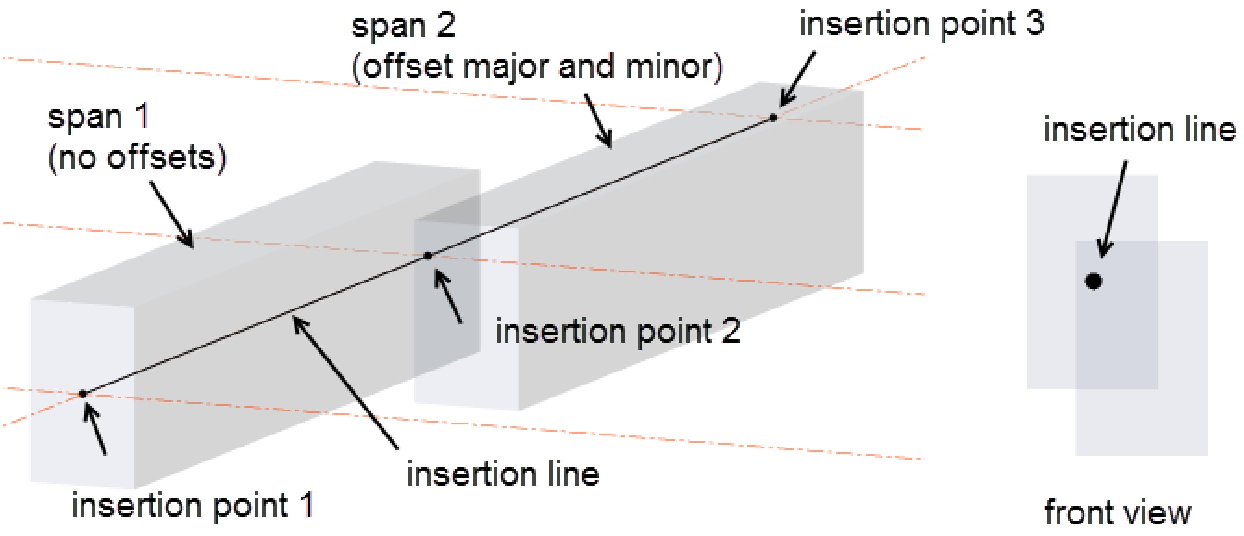


TIP To see solver elements, solver nodes and rigid offsets: open a Solver View, and then in Scene Content select **1D Elements> Geometry & RigidOffset** and **Solver Nodes> Geometry**.

As a consequence of this method of modeling, you are freer to simplify the grid layout in order to create the structure more effectively, and then employ column offsets to position each column exactly, knowing that during analysis the program automatically assumes the column is located at its centroid as shown in the plan view.

Rigid offsets example 2 - concrete beam

Consider the two span concrete beam shown below - this has been inserted with both major and minor axis offsets applied to span 2 only.



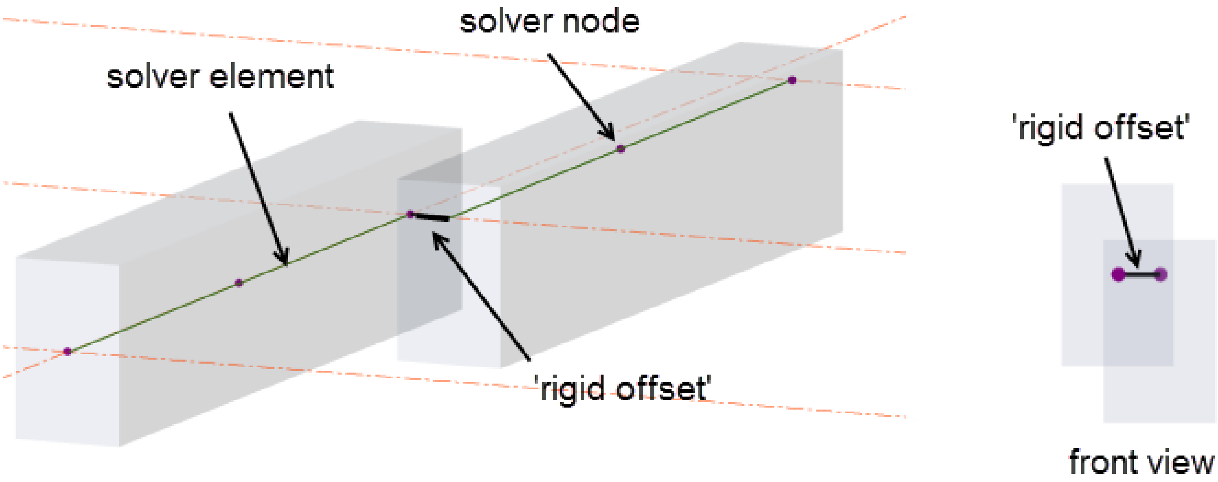
NOTE For concrete beams:

- The **minor** snap points and offsets **are** structurally significant and have an effect on the positioning of the 1D solver elements.
- The **major** snap points and offsets **are not** structurally significant.

In the minor direction beam solver elements are always located at the center of each beam section - as beam widths or minor offsets may vary, this may result in the introduction of lateral rigid offsets to make the connection between spans.

In the major direction beam solver elements are always created at the same level as the insertion line used to position the beam.

Consequently, for this particular example a lateral rigid offset is required to make the connection between the two spans.

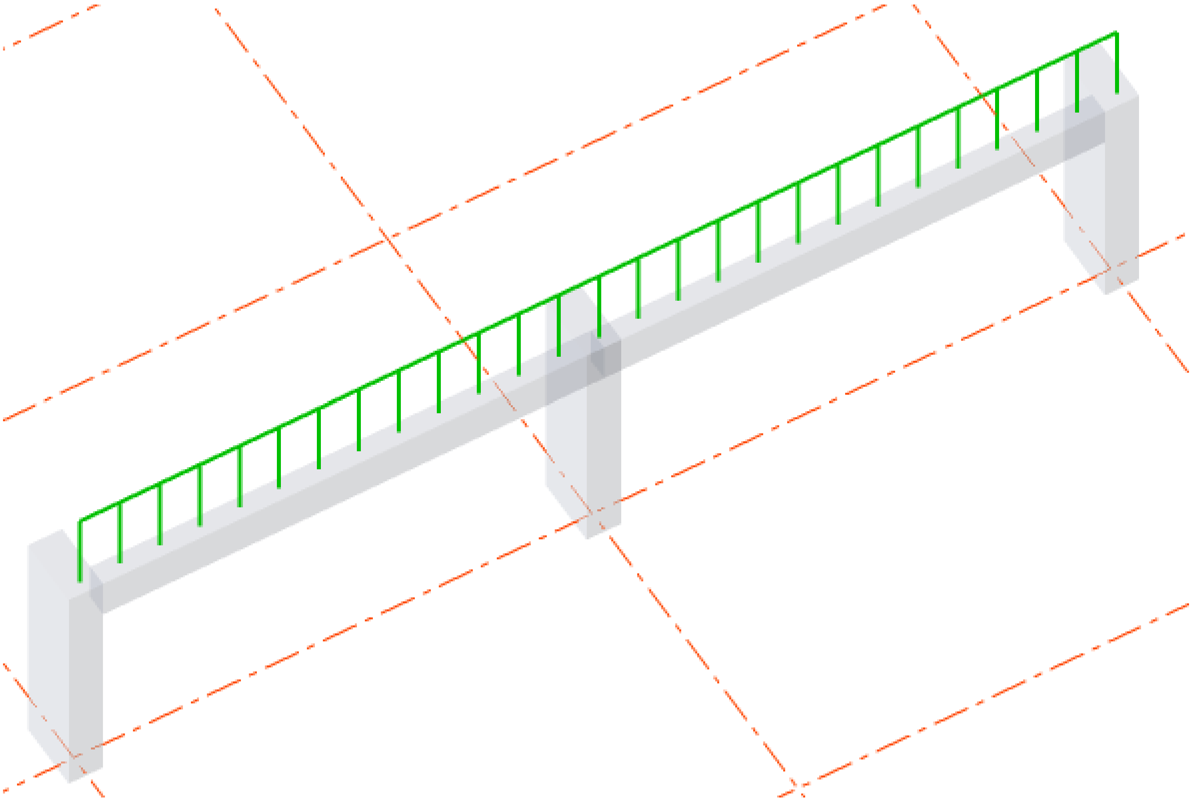


TIP To see solver elements, solver nodes and rigid offsets: open a Solver View, and then in Scene Content select **1D Elements> Geometry & RigidOffset** and **Solver Nodes> Geometry**.

Rigid zones examples

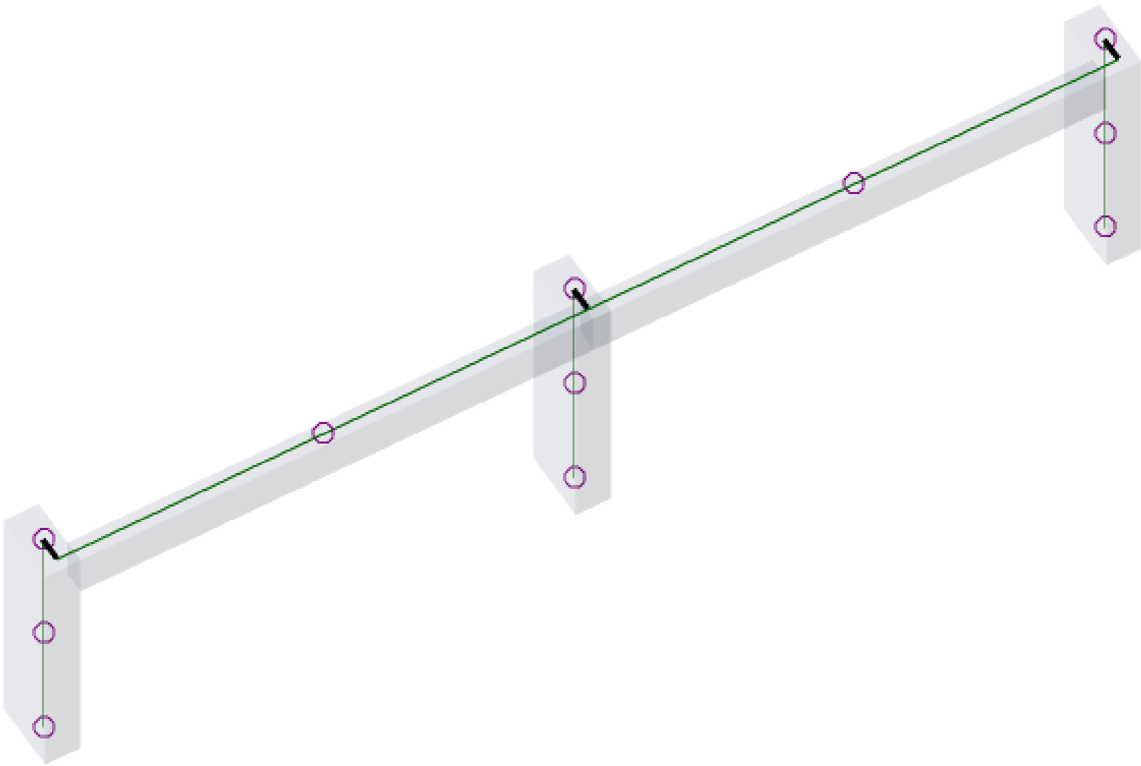
Rigid zones example 1 - fixed ended beam

Consider the following 2 span beam supported on columns and loaded with a udl:

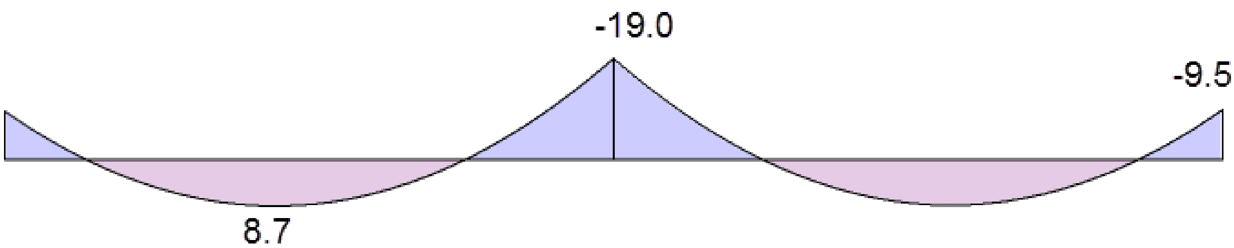


Rigid Zones Not Applied

The analysis model is simply constructed from the solver elements with rigid offsets applied as necessary to connect the beam solver elements to the column solver elements.



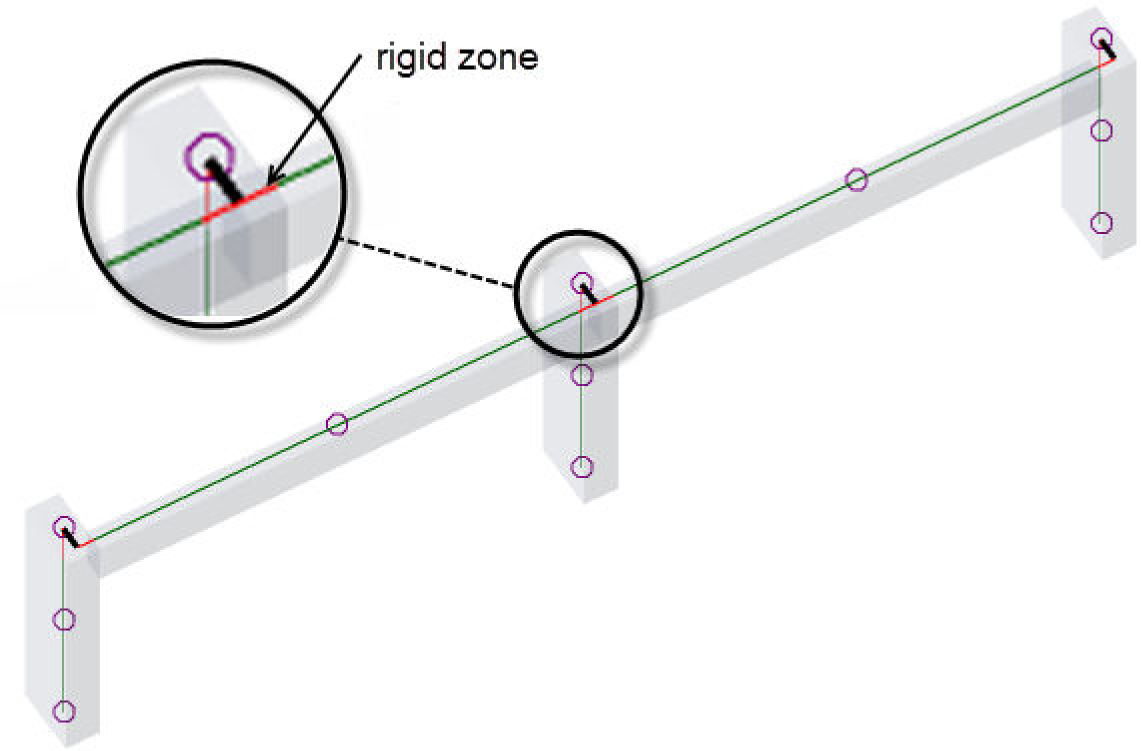
The resulting beam bending moment diagram is as follows:



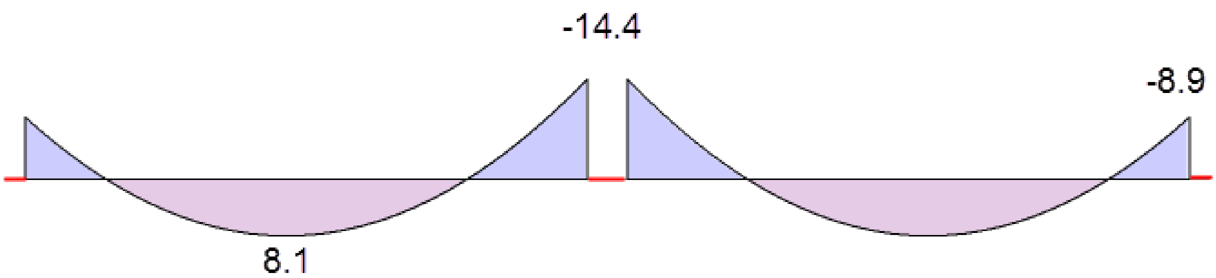
Rigid Zones Applied (default 50%)

Rigid Zones are activated via Model Settings, and this is also where the percentage of rigidity of the zones can be specified. Initially for this example it will be left at the default of 50%.

The revised solver model is as shown below, note the rigid zones that have been formed where the columns and beams connect:



The beam bending moment diagram for the revised model is as shown below.

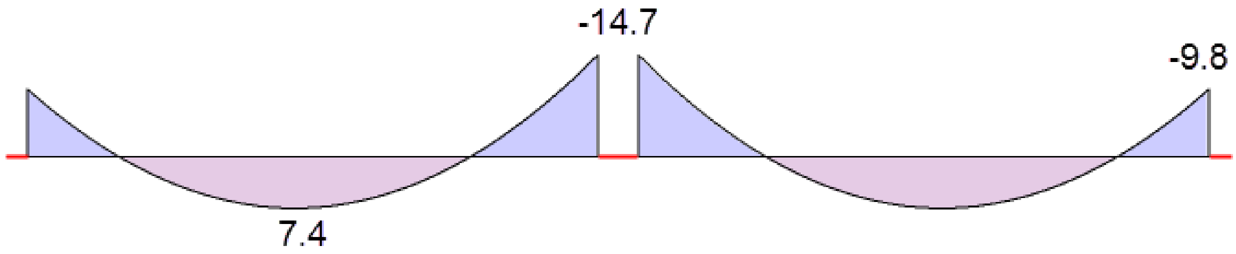


NOTE The above diagram was generated from a Results View to illustrate that a “gap” is formed in the diagram where there are rigid zones. It should be noted that when the same result is displayed in a Load Analysis View the gap is removed, leaving only the non-rigid length of the member displayed.

We might expect the extra stiffness introduced at the supports to increase the hogging moments and reduce the sagging moments, however because the element end forces are now reported at the rigid zone boundaries (rather than the ends of the solver elements) - in this example the main effect is that the hogging moment over the central column support is substantially reduced.

Rigid Zones Applied (100%)

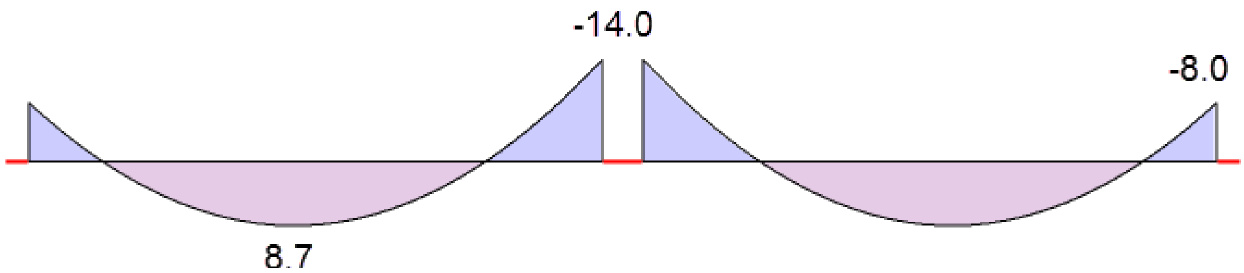
To investigate the effect of the percentage rigidity an additional run is made with the percentage rigidity increased to 100%. The bending moment diagram that results is shown below:



As expected the extra stiffness introduced at the supports increases the hogging moments and reduce the sagging moments in comparison to the run at 50%.

Rigid Zones Applied (0%)

If the percentage rigidity is reduced to 0% the bending moment is as shown below:



If this result is compared to the run in which rigid zones were not applied, it is clear that although the sagging moments are identical, the hogging moments that are reported are not the same. This is because, although the total elastic length of a member is the same in the two models, the position of the start and end of design members is different (being taken at the rigid zone boundaries when applied).

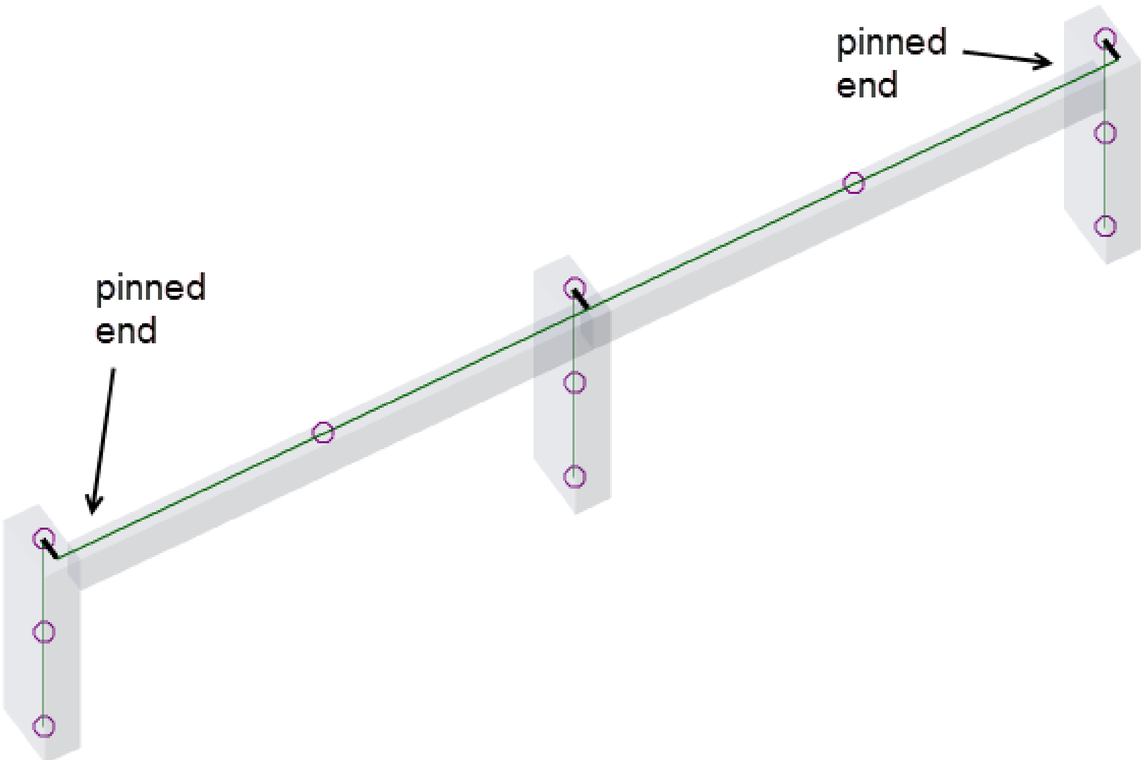
Rigid zones example 2 - pin ended beam

When rigid zones are applied to a pin ended member, the end release is shifted from the end of the solver element to the rigid zone boundary.

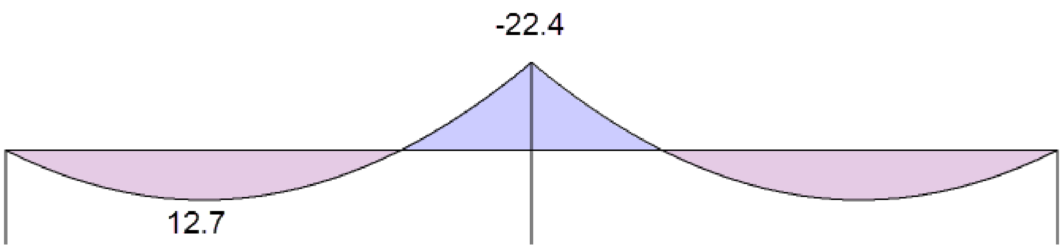
To illustrate this effect the model illustrated in Rigid zones example 1 is modified to have pinned connections introduced at the two remote ends of the beam.

Rigid Zones Not Applied

The analysis model is constructed from the solver elements with rigid offsets applied to connect the beam and column solver elements. Releases are formed at the two remote ends of the beam solver elements.



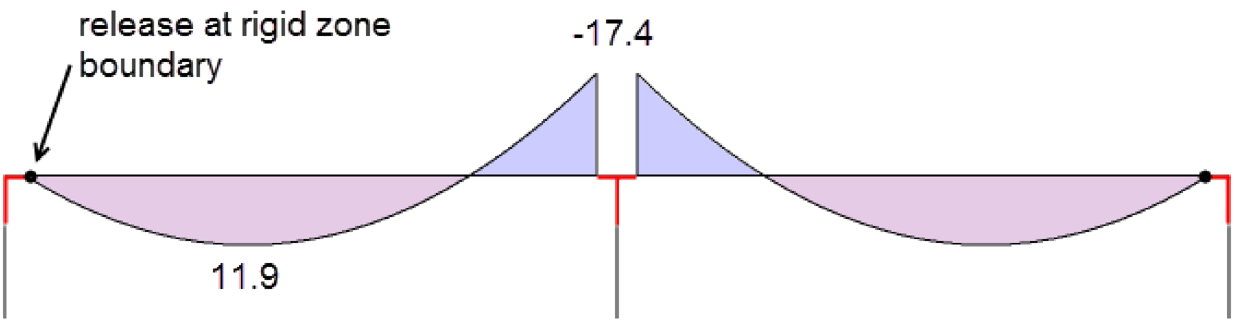
The beam bending moment diagram is as follows:



Rigid Zones Applied (0%)

For comparison, rigid zones are then introduced, (with 0% rigidity in order to keep the total elastic length of the beams the same in both models).

Because the releases are moved to the rigid zone boundaries, this has the effect of reducing the moments in the beams.



How meshed walls are represented in solver models

Meshed walls have isotropic stiffness properties and resist loads in all directions.

Wall beam elements

Wall beam elements are inserted into walls primarily to collect slab mesh nodes and line elements. For meshed walls, they are generated along the top of the wall and also at intermediate levels where an object (e.g. slab, beam, truss) is physically connected. A wall beam element is also generated along the bottom edge of the wall if Generate Support is not selected.

Sloping wall beam elements can be generated by sloping top or bottom edges or connected sloping slabs.

Where horizontal wall beam elements are required, they are generated across the entire width of the wall at that level.

Wall beam elements can also be created where certain properties, (e.g. thickness), differ across panel boundaries.

2D solver elements

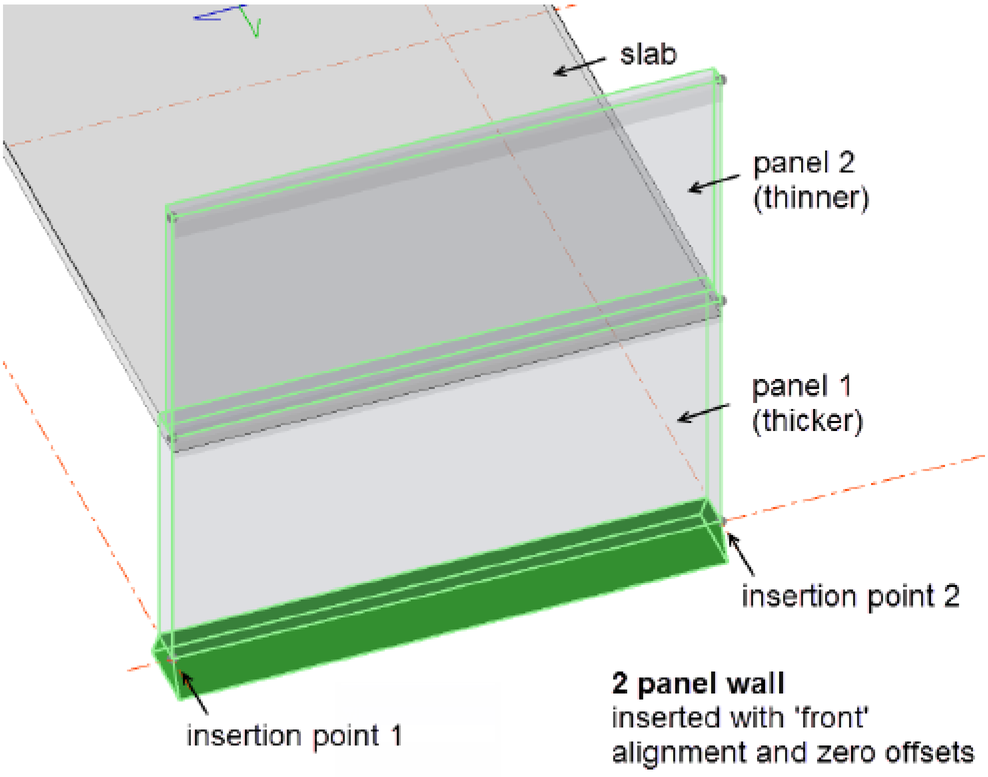
For meshed walls the type of 2D solver element used will depend on whether the wall mesh type is set to Quad only, Tri only, or, Quad dominant.

Modification Factors

Different modification factors applied to meshed walls in the analysis depending on the Material type that has been applied.

Meshed concrete wall example

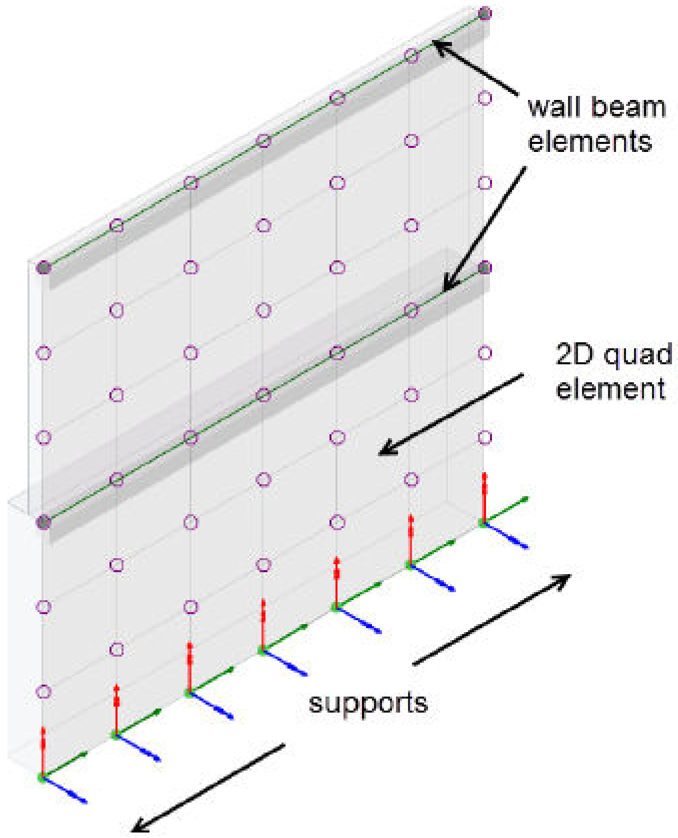
Consider the following two stack wall supporting a slab; the wall has different thickness panels aligned to produce a flush surface on one face.



1D and 2D solver elements for each wall panel are always located along the insertion line used to position the wall originally, irrespective of any alignment offsets that have been specified.

Quad only

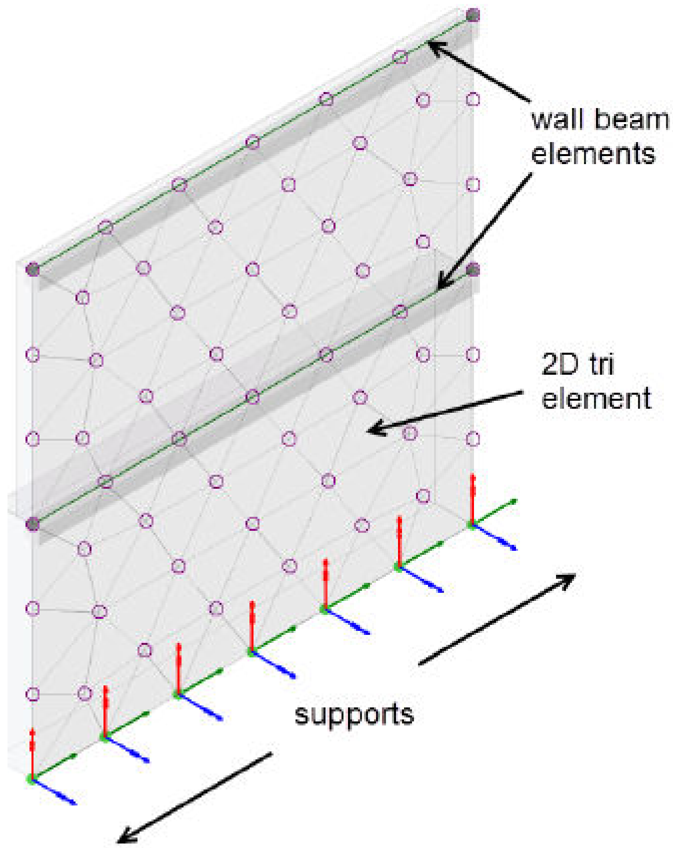
In this two stack example, when the Wall Mesh Type is set to Quad only, solver elements are formed as shown below:



NOTE To see solver elements, solver nodes and 2D elements: open a Solver View, and then in Scene Content check 1D Elements> Geometry, 2D Elements> Geometry and Solver Nodes> Geometry.

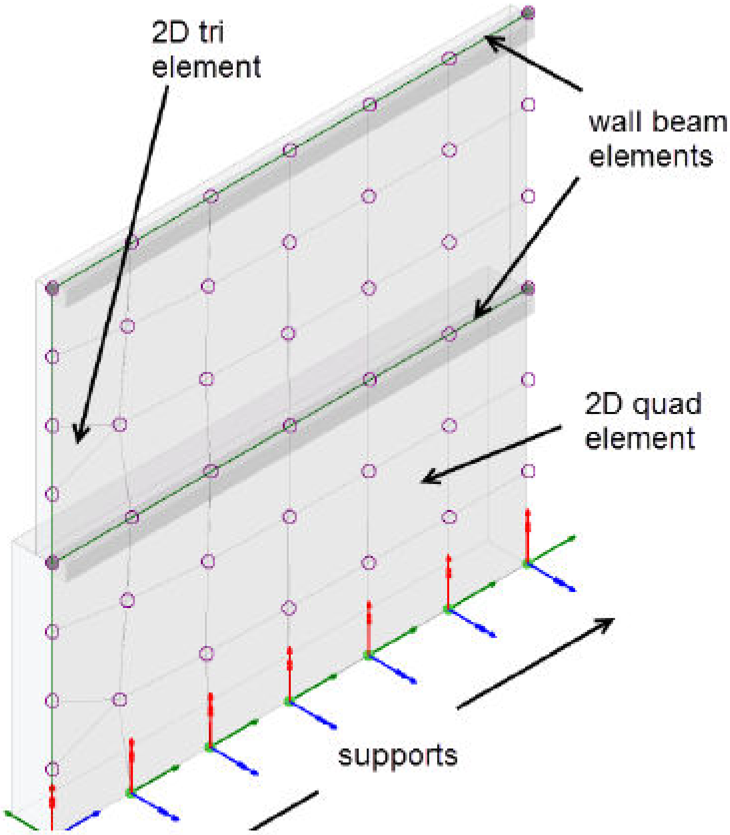
Tri only

In this two stack example, when the Wall Mesh Type is set to Tri only, solver elements are formed as shown below:



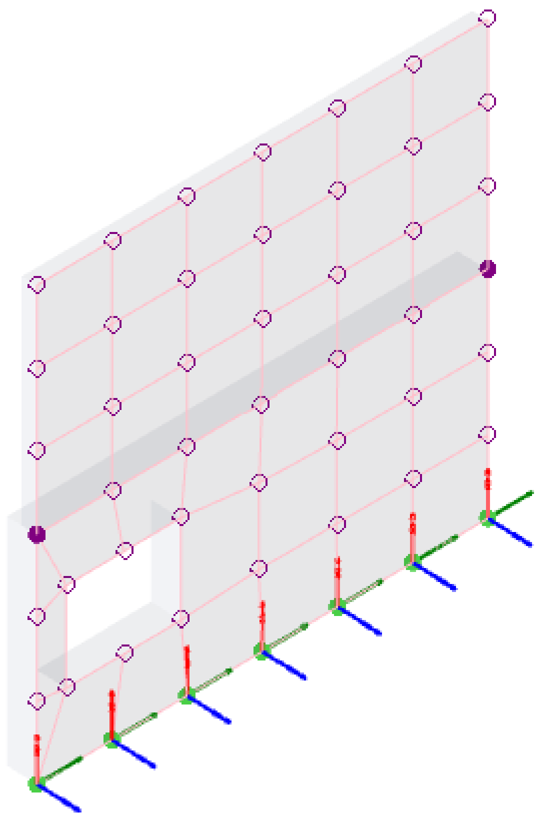
Quad dominant

In this two stack example, when the Wall Mesh Type is set to Quad dominant, solver elements are formed as shown below:



Wall openings

When wall openings are introduced, the mesh will adjust to form around the openings.



See also: Meshed wall openings analysis model (page 242)

How mid-pier walls are represented in solver models

Wall beam and wall column elements

Wall beam elements are inserted into walls primarily to collect slab mesh nodes and line elements. For mid-pier walls, they are generated along the top and bottom edges of the wall and also at intermediate levels where an object (e.g. slab, beam, truss) is physically connected.

NOTE Only horizontal wall beam elements can be generated in mid-pier walls - sloping wall beam elements cannot be generated - this will be indicated by an error in validation.

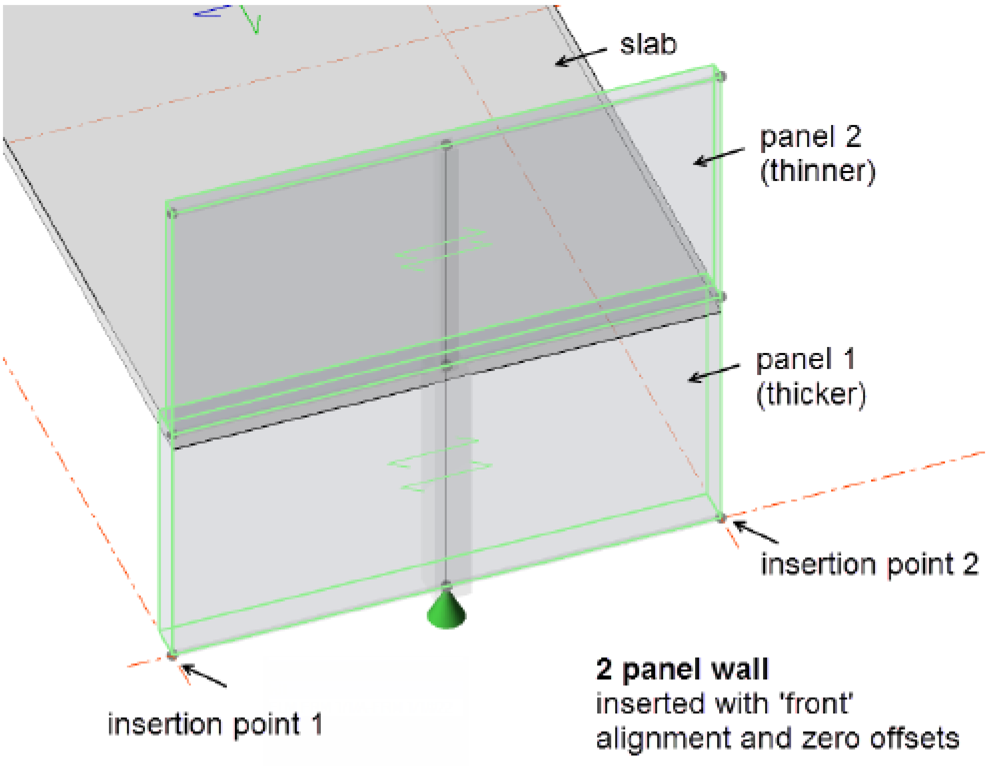
Each mid-pier wall object also has a single vertical wall column element in the middle of the wall, from the top to the bottom level.

Modification Factors

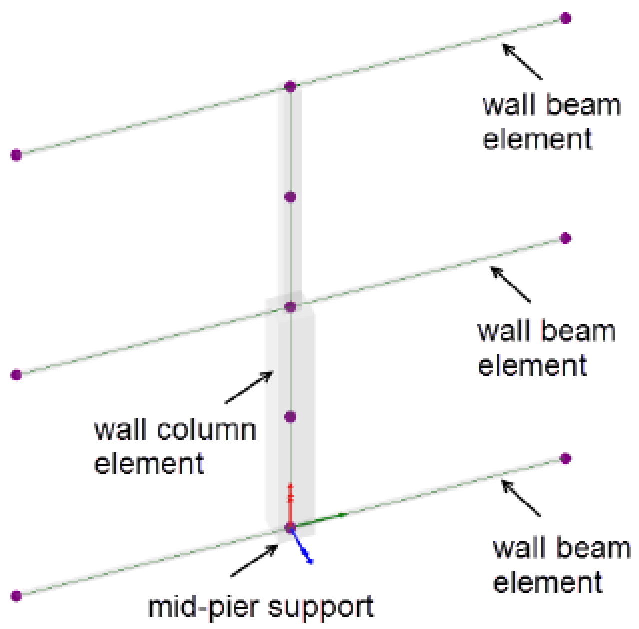
The modification factors applied to concrete walls in the analysis depend on whether they have been specified as meshed or mid-pier.

Mid-pier wall example

Consider the following two stack mid-pier wall supporting a slab; the wall has different thickness panels aligned to produce a flush surface on one face.



The wall beam and wall column elements are always located along the insertion line used to position the wall originally, irrespective of any alignment offsets that have been specified, so for this example, the elements are formed as shown below:



NOTE To see wall beam elements, wall column elements and solver nodes: open a Solver View, and then in Scene Content select 1 D Elements> Geometry and Solver Nodes> Geometry.

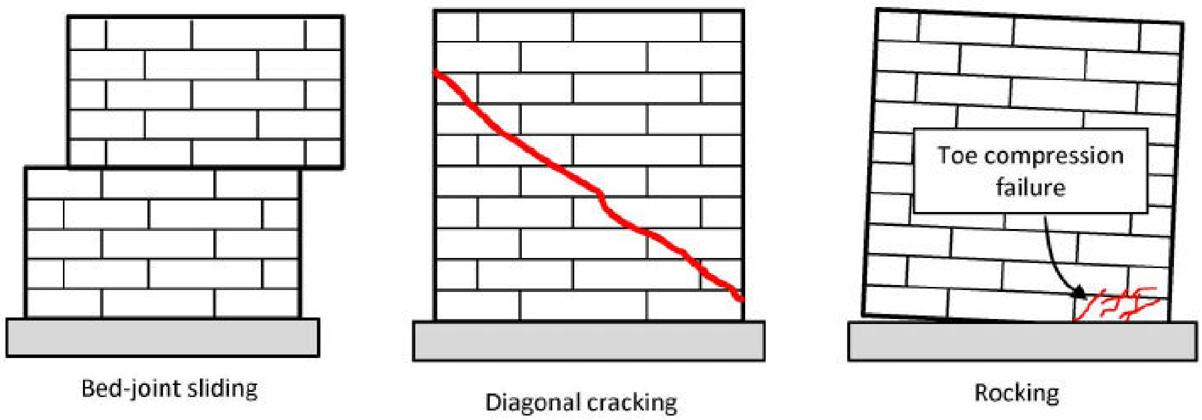
How shear only walls are represented in solver models

Shear only walls resist in-plane shear only and have no out of plane stiffness or load bearing resistance.

Background

The ‘shear only wall’ is configured to allow, as the name implies, shear forces only to be resisted. These arise from frame action in the lateral load resisting system when subject to lateral loads. The behaviour is typical of unreinforced masonry walls built into a steel or concrete frame. Under lateral loading the top of the column adjacent to the wall panel bears on a relatively short length of wall, creates compression in some width of masonry in the diagonal and exits the wall panel at the toe of the opposite corner.

A masonry wall panel resists this force couple at each corner primarily as a shear panel. The failure mechanism either follows a stepped pattern through the joints or by shear bond along the bed joints. The compression strut manifests as a failure in diagonal tension across the bed joints in a stepped fashion. Failure can also occur by local crushing in the top left or bottom right of the wall (or vice versa). These failure modes are depicted in the figure below.



The third failure mode mentioned, crushing in the corners, is very difficult to model and is believed to have a minor influence on overall behaviour. It is thought to be self compensating to some extent because when the masonry begins to crush more length of wall is brought into play.

The diagonal cracking and sliding in the bed joint are the primary effects of the wall acting as a shear panel. This is the only behaviour that ‘shear only walls’ are able to model.

The compression strut that is also inferred by the diagonal cracking generates push-pull forces in the frame and complementary axial forces in the beams. These are incorporated in the Tekla Structural Designer implementation by the use of special ‘Link Elements’.

The oft used and simplest model of a single brace or pair of braces from corner to corner of the wall panel can adequately represent the lateral stiffness of the infilled frame but introduces unwanted axial forces (from gravity loads), particularly in columns. The Tekla Structural Designer implementation is a significant improvement on this simplest model and requires only the determination of the spring stiffness associated with the shear behaviour.

Crisafulli (2007)[1] provides a formula for the stiffness, k_s, of the shear spring as,

k_s	=	$\gamma_s \times A_{ms} \times E_m/d_m \times \cos 2\theta$
Where,		
A_{ms}	=	total area of equivalent strut based on a width of strut of the order of ¼ to ⅓ of the diagonal length of the panel

E_m	=	Elastic modulus of the masonry
d_m	=	diagonal length of the wall panel
θ	=	the angle of the 'strut' with the horizontal

The factor γ_s is the proportion of total stiffness that is assigned to the spring whilst the remainder is provided in the Crisafulli model by a pair of 'masonry struts'. In Tekla Structural Designer γ_s is 1.0 i.e. all of the stiffness is provided by the spring whilst the force effects of the 'masonry struts' are replicated by the special 'Link Element'.

Other formulations for the spring stiffness are likely to exist in the literature.

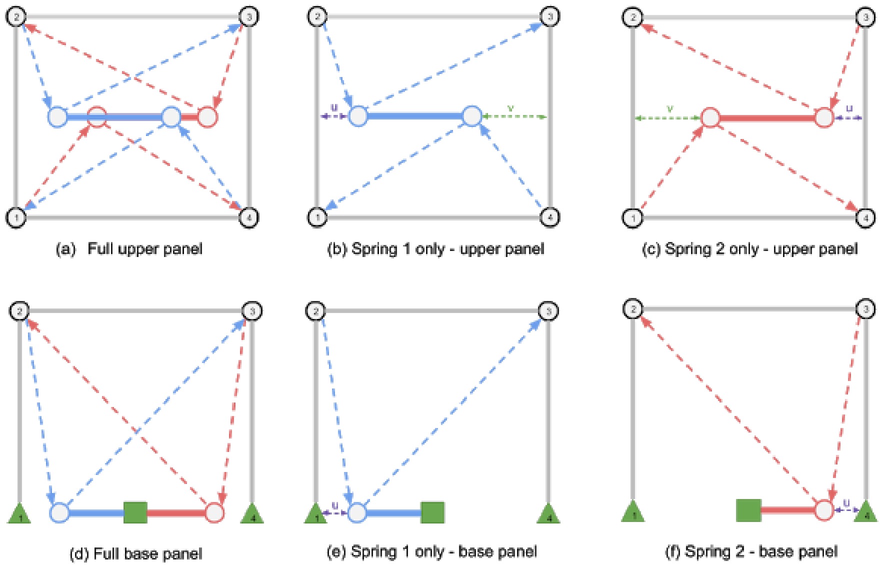
In Tekla Structural Designer, the spring is a either a linear or non-linear uniaxial spring, the former being used when all diaphragms are 'Rigid'.

One of the consequences of the configuration of 'shear only walls' adopted in Tekla Structural Designer is that no loads can be applied out of plane and no members can be connected into the main body of the wall panel. Similarly, the wall panel must be completely surrounded by column and beam members to ensure transfer of lateral loads. For this type of shear wall the head detail is assumed to be such that there is no load transfer from the beam above to the head of the wall.

1 *Crisafulli F. J. and Athol J. C., Proposed macro-model for the analysis of infilled frame structures, Bulletin of the New Zealand Society for Earthquake Engineering, Vol. 40 No. 2 June 2007.*

Solver model in Tekla Structural Designer

Shear only wall panels are modelled using two axial springs between 'panel nodes' connected to 'corner nodes' by link elements. How these are configured differs for interstory panels and the base panel.



Configuration of shear only walls for interstory and base panels. (a) & (d) show the full panel, (b) & (e) show the right to left spring assembly, (c) & (f) show the left to right spring assembly.

NOTE The axial spring and link elements are only shown in the Solver View used for analysis. The above illustrations are not to scale, the actual u and v dimensions being 10mm (25/64 in.) and 20mm (50/64in.)

Interstory panels

For each interstory panel, two springs, each with a pair of nodes are created and connected to the 'corner' nodes where the panel connects to the beam-column node. The connection is made using four special 'Link Elements'. The orientation of the axial spring means there is stiffness only in the plane of the wall, specifically only in the horizontal direction. The Link Elements coordinate systems and their degrees of freedom are configured such that the panel operates in-plane and is stable out-of-plane whilst not generating any untoward moments and forces.

Base panel

Where a base panel exists, a single fixed base is created. Two springs are created at the same level, and connected to the 'corner' nodes where the panel connects to the beam-column node using two Link Elements. The orientation of the axial springs means there is stiffness only in the plane of the wall, specifically only in the horizontal direction. The Link Element coordinate system and their degrees of freedom are configured such that the panel operates in-plane and is stable out-of-plane whilst not generating any untoward moments and forces. Only horizontal reaction is produced at the support and this must be distributed manually along the wall footing if required.

Walls supported on meshed slabs and foundation mats are treated as base panels with the springs and consequent forces applied to a 'seeded' node in the mesh.

Self weight

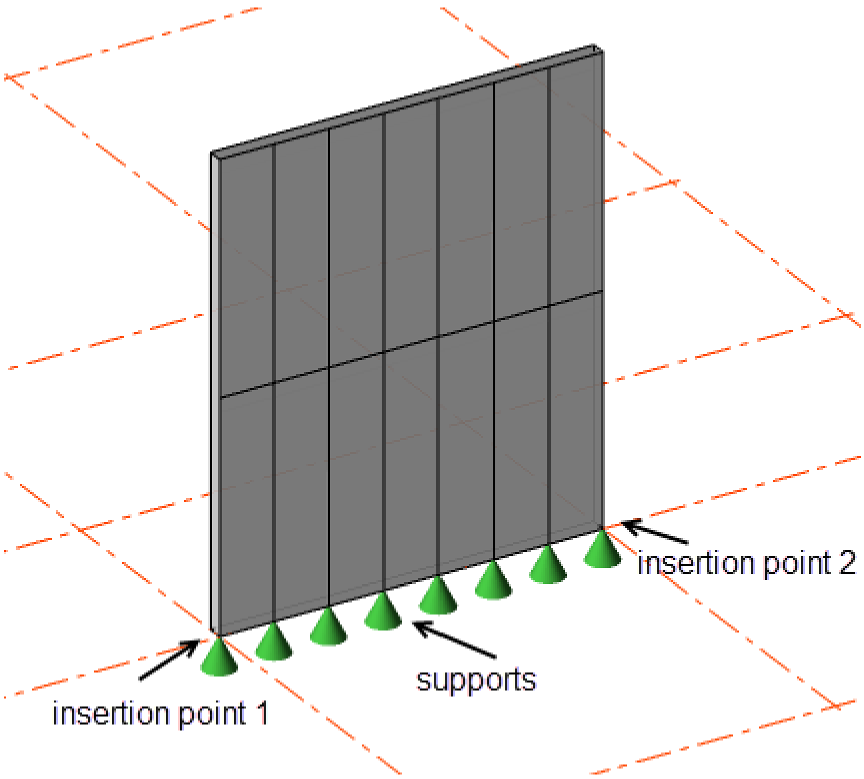
The self weight of each panel is automatically calculated by Tekla Structural Designer and applied to the supporting beam. For a base panel this applied directly to the wall support. For meshed slabs and foundation mats the wall is treated as a base panel and the self weight is applied to a 'seeded' node in the mesh.

How bearing walls are represented in solver models

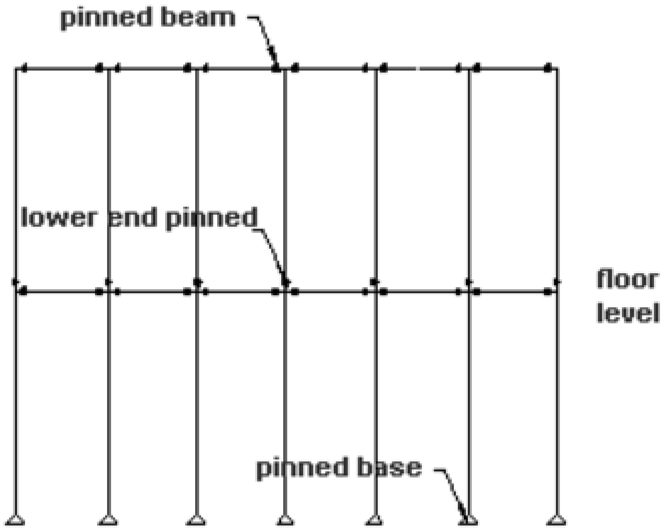
Bearing walls have no in-plane or out-of-plane stiffness and resist vertical load only.

For bearing walls the alignment (Front, Middle, or Back) specified in the wall properties is not structurally significant as it has no effect on the positioning of the solver elements in the solver model.

Consider the two stack bearing wall shown below.

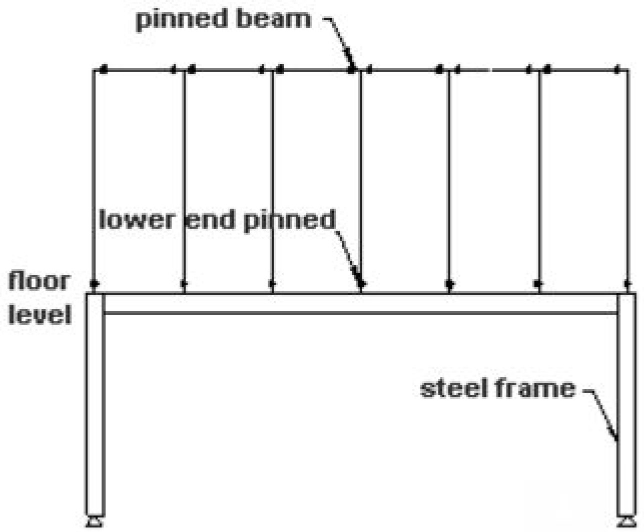


The solver model for this wall is formed using a series of vertical "wall column" and horizontal "wall beam" solver elements. The beams have pinned ends and are placed at the top of the wall spanning between the columns. The next panel above is pinned to the one below and similarly the lower end of a column is pinned to a supporting beam. At the lowest level the column is 'fixed' to a pinned support.

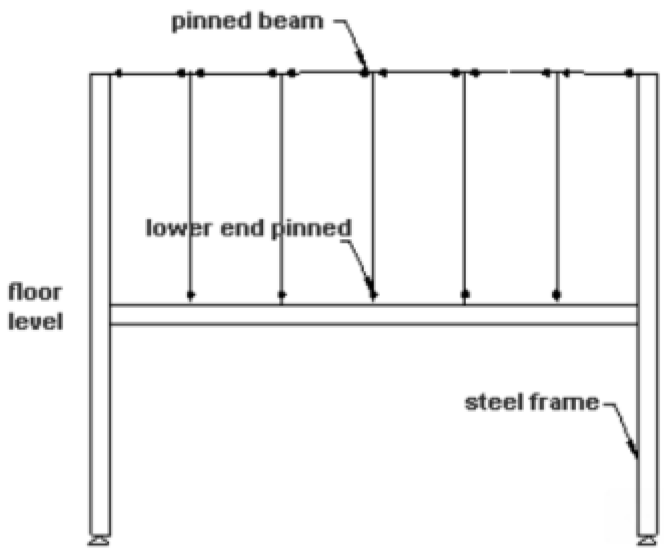


Members supported by the wall either (fortuitously) bear directly on one of the wall columns or on one of the wall beams at the head of the wall. All wall columns and wall beams in an individual bearing wall are given properties automatically by Tekla Structural Designer, based on the width of the bearing wall with which they are associated.

If the bearing wall did not continue to the lowest level, but was instead supported by a transfer beam, then at the lowest level the wall columns would have pinned ends and no supports would be introduced.



For bearing walls that are defined between other vertical column members e.g. steel columns, the wall columns at the edge of the panel are omitted and the associated wall beam is connected to the steel column (for example) and the adjacent wall column - as below.



Wall columns at the edge of the panel are also omitted when it is defined between concrete walls.

Irrespective of whether the wall spans between other vertical column members or not - any load applied to the wall beam at the edge of the panel is shared between the end column and the first internal column. This can result in some load being `lost' directly into the supports.

Load transfer in the bearing wall model is not the same as it would be in for example, a masonry wall. A point load applied at the top of a masonry wall would result in a distributed load on any beam supporting the masonry wall, whereas in a bearing wall the supporting beam would be subjected to a pair of point loads, (or possibly even a single point load if the applied load coincides exactly with a wall column location).

Self weight of the bearing wall is concentrated in the wall beams so seismic weight is concentrated at the top of the wall and not split between the floor above and below.

View tabular solver model data and results

Solver model data views can be used to view tabulated node and element data and specific results for the different solver models.

The full list of tabular data view types is shown below.

NOTE Some of the view types are only available for specific solver model types.

- View solver model object properties (page 561)
- View tabular results for support reactions (page 592)
- View tabular results for nodal deflections (page 592)
- View tabular results for solver element end forces (page 593)
- View tabular results for wall lines (page 594)
- View tabular results for result lines (page 594)
- View tabular results for core lines (page 595)
- View the summed mass for modal mass combinations (page 595)
- View the dynamic masses for modal mass combinations (page 596)
- View active masses by node (page 596)
- View modal frequencies and modal masses (page 597)
- View buckling factors (page 597)

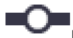
View tabulated solver node and element data

To view node coordinate and degrees of freedom, or solver element properties in tables, see the following instructions.

View tabulated node coordinates and degrees of freedom

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the solver model result type required.
3. In the **View Type** list, select  **Nodes**.

   Tekla Structural Designer displays the node coordinates and degrees of freedom in a table.

View tabulated solver element properties

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select the solver model type required.
3. In the **View Type** list, select  **Elements**. Tekla Structural Designer displays solver element properties in a table.

View tabular results for support reactions

In order to view the results for support reactions, or nodal forces, in tables, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select the desired solver model type.

   NOTE You can view support reactions for all solver model types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis
3. In the **View Type** list, select **Nodal Forces**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.

View tabular results for nodal deflections

To view the deflections of the nodes in your model as tables, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired solver model type.

   NOTE You can view nodal deflections for all solver model types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

3. In the **View Type** list, select **Nodal Deflections**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.

View tabular results for solver element end forces

To view the raw results for element end forces in tables, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired solver model type.

   NOTE You can view solver element end forces for all solver model types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

3. In the **View Type** list, select **Element End Forces**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.

   Tekla Structural Designer displays the raw analysis results of element end forces. This means that no axial load reductions have been applied.

   NOTE The asterisk next to certain element numbers signifies that the results are actually output at the end of a rigid arm that you have modeled, rather than at the node itself.

View tabular results for wall lines

To view the results of the wall lines in your model as tables, see the following instructions. The result tables include information on such aspects as the location, position, axial force, and torsion of the wall lines.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired solver model type.

   NOTE You can view wall line forces for all solver model types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

3. In the **View Type** list, select **Wall Lines**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope. Tekla Structural Designer displays the wall line results in a table.

View tabular results for result lines

To view the results of the result lines in your model as tables, see the following instructions. The result tables include information on such aspects as the location, position, axial force, and torsion of the result lines.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired analysis type.

   NOTE You can view 2D result lines forces for all analysis types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

3. In the **View Type** list, select 2D **Result Lines**.
4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope. Tekla Structural Designer displays the wall line results in a table.

View tabular results for core lines

To view the results of the core lines in your model as tables, see the following instructions.

The result tables include information on such aspects as the location, position, axial force, and torsion of the wall lines.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the desired analysis type.

   NOTE You can view core line forces for all analysis types except for:

   - 1st order modal analysis
   - 2nd order buckling analysis

   ---

3. In the **View Type** list, select **Core Lines**.

4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope. Tekla Structural Designer displays the wall line results in a table.

View tabular results for mode shapes

To view the mode shapes analyzed in a 1st order modal or 2nd order buckling analysis, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.

2. In the upper list of the **Result Type** group, select the solver model type.

   NOTE You can only view mode shapes for the following solver model types:

   - 1st order modal analysis
   - 2nd order buckling analysis

   ---

3. In the **View Type** list, select **Mode Shape**.

4. In the **Loading** list at the bottom of the window, select the desired loadcase, combination, or envelope.

5. In the lower list of the **Result Type** group, select the mode.

View the summed mass for modal mass combinations

In order to view the summed total mass of the modal mass combinations in your model, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Summed Mass**.

   Tekla Structural Designer displays the total mass of the modal mass combinations in a table.

View the dynamic masses for modal mass combinations

To view the total dynamic masses for modal mass combinations, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Dynamic Masses**. Tekla Structural Designer displays the total dynamic masses for the modal mass combinations in a table.

View active masses by node

To view the dynamic active masses of the nodes in your model, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Active Masses by Node**.

   Tekla Structural Designer displays the dynamic active masses of the nodes in your model.

View total masses by node

To view a table of the total masses of the nodes in your model, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Total Masses by Node**.

   Tekla Structural Designer displays the dynamic total masses of each node in your model in a table.

View modal frequencies and modal masses

In order to view the modal frequencies and modal masses of a particular load combination, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of the **Result Type** group, select **1st Order Modal**.
3. In the **View Type** list, select **Modal Frequencies**.
4. In the **Loading** list, select the desired load combination.

   Tekla Structural Designer displays the modal frequencies and modal masses of the selected load combination in a table.

View buckling factors

To view the buckling factors analyzed in the 2nd order buckling analysis in a table, see the following instructions.

1. On the **Analyze** toolbar, click **Model & Results**.

   A **Solver Model Data** view opens in a new tab, and the **Result Type** and **View Type** groups appear in the ribbon.
2. In the upper list of **Result Type** group, select **2nd Order Buckling**.
3. In the **View Type** list, select **Buckling Factors**.
4. In the **Loading** list, select the desired loadcase or combination.

7 Design models

From the **Design** toolbar you can batch design all the members and walls in your model and (separately) batch design all the slabs. From the same toolbar you can also check the response of floors to dynamic excitation.

As an alternative to batch design you might choose to selectively design parts of the model as required.

To get to know about combined analysis and member design and selective member design, see:

- Design steel members and cast-in-place concrete beams, columns and walls (page 599)

To design slabs, see

- Design slabs and run punching shear checks (page 616)

If working to either the Eurocode or US headcode, you may also want to introduce more/less conservatism to the design process by specifying user defined utilation ratios:

- Apply user defined utilization ratios (page 613)

To check the response of floors to dynamic excitation, see:

- Create and run floor vibration checks (page 632)

As part of the design process, drift/sway, seismic drift, and wind drift checks are automatically performed as required. To find out more about these checks, see:

- Drift, sway, seismic drift, wind drift, and overall displacements (page 662)

NOTE Foundation design topics are covered separately, see Create and design foundations (page 688)

Certain member types can be designed using Tekla Tedds, see:

- Design timber and precast members using Tekla Tedds (page 632)

7.1 Design steel members and cast-in-place concrete beams, columns and walls

Before commencing the design you should take a moment to ensure design options and autodesign settings are set as required. If working to either the Eurocode , US, or India headcode, you may also want to introduce more/less conservatism to the design process by specifying user defined utilation ratios.

- Apply and modify design options (page 599)
- Autodesign versus check design (page 600)
- Apply user defined utilization ratios (page 613)

Having set the properties and options as required you can then proceed to rapidly assess all steel members and/or concrete members/walls in your model:

- Combined analysis and member design

Alternatively, you may find it more efficient to run an analysis in the first instance and then:

- Check selected members and walls (page 606)
- Design selected members and walls (page 609)

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see in the Seismic analysis and design handbook.

See also

Design timber and precast members using Tekla Tedds (page 632)

Apply and modify design settings

The design options in Tekla Structural Designer allow you to adjust the way in which your model is designed.

The different Design Settings (page 1064) sub pages enable you to, for example:

- set which analysis type is run for the design, (on the Analysis sub page)
- set reinforcement and other parameters for concrete design, (on the Concrete sub page)
- set the levels at which forces can be ignored, (on the Design Forces sub page)
- set which members to design using groups, (on the Design Groups sub page)

- control how Autodesign settings will be reset after the design, (on the Auto design sub page)

Modify design settings in the current project

1. On the **Design** tab, click **Settings**.
2. Modify the Design Settings (page 1064) according to your needs.
3. After making the changes, do one of the following:
   - To apply the changes to the current project, click **OK**.
   - To save the changes as defaults for future projects, click **Save...**

   TIP To revert to the default design settings of the active settings set, click **Load...**

Modify design settings defaults for future projects

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Go to the Design Settings (page 1064) page.
4. Select the settings set that you want to modify.
5. Modify the settings according to your needs.
6. To save the changes to the settings set, click **OK**.

See also

Design Settings (page 1064)

Autodesign versus check design

Before you begin the design process you should select a suitable design mode for each member in your model. The available design mode options are check design mode and autodesign mode.

Each member in your model is set to one of two design modes:

- Check design mode:

  You assign the desired section size (for steel members), or section size and reinforcement (for concrete members). Then, Tekla Structural Designer determines if the section or reinforcement is sufficient

- Autodesign mode:

  For steel members, you select the desired section type. Tekla Structural Designer then determines a suitable size for the selected section type.

  For concrete members, you assign the desired section size.Tekla Structural Designer then automatically determines a suitable reinforcement configuration.

TIP To quickly review and update the mode applied to all members in the model, open a review view and use the **Auto/Check Design** command to switch between the two modes.

NOTE The Autodesign setting is only considered when you choose to design multiple members, as explained below:

- If you choose to run a **combined analysis and design** (Design Steel, Design Concrete, Design All); or choose a **selective design** of multiple members (Design model, Design plane, Design Selection etc.) - members will either be designed or checked according to their individual 'auto-design' setting.
- If you choose a **check** (Check model, Check plane, Check Selection, Check Member etc.) - each member will be checked irrespective of its **'auto**-design' setting.
- If you **design** an individual member (Design Member, Design Wall etc.) - the member will be designed irrespective of its 'auto-design' setting.

See also

Select whether to design steel, concrete, or all (page 604)

Select between static and gravity design (page 605)

Combined analysis and member design

Run a combined analysis and design in order to rapidly check, or design; every concrete, steel, or, concrete and steel menber in the model for active gravity, static, or RSA load combinations.

Overview

Located on the **Design** toolbar, three groups of commands can be used for combined analysis and design as follows:

- **Design steel** - to design all steel beams and columns in your model.
- **Design concrete** - to design all concrete beams, columns and walls.
- **Design all** - to design all steel beams and columns, and concrete beams, columns and walls.

By selecting a **Gravity**, **Static**, or **RSA** option, you can control which classes of combination are considered in the analysis and design process.

At the end of the chosen process the **Review** tab and the review view open so that you can inspect the results

NOTE As design validation is performed automatically as part combined analysis and design, there is no requirement to run design validation manually beforehand.

NOTE In combined analysis and design each member is either designed or checked according to its individual 'auto-design' setting.

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see in the Seismic analysis and design handbook.

Run Design Steel (Gravity)

- On the **Design** tab, click **Design Steel (Gravity)**.

  Tekla Structural Designer analyzes gravity combinations only, and then designs all steel beams and columns in the model.

Run Design Steel (Static)

- On the **Design** tab, click **Design Steel (Static)**.

  Tekla Structural Designer analyzes all static combinations and then designs all steel beams and columns in the model.

Run Design Steel (RSA)

NOTE RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 407) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see in the Seismic analysis and design handbook.

- On the **Design** tab, click **Design Steel (RSA)**.

  Tekla Structural Designer analyzes all Seismic RSA combinations and then designs all steel beams and columns in the model.

Run Design Concrete (Gravity)

- On the **Design** tab, click **Design Concrete (Gravity)**.

  Tekla Structural Designer analyzes gravity combinations only, and then designs all concrete beams, columns and walls in the model.

Run Design Concrete (Static)

- On the **Design** tab, Click **Design Concrete (Static)**.

  Tekla Structural Designer analyzes all static combinations and then designs all concrete beams, columns and walls in the model.

Run Design Concrete (RSA)

NOTE RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 407) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see in the Seismic analysis and design handbook.

- On the **Design** tab, Click **Design Concrete (RSA)**.

  Tekla Structural Designer analyzes all Seismic RSA combinations and then designs all concrete beams, columns, and walls in the model.

Run Design All (Gravity)

- On the **Design** tab, click **Design All (Gravity)**.

  Tekla Structural Designer analyzes gravity combinations only, and then designs all steel and concrete beams and columns and all concrete walls in the model.

Run Design All (Static)

- On the **Design** tab, click **Design All (Static)**.

  Tekla Structural Designer analyzes all static combinations and then designs all steel and concrete beams and columns and all concrete walls in the model.

Run Design All (RSA)

NOTE RSA load combinations must be set up in advance of RSA Design by running the Seismic Wizard (page 407) (choosing the option to use Modal Response Spectrum Analysis).

NOTE Additional considerations may be necessary when designing members and walls for seismic analysis, see in the Seismic analysis and design handbook.

- On the **Design** tab, Click **Design All (RSA)**.

  Tekla Structural Designer analyzes all Seismic RSA combinations and then designs all steel and concrete beams and columns and all concrete walls in the model.

Select whether to design steel, concrete, or all

The appropriate design command depends on the materials you have used in the model. For more information on selecting the appropriate command, see the following paragraphs.

In simple terms:

- If your model consists of steel members only, you can run the design by using the **Design Steel** commands.
- If your model consists of concrete members only, you can run the design by using the **Design Concrete** commands.
- If your model consists of a mix of both concrete and steel members, you can run the design by using the **Design All** commands.

TIP For structures that are mostly steel but have a few concrete members: instead of running Design All, you could run Design Steel (in order to focus on the steel design) before switching to Design Concrete for the remaining members.

In this way during the steel design phase you are not running grillage and FE chasedown analyses when they are not required .

For more details on how each command affects the analysis process, see the following:

Design steel

- Performs a 3D analysis.
- Does not perform grillage chasedown or FE chasedown analysis.
- Designs or checks all steel elements and shear walls.
- Does not design or check concrete beams or columns.

Design concrete

- Performs a 3D analysis and a grillage chasedown analysis.
- May also be required to perform an FE chasedown analysis.
- Designs or checks all concrete beams, columns and shear walls.
- Does not design or check steel elements.

Design all

- Performs a 3D analysis and a grillage chasedown analysis.
- May also be required to perform an FE chasedown analysis.
- Designs or checks all concrete beams, columns and shear walls.
- Designs or checks all steel elements.

See also

Select between static and gravity design (page 605)

Select between static and gravity design

If you are not sure whether you should use the **Design (Static)** or **Design (Gravity)** commands when designing the model, see the following paragraphs for more details on each option.

Design (Gravity)

Although your final design should consider all combinations, designing for gravity combinations can be a useful way to rapidly pre-size the members in the model that are not subjected to lateral loads.

Designing members for gravity combinations is more common for steel structures than concrete structures.

The gravity design commands involve the use of first order analysis on a limited set of design combinations as follows:

- **Design Steel (Gravity)**: rapid gravity sizing of the majority of steel members
- **Design Concrete (Gravity)**: design of concrete members for gravity combinations only

- **Design All (Gravity)**: design of all steel and concrete members for gravity combinations only

Because you may not have created lateral systems at this stage, the column nodes that are not in a rigid floor diaphragm are, by default, fixed horizontally. To change the default setting, go to **Home** --> **Settings** --> **Design Settings** --> **General**.

Design (Static)

Once initial member sizes have been adequately sized, you should make a sway sensitivity assessment (ACI/AISC), as this can affect the choice of analysis type (ACI/AISC) used in the final static design.

The final static design is invoked by selecting the suitable **Design (Static)** command:

- **Design Steel (Static)**: full design of all steel members
- **Design Concrete (Static)**: full design of all concrete members
- **Design All (Static)**: full design of all steel and concrete members

For each command above, Tekla Structural Designer performs a 3D analysis for all active combinations to establish a set of design forces. The 3D analysis can be either first or second order, depending on the settings specified on the **Analysis** page of the **Design Settings** dialog box.

If the model contains concrete members, Tekla Structural Designer also performs a Grillage chasedown and potentially FE chasedown analysis to establish additional sets of design forces.

All members are checked or designed for the appropriate design requirements. Gravity members are only checked for gravity combinations, while lateral members are checked for all combinations. Only active combinations are checked.

See also

Select whether to design steel, concrete, or all (page 604)

Check selected members and walls

There are various ways to check specific members only, using the results of the analyses that have already been performed.

Using a selective approach can be a more efficient way to work with large models.

Check an individual member, wall, or core

- Do one of the following:

To	Do this
Check a member/wall from a 2D or 3D view	1. Hover the mouse pointer over the member. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right click and select **Check Member > Static/RSA**, or **Check Wall > Static/RSA** as required.
Check a member from the Structure Tree	1. In the Members branch locate the member reference. 2. Right click and select **Check Member > Static/RSA** as required.
Check a wall from the Structure Tree	1. In the Walls branch locate the wall reference. 2. Right click and select **Check Wall > Static/RSA** as required.
Check all members in a core from the Structure Tree	1. In the Cores branch locate the core reference. 2. Right click and select **Check member > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check selected members and walls

1. In the active view, select the members to check.
2. Right-click, then in the context menu, select **Check Selection > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members in a level, slope, or frame

- In the **Project Workspace**, click **Structure** tab.
- In the **Structure** tree, do one of the following:

To	Do this
Check all members in a construction level	Right click the required **Level** and select **Check plane > Static/RSA** as required.
Check all members and walls in a frame	Right click the required **Frame** and select **Check plane > Static/RSA** as required.
Check all members in a slope	Right click the required **Slope** and select **Check plane > Static/RSA** as required.

NOTE These commands use the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Members** tree.
3. In the context menu, select **Check members > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all walls

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Walls** tree.
3. In the context menu, select **Check walls > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members and walls

1. In the **Project Workspace**, click **Structure** tab.
2. Right click on the **Structure** branch.
3. In the context menu, select **Check model > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members of a particular section or type

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Members** tree, right-click the section type or size you want to check.
3. In the context menu, select **Check members > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Check all members in a group

1. In the **Project Workspace**, click **Groups** tab.
2. In the **Design** tree, right-click the member group you want to check.
3. In the context menu, select **Check Group > Static/RSA** as required.

NOTE
- **Check Groups** uses the existing analysis results, (even if the analysis status is 'Out of Date').

  It is not available unless the **Design Groups** option is checked for the member type in **Design Settings**.

Check all members and walls in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to check.
3. In the context menu, select **Check sub structure > Static/RSA** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Design selected members and walls

There are various ways to design specific members only, using the results of the analyses that have already been performed.

Using a selective design approach can be a more efficient way to work with large models.

NOTE In all of the selective design methods, each member in the selection will be individually checked or designed according to its individual autodesign setting, (while also taking account of any member design groups that are active). The exception being that if there is only a single member in the selection it will always be designed irrespective of its autodesign setting.

Design an individual member, wall, or core

Design Member designs an individual member (taking into account any member design groups that are active).

Design Wall designs an individual wall.

Both these commands ignore the auto-design setting of the member/wall. (i.e. a design is always performed even if the autodesign property is off.)

- Do one of the following:

To	Do this
Design a member/wall from a 2D or 3D view	1. Hover the mouse pointer over the member/ wall. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right click and select **Design Member > Static/RSA**, or **Design Wall > Static/RSA** as required.
Design a member from the Structure Tree	1. In the Members branch locate the member reference. 2. Right click and select **Design Member > Static/RSA** as required.
Design a wall from the Structure Tree	1. In the Walls branch locate the wall reference. 2. Right click and select **Design Wall > Static/RSA** as required.
Design all members in a core from the Structure Tree	1. In the Cores branch locate the core reference. 2. Right click and select **Design member > Static/RSA** as required.

NOTE The **Design Member** and **Design Wall**commands both use the existing analysis results, (even if the analysis status is 'Out of Date').

Design selected members and walls

Design Selection performs a check or design of each entity in the current selection, according to each entity's autodesign setting, (taking into account any member design groups that are active).

1. In the active view, drag a box to make your selection.
2. Right-click, then in the context menu, select **Design Selection > Static/RSA** as required.

   The selected members/walls are designed/checked.

   If any members in the selection are in a design group, other members in the same group are updated and checked as required.

NOTE The **Design Selection** command uses the existing analysis results, (even if the analysis status is 'Out of Date').

Interactively design a concrete member

Interactive design is only available for concrete beams columns and walls. A dialog opens showing the existing design of the highlighted member for the currently supplied reinforcement. From here you can interactively modify the reinforcement and instantly see the result.

1. In the active view, hover the mouse pointer over the concrete member you want to interactively design.
2. Right click and select **Interactive Design > Static/RSA** as required.

   The interactive dialog opens from where the member can be designed.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

Design all members in a level, slope, or frame

Design plane performs a check or design of each entity in the plane, according to each entity's autodesign setting, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Structure** tree, do one of the following:

To	Do this
Design all members in a construction level	Right click the required **Level** and select **Design plane > Static/RSA** as required.
Design all members and walls in a frame	Right click the required **Frame** and select **Design plane > Static/RSA** as required.
Design all members in a slope	Right click the required **Slope** and select **Design plane > Static/RSA** as required.

NOTE

- The **Design plane** command uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It requires at least one member in the selection to be in auto-design mode.
- If any grouped members are designed and the grouped member design option is on, all members of the group are considered in the design even if they are not in the selected level, slope, or frame.

Design all members

Design members performs performs a check or design of every member according to each member's autodesign setting, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Members** tree.
3. In the context menu, select **Design members > Static/RSA** as required.

NOTE
- **Design members** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available until at least one member is in auto-design mode.

Design all walls

Design walls performs performs a check or design of every wall according to each wall's autodesign setting.

1. In the **Project Workspace**, click **Structure** tab.
2. Right-click the **Walls** tree.
3. In the context menu, select **Design walls > Static/RSA** as required.

NOTE
- **Design walls** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available until at least one wall to be in auto-design mode.

Design all members and walls

Design model performs performs a check or design of every member/wall in the model according to the individual member/wall autodesign settings, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. Right click the **Structure** branch.
3. In the context menu, select **Design model > Static/RSA** as required.

NOTE
- **Design model** uses the existing analysis results, (even if the analysis status is 'Out of Date').
- It is not available until at least one member is in auto-design mode.

Design all members of a particular section or type

Design members performs performs a check or design the selected members according to each member's autodesign setting, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Members** tree, right-click the section type or size you want to design.

3. In the context menu, select **Design members > Static/RSA** as required.

NOTE	• **Design members** uses the existing analysis results, (even if the analysis status is 'Out of Date'). • It is not available until at least one member is in auto-design mode.

Design all members in a group

Design Group performs performs a check or design of the selected member group according to the autodesign settings in the group.

1. In the **Project Workspace**, click **Groups** tab.
2. In the **Design** tree, right-click the member group you want to design.
3. In the context menu, select **Design Group > Static/RSA** as required.

NOTE	• **Design Group** uses the existing analysis results, (even if the analysis status is 'Out of Date'). • It is not available unless the **Design Groups** option is checked for the member type in **Design Settings**.

Design all members and walls in a sub structure

Design sub structure performs performs a check or design of all members/ walls in the selected sub structure according to their individual autodesign settings, (taking into account any member design groups that are active).

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure you want to design.
3. In the context menu, select **Design sub structure > Static/RSA** as required.

NOTE	• **Design sub structure** uses the existing analysis results, (even if the analysis status is 'Out of Date'). • It is not available until at least one member in the sub structure is in auto-design mode.

Apply user defined utilization ratios

By default designs and checks are performed against a utilization ratio (U/R) of 1.0.

To facilitate design optimization Tekla Structural Designer allows you to specify user defined U/Rs on an individual member basis (currently for Eurocode, US, and India head codes only).

NOTE Utilization ratio = actual performance value / maximum allowable performance value.

Overview of user defined U/R

When applied to the autodesign process user defined U/Rs introduce a measure of:

- Conservatism - if the value is less than 1.0
- Non conservatism - if the value is greater than 1.0

By not applying user defined U/Rs to the check process, you are still able to work to the original U/R of 1.0 when working in check mode.

User defined U/Rs can be applied to the following objects:

- Members; steel and concrete columns and beams, composite beams, steel joists and braces. This includes separate settings for each span/stack/panel of continuous beams/columns/walls.
- Steel Trusses - includes; Truss member top/bottom, Truss member side/ internal.
- Portal frames (Individual members within Frames are treated as steel columns and steel beams as appropriate).
- Concrete Walls and Slabs including panels, patches, and punching shear checks.
- Foundations; Pad Bases (Spread Footings), Strip Bases, Pile Caps, Mat and Piled Mat Foundation slabs and Piles.

Apply user defined U/R for autodesign only

Applying a value of utilization ratio < 1.0 for autodesign only introduces a measure of conservatism into the autodesign process but doesn't fail the member until the ratio default value of 1.0 is exceeded.

1. Select the required element or elements.

   The properties of the selected elements are displayed in the **Properties** window
2. Expand **Utilization ratio** and select **Apply (to autodesign)**, but do **not** select **Apply to check**
3. Enter the **Ratio limit** as required.

NOTE For continuous beams/columns/walls you can apply separate ratio limits for the individual spans/stacks/panels if required.

When an **Autodesign** is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. When a **Check** is performed, the check will pass provided the utilization ratio is less than 1.0.

Apply user defined U/R for autodesign and check

Applying a value of utilization ratio < 1.0 for autodesign and check introduces a measure of conservatism into the autodesign process and fails the member when the ratio value that was set is exceeded.

1. Select the required element or elements.

   The properties of the selected elements are displayed in the **Properties** window

2. Expand **Utilization ratio** and select both**Apply (to autodesign)** and **Apply to check**

3. Enter the **Ratio limit** as required.

NOTE For continuous beams/columns/walls you can apply separate ratio limits for the individual spans/stacks/panels if required.

When an **Autodesign** is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. When a **Check** is performed, the check will pass provided the utilization ratio is less than the ratio limit.

See also

Review and modify user defined utilization ratios (page 757)

Related video

User defined utilization ratio

Validate the model for design issues

If required, you can manually validate the model for design issues before the model is submitted for design in order to trap errors that will cause the design to fail.

Run design validation

NOTE As design validation is performed automatically as part of all the combined analysis and design processes, there is no requirement to run design validation manually beforehand in those cases. You might however elect to run the validation manually when performing selective member and wall checking or design.

NOTE Design validation cannot be performed if model validation is outdated.

- On the **Design** tab, click **Validate**.

  Tekla Structural Designer performs the design validation checks. If your model contains any design validation issues, warning messages appear in the **Status** tab of the **Project Workspace**.

Adjust the conditions considered in design validation

1. On the **Home** tab, click **Model Settings**.
2. Go to the **Validation** page.
3. Select the conditions that you want Tekla Structural Designer to consider during validation checks.
4. Click **OK**.

7.2 Design slabs and run punching shear checks

Slab design in Tekla Structural Designer requires a certain amount of user interaction, (which is why slabs are not considered when any of the combined analysis and design commands are run).

Slab design can therefore be considered in isolation, the process being able to be broken down into the following discreet steps:

1. Create slab patches over columns, beams, walls, or panels
2. Design slabs (the slab areas that lie outside the patches)
3. Design patches (the slab areas inside the patches)
4. Create punching shear checks as required
5. Design punching shear (page 629) for the whole model or individually

See also

Apply user defined utilization ratios (page 613)

Create and modify patches

You can apply rectangular patches of reinforcement to individual slab items to act in addition to the background reinforcement. After selecting the patch type, you can place patches in the model by clicking or boxing around elements in 2D or 3D views.

Overview of patches and patch types

Patches are used during the design of concrete slabs as a way of managing the physical and design data. Each patch defines a rectangular area of slab within which FE analysis results are collected, enabling Tekla Structural Designer to perform the design process. Design moments are calculated along result strips, embedded within each patch. Depending on the patch type, a patch can contain up to 6 result strips, catering for up to 3 strips of reinforcement in each of two perpendicular directions.

Tekla Structural Designer contains the following patch types:

- Column patch: can be placed at column stack heads
- Beam patch: can be placed along beams
- Wall patch: can be placed along walls
- Panel patch: can be placed at a specified position within the panel boundary
    - not restricted to a centralized position or to existing within one panel
    - can be positioned under loads

Patches can be either on the top or the bottom of the slab and may or may not have reinforcement defined in them. If you have not defined any reinforcement, Tekla Structural Designer uses the background reinforcement. If you have defined reinforcement, then for the top/bottom, x/y direction, you can optionally use the sum of the background + patch reinforcement option, if the patches are reasonably aligned.

NOTE Any patches may overlap on the plan view. However, during design, the eventual overlap is ignored.

Create column patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Column**.
3. In the **Properties** window, adjust the patch properties according to your needs.
    a. To specify the patch size, type values in the **Lx** and **Ly** fields.
    b. To specify the layer of the patch, select the desired option in **Surface**.

c. To have the reinforcement automatically designed, select the **Autodesign** option.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Place a patch over a specific column	• Click a column node within the slab. Tekla Structural Designer creates a patch to the selected column.
Place multiple patches	a. Move the mouse pointer to one corner of an imaginary box that will encompass the columns to which you want to create patches. b. Hold down the left mouse button. c. Drag the mouse pointer to the opposite corner of the box. d. Release the mouse button. Tekla Structural Designer creates patches to all columns entirely within the box.

Create beam patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Beam**.
3. In the **Properties** window, define the required patch width (perpendicular to the beam span) and the center slab width.

   Tekla Structural Designer recalculates the two end strips accordingly. The strips cannot be modified.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Place a patch over a specific beam span	• Click the required beam span. Tekla Structural Designer creates a patch centered on and orientated to the beam center line.
Place multiple patches	a. Move the mouse pointer to one corner of an imaginary box that will encompass the beams to which you want to create patches. b. Hold down the left mouse button. c. Drag the mouse pointer to the opposite corner of the box. d. Release the mouse button. Tekla Structural Designer creates patches to all beams entirely within the box.

Create wall patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Wall**.
3. In the **Properties** window, adjust the patch properties according to your needs.
    a. To specify the layer of the patch, select the desired option in **Surface**.
    b. To have the reinforcement automatically designed, select the **Autodesign** option.
    c. Define the required patch width perpendicular to the wall.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this

Create a patch along a specific wall	a. In the **Properties** window, set **Create Mode** to **Single Patch Along Wall**. b. Click the required wall. Tekla Structural Designer creates a patch, centered on and orientated to the wall center line.
Place an internal patch and two end patches along a specific wall	a. In the **Properties** window, set **Create Mode** to **Internal With End Patches**. b. Click the required wall. Tekla Structural Designer creates an internal patch and two end patches, centered and orientated to the wall center line.
Place an end patch at one end of a specific wall:	a. In the **Properties** window, set **Create Mode** to **End Patch at Wall End**. b. Click near the required end of the wall. Tekla Structural Designer creates a patch at the end of the wall that is closest to the point that you clicked.
Place an internal patch part way along a specific wall	a. In the **Properties** window, set **Create Mode** to **Internal Patch**. b. Click to define the start point of the patch along the required wall. c. Click to define the end point of the patch along the required wall. Tekla Structural Designer creates a patch between the selected points.
Create patches along multiple walls	a. In the **Properties** window, set **Create Mode** to **Single Patch Along Wall** or **Internal With End Patches**. b. Move the mouse pointer to one corner of an imaginary box that will encompass the walls to which you want to create patches. c. Hold down the left mouse button. d. Drag the mouse pointer to the opposite corner of the box. e. Release the mouse button. Tekla Structural Designer creates patches to all walls entirely within the box.

Create panel patches

1. On the **Design** toolbar, click the arrow on the left side of the **Patch** list.
2. In the list, select **Patch Panel**.
3. In the **Properties** window, adjust the patch properties according to your needs.
    a. To specify the patch size, type values in the **Lx** and **Ly** fields.
    b. To specify the layer of the patch, select the desired option in **Surface**.

c. To have the reinforcement automatically designed, select the **Autodesign** option.

NOTE Only select the **Consider patch surface moments only** option if you do not want Tekla Structural Designer to consider the opposite surface.

One example where you would need the option is when the check of slab reinforcement in the opposite surface fails, but another patch exists in the same location at that face. When the second patch is designed, additional reinforcement is provided at the surface, so the surface check is not required for the original patch.

4. According to your needs, do one of the following:

To	Do this
Place the patch at the centroid of a specific panel	a. In the **Properties** window, ensure that the **Create Patch at Centroid** option is selected. b. Click anywhere within the required panel. Tekla Structural Designer creates a patch at the panel centroid.
Place the patch elsewhere within a specific panel	a. In the **Properties** window, ensure that the **Create Patch at Centroid** option is cleared. b. Still in the **Properties** window, do one of the following: • To define the patch by its corners, select the **Define corner points** option. • To define the patch by its center, clear the **Define corner points** option. c. In the panel, click the points that define the patch. Tekla Structural Designer creates a patch at the selected position.
Create multiple centroid patches	a. In the **Properties** window, ensure that the **Create Patch at Centroid** option is selected. b. Move the mouse pointer to one corner of an imaginary box that will encompass the panels to which you want to create patches. c. Hold down the left mouse button. d. Drag the mouse pointer to the opposite corner of the box. e. Release the mouse button. Tekla Structural Designer creates patches to the centroids of all panels entirely within the box.

Modify patch properties

1. Hover the mouse pointer over the patch that you want to modify.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, use the arrow keys to navigate to the patch name and press **Enter**.
3. In the **Properties** window, modify the patch properties according to your needs.
   a. To specify the patch size, type values in the **Lx** and **Ly** fields.
   b. To specify the layer of the patch, select the desired option in **Surface**.
   c. To have the reinforcement automatically designed, select the **Autodesign** option.
   d. To specify the directions of the reinforcement that you want to design, select the appropriate option in **Consider Strips**.
   e. To specify the type and direction of the reinforcement that you want to design, select the appropriate option in **Reinforcement**.
   f. To specify the required width for each strip, type a value in the **Width** field.
   g. To specify whether each strip should be designed for the average or maximum design force, select the appropriate option in **Design Force**.

Resize patches

1. Hover the mouse pointer over the patch that you want to resize.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, use the arrow keys to navigate to the patch name and press **Enter**.
3. Hover the mouse pointer over the patch edge or corner that you want to adjust.

   The **Select Entity** tooltip appears.
4. In the **Select Entity** tooltip, use the arrow keys to navigate to the edge or corner node and press **Enter**.
5. Hold down the left mouse button and drag to move the edge or corner.

Design and check slabs

After creating any necessary patches, you can determine the top and bottom reinforcement requirements in all the slab item areas that lie outside of the patch areas. Because patch areas are excluded from slab item design, you should create patches before designing slab items.

NOTE Slabs can only be designed/checked after slab design moments have been established from a suitable analysis. If valid analysis results do not yet exist you would need to create them (by, for example, clicking **Analyze All (Static)** from the **Analyze** toolbar).

Check an individual slab item

- Do one of the following:

To	Do this
Check an individual slab from a 2D or 3D view	1. Hover the mouse pointer over the slab. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Check Panel**.
Check an individual slab from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs branch, locate the slab reference. 3. Right click and select **Check Panel**.

NOTE **Check Panel** uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

-

Check all slab items

1. In the **Project Workspace**, click **Structure** tab.
2. Right click on the **Structure** branch.
3. In the context menu, select **Check model slabs** as required.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

-

Check all slab items on a single floor

- Do one of the following:

To	Do this
Check all slabs in a floor from a 2D view	1. Open a 2D view of the floor that you plan to design. 2. Right-click anywhere in the view. 3. In the context menu that appears, select **Check Slabs**. Tekla Structural Designer checks the reinforcement of all slab items on the floor.
Check all slabs in a floor from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Levels branch, right click the required **Level** and select **Check plane slabs**.

NOTE This command uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

•

Check all slab items in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure containing the slabs you want to check.
3. In the context menu, select **Check sub structure slabs**.

   Tekla Structural Designer checks the reinforcement of all slab items on the floor.

See also:

•

Design an individual slab item

• Do one of the following:

To	Do this
Design an individual slab from a 2D or 3D view	1. Hover the mouse pointer over the slab. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Design Member**.
Design an individual slab from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs branch, locate the slab reference. 3. Right click and select **Design panel**.

NOTE **Design panel** uses the existing analysis results, (even if the analysis status is 'Out of Date').

See also:

•

Design all slab items

- On the **Design** tab, click **Design Slabs**.

  Tekla Structural Designer designs or checks all slab items in the model according to their autodesign settings.

See also:

•

Design all slab items on a single floor

- Do one of the following:

To	Do this
Design all slabs in a floor from 2D view	1. Open a 2D view of the floor that you plan to design. 2. Right-click anywhere in the view. 3. In the context menu that appears, select **Design Slabs**.
Design all slabs in a floor from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Levels branch, right click the required **Level** and select **Design plane slabs**.

See also:

•

Design all slab items in a sub structure

1. In the **Project Workspace**, click **Structure** tab.
2. In the **Sub Structures** tree, right-click the sub structure containing the slabs you want to design.
3. In the context menu, select **Design sub structure slabs**.

   Tekla Structural Designer designs all slab items in the sub structure, possibly selecting new reinforcement for them.

See also:

•

Design and check patches

When that the slab item designs resulting from the slab or mat design process are satisfactory, you can then proceed to design patches. You should design slab items before designing patches, as the additional patch reinforcement requirement takes account of the existing level of reinforcement provided by the slab items.

NOTE For each strip within each patch, Tekla Structural Designer determines the area of the required steel ($A_{s,reqd}$) and the area of the provided steel ($A_{s,prov}$). The $A_{s,prov}$ calculation always considers the patch reinforcement, but if necessary and allowed, you can also include additional background reinforcement in the calculation.

Check an individual patch

1. Hover the mouse pointer over the patch that you want to check.
2. Right-click the patch.
3. In the context menu that appears, select **Check Slab Patch**.

   Tekla Structural Designer displays the results of the check in a new dialog.

Check all patches in the model

- The same command can be accessed in two ways:
  - In a 3D Structural View, right click on the background and choose **Check Patches** from the context menu.
  - In the **Project Workspace** Structure tree, right-click on the **Structure** branch and choose from the context menu.

  Tekla Structural Designer checks the reinforcement of all patches in the model.

Check all patches on a single floor

1. Open a 2D view of the floor that you plan to design.
2. Right-click anywhere in the view.
3. In the context menu that appears, select **Check Patches**.

   Tekla Structural Designer checks the reinforcement of all patches on the floor.

Design an individual patch

1. Hover the mouse pointer over the patch that you want to design.
2. Right-click the patch.
3. In the context menu that appears, select **Design Slab Patch**. Tekla Structural Designer displays the results of the design in a new dialog.

   The results of the design are displayed in a new dialog.

Design or check all patches in the model

- The same command can be accessed in three ways:
  - On the **Design** tab, click  **Design Patches**
  - In a 3D Structural View, right click on the background and choose **Design Patches** from the context menu.
  - In the **Project Workspace** Structure tree, right-click on the **Structure** branch and choose from the context menu.

  Tekla Structural Designer designs or checks all patches in the model according to their autodesign settings.

Design all patches on a single floor

1. Open a 2D view of the floor that you want to design.
2. Right-click anywhere in the view.
3. In the context menu that appears, select **Design Patches**.

   Tekla Structural Designer designs all patches on the floor, possibly selecting new reinforcement for them.

Create punching shear checks

You can apply punching shear checks to flat slabs and foundation mats by using the **Add Check** command on the **Design** tab.

Punching check locations

You can apply punching shear checks as follows:

In flat slabs:

- Concrete columns supporting a flat slab with and without drops
- Concrete columns supported by a flat slab
- Concrete columns through a flat slab with or without drops

- Concrete walls supporting a flat slab
- Concrete walls supported by a flat slab
- Concrete walls through a flat slab
- Steel columns supported by a flat slab
- Point loads at either face of a flat slab

In foundation mats:

- Concrete columns supported by a foundation mat
- Concrete walls supported by a foundation mat
- Steel columns supported by a foundation mat
- Point loads on a foundation mat
- individual piles supporting a foundation mat

Punching check axis orientation

- When you apply a punching shear check to a column, the check Y and Z axes are automatically orientated to align with the column major and minor axes. The two axis systems are locked together, so if the column is rotated, the punching check axes also rotate.
- When you apply a punching shear check to a point load, you must manually define the punching shear check Z axis orientation. You can do this in the **Properties** window by specifying the point load orientation in relation to the global Y axis.

  NOTE Point load properties (including point load orientation, point load breadth, and point load depth) are not applicable when you apply a check to a column.

Create punching check items

NOTE Punching check items can generally be created in 2D and 3D Structural or Review Views, apart from checks around walls which can only be created in Structural Views, but **not** currently in Review Views.

Punching check items are the objects to which the punching shear information and calculations are attached. To create punching check items, do the following:

1. On the **Design** tab, click **Add Check**.
2. Go to the **Properties** window-
3. Ensure that the location for the slab tension reinforcement is correct.

4. Adjust the remaining properties according to your needs.

NOTE The point load properties are only relevant if you are adding the check to a point load position.

5. To create the check, do one of the following:
   - Hover the mouse pointer over the desired column node, wall node, or point load. When the **Pick Punching Check Location** tooltip appears, click to add the punching shear at the selected node.
   - Hold down the left mouse button and drag a box to add punching check items to all potential check locations within the box.

Specify stud rail reinforcement

RESTRICTION Stud rail reinforcement is only available for Eurocode and ACI head codes.

You can specify an orthogonal or circular arrangement of stud rails, and then check or design it as required.

1. Create a punching check item as instructed above.
2. Go to the **Properties** window.
3. Select the **Use reinforcement** option.
4. Select whether you want to autodesign the reinforcement.
5. Adjust the remaining reinforcement properties according to your needs.

Modify the properties of existing punching check items

1. Right-click the punching check item.
2. In the context menu, select **Edit Reinforcement**.

Design and check punching shear

Overview of the Design Punching Shear command

The **Design Punching Shear** command calculates an applied load on the slab, accounting for the difference in column/wall axial load and bending. In addition, the command checks the slab shear stresses, accounting for the reinforcement present (background and column/general patches).

Provided that you are using the Eurocode or ACI head code, you can also design the punching reinforcement as an orthogonal or circular arrangement of stud rails at the following locations:

In flat slabs:

- Concrete columns supporting a flat slab without drops
- Concrete columns supported by a flat slab
- Concrete columns through a flat slab without drops
- Point loads at either face of a flat slab

In foundation mats:

- Concrete columns supported by a foundation mat
- Point loads on a foundation mat

Column head drops and the presence of openings within a certain distance of the punching shear boundary are taken into account in the punching shear calculations.

Where punching checks are closely spaced and the perimeters overlap the checks are beyond scope.

NOTE
- Since the checks are dependent on using the correct levels of slab reinforcement (typically provided by patch reinforcement), do not check punching before designing the patches.
- View punching checks applied to walls with caution. Their applicability to long walls is particularly questionable, as the check does not consider the potential for stress concentrations at the ends of the wall.

Check punching shear for an individual punching check item

- Do one of the following:

To	Do this
Check an individual punching check item from a 2D or 3D view	1. Hover the mouse pointer over the punching check. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Check Punching Shear**.
Check an individual punching check item from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs> Punching Checks branch, locate the check reference. 3. Right click and select **Check Punching Shear**.

Check all punching check items

1. In the **Project Workspace**, click **Structure** tab
2. Right click the Structure branch and select **Check Punching Shear**

Check all punching shear check items on a floor

1. In the **Project Workspace**, click **Structure** tab.
2. In the Levels branch, right click the required **Level** and select **Check Punching Shear**.

Design all punching check items

- Do one of the following:

To	Do this
Design all punching check items from the ribbon	1. On the **Design** tab, click **Design Punching Shear**.
Design all punching check items from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. Right click the Structure branch and select **Design Punching Shear**.

Tekla Structural Designer performs punching shear checks for all punching check items in the model based on their individual auto-design setting. If the punching check item does not connect to a flat slab, punching shear is flagged as beyond scope.

Design all punching shear check items on a floor

1. In the **Project Workspace**, click **Structure** tab.
2. In the Levels branch, right click the required **Level** and select **Design Punching Shear**.

Design an individual punching check item

- Do one of the following:

To	Do this
Design an individual punching check item from a 2D or 3D view	1. Hover the mouse pointer over the punching check. 2. If a different entity is highlighted, press the <down arrow> cursor key until the correct reference is shown in the Select Entity tooltip. 3. Right-click, and from the context menu that appears, select **Design Punching Shear**.

Design an individual punching check item from the Structure Tree	1. In the **Project Workspace**, click **Structure** tab. 2. In the Slabs> Punching Checks branch, locate the check reference. 3. Right click and select **Design Punching Shear**.

Tekla Structural Designer displays the results of the design in a new dialog box.

7.3 Design timber and precast members using Tekla Tedds

Timber and precast members that have been modelled and analysed in Tekla Structural Designer can then be designed in Tekla Tedds if a licence is available.

For further information, see:

-
-

7.4 Create and run floor vibration checks

To establish the response of the floor to dynamic excitation you can apply a floor vibration checks. These are applied over a user-defined rectangular or polygon shaped slab area by using the **Add Check** command in the **Floor Vibration** group. In order to create the check, you need to identify a primary beam, secondary beam and critical slab item. In addition, you must specify associated data in order to perform the floor vibration calculation.

See also

Create and modify floor vibration checks (page 632)

Run floor vibration checks (page 635)

Create and modify floor vibration checks

To establish the response of the floor to dynamic excitation you can apply a floor vibration checks. These are applied over a user-defined rectangular or polygon shaped slab area by using the **Add Check** command in the **Floor Vibration** group. In order to create the check, you need to identify a primary beam, secondary beam and critical slab item. In addition, you must specify associated data in order to perform the floor vibration calculation.

Vibration of Floors to SCI P354 Handbook; Vibration of Floors to DG11 Handbook; Vibration of Floors to SCI P354 Example

Create floor vibration check items

Floor vibration check items are the objects to which the floor vibration check information and calculations are attached.

NOTE Floor vibration check items can only be created in 2D Views.

1. On the **Design** tab, click **Add Check**.
2. Go to the **Properties** window.
3. In **FloorPlateDefinition**, select whether you want to create a rectangular or polygonal slab.
    a. If you selected the **Rectangular** option, adjust the local x angle to define the angle at which the rectangle will be drawn, if necessary.
4. To create a check area, in the model, do one of the following depending on the slab shape:
    - **Rectangular**: Click once to define the first corner of the rectangle, and again to define the opposite corner.
    - **Polygon**: Click to define the corners of the polygon. In order to close the shape, click the first point again.
5. Click to define the primary beam.

    The primary beam properties appear in the **Properties** window.
6. Click to define the secondary beam.

    The secondary beam properties appear in the **Properties** window.
7. Click to define the critical slab item.

    NOTE If the critical slab item is a composite slab, Tekla Structural Designer automatically defines the slab item properties. In other cases, you need to input the properties yourself.

    The critical slab item properties appear in the **Properties** window.
8. In the **Properties** window, review and adjust the check item properties.
9. Click anywhere in the model to create the check item.

    Tekla Structural Designer highlights the primary beam, secondary beam and critical slab item of the current check item.
10. Create new checks, or press **Esc** to finish creating floor vibration check items.

Create floor vibration checks that consider two or three adjoining spans

Creating two-span or three-span floor vibration checks is almost identical to that required for single spans. When creating the check item, do the following:

- In the **Properties** window, set the **Adjoining Spans** property of a primary and/or secondary beam to **Two span** or **Three span**.

NOTE If the **Two span** option is selected, note the following:

- When you hover the mouse pointer to select the beam, only beams of two or more spans are available.
- When a beam is highlighted, note that the beam directly under the mouse pointer will become the critical beam, and the second highlighted beam will become the adjoining beam.
- The second highlighted beam (the adjoining beam) will be the beam closest to the mouse pointer position. That's why, in order to highlight the adjoining beam at a particular end of the critical beam, you can move the mouse pointer toward that end of the critical beam.

NOTE If the **Three span** option is selected, note the following:

- When you hover the mouse pointer to select the beam, only beams of three or more spans are available to be highlighted,
- When a beam is highlighted, note that the beam directly under the mouse pointer will become the critical beam, and the second highlighted beam will become the adjoining beam.
- The second highlighted beam (the adjoining beam) will be the beam closest to the mouse pointer position. That's why, in order to highlight the adjoining beam at a particular end of the critical beam, you can move the mouse pointer toward that end of the critical beam.

Modify the properties of existing floor vibration check items

1. In **Scene Content**, ensure that Floor Vibration Checks have been turned on.
2. Hover the mouse pointer over the slab area where the floor vibration check item lies.

   The **Select Entity** tooltip appears.
3. Navigate to the floor vibration check item name by using the arrow keys, and click **Enter**.

   The floor vibration check properties appear in the **Properties** window.

4. In the **Properties** window, adjust the check item properties according to your needs.

Run floor vibration checks

Check vibration for all floor vibration check items

- On the **Design** tab, click **Check Floor Vibration**.

  Tekla Structural Designer performs floor vibration checks for all floor vibration check items in the model.

Check floor vibration for an individual floor vibration check item

1. Hover the mouse pointer over the floor vibration check item that you want to check.

   The **Select Entity** tooltip appears.
2. If necessary, navigate to the floor vibration check item name by using the arrow keys.
3. Right-click the check item.
4. In the context menu, select **Check Floor Vibration**. Tekla Structural Designer displays the results of the design in a new dialog box.

7.5 Create and check steel connections

Check simple connection resistance

Tekla Structural Designer provides a quick and simple way of checking the shear capacity of simple beam connections, and the axial capacity of brace connections.

Overview

Tekla Structural Designer has a Connection Resistance database for simple beams and braces. For simple beams the database contains pre-defined types with pre-defined resistances (page 895) to Eurocode and US head codes.

You can add user-defined connection types (page 898) and user-defined resistances (page 898) to the database for any head code, for steel and cold formed materials. Once defined, these can then be used across all projects.

Each resistance value in the database has an 'active' flag which must be 'on' for the resistance to be considered. (Simple beams and braces in the model that have no active resistances are not checked.)

Having configured the active resisistances in the database as required, the checks are then performed in the model as follows:

- For each simple beam the active shear resistances for each available connection type are read from the database.
- These resistances are compared to the maximum shear force at the connection (determined from a 3D analysis)
- The connection configuration is optimized accordingly for each connection type.
- A similar process is then followed for each brace, by comparing the active axial resistances to the maximum applied axial force.

Results can be displayed in a review data table, or output via a connection resistance report.

Specify 'active' connection resistances (Eurocodes)

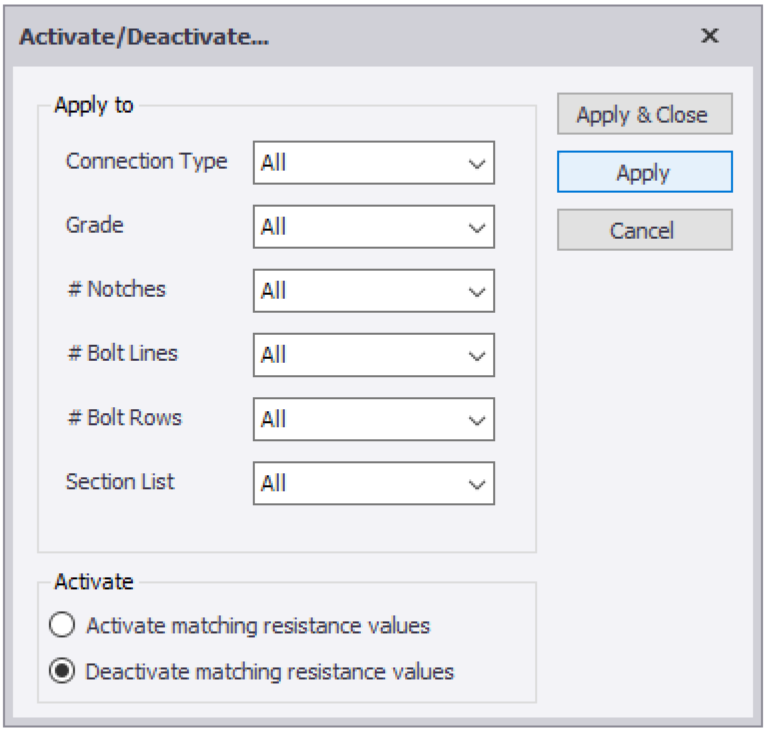
Since only active resistances are considered in the check, you can deactivate resistances for those connection configurations you do not want to be considered.

The active status of resistances can be set on or off directly (by ticking or unticking a checkbox in the Active column) but a quicker way of changing the active status is with the Activate/Deactivate dialog.

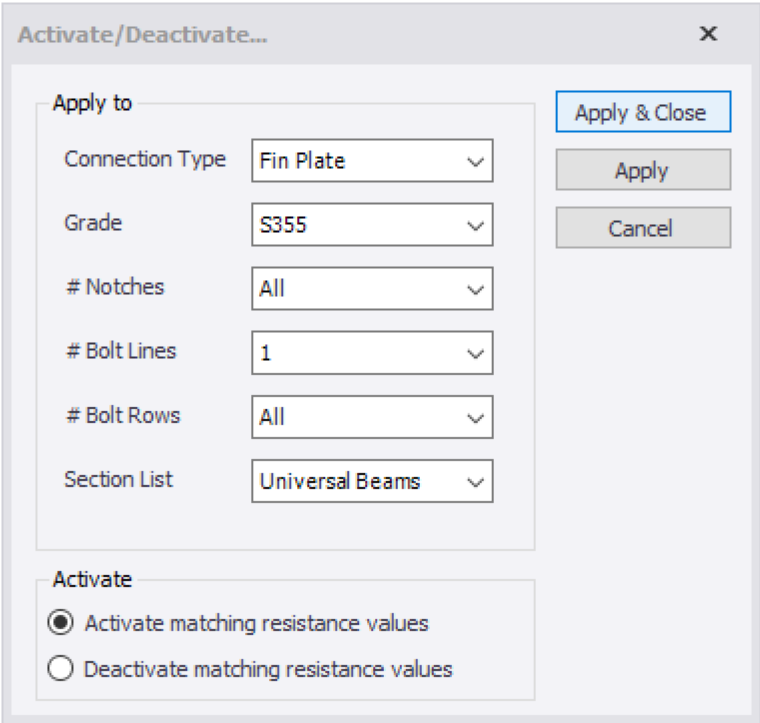
1. In the Connection Resistance dialog (page 1174), click **Activate/ Deactivate...**
2. Set the filters in the dialog, then set either Activate or Deactivate, as required.
3. Click **Apply** if you want to change further Active settings, or click **Apply & Close** to finish.

Example: From the pre-defined connection types you want only Fin Plates to be active, only for S355 Universal Beams, and only those Fin Plates with 1 line of bolts. There are a number of ways to achieve this using **Activate/ Deactivate** but one way would be as follows:

1. Click **Activate/Deactivate...**
2. Deactivate everything - select 'All' in each of the Activate/Deactivate filters, select 'Deactivate matching resistance values', then **Apply**



3. To activate only Fin Plates for S355 Universal Beams with 1 line of bolts
   a. select Fin Plate in the Connection Type filter,
   b. select S355 in the Grade filter,
   c. select 1 in the # Bolt Lines filter,
   d. select Universal Beams in the Section List filter,
   e. select 'All' in the other filters (# Notches, # Bolt Rows),
   f. select 'Activate matching resistance values',
   g. click **Apply & Close**



Following step 3, you will find ticks in the Active column only for Fin Plates with S355 Universal Beams with 1 line of bolts, *but note these changes are not saved to the database, and do not become active in Reports or in Tabular Data, until clicking OK in the main dialog.*

Specify 'active' connection resistances (US)

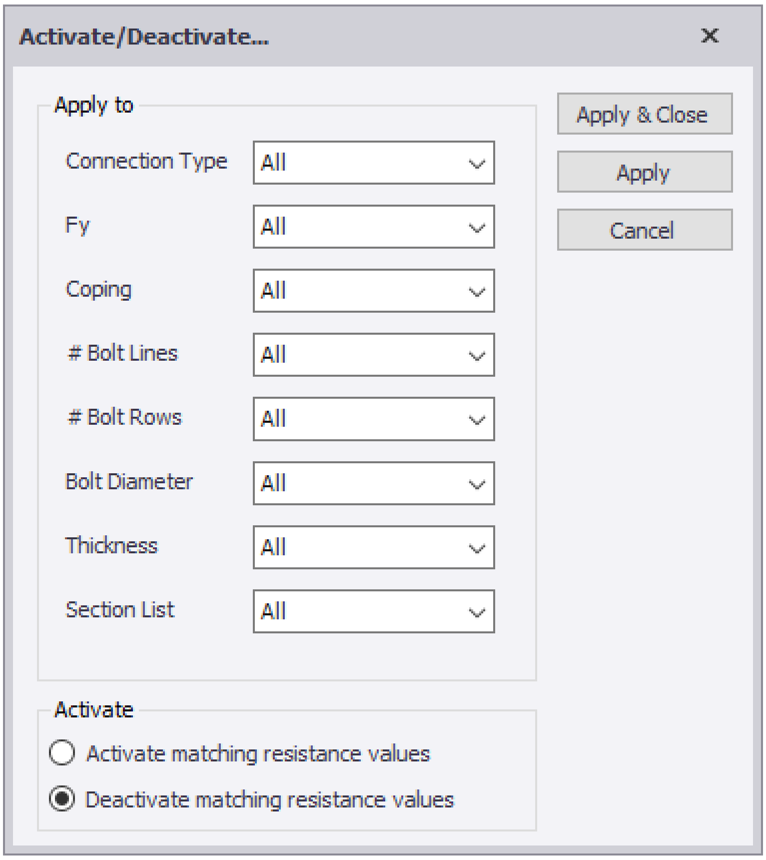
Since only active resistances are considered in the check, you can deactivate resistances for those connection configurations you do not want to be considered.

The active status of resistances can be set on or off directly (by ticking or unticking a checkbox in the Active column) but a quicker way of changing the active status is with the Activate/Deactivate dialog.

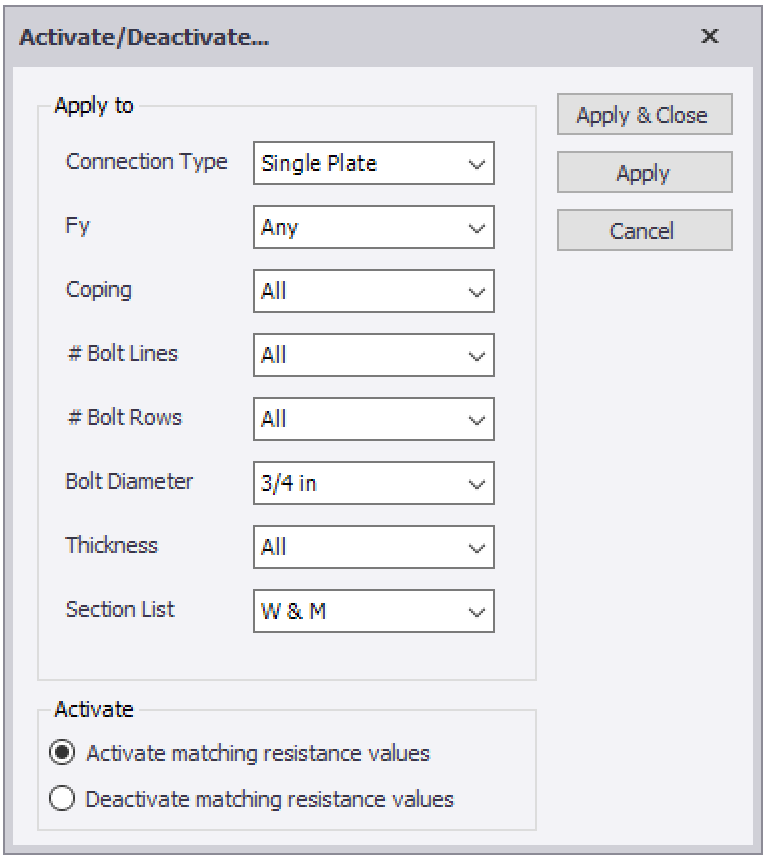
1. In the Connection Resistance dialog (page 1174), click **Activate/ Deactivate...**
2. Set the filters in the dialog, then set either Activate or Deactivate, as required.
3. Click **Apply** if you want to change further Active settings, or click **Apply & Close** to finish.

Example: From the pre-defined connection types you want only Single Plates to be active, for W & M beams of 'Any' Fy, only for 3/4 in bolts, but for all defined thicknesses of plate. There are a number of ways to achieve this using **Activate/Deactivate** but one way would be as follows:

1. Click **Activate/Deactivate...**
2. Deactivate everything - select 'All' in each of the Activate/Deactivate filters, select 'Deactivate matching resistance values', then **Apply**



3. To activate only Single Plates for W & M Beams of 'Any' Fy, with 3/4 in diameter bolts and all plate thickness
   a. select Single Plate in the Connection Type filter,
   b. select Any in the Fy filter, select 3/4 in in the Bolt Diameter filter,
   c. select W & M in the Section List filter,
   d. select 'All' in the other filters (Coping, # Bolt Lines, # Bolt Rows, Thickness),
   e. select 'Activate matching resistance values',
   f. click **Apply & Close**



Following step 3, you will find ticks in the Active column only for Single Plates, with Any Fy selected, and W & M Beams with 3/4 in diameter bolts, *but note these changes are not saved to the database, and do not become active in Reports or in Tabular Data, until clicking OK in the main dialog.*

Run resistance checks

Provided you have an appropriate set of 'active' resistances, the checks are performed automatically, using the current analysis results - you can proceed directly to a review of the check results.

- If necessary, analyse the model to generate an up-to-date set of analysis results,
- Review the check results, see: ***Display connection resistance checks in a review data table*** below.

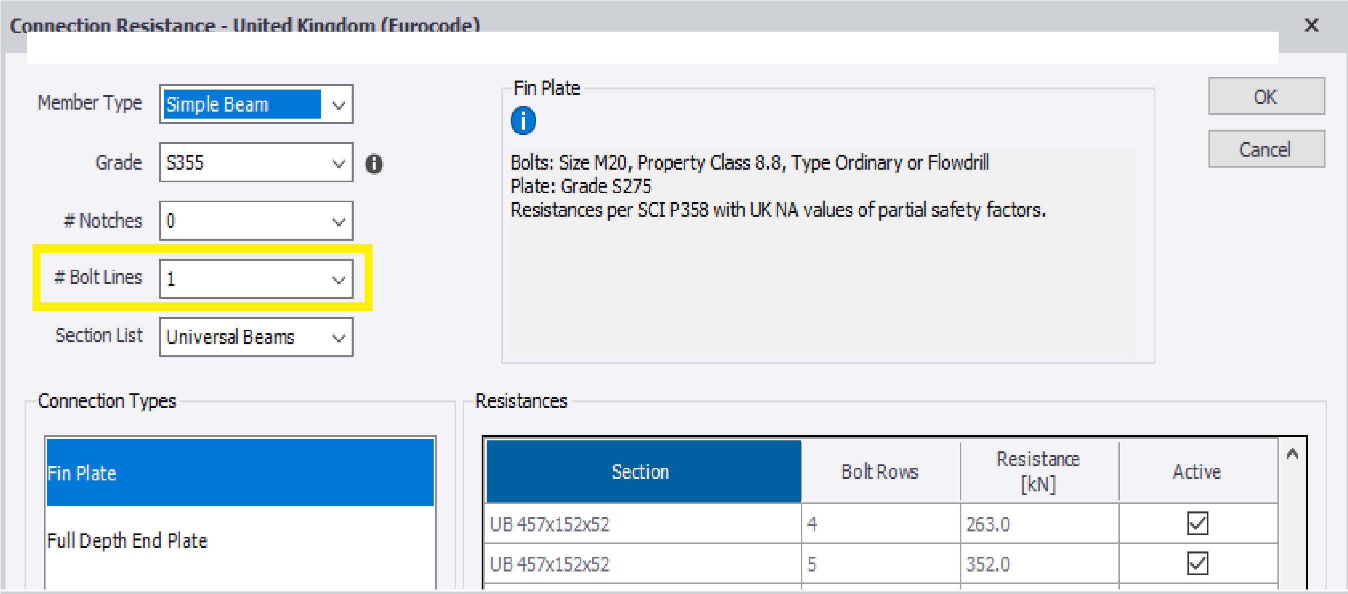
The connection optimization process

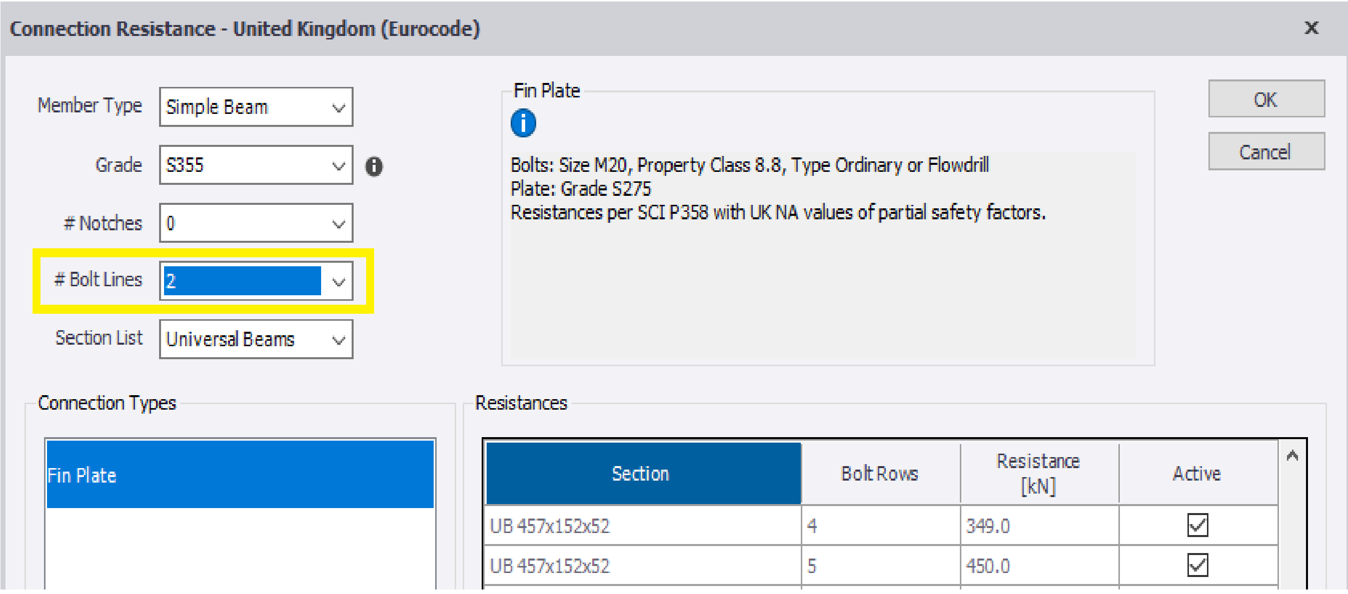
Tekla Structural Designer carries out an optimization process to find the first passing Active resistance based on the name of the Connection Type and the number of Bolt Lines and Bolt Rows assigned to that name.

For Eurocodes, separate check results are reported for each 'active' connection type and each notch variation.

For US codes, separate check results are reported for each 'active' connection type and each coping variation.

Example: to Eurocode, a Fin Plate for an S355 UB 457x152x52 with 1 and then 2 lines of bolts has *pre-defined* resistances as shown below. All resistances are indicated as Active.





If an S355 UB 457x152x52 in a model is found to have an applied major shear force of 250 kN then the connection resistance check will report that a Fin Plate with 1 bolt line and 4 bolt rows (resistance 263 kN) is adequate.

If the applied major shear force is 300 kN then the connection resistance check will report that a Fin Plate with 1 bolt line and 5 bolt rows (resistance 352 kN) is adequate.

If the applied major shear force is 355 kN then the connection resistance check will report that a Fin Plate with 2 bolt lines and 5 bolt rows (resistance 450 kN) is adequate.

For each of the applied major shear forces given above only the *first passing* resistance is reported, by looping through the *Active* bolt rows for 1 bolt line followed, where necessary, by looping through the *Active* bolt rows for 2 bolt lines.

If Tekla Structural Designer gets to the end of these loops and still doesn't find a passing resistance then it reports the last resistance found and a fail status e.g. if the applied major shear force is 455 kN then the connection resistance check will report that a Fin Plate with 2 bolt lines and 5 bolt rows (resistance 450 kN) fails.

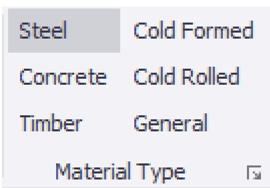
Now suppose that the UB 457x152x52 with 1 bolt line and 5 bolt rows (resistance 352 kN) is *not* Active. The first shear check above (applied force 250 kN) would be unchanged but the second check (applied force 300 kN) would now report that a Fin Plate with 2 bolt lines and 4 bolt rows (resistance 349 kN) is adequate. This illustrates the interplay of the resistance Active status with the optimization process.

Display connection resistance checks in a review data table

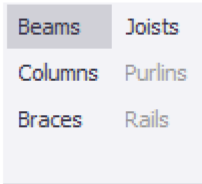
To create a tabular connection resistance summary:

1. If necessary, change the view regime (page 89) to a **Review View**.
2. On the **Review** ribbon tab, click **Tabular Data**.

   A **Review Data** tab opens on the ribbon and a **Review Data View** is displayed.
3. On the **Review Data** ribbon tab, in the list in the **View Type** group, select **Connection Resistance**.
4. In the **Material Type** group, select the material (either **Steel**, or **Cold Formed**).

   The available selections in the **Characteristic** group (and the other groups) are reduced to those appropriate to the selected material.
5. In the **Characteristic** group, select the required characteristic (either **Beams**, or **Braces**).



6. In the **Construction** group, select **Composite**, **Non-composite**, or both.

7. In the **Fabrication** group, select the required fabrication types.

A resistance check is provided for the filtered selection for every section where a resistance has been specified in the database.

NOTE A resistance check is not provided for beams that do not have simple connections.

NOTE If there is uplift in any load combination then the maximum uplift shear force will be displayed, with a warning that connection resistance is not checked for uplift condition.

NOTE Each time you re-analyse the building, if you have the Connection Resistance data table open you must close and then re-open it to update the results.

Create and display a connection resistance report

1. On the **Report** tab, use the droplist to select the existing **Connection Resistance** report.
2. Click **Model Report...**
3. Review the report structure and adjust as required.

4. Click **OK** to save the report.
5. Click **Show Report**

Related video

Predefined connection resistance database for Eurocode and AISC

Create and check column base plates

Base plates can only be created under steel columns. How these can be checked depends on the head code being worked to.

Create column base plates

Tekla Structural Designer automatically adds a base plate on creation of a new steel column if a support is required.

NOTE If an existing steel column is missing a base plate you can add one by clicking the (Add) Base Plate button (page 207) on the Design ribbon and then select the column.

Check column base plates

How base plates are checked depends on the head code being worked to.

- **AISC and Eurocode**

  Base plates can be checked directly in Tekla Structural Designer, but cannot be designed in Tekla Connection Designer. For details, see: in the Steel Design Handbook

- **British Standards**

  Base plates **cannot** be checked directly in Tekla Structural Designer, but **can** be designed in Tekla Connection Designer. For details, see: Create and design other connections (page 655)

- **Australian & Indian Standards**

  Base plates **cannot** be checked in Tekla Structural Designer, or designed in Tekla Connection Designer.

Create and size SidePlate connections

SidePlates are a US proprietary connection system that can be applied in Tekla Structural Designer but are considered separately to other connection types.

Click the following links to find out more:

- For background to their usage in Tekla Structural Designer, see

  SidePlate connections theory (page 646)
- For modeling instructions, see

  Create SidePlate connections (page 653)

SidePlate connections theory

Overview of SidePlate connections

SidePlate® is a type of moment connection for connecting I or HSS section beams to I, HSS, built up box or built up WF (cruciform) section columns.

SidePlates are welded to the column section in the fabrication shop. Plates or angles are similarly welded to beam ends.

The column and beam units are transferred to site where the columns are erected and the beams are lifted into place and either bolted or welded to the columns. SidePlate connections are used on buildings of 1 to 30 stories tall.

Within the structure, the beam end is held in position by SidePlates. The net result is moment connection with a very stiff section of column and a stiffened beam end.

SidePlate connection types can be:

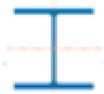
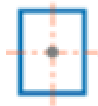
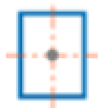
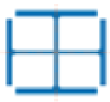
- Non-Seismic moment connections
  - A beam/column moment connection anywhere in the structure
- Seismic moment connections
  - A seismic beam/column moment connection within any of the following seismic force resisting systems Ordinary Moment Frame (OMF), Intermediate Moment Frame (IMF) or Special Moment Frame (SMF).

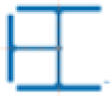
SidePlate connections work for all WF and HSS sections in the AISC steel book, and for all UB and UKB sections. However it is worth noting the SidePlate datafile may NOT have a full set of values for all 'clear spans' or all %M_ps for all these sections; if this is the case then no SidePlate connection can be applied at the joint.

Tekla Structural Designer can perform an initial sizing for the SidePlate connection, but the final detailed design of the SidePlate connection has to be undertaken in SidePlate® software.

Permitted SidePlate connections in Tekla Structural Designer

RESTRICTION In Tekla Structural Designer, sideplate connections are currently only permitted for US Customary sections and not for metric sections.

Column		Side Plate Permitted Connections					Tekla Structural Designer Status
		Non-Seismic Moment Connections		Seismic Moment Connections			
		Beam		Beam			
		I	HSS	I	HSS		
WF	Flange	yes	yes	yes	yes		Modeled Analyzed Designed
	Web	yes	yes	Warning	Warning		
HSS	Flange	yes	yes	yes	yes		Modeled Analyzed Designed
	Web	yes	yes	yes	yes		
Plated box	Flange	yes	yes	yes	yes		Modeled Analyzed **Not Designed**
	Web	yes	yes	yes	yes		
WF + 2xWT	Flange	yes	yes	yes	yes		**Not Modeled** **Not Analyzed** **Not Designed**
	Web	yes	yes	yes	yes		

		Side Plate Permitted Connections					
		Non-Seismic Moment Connections		Seismic Moment Connections			Tekla Structural Designer Status
		Beam		Beam			
Column		I	HSS	I	HSS		
WF + 1xWT	Flange	yes	yes	yes	yes		**Not Modeled** **Not Analyzed** **Not Designed**
	Web	yes	yes	Warning	Warning		

SidePlate workflow in Tekla Structural Designer

A brief description of the SidePlate workflow in Tekla Structural Designer is given here with more detail available in the subsequent topics.

- **Modeling**: You define SidePlate connections by changing properties of the relevant beam or beams.

> **NOTE** The **Update Connections** process used for defining other connection types is **not** applicable for SidePlate.
>
> Connection objects are not created for any point on a steel column where *any* of the beams attached at that point have a SidePlate connection.

- **Visualisation**: Each connection is initially shown in the graphics as a simple box. After sizing has been performed it is shown as a pair of plates.
- **Validation**: Some preliminary checks are carried out in Model validation (page 338), (additional checks are done subsequently in **Initial sizing** and **Analysis model adjustment**). Any problems do not stop the analysis and design process but mean the relevant beam and or column will not benefit from the enhanced properties.
- **Initial sizing**: The initial SidePlate dimensions are determined during the Design Steel (Static & RSA) and Design All (Static & RSA) processes, using data provided by SidePlate and incorporated into Tekla Structural Designer.

> **NOTE** After the initial sizing process, any changes to loading, (individual loads, loadcases and combinations), will not trigger change control for the SidePlates, i.e. they will keep those initial sizes. You can

reset the status using Review mode (page 754) if you decide that the sizing process needs to be run again.

- **Analysis model adjustment**: Where valid SidePlate dimensions exist, the analysis model passed to the solver is adjusted to reflect the increased stiffness of the SidePlate connections. This is done during stand-alone analysis as well as Design.
- **Steel member design**: If SidePlate connections exist, they are taken into account in the member design for steel beams and columns. The changes affect both static and seismic design.
- **Review**: The SidePlate connection status can be seen easily in a Review view (page 754), and tooltips give more information.
- **Reports**: A “SidePlate Connection Report” can be generated.
- **Final design**: SidePlate developers can use the **Tekla Structural Designer API** in order to fully design a SidePlate connection defined and initially sized in Tekla Structural Designer

NOTE Design of SidePlate connections is beyond scope for both Tekla Connection Designer and export to the IDEA StatiCa Connection Design program.

SidePlate 'joint' and 'connection' terminology

NOTE
- A SidePlate **joint** exists at a column node and consists of 1 to 4 valid connections.
- Every suitable beam framing into the column node can create a SidePlate **connection**.

A SidePlate connection exists at End 1 and / or End 2 of a steel beam where **Apply SidePlate Connection** is true. It will still exist even if other data makes it invalid. The connection position is the reference point at the relevant end of the beam.

All connections sharing the same position on a column are considered as a single SidePlate Joint. No attempt is made to merge separate SidePlate joints even where the relevant levels are close.

Various checks are performed on each connection and each joint, and the analysis model is only be adjusted if they all pass. In particular, if the depths of two SidePlate joints overlap then a warning is generated and the analysis model is not be adjusted.

SidePlate validation

SidePlate validation is performed during model validation.

If warnings are issued, this will typically result in the Connection Status being set to "Not Applied" in the **Properties** window.

NOTE Until the “Not Applied” Connection Status is resolved, the connection will not be considered for the initial sizing process.

Initial sizing of the SidePlates

The initial SidePlate dimensions are determined during the Design Steel (Static & RSA) and Design All (Static & RSA) processes.

The initial sizing process is only performed where the Sizing Status is “Not Performed” and the Connection Status is **not** “Not Applied”.

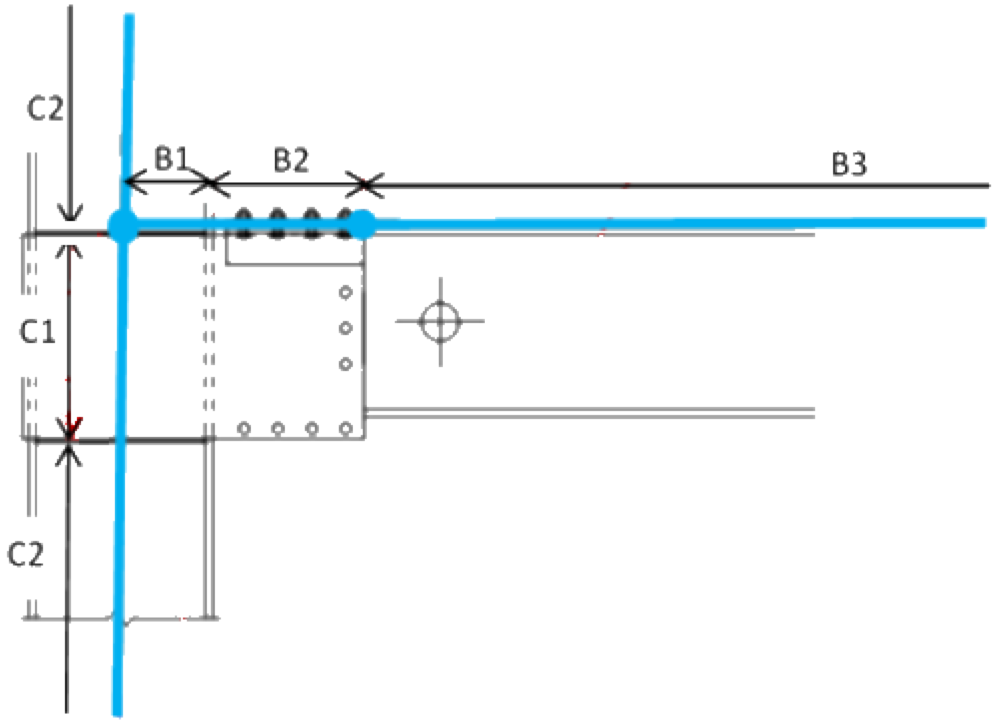
For each connection that is considered, a number of checks are performed in order to verify that SidePlate can be applied. If any of the checks fail, the Connection Status is set to "Not Applied".

NOTE After the initial sizing process, any changes to loading, (individual loads, load cases and combinations), will not trigger change control for the SidePlates, i.e. they will keep those initial sizes. You can reset the status using Review mode if you decide that the sizing process needs to be run again.

Analysis model adjustment

Analysis model adjustment is only performed for valid SidePlate connections that have successfully completed the initial sizing process, i.e. where the Sizing Status is **not** “Not Performed”.

Both the geometry and the properties of the Solver model are dependent upon the size of the SidePlate connection.



For the beam

- B2 – a new 1d element on the beam, with stiffened beam properties.
- B1 – a rigid offset on the end of B2
- B3 – normal bare beam properties

For the Column

- C1 – a rigid offset at the top of the column 1d element
- C2 – normal column properties

Design/Analysis processes and recommended workflows

Tekla Structural Designer can automatically size members throughout a structure – we call this Autodesign. The process is complex and involves sophisticated initial sizing, multiple analysis runs with repeated design cycles. The interweaving of initial sizing for SidePlate connections into the Autodesign routines is a highly complex process and for the first implementation remains beyond scope. Instead you have to manually reduce section sizes to take advantage of the SidePlate advantages.

Any beams and columns not affected by SidePlate connections can be AutoDesigned as normal during this process.

The following examples show the results for 2 different workflows where each step is carried out in the exact order specified and assuming that there are no model issues.

Recommended Workflow

- Full model created including loadcases and combinations - AutoDesign left checked for all columns, beams etc.
- Check “Apply SidePlate Connection” for selected beams - also set Utilization Ratio to 1.25 for those beams.
- Set Utilization Ratio to 1.1 for selected columns
- Run 1st order linear analysis
  - Validation will generate SidePlate AutoDesign warnings where relevant
  - SidePlate Initial Sizing will not be performed because not Static Design
  - Adjust Solver Model will not be performed because no Connections will have Sizing Status not “Not Performed”.
- Run Gravity Design Steel
  - Validation will generate SidePlate AutoDesign warnings where relevant
  - SidePlate Initial Sizing will not be performed because not Static Design
  - Adjust Solver Model will not be performed because no connections will have Sizing Status not “Not Performed”.
- Reset Utilization Ratio to 1.0 for relevant beams and columns
- Run Static Design Steel
  - No SidePlate related validation warnings
  - SidePlate Initial Sizing is performed for all connections
  - Adjust Solver Model will be performed for subsequent analysis
- Manually adjust section sizes to achieve further size savings with SidePlate connections
  - SidePlate Connection Status will be reset where necessary
- Run Static Design Steel
  - No SidePlate related validation warnings
  - SidePlate Initial Sizing is performed only where section sizes have been changed
  - Adjust Solver Model will be performed

At the end of this workflow, any subsequent stand-alone analysis will also use the adjusted solver model.

Other Workflow

- Full model created including loadcases and combinations - AutoDesign left checked for all columns, beams etc.
- Check “Apply SidePlate Connection” for selected beams.

- Run Static Design Steel
    - Validation will generate SidePlate AutoDesign warnings where relevant
    - SidePlate Initial Sizing will not be performed because Connection Status will be set to Warning
    - Adjust Solver Model will not be performed because no Connections will have Sizing Status not “Not Performed”.
- Run Static Design Steel
    - No SidePlate related validation warnings
    - SidePlate Initial Sizing is performed for all Connections
    - Adjust Solver Model will be performed for subsequent analysis

Effects of SidePlate on steel member design

Any beams and columns not affected by SidePlate connections can be AutoDesigned as normal.

Beams and columns affected by SidePlate connections cannot be AutoDesigned.

Where SidePlate connections exist, changes are required for both static and seismic member design.

For static design

- Beam and column buckling lengths are affected.
- Beam and column design checks are adjusted so that they are applied outside the physical length of any SidePlate connection

For seismic design

- OMF frames - no change
- IMF frames – various changes
- SMF frames – various changes

Create SidePlate connections

To add SidePlate connections:

1. Select the steel beam(s) to which you want to apply SidePlate connections.
2. In the **Properties** window, expand the SidePlate heading and select **Apply SidePlate connection** at one, or both ends as required.
3. In the **Properties** window, select the SidePlate **Connection type** that is appropriate to the SFRS type that has been specified, or, if not in a seismic frame, set the type as **SidePlate MF** (moment frame).

    Each connection is initially shown in the graphics as a simple box. After sizing has been performed it will be shown as a pair of plates.

See also

[Beam properties - SidePlate (page 654)]
[Modify SidePlates (page 754)]

Beam properties - SidePlate

For steel beams the **Properties** window has a SidePlate group within which an **Apply SidePlate Connection** check box can be selected for End 1 and End 2 of the beam.

When selected the following properties are displayed:

- Seismic Provision - depending on the head code, one of:
  - Non-Seismic (ACI/AISC only)
  - Seismic OMF
  - Seismic IMF (ACI/AISC only)
  - Seismic SMF
- Field Connection, (user option if Connection Type is Seismic SMF, otherwise fixed to Bolted)
  - Bolted (default)
  - Welded
- Datafile look up criteria (readonly)
  - Use %Mp (if Connection Type is Non-Seismic)
  - Use clear span (otherwise)
- Connection Status - one of
  - Unknown (default)
  - Pass
  - Warning
  - Not Applied
- Failure Reason - one of
  - Unknown (default)
  - Datafile Look Up Failed
  - Geometric Compatibility
  - Beam Element too short for Rigid Offset
  - Column Element too short for Rigid Offset
- Sizing Status - one of
  - Not Performed (default)

    - Initial
    - Final
- SidePlate size and other data set during the Initial Sizing Process, (Invisible if Sizing Status is Not Performed)
    - DimA – the outstand distance required for the SidePlate connection
    - DimB – the depth required for the SidePlate connection
    - Tsp – the thickness of the SidePlates
    - DimH – the length of the SidePlates
    - $\%M_p$- if calculated during initial sizing
    - Clear span - if calculated during initial sizing

NOTE These properties are for the start and end of the whole beam. Where a steel beam is multi-span it is not possible to apply SidePlate connections to the internal points.

Create and design other connections

NOTE These topics relate to the *design* of connections using either Tekla Connection Designer, or IDEA StatiCa; they do not cover checking of simple connection resistance (page 635), or checking of AISC/Eurocode base plates, which are separate topics.

Overview

Before connections can be designed in Tekla Structural Designer valid connection objects have to be created in the model. This is achieved by running **Update connections** from the Project Workspace. This applies a set of rules (page 658) to create and organize connections into the following types:

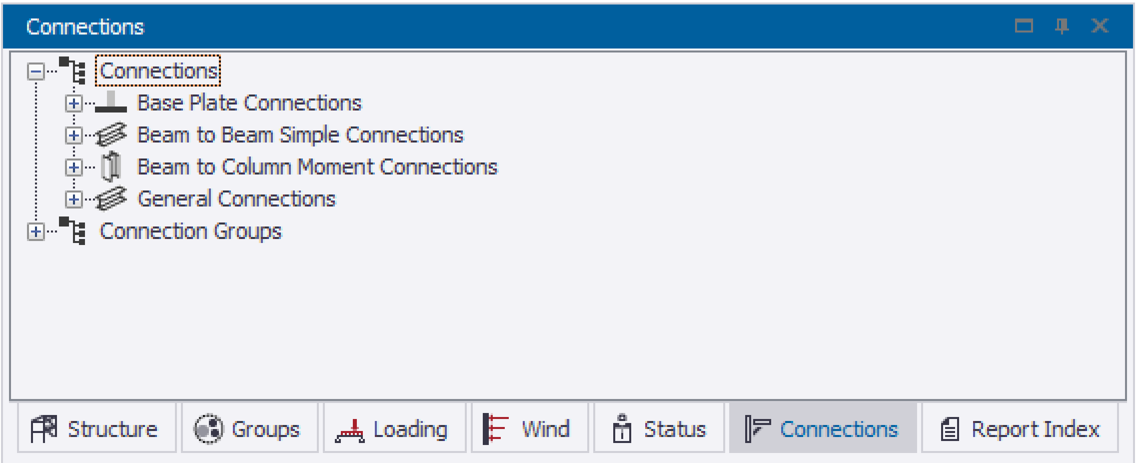
- Base plate connections - BS head code only

    NOTE Base plates to AISC or Eurocode head codes are handled differently, see:

- Beam to beam moment connections
- Beam to beam simple connections
- Beam to column moment connections
- Beam to column simple connections
- Column splice connections

- General connections

Valid connections are then displayed in the scene views and are also listed in the **Connections** tree.



Provided a license of Tekla Connection Designer is available, some of these can then be designed or exported to Tekla Connection Designer. If you have a license of IDEA StatiCa you can also export some types to that software for design.

Type	Connection Design Options
Base plate connections	• Design Connection (BS head code only) • Export to Tekla Connection Designer (BS head code only)
Beam to beam moment connections	• Design Connection (Eurocode and BS head codes only) • Export to Tekla Connection Designer (Eurocode and BS head codes only) • Export to IDEA StatiCa
Beam to beam simple connections	• Export to IDEA StatiCa
Beam to column moment connections	• Design Connection (Eurocode and BS head codes only) • Export to Tekla Connection Designer (Eurocode and BS head codes only) • Export to IDEA StatiCa
Beam to column simple connections	• Export to IDEA StatiCa

Type	Connection Design Options
Column splices	• Design Connection (Eurocode and BS head codes only) • Export to Tekla Connection Designer (Eurocode and BS head codes only) • Export to IDEA StatiCa
General Connections	• Export to IDEA StatiCa

See also

- Recommended workflows for specific connection types (page 658)
- Limitations when using Tekla Connection Designer with Tekla Structural Designer (page 660)

Update connections

To create connection objects for the first time, or to recreate connections after changes to the model:

1. Click on the **Connections** tree in the **Project Workspace**
2. Right-click the Connections branch.
3. Choose **Update Connections** from the context menu.

   This applies a set of rules (page 658) to determine all valid steel connections in the model. The resulting connections are listed in the Connections Tree and are also shown by bounding boxes in the Scene Views.

   NOTE If the model has been changed so that new connections exist, these are not created automatically; you are required to update connections as required.

Design connections

Design using Tekla Connection Designer is initiated in one of two ways:

1. From the Project Workspace:
   - Expand the appropriate branch in the **Connections** tree
   - Right click on the required connection reference and choose **Design connection** from the context menu.
2. From a scene view:
   - Right click on the connection object and choose **Design connection** from the context menu.

See also

Recommended workflows for specific connection types (page 658)

Limitations when using Tekla Connection Designer with Tekla Structural Designer (page 660)

Export connections to another application for design (page 661)

Steel connection formation rules

The following rules relate to connection formation in Tekla Structural Designer.

1. beam to column connection
    a. beam connections to the column base are ignored
    b. column must be a symmetric section
    c. more then 1 beam attached to the same column face prevents any connection
    d. a continuous beam connected to the column flanges prevents any connection
    e. a beam with a fully fixed or moment end condition connected to the column web prevents any connection
2. beam to beam - for a particular construction point (node)
    a. any column at the node prevents the connection
    b. only two beam ends can be connected, a 3rd beam at the same node prevents any connection.
    c. connections cannot be formed at beam internal nodes
    d. the beams being connected must be symmetric and of the same section size
    e. connected beam ends cannot be free or pinned
    f. top flange haunches prevent the connection
    g. beam ends being connected must have no end plates specified, or both have the same end plate
    h. beam ends being connected must have no bottom haunch specified, or both have the same bottom haunch
3. any non-steel member at the node prevents any connection
4. braces are ignored in the procedure

Recommended workflows for specific connection types

NOTE Connection design requires a Tekla Connection Designer licence.

Portal frame connections

Although it is possible to design portal frame connections in Tekla Structural Designer - i.e. Base Plate, Eaves, and Apex moment connections; it should be noted that the design forces used for connection design within Tekla Structural Designer are those from its own elastic analysis and may not be appropriate.

A more correct approach would be to derive the connection forces from a Tekla Portal Frame Designer plastic analysis. To do this you would need to export the portal frame to Tekla Portal Frame Designer and then design both the frame and its connections within Tekla Portal Frame Designer.

Note that it in addition to the connection forces (usually) being fundamentally different between Tekla Portal Frame Designer and Tekla Structural Designer there are other potential differences also:

- for eaves connections - in Tekla Portal Frame Designer the connection design forces are taken at the column face (rather than the rafter/column intersection),
- for column bases - in Tekla Portal Frame Designer they can be pinned at ULS but % fixity can be taken for SLS, while Tekla Structural Designer will always give an analysis moment for % fixity regardless of ULS/SLS and this moment would be designed for.

NOTE Connection design via Tekla Portal Frame Designer requires both a Tekla Portal Frame Designer and a Tekla Connection Designer licence.

Base plate connections (not in portal frames)

Base plate connections can be edited directly in Tekla Structural Designer.

Provided they are in elastically analyzed frames they can be also be designed using the design forces obtained from Tekla Structural Designer.

NOTE The analysis results used for the connection design follow the selection (1st order or 2nd order) made on the Analysis page of the Tekla Structural Designer Design Options.

Base plate connections can be exported to Tekla Connection Designer if you require, in which case if you edit the connection geometry while in Tekla Connection Designer you should use the “Return Connection to Tekla Structural Designer” command (under the “Connection” menu) to return the edited connection data back to Tekla Structural Designer.

Other connections (not in portal frames)

The recommended workflow for connection types other than Base Plate Connections is:

- Select the connection(s) to be designed and export to Tekla Connection Designer always.
- Design in Tekla Connection Designer and print the reports.
- While the “Return Connection to Tekla Structural Designer” command can be used, this does not currently return all connection data to Tekla Structural Designer (principally the data derived from the connected elements). Hence, if changes are made in Tekla Structural Designer and the analysis and/ or design processes run and the connection exported again for checking, the engineer should review the connection data and may need to re-enter some of the settings previously made.

NOTE While the “Return Connection to Tekla Structural Designer” command can be used In Tekla Connection Designer, this does not currently return all connection data to Tekla Structural Designer (principally the data derived from the connected elements). Hence, if changes are made in Tekla Structural Designer and the analysis and/or design processes are re-run; if the connection is exported again for checking, the engineer would need to review the connection data as it may be necessary to re-enter some of the settings previously made.

See also

Limitations when using Tekla Connection Designer with Tekla Structural Designer (page 660)

Limitations when using Tekla Connection Designer with Tekla Structural Designer

General

- The analysis results used for the connection design follow the selection (1st order or 2nd order) made on the Analysis page of the Tekla Structural Designer Design Options.
- It is always necessary to export all connection types to Tekla Connection Designer for reporting purposes.

Base plate connections

- Base Plate Connections only deal with major axis shear - only one value for shear is available in the Combinations grid in Tekla Connection Designer. (This is a limitation in the Green Book also.)

Moment connections

- While the “Return Connection to TSD” command can be used, this does not currently return all connection data to Tekla Structural Designer (principally the data derived from the connected elements). Hence, if changes are made in Tekla Structural Designer and the analysis and/ or design processes run and the connection exported again for checking, the engineer should review the connection data and may need to re-enter some of the settings previously made.
- For Beam to Column moment connections, column forces are exported and displayed on the "Column" tab of the Combinations page.
  - When designed to Eurocodes, there is a reduction factor for column web in compression to allow for coincident forces in the column itself i.e. moment and axial force. This is called kwc and in most cases is 1.0 i.e. no reduction. But where the stress due to the column forces is high (> 0.7 fy) then kwc is < 1.0. See Clause 6.2.6.2 (2).
  - These forces are NOT populated by the TCD integration link and must be entered manually by the engineer.

Simple connections

- It is not currently possible to transfer Simple Connections from Tekla Structural Designer to Tekla Connection Designer.

Hollow section connections

- It is not currently possible to transfer Simple Connections from Tekla Structural Designer to Tekla Connection Designer.

Column splice connections

- Splice connections with significant ‘real’ moments (other than eccentricity moments) cannot currently be transferred from Tekla Structural Designer to Tekla Connection Designer.

NOTE Significent 'real' moments being greater than the Ignore forces below (page 1097) limit in Design Settings.

Export connections to another application for design

Click the following links to find out more about exporting specific connections into another application for design:

- Recommended workflow for TCD connection types (page 658)
- Export to Tekla Connection Designer (page 127)
- Export to Tekla Portal Frame Designer (page 128)

- Export to IDEA StatiCa Connection Design (page 152)

7.6 Drift, sway, seismic drift, wind drift, and overall displacements

As part of the static design process the model is automatically checked on a level by level basis for drift (page 662) (ACI/AISC), or sway (page 667) (other head codes). Also any seismic load combinations are checked for seismic drift (page 671) and wind load combinations checked for wind drift (page 675)/ overall wind drift (page 679).

Click the links below to find out more:

- Drift check (page 662)
- Sway check (page 667)
- Seismic drift check (page 671)
- Wind drift check (page 675)
- Overall wind drift check (page 679)
- Overall displacement (page 681)

Drift check

By default, this check is applied to all columns (of all materials) and all walls at every level in the model.

NOTE The drift check is only relevant to models that use the ACI/AISC head code.

Results are presented in terms of levels, with detailed results for individual column/wall stacks/panels also being available.

For levels set as floors a **Check for drift** option is displayed in Level Properties (page 933). By unselecting this option, individual levels can be excluded from the check reporting.

In this check the story drift in each direction (i.e. the difference between top and bottom deflection of the stack) is calculated, and from this the stability coefficient is determined as follows:

Stability coefficient = Δ_2/Δ_1

Where

Δ_2= second order drift

Δ_1= first order drift

A Warning status is displayed against any stack in which the ratio exceeds 1.71 and a Beyond Scope status is applied if it exceeds 2.85 (as this indicates the structure lacks stiffness in whole or in part).

NOTE Stability coefficients can only be calculated accurately by performing a second order analysis, however indicative values can still be displayed when only a first order analysis has been performed. These are determined from the column's elastic critical load factor, λ:

$(\Delta_2/\Delta_1) = 1/(1- 1/\lambda)$

Approximate second order drift:

$\Delta_2 = \Delta_1/(1- 1/\lambda)$

A workflow for running the check is outlined below:

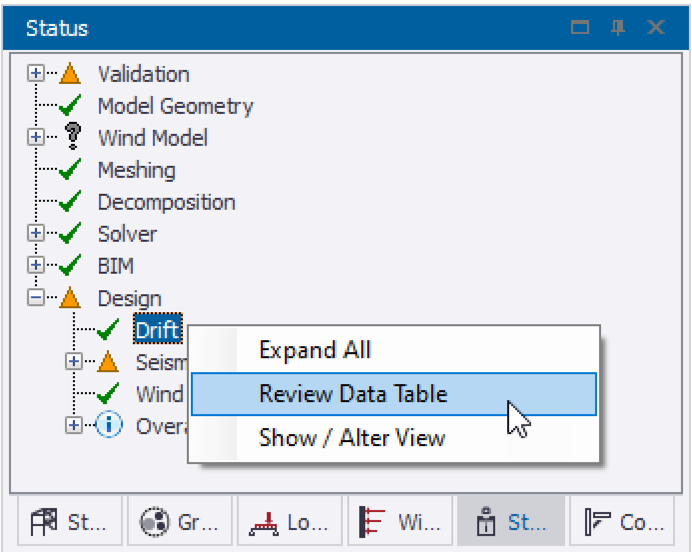
Run the check

Load combinations are automatically checked on a level by level basis for drift by running a static analysis or design.

Review the check status and details

Once the check has been run, an overall status is reported in the **Design** branch of the Status Tree (page 82), the overall status being the worst status from each of the individual levels.

Right clicking on the status opens a context menu.

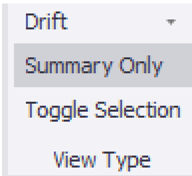


From this menu you can display the drift check summary in a Review Data Table (page 770).

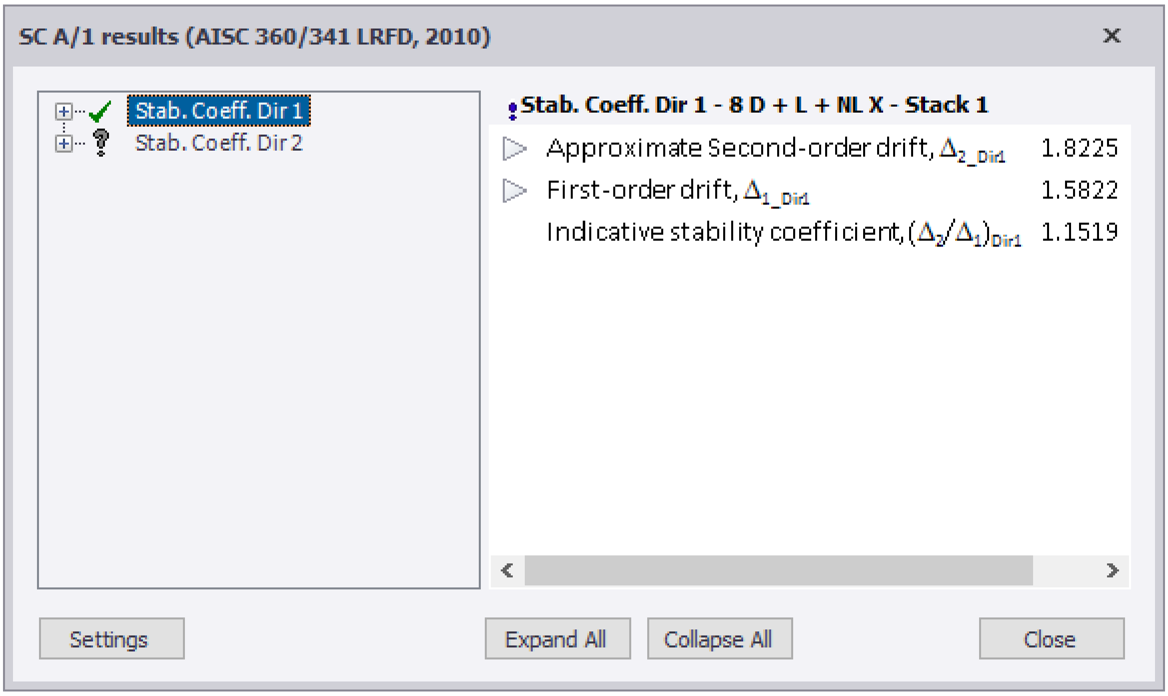
Review Data

Drift												
Level	Ref.	Stack	Combination Dir 1	Drift 1st order Dir 1 [mm]	Drift 2nd order Dir 1 [mm]	Ratio Dir 1	Combination Dir 2	Drift 1st order Dir 2 [mm]	Drift 2nd order Dir 2 [mm]	Ratio Dir 2	Status	Details
St. 2 (2)	SC A/1	2	8 D + L + NL X	0.7615	0.8131	1.0678	-	-	-	-	Pass	Details...
St. 1 (1)	SC A/4	1	8 D + L + NL X	1.5822	1.8225	1.1519	-	-	-	-	Pass	Details...

By unselecting the **Summary Only** option in the ribbon, the table can also display detailed results on a stack by stack basis if required.



The checks can be investigated in more detail by clicking the **Details** button at the end of any row.



This displays the stability coefficients in both directions (RatioDir1,RatioDir2). These are calculated as follows:

Stability coefficient = Δ_2/Δ_1

Where

Δ_2= second order drift

Δ_1= first order drift

A Warning status is displayed against any stack in which the ratio exceeds 1.71 and a Beyond Scope status is applied if it exceeds 2.85 (as this indicates the structure lacks stiffness in whole or in part).

Stability coefficients can only be calculated accurately by performing a second order analysis, however indicative values can still be displayed when only a first order analysis has been performed. These are determined from the column's elastic critical load factor, λ:

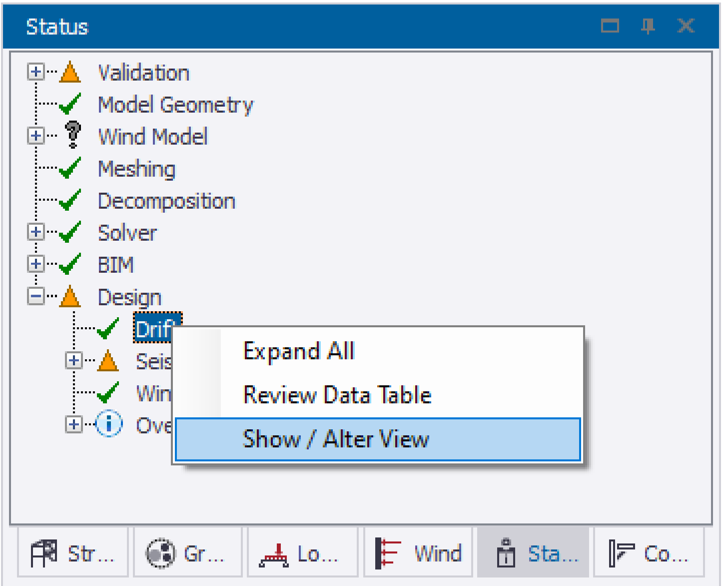
$(\Delta_2/\Delta_1) = 1/(1- 1/\lambda)$

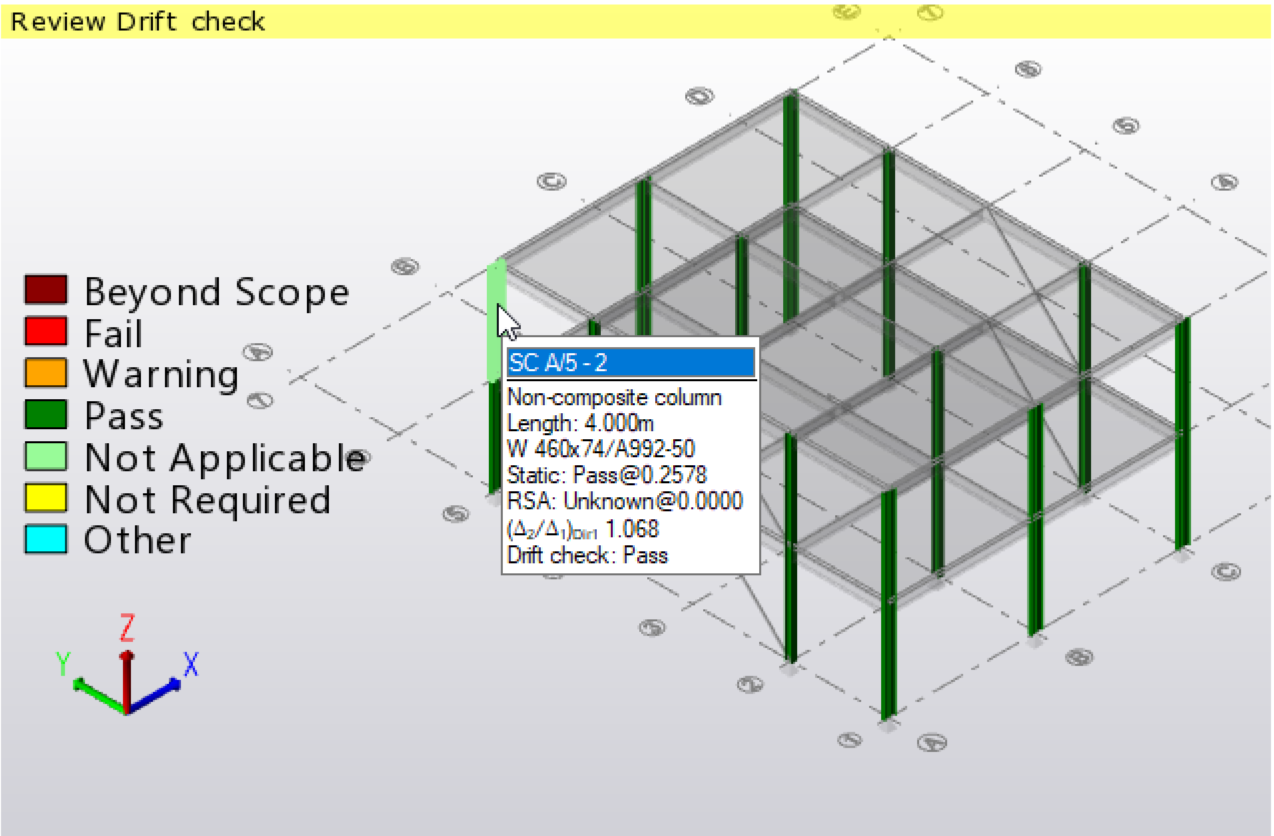
Approximate second order drift:

$\Delta_2= \Delta_1/(1- 1/\lambda)$

Review the check graphically

From the right click menu in the Status Tree you are also provided with an option to graphically review the checks in a Show/Alter State View (page 736).





The color coded legend makes it easy to see individual failing members. By hovering the cursor over a member, its drift results are displayed in the tooltip.

From the same Show/Alter State view, by changing the Mode in the **Properties** window, you also have the option to switch off checks for specific stacks (page 736) if you require.

Switching off inappropriate checks and merging stack lengths

After reviewing you might choose to switch off checks for selected columns and walls, or individual stacks/panels within selected columns and walls: this can be done directly via the (page 684), or graphically in a Show/Alter State view (as described in the previous section).

You might also choose to take advantage of the option to merge short stacks automatically (page 681) where they are less than a specified limit. Stacks check lengths can also be manually adjusted (page 685) by merging them together if required.

An option is also provided which allows you to switch off checks for specific levels (page 685) if you decide they are inappropriate, (for example if the story height is very short). The checks at that level are still performed, but they are excluded from the tabular review data display.

Printing calculations

The drift check results table is available either by creating a Building Analysis & Drift Checks report (page 841), or by including the Analysis>Drift chapter in a custom report.

Sway check

By default, this check is applied to all columns (of all materials) and all walls at every level in the model.

NOTE The sway check is not relevant to models that use the ACI/AISC head code.

Results are presented in terms of levels, with detailed results for individual column/wall stacks/panels also being available.

For levels set as floors a **Check for drift** option is displayed in Level Properties (page 933). By unselecting this option, individual levels can be excluded from the check reporting.

In this check the story sway in each direction (i.e. the difference between top and bottom deflection of the stack) is calculated and from this the elastic critical load factor is determined.

A workflow for running the check is outlined below:

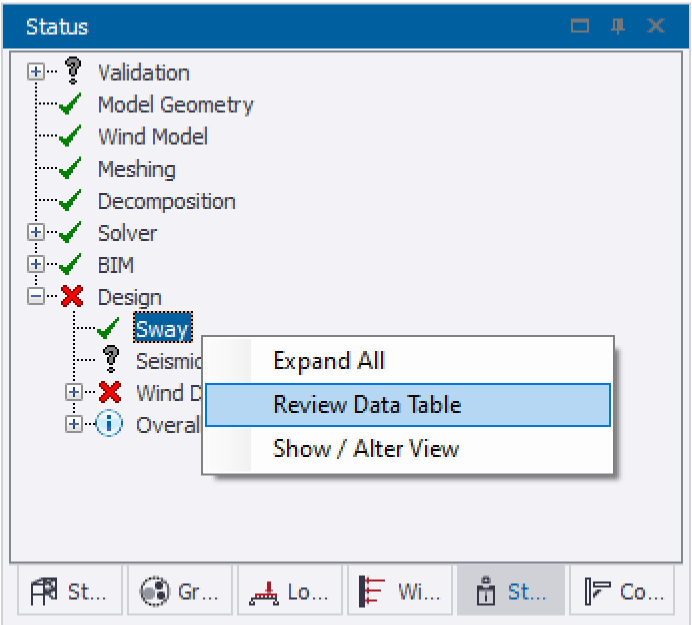
Run the check

Load combinations are automatically checked on a level by level basis for sway by running a static analysis or design.

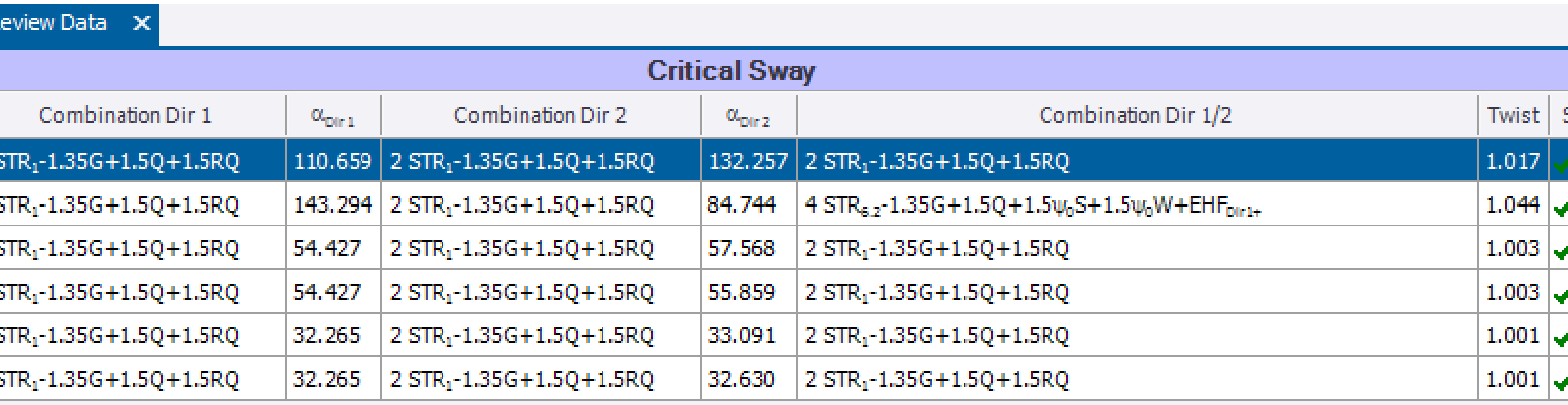
Review the check status and details

Once the check has been run, an overall status is reported in the **Design** branch of the Status Tree (page 82), the overall status being the worst status from each of the individual levels.

Right clicking on the status opens a context menu.



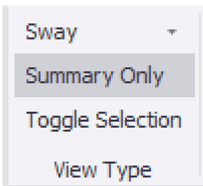
From this menu you can display the sway check summary in a Review Data Table (page 765).



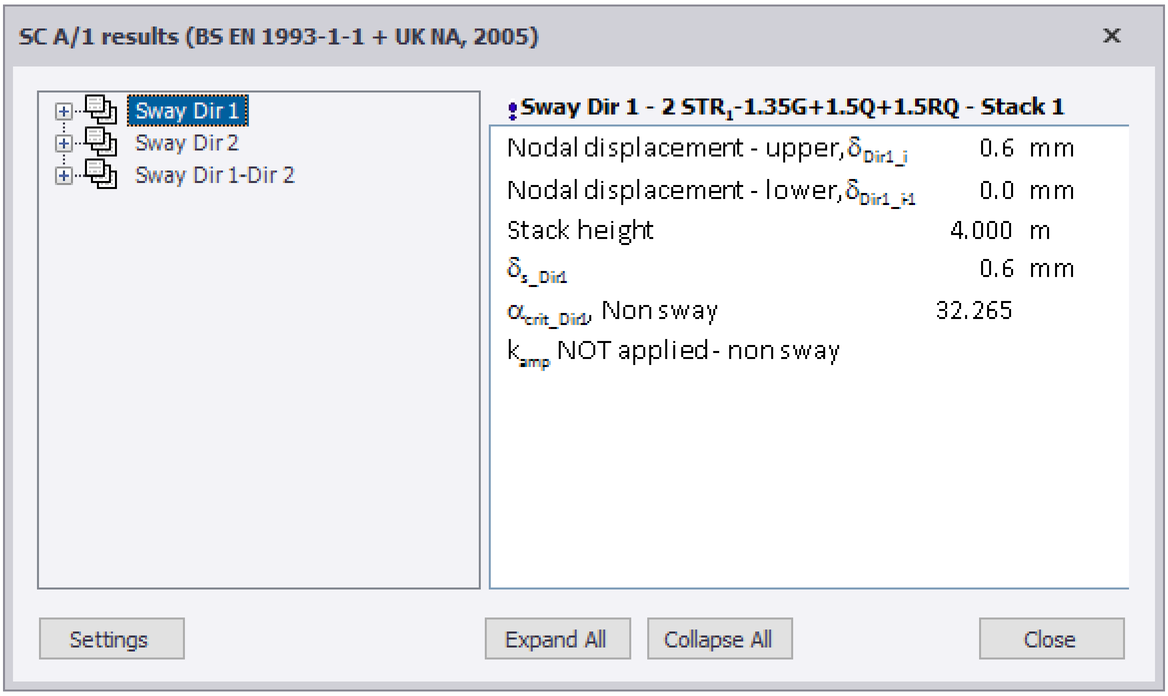
eview Data

Critical Sway					
Combination Dir 1	$\alpha_{Dir 1}$	Combination Dir 2	$\alpha_{Dir 2}$	Combination Dir 1/2	Twist
STR$_1$-1.35G+1.5Q+1.5RQ	110.659	2 STR$_1$-1.35G+1.5Q+1.5RQ	132.257	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.017
STR$_1$-1.35G+1.5Q+1.5RQ	143.294	2 STR$_1$-1.35G+1.5Q+1.5RQ	84.744	4 STR$_{6.2}$-1.35G+1.5Q+1.5ψ_0S+1.5ψ_0W+EHF$_{Dir1+}$	1.044
STR$_1$-1.35G+1.5Q+1.5RQ	54.427	2 STR$_1$-1.35G+1.5Q+1.5RQ	57.568	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.003
STR$_1$-1.35G+1.5Q+1.5RQ	54.427	2 STR$_1$-1.35G+1.5Q+1.5RQ	55.859	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.003
STR$_1$-1.35G+1.5Q+1.5RQ	32.265	2 STR$_1$-1.35G+1.5Q+1.5RQ	33.091	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.001
STR$_1$-1.35G+1.5Q+1.5RQ	32.265	2 STR$_1$-1.35G+1.5Q+1.5RQ	32.630	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.001

By unselecting the **Summary Only** option in the ribbon, the table can also display detailed results on a stack by stack basis if required.

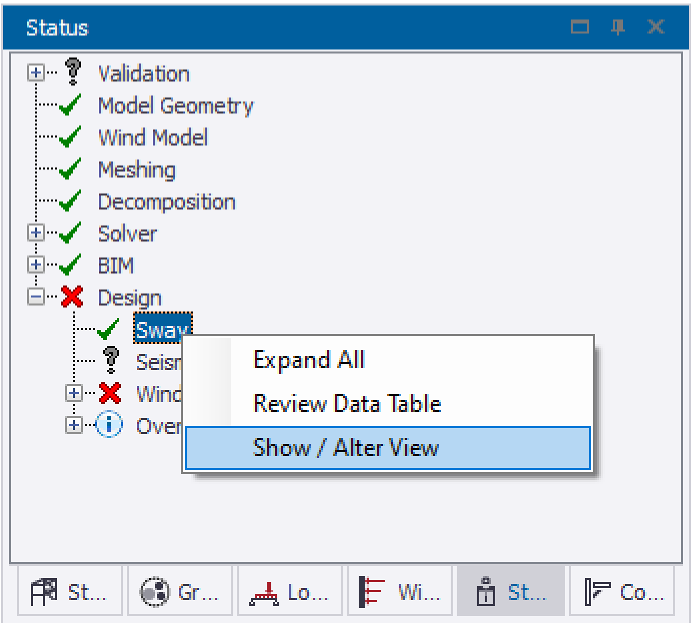


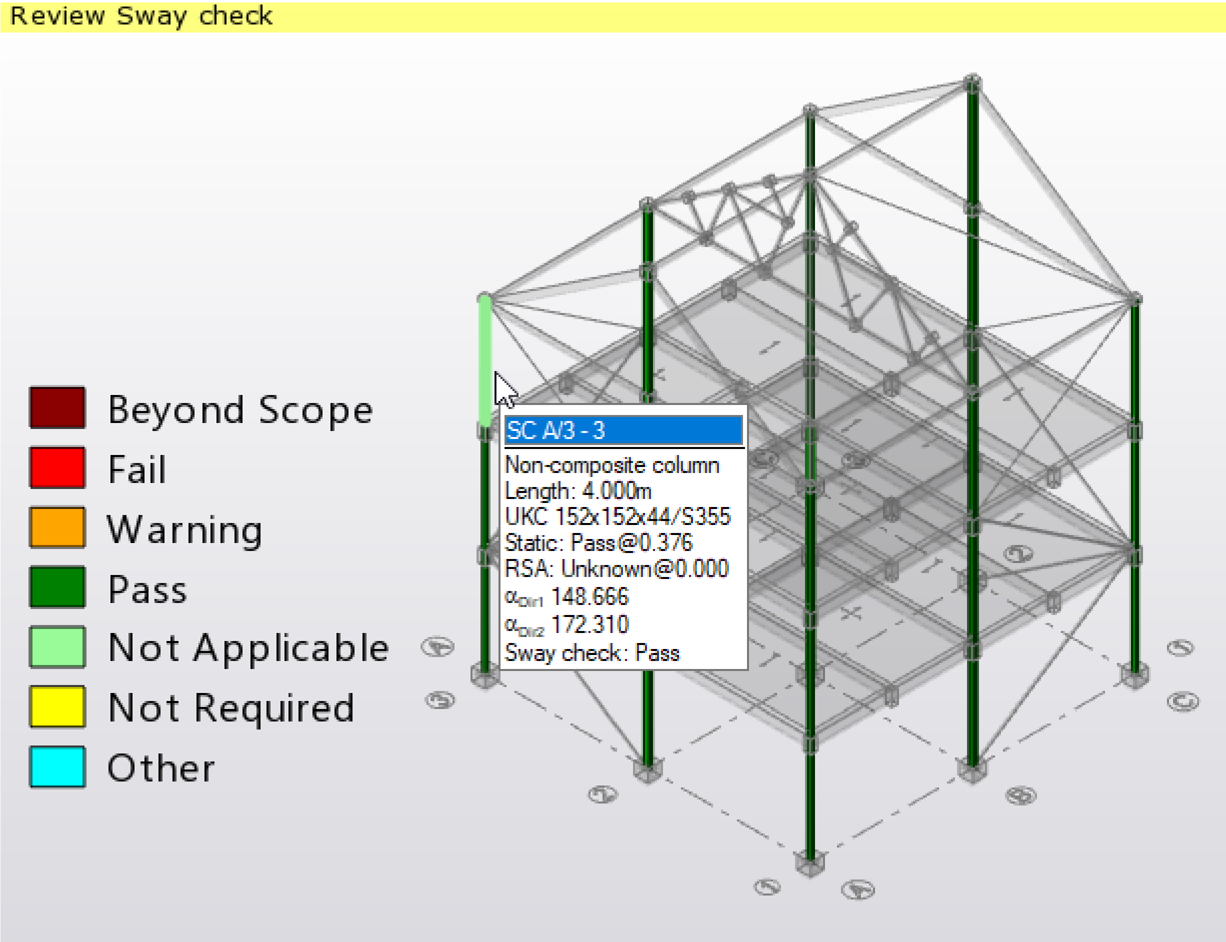
The individual checks can be investigated in more detail by clicking the **Details** button at the end of any row.



Review the check graphically

From the right click menu in the Status Tree you are also provided with an option to graphically review the checks in a Show/Alter State View (page 756).





The color coded legend makes it easy to see any individual failing members. By hovering the cursor over a member, its sway results are displayed in the tooltip.

From the same Show/Alter State view, by changing the Mode in the **Properties** window, you also have the option to switch off checks for specific stacks (page 756) if you require.

If required, sway values can also be displayed graphically (page 503) in a Results View.

Switching off inappropriate checks and merging stack lengths

After reviewing you might choose to switch off checks for selected columns and walls, or individual stacks/panels within selected columns and walls: this can be done directly via the (page 683), or graphically in a Show/Alter State view (as described in the previous section).

You might also choose to take advantage of the option to merge short stacks automatically (page 681) where they are less than a specified limit. Stacks check lengths can also be be manually adjusted (page 685) by merging them together if required.

An option is also provided which allows you to switch off checks for specific levels (page 685) if you decide they are inappropriate, (for example if the story height is very short). The checks at that level are still performed, but they are excluded from the tabular review data display.

Printing calculations

The sway check results table is available either by creating a Building Analysis & Drift Checks report (page 841), or by including the Analysis>Sway chapter in a custom report.

Seismic drift check

By default this check is applied to all columns (of all materials), and all walls at every level in the model.

Results are presented in terms of levels, with detailed results for individual column/wall stacks/panels also being available.

For levels set as floors a **Check for drift** option is displayed in Level Properties (page 933). By unselecting this option, individual levels can be excluded from the check reporting.

In this check the seismic stability coefficient θ in each direction is checked against the limit θ_{max} and then the stack design story drift Δ_{des} is determined in each direction and checked against the allowable story drift Δ_a

A workflow for running the check is outlined below:

Configure the check and set the limit

The check is configured on the Localization page of the **Seismic Wizard**.

An option is provided to override the allowable story drift factor and a second option allows for the seismic drift check to be skipped entirely if required.

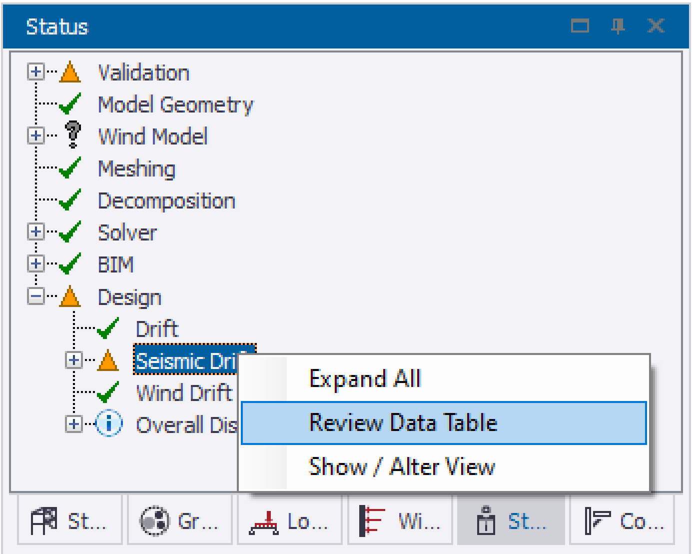
Run the check

Seismic load combinations are automatically checked on a level by level basis for drift by running an appropriate analysis or design.

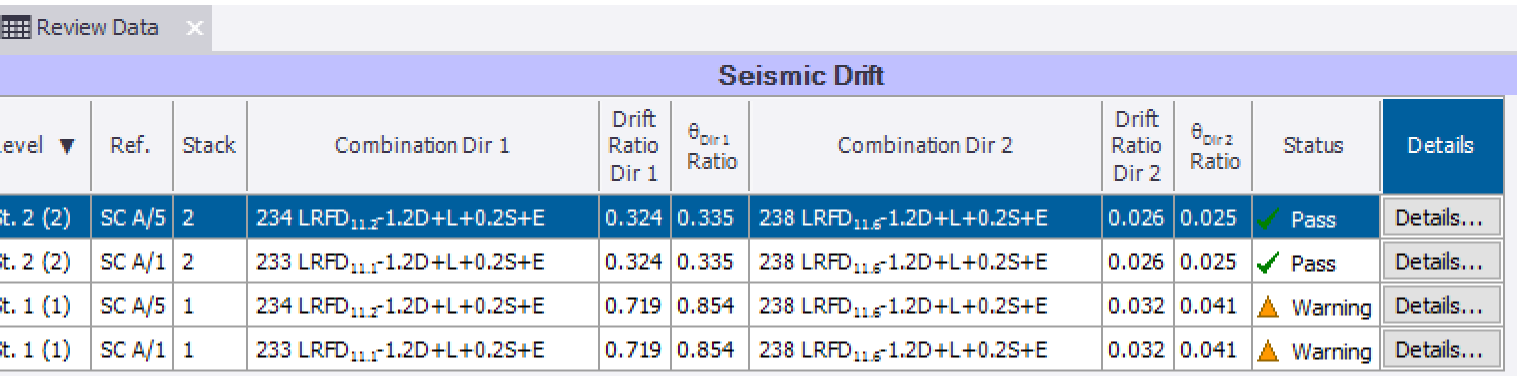
Review the check status and details

Once the check has been run, an overall status is reported in the **Design** branch of the Status Tree (page 82), the overall status being the worst status from each of the individual levels.

Right clicking on the status opens a context menu.



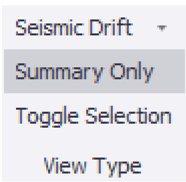
From this menu you can display the seismic drift check summary in a Review Data Table (page 774), an ASCE example of this table is shown below.



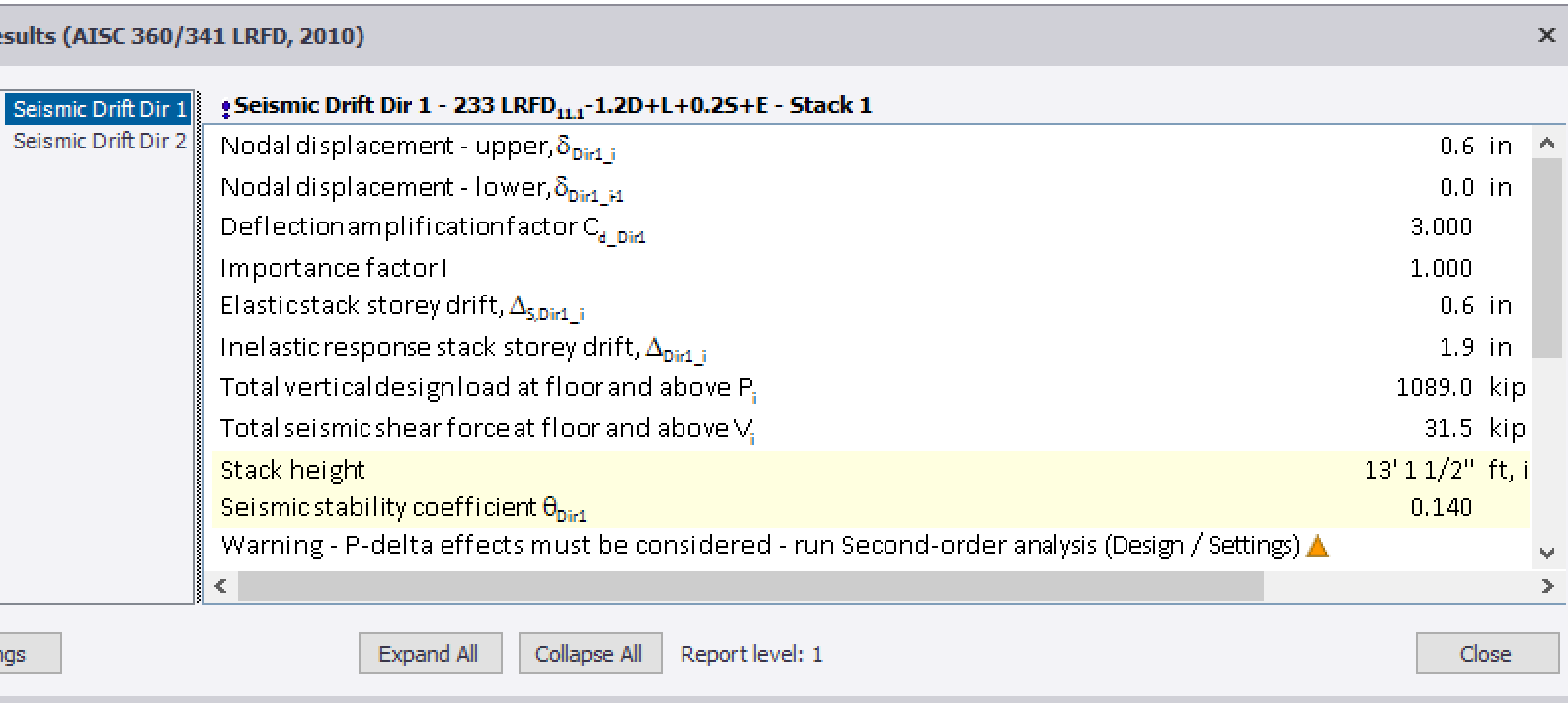
Review Data

Seismic Drift											
evel	Ref.	Stack	Combination Dir 1	Drift Ratio Dir 1	$\theta_{Dir 1}$ Ratio	Combination Dir 2	Drift Ratio Dir 2	$\theta_{Dir 2}$ Ratio	Status	Details
t. 2 (2)	SC A/5	2	234 LRFD$_{11.2}$-1.2D+L+0.2S+E	0.324	0.335	238 LRFD$_{11.6}$-1.2D+L+0.2S+E	0.026	0.025	Pass	Details...
t. 2 (2)	SC A/1	2	233 LRFD$_{11.1}$-1.2D+L+0.2S+E	0.324	0.335	238 LRFD$_{11.6}$-1.2D+L+0.2S+E	0.026	0.025	Pass	Details...
t. 1 (1)	SC A/5	1	234 LRFD$_{11.2}$-1.2D+L+0.2S+E	0.719	0.854	238 LRFD$_{11.6}$-1.2D+L+0.2S+E	0.032	0.041	Warning	Details...
t. 1 (1)	SC A/1	1	233 LRFD$_{11.1}$-1.2D+L+0.2S+E	0.719	0.854	238 LRFD$_{11.6}$-1.2D+L+0.2S+E	0.032	0.041	Warning	Details...

By unselecting the **Summary Only** option in the ribbon, the table can also display detailed results on a stack by stack basis if required.

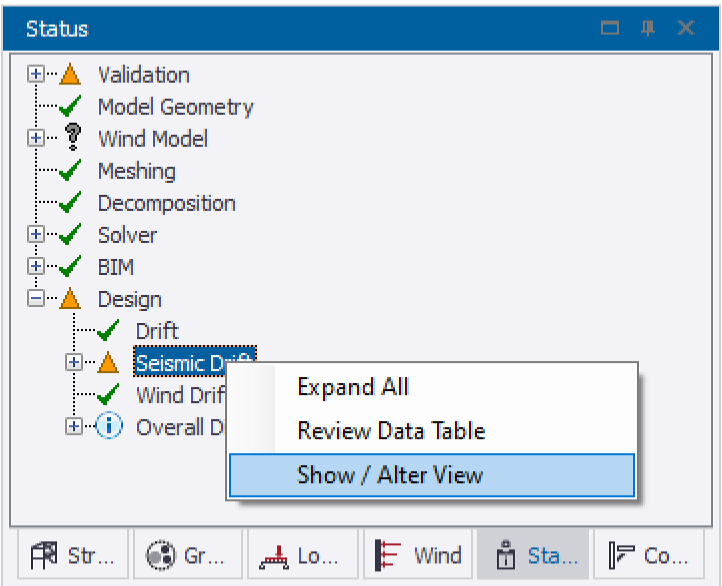


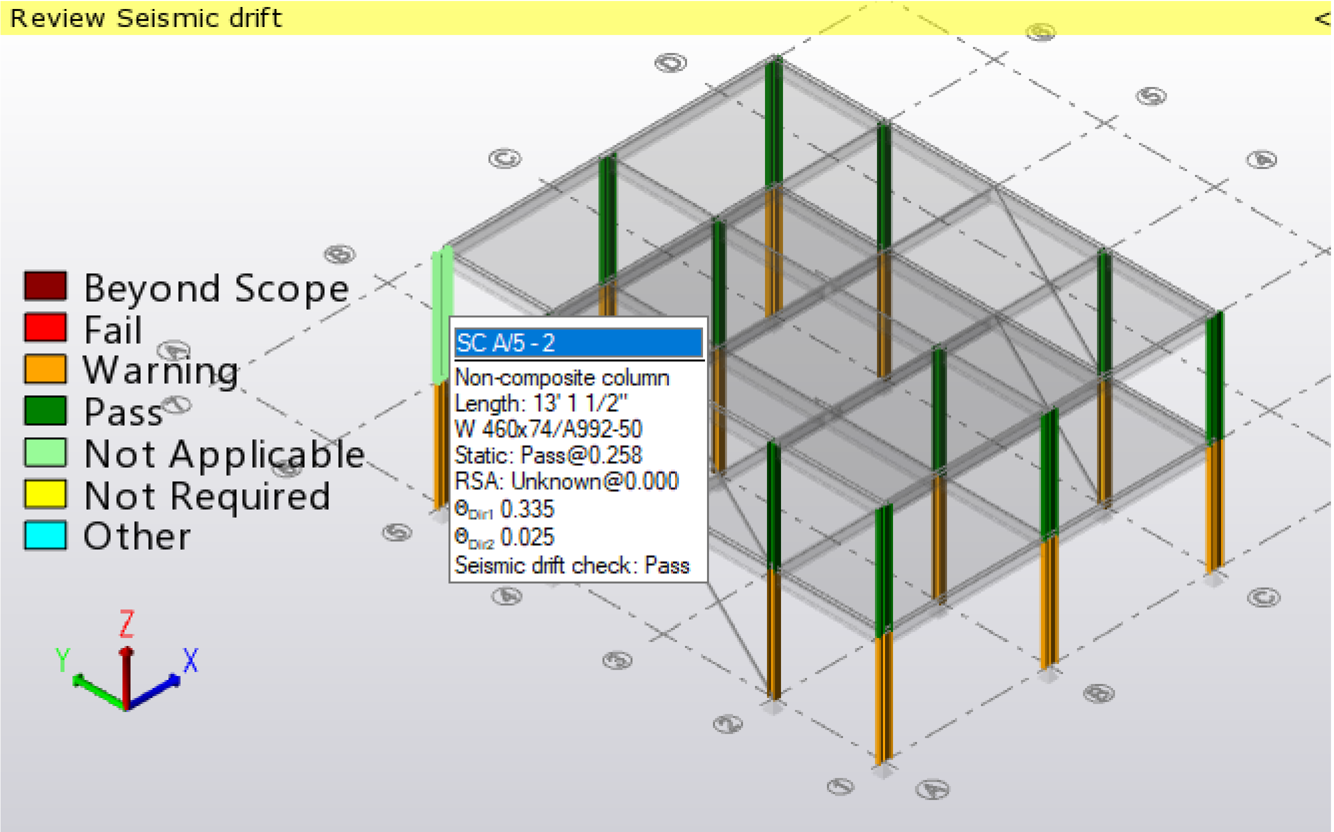
The individual checks can be investigated in more detail by clicking the **Details** button at the end of any row.



Review the check graphically

From the right click menu in the Status Tree you are also provided with an option to graphically review the checks in a Show/Alter State View (page 752).





The color coded legend makes it easy to see individual failing members. By hovering the cursor over a member, its seismic drift results are displayed in the tooltip.

From the same Show/Alter State view, by changing the Mode in the **Properties** window, you also have the option to switch off checks for specific stacks (page 753) if you require.

Switching off inappropriate checks and merging stack lengths

After reviewing you might choose to switch off checks for selected columns and walls, or individual stacks/panels within selected columns and walls: this can be done directly via the (page 684), or graphically in a Show/Alter State view (as described in the previous section).

You might also choose to take advantage of the option to merge short stacks automatically (page 681) where they are less than a specified limit. Stacks check lengths can also be be manually adjusted (page 685) by merging them together if required.

An option is also provided which allows you to switch off checks for specific levels (page 685) if you decide they are inappropriate, (for example if the story height is very short). The checks at that level are still performed, but they are excluded from the tabular review data display.

Printing calculations

The seismic drift check results table is available either by creating a Seismic Design report (page 847), or by including the Analysis>Seismic Drift chapter in a custom report.

Wind drift check

By default this check is applied to all columns (of all materials), and all walls at every level in the model.

Results are presented in terms of levels, with detailed results for individual column/wall stacks/panels also being available.

For levels set as floors a **Check for drift** option is displayed in Level Properties (page 933). By unselecting this option, individual levels can be excluded from the check reporting.

In this check the lateral drift resulting from the analysis is determined for each wind load combination using the combination SLS (Service Level) factors - (which can be < 1.0). This is then compared against a user-defined limit.

A workflow for running the check is outlined below:

Configure the check and set the limit

The check is configured from **Design Settings**.

On the **Drift Checks** page (if working to the ACI/AISC head code) or **Sway & Drift Checks** page (if working to any other head code) you can enter the limit required.

NOTE By default the limit is set as 1/300 of the story height, in line with Eurocode 3 recommendations.

Individual limits can also be applied as overrides to selected stacks (page 685) if required.

By default the check is performed in the resultant wind drift direction only, but by unselecting **Check for Resultant Wind Drift** it can be performed in each of Building Directions 1 & 2 if required.

By default the check only considers the effects of the wind loadcase(s) in wind combinations, but by unselecting **Check wind cases only** it will consider the effects of all loadcases in wind combinations (which would include drift induced by gravity loads).

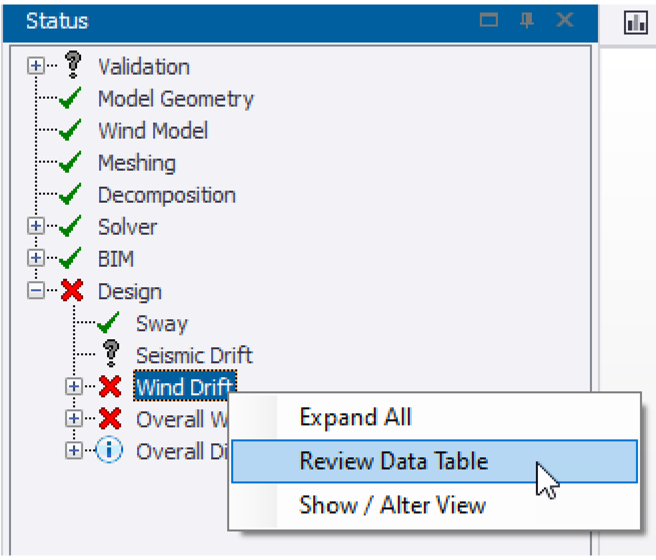
Run the check

Wind load combinations are automatically checked on a level by level basis for wind drift during the structure static design, and also when any 3D analysis is run in isolation. If a sub-set of combinations are considered for analysis then only those combinations are considered in the wind drift checks, (allowing engineers working on larger structures to investigate and optimize the lateral load resisting systems more rapidly).

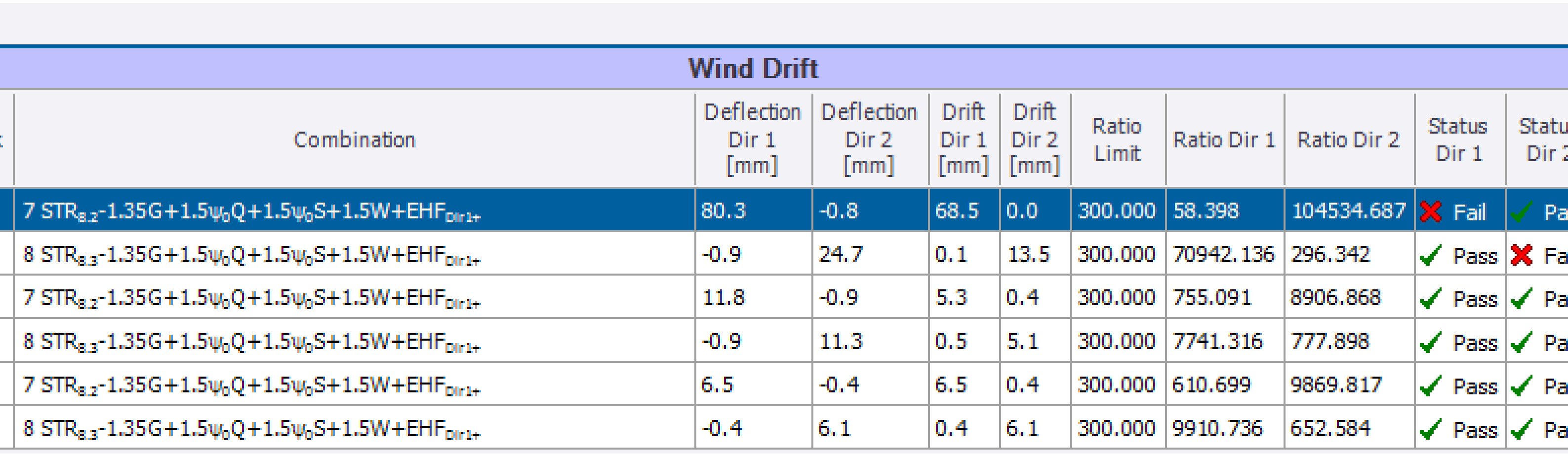
Review the check status and details

Once the check has been run, an overall status is reported in the **Design** branch of the Status Tree (page 82), the overall status being the worst status from each of the individual levels.

Right clicking on the status opens a context menu.



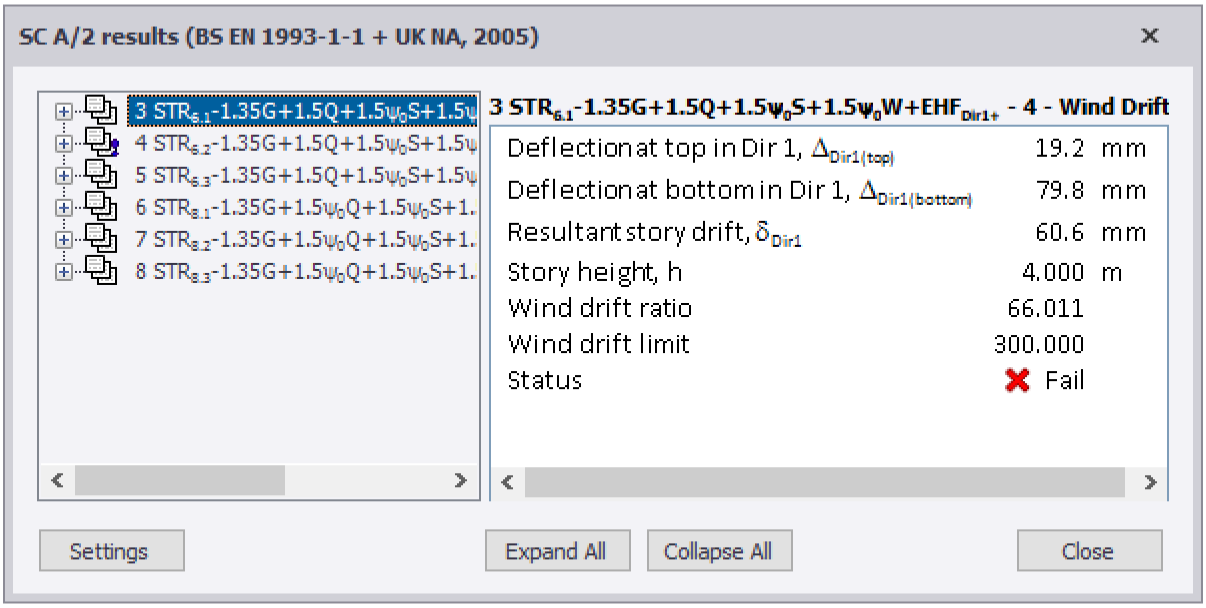
From this menu you can display the wind drift check summary in a Review Data Table (page 778), an example of this table for a Eurocode model is shown below.



Wind Drift									
Combination	Deflection Dir 1 [mm]	Deflection Dir 2 [mm]	Drift Dir 1 [mm]	Drift Dir 2 [mm]	Ratio Limit	Ratio Dir 1	Ratio Dir 2	Status Dir 1	Statu Dir 2
7 $STR_{8.2}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1+}	80.3	-0.8	68.5	0.0	300.000	58.398	104534.687	Fail	Pa
8 $STR_{8.3}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1+}	-0.9	24.7	0.1	13.5	300.000	70942.136	296.342	Pass	Fa
7 $STR_{8.2}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1+}	11.8	-0.9	5.3	0.4	300.000	755.091	8906.868	Pass	Pa
8 $STR_{8.3}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1+}	-0.9	11.3	0.5	5.1	300.000	7741.316	777.898	Pass	Pa
7 $STR_{8.2}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1+}	6.5	-0.4	6.5	0.4	300.000	610.699	9869.817	Pass	Pa
8 $STR_{8.3}$-1.35G+1.5ψ_0Q+1.5ψ_0S+1.5W+EHF_{Dir1+}	-0.4	6.1	0.4	6.1	300.000	9910.736	652.584	Pass	Pa

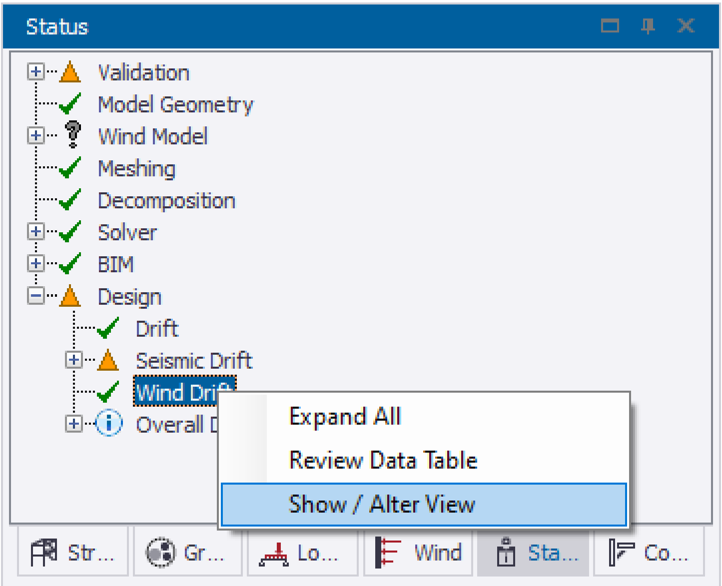
By unselecting the above **Summary Only** option in the ribbon, the table can also display detailed results on a stack by stack basis if required.

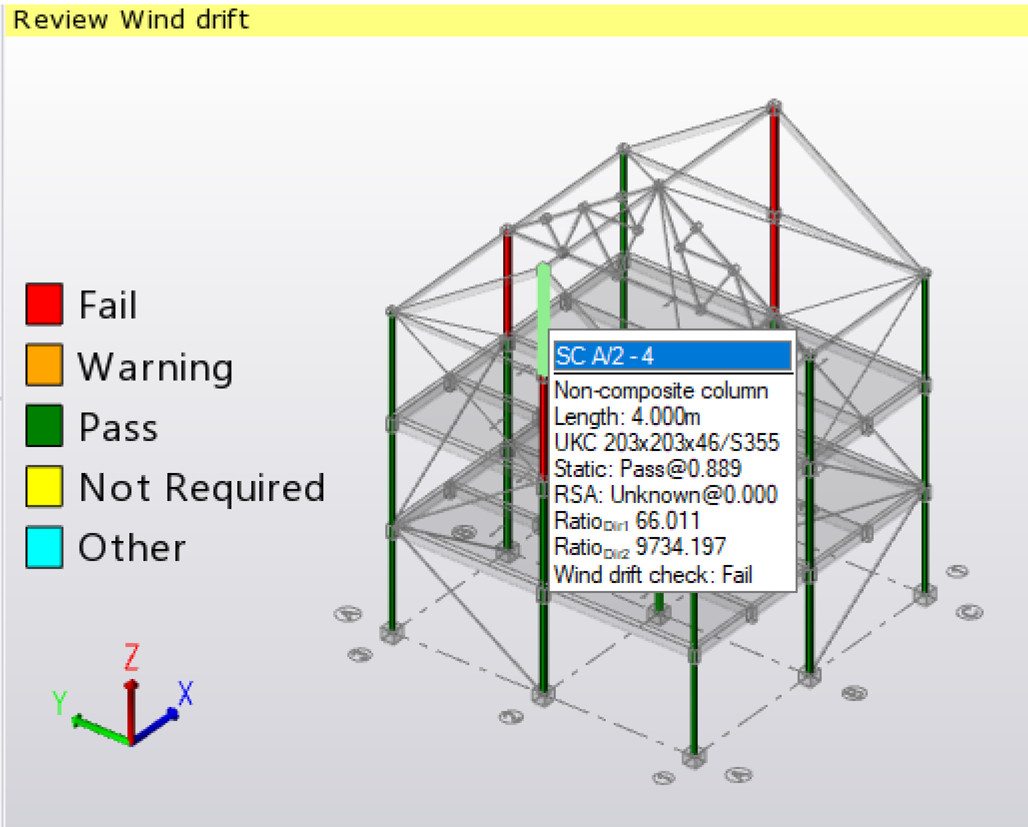
The individual checks can be investigated in more detail by clicking the **Details** button at the end of any row.



Review the check graphically

From the right click menu in the Status Tree you are also provided with an option to graphically review the checks in a Show/Alter State View (page 760).





This color coded legend makes it easy to see individual failing members. By hovering the cursor over a member, its wind drift results are displayed in the tooltip.

From the same Show/Alter State view, by changing the Mode in the **Properties** window, you also have options to switch off checks (page 761), or set a different drift limit (page 761) for specific stacks if you require.

Switching off inappropriate checks and merging stack lengths

After reviewing you might choose to switch off checks for selected columns and walls, or individual stacks/panels within selected columns and walls: this can be done directly via the (page 684), or graphically in a Show/Alter State view (as described in the previous section).

You might also choose to take advantage of the option to merge short stacks automatically (page 681) where they are less than a specified limit. Stacks check lengths can also be be manually adjusted (page 685) by merging them together if required.

An option is also provided which allows you to switch off checks for specific levels (page 685) if you decide they are inappropriate, (for example if the story height is very short). The checks at that level are still performed, but they are excluded from the tabular review data display.

Printing calculations

The wind drift check results table is available either by creating a Building Analysis & Drift Checks report (page 841), or by including the Analysis>Wind Drift chapter in a custom report.

NOTE The Wind Drift chapter has a Settings option that can be used to dramatically reduce the number of report pages. By default all columns and walls are output, but you can specify instead the most critical 'XX' columns and walls, (where 'XX' is a number of your choice).

The same Settings option also allows you to choose between reporting only the critical stack/all stacks, and reporting only the critical combination/all combinations.

Overall wind drift check

When activated, by default this check considers all columns (of all materials), and all walls at every level in the model.

In this check the overall wind drift ratio, (H / Δ_{max}) is compared against a user defined limit.

For levels set as floors a **Check for drift** option is displayed in Level Properties (page 933). This is used in the above calculation as follows:

H	=	Distance between the highest and lowest levels with **Check for drift** selected.
Δ_{max}	=	Maximum lateral deflection at the highest level with **Check for drift** selected.

By default the Δ_{max} is calculated in the resultant wind drift direction only:

$\Delta_{max} = \Delta_{max,Resultant}$

By unselecting in Design Settings (page 682) it is calculated as follows:

$\Delta_{max} = \max [\Delta_{max, Dir1}, \Delta_{max, Dir2}]$

A workflow for running the check is outlined below:

Configure the check and set the limit

The check is activated from **Design Settings**.

On the **Drift Checks** page (if working to the ACI/AISC head code) or **Sway & Drift Checks** page (if working to any other head code), select **Check for Overall Wind Drift** and then enter the limit required.

NOTE By default the check is activated for new models with the limit set as 500.

By default the check is performed in the resultant wind drift direction only, but by unselecting **Check for Resultant Wind Drift** it can be performed in each of Building Directions 1 & 2 if required.

By default the check only considers the effects of the wind loadcase(s) in wind combinations, but by unselecting **Check wind cases only** it will consider the effects of all loadcases in wind combinations (which would include drift induced by gravity loads).

Run the check

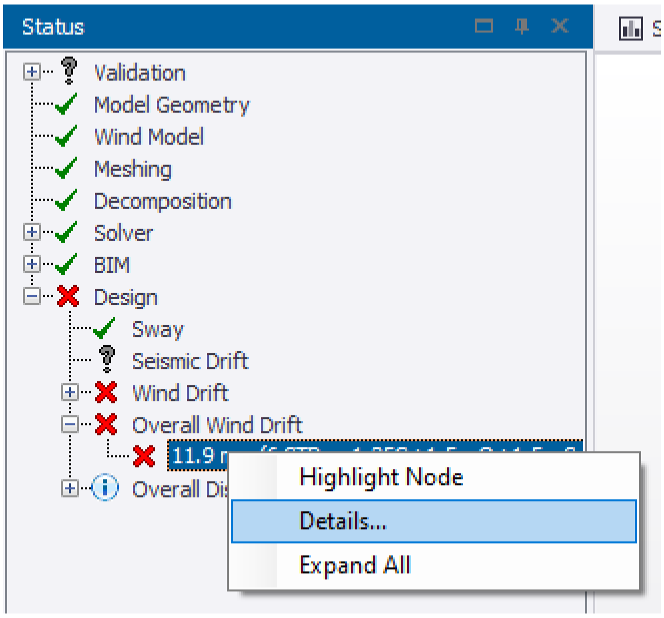
Provided the check has been activated, wind load combinations are automatically checked for overall wind drift during the structure static design, and also when any 3D analysis is run in isolation. If a sub-set of combinations are considered for analysis then only those combinations are checked, (allowing engineers working on larger structures to investigate and optimize the lateral load resisting systems more rapidly).

In performing the checks, the maximum lateral deflection (Δ_{max}) associated with the highest level of active check for drift is determined for each combination using the combination SLS (Service Level) factors - (which can be < 1.0). The ratio of H / Δ_{max} is then compared against a user-defined limit.

Review the check status and details

Once the check has been run, an **Overall Wind Drift** status is reported in the **Design** branch of the Status Tree (page 82). Beneath the status the node with maximum wind drift deflection value and its associated combination are listed.

Right clicking on the value opens a context menu, from which you can choose **Highlight Node** to see where the maximum occurs in a solver view.



You can also choose Details... to open a dialog which shows the results of the check.

Overall displacement

By expanding the **Design** branch of the Status Tree (page 82), you are able to review the maximum positive and negative overall displacement results from the 3D Analysis for both Strength and Service combinations.

Displacements can also be viewed graphically in the Results View by using the buttons on the Deflections group.

Common tasks for sway and drift checks

Automatically merge short stacks

Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of sway/drift, wind drift and seismic drift checks only, you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.

A setting is provided in **Design Settings** to automatically do this when stacks are less than a given length.

- With the setting off, the check considers all stacks of all columns/walls apart from those that have been manually excluded.
- With the setting on, any stacks that are less than the limit specified are merged with the stack below, and the check is performed on the merged stack length. The check is not performed on single stack columns/walls that are less than this limit .

To turn the setting on:

1. On the **Design** ribbon, click **Settings**.
2. On the left hand side of the **Design Settings** dialog, click:
   - **Drift Checks** (if working to the ACI/AISC head code)
   - **Sway & Drift Checks** (if working to any other head code)
3. Select **Merge short stacks**.
4. Enter the minimum stack length, (below which the stack will be merged with the next adjacent stack).
5. Click **OK**

Set the wind drift limit

1. On the **Design** ribbon, click **Settings**.
2. On the left hand side of the **Design Settings** dialog, click:
   - **Drift Checks** (if working to the ACI/AISC head code)
   - **Sway & Drift Checks** (if working to any other head code)
3. Enter the **Wind Drift Limit** required.
4. Click **OK**

The limit is applied to all columns and walls, but can be overridden if required for specific members via in the member properties.

Choose resultant or directional wind drift checks

1. On the **Design** ribbon, click **Settings**.
2. On the left hand side of the **Design Settings** dialog, click:
   - **Drift Checks** (if working to the ACI/AISC head code)
   - **Sway & Drift Checks** (if working to any other head code)
3. Select or unselect **Check for Resultant Wind Drift**.
   - Selected - the wind drift limit is checked against the resultant wind drift
   - Unselected - two checks are performed, one each for Building Directions 1 & 2.
4. Click **OK**

Consider wind cases only for the wind drift check

A **Check wind cases only** design setting (default on) is provided which applies to the check as follows:

- With the setting on, the Wind Drift check only considers the effects of the wind loadcase(s) in wind combinations.
- With the setting off, the check considers the effects of all loadcases in wind combinations (which would include drift induced by gravity loads).

1. On the **Design** ribbon, click **Settings**.
2. On the left hand side of the **Design Settings** dialog, click:
   - **Drift Checks** (if working to the ACI/AISC head code)
   - **Sway & Drift Checks** (if working to any other head code)
3. Select or unselect **Check wind cases only** as required.
4. Click **OK**

Switch off sway checks for selected columns/walls

Selected column stacks or wall panels can be excluded from the sway check as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.
2. Expand the appropriate **Sway & Drift Checks** heading:
   - under **All stacks**, **All Panels** - to exclude the entire column/wall
   - under the required **Stack** or **Panel** - to exclude a specific stack/panel
3. Unselect **Sway/Seismic drift checks**

Switch off drift checks for selected columns/walls

Selected column stacks or wall panels can be excluded from the drift check as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.
2. Expand the appropriate **Drift Checks** heading:
   - under **All stacks**, **All Panels** - to exclude the entire column/wall
   - under the required **Stack** or **Panel** - to exclude a specific stack/panel
3. Unselect **Drift/Seismic drift checks and consider for seismic torsion**

Switch off seismic drift checks for selected columns/walls

Selected column stacks or wall panels can be excluded from the seismic drift check as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.
2. Expand the appropriate **Sway & Drift Checks*** heading:
   - under **All stacks**, **All Panels** - to exclude the entire column/wall
   - under the required **Stack** or **Panel** - to exclude a specific stack/panel
3. Unselect:
   - **Drift/Seismic drift checks and consider for seismic torsion** (if working to the ACI/AISC head code)
   - **Sway/Seismic drift checks** (if working to any other head code)

   NOTE *If working to the ACI/AISC head code this heading is shortened to **Drift Checks**

Switch off wind drift checks for selected columns/walls

Selected column stacks or wall panels can be excluded from the wind drift check as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.
2. Expand the appropriate **Sway & Drift Checks*** heading:
   - under **All stacks**, **All Panels** - to exclude the entire column/wall
   - under the required **Stack** or **Panel** - to exclude a specific stack/panel

3. Unselect **Wind drift check**

NOTE *If working to the ACI/AISC head code this heading is shortened to **Drift Checks**

Switch off tabular results for an entire level

The sway/drift, seismic drift and wind drift check results for all column stacks and wall panels at a specific level can be excluded from the tabular review data as follows:

1. Go to the **Project Workspace**.
2. In the **Structure** tree, expand **Levels** then select the level at which you want the tabular results to be switched off.
3. In the **Properties** window, unselect the **Check for drift** option.

Override the wind drift limit for selected columns/walls

The wind drift limit specified in Design Settings can be overridden for selected column stacks or wall panels as follows;

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.
2. Expand the appropriate **Sway & Drift Checks*** heading:
   - under **All stacks**, **All Panels** - to override the entire column/wall
   - under the required **Stack** or **Panel** - to override a specific stack/panel
3. Select **Override Wind Drift Limit**
4. Enter the required **Wind drift ratio limit**.

NOTE *If working to the ACI/AISC head code this heading is shortened to **Drift Checks**

Adjust column stack or wall panel check lengths

Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of sway/drift, wind drift and seismic drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.

To do this manually proceed as follows:

1. Select the desired column(s) or wall(s).

   The column or wall properties are displayed in the **Properties** window.

2. Under the required **Stack** or **Panel** group, unselect the following setting:

   - **Drift Checks > Merge with stack below** (if working to the ACI/AISC head code)
   - **Sway & Drift Checks > Merge with stack below** (if working to any other head code)

Considerations for non-linear models with Tension Only bracing

For non-linear models with Tension Only bracing it is essential that the 'X Brace' pattern is used to input the braces as brace pairs rather than creating them individually. For linear analysis one of the braces in a brace pair is automatically inactivated, ensuring that the model's lateral stiffness, and hence lateral drift, is reasonably correct. If braces are input individually this will not be the case.

Perform checks

The checks are performed either by running Analyze All (Static) from the Analyze ribbon, or by running any of the Design (Static) commands from the Design ribbon.

Review tabular results in a Data Table

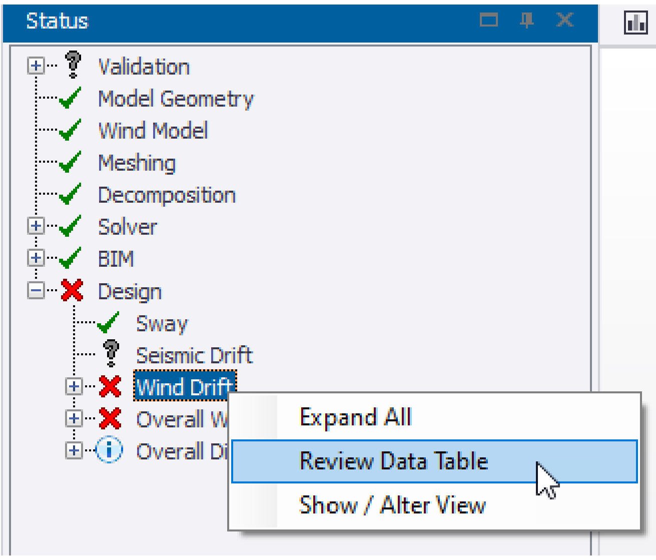
Tabular results can either be reported in summary, or in detail.

- Summary - the column/wall with most critical result from all combinations is reported for each direction at each level
- Detailed - the critical result in each direction at each level is reported for all columns/walls

The results can also be filtered by material type, characteristic and fabrication as required.

To access the tabular results from the Project Workspace Status Tree proceed as follows:

1. Expand the Design branch of the Project Workspace Status Tree.
2. Right-click on the required check.
3. Select **Review Data Table**



For further information, see:

- Review drift check tabular results (page 770)
- Review sway check tabular results (page 765)
- Review seismic drift check tabular results (page 773)
- Review wind drift check tabular results (page 777)

Review results graphically in a Show / Alter State view

To access the graphical results from the Project Workspace Status Tree proceed as follows:

1. Expand the Design branch of the Project Workspace Status Tree.
2. Right-click on the required check.
3. Select **Show / Alter View**

For further information, see:

- Review and modify sway checks (page 756)
- Review and modify drift checks (page 736)
- Review and modify seismic drift (page 752)
- Review and modify wind drift checks (page 760)

Create a report

Each of the results tables can be included in printed output by adding the Drift, Sway, Seismic Drift and Wind Drift chapters from the Analysis branch into your model report.

8 Create and design foundations

You can create and design both isolated and mat foundations using the **Foundations** toolbar.

8.1 Create isolated foundations

Tekla Structural Designer allows you to create pad base, strip base and pile cap isolated foundations. Before you can create a pile cap the Pile Catalogue must contain at least one pile type.

Click the links below to find out more:

- Create pad bases and strip bases (page 688)
- Create a pile type catalogue (page 690)
- Create pile caps (page 690)

See also

Design isolated foundations (page 692)

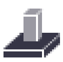
Create pad bases and strip bases

You can create both pad base columns and strip base walls in your model. Pad bases and strip bases are both isolated foundations. The difference is that pad bases support a column, while strip bases support a wall.

Create pad base columns

A pad base is an isolated foundation that supports a single column. To place pad bases in your model, see the following instructions.

Create a pad base under a specific column

1. On the **Foundations** tab, click  **Pad Base Column**.

2. In the **Properties** window, adjust the pad base properties according to your needs.
3. Click anywhere in the wall under which you want to place the pad base.

   Tekla Structural Designer places a pad base under the selected column.

Create multiple pad base columns

1. On the **Foundations** tab, click  **Pad Base Column**.
2. In the **Properties** window, adjust the pad base properties according to your needs.
3. Move the mouse pointer to a corner of an imaginary box that will encompass the columns under which you want to place pad bases.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the opposite corner of the box.
6. Release the mouse button.

   Tekla Structural Designer places pad bases under all columns within the box.

Create strip base walls

A strip base is an isolated foundation that supports a single wall. To place strip bases in your model, see the following instructions.

Create a strip base under a specific wall

1. On the **Foundations** tab, click  **Strip Base Wall**.
2. In the **Properties** window, adjust the strip base properties according to your needs.
3. Click anywhere in the wall under which you want to place the strip base.

   Tekla Structural Designer places a strip base under the selected wall.

Create multiple strip bases

1. On the **Foundations** tab, click  **Strip Base Wall**.
2. In the **Properties** window, adjust the strip base properties according to your needs.
3. Move the mouse pointer to a corner of an imaginary box that will encompass the walls under which you want to place strip bases.
4. Hold down the left mouse button.

5. Drag the mouse pointer to the opposite corner of the box.
6. Release the mouse button.

   Tekla Structural Designer places strip bases under all walls within the box.

Create a pile type catalogue

In order to use pile foundations in your model, you must create a catalogue of pile types. To do so, see the following instructions.

1. On the **Foundations** tab, click **Catalogue**.

   The **Pile Catalogue** opens.
2. To create a new pile type, click **Add...**

   The **Edit Pile Type** dialog box opens.
3. On the different pages of the **Edit Pile Type** dialog box, define the properties of the new pile type.
4. Click **OK**.

   Tekla Structural Designer creates the pile type.
5. Repeat steps 2–4 as necessary.

TIP In the **Pile Catalogue**, you can also modify or delete the existing pile types:

- To modify a pile type, select the pile type and click **Edit...**
- To delete a pile type, select the pile and click **Delete**.

Create pile caps

A pile cap is an isolated piled foundation that supports a single column.

NOTE Before you can create pile caps, the **Pile Catalogue** must contain at least one pile type.

To create pile types, see Create a pile type catalogue (page 690).

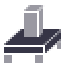
Create pile cap under a specific column

1. On the **Foundations** tab, click **Pile Cap Column**.
2. In the **Properties** window, adjust the pile cap properties according to your needs.

3. Click anywhere in the column under which you want to place the pile cap.

   Tekla Structural Designer places a pile cap under the selected column.

Create multiple pile caps

1. On the **Foundations** tab, click  **Pile Cap Column**.
2. In the **Properties** window, adjust the pile cap properties according to your needs.
3. Move the mouse pointer to a corner of an imaginary box that will encompass the columns under which you want to place pile caps.
4. Hold down the left mouse button.
5. Drag the mouse pointer to the opposite corner of the box.
6. Release the mouse button.

   Tekla Structural Designer places pile caps under all columns within the box.

Create a user-defined pile arrangement

If necessary, you can modify the pile arrangement in a pile cap and create user-defined pile arrangements. Creating user-defined pile arrangements can be particularly useful when you want to check a pile cap where the pile positions on site do not exactly match the originally specified arrangement.

Do the following:

1. In the model, select the pile cap that you want to modify.
2. In the **Properties** window, go to **Pile arrangement**.
3. Click the **...** button next to **Pile arrangement**.

   The **Pile Arrangement** dialog box opens.
4. In the **Pile Arrangement** dialog box, select the **User Defined Arrangement** option.

   NOTE If the **Auto-design piles** option is selected in the **Properties** window, Tekla Structural Designer asks you whether you want to turn off the option. Click **Yes**.

5. Specify the pile arrangement properties according to your needs:
   a. Select the pile type.
   b. Define the pile length and width.
   c. If necessary, define the eccentricity of the pile cap centroid from the centroid of the column it supports in X and Y directions.

TIP To add more piles into the arrangement, click **Add**.

6. If necessary, in the table below the properties, adjust the position of piles by defining the eccentricity of the pile centroid from the pile cap centroid.
7. Click **OK**.

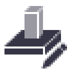
   Tekla Structural Designer creates the pale arrangement that you defined.

8.2 Design isolated foundations

You can design isolated foundations (pad bases, strip bases, and pile caps) either in a batch or individually.

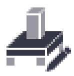
NOTE Before bases or pile caps can be designed, you have to run an analysis to establish the design forces. You can run either the **Analyze All (Static)** command or one of the combined analysis and design commands (such as **Design All**).

Design or check all pad bases and strip bases

1. On the **Foundations** tab, click  **Design Pad Bases**.

   Tekla Structural Designer designs or checks all pad bases and strip bases in the model according to their individual auto-design settings. In the process, Tekla Structural Designer considers all active static and RSA combinations .

Design or check all pile caps

1. On the **Foundations** tab, click  **Design Pile Caps**.

   Tekla Structural Designer designs or checks all pile caps in the model according to their individual auto-design settings. In the process, Tekla Structural Designer considers all active static and RSA combinations .

Check an individual isolated foundation

1. Hover the mouse pointer over the foundation that you want to check.

   The **Select Entity** tooltip appears.

2. In the **Select Entity** tooltip, navigate to the foundation name by using the arrow keys.
3. Right-click the foundation.
4. In the context menu, select **Check Member**. Tekla Structural Designer displays the results of the check in a new dialog box.

Design an individual isolated foundation

1. Hover the mouse pointer over the foundation that you want to design.

   The **Select Entity** tooltip appears.
2. In the **Select Entity** tooltip, navigate to the foundation name by using the arrow keys.
3. Right-click the foundation.
4. In the context menu, select **Design Member**. Tekla Structural Designer displays the results of the design in a new dialog box.

See also

Apply user defined utilization ratios (page 613)

8.3 Create mat foundations

Mat foundations support multiple columns and walls. Mats can be ground bearing, supported on piles, or both. Tekla Structural Designer contains he following commands for creating mat foundations:

- **Minimum Area**: creates an overhanging polygonal mat to minimize the required area
- **Rectangular**: creates an overhanging rectangular mat either at a specified angle to the global axes or at the smallest rectangular area aligned to the global axes
- **Strip**: creates a constant width mat along a series of points that do not have to be on the same line
- **Area**: creates an overhanging polygonal mat when you identify points around its outline
- **Bays**: creates a mat with no overhang when you click within a closed grid area
- **Pile**: creates individual piles underneath existing mats
- **Pile Array**: creates an array of piles underneath existing mats

Click the links below to find out more:

- Create mats (page 694)
- Place piles and pile arrays in mats (page 695)

See also

Create slab or mat openings (page 265)

Add overhangs to existing slab or mat edges (page 267)

Create column drops (page 270)

Split and join slabs and mats (page 273)

Design isolated foundations (page 692)

Create mats

In Tekla Structural Designer, you can create various types of mat foundations, including minimum area mats, rectangular mats, strip mats, area mats, and mats with bays. For detailed information on creating mats, see the following instructions.

Create a minimum area or rectangular mat

1. On the **Foundations** tab, click either  **Minimum Area** or  **Rectangular**.
2. In the **Properties** window, adjust the mat properties according to your needs.
3. Do one of the following:

To	Do this
Create a mat in a 3D view	a. Click anywhere on the first column or wall to be supported. b. Click to define the other necessary columns and walls. c. To create the mat, click the last column or wall again or press **Enter**.
Create a mat in a 2D view	a. Do one of the following: 1. To select multiple walls, hold down the left mouse button and drag a box from right to left to encompass the walls under which you want to create the mat. 2. To select multiple columns, hold down the left mouse button and drag a box from left to right to encompass the columns under which you want to create the mat.

	3. To add individual columns and walls, click each member individually. b. To create the mat, click one of the previously selected members or press **Enter**.

Create a strip mat

1. Open a 2D view at the base level where you want to place the strip mat.
2. On the **Foundations** tab, click **Strip**.
3. In the **Properties** window, adjust the strip width and other mat properties according to your needs.
4. In the model, click the start point of the strip.
5. Click to define the other necessary points.
6. To create the strip, click the last point again or press **Enter**.

Create an area mat

Either click once again on the last point, or press <Enter> to create the mat.

1. On the **Foundations** tab, click **Area**.
2. In the **Properties** window, adjust the mat properties according to your needs.
3. In the model, click a construction point at a corner of the area.
4. Click to define the remaining corner points.
5. To create the mat, click the last point again or press **Enter**.

Create a mat within bays

1. On the **Foundations** tab, click **Bays**.
2. In the **Properties** window, adjust the mat properties according to your needs.
3. To select a bay, do one of the following:
   - Click within an enclosed area defined by grid lines.
   - Hold down the left mouse button and drag a box around the required area.

Place piles and pile arrays in mats

A piled mat can either be supported on piles alone, or by a combination of ground springs and piles. Piles can either be placed individually at specific locations within the mat, or an array of equally spaced piles can be created. To place piles in mats and define their inclination, see the following instructions.

Specify if a piled mat is ground bearing

1. Click the desired mat.
2. In the **Properties** window, go to **Soil Parameters**.
3. Do one of the following:

To	Description
Create a mat supported on piles alone	• Clear the **Use Ground Bearing Springs** option.
Create a mat supported by a combination of ground spring and piles	• Ensure that the **Use Ground Bearing Springs** option is selected.

Place an individual pile in a mat

1. On the **Foundations** tab, click **Pile**.
2. Click the mat in which you want to place the pile.
3. Click the pile position in the selected mat.

   Tekla Structural Designer places the pile in the selected position.

Place a pile array in a mat

1. On the **Foundations** tab, click **Pile Array**.
2. In the **Properties** window, adjust the pile type, spacing, and pile direction according to your needs.

   TIP To preview the array, hover the mouse pointer over a mat.

3. Click an individual mat, or hold down the left mouse button and drag a box around multiple mats.
4. If necessary, add further mats to the selection by clicking or boxing them.
5. To create the mat, click the last mat you selected again.

   Tekla Structural Designer places the piles in the mat.

6. Repeat steps 2–4 to place more piles or press **Esc** to finish placing piles.

Specify the pile direction of an inclined pile

1. On the **Foundations** tab, click **Pile**.
2. In the **Properties** window, do one of the following:

To	Do this
Specify the direction as a vector	a. Adjust the X and Y components of the vector. **TIP** If you do not want the pile to slope in the global X or Y direction, leave the appropriate value as 0. b. Type a negative value in the Z component field. For example: To create an inclined pile of 45 degrees in positive X, define the components as follows: `X = 1.0, Y = 0.0, Z = -1.0`. To create an inclined pile of 45 degrees in negative Y, define the components as follows: `X = 0.0, Y = 0.5, Z = -1.0`
Specify the direction as an angular measurement	**NOTE** You can only specify the direction as an angular measurement before the pile is first positioned. a. Select the **Direction by Angles** option. b. Define the inclination of the pile from the vertical. c. Define the azimuth (the horizontal angle measured clocwise about the positive global Z direction. d. If necessary, define the rotation about the longitudinal axis of the pile. **NOTE** Since the pile is only checked for axial capacity, the rotation angle has no effect on the pile checks. However, you can define the rotation to vary how Tekla Structural Designer displays the moment and shear force values of the pile. For example: To create an inclined pile of 45 degrees in positive X, define the angles as follows: `inclination = 45 degrees, azimuth = 90 degreeS` To create an inclined pile of 30 degrees in negative Y, define the angles as follows: `inclination = 30 degrees, azimuth = 0.0 degrees`

8.4 Design mat foundations

Design or check all mats in the model

1. On the **Foundations** tab, click **Design Mats**.

   Tekla Structural Designer designs or checks all mats in the model according to their auto design settings.

Check all mats in a single floor

1. Open a 2D view of the floor that you want to design.
2. Right-click anywhere in the scene view.
3. In the context menu, select **Check Slabs**.

   Tekla Structural Designer checks all slab items (including mat foundations) in the floor regardless of their autodesign settings.

Design all mats in a single floor

1. Open a 2D view of the floor that you want to design.
2. Right-click anywhere in the scene view.
3. In the context menu, select **Design Slabs**. Tekla Structural Designer designs all slab items (including mat foundations) in the floor regardless of their autodesign settings.

Check an individual mat

1. Hover the mouse pointer over the mat that you want to check.
2. Right-click the mat.
3. In the context menu that appears, select **Check Panel**.

   Tekla Structural Designer displays the results of the check in a new dialog box.

Design an individual mat

1. Hover the mouse pointer over the mat that you want to design.

2. Right-click the mat.
3. In the context menu that appears, select **Design Member**.

   Tekla Structural Designer displays the results of the design in a new dialog box.

See also

Apply user defined utilization ratios (page 613)

9 Review models

By selecting the **Review** view regime, you are able to graphically display the design status of members, slabs, foundations and connections. In the same view you can interrogate or modify a variety of model parameters and properties.

In addition, by clicking **Tabular Data** from the toolbar, you can open a **Review Data** view, from where you can display tables of Sway/Drift Results, Design Summaries and Material Lists.

TIP The **Review** view opens automatically at the end of the structure design process, but you can access it at any time by clicking the icon in the **Status bar**

9.1 Review designs

By graphically reviewing the design status and ratios you are able to quickly assess the design efficiency.

Click the links below to find out more:

- Set the design type to review (page 700)
- Review member design (page 701)
- Review foundation and pile design (page 702)
- Review slab and mat design (page 703)
- Design review filters (page 705)

Set the design type to review

You can set a specific design type to focus on when reviewing the member, foundation, or slab designs.

The available design type options are Static, RSA, and Combined (i.e. Static and RSA together).

To set the design type, do the following:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, in the **Type** list, select the desired design type.
   - Static
   - RSA
   - Combined

See also

Review member design (page 701)

Review foundation and pile design (page 702)

Review slab and mat design (page 703)

Design review filters (page 705)

Review member design

Review member design status

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, in the **Design** group, click **Status**.

   Color codes display the design status of each member.

   Tekla Structural Designer uses the following classifications:

Status	Description
Pass	The member has passed all design checks.
Fail	The member has failed one or more design checks.
Warning	Although the member has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.

Status	Description
Error	The member currently cannot be designed because an error has occurred.
Beyond Scope	The member cannot be designed because its design is beyond the scope of Tekla Structural Designer.
Unknown	The member has not been designed.

Review member design ratios

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, in the **Design** group, click **Ratio**.

   Color codes display the design ratio of each member. **N/A** is assigned to members that have no ratio or whose ration is smaller than the lowest band. **N/A** also indicates members that either are beyond scope or have yet to be designed.

Review member depth ratios

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, in the **Design** group, click **Depth Ratio**.

   All steel and concrete beams are color coded to indicate their span to depth utilization ratios.

See also

Design review filters (page 705)

Review design summary tabular results (page 763)

Review foundation and pile design

Review foundation or pile status

1. Open a view and change the view regime (page 89) to **Review View**.
2. Go to the **Review** ribbon.
3. Either in the **Foundations** group or the **Piles** group, click **Status**.

   Color codes display the design status of each foundation or pile.

   Tekla Structural Designer uses the following classifications:

Status	Description
Pass	The foundation or pile has passed all design checks.
Fail	The foundation or pile has failed one or more design checks.
Warning	Although the foundation or pile has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.
Error	The foundation or pile currently cannot be designed because an error has occurred.
Beyond Scope	The foundation or pile cannot be designed because its design is beyond the scope of Tekla Structural Designer.
Unknown	The foundation or pile has not been designed.

Review foundation or pile ratios

1. Open a view and change the view regime (page 89) to **Review View**.
2. Go to the **Review** ribbon.
3. Either in the **Foundations** group or the **Piles** group, click **Ratio**.

   Color codes display the design ratio of each foundation or pile. **N/A** is assigned to members that have no ratio or whose ration is smaller than the lowest band. **N/A** also indicates foundations and piles that are either beyond scope or have yet to be designed.

See also

Design review filters (page 705)

Review design summary tabular results (page 763)

Review slab and mat design

Review slab and mat design status

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, in the **Slab/Mat Design** group, click **Status**.

Color codes display the design status of each slab and mat.

Tekla Structural Designer uses the following classifications:

Status	Description
Pass	The slab item has passed all design checks.
Fail	The slab item has failed one or more design checks.
Warning	Although the slab item has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.
Error	The slab item currently cannot be designed because an error has occurred.
Beyond Scope	The slab item cannot be designed because its design is beyond the scope of Tekla Structural Designer.
Unknown	The slab item has not been designed.

Review slab and mat design ratios

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, in the **Slab/Mat Design** group, click **Ratio**.

   Color codes display the design ratio of each slab item. **N/A** is assigned to members that have no ratio or whose ration is smaller than the lowest band. **N/A** also indicates slab items that are either beyond scope or have yet to be designed.

Filter slab and mat design information

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, click the list in the **Slab/Mat Design** group.
3. In the list that appears, select one of the following filtering options according to your needs:

Filter	Description
Overall	The governing Top X, Top Y, Bottom X, Bottom Y, or span depth result is displayed.
Reinforcement	The governing Top X, Top Y, Bottom X, or Bottom Y result is displayed.

Filter	Description
Top X	The result is only displayed for top reinforcement in the X direction.
Top Y	The result is only displayed for top reinforcement in the Y direction.
Bottom X	The result is only displayed for bottom reinforcement in the X direction.
Bottom Y	The result is only displayed for bottom reinforcement in the Y direction.
Span Depth	The span to depth result is displayed.
Bearing Pressure	The bearing pressure result is displayed.

See also

Design review filters (page 705)

Review design summary tabular results (page 763)

Design review filters

Design review filters help the user to focus on specific results.

The filters are displayed in the **Properties** window when **Status** or **Ratio** are selected from any group on the **Review** ribbon.

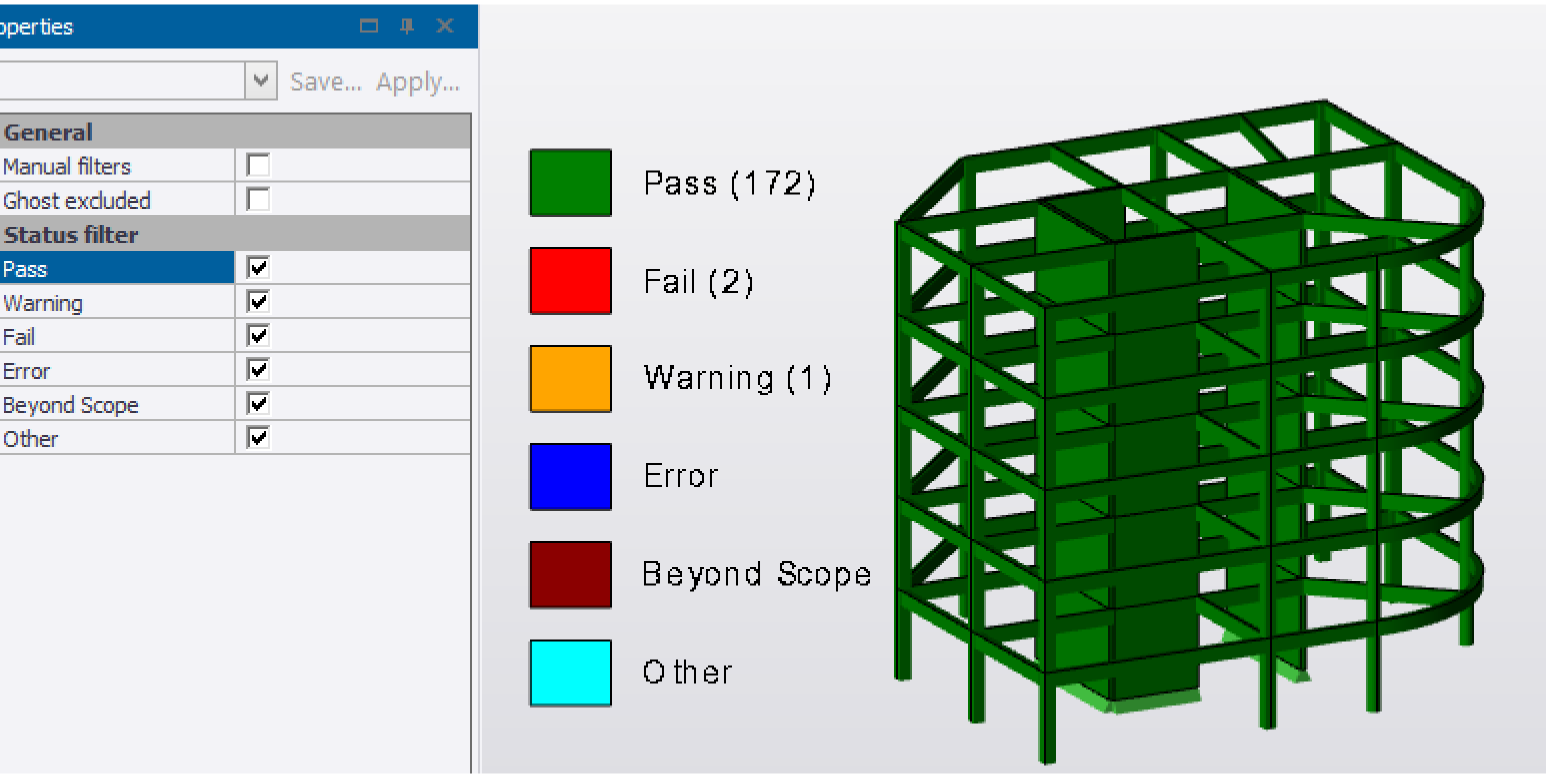
The following filter types are available and can be used individually or in combination:

- **Status filter**
- **Utilization ratio filter**
- **Entity type filter**

TIP These filters can be particularly useful when working with large models.

Working with the Status filter

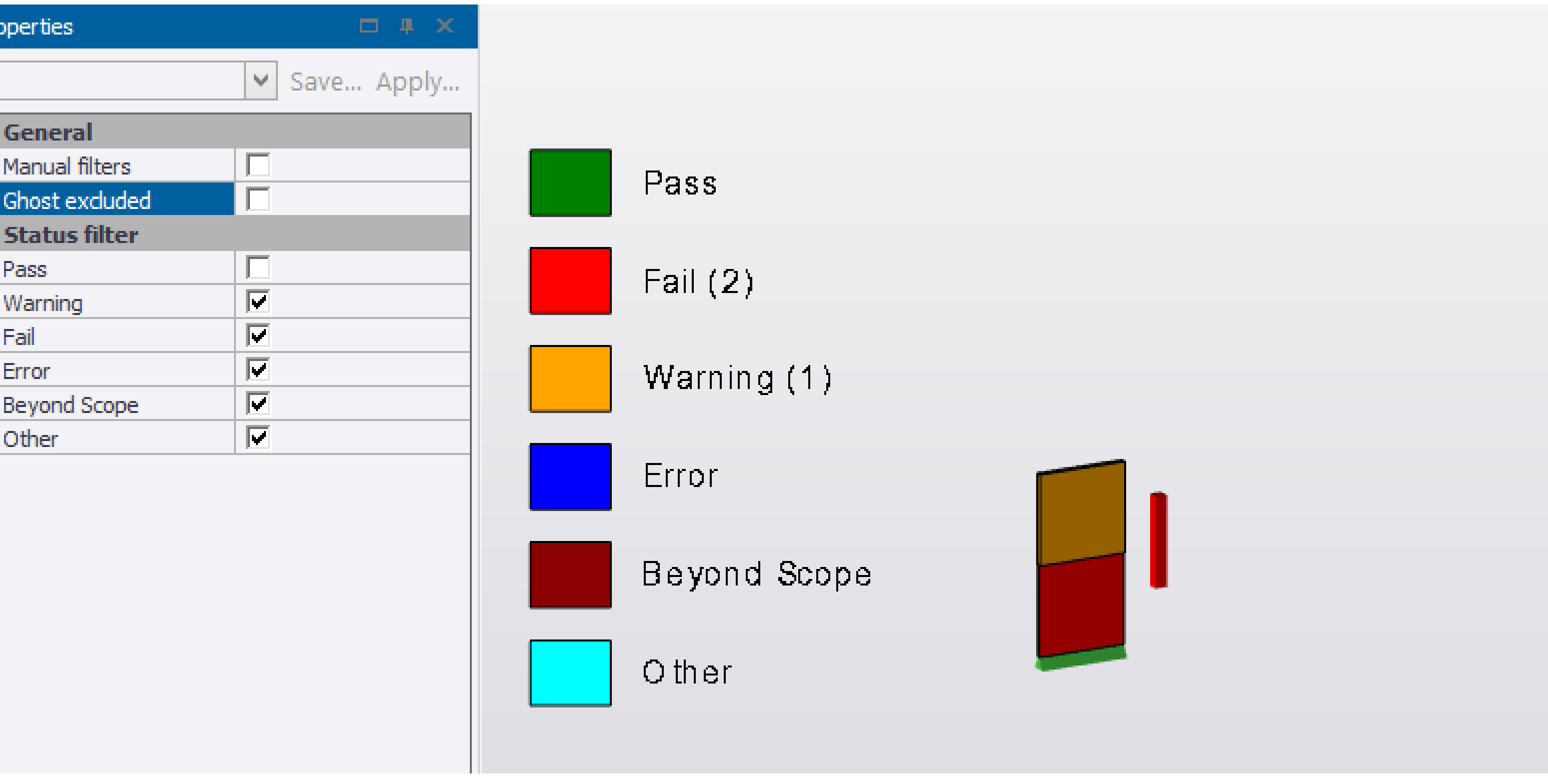
The **Status filter** is always active in the **Properties** window when **Status** is selected from the **Review** ribbon.



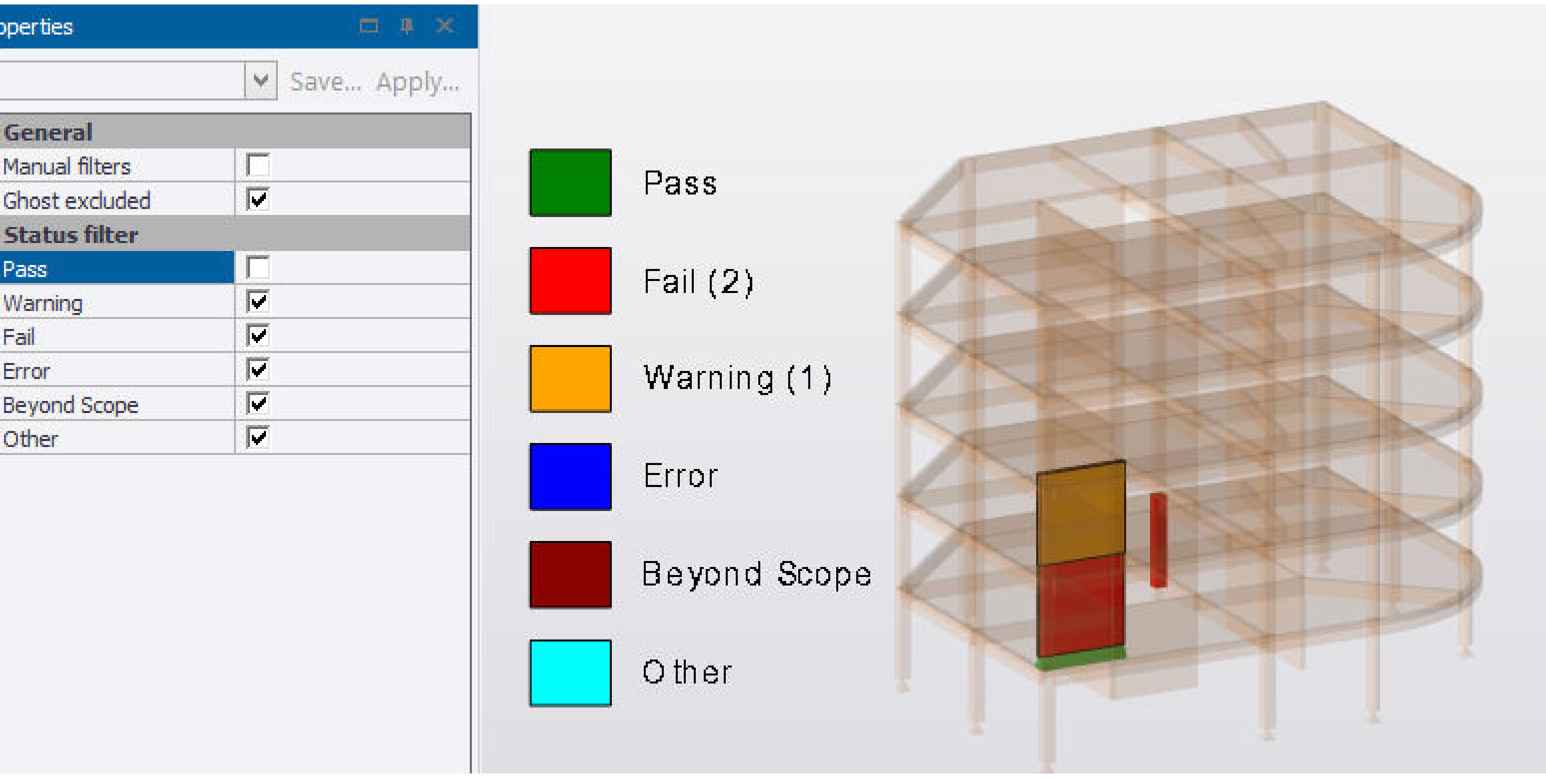
All entities are assigned a design status from the below list:

- **Pass** - The entity has passed all design checks.
- **Warning** - Although the entity has passed the design checks, one or more warnings have been issued. Review the warnings before deciding if you need to take further action.
- **Fail** - The entity has failed one or more design checks.
- **Error** - The entity currently cannot be designed because an error has occurred.
- **Beyond Scope** - The entity cannot be designed because its design is beyond the scope of Tekla Structural Designer.
- **Unknown** - The entity has not been designed.

In the above model the legend indicates that two members are failing and another has a warning but their locations are not immediately apparent. Deselecting a status in the **Properties** window causes entities of that status to be hidden in the **Review View**, so in this instance deselecting the Pass status helps to focus on where the problems are occurring.



By also selecting **Ghost excluded** the filtered results can be displayed in relation to the rest of the model.

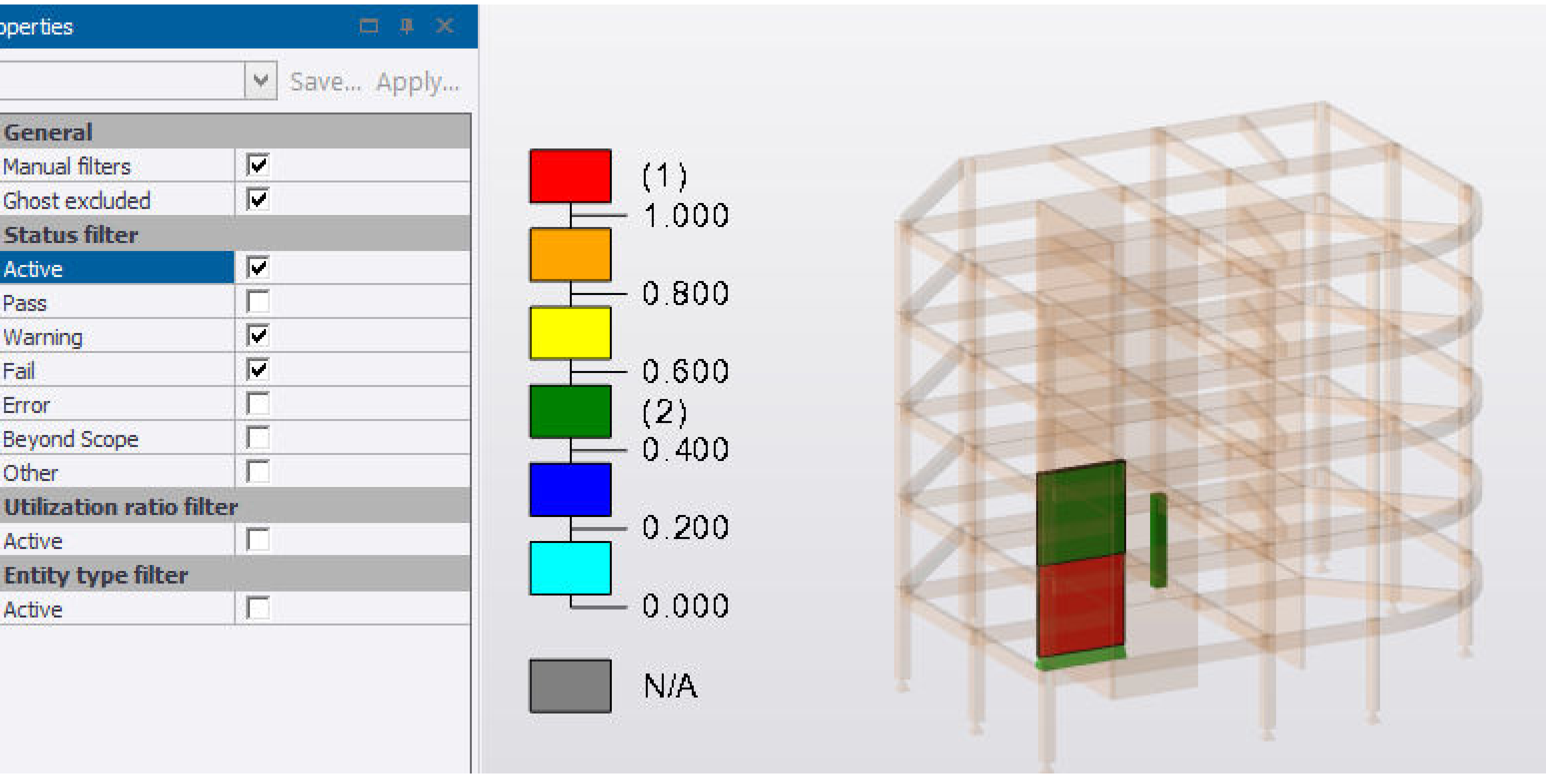


If required, you can also apply the **Status filter** when investigating ratios. This is achieved as follows:

1. From the **Review** ribbon select **Ratio**
2. Select **Manual filters** in the **Properties** window.

   The **Status filter** group is displayed.
3. Select **Active** and choose the statuses by which to filter the display of ratios.

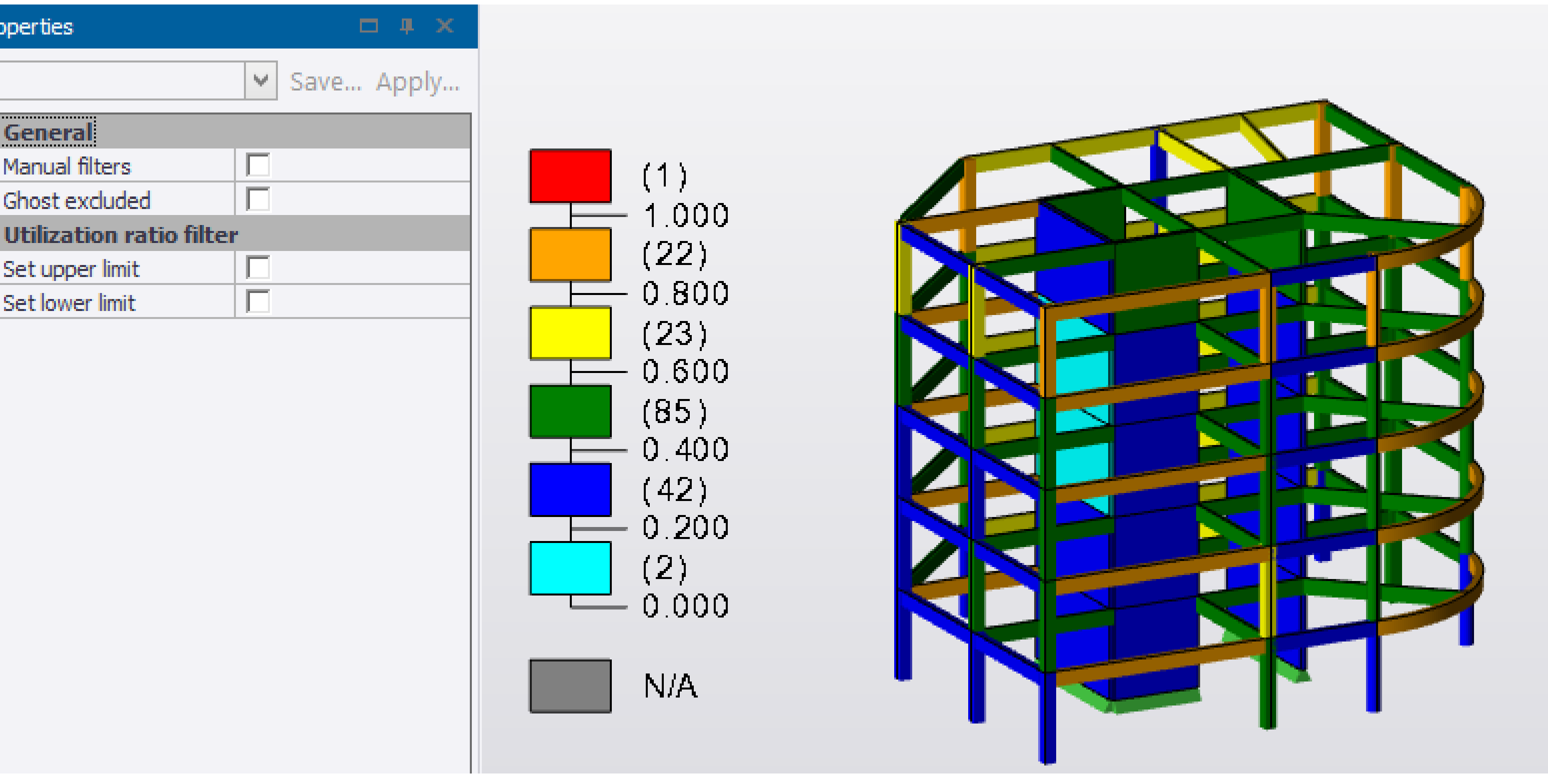
   The result is that ratios are then only displayed for those members in the chosen **Status filter** categories.



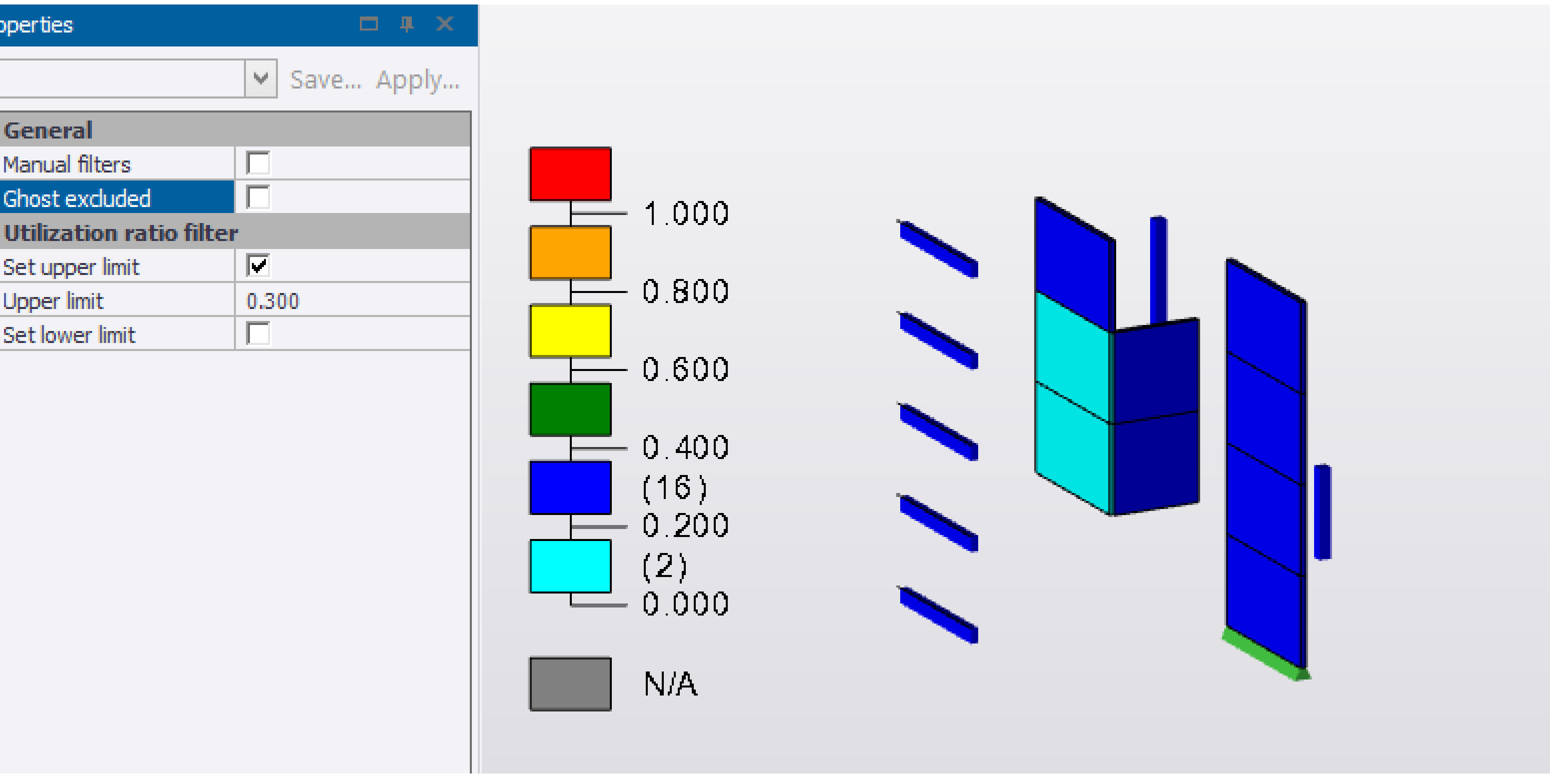
Working with the Utilization ratio filter

All entities that have been designed are assigned a utilization ratio. The **Utilization ratio filter** allows you to display only those entities within a given utilization range.

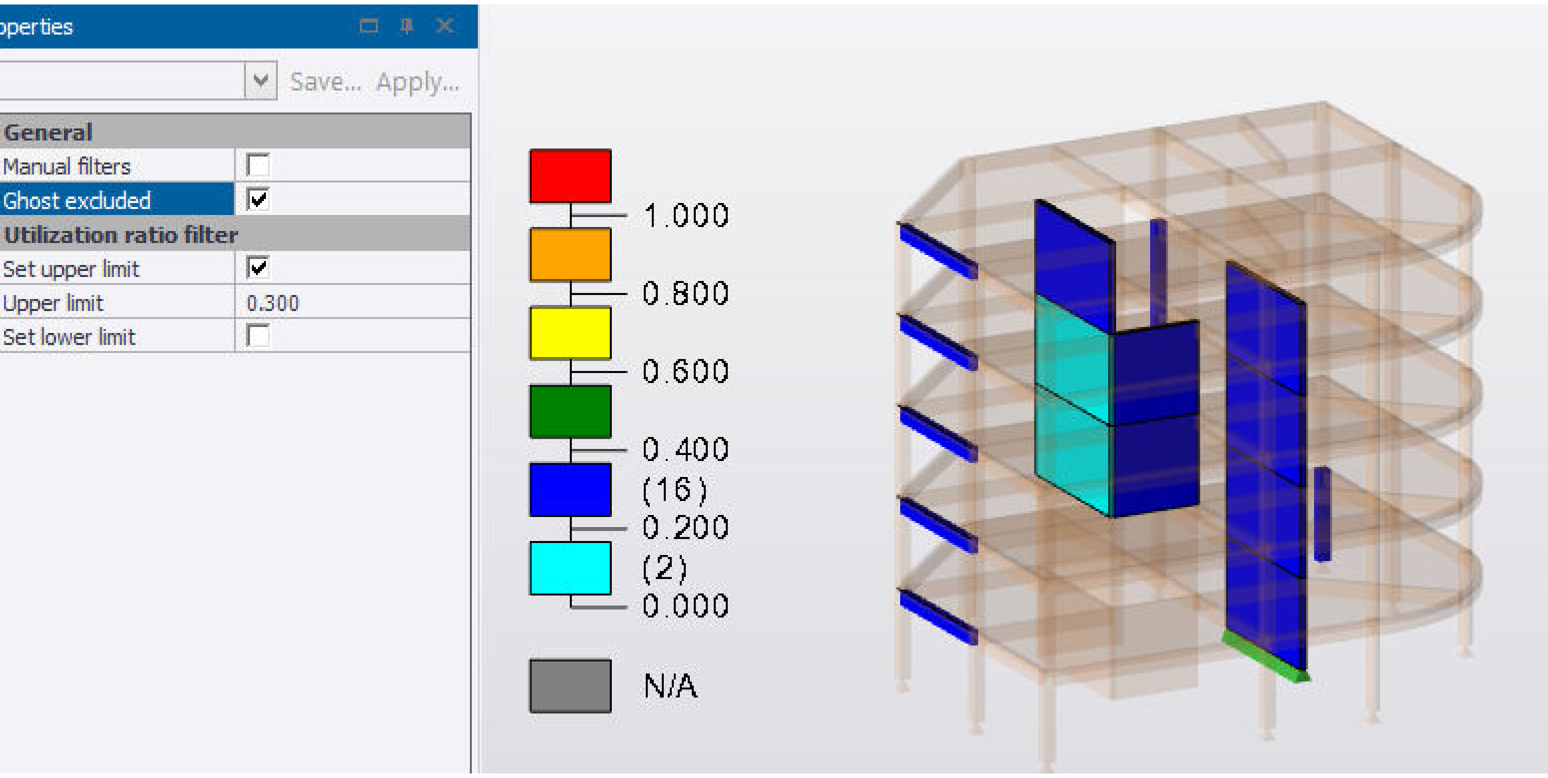
This filter is always active in the **Properties** window when **Ratio** is selected from the **Review** ribbon.



Initially entities with any ratio are displayed, but by selecting upper and/or lower limits, entities outside the defined range are hidden in the **Review View**.



By selecting **Ghost excluded** the filtered results can be displayed in relation to the rest of the model.

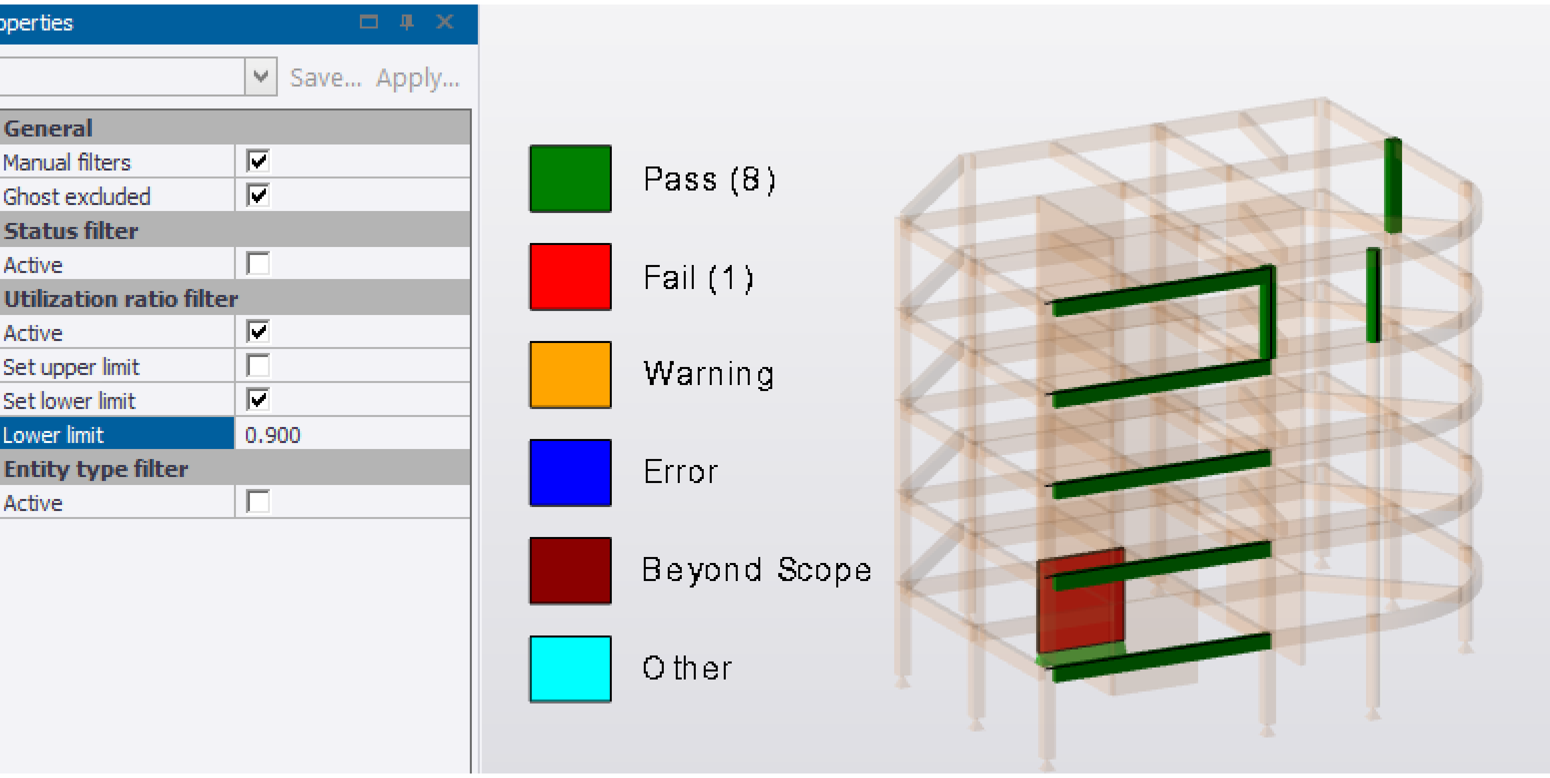


If required, you can also apply the **Utilization ratio filter** when investigating statuses. This is achieved as follows:

1. From the **Review** ribbon select **Status**
2. Select **Manual filters** in the **Properties** window.

   The **Utilization ratio filter** group is displayed.
3. Select **Active** and set the upper and/or lower limits by which to filter the display.

   The result is that statuses are then only displayed for those members within the specified utilization ratio limits.



Working with the Entity type filter

Unless they have been unselected in **Scene Content**, all relevant entity types are initially displayed when either **Status** or **Ratio** are selected from the **Review** ribbon.

You can use the **Entity type filter** to focus on specific types only, it is made active as follows:

1. Select **Manual filters** in the **Properties** window

   The **Entity type filter** group is displayed.

2. Select **Active**

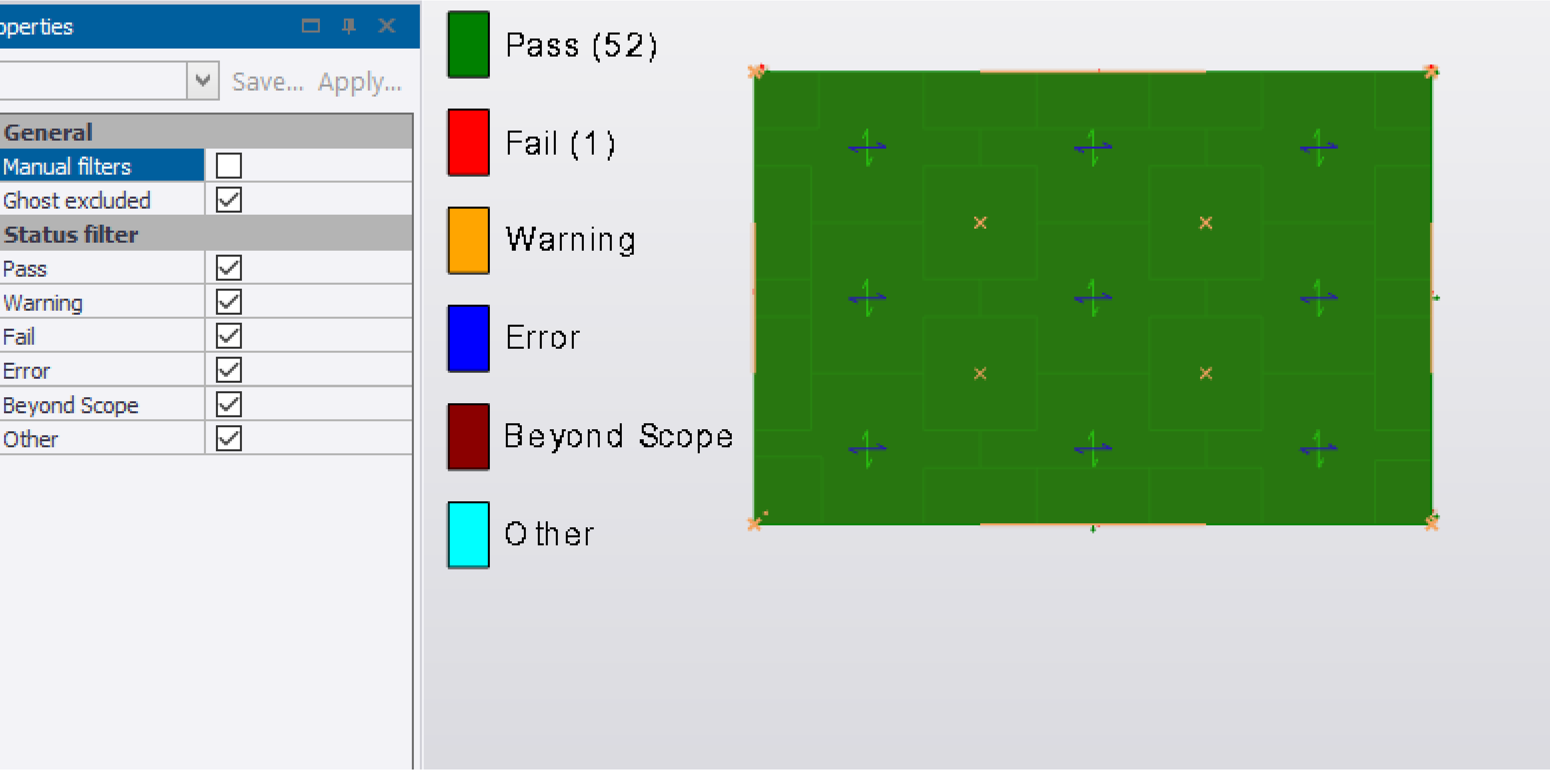
A list of designable entity types is displayed in the **Properties** window.

Deselecting an entity type from the list causes entities of that type to either be hidden in the **Review View**, or displayed as ghosted if **Ghost excluded** is selected.

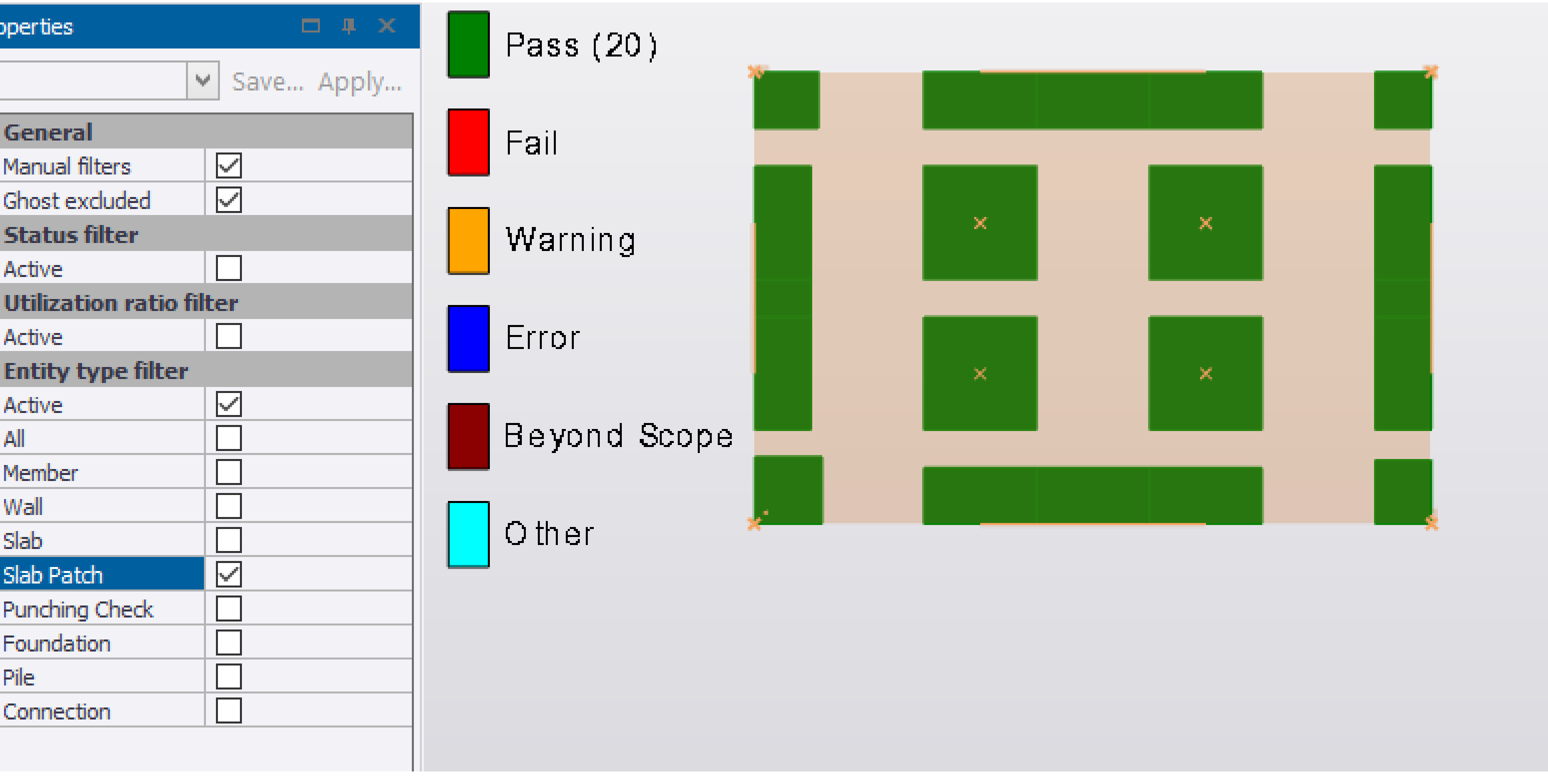
NOTE Clicking 'All' toggles between all entity types being selected, or unselected.

Example:

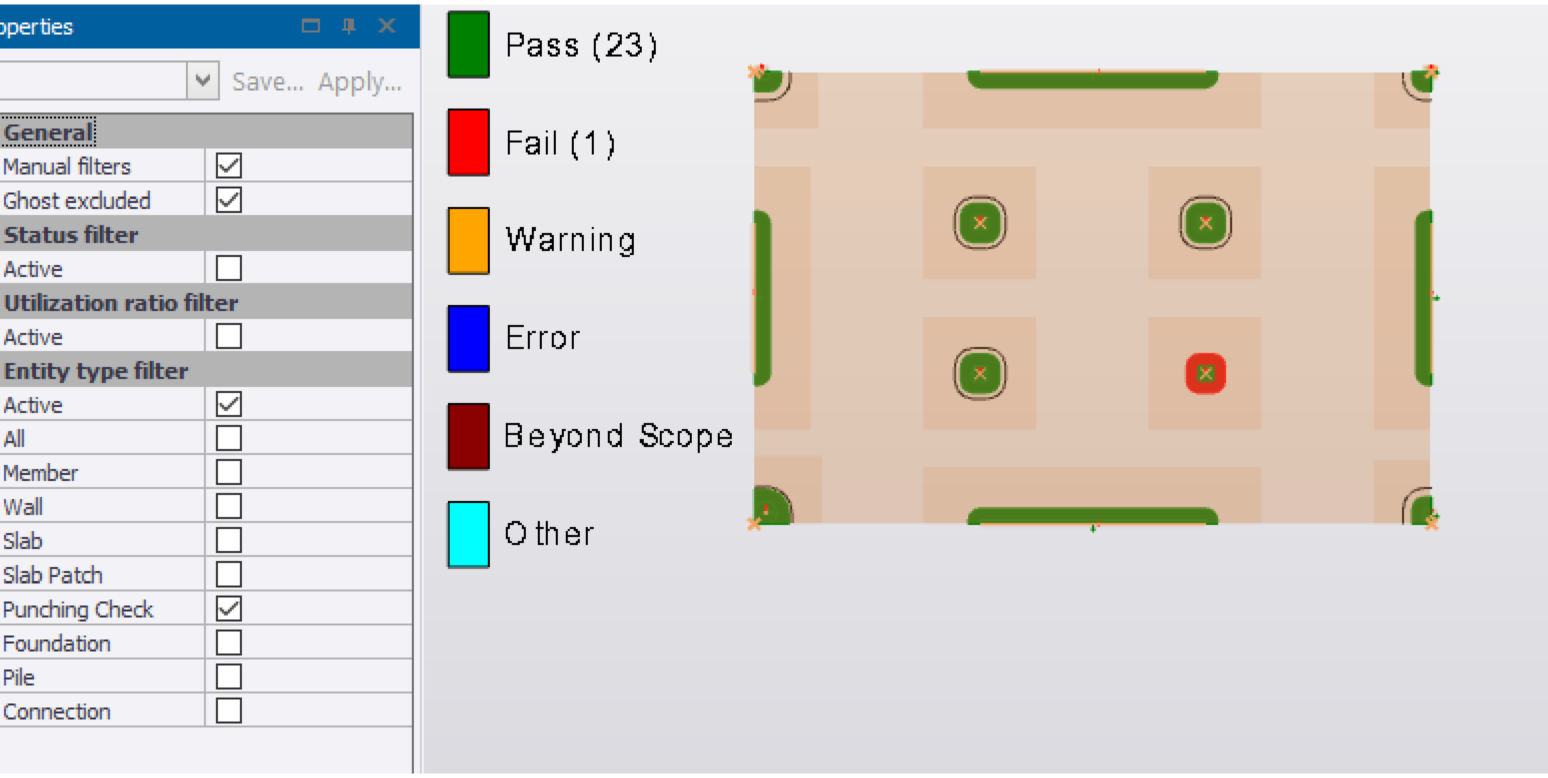
The below view shows slab status at a particular level. Initially, the **Entity type filter** is not active and as a result the **slab**, **slab patch**, and **punching check** statuses are all displayed making interpretation difficult. Note that a Fail is reported, but it is not immediately apparent where it is.



By activating the **Entity type filter** you can investigate the status for specific entity types. The below view is filtered for patches only, which are all passing.



The below view is filtered for punching checks only and clearly reveals where the Fail occurs.



9.2 Review model properties (show/alter state)

The **Show/Alter State** group of commands on the **Review** toolbar allow you to graphically review and modify various model properties. For each of the commands, entities are color coded in accordance with a legend, and the entity color changes to reflect any changes as they are made.

Many of the commands can only be accessed by selecting an appropriate attribute in the **Properties** window after first clicking Show/Alter State (page 725) on the toolbar, while some of the commands can only be accessed directly from the toolbar.

The **Show/Alter State** commands accessed directly from the toolbar are as follows:

- **Embodied Carbon** - which is used to:
  - graphically review and override the carbon factors (page 734) applied to the model,
  - and focus in on high carbon usage and inefficiency (page 758)
- Auto/Check Design (page 717) - to graphically review or modify autodesign settings of a member or slab
- Diaphragm On/Off - to graphically review or modify diaphragm settings

- Fixed/Pinned (page 719) - to graphically review or modify member end fixities
- BIM Status (page 720) - to graphically assess the BIM Status, and also to exclude members and panels from the import/export process
- Slab / Foundation Reinforcement - to graphically review and edit or rationalize panel, patch or foundation reinforcement
- Section/Material Grade (page 722) - to graphically copy section sizes, or, (by changing the Attribute), material grades (page 722)
- Copy Properties (page 723) - to copy a specified element parameter (such as web openings, shear connectors, or transverse reinforcement) from a source member to valid target members.
- Report Filter (page 723) - to graphically review and modify report filters
- Sub Structures (page 724) - to graphically review and modify sub structures
- Concrete Beam Flanges (page 724) - to graphically set whether flange are considered
- Column Splices (page 725) - to graphically review and modify splice positions within steel columns.
- Property Sets - to graphically review or apply property sets to members, slab items and roof panels.
- UDA - to graphically review and modify the values of user defined attributes that have been attached to the model.
- Show/Alter State (page 725) - to review and modify various model properties

Modify autodesign settings

You can graphically modify autodesign settings of members and slabs from a **Review View**.

To modify the autodesign settings of a member or slab, do the following:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, either:
    a. Click **Auto/Check Design**, or,
    b. Click **Show/Alter State** and choose **Autodesign** from the **Attribute** list.

    Members and slabs are color coded according to their auto design setting.
3. In the **Properties** window, select the appropriate mode.
4. Do one of the following:

- Click an individual entity to modify its auto design setting.
- Hold down the left mouse button and drag a box from left to right to modify the auto design setting of all members within the box.
- Hold down the left mouse button and drag a box from right to left to modify the auto design setting of all members within the box or cut by the box.

Review and modify diaphragm settings

The **Diaphragm On/Off** command on the **Review** tab allows you to select whether the slab items, roof panels, and nodes in the model are included

Review the diaphragm settings

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, either:
    a. Click **Diaphragm On/Off**, or
    b. Click **Show/Alter State** and choose **Diaphragm** from the **Attribute** list.

    Slab items, roofs and nodes are color coded according to their **Include in diaphragm** setting.

Modify the diaphragm settings of slab items or roofs

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, either:
    a. Click **Diaphragm On/Off**, or
    b. Click **Show/Alter State** and choose **Diaphragm** from the **Attribute** list.

    Slab items, roofs and nodes are color coded according to their **Include in diaphragm** setting.
3. In the **Properties** window, set **Entity type** to **Slab Item** or **Roof**.
4. Do one of the following:
    - Click an individual slab item or roof panel to switch between including and excluding the element from the diaphragm.
    - Hold down the left mouse button and drag a box from left to right to switch between including and excluding slab items and roof panels within the box from the diaphragm.

- Hold down the left mouse button and drag a box from right to left to switch between including and excluding slab items and roof panels within the box or cut by the box from the diaphragm.

Include or remove solver nodes from the diaphragm

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, either:
   a. Click **Diaphragm On/Off**, or
   b. Click **Show/Alter State** and choose **Diaphragm** from the **Attribute** list.

   Slab items, roofs and nodes are color coded according to their **Include in diaphragm** setting.
3. In the **Properties** window, set **Entity type** to **Solver Node**.
4. Do one of the following:
   - Click an individual slab item or roof panel to switch between including and excluding the element from the diaphragm.
   - Hold down the left mouse button and drag a box to switch between including and excluding the nodes within the box in diaphragm.

Modify end fixity

The **Fixed/Pinned** command allows you to graphically review and modify the end fixity of all members in the model.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Fixed/Pinned**.
3. Go to the **Properties** window.
4. In the **Attribute** list, ensure that **Fixed/Pinned** is selected.
5. With the **Mode** set as **Toggle**, do one of the following:
   - Click an individual member to switch its end fixity between the valid end fixity types.
   - Hold down the left mouse button and drag a box from left to right to alter the fixity settings of all members within the box.
   - Hold down the left mouse button and drag a box from right to left to alter the fixity settings of all members within the box or cut by the box.

6. Alternatively, with the **Mode** set as **Set On**, select the **FixityType** required and then click an individual member, or drag a box, as above to apply the chosen fixity.

NOTE If the end fixity of a member is labeled **Cantilever**, or **N/A**, the end fixity at end 1 of the element is different from the fixity at end 2. You can only edit a mixed end fixity by right-clicking on the member and editing its properties in the Properties dialog (page 63).

Modify BIM status

The **BIM Status** command allows you to graphically modify the BIM status and to exclude members and panels from the BIM import or export process.

To include or exclude members and panels from the BIM process, do the following:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **BIM Status**.
3. Do one of the following:
   - Click an individual member or panel once to exclude it or twice to re-include it.
   - Hold down the left mouse button and drag a box from left to right to modify the exclude setting of all members within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the exclude setting of all members within the box or cut by the box.

Copy or modify slab and foundation reinforcement

You can graphically review and modify the bar/mesh size and spacing applied in each layer and direction for slab items, mats, patches and isolated foundations. Tekla Structural Designer also allows you to copy reinforcement from one slab item, mat, patch or isolated foundation to another.

TIP If you open two **Review View**s side by side, you can then use the first one to modify the reinforcement while displaying the design status in the second one. This way, each change you make to the reinforcement immediately updates the design status.

Copy reinforcement

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, either:
   a. Click **Auto/Check Design**, or,
   b. Click **Show/Alter State** and choose **Autodesign** from the **Attribute** list.

   Members and slabs are color coded according to their auto design setting.
3. Ensure that auto design is set off for the slab items and patches that you want to modify.
4. On the **Review** tab, click **Slab / Foundation Reinforcement**.
5. Go to the **Properties** window.
6. Set **Entity type** to either **Slab Item**, **Slab Patch** or **Isolated Foundation**, depending on the reinforcement that you want to copy.
7. In **Reinforcement Direction**, select the direction that you want to modify.
8. In **Surface**, select the slab layer that you want to modify.
9. Click the slab item or patch whose reinforcement you want to copy.
10. Click the slab items or patches to which you want to copy the reinforcement.

    The slab reinforcement is copied to the selected slab items or patches.

    TIP To ensure that the updated reinforcement is sufficient, in the **Slab/Mat Design** group, click **Status**.

Modify reinforcement

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, either:
   a. Click **Auto/Check Design**, or,
   b. Click **Show/Alter State** and choose **Autodesign** from the **Attribute** list.

   Members and slabs are color coded according to their auto design setting.
3. Ensure that auto design is set off for the slab items and patches that you want to modify.
4. On the **Review** tab, click **Slab / Foundation Reinforcement**.
5. Go to the **Properties** window.
6. Set **Entity type** to either **Slab Item**, **Slab Patch** or **Isolated Foundation**, depending on the reinforcement that you want to modify.

7. In **Reinforcement Direction**, select the direction that you want to modify.
8. In **Surface**, select the slab layer that you want to modify.
9. If you want to modify bars, in **Bar Parameters**, select the properties that you want to modify.
10. In **Apply**, select the bar properties that you want to apply to the slab.
11. Click the slab item or patch whose reinforcement you want to update.

    TIP To ensure that the updated reinforcement is sufficient, in the **Slab/Mat Design** group, click **Status**.

Copy section sizes

If necessary, you can graphically copy the section size applied to a member to other members in the model. For more information, see the following paragraphs.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Section**.
5. Set **Mode** to **Copy**.
6. In **Material type**, select the desired material type.

    NOTE You can use a characteristic filter to reduce the elements displayed in the model.

7. In the model, click the element from which you want to copy a section size.
8. Click the elements to which you want to copy the section size.

    The section size is copied to the selected elements.

Copy material grades

If necessary, you can graphically review the material grade applied to the members in your model. Tekla Structural Designer also allows you to copy the grade of one member to another member with the same material type . For more information, see the following paragraphs.

RESTRICTION You can only copy material grades to and from steel, cold formed, cold rolled, concrete, or timber members.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Material Grade**.
5. In **Material Type**, select the desired material type.

   TIP If necessary, use a characteristic filter to reduce the elements displayed in the **Review View**.

6. Set **Mode** to **Copy**.
7. Click the element whose grade you want to copy.
8. Click the elements to which you want to copy the grade.

   The material grade is copied to the selected elements.

Copy properties

The **Copy Properties** command allows you to graphically copy a specified element parameter (such as web openings, shear connectors, or transverse reinforcement) from a source member to other valid target members. To copy properties from one member to another, see the following instructions.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Copy Properties**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select the parameter that you want to copy.
5. In the model, click the member from which you want to copy the parameter.
6. Do one of the following:
   - Click an individual member to which you want to copy the parameter.
   - Hold down the left mouse button and drag a box around multiple members to which you want to copy the parameter.

   Tekla Structural Designer copies the parameter to the selected members.

Review and modify member filters

The **Report Filter** command allows you to graphically review and modify report filters that you have defined for the members in your model. The

Report Filter command remains unavailable until you have defined a member filter.

TIP To define a report filter, do the following:

1. Go to the **Report** tab.
2. In the **Filters** group, click **Members**.
3. Define the member filter according to your needs.

---

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Report Filter**.
3. Do one of the following:
   - Click an individual member to switch between including and excluding the member from the selected member filter.
   - Hold down the left mouse button and drag a box from left to right to switch between including and excluding the members within the box from the selected member filter.
   - Hold down the left mouse button and drag a box from right to left to switch between including and excluding the members within the box or cut by the box from the selected member filter.

Review sub structures

The **Sub Structures** command allows you to graphically create, review and modify sub structures for modeling purposes. The command can be especially useful in large models, as individual sub structures can then be differentiated by color and worked on in separate views.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Sub Structures**.

   The elements in the model are color coded based on whether they belong to sub structures.

See also

Manage sub structures (page 921)

Review concrete beam flanges

The **Concrete Beam Flanges** command allows you to graphically review the flanges of concrete beams. See detailed instructions on how to review conrete beam flanges in the following paragraphs.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Concrete Beam Flanges**.

   Each concrete beam is color coded to indicate if its flanges are considered and flange widths determined.

Use of beam flanges

Review and modify column splice positions

The **Column Splices** command allows you to graphically review and modify splice positions within steel columns. For detailed instructions on how to review and modify splice positions, see the following paragraphs.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Column Splices**.

   All potential splice locations are color coded to indicate if they are on or off.
3. Do one of the following:
   - Click a potential splice location to alter its setting between on or off.
   - In a 2D view, hold down the left mouse button and drag a box to alter the setting of multiple splices simultaneously.

Review and apply property sets

The command provides a means to graphically review or apply property sets to entities.

- Review where property sets have been applied (page 914)
- Apply property sets to existing entities (page 913)

Copy or modify user-defined attributes

If necessary, you can graphically review and modify the values of attributes that have been attached to the model.

Show/alter state

The **Show/Alter State** commands allow you to efficiently review and modify various model properties. For each of the commands, entities are color coded in accordance with a legend, and the entity color changes to reflect any changes as they are made.

In the **Properties** window, you can use filters for most **Show/Alter State** commands. The filters allow you to control both the visibility of entities by their current setting and their adjustability by their type and characteristic. Inactive but visible entities are shaded a light transparent grey whose opacity you can also control, if necessary.

To access the **Show/Alter State** commands, do the following:

1. Open a **Review** view.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. Choose the command required from the **Attribute** list:

Option	Description
(page 727)	set single span members to be active/inactive
(page 723)	copy all properties between members
(page 728)	set concrete members as either cracked or uncracked
(page 717)	review or modify autodesign settings
(page 720)	modify BIM status, or exclude members/panels from BIM import/export
(page 731)	modify slenderness settings in each direction for concrete columns and walls
(page 732)	review and apply camber
(page 734)	mark beam and column ends as cantilevers
(page 734)	allows you to graphically review and override the carbon factors applied to the model
(page 735)	review and copy specified deflection limits between steel beams
(page 718)	select the slab items, roof panels, and nodes to be included in diaphragms
	review check status, or specify members to be checked
(page 737)	review and apply fire proofing
(page 719)	review and modify member end fixity
(page 739)	review and modify the gravity only setting of beams and columns
(page 739)	review and set imposed load reductions
(page 741)	review and set live load reductions (US headcode)
(page 722)	review and copy the material grade between members
(page 743)	set the composite beams to which an effective width override applies
(page 743)	modify assumed punching shear check positions
(page 744)	copy the connector layout from one beam to others
	review and modify lateral restraint settings of steel beams, columns, trusses and portal frames

(page 752)	review and modify rotational stiffness settings of beam ends
(page 722)	review and copy section sizes between members
(page 752)	review check status, or specify members to be checked
	review or modify SFRS type and direction
(page 754)	copy shear connector properties between composite beams
(page 754)	review the status and locations where SidePlate connections have been applied to steel beams
	review and copy specified size constraints for steel beams, columns, and braces
(page 755)	modify the stud auto layout setting of composite beams
	review check status, or specify members to be checked
(page 756)	modify the transverse reinforcement of composite beams
(page 725)	review and modify user defined attributes applied to members
	review and modify user defined utilization ratios
(page 758)	allows you to graphically focus in on high carbon usage and inefficiency.
(page 760)	copy web opening properties between members
(page 760)	copy westok opening properties between members
	review check status, specify members to be checked and set the limit
(page 761)	set the members/ancillaries/equipment to which open structure wind loading applies

TIP You can use **Ctrl** + **A** as a shortcut to quickly scroll the **Attribute** list.

Modify active / inactive settings

Initially all members are active. If a member is set inactive it does not participate in the analysis but any load applied to (or decomposed) to it is still accounted for.

Only single span beams, braces, analysis elements, purlins and rails have the potential to be inactive. They each therefore have an **Active** setting in the properties which defaults to 'on', but can be unchecked. In the **Review View** these members are color coded allowing the same setting to be toggled graphically.

NOTE Members cannot be inactive if:

- They are multi-span
- They support another member (active or inactive)

To modify the active setting of a member graphically do the following:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State** .
3. Go to the **Properties** window.
4. In the **Attribute** list select **Active**.

   Members are color coded according to their active setting.
5. In the **Properties** window, select the appropriate mode.
6. Do one of the following:
   - Click an individual entity to modify its active setting.
   - Hold down the left mouse button and drag a box from left to right to modify the active setting of all members within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the active setting of all members within the box or cut by the box.

Modify assumed cracked settings

You can graphically review and modify the assumed cracked setting of the concrete beams, columns and walls in a Review View using Show/Alter State - *Assume cracked*.

Review assumed cracked settings

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Review**
6. Set the **Entity type**

   The member color coding indicates the cracked, partially cracked or uncracked status.

   NOTE By hovering the cursor over an individual member or panel a tooltip is displayed. This reports the cracked status, and also the EI_{major}, EI_{minor} values. For partially cracked members/panels the percentage is also given.

Set as fully cracked or uncracked

1. Open a view and change the view regime (page 89) to **Review View**.

2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Set On**
6. Set **Assume cracked** to one of the following:
   - **Yes** - click or box around concrete member(s) to set to cracked.
   - **No**- click or box around concrete member(s) to set to uncracked.

   The member color coding updates to reflect the cracked or uncracked status.
7. Select **Entire member** to apply the new setting to all spans/stacks, or leave it unselected to update individual spans/stacks only.
8. Click or box around those concrete member(s) you want to set.

   The member color coding updates to reflect the new cracked or uncracked status.

Set as partially cracked

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Set On**
6. Set **Assume cracked** to **Partially**
7. Select **Set Percentage**
8. Enter the **Percentage** required.
9. Select **Entire member** to apply the new percentage to all spans/stacks, or leave it unselected to update individual spans/stacks only.
10. Click or box around those concrete member(s) you want to apply the percentage to.

    The member color coding updates to reflect the partially cracked status.

Toggle assumed cracked settings

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Toggle**

6. Click or box around those concrete member(s) you want to toggle.

   The assumed cracked status of the members toggles between cracked, partially cracked, and uncracked.

Review Wall Stress

To review/update the cracked status of all wall panels to be compatible with the stress levels for the loadcases/ combinations that have been analyzed, proceed as follows:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Assume cracked**.
5. Set the **[M]ode** to **Review Wall Stress**.
6. Check the yellow command prompt to confirm which analysis type is being used for the review, (this will be the type currently specified in **Design Settings>Analysis**).

Review Wall Stress Assume cracked (First-order linear): Select entity to Set compatible

7. Set the **Result type** to control which factors to use in combinations for the review.
   - **Strength** factors, or
   - **Service** factors
8. Set the **Stress type** to be considered.
   - **Max tension - Y**, or
   - **In-plane tension - Y**
9. Set the **Stress threshold** above which walls should crack. The existing cracked/uncracked status of each panel is displayed, along with an indiation of whether the stress threshold has been exceeded.
10. To make panel status compatible with the stress threshold for one or more panels:
    a. click an individual panel, or
    b. box around multiple panels, or
    c. to make all panels compatible, click Auto update then click the Auto update button

    Panel status is made compatible, i.e.

    - If uncracked and the stress threshold has been exceeded, the status is changed to cracked.

- If partially cracked and the stress threshold has been exceeded, the status is changed to cracked.
- If cracked and the stress threshold has not been exceeded, the status is changed to uncracked.

Review Wall Stress - limitations and assumptions

NOTE The following limitations and assumptions apply to **Review Wall Stress**

1. Only meshed wall panels are considered, mid-pier walls are excluded.
2. The analysis type specified in **Design Settings>Analysis** determines which set of analysis results are used for the review.
3. For the above analysis type, the loadcases and combinations considered are those from the **last performed building analysis of that type.**
    - This gives you the flexibility to review the stresses for a subset of loadcases/combinations if required.
    - By running Analyse All you can ensure all active gravity, lateral and seismic (not seismic RSA) combinations have been considered.
        - Stresses are available for seismic but not for seismic RSA combinations - these are beyond scope. (Review Wall Stress is more particularly aimed at determining the cracked status of panels under wind loading).
4. Chasedown results are **not** considered.
5. Axial load reductions are always taken into account.
6. The maximum stress value from all nodes for all analysed cases/ combinations is reported in a tooltip for each panel. This value is used to determine if the stress threshold has been exceeded.
    - The reinforcement content in the panel is not considered in determining the stress value.
7. The loading droplist in the status bar does not filter the status for individual cases/combinations.

Modify slenderness settings

The **Braced/Bracing** command provides a means to graphically review and modify the slenderness settings in each direction for concrete columns and walls.

To modify slenderness settings, do the following:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Braced/Bracing**.

5. Select the direction under consideration.

   Members are color coded according to their slenderness setting in the selected direction.

6. Do one of the following:

   - Click an individual column or wall to modify its slenderness setting between braced and bracing.
   - Hold down the left mouse button and drag a box from left to right to modify the slenderness setting of all concrete columns and walls within the box.
   - Hold down the left mouse button and drag a box from right to left to modify the slenderness setting of all concrete columns and walls within the box or cut by the box.

Review and set camber

The Show/Alter State **Camber** attribute allows you to graphically set and review the camber properties applied to steel beams.

Review camber

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Review**

   The member color coding indicates where and how camber has been applied to beams.

Set on

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Set On**
6. Set **Apply as** to **Value/Proportion of span/Proportion of deflection** as required.

   a. Enter the value or proportion information required to definer the camber.

7. Enter any beam length, or web thickness limits that you want to use to limit when the camber should be applied.
8. Select **Entire member** to apply the new setting to all spans, or leave it unselected to update individual spans only.
9. Click or box around those beam(s) you want to set.

   The member color coding updates to reflect the details that have been applied.

   - Members color coded as 'On' have the camber that is shown in the properties window.
   - Members color coded as 'Other details' have other camber properties. These are displayed in a tooltip when you hover the cursor over each member.

Set off

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Set Off**
6. Select **Entire member** to switch off camber for all spans, or leave it unselected to update individual spans only.
7. Click or box around those beam(s) for which you want to switch off camber.

   The member color coding updates to reflect the details that have been applied.

Toggle camber settings

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Toggle**
6. Click or box around those beam(s) you want to toggle.

   The camber setting toggles between on and off.

Copy camber settings

1. Open a view and change the view regime (page 89) to **Review View**.

2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Camber**.
5. Set the **[M]ode** to **Copy**
6. In the model, click the beam from which you want to copy the camber properties.
7. Click the beam to which you want to copy the camber properties. The camber properties are copied to the selected beam.
8. Continue copying the same camber properties by clicking the desired beams; or press **Esc** to select a new beam to copy the camber properties from; or press **Esc** twice to finish.

Apply cantilever ends

The **Cantilever end** command allows you to graphically apply cantilever ends where required to beams and columns.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Fixed/Pinned**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Cantilever end**.
5. Set **Mode** to **Set On**, **Set Off**, or **Toggle** as required.
6. Do one of the following:
   - Click an individual member end to set/unset a cantilever end as per the **Mode**.
   - Hold down the left mouse button and drag a box from left to right to alter the cantilever end settings of all members entirely within the box.
   - Hold down the left mouse button and drag a box from right to left to alter the cantilever end settings of all members within the box or cut by the box.

Review carbon factors

The Show/Alter State **Carbon factors** attribute allows you to graphically review and override the carbon factors applied to the model.

Review carbon factors

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Carbon factors**.

5. Set **Mode** to **Review**.
6. Set the **Carbon source category** to be reviewed

> **NOTE** Any entity in the model can have *multiple* carbon sources that relate to it.

Set override

> **NOTE** In general, rather than applying a carbon factor override you should consider adding it instead to the list of embodied carbon factors that can be applied.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Carbon factors**.
5. Set **Mode** to **Set override**.
6. Set the **Carbon source category** to be overridden.
7. Enter the required **Carbon factor**.
8. In the model, click the entities to which the override applies.
9. Continue applying overrides or press **Esc** to finish.

Remove override

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Carbon factors**.
5. Set **Mode** to **Remove override**.
6. Set the **Carbon source category** override to be removed.
7. In the model, click the entities from which the override is to be removed.
8. Continue removing overrides or press **Esc** to finish.

Review and copy deflection limits

You can graphically review specified deflection limits and copy the limits from one steel beam to others.

Review deflection limits

1. Open a view and change the view regime (page 89) to **Review View**.

2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Deflection Limits**.
5. Set **Mode** to **Review**.
6. Set **Load type** to the load type that you want to review.

Copy deflection limits

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Deflection Limits**.
5. Set **Mode** to **Copy**.
6. In the model, click the beam from which you want to copy the deflection limits.
7. Click the beam to which you want to copy the deflection limits.

   The deflection limits are copied to the selected beam.
8. Continue copying the limits by clicking the desired beams or press **Esc** to finish.

Review and modify drift checks

You can graphically review drift check results and set the members to be checked for columns and walls.

Review drift checks

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Drift check**.
5. Set **Mode** to **Review**.

Set drift checks on or off

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Drift check**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.

6. In the model, click the column or wall for which you want to set the check.
    a. If **Mode** is **Set On** - the check is switched on
    b. If **Mode** is **Set Off** - the check is switched off
    c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Apply fire proofing

The Show/Alter State **Fire proofing** attribute allows you to graphically set and review the fire proofing applied to 1D member types.

See also

Fire proofing (page 950)

Review fire proofing

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Fire proofing**.
5. Set the **[M]ode** to **Review**

    The member color coding indicates the fire proofing status. If working to Eurocodes the beams that have been set to have a fire resistance check are also indicated.

    NOTE By hovering the cursor over an individual member or panel a tooltip is displayed. This reports the cracked status, and also the EI_{major}, EI_{minor} values. For partially cracked members/panels the percentage is also given.

Set as protected

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Fire proofing**.
5. Set the **[M]ode** to **Set On**
6. Set **Checks** to **Protected**
7. Select which of the protection properties (exposure, shape, thickness & density) you want to apply proofing of the

8. Select **Entire member** to apply the new setting to all spans/stacks, or leave it unselected to update individual spans/stacks only.
9. Click or box around those 1D member(s) you want to set.

   The member color coding updates to reflect the details that have been applied.

   - Members color coded as 'On' have the fire proofing that is shown in the properties window.
   - Members color coded as 'Other details' have other fire proofing properties. These are displayed in a tooltip when you hover the cursor over each member.

Set check for fire resistance (Eurocode only)

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Fire proofing**.
5. Set the **[M]ode** to **Set On**
6. Set **Checks** to **Check for fire resistance**
7. Select **Set exposure** to specify whether the member is exposed on all sides or 3 sides only.
8. Select **Entire member** to apply the new setting to all spans, or leave it unselected to update individual spans only.
9. Click or box around those beam(s) you want to set.

   The member color coding updates to reflect the details that have been applied.

   - Members color coded as 'On' have the fire resistance that is shown in the properties window.
   - Members color coded as 'Other details' have other fire resistance properties. These are displayed in a tooltip when you hover the cursor over each member.

Toggle fire proofing settings

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Fire proofing**.
5. Set the **[M]ode** to **Toggle**

6. If working to Eurocodes, choose whether you want to toggle the protection or fire check settings.
7. Click or box around those 1D member(s) you want to toggle.

   The member properties toggle between on and off.

Modify gravity only settings

If necessary, you can review and modify the gravity only setting of the beams and columns in your model. For more information, see the following paragraphs.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Gravity only**.

   All members are color coded to indicate their gravity only setting.
5. Do one of the following:
   - Click a steel beam or column to set its gravity only setting on or off.
   - Hold down the left mouse button and drag a box from left to right to set on or off the gravity only setting of all steel beams and columns within the box.
   - Hold down the left mouse button and drag a box from right to left to set on or off the gravity only setting of all steel beams and columns within the box or cut by the box.

Review and set imposed load reduction

By selecting the **Imposed load reduction** attribute in **Show/Alter State** you can graphically review and set imposed load reduction parameters.

TIP Graphical editing will typically be more effective than the alternative of opening the **Properties** dialog box for each member individually.

Review load reduction attributes

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Imposed load reduction**
5. Set **Mode** to **Review**.
6. Set the **Load reduction attribute** as required:

- Applied imposed load reduction factor
- Reduce imposed loads by
- Count the floor as being supported*
- Number of floors carried*

* Not applicable if working to the Australian headcode.

The view updates to display the selected load reduction attribute information.

TIP Hover the cursor over a member to see all the above attributes in a tooltip.

Set 'Reduce imposed loads by' percentage for beams/slabs

To graphically reduce imposed loads on beams and slabs by a percentage proceed as follows:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Imposed load reduction**
5. Set **Mode** to **Set**.
6. Set the **Load reduction attribute** to **Reduce imposed loads by**.
7. Enter the percentage.
8. Set the **Entity type** that you want to modify.
9. Click or box around entities in the Review view to apply the new percentage. The view updates to display the revised reduction percentages.

TIP Hover the cursor over entities that don't have the currently selected percentage reduction to see their attributes in a tooltip.

Set 'Count floor as being supported' for columns/walls

NOTE Not applicable if working to the Australian headcode.

To graphically set the 'Count floor as being supported' property at specific levels for columns and walls proceed as follows:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Imposed load reduction**

5. Set **Mode** to **Set**.
6. Set the **Load reduction attribute** to **Count the floor as being supported**.
7. Check or uncheck the **Count the floor as being supported** checkbox as required.
8. Click on a column or wall just above/below a specific floor to set/unset the **Count the floor as being supported** property at that level for the selected column or wall.

TIP Set the **Entity type** to **Any**, **Column**, or **Wall** to work on specific entities only.

Set 'Assume extra floors supported' for columns/walls

NOTE Not applicable if working to the Australian headcode.

To graphically set the 'Assume extra floors supported' property for columns and walls proceed as follows:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Imposed load reduction**
5. Set **Mode** to **Set**.
6. Set the **Load reduction attribute** to **Assume extra floors supported**.
7. Enter the **Assume extra floors supported** number.
8. Click or box around columns and walls to apply the new value.

Review and set live load reduction

By selecting the **Live load reduction** attribute in **Show/Alter State** you can graphically review and set live load reduction parameters.

TIP Graphical editing will typically be more effective than the alternative of opening the **Properties** dialog box for each member individually.

Review load reduction attributes

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Live load reduction**

5. Set **Mode** to **Review**.
6. Set the **Load reduction attribute** as required:
   - Applied live load reduction factor
   - KLL
   - Reduce live loads by
7. If reviewing the applied live load reduction factor, choose the loadcase type to review:
   - Live
   - Roof Live

   The view updates to display the selected load reduction attribute information.

TIP Hover the cursor over a member to see all the above attributes in a tooltip.

Set 'Reduce live loads by' percentage for slabs

To graphically reduce live loads on slabs by a percentage proceed as follows:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Live load reduction**
5. Set **Mode** to **Set**.
6. Set the **Load reduction attribute** to **Reduce live loads by**.
7. Enter the percentage.
8. Click or box around slabs in the Review view to apply the new percentage. The view updates to display the revised reduction percentages.

TIP Hover the cursor over slabs that don't have the currently selected percentage reduction to see their attributes in a tooltip.

Set KLL factor

To graphically set the KLL property proceed as follows:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Live load reduction**
5. Set **Mode** to **Set**.

6. Set the **Load reduction attribute** to **KLL**.
7. Enter the KLL value.
8. Uncheck **Entire member** if you only want the value to apply to individual stacks/spans.
9. Click or box around entities to apply the new value.

TIP Set the **Entity type** to **Beam**, **Column**, or **Wall** to work on specific entities only.

Override effective width

The Show/Alter State **Override Effective Width** attribute allows you to graphically set the composite beams to which an effective width override applies.

NOTE For the width to be overridden the **Update effective width prior to design check** setting must be selected in **Design > Settings > Steel > Composite beam**.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, ensure that **Override Effective Width** is selected.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the composite beams to which an effective width override will apply.
    a. If **Mode** is **Set On** - the setting is switched on
    b. If **Mode** is **Set Off** - the setting is switched off
    c. If **Mode** is **Toggle** - the setting toggles between on and off
7. Continue applying the setting or press **Esc** to finish.

Modify punching shear check position

If necessary, you can graphically modify the assumed punching shear check position. This allows you to determine the loaded perimeter when the perimeter is close to a free edge.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Punch check position**.

5. To change the punching shear check position, do one of the following:

To	Do this
Switch the punching shear check position between Edge Z and Internal	• Click the Z edge of a punching check item.
Switch the punching shear check position between Edge Y and Internal	• Click the Y edge of a punching check item.
Switch the punching shear check position between Corner, Edge Z, Edge Y and Internal	• Click a corner of a punching check item.

NOTE Clicking an internal punching check item does not change the punching shear check position.

TIP You can also box around multiple punching check item to switch their position. Do one of the following:

- Hold down the left mouse button and drag a box from left to right to switch the position setting of all punch check items within the box.
- Hold down the left mouse button and drag a box from right to left to switch the position setting of all punch check items within the box or cut by the box.

Copy quick connector layout

You can graphically copy the connector layout from one beam to another. For more information, see the following paragraphs.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Quick Connector Layout**.
5. Click on the beam with the layout that you want to copy (must be valid).

   The composite beams in the model are color coded to indicate the source and valid targets.
6. To select the beams to apply the layout to, do one of the following:
   - Click an individual valid target beam.
   - Hold down the left mouse button and drag a box from left to right to totally enclose the target beams.
   - Hold down the left mouse button and drag a box from right to left to select target beams within the box or cut by the box.

   The connector layout is copied from the source to the target beam(s).

Review and modify restraints

By selecting the **Restraints** attribute in **Show/Alter State** you can graphically review and modify continuous and discrete restraints for beams, columns, trusses and portal frames.

TIP Graphical editing of restraints will typically be more effective than the alternative of opening the **Properties** dialog box for each member individually.

Review restraints

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Restraints**.
5. Set **Mode** to **Review**. For details of the restraint type choices etc. that are presented in Review mode, see the **Options overview** topics below.
6. Set the **Continuous** box as required:
   - Checked: shows continuous restraints, each sub-member being colour coded in accordance with the legend.
   - Unchecked: shows discrete restraints, each internal node at which other members connect being colour coded in accordance with the legend.
7. Set the Restraint Type that you want to review.
8. Set the Entity type that you want to review.

Toggle restraints

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Restraints**.
5. Set **Mode** to **Toggle**. For details of the restraint type choices etc. that are presented in Toggle mode, see the **Options overview** topics below.
6. Set the **Entity type** that you want to modify.
7. Leave the **Continuous** box checked to set continuous restraints along sub-members, or uncheck it to set discrete restraints at those points where other members connect.
8. Set the **Restraint Type** as required.
9. In the model:
   - click the member at the location where want to toggle the restraint.

- drag a box around multiple members to toggle the restraint at all valid locations within the box.

  The type of restraint at the selected location(s) changes to the next restraint type shown in the legend. (Simply click again as necessary until the desired restraint type has been applied.)

10. Continue toggling the restraints by clicking the desired members or press **Esc** to finish.

Set restraints

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Restraints**.
5. Set **Mode** to **Set**. For details of the restraint type choices etc. that are presented in Set mode, see the **Options overview** topics below.
6. Set the **Entity type** that you want to modify.
7. Leave the **Continuous** box checked to set continuous restraints along sub-members, or uncheck it to set discrete restraints at those points where other members connect.
8. Set the **Restraint Type** as required.
9. Set the **Restraint** as required.
10. In the model:
    - click the member at the location where want to set the restraint.
    - drag a box around multiple members to set the same restraint at all valid locations within the box.
11. Continue setting the checks by clicking/boxing the desired members or press **Esc** to finish.

Options overview (US headcode)

The restraint options displayed in the **Properties** window depend on the **Entity type** filter. When this is set to either **Beam**, **Column**, or **Truss Top/ Bottom Member**, the options are:

Command or option	Description
[M]ode	• **Review** - sub-members/discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking sub-members/discrete points of restraint toggles the applicable restraints.

Command or option	Description
	• **Set** - clicking sub-members/discrete points of restraint sets the selected restraint.
Continuous	• Checked - enables the review/setting of **sub-member continuous restraints** • Unchecked - enables the review/setting of **discrete restraints** at those points where other members connect.
Restraint Type	The restraint type determines which page(s) of properties are being edited. • **LTB** - enables the review/setting of the check-boxes located on the **Restraints (LTB)** property page. • **Compression** - enables the review/setting of the check-boxes located on the **Restraints (Comp)** property page. • **LTB & Compression Out of Plane** - enables the simultaneous review/setting of check-boxes on both the the **Restraints (LTB)** and **Restraints (Comp)** property pages.
Restraint	When in **Set** Mode, this setting determines which check-boxes on the selected restraint property page are checked and un-checked.
	For **LTB** restraint types (and Entity type Beam or Truss): • **Top** - Top flange check-box is checked, Bottom flange check-box is unchecked. • **Bottom** - Top flange check-box is unchecked, Bottom flange check-box is checked. • **Top and Bottom** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **LTB** restraint types (and Entity Type Column): • **Face A** - Face A check-box is checked, Face C check-box is unchecked. • **Face C** - Face A check-box is unchecked, Face C check-box is checked. • **Face A & C** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **Compression** restraint types: • **Out of Plane** - Out of plane check-box is checked, In plane check-box is unchecked.

Command or option	Description
	• **In Plane** - In plane check-box is checked, Out of plane check-box is unchecked. • **Out of Plane & In Plane** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **LTB & Compression Out of Plane** restraint type: • **Top & Bottom & Out of Plane** - Top flange, Bottom flange, and Out of Plane are checked. • **Top & Out of Plane** - Top flange and Out of Plane are checked, Bottom flange is unchecked. • **Bottom & Out of Plane** - Bottom flange and Out of Plane are checked, Top flange is unchecked. • **Unrestrained** - Top flange, Bottom flange, and Out of Plane are checked. **NOTE** For each of the above, the In Plane check-box is unaffected, retaining its existing setting.

Options overview (other head codes)

The restraint options displayed in the **Properties** window depend on the **Entity type** filter. When this is set to either **Beam**, **Column**, or **Truss Top/ Bottom Member**, the options are:

Command or option	Description
[M]ode	• **Review** - sub-members/discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking sub-members/discrete points of restraint toggles the applicable restraints. • **Set** - clicking sub-members/discrete points of restraint sets the selected restraint.
Continuous	• Checked - enables the review/setting of **sub-member continuous restraints** • Unchecked - enables the review/setting of **discrete restraints** at those points where other members connect.
Restraint Type	The restraint type determines which page(s) of properties are being edited.

Command or option	Description
	• **Lateral** - enables the review/setting of the check-boxes located on the **Lateral restraints** property page. • **Strut** - enables the review/setting of the check-boxes located on the **Strut restraints** property page • **Lateral & Strut Minor** - enables the simultaneous review/setting of check-boxes on both the the **Lateral restraints** and **Strut restraints** property pages.
Restraint	When in **Set** Mode, this setting determines which check-boxes on the selected restraint property page are checked and un-checked.
	For **Lateral** restraint types (and Entity type Beam or Truss): • **Top** - Top flange check-box is checked, Bottom flange check-box is unchecked. • **Bottom** - Top flange check-box is unchecked, Bottom flange check-box is checked. • **Top and Bottom** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **Lateral** restraint types (and Entity Type Column): • **Face A** - Face A check-box is checked, Face C check-box is unchecked. • **Face C** - Face A check-box is unchecked, Face C check-box is checked. • **Face A & C** - both check-boxes are checked. • **Unrestrained** - both check-boxes are unchecked.
	For **Strut** restraint types: • Major - Major check-box is checked, Minor check-box is unchecked. • Minor - Minor check-box is checked, Major check-box is unchecked. • Major and Minor - both check-boxes are checked. • Unrestrained - both check-boxes are unchecked.
	For **Lateral & Strut Minor** restraint type: • **Top & Bottom & Minor** - Top flange, Bottom flange, and Minor are checked. • **Top & Minor** - Top flange and Minor are checked, Bottom flange is unchecked.

Command or option	Description
	• **Bottom & Minor** - Bottom flange and Minor are checked, Top flange is unchecked. • **Unrestrained** - Top flange, Bottom flange, and Minor are checked. **NOTE** For each of the above, the Major check-box is unaffected, retaining its existing setting.

Portal Frame options (US headcode)

The restraint options displayed in the **Properties** window when the **Entity type** filter is set to **Portal Frame** are:

Command or option	Description
[M]ode	• **Review** - discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking discrete points of restraint toggles the applicable restraints. • **Set** - clicking discrete points of restraint sets the selected restraint.
Restraint	When in **Set** Mode, this setting determines which check-boxes on the restraint property pages are checked and un-checked. Rafters: • **Torsional** - Top flange, Bottom flange, and Out of Plane are checked. • **Outer** - Top flange is checked, Bottom flange is unchecked. • **Inner** - Top flange is unchecked, Bottom flange is checked. • **Unrestrained** - Top flange, Bottom flange, and Out of Plane are unchecked. Columns: • **Torsional** - Face A, Face C, and Out of Plane are checked. • **Outer** - Face A is checked, Face C is unchecked. • **Inner** - Face A is unchecked, Face C is checked.

Command or option	Description
	• **Unrestrained** - Face A, Face C, and Out of Plane are unchecked. **NOTE** For each of the above, any checkbox not specifically mentioned retains its existing setting.

Portal frames options (other head codes)

The restraint options displayed in the **Properties** window when the **Entity type** filter is set to **Portal Frame** are:

Command or option	Description
[M]ode	• **Review** - discrete points of restraint are color coded to indicate their restraint settings • **Toggle** - clicking discrete points of restraint toggles the applicable restraints. • **Set** - clicking discrete points of restraint sets the selected restraint.
Restraint	When in **Set** Mode, this setting determines which check-boxes on the restraint property pages are checked and un-checked. Rafters: • **Torsional** - Top flange, Bottom flange, and Minor are checked. • **Outer** - Top flange is checked, Bottom flange is unchecked. • **Inner** - Top flange is unchecked, Bottom flange is checked. • **Unrestrained** - Top flange, Bottom flange, and Minor are unchecked. Columns: • Torsional - Face A, Face C, and Minor are checked. • Outer - Face A is checked, Face C is unchecked. • Inner - Face A is unchecked, Face C is checked.

Command or option	Description
	• Unrestrained - Face A, Face C, and Minor are unchecked. **NOTE** For each of the above, any checkbox not specifically mentioned retains its existing setting.

Related video

Show/Alter State Restraints

Apply rotational stiffness to a beam end

The **Rotational stiffness** command allows you to graphically apply partially fixity/linear springs to the ends of beams.

Related video

Automatic modelling of partial fixity

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Fixed/Pinned**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Rotational stiffness**.
5. Set **Mode** to **Set On**.
6. Set **Direction** to **Major**, or **Minor** as required.
7. Set **Rotational stiffness** to **Partially fixed**, or **Spring linear** as required.
8. If **Partially fixed**, set the percentage fixity value; or if **Spring linear** set the spring stiffness value as required.
9. To apply the new setting do one of the following:
    - Click an individual beam end.
    - Hold down the left mouse button and drag a box from left to right to apply to all beam ends entirely within the box.
    - Hold down the left mouse button and drag a box from right to left to apply to both ends of all beams within the box or cut by the box.

Review and modify seismic drift

You can graphically review seismic drift check results and set the members to be checked for columns and walls.

Review seismic drift

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Seismic drift**.
5. Set **Mode** to **Review**.

Set seismic drift checks on or off

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Seismic drift**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.
   a. If **Mode** is **Set On** - the check is switched on
   b. If **Mode** is **Set Off** - the check is switched off
   c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Review and modify SFRS settings

If necessary, you can graphically review and modify the the types and directions of seismic force resisting systems (SFRS) in the model. For more information, see the following paragraphs.

Review SFRS type and direction settings

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **SFRS**.

   Members and walls are color coded according to their SFRS type and direction settings.

Modify SFRS type & direction settings

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.

4. In the **Attribute** list, select **SFRS**.

   Members and walls are color coded according to their SFRS direction setting.

5. Set **Mode** to one of the following:

   - **Update Type & Direction**: includes a member in the selected SFRS type & direction.
   - **Remove from SFRS**: removes the member from SFRS.

6. Click the desired member.

   Depending on the set mode, Tekla Structural Designer includes the member in the SFRS type & direction, or removes the member from SFRS.

Copy shear connectors

If necessary, you can copy the composite beam shear connector properties from a source beam to other composite beams. Do the following:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Shear Connectors**.
5. In the model, click the composite beam from which you want to copy the shear connector properties.
6. Click the composite beams to which you want to copy the shear connect properties.

   The properties are copied to the selected beams.

Modify SidePlates

The **SidePlates** command allows you to graphically review and modify the SidePlate connection status.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **SidePlates**.
3. Do one of the following:

   - Click on the first half of a beam to reset end 1, the second half to reset end 2 and the middle to reset both ends simultaneously.
   - Hold down the left mouse button and drag a box from right to left to reset all the beam ends within the box or cut by the box.

See also

Create and size SidePlate connections (page 645)

Review and copy size constraints

If necessary, you can graphically review specified size constraints for steel beams, columns, and braces, and copy the constraints from one steel member to others. For more information, see the following paragraphs.

Review size constraints

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Size Constraints**.
5. Set **Mode** to **Review**.
6. In **Size constraint**, select the size constraint that you want to review.

Copy size constraints

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Size Constraints**.
5. Set **Mode** to **Copy**.

   TIP Select the **By constraint type** option if you want to only copy an individual constraint type.

6. In the model, click the member from which you want to copy the size constraints.
7. Click the member to which you want to copy the size constraints.
8. Continue copying the size constraints by clicking the desired members or press **Esc** to finish.

Modify stud auto layout

If necessary, you can graphically modify the stud auto layout setting of composite beams in the model. For more information, see the following paragraphs.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.

4. In the **Attribute** list, select **Stud auto layout**.

   The composite beams in the model are color coded according to their auto layout setting.

5. Do one of the following:

   - Click an individual composite beam to alter its auto layout setting.
   - Hold down the left mouse button and drag a box from left to right to alter the auto layout setting of all composite beams within the box.
   - Hold down the left mouse button and drag a box from right to left to alter the auto design setting of all composite beams within the box or cut by the box.

Review and modify sway checks

You can graphically review sway check results and set the members to be checked for columns and walls.

Review sway checks

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Sway check**.
5. Set **Mode** to **Review**.

Set sway checks on or off

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Sway check**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.

   a. If **Mode** is **Set On** - the check is switched on

   b. If **Mode** is **Set Off** - the check is switched off

   c. If **Mode** is **Toggle** - the check toggles between on and off

7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Copy transverse reinforcement

If necessary, you can copy beam transverse reinforcement from one composite beam to others in your model. Do the following:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Transverse reinforcement**.

   The composite beams in the model are color coded based on whether you can copy transverse reinforcement properties from them.
5. Click the composite beam from which you want to copy transverse reinforcement properties.
6. Click the composite beams to which you want to copy the transverse reinforcement properties.

   The properties are copied to the selected beams.

Review and modify user defined U/R

From a **Review View** you can graphically review user defined utilization ratios, set the members to which they are applied, and set whether they apply to autodesign alone, or both autodesign and check.

Review user defined U/R

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.

Apply a user defined U/R for autodesign only

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.
5. Set **Mode** to **Apply to Autodesign**.
6. Select **Set limit ratio**.
7. Specify the required **Ratio limit**
8. If required, set **Entity type** and **Material** to make the command easier to apply.

9. In the model, click those column/wall stacks, beam spans, or objects for which you want to apply the user defined U/R.
10. Continue setting the limit by clicking the desired columns and walls or press **Esc** to finish.

Apply a user defined U/R for autodesign and check

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.
5. Set **Mode** to **Apply to Autodesign+Check**.
6. Select **Set limit ratio**.
7. Specify the required **Ratio limit**
8. If required, set **Entity type** and **Material** to make the command easier to apply.
9. In the model, click those column/wall stacks, beam spans, or objects for which you want to apply the user defined U/R.
10. Continue setting the limit by clicking the desired columns and walls or press **Esc** to finish.

Turn off user defined U/R

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **User defined U/R**.
5. Set **Mode** to **Set Off**.
6. If required, set **Entity type** and **Material** to make the command easier to apply.
7. In the model, click those column/wall stacks, beam spans, or objects for which you want to turn off user defined U/R.
8. Continue by clicking the desired stacks/spans/objects or press **Esc** to finish.

See also

Apply user defined utilization ratios (page 613)

Review utilization and embodied carbon

The Show/Alter State **Utilization and embodied carbon** attribute allows you to graphically focus in on high carbon usage and inefficiency.

Review utilization and embodied carbon

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Utilization and embodied carbon**.
5. Set the **Entity type** and **Variant** to review.
6. Set the **Carbon source category** to be review.

   NOTE Any entity in the model can have *multiple* carbon sources that relate to it, for example:

   - A concrete beam has concrete and reinforcement
   - A composite beam has steel, shear studs, and potentially transverse shear reinforcement

   For example

Set filters to focus on where the carbon impact might be reduced

1. Set the utilization filter as required.
   - Value below - only entities with utilization below the specified value are displayed
   - Value above - only entities with utilization above the specified value are displayed
   - Lowest X % - only the lowest specified percentage of utilizations are displayed
   - Highest X % - only the highest specified percentage of utilizations are displayed
2. Set the embodied carbon filter as required.
   - Value below - only entities with a carbon usage below the specified value are displayed
   - Value above - only entities with a carbon usage above the specified value are displayed
   - Lowest X % - only the lowest specified percentage of carbon usages are displayed
   - Highest X % - only the highest specified percentage of carbon usages are displayed

Typically, filters would be applied to detect where *high* carbon usage occurs in combination with *low* utilization, as this informs where the carbon impact could be reduced.

For example: Highest 50% embodied carbon combined with lowest 50% utilization.

Copy web openings

If necessary, you can copy web opening properties from one member to others. Do the following:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Web Openings**.

   The members in the model are color coded based on whether you can copy web opening properties from them.
5. In the model, click the member from which you want to copy web opening properties.
6. Click the members to which you want to copy the web opening properties.

   The properties are copied to the selected members.

Copy westok openings

If necessary, you can copy westok opening properties from one member in your model to others. Do the following:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** tab, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Westok Openings**.

   The members in the model are color coded based on whether you can copy westok opening properties from them.
5. In the model, click the member from which you want to copy westok opening properties.
6. Click the members to which you want to copy the westok opening properties.

   The properties are copied to the selected members.

Review and modify wind drift checks

You can graphically review wind drift check results, set the members to be checked and set the check limits for columns and walls.

Review wind drift checks

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Wind drift**.
5. Set **Mode** to **Review**.

Set wind drift checks on or off

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Wind drift**.
5. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
6. In the model, click the column or wall for which you want to set the check.
    a. If **Mode** is **Set On** - the check is switched on
    b. If **Mode** is **Set Off -** the check is switched off
    c. If **Mode** is **Toggle** - the check toggles between on and off
7. Continue setting the checks by clicking the desired columns and walls or press **Esc** to finish.

Set wind drift checks on and set the ratio limit

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, select **Wind drift**.
5. Set **Mode** to **Set On**.
6. Select **Set limit ratio**.
7. Specify the required **Wind drift ratio limit**
8. In the model, click the column or wall for which you want to set the limit.
9. Continue setting the limit by clicking the desired columns and walls or press **Esc** to finish.

Modify wind loading

The Show/Alter State **Wind Loading** attribute allows you to graphically set the members/ancillaries/equipment to which open structure wind loading applies.

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** toolbar, click **Show/Alter State**.
3. Go to the **Properties** window.
4. In the **Attribute** list, ensure that **Wind Loading** is selected.
5. If you want to apply the setting to individual stacks/spans within members as opposed to the entire member, unselect **Entire member**.
6. Set **Mode** to **Set On** , or **Set Off** or **Toggle**.
7. In the model, click the members/ancillaries/equipment for which you want to set the check.
    a. If **Mode** is **Set On** - the setting is switched on
    b. If **Mode** is **Set Off** - the setting is switched off
    c. If **Mode** is **Toggle** - the setting toggles between on and off
8. Continue applying the setting or press **Esc** to finish.

9.3 Review tabular data

The **Tabular Data** command allows you to view design summaries, sway & drift results, material lists, connection resistances, embodied carbon and more in tabular views. Tabular data can be filtered, sorted and exported to Excel.

- Review design summary tabular results (page 763)
- Review inter-story shear tabular results (page 764)
- Review sway check tabular results (page 765)
- Review story shear tabular results (page 768)
- Review drift check tabular results (page 770)
- Review seismic drift check tabular results (page 773)
- Review wind drift check tabular results (page 777)
- Review connection resistance tabular results (page 643)
- Review material list tabular results (page 782)
- Review embodied carbon detail (page 808)
- Review embodied carbon overview (page 809)
- Review floored area tabular results (page 810)
- Filter tabular data (page 811)

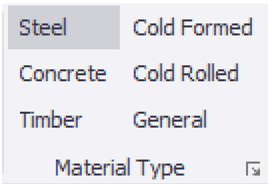
- Export tabular results to Excel (page 811)

Review design summary tabular results

To review a tabular **Design Summary**:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Design Summary**
4. In the **Result Type** group, select the analysis type for which the design summary is required.
5. In the **Material Type** group, select the material.

   The available selections in the **Characteristic** group (and the other groups) are reduced to those appropriate to the selected material.
6. In the **Characteristic** group, select the required characteristic.

   The available selections in the **Construction** group are reduced to those appropriate to the selected characteristic.
7. If applicable, in the **Construction** group, select the construction.

The available selections in the **Fabrication** group are reduced to those appropriate to the selected construction.

8. In the **Fabrication** group, select the fabrication.

The tabular design summary is displayed for every entity of the selected material and type in the structure.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

See also

Export tabular results to Excel (page 811)

Review member design (page 701)

Review inter-story shear tabular results

Inter-story shear can be reviewed by creating an **Inter-story Shear** tabular data view.

Inter-story Shear tabular data views:

- are displayed for the loadcase/combination selected in the **Loading** list
- are by default displayed for all levels
- but can have level filters applied (page 811)
- can be exported to a spreadsheet (page 811)

Create inter-story shear tabular results

To display **Inter-story Shear** as tabular data:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Inter-story Shear**

4. Choose the required loadcase/combination in the **Loading** list.

   The results are displayed as specified.

   If required you can configure a filter to limit the display to selected levels. To do this, see: Filter tabular data (page 811)

Review sway check tabular results

NOTE The sway check is not applicable if using the ACI/AISC head code.

Sway check results can be reviewed by creating a **Sway** tabular data view.

Sway tabular data views:

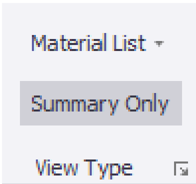
- can either be displayed in summary or detail:
  - **Summary Only** (default) - the column/wall with most critical result for the selected combinations is reported for each direction at each level
  - **Detailed** - the critical ratio in each direction at each level is reported for all columns/walls for the selected combinations
- are displayed for all columns /walls (default), but you can also select by material, characteristic and fabrication
- can have filters applied (page 811)
- can be examined in more detail by clicking the **Details...** button at the end of each row
- are linked to the 3D view: double-clicking a row in the table locates the row entity in the 3D view
- can be exported to a spreadsheet (page 811)
- are not suitable for direct printing, but the data can be included in a Building Analysis Checks (page 841) report which can then be printed.

Review sway check tabular results from the Project Workspace Status Tree

1. Go to the **Project Workspace** and in **Status** tree expand the **Design** branch.
2. Right click on **Sway** and from the context menu select **Review Data Table**
3. On the **Review Data** ribbon, choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:

   a. In the **Material Type** group, select the material(s).
   b. In the **Characteristic** group, select the required characteristic(s).
   c. In the **Fabrication** group, select the fabrication type(s).

4. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.
5. Choose if you want to create a summary or a detailed report by selecting/ unselecting **Summary Only** in the **View Type** group.



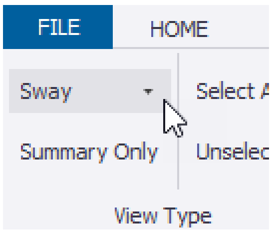
The tabular drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

Review sway check tabular results from a Review View

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

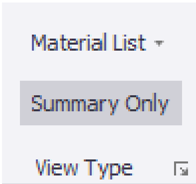
   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Sway**



4. Choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.

- To display results for a subset of materials/member types proceed as follows:
    a. In the **Material Type** group, select the material(s).
    b. In the **Characteristic** group, select the required characteristic(s).
    c. In the **Fabrication** group, select the fabrication type(s).

5. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
    - In the **Loading** list, select **All Combinations**.
    - In the **Loading** list, select a specific combination.
6. Choose if you want to create a summary or a detailed report by selecting/ unselecting **Summary Only** in the **View Type** group.

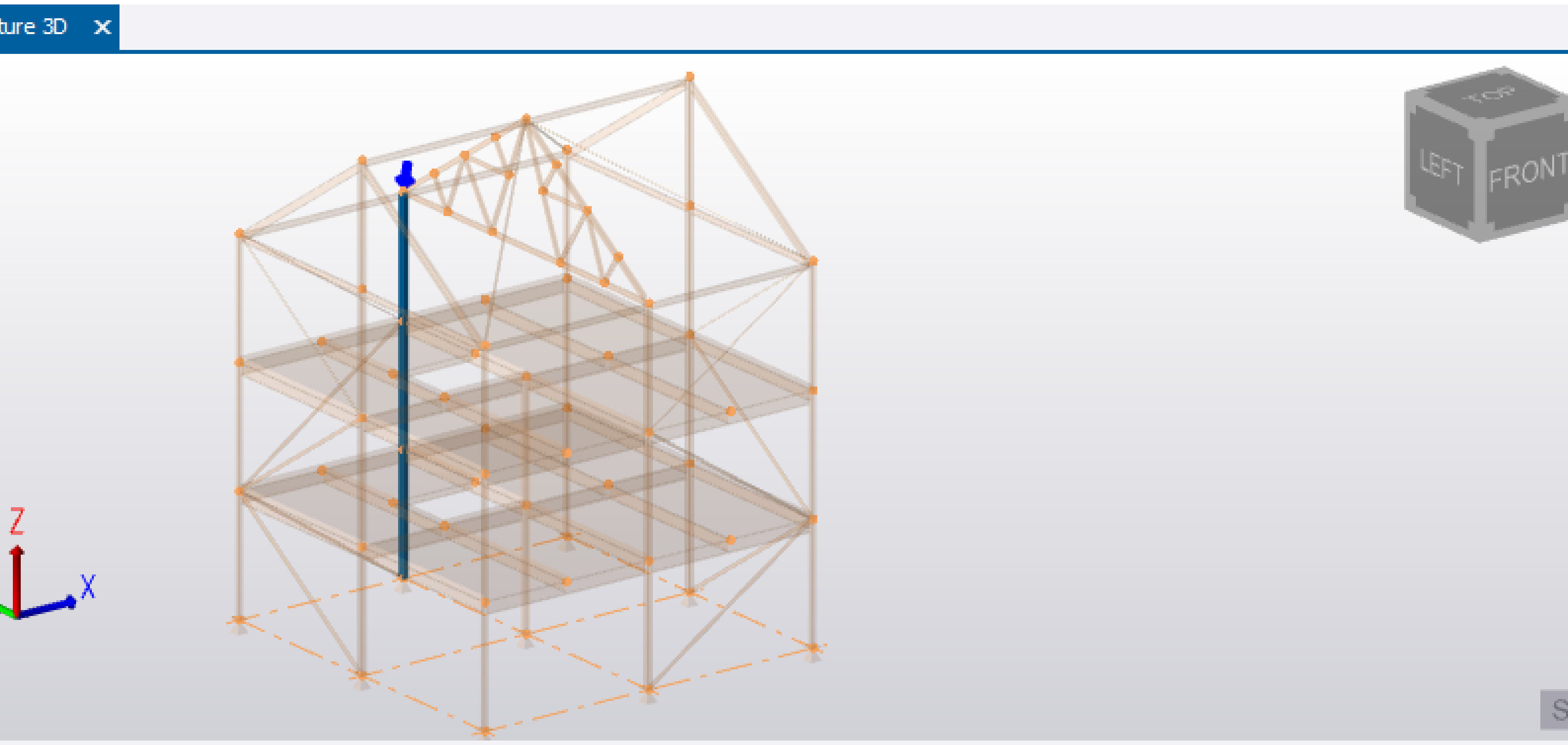


The tabular drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

Locate check in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Critical Sway

Ref.	Stack	Combination Dir 1	$\alpha_{Dir 1}$	Combination Dir 2	$\alpha_{Dir 2}$	Combination Dir 1/2	Twist	Status	Details
SC C/2	3	2 STR$_1$-1.35G+1.5Q+1.5RQ	110.659	2 STR$_1$-1.35G+1.5Q+1.5RQ	132.257	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.017	✓ Pass	Details...
SC B/3	3	2 STR$_1$-1.35G+1.5Q+1.5RQ	143.294	2 STR$_1$-1.35G+1.5Q+1.5RQ	84.744	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.041	✓ Pass	Details...
SC A/1	2	2 STR$_1$-1.35G+1.5Q+1.5RQ	54.427	2 STR$_1$-1.35G+1.5Q+1.5RQ	57.568	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.003	✓ Pass	Details...
SC C/1	2	2 STR$_1$-1.35G+1.5Q+1.5RQ	54.427	2 STR$_1$-1.35G+1.5Q+1.5RQ	55.859	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.003	✓ Pass	Details...
SC A/1	1	2 STR$_1$-1.35G+1.5Q+1.5RQ	32.265	2 STR$_1$-1.35G+1.5Q+1.5RQ	33.091	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.001	✓ Pass	Details...
SC C/1	1	2 STR$_1$-1.35G+1.5Q+1.5RQ	32.265	2 STR$_1$-1.35G+1.5Q+1.5RQ	32.630	2 STR$_1$-1.35G+1.5Q+1.5RQ	1.001	✓ Pass	Details...

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review story shear tabular results

To create a tabular floored area summary:

1. Open a view and change the view regime (page 89) to **Review View**.

2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.

3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Inter-story Shear** or **Cumulative Story Shear** as required.

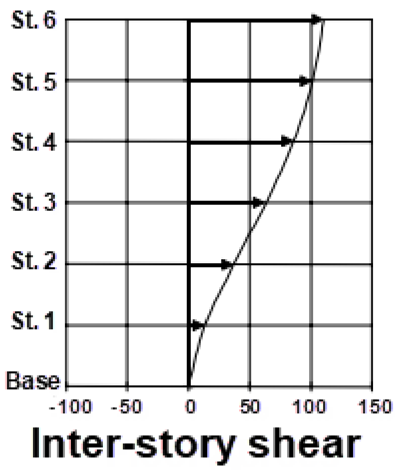
A table of story shears is displayed.

See also

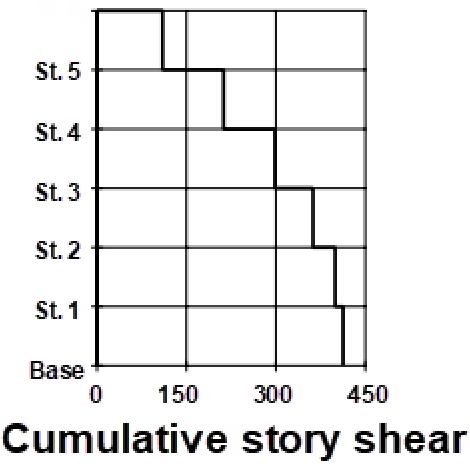
Inter-story shear and cumulative story shear (page 769)

Inter-story shear and cumulative story shear

The Inter-story Shear (also known as Floor Forces) table displays the sum of the lateral forces applied at each level in the structure.



Cumulative story shears are calculated by summing the floor forces from the top to the bottom of the structure, consequently they should get bigger as you move down.



Review drift check tabular results

NOTE The drift check is only applicable if using the ACI/AISC head code.

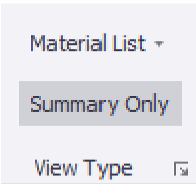
Drift check results can be reviewed by creating a **Drift** tabular data view.

Drift tabular data views:

- can either be displayed in summary or detail:
  - **Summary Only** (default) - the column/wall with most critical ratio for the selected combinations is reported for each direction at each level
  - **Detailed** - the critical ratio in each direction at each level is reported for all columns/walls for the selected combinations
- are displayed for all columns /walls (default), but you can also select by material, characteristic and fabrication
- can have filters applied (page 811)
- can be examined in more detail by clicking the **Details...** button at the end of each row
- are linked to the 3D view: double-clicking a row in the table locates the row entity in the 3D view
- can be exported to a spreadsheet (page 811)
- are not suitable for direct printing, but the data can be included in a Building Analysis Checks (page 841) report which can then be printed.

Review drift check tabular results from the Project Workspace Status Tree

1. Go to the **Project Workspace** and in **Status** tree expand the **Design** branch.
2. Right click on **Drift** and from the context menu select **Review Data Table**
3. On the **Review Data** ribbon, choose if you want results displayed for **all** available material and member types, or a subset only:
    - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
    - To display results for a subset of materials/member types proceed as follows:
        a. In the **Material Type** group, select the material(s).
        b. In the **Characteristic** group, select the required characteristic(s).
        c. In the **Fabrication** group, select the fabrication type(s).
4. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
    - In the **Loading** list, select **All Combinations**.
    - In the **Loading** list, select a specific combination.
5. Choose if you want to create a summary or a detailed report by selecting/ unselecting **Summary Only** in the **View Type** group.



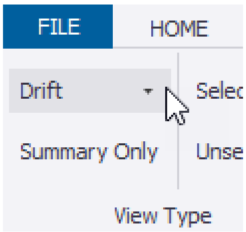
The tabular drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

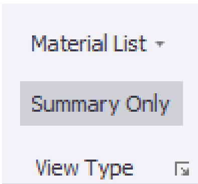
Review drift check tabular results from a Review View

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

    A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Drift**



4. Choose if you want results displayed for **all** available material and member types, or a subset only:
    - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
    - To display results for a subset of materials/member types proceed as follows:
        a. In the **Material Type** group, select the material(s).
        b. In the **Characteristic** group, select the required characteristic(s).
        c. In the **Fabrication** group, select the fabrication type(s).
5. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
    - In the **Loading** list, select **All Combinations**.
    - In the **Loading** list, select a specific combination.
6. Choose if you want to create a summary or a detailed report by selecting/ unselecting **Summary Only** in the **View Type** group.

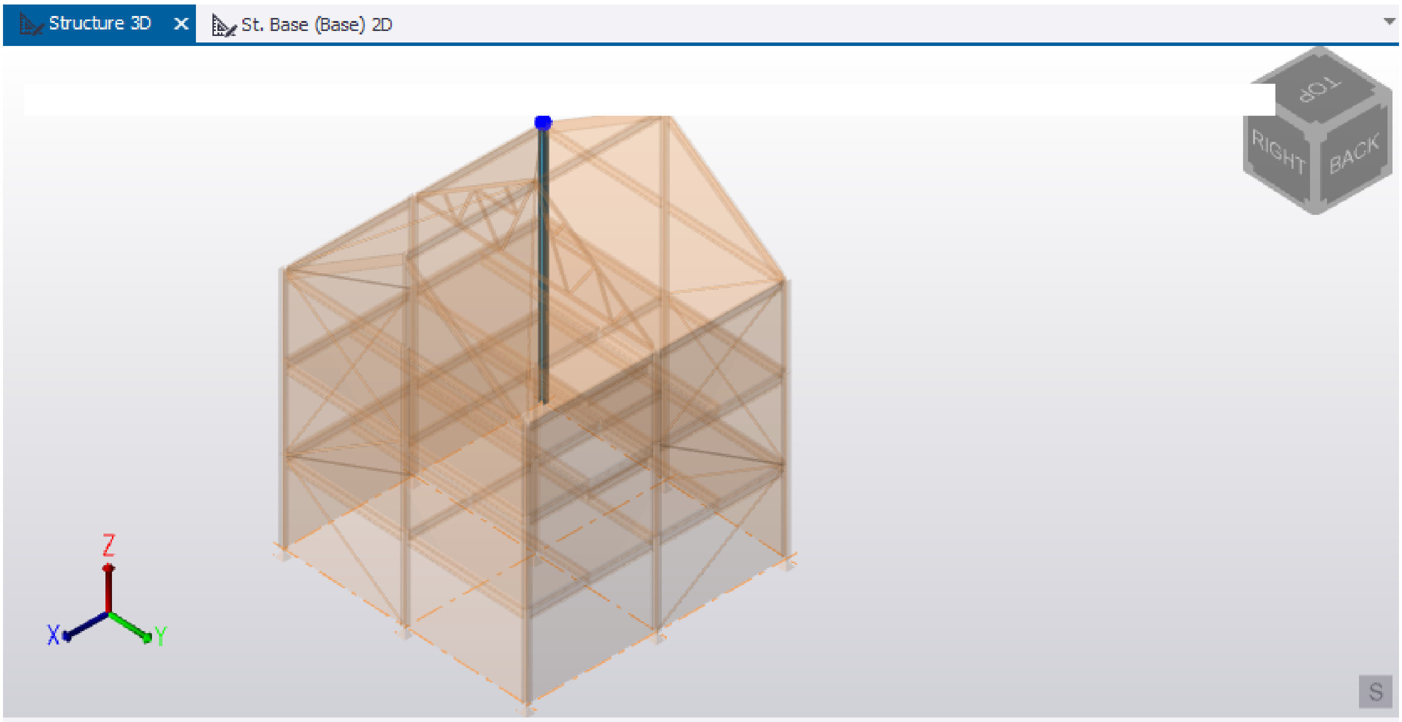


The tabular drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

Locate check in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Review Data

Drift

Level	Ref.	Stack	Combination Dir 1	Drift 1st order Dir 1 [mm]	Drift 2nd order Dir 1 [mm]	Ratio Dir 1	Combination Dir 2	Drift 1st order Dir 2 [mm]	Drift 2nd order Dir 2 [mm]	Ratio Dir 2	Status	Details
Top	SC C/2	3	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	43.7	44.2	1.012	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	2.3	2.3	1.003	Pass	Details...
Top	SC B/3	3	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	0.8	0.8	1.002	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	4.5	4.5	1.005	Pass	Details...
St. 2 (2)	SC A/1	2	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	1.2	1.2	1.011	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	2.7	2.8	1.013	Pass	Details...
St. 2 (2)	SC C/1	2	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	1.2	1.2	1.011	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	2.7	2.7	1.013	Pass	Details...
St. 1 (1)	SC A/1	1	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	1.5	1.5	1.019	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	3.5	3.5	1.022	Pass	Details...
St. 1 (1)	SC C/1	1	18 $ASD_{9.1}$-D+0.75L+0.75Lr+0.45W	1.5	1.5	1.019	46 $LRFD_{10.4}$-1.2D+L+0.5S+W	3.4	3.5	1.023	Pass	Details...

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review seismic drift check tabular results

Seismic drift check results can be reviewed by creating a **Seismic drift** tabular data view.

Seismic drift tabular data views:

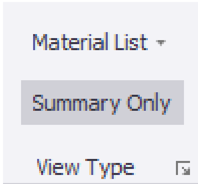
- can either be displayed in summary or detail:
  - **Summary Only** (default) - the column/wall with most critical ratio for the selected combinations is reported for each direction at each level
  - **Detailed** - the critical ratio in each direction at each level is reported for all columns/walls for the selected combinations
- are displayed for all columns /walls (default), but you can also select by material, characteristic and fabrication
- can have filters applied (page 811)
- can be examined in more detail by clicking the **Details...** button at the end of each row
- are linked to the 3D view: double-clicking a row in the table locates the row entity in the 3D view
- can be exported to a spreadsheet (page 811)
- are not suitable for direct printing, but the data can be included in a Building Analysis Checks (page 841) report which can then be printed.

Review seismic drift check tabular results from the Project Workspace Status Tree

1. Go to the **Project Workspace** and in **Status** tree expand the **Design** branch.
2. Right click on **Seismic Drift** and from the context menu select **Review Data Table**
3. On the **Review Data** ribbon, choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
4. Choose if you want critical results determined from **All Combinations**, or a specific combination only:

- In the **Loading** list, select **All Combinations**.
- In the **Loading** list, select a specific combination.

5. Choose if you want to create a summary or a detailed report by selecting/ unselecting **Summary Only** in the **View Type** group.



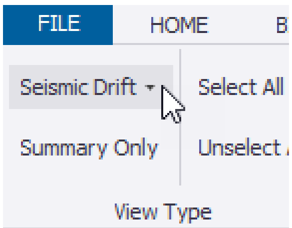
The tabular results are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

Review seismic drift check tabular results from a Review View

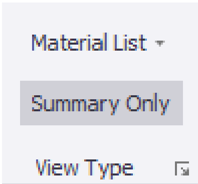
1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Seismic drift**



4. Choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).

5. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.
6. Choose if you want to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.

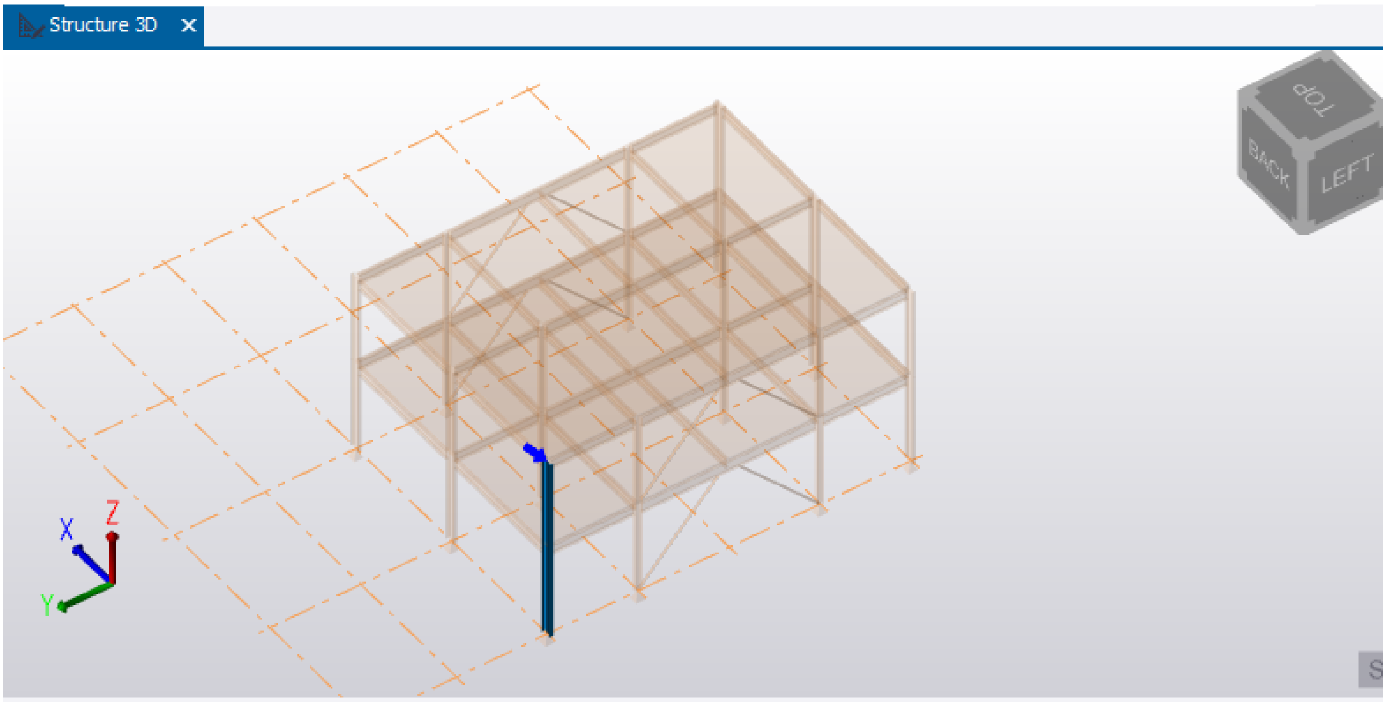


The tabular results are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

Locate check in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Review Data

Seismic Drift

Level	Ref.	Stack	Combination Dir 1	Drift Ratio Dir 1	$\theta_{Dir\,1}$ Ratio	Combination Dir 2	Drift Ratio Dir 2	$\theta_{Dir\,2}$ Ratio	Status	Details
St. 2 (2)	SC A/5	2	234 $LRFD_{11.2}$-1.2D+L+0.2S+E	0.3245	0.3351	238 $LRFD_{11.6}$-1.2D+L+0.2S+E	0.0257	0.0246	Pass	Details...
St. 2 (2)	SC A/1	2	233 $LRFD_{11.1}$-1.2D+L+0.2S+E	0.3245	0.3351	238 $LRFD_{11.6}$-1.2D+L+0.2S+E	0.0257	0.0246	Pass	Details...
St. 1 (1)	SC A/5	1	234 $LRFD_{11.2}$-1.2D+L+0.2S+E	0.7194	0.8536	238 $LRFD_{11.6}$-1.2D+L+0.2S+E	0.0322	0.0413	Warning	Details...
St. 1 (1)	SC A/1	1	233 $LRFD_{11.1}$-1.2D+L+0.2S+E	0.7194	0.8536	238 $LRFD_{11.6}$-1.2D+L+0.2S+E	0.0322	0.0413	Warning	Details...

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review wind drift check tabular results

Wind drift check results can be reviewed by creating a **Wind drift** tabular data view.

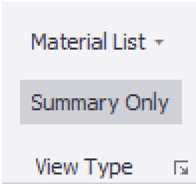
Wind drift tabular data views:

- can either be displayed in summary or detail:

    - **Summary Only** (default) - the column/wall with most critical ratio for the selected combinations is reported for each direction at each level
    - **Detailed** - the critical ratio in each direction at each level is reported for all columns/walls for the selected combinations
- are displayed for all columns /walls (default), but you can also select by material, characteristic and fabrication
- can have filters applied (page 811)
- can be examined in more detail by clicking the **Details...** button at the end of each row
- are linked to the 3D view: double-clicking a row in the table locates the row entity in the 3D view
- can be exported to a spreadsheet (page 811)
- are not suitable for direct printing, but the data can be included in a Building Analysis Checks (page 841) report which can then be printed.

Review wind drift check tabular results from the Project Workspace Status Tree

1. Go to the **Project Workspace** and in **Status** tree expand the **Design** branch.
2. Right click on **Wind Drift** and from the context menu select **Review Data Table**
3. On the **Review Data** ribbon, choose if you want results displayed for **all** available material and member types, or a subset only:
    - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
    - To display results for a subset of materials/member types proceed as follows:
        a. In the **Material Type** group, select the material(s).
        b. In the **Characteristic** group, select the required characteristic(s).
        c. In the **Fabrication** group, select the fabrication type(s).
4. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
    - In the **Loading** list, select **All Combinations**.
    - In the **Loading** list, select a specific combination.
5. Choose if you want to create a summary or a detailed report by selecting/ unselecting **Summary Only** in the **View Type** group.



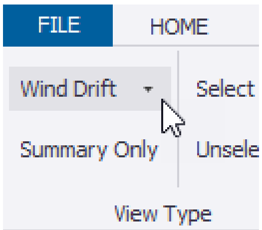
The tabular wind drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

Review wind drift check tabular results from a Review View

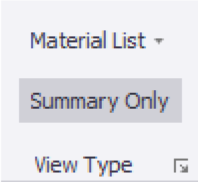
1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.
3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Wind drift**



4. Choose if you want results displayed for **all** available material and member types, or a subset only:
   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection**.
   - To display results for a subset of materials/member types proceed as follows:
     a. In the **Material Type** group, select the material(s).
     b. In the **Characteristic** group, select the required characteristic(s).
     c. In the **Fabrication** group, select the fabrication type(s).
5. Choose if you want critical results determined from **All Combinations**, or a specific combination only:
   - In the **Loading** list, select **All Combinations**.
   - In the **Loading** list, select a specific combination.

6. Choose if you want to create a summary or a detailed report by selecting/ unselecting **Summary Only** in the **View Type** group.

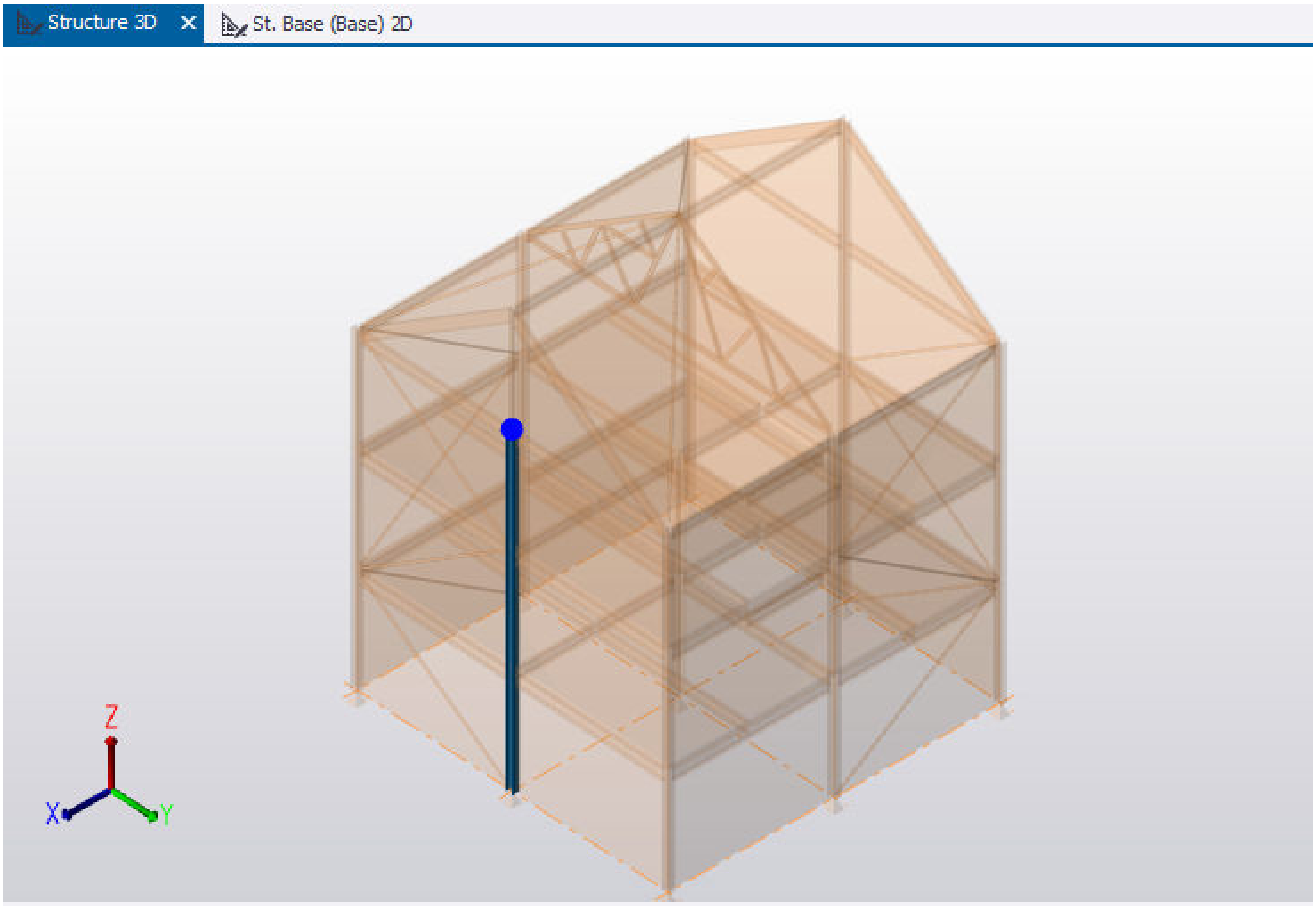


The tabular wind drift checks are displayed as specified.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

Locate check in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Review Data

Level	Ref.	Stack	Combination	Deflection Dir 1 [mm]	Deflection Dir 2 [mm]	Drift Dir 1 [mm]	Drift Dir 2 [mm]	Ratio Dir 1	Ratio Dir 2	Status Dir 1	Status Dir 2	Details
Top	SC C/2	3	50 LRFD$_{12.4}$-0.9D+W	26.2	4.8	26.3	1.1	152.012	3503.178	Fail	Pass	Details...
Top	SC B/3	3	50 LRFD$_{12.4}$-0.9D+W	-0.1	5.7	0.0	2.0	170421.311	1972.091	Pass	Pass	Details...
St. 2 (2)	SC A/1	2	47 LRFD$_{12.1}$-0.9D+W	3.7	-0.2	1.7	0.1	2370.157	41721.556	Pass	Pass	Details...
St. 2 (2)	SC C/1	2	50 LRFD$_{12.4}$-0.9D+W	-0.1	3.7	0.0	1.6	2.301E+06	2482.328	Pass	Pass	Details...
St. 1 (1)	SC A/1	1	47 LRFD$_{12.1}$-0.9D+W	2.0	-0.1	2.0	0.1	1998.949	53404.925	Pass	Pass	Details...
St. 1 (1)	SC A/1	1	50 LRFD$_{12.4}$-0.9D+W	-0.1	2.1	0.1	2.1	42301.068	1930.097	Pass	Pass	Details...

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review material list tabular results

The material content of the model can be readily assessed in the form of a **Material List**. This can be created and viewed either in a *tabular data view* or a *report*, according to your needs.

- Material List tabular data view:
  - created for a specific material and entity type
  - displayed on screen (not suitable for direct printing)
  - can be sorted in ascending/descending order of a column by clicking a column heading
  - row content can be located within a 3D view by double-clicking a row
  - can be exported to a spreadsheet
- Material List report:
  - created for *multiple* materials and entity types
  - displayed in a format suitable for printing
  - can be exported to a spreadsheet

Two levels of content can be generated:

- **Summary Only** - this is the default level and is very condensed, for example:
  - total quantities only for each beam/ column section size
  - total quantities for each slab
  - total weight of reinforcement (by mass, and by mass/unit volume)
- **Detailed** - this is a longer report, for example:
  - quantities for each span/ stack/ panel for each concrete beam / column / wall
  - quantities for each section size of a given length for beams and columns in other materials
  - quantities for each slab item
  - quantities for individual reinforcement bar/mesh types

The topics in the section all relate to the tabular data view. To find out how to create a material list report, see: Material Listing report (page 844)

Create material list tabular results

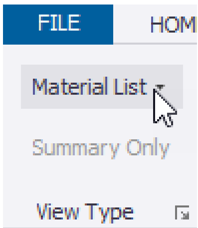
To create a material list as tabular data:

1. Open a view and change the view regime (page 89) to **Review View**.

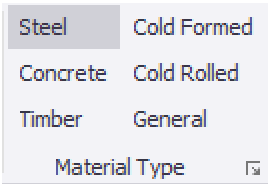
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.

3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Material List**



4. In the **Material Type** group, select the material.

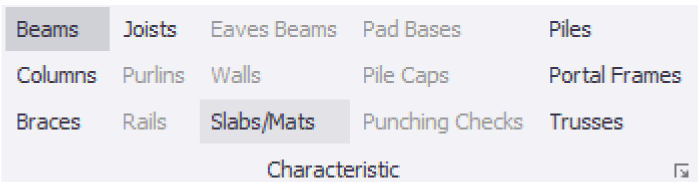


   The available selections in the **Characteristic** group (and the other groups) are reduced to those appropriate to the selected material.

NOTE To find out exactly what summary and detailed level output is produced for each material type and characteristic, see:

- Steel material lists (page 786)
- Concrete material lists (page 792)
- Timber material lists (page 803)
- Cold formed material lists (page 805)
- General material lists (page 806)

5. In the **Characteristic** group, select the required characteristic.



   The available selections in the **Construction** group are reduced to those appropriate to the selected characteristic.

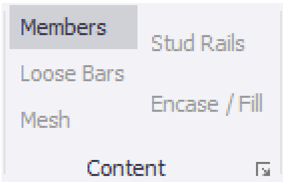
6. If applicable, in the **Construction** group, select the construction.

The available selections in the **Fabrication** group are reduced to those appropriate to the selected construction.

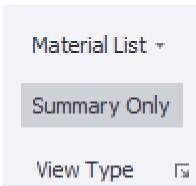
7. In the **Fabrication** group, select the fabrication.

The available selections in the **Content** group are reduced to those appropriate to the selected fabrication.

8. In the **Content** group, select the content.



9. If you have selected **Members** in the **Content** group, you have the option to create a summary or a detailed report by selecting/unselecting **Summary Only** in the **View Type** group.



The tabular material list is displayed for every entity of the selected material and type in the structure.

If required you can configure a filter to limit the display to sub-parts of the structure only (selected levels, selected frames etc.) To do this, see: Filter tabular data (page 811)

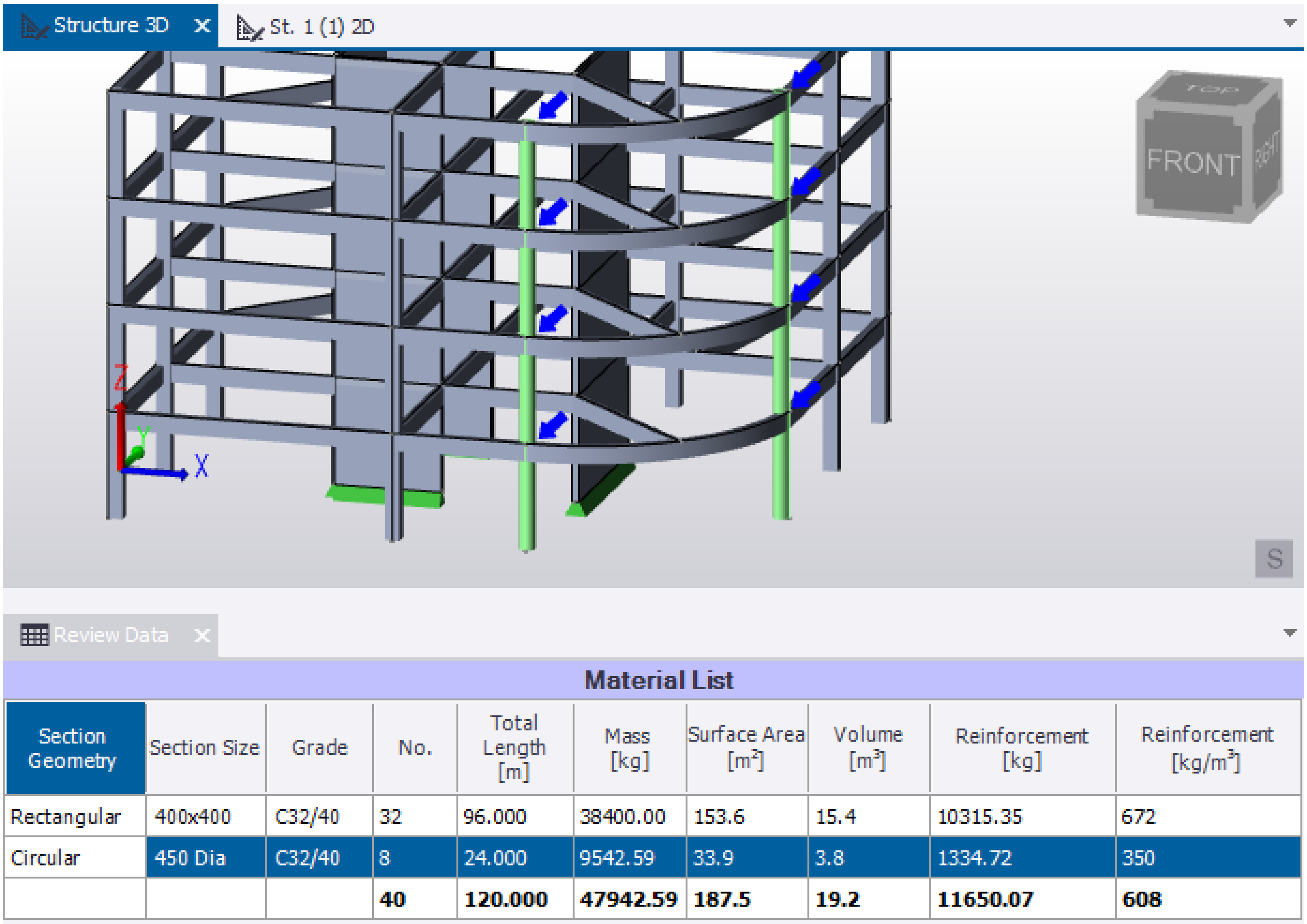
See also:

- Steel material lists (page 786)

- [Concrete material lists (page 792)]
- [Timber material lists (page 803)]
- [Cold formed material lists (page 805)]
- [General material lists (page 806)]

Locate material list rows in a 3D view

Double-clicking on a specific row in the material list tabular data view locates where the row content is in the model by highlighting it in a 3D view.



Material List									
Section Geometry	Section Size	Grade	No.	Total Length [m]	Mass [kg]	Surface Area [m^2]	Volume [m^3]	Reinforcement [kg]	Reinforcement [kg/m^3]
Rectangular	400x400	C32/40	32	96.000	38400.00	153.6	15.4	10315.35	672
Circular	450 Dia	C32/40	8	24.000	9542.59	33.9	3.8	1334.72	350
			40	**120.000**	**47942.59**	**187.5**	**19.2**	**11650.07**	**608**

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Export material list to Excel

1. Create the material list and ensure that it contains the necessary information.
2. On the **Review Data** ribbon tab, click **Excel**.
3. Click **Yes** to confirm exporting the report.

   The material list opens in Excel

Material lists for steel

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length
- Composite beam reports also include the number of connectors, the transverse reinforcement associated with the beam (at detailed level only), and the weight of reinforcement

Construction	Fabrication	Content	View Type	Description
Composite	• Rolled • Plated • Westok Plated • Fabsec Plated • DELTABEAM	Members	Summary	One row for each size: size, grade, number (of that size), no. of connectors, total length, mass, surface area, reinforcement
			Detailed	One row for each size & length: size, grade, number (of that size & length), no. of connectors, length, mass, surface area, reinforcement

Construction	Fabrication	Content	View Type	Description
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
	Westok Cellular	Members	Summary	One row for each size: size, grade, number (of that size), no. of connectors, total length, gross mass, net mass, gross surface area, net surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), no. of connectors, length, gross mass, net mass, gross surface area, net surface area
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass

Construction	Fabrication	Content	View Type	Description
		Mesh	Mesh	One row for each type: type, unit mass, total area, total mass
Non-composite	• Rolled • Plated • Westok Plated • Fabsec Plated • DELTABEAM	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area
	Westok Cellular	Members	Summary	One row for each size: size, grade, number (of that size), total length, gross mass, net mass, gross surface area, net surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, gross mass, net mass, gross

Construction	Fabrication	Content	View Type	Description
				surface area, net surface area

Columns, Braces, Joists

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length

Characteristic	Fabrication	Content	View Type	Description
Columns	• Rolled • Plated	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area
	• Concrete Filled • Concrete Encased	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size &

Characteristic	Fabrication	Content	View Type	Description
				length), length, mass
		Encase/Fill	Detailed	One row for each size: size, grade, number (of that size), length, mass
Braces	Rolled	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area
Joists	N/A	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass
			Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass

Slabs/Mats

NOTE	• Summary reports provide totals for the entire slab • Detailed reports are given on a slab item basis

Characteristic	Content	View Type	Description
Slabs/Mats	Members	Summary	One row for each slab: level, slab, depth, grade, mass, gross surface area, net surface area, volume
		Detailed	One row for each slab item: level, slab, depth, grade, mass, gross surface area, net surface area, volume

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Portal Frames

Characteristic	Content	View Type
Portal Frames	Members	Detailed

Trusses

Characteristic	Content	View Type
Trusses	Members	Detailed

Material lists for concrete

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column. This can be useful for example when wanting to sort members by how much reinforcement they contain.

Beams and Columns

NOTE
- Summary reports provide total quantities for each section size
- Detailed reports provide quantities on a span/stack basis
  - When group design is active this reduces to every detailing group span/stack
- Reinforcement running between spans/stacks is shared
- No additional detailing allowance has been applied
- Reinforcement quantities are given in terms of weight and weight/unit volume

Characteristic	Fabrication	Content	View Type	Description
Beams	Cast-in-place	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
			Detailed	One row for each span: Group/ Member ref., section geometry, size, grade, number (of that ref), length, mass,

Characteristic	Fabrication	Content	View Type	Description
				surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
	Precast	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume
			Detailed	One row for each span: Member reference, section geometry, size, grade, number, length, mass, surface area, volume

Characteristic	Fabrication	Content	View Type	Description
	Post-tensioned	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume
			Detailed	One row for each span: Member reference, section geometry, size, grade, number, length, mass, surface area, volume
Columns	Cast-in-place	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
			Detailed	One row for each size: section geometry, size, grade, number (of that size), total length,

Characteristic	Fabrication	Content	View Type	Description
				mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
	Precast	Members	Summary	One row for each size: section geometry, size, grade, number (of that size), total length, mass, surface area, volume
			Detailed	One row for each stack: Member reference, section geometry, size, grade, number, length, mass, surface area, volume

Walls

NOTE

- Summary reports provide total quantities for each wall
- Detailed reports provide quantities on a wall panel basis
- Reinforcement running between panels is shared
- No additional detailing allowance has been applied
- Reinforcement quantities are given in terms of weight and weight/unit volume

Characteristic	Fabrication	Content	View Type	Description
Walls	Cast-in-place	Members	Summary	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
			Detailed	One row for each panel: Member reference, gross mass, mass, gross surface area, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass

Characteristic	Fabrication	Content	View Type	Description
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
	Precast	Members	Summary	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume
			Detailed	One row for each panel: Member reference, gross mass, mass, gross surface area, surface area, volume
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

- Patch and punching reinforcement is approximately distributed by sharing it equally between slab items it touches - it is not split based on covered areas
- Reinforcement allows for laps at the slab item boundary
- An approximate user defined additional detailing allowance (specified in Design Settings for cast-in-place slabs) is applied to all reinforcement defined in any slab type.
- Punching reinforcement weight is approximated (based on ACI guidance)
- Precast and post-tensioned slabs are not designed, but the reinforcement that is specified in the slab properties is reported. The post-tensioned tendon information is not reported and must be allowed for in some other way.

Construction	Fabrication	Content	View Type	Description
Composite	N/A	Members	Summary	One row for each slab: level, slab, depth, manufacturer, reference, gauge, grade, mass, gross surface area, net surface area, volume, reinforcement
			Detailed	One row for each slab item: level, slab, depth, manufacturer, reference, gauge, grade, mass, gross surface area, net surface area, volume, reinforcement
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total

Construction	Fabrication	Content	View Type	Description
				mass inc. allowance
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass inc. allowance
Non-composite	Cast-in-place	Members	Summary	One row for each slab: level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
			Detailed	One row for each slab item: level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement

Construction	Fabrication	Content	View Type	Description
				(mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass inc. allowance
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass inc. allowance
	Precast	Members	Summary	One row for each slab: level, slab, depth, manufacturer, reference, unit depth, topping, grade, mass, gross surface area, net surface area, volume, reinforcement
			Detailed	One row for each slab item: level, slab, depth, manufacturer, reference, unit depth, topping, grade, mass,

Construction	Fabrication	Content	View Type	Description
				gross surface area, net surface area, volume, reinforcement
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass inc. allowance
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass inc. allowance
	Post Tensioned	Members	Summary	One row for each slab: level, slab, type, depth, grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
			Detailed	One row for each slab item: level, slab, type, depth,

Construction	Fabrication	Content	View Type	Description
				grade, mass, gross surface area, net surface area, volume, reinforcement, punching reinforcement, total reinforcement (mass), total reinforcement (mass/unit area), total reinforcement (mass/ unit volume)
		Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass inc. allowance
		Mesh	Detailed	One row for each type: type, unit mass, total area, total mass inc. allowance

Punching checks

NOTE The weight of punching shear rails is approximated based on ACI guidance.

Content	View Type	Description
Stud Rails	Detailed	One row for each rail ref: ref., diameter,stud height, studs per rail, stud spacing, rail length, total no. of rails, approx. mass

Pad Bases, Pile Caps and Piles

NOTE
- Reports provide quantities for each individual base
- Reinforcement quantities are given in terms of weight and weight/unit volume

Characteristic	Content	View Type	Description
• Pad Bases • Pile Caps	Members	Detailed	One row for each base: member reference, grade, mass, surface area, volume, reinforcement (mass), reinforcement (mass/unit volume)
	Loose Bars	Detailed	One row for each type: type, unit mass, total length, total mass
	Mesh	Detailed	One row for each type: type, unit mass, total area, total mass
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Material lists for timber

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams, Columns, Braces

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length

Characteristic	View Type	Description
• Beams • Columns • Braces	Summary	One row for each size: size, grade, number (of that size), total length, mass
	Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

Characteristic	Content	View Type	Description
Slabs/Mats	Members	Summary	One row for each slab: level, slab, depth, grade, mass, gross surface area, net surface area, volume
		Detailed	One row for each slab item: level, slab, depth, grade, mass, gross surface area, net surface area, volume

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Trusses

Characteristic	Content	View Type
Trusses	Members	Detailed

Material lists for cold formed

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams, Columns, Braces

Characteristic	Content	View Type	Description
• Beams • Columns • Braces	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
		Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Trusses

Characteristic	Content	View Type
Trusses	Members	Detailed

Material lists for general materials

TIP When viewing a Material List tabular data view, clicking a column header sorts the table by that column.

Beams, Columns, Braces

NOTE
- Summary reports provide quantities for each section size
- Detailed reports provide quantities for each section size of a given length

Characteristic	Content	View Type	Description
• Beams • Columns • Braces	Members	Summary	One row for each size: size, grade, number (of that size), total length, mass, surface area
		Detailed	One row for each size & length: size, grade, number (of that size & length), length, mass, surface area

Walls

Characteristic	Content	View Type	Description
Walls	Members	Summary	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume
		Detailed	One row for each wall: reference, gross mass, mass, gross surface area, surface area, volume

Slabs/Mats

NOTE
- Summary reports provide totals for the entire slab
- Detailed reports are given on a slab item basis

Characteristic	Content	View Type	
Slabs/Mats	Members	Summary	One row for each slab: level, slab, depth, grade, mass, gross surface area, net surface area, volume
		Detailed	One row for each slab item: level, slab, depth, grade, mass, gross surface area, net surface area, volume

Piles

Characteristic	Content	View Type	Description
Piles	Members	Detailed	One row for each pile type: reference, dimension, shape, installation type, installation length, number, total length

Review embodied carbon detail

Embodied carbon can be reviewed in depth by creating an **Embodied Carbon Detail** tabular data view.

Embodied Carbon Detail tabular data views:

- are displayed for one construction type at a time according to the material, characteristic, construction and fabrication type selected in the ribbon
- display a single row listing the total embodied carbon mass per member reference when **Summary Only** is selected
- display separate rows listing the quantities, units, carbon factors, and embodied carbon masses for each material that comprises the member reference when **Summary Only** is unselected
- can have filters applied (page 811)
- are linked to the 3D view: double-clicking a row in the table locates the row entities in the 3D view
- can be exported to a spreadsheet (page 811)
- are not suitable for direct printing, (but could be printed if exported to a spreadsheet).

NOTE The embodied carbon mass totals are also included in the Review material list tabular results (page 782).

Create embodied carbon detail tabular results

To display an embodied carbon detail as tabular data:

1. Open a view and change the view regime (page 89) to **Review View**.
2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.

3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Embodied Carbon Detail**
4. Choose the construction type by selecting the required material, characteristic, construction and fabrication type from the ribbon.
5. Choose if you want summary or detailed results:
   - To display summary only, in the **View Type** group, select **Summary Only**.
   - To display more detailed results, unselect **Summary Only**.

   The results are displayed as specified.

   If required you can configure a filter to limit the display to selected levels, frames etc. To do this, see: Filter tabular data (page 811)

Locate tabular data in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review embodied carbon overview

Embodied carbon can be reviewed in summary form by creating an **Embodied Carbon Overview** tabular data view.

Embodied Carbon Overview tabular data views:

- are reported for each plane when **Group by Plane** is selected, or each construction type when unselected
- are either displayed for the whole model, or according to the materials, characteristics, construction and fabrication types selected in the ribbon
- can have filters applied (page 811) (selected UDAs, frames, or groups)
- are linked to the 3D view: double-clicking a row in the table locates the row entities in the 3D view
- can be exported to a spreadsheet (page 811)
- are not suitable for direct printing, but the data can be included in an Embodied Carbon report (page 842) which can then be printed.

Create embodied carbon overview tabular results

To display an embodied carbon overview as tabular data:

1. Open a view and change the view regime (page 89) to **Review View**.

2. On the **Review** ribbon, click **Tabular Data**

   A **Review Data** tab is added to the ribbon and a **Review Data View** opens in a new window.

3. On the **Review Data** ribbon, in the list in the **View Type** group, select **Embodied Carbon Overview**

4. Choose if you want results displayed for **all** available material and member types, or a subset only:

   - To display results for all available material and member types, in the **View Type** group, click **Toggle Selection** so that all construction types are selected.
   - To display results for a subset of construction types proceed as follows:

     a. In the **Material Type** group, select the material(s).

     b. In the **Characteristic** group, select the required characteristic(s).

     c. In the **Fabrication** group, select the fabrication type(s).

5. Choose if you want results reported by plane, or by construction type:

   - For the former, in the **View Type** group, select **Group by Plane**.
   - For the latter, unselect **Group by Plane**.

   The **Embodied Carbon Overview** is displayed as specified.

Locate tabular data in a 3D view

Double-clicking on a specific row in the tabular data view locates where the row content is in the model by highlighting it in a 3D view.

TIP Make use of the “ghost unselected” feature to make highlighted entities clearer.

Review floored area tabular results

To create a tabular floored area summary:

1. If necessary, change the view regime (page 89) to a **Review View**.
2. On the **Review** ribbon tab, click **Tabular Data**.

   A **Review Data** tab opens on the ribbon and a **Review Data View** is displayed.

3. On the **Review Data** ribbon tab, in the list in the **View Type** group, select **Floored Area**.

A table of floored areas is displayed.

Filter tabular data

You can apply model filters to enable selective display of tabular data. The filters available depend on the view type displayed.

Create and apply filters

1. In the **Filters** group of the **Review Data** tab, select the desired filter type.

   The **Select filter items** list opens.

2. In the **Select filter items** list, select the necessary options to define the filter requirements.

3. Click **OK**.

   Tekla Structural Designer applies the filter.

Edit filters

NOTE If a filter type is not applicable for the current tabular data view type, it will be unavailable.

1. On the **Review Data** tab, click either **Filter Items...**

   The **Select filter items** list opens.

2. In the **Select filter items** list, select the necessary options to define the filter requirements.

3. Click **OK**.

   Tekla Structural Designer applies the filter.

Export tabular results to Excel

1. Create the tabular data view and ensure that it contains the necessary information.

2. On the **Review Data** ribbon tab, click **Excel**.

3. Click **Yes** to confirm exporting the report.

   The tabular data opens in Excel

10 Calculate slab deflections

In Tekla Structural Designer you can choose to adopt a rigorous approach to slab deflection calculation using iterative cracked section analysis.

NOTE Tekla Structural Designer's iterative cracked section analysis for slab deflections is only available for the Eurocode and ACI/AISC Head Code.

- To begin, see Get started with slab deflection analysis (page 812)
- To define event sequences in Tekla Structural Designer, see Work with event sequences (page 813).
- To set up the deflection checks and place check lines, see Work with check lines (page 815).
- To perform the analysis, see Run a slab deflection analysis (page 817)
- To investigate the results, see Slab deflection results and reports (page 818).

Related video

Rigorous Slab Deflection Design (ACI)

Rigorous Slab Deflection (Eurocode)

10.1 Get started with slab deflection analysis

Two different ways of checking slab deflections in Tekla Structural Designer are available. Either deemed-to-satisfy checks, or a rigorous theoretical deflection estimation. These are introduced in the topic .

If you decide to adopt the latter approach the video links at the bottom of this page are a good start point, and the topic also provides an overview of the steps required.

Many topics are then discussed in more detail in the , which also provides Tutorial worked examples for you to download and work through.

Related video

Rigorous Slab Deflection Design (ACI)

Rigorous Slab Deflection (Eurocode)

10.2 Work with event sequences

Event sequences are edited in the (page 1194).

Add an event to the end of the event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. Click **Add** to Add the Event as the last event in the sequence.
4. Name the new event and specify the event parameters according to your needs. If you want the same parameters to apply to custom events, ensure the **Update custom event sequences** box is selected.
5. Click **OK**.

Insert an event within the event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. Select the event above which you want to insert the new event.
4. Click **Insert** to insert the event.
5. Name the new event and specify the event parameters according to your needs. If you want the same parameters to apply to custom events, ensure the **Update custom event sequences** box is selected.
6. Click **OK**.

Re-order events in the event sequence

You can Add, Insert, Remove and Re-Order Events from the Model Event Sequence page.

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. In the **Drawing Variant** list, select the Event you want to move.
4. Click**Move Up** or **Move Down** until the Event is in the required position.
5. Click **OK**.

Remove an event from the event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page.
3. Select the event which you want to remove.
4. Click **Remove** to remove the event.
5. Click **OK**.

Edit event parameters

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page, or specific custom event sub page as required.
3. Edit the event parameters according to your needs. If editing the Model Event Sequence and you want the same parameters to apply to custom events, ensure the **Update custom event sequences** box is selected.
4. Click **OK**.

Edit event loadcases

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Model Event Sequence** sub page, or specific custom event sub page as required.
3. In the left hand pane, expand the Event Sequence if required so that you can select the Event.

   The available and included loadcases are displayed.

4. Move loadcases between the available and included lists as required. If editing the Model Event Sequence and you want the same loadcases to apply to custom events, ensure the **Update custom event sequences** box is selected.
5. Click **OK**.

Create a custom event sequence

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Event Sequences** sub page.
3. Click **Add** to add a new Custom Event.
4. In the right hand pane click on the event name to rename it.
5. In the left hand pane click on the event name in order to define the event parameters in the sequence.
6. Click **OK**.

Apply a custom event sequence to a submodel

1. On the **Slab Deflection** tab, click **Event Sequences**

   The **Load Event Sequences** dialog box opens.
2. Go to the **Submodels** sub page.

   The table displays the event sequence assigned to each submodel.
3. Use the droplist to change the Event Sequence assigned to a specific submodel as required.
4. Click **OK**.

10.3 Work with check lines

User defined Check Lines can be placed across 2D element meshes, They are similar to 2D results strips, (but with zero width and different properties).

From these check lines, deflection results are determined from the shell/plate/membrane nodal analysis results - these can then be checked against specified limits for slab design purposes.

Engineering judgment is required when positioning the check lines to ensure worst case deflections are obtained.

Create the deflection checks to be applied to check lines

1. On the **Slab Deflection** tab, click **Deflection Checks**

   The Slab Deflection Check Catalogue (page 1211) is displayed, containing any checks that have been defined.

2. Click **Add**

   A new check is created in the table.

3. Select the name in order to give it a more descriptive title.
4. Choose the check **Type**: (Total, Instantaneous, or Differential).
   a. If the Type is Differential, select the **Start Event** from the droplist.
5. Select the **Event** to check from the droplist.
6. Enter the **Deflection Limit** to be checked.
7. Select **Use in new Check Lines** if you want this check to be performed in each check line that is created.

Create a check line

Check lines can only be created while in the **Slab Deflections View** regime.

1. Open a 2D view of the FE mesh where the check line is to be placed.
2. On the **Slab Deflection** tab, click **Create**

   The Check Line properties are displayed in the **Properties** window.

3. Adjust the properties to specify:
   a. The number of stations
   b. The upper and lower Flat Zone Reductions
   c. Up to 6 deflection checks to be performed (these can either be selected from the predefined Deflection Checks Catalogue, or a new check can be defined)
4. Click a point where the strip is to start.
5. Click a 2nd point where the strip is to end. (Neither start or end points have to match nodes in the mesh.)

   A check line is created between the points that you identified.

6. Either continue to place further check lines, or if done, press **Esc** to exit.

Delete a check line

To be able to delete a check line you must first ensure that Slab Deflection Check Lines are switched on in Scene Content.

1. Open a view displaying the check lines to be deleted.
2. Click or press **Delete**
3. Click on the check lines to be deleted.

10.4 Run a slab deflection analysis

The same basic analysis process is followed irrespective of whether the current level (sub-model), a selected level, or all slabs in the model are analyzed.

In simple terms, events are considered in sequence.

For each event:

- An iterative cracked section analysis including long term effects determines the deflection at the end of the event.
- An additional analysis using the determined state of cracking along with short term cracked properties is undertaken to calculate the total instantaneous deflection associated with the event.
- The state of cracking is carried forward to the next event as the starting point.

To run the analysis for specific or all sub models see the following instructions.

Run a slab deflection analysis for the current sub model

To analyze a specific sub model:

1. Open a view of the required sub model from the Structure Tree in the Project Workspace.
2. On the **Slab Deflection** tab, click **Analyze Current** At the end of the analysis the active sub model view switches into a **Slab Deflections View** regime and the Results group is available on the ribbon - ready for reviewing the results graphically.

Run a slab deflection analysis for all sub models

To analyze all sub models:

1. On the **Slab Deflection** tab, click **Analyze All** At the end of the analysis the active view switches into a **Slab Deflections View** regime and the Results group is available on the ribbon - ready for reviewing the results graphically.

Run a slab deflection analysis for selected sub models

To analyze selected sub models:

1. On the **Slab Deflection** tab, click **Analyze Selected** At the end of the analysis the active view switches into a **Slab Deflections View** regime and the Results group is available on the ribbon - ready for reviewing the results graphically.

10.5 Slab deflection results and reports

Click the links below to find out more:

- Display slab deflection analysis results (page 818)
- Display check line results (page 820)
- Display slab deflection status and utilization (page 821)
- Slab deflection optimization (page 823)
- Slab Deflection Reports
- Slab Reinforcement (page 720)

Display slab deflection analysis results

Once a slab deflection analysis has been performed analysis results can be reviewed for the chosen level (sub-model), or the entire structure - dependent upon your chosen analysis.

The following results are available for display in the **Slab Deflections View** regime by making appropriate selections from the **Results** group of the **Slab Deflection** ribbon.

- **Deflections**.
- **Extent of Cracking** at any load event.
- **Relative Stiffness** in a particular result direction for any specified event.
- **Effective Reinforcement** for a chosen result direction for each FE element.

Display deflection contours

Three deflection types are available for review:

- **Total** deflection at the end of any event.

- **Differential** deflection between any two events (Start of Event and End of Event).

  For further details, see:
- **Instantaneous** deflection (not actually needed for TR 58).
  - This is the deflection when the entire event loading is applied to a version of the model using the established extent of cracking along with short term analysis properties.
  - US codes require an assessment of the instantaneous deflection associated with the imposed load only. This is achieved by adding extra events at the same time as the final event where only the required imposed load is applied.

To display deflection contours:

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Deflections**
2. Then from the same group:
   a. Make your selection from the **Result Direction** list
   b. Make your selection (Instantaneous, Total, or Differential) from the **Result Type** list
3. From the **Results** group select the **Event**, and if viewing differential deflections also select the **Start Event**.

   NOTE Hover the cursor over a specific solver node in order to see the nodal displacement at that location displayed in a tooltip.

Display extent of cracking

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Extent of Cracking**
2. Make your selection from the **Result Direction** list
3. From the **Results** group select the **Event**.

   NOTE Hover the cursor over a specific solver element in order to see the detailed cracking calculation parameters for that element displayed in a tooltip.

Display relative stiffness

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Relative Stiffness**
2. Make your selection from the **Result Direction** list

3. From the **Results** group select the **Event**.

NOTE Hover the cursor over a specific solver element in order to see the detailed relative stiffness calculation parameters for that element displayed in a tooltip.

Display effective reinforcement

1. From the **Slab Deflection** ribbon **Results** group **Result** list select **Effective Reinforcement**
2. Make your selection from the **Result Direction** list

NOTE Hover the cursor over a specific solver element in order to see the detailed effective reinforcement calculation parameters for that element displayed in a tooltip.

Display check line results

Once a slab deflection analysis has been performed, the Check Line results are available for display in the **Slab Deflections View** regime.

Display deflections along all check lines

1. From the **Slab Deflection** ribbon **Check Lines** group click **Deflections**

   The deflected shape of each check line and an accompanying legend is displayed in the current view.

   For clarity you may want to switch off other results to obtain a clearer view.

Display detailed deflections and average slopes along an individual check Line

1. Right click on the check line you want to view and select Open deflections check view from the context menu that is displayed.
2. Select the Result Type and the Event from the Loading Analysis toolbar.

   The deflections and average slopes are displayed in a deflections view.

Display check line status and utilization

Every check line can have up to six different deflection checks assigned to it for different events. Every check line will therefore have a Pass/Fail status and a critical Utilization ratio

Once check lines are set up and any re-analysis is undertaken i.e. due to optimization of a slab design, adding reinforcement etc. then the check lines are automatically re-checked so you will obtain instant feedback on status and utilization.

To view check line status or utilization:

1. From the **Slab Deflection** ribbon **Check Lines** group
   - Click **Status**
   - Click **Utilization**

   Color codes are used to graphically display the status or utilization of each Check Line.

Display slab deflection status and utilization

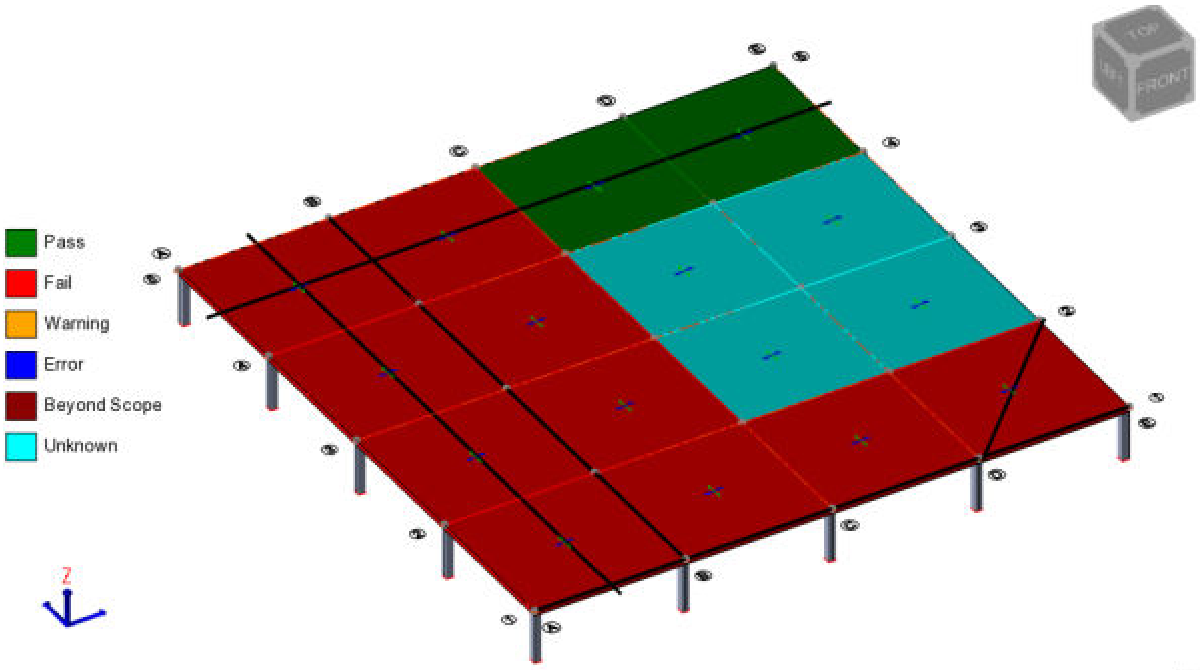
Once a slab deflection analysis has been performed, the Slab deflection status and utilization are available for display in the **Slab Deflections View** regime.

Display slab deflection status

1. From the **Slab Deflection** ribbon **Slab Deflection** group click **Status**

   Color codes are used to graphically display the status of each slab item.

In the screenshot below;



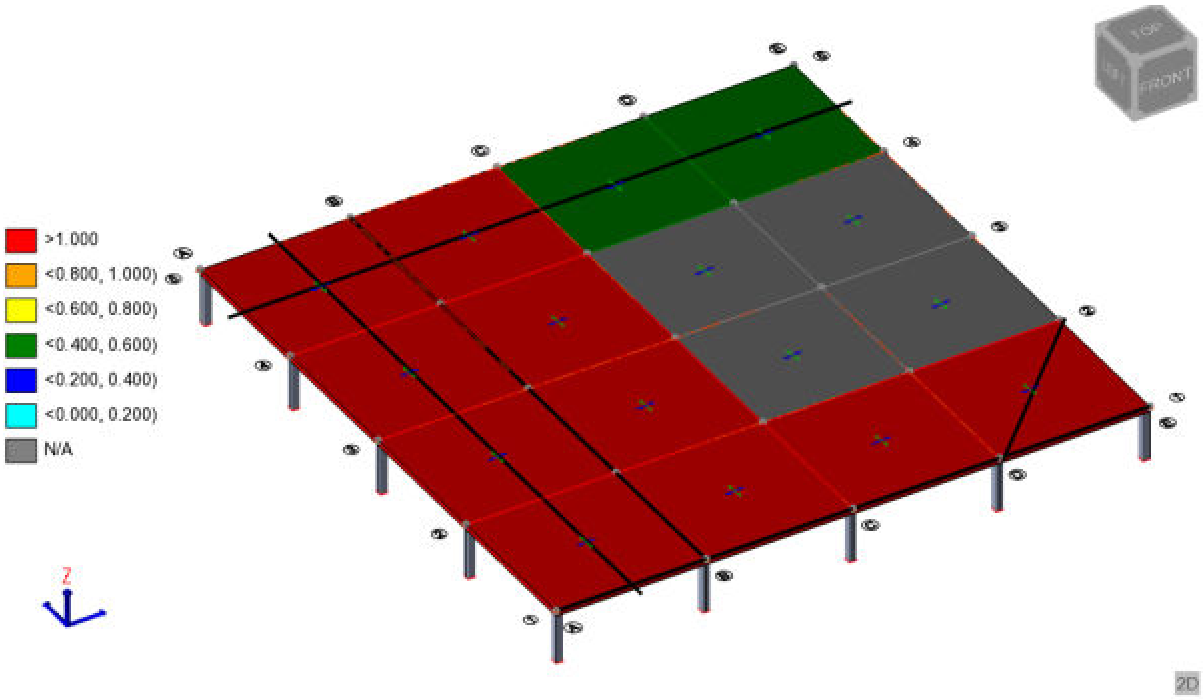
- No check lines cross the slab items between C-D / 2-3 and C-E / 3-4 so the slab reports Unknown as no checks have been performed
- One passing check line crosses the slabs between C-E / 4-5 so the slab reports a Pass.
- A fail at any point in a check line causes the status of every slab item crossed by the check line to report as a Fail. Hence, all other slab items Fail.

Display slab deflection utilization

1. From the **Slab Deflection** ribbon **Slab Deflection** group click **Utilization**

   Color codes are used to graphically display the utilization of each slab item.

The worst utilization from all associated check lines is displayed.



Slab deflection optimization

If you find that a slab either fails deflection checks (or passes with a low utilization) then changes will need to be made. There are many areas where adjustments can be made.

- Adjust slab/panel depths?
- Adjust reinforcement?
- Re-consider the Event timings and loadings?
- Adjust other settings?

Any or all changes are possible.

When changes are necessary we would recommend that it will be more efficient to work on one level at a time. The automatic re-checking of check lines will help considerably with this optimization.

View slab deflection reports

A Slab Deflection Check Lines report can be generated for multiple check lines from the Reports toolbar. The same report can also be generated for individual check lines from the right click menu.

View an individual check line report

1. Right click on the check line you want the report for and select **Report for Member** from the context menu that is displayed.

   A report is displayed consisting of a defection check summary table.

   NOTE Optionally the report can be configured to include a picture showing the Check Line location within the slab, (by clicking Member Report on the Reports ribbon and choosing the Slab Deflection Check Line as the Member Type.)

View all/multiple check line reports

1. From the **Report** tab click **Model Report...**

   A report is displayed consisting of a defection check summary table.

2. Create a report that includes the Slab Deflection Check Lines chapter (located under Concrete>Slab/Mat Design per Plane).
3. Apply Model filters to the report as required.

View an effective modulus report

A slab Effective Modulus Report can be exported to Excel for an individual slab panel, or all panels.

1. Right click on a slab panel and select **Export Eff. Modulus Report to Excel** from the context menu that is displayed.
2. Choose whether to export for the current slab item, or all slab items from the sub menu that is displayed.

   The report opens in Excel, (assuming Excel is installed).

11 Create reports and drawings

Having designed your model in Tekla Structural Designer you can then output the results in the form of reports and/or drawings.

11.1 Create and modify reports

You can tailor a wide range of different reports according to your needs by using the **Report** toolbar.

NOTE Initially, only the **Contents** group and **Filters** group are displayed on the **Report** toolbar. Only when a report is displayed and the **Report View** is active, do the **Appearance** group, **Navigation** group, and **Export** group become available.

Report terminology

The following terms are used to described specific aspects of the report creation process in Tekla Structural Designer.

Model reports

Model reports are used to set up the printed output for either the entire structure or a part of it. Model reports are configured by selecting specific output categories (referred to as *chapters*) in a list of all the available output categories. You can include entire chapters, or just the headings within the chapter you require.

The combination of selected chapters is referred to as the *report structure*. If necessary, you can apply filters to individual headings in the report structure to limit the output that is produced.

Model report chapters include:

- Structure:

  Structure data (under headings such as **Loadcases** or **Wind Data**).

- Analysis:

  Analysis model properties and results

- Concrete, Steel, Timber, Cold Formed, Cold Rolled, and General Material:

  Member reports for each different member types. Each member report can be configured to contain as much or as little output as needed. You can also decide to include design summary tables for the members that Tekla Structural Designer can design.

- Beam End Forces, Bracing Forces, and Foundation Reactions:

  Chapters for specific sets of analysis results.

- Picture:

  A 3D view of the entire structure. Applying model filters also allows you to view a picture of selected sub structures, frames, or levels. If necessary, you can also include applied loads in the picture by applying a load filter.

- View:

  The current display in any 2D or 3D view can be saved as a view configuration, and then, included in the model report as a View chapter.

- Analysis Diagram:

  A 3D diagram of the entire structure. Applying model filters also allows you to view a picture of selected sub structures, frames, or levels. If necessary, you can also display the analysis results by applying an analysis method filter.

- Material Listing:

  Tabulated quantities of materials.

- Revision History:

  The revision history that has been recorded in the **Project Wiki**.

Member reports

Member reports are only output as part of a created model report or by right-clicking on an individual member and selecting **Report for Member**.

Member report chapters include:

- Picture:

  The 3D view that you receive by right-clicking a member and selecting **Open [element name] view**.

- Drawing:

  The DXF file that you receive by right-clicking a member and selecting **Generate Detailing Drawing...**

- Loading:

  The table that you receive by right-clicking a member and selecting **Show Member Loading**.
- Different force diagrams and Deflected Shape Diagram:

  The diagrams that you receive by right-clicking a member and selecting **Open Load Analysis View**.

Active model report

The active model report is the default report displayed in the list on the left corner of the **Report** toolbar, unless another report view is already active.

To specify the active model report, do the following:

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select the desired report.
3. Click **>> Active**.

Active member report

The active member report is the report that is generated when you right-click a member in a 2D or 3D view and select **Report for Member**.

To specify the active model report, do the following:

1. On the **Report** tab, click **Member Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select the desired report.
3. Click **>> Active**.

Active and inactive chapters

In the **Report Contents** dialog box, each chapter in the report structure is marked as either *active* or *inactive*. Only the chapters that are marked as *active* are included in the report that you generate.

Report filters

You can use filters to limit the amount of output in the report.

Tekla Structural Designer offers the following filters for different data types:

- Model filters:

  If necessary, you can filter the report for selected levels, frames, or planes. You can also filter certain data types for selected beams, columns, or walls.

If you have applied user-defined attributes to your model, you can also use them to filter the report output.

- Loading filters:

  If necessary, you can filter the report for selected loadcases and combinations.

Available styles

A number of sample reports are available in the **Available Styles** list of the **Report Contents** dialog box. The reports serve as templates and can be modified to suit the model in question. If you cannot find a report that can be customized to meet your needs, you can add further reports to the list by clicking **Add**.

If you start a new project, the same default reports are available. However, they are reset to the default report structures. It is not currently possible to save customized reports from the **Report Contents** dialog box to apply them to other projects.

Create reports

Tekla Structural Designer allows you to create two kinds of reports: you can either create a report based on the entire model or an individual member in the model.

Configure and display model reports

To create model reports and display them, see the following instructions.

Configure a model report

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.

2. Select the report:

   - To select an existing report, click the report name in the **Available Styles** list.

   - To add a report, click **Add** and type the report name in the **Active Style** field.

3. Review the report structure and modify it (page 831) according to your needs.

4. To limit the output to selected levels, frames, planes, or sub structures, apply a model filter (page 832).

TIP You can further limit the output of **Loadcases** and **Combinations** sub chapters by applying a loading filter (page 832).

5. If you have created specific view configurations for the model and want to include them in the report, include a separate **View** chapter for each view configuration.
6. If the report structure includes any member chapters (such as beams, columns, or walls), select an appropriate member report style for each member chapter.
7. Click **OK**.

NOTE If you have included a drawing in the report structure, specify appropriate settings for the drawing as follows:

a. Right-click the drawing in the report structure.
b. Click **Settings...**
c. Define the drawing settings according to your needs.
d. Click **OK**.

Display a model report

1. In the list on the far left of the **Report** tab, select the report that you want to view.
2. On the **Report** tab, click **Show Report**.

   Tekla Structural Designer displays the report in a new window. If the report contains loading analysis views of individual members (such as force diagrams or pictures), the views are displayed for each member type according to the selected member report style.

See also

Export reports (page 838)

Print reports (page 838)

Format reports (page 833)

Configure and display member reports

To create member reports, see the following instructions.

Configure a member report

1. On the **Report** tab, click **Member Report...**

   The **Report Contents** dialog box opens.
2. In the **Member Type** list, select the desired member.

3. Select the report:
   - To select an existing report, click the report name in the **Available Styles** list.
   - To add a report, click **Add** and type the report name in the **Active Style** field.
4. Review the report structure and modify it (page 831) according to your needs.
5. To control the level of output, modify the report structure (page 834) or loading filters according to your needs.
6. Click **OK**.

NOTE If you have included a drawing in the report structure, specify appropriate settings for the drawing as follows:

a. Right-click the drawing in the report structure.

b. Click **Settings...**

c. Define the drawing settings according to your needs.

d. Click **OK**.

Display a member report

1. Ensure that you have selected and activated the member report style (page 830).
2. Hover the mouse pointer over the desired member.
3. Once the outline of the member is high-lighted, right-click the member.
4. In the context menu, select **Report for Member**.

   Tekla Structural Designer opens a report for the selected member in a new window.

See also

Format reports (page 833)

Export reports (page 838)

Select the member report style

Different member report styles produce different levels and types of output both in model reports and member reports. To select the member report style, see the following instructions.

Select the member report style used in a model report

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.

2. Select the desired report type.
3. Right-click the member chapter in the report structure.
4. In the context-menu, go to **Style**.
5. Click the desired member report style.

Select the member report style used in an individual member report

1. On the **Report** tab, click **Member Report...**

   The **Report Contents** dialog box opens.

2. In the **Member Type** list, select the desired member.
3. In the **Available Styles** list, select the desired member report style.
4. Click **>> Active**.

See also

Configure and display member reports (page 829)

Configure and display model reports (page 828)

Format reports (page 833)

Export reports (page 838)

Modify the report structure

Once you have selected the report type, you can modify the report structure according to your needs. For more information, see the following instructions.

To aid the report structuring process, an option is provided to display the structure as either a Flat, or a Hierarchical layout.

1. According to your needs, click either **Model Report...** or **Member Report...**

   The **Report Contents** dialog box opens.

2. In the **Report Contents** dialog box, do one or more of the following:

   - To include chapters and options in the report, drag them from the left column to the right column.
   - To remove unwanted chapters and options permanently from the report, drag them from the right column to the left column.
   - To exclude a specific chapter from the report but maintain the current report structure, right-click the chapter and clear the **Active** option.

- To re-arrange the report order, drag and drop chapters within the right column.
- To apply a filter to a chapter, right-click the desired chapter. For more information, see Filter reports (page 832).

TIP In **View Mode**, you can select if you want to display the structure as flat or hierarchical.

---

3. Click **OK**.

See also

Configure and display member reports (page 829)

Filter reports

Tekla Structural Designer contains two main categories of filters that you can apply to chapters in the report structure. The categories are model filters and loading filters. For more information, see the following paragraphs.

You can apply model filters to enable selective output based on selected:

- Levels
- Frames
- Planes
- Groups
- Members
- Trusses
- Portal frames
- UDAs

Using loading filters enables selective output based on selected:

- Loadcases
- Combinations
- Envelopes

See also

Configure and display model reports (page 828)

Create filters

1. In the **Filters** group of the **Report** tab, select the desired filter type.

   The **Edit filters** dialog box opens.

2. Click **Add**.
3. Type a name for the new filter.
4. In the **Selected items** list, select the necessary options to define the filter requirements.
5. Click **OK**.

   Tekla Structural Designer creates the filter. You can now apply the filter to a specific report.

   TIP If you create a member filter, you can later review and modify it graphically by clicking **Report Filter** on the **Review** tab.

Apply filters

NOTE If a filter type is not applicable for the selected chapter, it will be unavailable.

---

1. On the **Report** tab, click either **Model Report...** or **Member Report...**

   The **Report Contents** dialog box opens.
2. Select the report that you want to filter.

   In the right column, any chapters that can be filtered are displayed in blue text.
3. Right-click a chapter.
4. In the context menu, select the desired filter type and filter name.

   TIP If you cannot find an appropriate filter, do the following:

   a. In the context menu, go to the filter type and click **Edit\New...**
   b. In the **Select filter** dialog box, click **Add**.
   c. Type the name of the filter in the **Active filter** field.
   d. In the **Selected items** list, select the necessary options to define the filter requirements.
   e. Click **OK**.

   ---

5. Click **OK**.

   Tekla Structural Designer applies the filter to the selected report chapter.

Format reports

Tekla Structural Designer allows you to customize the appearance of reports in a number of ways. You can, for example, configure paragraphs styles and tables, and adjust header and footer information.

Adjust and apply report settings

Report settings allow you to modify the appearance of reports. You can adjust, for example, paragraph styles, page margins and numbering, tables, and headers or footers.

1. On the **Report** tab, click **Settings**.

   The **Settings** dialog box opens.

2. In the **Settings** dialog box, do one or more of the following to adjust the report settings:

To	Do this
Customize the paragraph styles in the different areas of the report	a. Go to the **Styles** page. b. Adjust the settings according to your needs.
Adjust the page margin width, page numbering, and margin frame	• Go to the **Page Options** page. • Adjust the settings according to your needs.
Adjust the appearance of tables	• Go to the **Table Options** page. • Adjust the settings according to your needs.
Define whether headers and footers are displayed, adjust image width and paragraph spacing, and control page breaks	• Go to the **Document Options** page. • Adjust the settings according to your needs.
Adjust the appearance of fonts used in pictures and force diagrams	• Go to the **Picture Fonts** page. • Adjust the settings according to your needs.

3. Click **OK**.

   Provided that you adjusted the report settings of an active settings set, the new report settings are applied to the report.

See also

Report settings (page 1168)

Adjust report headers and footers

Tekla Structural Designer provides you with default page header and footer layouts for your reports. However, you can also create, use and save different layouts according to your needs.

Enter company details in the header

1. On the **Report** tab, create the desired report and click **Show Report**.

   The report is displayed.

2. On the **Report** tab, click **Edit Header**.

   The **Document headers/footers** dialog box opens.

3. Go to the **Headers** page.
4. In the **Available layouts** list, select an existing layout that contains an **Address** cell.
5. Go to the **Fields** page.
6. In the **Available fields** list, select **Address**.
7. In the list on the right, select the necessary company details and click **<<** to include them in the report.
8. To define the text displayed in each included field, select the field in the left **Available fields** list and type the desired text in the **Displayed value** field.
9. To insert the company logo in the header, select **Company Logo** in the left **Available fields** list and click **...** to browse to the desired image.

   TIP
   - Large logos are automatically resized to within the column width in which they are placed. However, using smaller images in PNG format saves memory and speeds up exporting reports.
   - To include more images in the header, click **Add Image Field**, type a name for the field and selected the desired image.

10. Click **OK**.

    The company details are displayed in the report header.

Enter project-specific details in the header

1. On the **Home** tab, click **Project Wiki**.

   The **Project Wiki** dialog box opens.

2. On the **Project Summary** page, type the project details in the available fields.
3. Click **OK**.

   The project details are saved.

   NOTE When you go back to the **Report** tab, remember to update the report to display the project details in the header.

Create a new header or footer layout

1. On the **Report** tab, create the desired report and click **Show Report**.

   The report is displayed.

2. On the **Report** tab, click **Edit Header**.

   The **Document headers/footers** dialog box opens.
3. According to your needs, go to the **Headers** or **Footers** page.
4. In the **Available layouts** section, click **New**.
5. Type a name for the new layout in the **Name** field.
6. According to your needs, do some of the following to adjust the header layout:

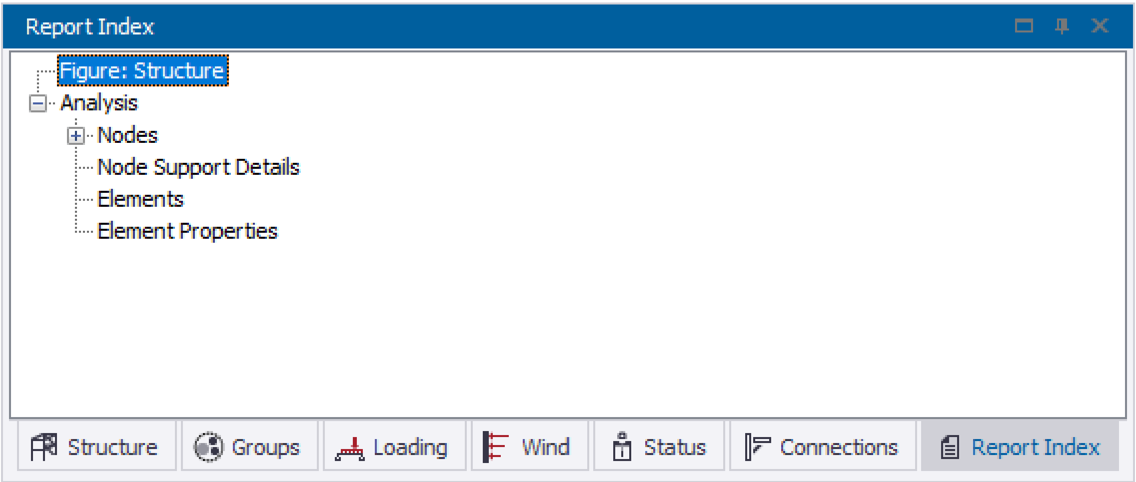
To	Do this
Remove an existing field and create an empty shell	a. Hover the mouse pointer over the field in the **Current layout** section until the field name is highlighted. b. Click the field that you want to remove it and replace it with an empty cell.
Place a field into an empty cell or replace an existing field	• Hold down the left mouse button and drag the field from the **Available fields** list into the desired cell.
Merge cells	• Hold down the left mouse button and drag over the cells that you want to merge. **NOTE** To unjoin previously merged cells, right-click the merged cell and in the context menu, select **Unjoin**.
Modify the field alignment within a shell	a. Right-click the required cell in the layout. b. In the context menu, set the horizontal and vertical alignment options as required.
Insert or remove rows or columns or change their alignment	a. Right-click the layout. b. In the context menu, select the desired command.
Change column type, width and alignment in the layout	a. Under the **Current layout** section, click **Edit...** b. Adjust the column properties according to your needs. c. Click **OK**.

7. Click **OK**.

   The new layout is created.

Navigate reports

Use the **Report Index** to quickly navigate to a specific section in a report.



You can also use the buttons in the Navigation group of the **Report** toolbar to move through the report pages.

Navigation using the Report Index

Open a report view then:

1. In the **Navigation** group of the **Report** toolbar, select **Report Index**

   The **Report Index** tab is displayed in the **Project Workspace**.
2. Click the **Report Index** tab in the **Project Workspace**.
3. Click on a specific heading or sub heading Report Index to move to that section in the report.

Navigation buttons in the Report toolbar

Open a report view then:

- Click  to move to the first page
- Click  to move to the previous page
- Click  to move to the next page
- Click  to move to the last page
- Click  to open the **Report Index** in the **Project Workspace**

Export reports

Tekla Structural Designer allows you to export your reports to PDF format, Microsoft Word, Excel, and Tekla Tedds. For detailed instructions, see the following paragraphs.

Export a report to PDF

1. Create the report and ensure that it contains the necessary information.
2. On the **Report** tab, click **PDF**.
3. Click **Yes** to confirm exporting the report.

   The report opens as a PDF file.

Export a report to Microsoft Word

1. Create the report and ensure that it contains the necessary information.
2. On the **Report** tab, click **Word**.
3. Click **Yes** to confirm exporting the report.

   The report opens in Microsoft Word.

Export a report to Excel

1. Create the report and ensure that it contains the necessary information.
2. On the **Report** tab, click **Excel**.
3. Click **Yes** to confirm exporting the report.

   The report opens in Excel

Export a report to Tekla Tedds

1. Create the report and ensure that it contains the necessary information.
2. On the **Report** tab, click **Tedds**.
3. Click **Yes** to confirm exporting the report.

   The report opens in Tekla Tedds

Print reports

To print any reports that you require, see the following instructions.

1. Create the report and click **Show Report**.

   Tekla Structural Designer displays the report.

2. On the **File** menu, go to **Print** --> **Print...**
3. Adjust the printing settings according to your needs.
4. Click **OK**.

Example reports

Tekla Structural Designer provides a number of standard reports that are installed by default. Even though the standard reports may not match your exact requirements, they can often serve as templates that you can modify according to your needs.

Beam End Forces report

The report is typically generated for steel or timber beams in order to provide connection design forces in one of the following configurations:

- End 1/End 2:

  6 forces are output in the support's local co-ordinate system for the selected loadcases, combinations, or envelopes. For each end of each beam span, a single row of data is output for loadcases, combinations, or envelopes in the selected loading filter (Fx, Fy, Fz, Mx, My, Mz). Select this option if you intend to design the connections at both ends of each beam independently.

- End 1/End 2 + Coincident:

  For each end of each beam span, a separate row is output for each loadcase, combination, or envelope according to the applied loading filter for the maximum and the minimum of each of the 6 forces that are output. In addition, the report contains the coincident forces in the other directions.

  Up to 12 rows of data can be output for a loadcase, combination, or envelope as follows:

  - maximum: Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - minimum: Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - maximum: Fy (+ coincident Fx, Fz, Mx, My, Mz)
  - minimum: Fy (+ coincident Fx, Fz, Mx, My, Mz)
  - maximum: Fz (+ coincident Fx, Fy, Mx, My, Mz)
  - minimum: Fz (+ coincident Fx, Fy, Mx, My, Mz)
  - maximum: Mx (+ coincident Fx, Fy, Fz, My, Mz)
  - minimum: Mx (+ coincident Fx, Fy, Fz, My, Mz)
  - maximum: My (+ coincident Fx, Fy, Fz, Mx, Mz)

- minimum: My (+ coincident Fx, Fy, Fz, Mx, Mz)
- maximum: Mz (+ coincident Fx, Fy, Fz, Mx, My)
- minimum: Mz (+ coincident Fx, Fy, Fz, Mx, My)

The loadcase, combination, or envelope names in which the maximum or minimum values occur are also reported.

- Higher End Only:

  For each beam span, two rows of data are output for the chosen loading filter: maximum (end 1, end 2): Fx, Fy, Fz, Mx, My, Mz; minimum (end 1, end 2): Fx, Fy, Fz, Mx, My, Mz. This configuration does not contain loadcase, combination, or envelope names because they could vary for each of the 6 values.

- Higher End Only + Coincident (initial default option):

  A separate row is output for the maximum and the minimum of each of the 6 forces, along with the coincident forces in the other directions. For each beam span, up to 12 rows of data can be output for the applied loading filter as follows:

  - maximum (end 1, end 2): Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - minimum (end 1, end 2): Fx (+ coincident Fy, Fz, Mx, My, Mz)
  - maximum (end 1, end 2): Fy (+ coincident Fx, Fz, Mx, My, Mz)
  - minimum (end 1, end 2): Fy (+ coincident Fx, Fz, Mx, My, Mz)
  - maximum (end 1, end 2): Fz (+ coincident Fx, Fy, Mx, My, Mz)
  - minimum (end 1, end 2): Fz (+ coincident Fx, Fy, Mx, My, Mz)
  - maximum (end 1, end 2): Mx (+ coincident Fx, Fy, Fz, My, Mz)
  - minimum (end 1, end 2): Mx (+ coincident Fx, Fy, Fz, My, Mz)
  - maximum (end 1, end 2): My (+ coincident Fx, Fy, Fz, Mx, Mz)
  - minimum (end 1, end 2): My (+ coincident Fx, Fy, Fz, Mx, Mz)
  - maximum (end 1, end 2): Mz (+ coincident Fx, Fy, Fz, Mx, My)
  - minimum (end 1, end 2): Mz (+ coincident Fx, Fy, Fz, Mx, My)

  The loadcase, combination, or envelope names in which the maximum or minimum values occur are also reported.

TIP To change the default **Beam End Forces** report configuration, do the following:

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.

2. In the **Available Styles** list, select **Beam End Forces**.
3. In the **Chapters and Options** list, expand the **Member End Forces** group.

4. Hold down the left mouse button and drag the desired configurations into the report structure.
5. Click **OK**.

To display a **Beam End Forces** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Beam End Forces**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Bracing Forces report

Tekla Structural Designer provides the bracing design force data for all loadcases and combinations in steel buildings.

To display a **Bracing Forces** report:

1. In the list on the left side of the **Report** toolbar, select **Bracing Forces**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Building Analysis & Drift Checks report

By default this report contains the following tables:

- Loadcase summary
- Drift check or sway check results (depending on head code)
- Wind drift check results

To display the **Building Analysis & Drift Checks** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Building Analysis & Drift Checks**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Building Design report

By default, the report contains the design results for the building at a summary level.

To display a **Building Design** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Building Design**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Building Loading report

By default, the report contains a picture of the model and the following tables of loading related input data:

- Action codes
- Resistance codes
- Combinations
- Wind data

To display a **Building Loading** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Building Loading**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Connection Resistance report

Tekla Structural Designer provides the bracing design force data for all loadcases and combinations in steel buildings.

To display a **Connection Resistance** report:

1. In the list on the left side of the **Report** toolbar, select **Connection Resistance**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Embodied Carbon report

The report summarizes the embodied carbon in the structure in two tables:

- Embodied Carbon Overview by Plane

  The plane reference, level, embodied carbon mass, floored area, and embodied carbon mass per floored area are output for each plane.

- Embodied Carbon Overview by Construction Type

  The embodied carbon mass is output each construction type.

To display an **Embodied Carbon** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Embodied Carbon**.
2. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Foundation Reactions report

The report summarizes the foundation design forces and is available in two different configurations:

- Foundation Reactions (initial default option):

  6 forces are output in the support's local co-ordinate system for the selected loadcases, combinations, or envelopes.

- Foundation Reactions + Coincident

  The maximum and minimum value of each of the 6 forces in the support's local co-ordinate system are output for the selected loading type. In addition, the report contains the coincident forces that exist in the other directions for the loading type in which the maximum or minimum value occurs. The maximum values are highlighted in red, whereas the minimum values are highlighted in blue.

  > **NOTE** The table for the Foundation Reactions + Coincident option never contains more than 12 rows, irrespective of the number of selected loadcases or combinations.

To display a **Foundation Reactions** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Foundation Reactions**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

In addition to model and loading filters, you can also reconfigure settings for each of the two configurations. These settings provide options for:

- choosing strength or service factors
- choosing to display all rows, or non-zero rows only
- excluding reactions from columns assigned to cores
- excluding reactions from walls assigned to cores

Industrial Structure Loading report

By default, this report contains tables for the following industrial structure loading data:

- Line Ancillary
- Area Ancillary
- Equipment

To display an **Industrial Structure Loading** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Industrial Structure Loading**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

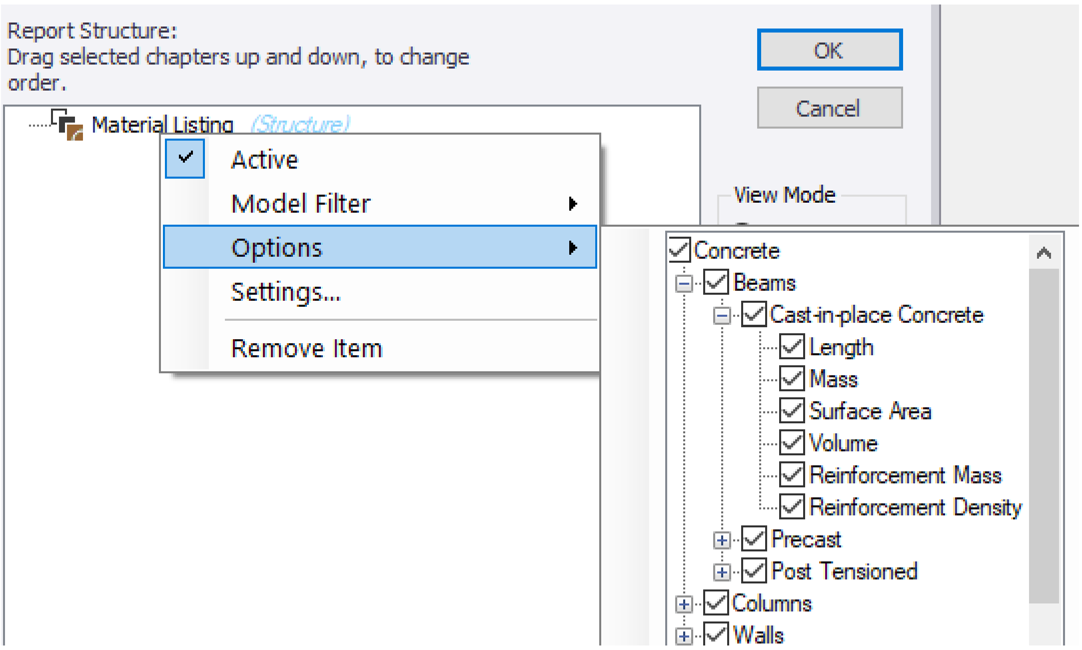
Material Listing report

To configure and then display a **Material Listing** report, do the following:

1. On the **Report** ribbon, select **Material Listing** from the list in the **Contents** group.
2. Click **Model Report...**

   The **Report Contents** dialog box opens.
3. The default report is created for all materials and entity types in the model and includes all associated properties. If required you can configure which materials/entity types are to be included in the report and also choose to exclude properties that you don't require. This is done as follows:

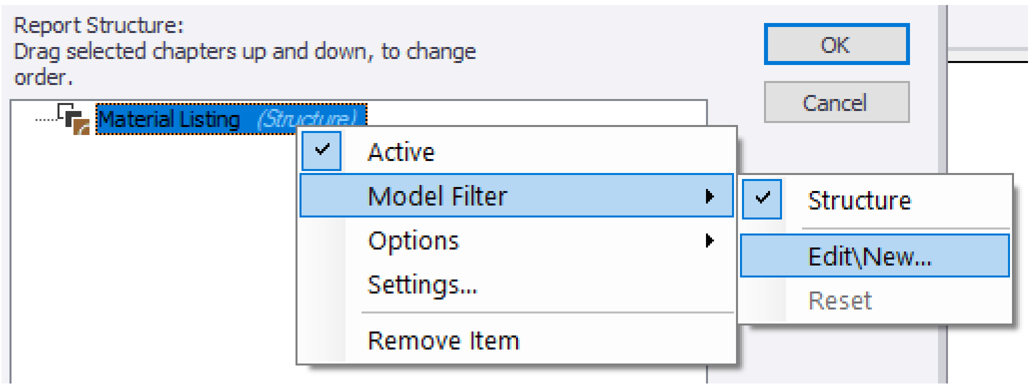
a. Right-click on the **Material Listing** chapter in the Report Structure and choose **Options** from the context menu.



b. Uncheck those materials/entity types and any properties that you don't want to include in the report.

4. The default report is created for the whole structure. If required you can configure a filter to limit the report to selected floor levels only, or other criteria as follows:

a. right-click on the **Material Listing** chapter once again and choose **Model Filter > Edit/New**



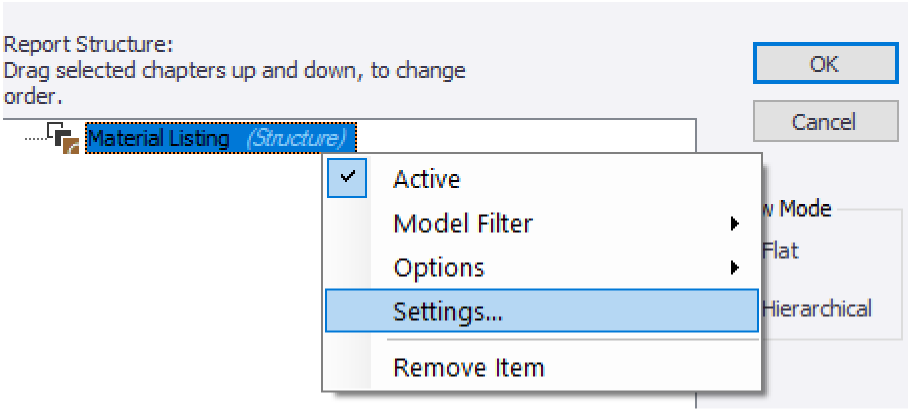
b. Click Add

c. Select the type of filter

d. Select the items to be included

e. Click OK

5. To choose between a summary or detailed report, right-click on the **Material Listing** chapter once again and choose **Settings**



- Uncheck **Show summary only** for a detailed report
- Leave it checked for a summary report

NOTE To find out exactly what summary and detailed level output is produced for each material type and characteristic, see:

- Steel material lists (page 786)
- Concrete material lists (page 792)
- Timber material lists (page 803)
- Cold formed material lists (page 805)
- General material lists (page 806)

---

6. Click **OK** to close the settings dialog.
7. Click **OK** to close the **Report Contents** dialog. The report is displayed according to the configuration choices that you have applied.

Member Design report

By default, the report contains the design results for each member at a summary level.

To display a **Member Design Calcs** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Member Design Calcs**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Open Structure Wind Load report

By default, this report contains tables of the open structure wind loads applied to the following entities:

- Member
- Line Ancillary
- Area Ancillary
- Equipment

To display an **Open Structure Wind Load** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Open Structure Wind Load**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Seismic Design report

By default, the report contain the following tables:

- Seismic loading summary
- Analysis seismic drift results

To display a **Seismic Design** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Seismic Design**.
2. Click **Show Report**.

TIP If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

Solver Model Data report

By default, the report contains a picture of the model and the following tables of analysis model input data:

- Nodes
- Node support details
- Elements

- Element members
- Element properties
- Contains to PDF

To display a **Solver Model Data** report, do the following:

1. In the list on the left side of the **Report** toolbar, select **Solver Model Data**.
2. Click **Show Report**.

> **TIP** If the resulting report displays too little or too much information, adjust the content by re-configuring the report (page 828).

See also

Configure and display member reports (page 829)

11.2 Create drawings

You can create drawings of individual levels and frames by using the commands on the **Draw** toolbar. As for single-member drawings, they are created by right-clicking the desired object and selecting the correct option in the context menu that appears. In addition, you can create drawings in batches using the **Drawing Management...** commands.

Drawing categories

In Tekla Structural Designer there are five main drawing categories. Each of the categories contains different drawing variants, and each variant has a number of specific options that allows you to configure their appearance. In addition, each variant contains a set number of drawing layers that are switched on and off according to the selected layer configuration. The appearance of each layer is controlled by the layer style. Drawing options, drawing layers, and layer styles can be managed in drawing settings.

See the different drawing categories and variants in the following table:

Category	Drawing variant	Description
Planar drawings	General arrangement	General arrangement (GA) of 2D levels and frames.
	Foundation reactions	GA that also contains support reactions in order to assist foundation design.

Category	Drawing variant	Description
	Loading plan	GA that also contains applied loads for the selected loadcase.
	Beam end forces	GA that also contains forces at the ends of steel beams in order to assist steel connection design.
	Column splice loads	Frame GA that also contains the splice loads at splice locations in steel columns in order to assist steel connection design.
Member details	Concrete beam detail	Beam reinforcement in elevation and section for each span. If necessary, you can also include a reinforcement quantity table.
	Concrete column detail	Column reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.
	Concrete wall detail	Wall reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.
	Non-concrete beam detail	Individual details for non-concrete beams.
	Non-concrete column detail	Individual details for non-concrete columns. When applicable, this can include the associated base plate detail.
	Base plate detail	Individual details for column base plates. Applicable to EC & US headcodes
Concrete member schedules	Concrete beam schedule	Tabular data created by building, by floor, or by

Category	Drawing variant	Description
		selected beams. The information shown in the schedule is based on the design groups.
	Concrete column schedule	A table containing cross sections through each stack for the selected columns. If necessary, you can also include a reinforcement quantity table.
	Concrete wall schedule	A table containing cross sections through each stack for the selected walls. If necessary, you can also include a reinforcement quantity table.
Slabs and mats	Slab/mat layout	Slab item, patch and punching shear reinforcement requirements. If necessary, you can also include a reinforcement quantity table for the reinforcement displayed with an added detailing allowance.
	Punching check detail	Punching shear reinforcement details for individual punching check locations.
Foundations	Isolated foundation detail	Individual foundation details in plan. If necessary, you can also include the detail in cross section and a reinforcement quantity table.
	Foundation layout	GA at foundation level displaying isolated bases, pile layouts, and mats. If necessary, you can also include isolated foundation details, a reinforcement quantity

Category	Drawing variant	Description
		table, and an isolated foundation schedule.

Adjust and apply drawing settings

To adjust the drawing settings of the current project or set them as default settings for future projects, see the following instructions.

Adjust drawing settings in the current project

1. On the **Draw** toolbar, click **Settings**

   The **Model Settings** dialog box opens.

2. Review and modify the drawing settings according to your needs.
3. Do one of the following:
   - To apply the changes to the current project, click **OK**.
   - To save the changes to the selected settings set, click **Save...**
   - To revert to the drawing settings specified in the selected settings set, click **Load...**

Adjust drawing settings in future projects

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.

2. Go to the **Drawings** page.
3. In the list at the top of the page, select the settings set that you want to modify.
4. Review and modify the drawing settings according to your needs.
5. To save the settings as defaults for future projects that use the selected settings set, click **OK**.

See also

Drawing settings (page 1126)

Create drawing scales

You can only create new drawing scales by accessing the **Settings** dialog box via the **Home** tab. For more information on creating scales, see the following instructions.

1. On the **Home** toolbar, click **Settings**.

2. Go to the **Drawings** page.
3. In the list at the top of the page, select the settings set that you want to modify.
4. In the **Drawing Variant** list, select the drawing category that you want to configure.
5. Type the desired scale in the **Scale** field.
6. Click **Add**.

   The new scale is added to the **Available Styles** list.
7. Click **OK**.

   TIP If necessary, you can also adjust the distance between independent drawing blocks by typing a value in the **Minimum Text Block Spacing** field. However, note that if the value is too great, text labels can end up far from the objects to which they refer.

Create, modify, or delete layer configurations

Layer configurations control which layers are displayed when the drawing is created. Tekla Structural Designer contains several default layer configurations that you can modify. If necessary, you can also create your own layer configurations.

Create a new layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Configurations** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer configurations you want to modify.
4. Click **Add**.
5. Name the new layer configuration.
6. Select or clear the layers according to your needs.
7. Click **OK**.

Modify a layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Configurations** sub page.
3. In the **Available Configurations** list, select the configuration that you want to modify.

4. Select or clear the layers according to your needs.
5. Click **OK**.

Copy a layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Configurations** sub page.
3. In the **Drawing Variant** list, select the drawing category whose properties you want to copy.
4. In the **Available Configurations** list, select the configuration that you want to copy properties from.
5. Click **Add copy...**

   The **Copy Drawing Item** dialog box opens.
6. Select the properties that you want to copy.
7. Name the copy.
8. Select or clear the layers according to your needs.
9. Click **OK**.

Delete a layer configuration

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Configurations** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer configurations you want to modify.
4. In the **Available Configurations** list, select the configuration that you want to remove.
5. Click **Remove**.

   The selected layer configuration is deleted for the selected drawing category.
6. Click **OK**.

Create, modify, or delete layer styles

Drawing styles control how a drawing is displayed, for example, what line types, fonts, and colors are used. Tekla Structural Designer contains several default layer styles that you can modify. If necessary, you can also create your own layer styles.

Create a new layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Styles** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer styles you want to modify.
4. Click **Add**.

   A new layer style is added to the **Available Styles** list.
5. Name the new layer style.
6. Adjust the layer properties according to your needs.
7. Click **OK**.

Modify an existing layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Styles** sub page.
3. In the **Available Styles** list, select the layer style that you want to modify.
4. Adjust the layer properties according to your needs.
5. Click **OK**.

Copy an existing layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.
2. Go to the **Layer Styles** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer styles you want to modify.
4. In the **Available Styles** list, select the layer style that you want to copy properties from.
5. Click **Add copy...**

   The **Copy Drawing Item** dialog box opens.
6. Select the drawing item that you want to copy.
7. Click **OK**.
8. Name the copied layer style.
9. Adjust the layer properties according to your needs.
10. Click **OK**.

Delete an existing layer style

1. On the **Draw** toolbar, click **Edit...**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to the **Layer Styles** sub page.
3. In the **Drawing Variant** list, select the drawing category whose layer styles you want to modify.
4. In the **Available Styles** list, select the layer style that you want to delete.
5. Click **Delete**.

   The selected layer configuration is deleted for the selected drawing category.

6. Click **OK**.

Create planar drawings

Planar drawings cover general arrangement drawings, beam end force drawings, column splice load drawings, foundation reaction drawings and loading plan drawings.

NOTE Before you create a drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** toolbar, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to **Settings**.
3. Click the drawing type whose options you want to review or adjust.
4. Adjust the drawing options according to your needs.
5. Click **OK**.

Create general arrangement drawings

General arrangement drawings display 2D levels and frames, and can only be created in 2D views. To create general arrangement drawings, do the following:

1. Open a 2D view displaying the part of the model that you want to include in the drawing.

   The part can be, for example, a construction level, frame, or sloped plane.

2. On the **Draw** toolbar, click **General Arrangement**.

   The **DXF Export Preferences** dialog box opens.

3. Select the layer configuration and layer style for the drawing.

4. Set the drawing scale.
5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create drawing scales (page 851)

Create beam end force drawings

Beam end force drawings are general arrangement drawings that also contain support reactions. They are typically created to assist connection design in steel structures and can only be created in 2D views. To create beam end force drawings, do the following:

NOTE Beam end forces are not displayed for concrete beams.

1. Open a 2D results view displaying the part of the model that you want to include in the drawing.

   The part can be, for example, a construction level, frame, or sloped plane.
2. In the **Loading** list, select the loadcase or combination that you want to display.
3. On the **Draw** toolbar, click **Beam End Forces**.

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.

8. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create drawing scales (page 851)

Create column splice load drawings

Column splice drawings are general arrangement drawings that also contain applied loads for the selected loadcase. Column splice drawings are typically created to assist connection design in steel structures. To create column splice load drawings, do the following:

NOTE Column splice loads are not relevant in concrete structures, and therefore, they are not displayed for concrete beams.

1. Open a frame view containing the steel frame for which you want to see the column splice loads.
2. On the **Status bar**, click  **Results View**.
3. In the **Loading** list, select the loadcase or combination that you want to display.
4. On the **Draw** toolbar, click **Column Splice Loads**.

   TIP If the **Column Splice Loads** command is not active, ensure that the current view is displayed in 2D.

   The **DXF Export Preferences** dialog box opens.
5. Select the layer configuration and layer style for the drawing.
6. Set the drawing scale.
7. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
8. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
9. Click **OK**.

The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create drawing scales (page 851)

Create foundation reaction drawings

Foundation reaction drawings are general arrangement drawings that also contain support reactions in order to assist foundation design. Foundation reaction drawings can only be created in 2D results views. To create foundation reaction drawings, do the following:

1. Open a 2D results view displaying the part of the model that you want to include in the drawing.

   Typically, the part is be the base construction level.
2. In the **Loading** list, select the loadcase or combination that you want to display.
3. On the **Draw** toolbar, click **Foundation Reactions**.

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create drawing scales (page 851)

Create loading plan drawings

Loading plan drawings are general arrangement drawings that also contain applied loads for the selected loadcase. Loading plan drawings can only be created in 2D scene views. To create loading plan drawings, do the following:

1. Open a 2D view displaying the part of the model that you want to include in the drawing.

   The part can be, for example, a construction level, frame, or sloped plane.

2. In the **Loading** list, select the loadcase or combination that you want to display.
3. On the **Draw** toolbar, click **Loading Plan**.

   The **DXF Export Preferences** dialog box opens.

4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create drawing scales (page 851)

Create member detail drawings

Member detail drawings include concrete beam detail drawings, concrete column detail drawings, concrete wall detail drawings, non-concrete beam detail drawings and non-concrete column detail drawings. For instructions on how to create different member detail drawings, see the following paragraphs.

NOTE Before you create a drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to **Settings** --> **Member Details**.
3. Click the drawing type whose options you want to review or adjust.
4. Adjust the drawing options according to your needs.
5. Click **OK**.

See also

Create drawing scales (page 851)

Create concrete beam detail drawing

Concrete beam drawings display the beam reinforcement in elevation and section for each span. If necessary, you can also include a reinforcement quantity table.

1. Hover the mouse pointer over the beam that you want to detail.
2. When the desired beam is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete column detail drawing

Concrete column drawings display the column reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.

1. Hover the mouse pointer over the column that you want to detail.
2. When the desired column is highlighted, right-click it.

3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete wall detail drawing

Concrete wall details drawings display the wall reinforcement in elevation and section. If necessary, you can also include a reinforcement quantity table.

1. Hover the mouse pointer over the wall that you want to detail.
2. When the desired wall is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create non-concrete beam detail drawing

Non-concrete beam detail drawings display individual steel beam details.

1. Hover the mouse pointer over the beam that you want to detail.
2. When the desired beam is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create non-concrete column detail drawing

Non-concrete column detail drawings display individual steel column details.

1. Hover the mouse pointer over the column that you want to detail.
2. When the desired column is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create base plate detail drawing

Base plate detail drawings display individual column base plate details.

1. Hover the mouse pointer over the base plate that you want to detail.
2. When the desired base plate column is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create slab and mat drawings

Slab and mat drawings refer to slab and mat layout drawings and punching shear check detail drawings. Slab and mat detail drawings are used to convey slab reinforcement and patch reinforcement requirements, and contain a quantity table for the reinforcement displayed with a detailing allowance added. Punching check detail drawings display an individual check detail in, and also contain the option to display the reinforcement quantities table.

See also

Create drawing scales (page 851)

Create slab or mat layout drawings

1. Open a 2D scene view displaying the slabs or mats that you want to include in the drawing.
2. On the **Draw** tab, click **Slab/Mat Detailing**.

   NOTE If the **Slab/Mat Detailing** command is not active, ensure that you are in a 2D scene view.

   The **DXF Export Preferences** dialog box opens.
3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.
5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create punching shear check detail drawings

1. Hover the mouse pointer over the punching shear check item that you want to detail.
2. When the desired punching shear check item is highlighted, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create foundation drawings

Foundation drawings refer to isolated foundation detail drawings and foundation layout drawings. Isolated foundation detail drawings display an individual foundation detail in plan. If necessary, you can also display the detail in cross section and a reinforcement quantity table. As for foundation layout drawings, they display the piling layout and the layout of isolated foundations. If necessary, you can also display isolated foundation details, a reinforcement quatity table, an isolated foundation schedule, and a pile location table.

See also

Create drawing scales (page 851)

Create isolated foundation detail drawings

NOTE Before you create a base cap detail drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to **Settings** --> **Isolated Foundation Detail**.
3. Adjust the drawing options on the **Content** and **Isolated Foundation Detail** sub pages according to your needs.
4. Click **OK**.

1. Hover the mouse pointer over the base cap or pile cap that you want to detail.
2. When the desired base cap or pile cap is highlighter, right-click it.
3. In the context menu, select **Generate Detailing Drawing...**

   The **DXF Export Preferences** dialog box opens.

4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.

7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create foundation layout drawings

1. Open a 2D scene view displaying the piles, bases or pile caps that you want to include in the drawing.
2. On the **Draw** tab, click **Foundation Layout**.

   NOTE If the command is not active, ensure that you are in a 2D scene view displayed in 2D.

   The **DXF Export Preferences** dialog box opens.
3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.
5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
7. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete member schedule drawings

Concrete member schedule drawings include concrete beam schedules, concrete column schedules and concrete wall schedules. In beam schedule drawings, the reinforcement is listed on a span by span basis, and contractors or specialist detailing firms can use the information to produce the necessary bar bending schedules. As for concrete column schedule drawings, they display a cross section through each stack of the selected columns, and if necessary, allow you to include a reinforcement quantity table. Concrete wall schedule drawings display a cross section through each stack of the selected walls, and if necessary, also allow you to include a reinforcement quantity table

NOTE Before you create a schedule drawing, ensure that the current drawing options meet your needs by doing the following:

1. On the **Draw** tab, click **Settings**

   The **Model Settings** dialog box opens on the **Drawings** page.

2. Go to **Settings**.
3. Adjust the drawing options on the sub pages according to your needs.
4. Click **OK**.

See also

Create drawing scales (page 851)

Create concrete beam schedule drawings

You can create beam schedule drawings by building, by floor or by selected beams. The information shown in the schedule is based on the different design groups. Beam schedule drawings are created in dxf format even though they do not include graphical information, so that you can add them to beam detail drawings.

1. Open a 2D scene view of a part displaying the beams that you want to include in a schedule. The part can be, for example, a construction level, frame, or sub model.
2. On the **Draw** tab, click **Beam Schedule**.

   NOTE If the **Beam Schedule** command is not active, ensure that you are in a 2D scene view that is displayed in 2D.

   The **DXF Export Preferences** dialog box opens.

3. Select the layer configuration and layer style for the drawing.
4. Set the drawing scale.
5. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
6. Do one of the following:
   - Accept the automatic file name.

- Clear the **Use automatic file name** option and name the file yourself.

7. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete column schedule drawings

1. Open a 3D view or a frame view displaying the columns that you want to include in the schedule.
2. On the **Draw** tab, click **Column Schedule**.

   The **Select column schedule content** dialog box opens.
3. Select the columns to be included and click **OK**.

   The **DXF Export Preferences** dialog box opens.
4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Create concrete wall schedule drawings

1. Open a 3D view or a frame view displaying the walls that you want to include in the schedule.
2. On the **Draw** tab, click **Wall Schedule**.

   The **Select wall schedule content** dialog box opens.
3. Select the walls to be included and click **OK**.

   The **DXF Export Preferences** dialog box opens.

4. Select the layer configuration and layer style for the drawing.
5. Set the drawing scale.
6. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
7. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
8. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Manage drawings in batches

Although you can create individual drawings according to your needs, creating a batch of drawings at once is often more efficient, especially when working with larger models.

The **Drawing Management...** command on the **Draw** tab allows you to:

- Select the drawing variant that you want to manage.
- Either add a drawing sheet manually or generate a series of drawing sheets.
- Select the frames, levels, or members for which you want to create drawings.
- Arrange the drawings on the drawing sheet either in a linear or a grid arrangement.
- Select the loadcases or combinations for load-dependent drawing variants.
- Create drawing revisions.
- View the revision history of drawings.
- Reset reinforcement marks on concrete detail drawings in order to remove gaps in the bar mark numbering.

Create or generate drawings in batches

To easily create multiple drawings of any particular variant , be they base plate details, punching check details, concrete member details, see the following instructions:

To place multiple drawings of a single variant, (base plate detail, punching check detail, concrete member detail etc.) on to one or more drawing sheets, see the following instructions:

See also

Specify the drawing layout (page 871)

Specify the loading for load-dependent drawings (page 871)

View drawings (page 872)

Review drawings (page 873)

Reset reinforcement marks in concrete detail drawings (page 872)

Create a new batch drawing manually

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. Click **Add**.

   A new drawing appears in the **Available Drawings** list.
4. Name the new drawing.
5. Click **Content...**

   The **Drawing Content** dialog box opens.
6. Drag the items that you want to include in the drawing from the left column to the right column.
7. Click **OK**.
8. Click **View Drawing...**

Generate a new concrete beam or column detail drawing

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.
2. In the **Drawing Variant** list, select either **Concrete Beam Detail** or **Concrete Column Detail**.

3. Click **Generate**.

   New drawings are created with automatically generated content. One drawing contains typical beams or columns, wheras additional drawings can contain any ungrouped beams or columns.

4. If necessary, click the available drawings to rename them, and type a new name.
5. Click **View Drawing...**
6. Select the drawing you want to create from the available drawings list.
7. Click **View Drawing...**

Specify the drawing layout

To specify the layout of the drawings that you have created, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Layout...**

   The **Drawing Layout** dialog box opens.

5. Define the direction and arrangement of the layout.
6. Click **OK**.

See also

Specify the loading for load-dependent drawings (page 871)

Specify the loading for load-dependent drawings

To specify the loading for load-dependent drawings, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Loading...**

   The **Select loading** dialog box opens.

5. Select the desired loadcases and combinations.
6. Click **OK**.

See also

Create or generate drawings in batches (page 869)

Reset reinforcement marks in concrete detail drawings

Each bar geometry used in the model has an associated reinforcement mark. However, if a model is designed and some of the bars fall out of use, the mark assignment is still retained. As a result, there may be gaps in the marks and marks starting at high numbers. To avoid this, you can reset all the reinforcement marks in the model. Do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.

2. In the **Drawing Variant** list, select either **Concrete Beam Detail** or **Concrete Column Detail**.

3. Click **Reset ALL Marks**.

   All reinforcement marks are reset.

See also

Create or generate drawings in batches (page 869)

View drawings

To view the drawings that you have created in the **Drawing Management** dialog box, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.

3. In the **Available Drawings** list, select the desired drawing.

4. Click **View Drawing...**

   The **DXF Export Preferences** dialog box opens.

5. Select the layer configuration and layer style for the drawing.

6. Set the drawing scale.

7. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.

8. Do one of the following:

   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.

9. Click **OK**.

The drawing opens in an available DXF application.

NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

See also

Create or generate drawings in batches (page 869)

Review drawings

To apply a revision to a drawing, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **Create Revision...**

   The **Create Drawing Revision** dialog box opens.

5. Name the revision and type a revision note.
6. Click **OK**.

   The drawings open in an available DXF tool.

See also

View the revision history of drawings (page 873)

View the revision history of drawings

To view the formerly applied revisions to a drawing, do the following:

1. On the **Draw** tab, click **Drawing Management...**

   The **Drawing Management** dialog box opens.

2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **History...**

   The **Drawing History** dialog box opens and allows you to view the revision history of the selected drawing.

See also

Review drawings (page 873)

Manage schedule drawings in batches

Although you can create individual schedule drawings according to your needs, creating a batch of schedule drawings at once is also possible. For more information, see the following instructions.

The **Schedule Management...** command on the **Draw** tab allows you to:

- Select the drawing variant that you want to manage.
- Create new drawing sheets.
- Select the members that you want to include in the drawing.
- Arrange the drawing layout.
- Create drawing revisions.
- Reset reinforcement marks in order to remove gaps in the bar mark numbering.

Create new schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. Click **Add**.

   A new drawing appears in the **Available Drawings** list.
4. Name the new drawing.
5. Click **Content...**

   The **Drawing Content** dialog box opens.
6. Drag the items that you want to include in the drawing from the left column to the right column.
7. Click **OK**.

Specify the schedule drawing layout

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. Click **Layout...**

   The **Drawing Layout** dialog box opens.
3. Define the direction and arrangement of the layout.
4. Click **OK**.

View schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **View Drawing...**

   The **DXF Export Preferences** dialog box opens.
5. Select the layer configuration and layer style for the drawing.
6. Set the drawing scale.
7. If necessary, to adjust the distance between independent lines of text, modify the minimum text block spacing.
8. Do one of the following:
   - Accept the automatic file name.
   - Clear the **Use automatic file name** option and name the file yourself.
9. Click **OK**.

   The drawing opens in an available DXF application.

   NOTE If preferred this drawing variant can be created as a batch (page 870), (allowing multiple drawings to be created on a single drawing sheet).

Reset reinforcement marks in schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Drawing Variant** list, select **Concrete Beam Schedule**, **Concrete Column Schedule**, or **Concrete Wall Schedule**.
3. Click **Reset ALL Marks**.

   All reinforcement marks are reset.

Review schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.

4. Click **Create Revision...**

   The **Create Drawing Revision** dialog box opens.
5. Name the revision and type a revision note.
6. Click **OK**.

   The drawings open in an available DXF tool.

View the revision history of schedule drawings

1. On the **Draw** tab, click **Schedule Management...**

   The **Schedule Management** dialog box opens.
2. In the **Available Styles** list, select the desired drawing category.
3. In the **Available Drawings** list, select the desired drawing.
4. Click **History...**

   The **Drawing History** dialog box opens and allows you to view the revision history of the selected drawing.

12 Manage models

Click the links below to find out more various model management tasks:

- Define and modify head codes and design codes (page 878)
- Define and modify units (page 879)
- Manage object references (page 881)
- Create and manage user-defined attributes (page 916)
- Manage settings sets (page 886)
- Manage materials (page 890)
- Manage properties and property sets (page 912)
- Manage sub structures (page 921)
- Working with large models (page 927)

See also

Model Settings (page 1032)

12.1 Apply and manage model settings

To apply and manage the various defaults and settings that apply to the current project, see the following instructions.

1. On the **Home** tab, click **Model Settings**.
2. Adjust the Model Settings (page 1032) according to your needs.
3. After adjusting the settings, do one of the following:

To	Do this
Apply the changes to the current project	• Click **OK**.
Save the changes back to the active settings set for future use	• Click **Save...**
Cancel the changes	• Click **Cancel**.

Revert to the model settings specified in the active settings set	• Click **Load...** **NOTE** Clicking **Load...** loads all of the model settings from the active settings set, not only those specified on the current page of the **Model Settings** dialog box. Analysis options and design options are not loaded.

See also

Define and modify head codes and design codes (page 878)
Define and modify units (page 879)
Manage object references (page 881)
Create and manage user-defined attributes (page 916)

Define and modify head codes and design codes

Tekla Structural Designer allows you to select from a range of international design codes of practice. Each new project initially adopts the codes that have been specified in the active settings set. However, you can also change the codes in the middle of a project.

See also

Design code settings (page 1033)

Change design codes in an existing project

WARNING If you change the head code in an existing project, the following will occur in the model:

- Some materials, steel sections, studs, decks and reinforcement may require re-selecting in the model to make them consistent with the new head code/unit system.
- Wind loading (if any) and wall/roof panel properties will be deleted. The wind wizard will need rerunning, wall/roof panel properties need resetting and the wind loadcases will need recreating.
- Seismic loading (if any) will be deleted. The seismic wizard will need re-running.
- All combinations will be deleted.

1. On the **Home** tab, click **Model Settings**.

   The **Settings** dialog box opens.
2. Go to **Design Codes**.
3. Select the head code as required.

   The head code controls which action and resistance codes are available for selection.
4. Set the action and resistance codes according to your needs.
5. Click **OK**.

Define default design codes for new projects

1. On the **Home** tab, click **Settings**.

   The **Settings** dialog box opens.
2. Select a suitable settings set, and click **>> Active** to make it active.
3. Go to **Design Codes**.
4. Check that the head code is set as required.

   The head code controls which action and resistance codes are available for selection.
5. Set the action and resistance codes according to your needs.
6. Click **OK**.

   Tekla Structural Designer retains the selected codes to apply them for each new project until you decide to modify them.
7. To use the set design codes in a new project, on the **Home** tab, click **New**. A new project opens with its design codes, (and all of its other model settings) copied from the active settings set.

Define and modify units

Tekla Structural Designer allows you to switch between Metric and US customary units. Furthermore, you can select the units which you want to use in the selected unit system. For example, if you use US customary units, you can input forces in either kip or lb, and lengths in feet, inches, or feet and inches.

Change units and unit precision in an existing project

1. On the **Home** tab, click **Model Settings**.

   The **Model Settings** dialog box opens.
2. Go to **Units**.
3. Select the unit system.
4. Set the units and unit precision that you want Tekla Structural Designer to use.
5. Click **OK**.

NOTE The length unit can be set appropriate for the type of dimension being input:

- **Fine Dimension** units are used for defining stud spacings, section size constraints, and other typically small distances.
- **Dimension** units are used for defining grid spacings, positioning members, positioning load locations, and so on. They are also used to control the accuracy of any measured dimension lines that you apply to the model.
- **Deflection** units are used for reporting deflection results.
- **Distance** units are used for defining large dimensions.

Once you have set the units according to your needs, Tekla Structural Designer will then expect input in the same format.

Example

If the **Dimension** units are set to **ft, in fract.**:

- To input a dimension of two feet, six and one quarter inches, you need to type `2' 6 1/4"`
- To input a series of irregular grid lines in the grid wizard at spacings of fifteen feet, followed by three spacings of twenty feet, six and one half inches, followed by one spacing of fifteen feet, you need to type **`15',3*20' 6 1/2",15'`**

Define the default units and unit precision for new projects

1. On the **Home** tab, click **Settings**.

   The **Settings** dialog box opens.
2. Select a suitable settings set, and click **>> Active** to make the set active.
3. Go to **Units**.

4. Select the unit system.
5. Set the units and unit precision that you want Tekla Structural Designer to use.

> **NOTE** The length unit can be set appropriate for the type of dimension being input:
>
> - **Fine Dimension** units are used for defining stud spacings, section size constraints, and other typically small distances.
> - **Dimension** units are used for defining grid spacings, positioning members, positioning load locations, and so on. They are also used to control the accuracy of any measured dimension lines that you apply to the model.
> - **Deflection** units are used for reporting deflection results.
> - **Distance** units are used for defining large dimensions.

6. Click **OK**.
7. To use the set units in a new project, on the **Home** tab, click  **New**. A new project opens with its units, (and all of its other model settings) copied from the active settings set.

Once you have set the units according to your needs, Tekla Structural Designer will then expect input in the same format.

Example

If the **Dimension** units are set to **ft, in fract.**:

- To input a dimension of two feet, six and one quarter inches, you need to type `2' 6 1/4"`
- To input a series of irregular grid lines in the grid wizard at spacings of fifteen feet, followed by three spacings of twenty feet, six and one half inches, followed by one spacing of fifteen feet, you need to type **`15',3*20' 6 1/2",15'`**

Manage object references

Tekla Structural Designer contains a flexible object referencing system that is specifically designed considering the use of multiple materials within the same model.

You can define the object references to meet your needs on the **References** page of the **Settings** dialog box.

See also

Object reference settings (page 1035)

Basics of object reference formats

Object reference formats are user definable. Each object type has its own reference format that has been built using individual components. You can customize the reference format by adding or removing components as required.

See some of the typical components in the following table:

Icon	Item	Further information
M	Material	Fully user-definable text. For example: • `S` for Steel • `C` for Concrete
C	Characteristic	Fully user-definable text. For example: • `B` for Beam • `C` for Column
L1	Start Level Reference	Not applicable
L2	End Level Reference	
P1	Start Point Reference	Grid or construction point reference. The start and end reference points P1 and P2 work from the grids that you define, and from the construction points that have been created automatically when you place members that do not lie between existing points. **NOTE** When reference points are constructed using two grid reference points, you must always use a separator. For example, type `A/11` to avoid confusion with A1 and 1.
P2	End Point Reference	
D	Direction	Applies for beams only.
X	Count	Tekla Structural Designer keeps a separate count for each level for each object type.

Icon	Item	Further information
		NOTE Tekla Structural Designer does not keep a count by material. For example, steel beams and concrete beams are included in the same count on a level. If the direction prefix is specified, Tekla Structural Designer keeps separate counts for each direction. Direction 1 is defined as objects falling within ± 45 degrees of the horizontal, and Direction 2 ± 45 degrees of the vertical of the global axis.
Not applicable	Reinforcement > Bar Layer: • T1 • T2 • B1 • B2	• T1 is the top surface reinforcement in the 1st (outer) layer. • T2 is the top surface reinforcement in the 2nd (inner) layer. • B1 is the bottom surface reinforcement in the 1st (outer) layer. • B2 is the bottom surface reinforcement in the 2nd (inner) layer.
	Reinforcement > Bar Size	Not applicable
	Reinforcement > Center Spacing	The center spacing between bars.
	Reinforcement > Detailing prefix	An item that is typically added in front of the bar size.
	Custom text	Fixed text that you can include at any position within the reference. For example: • `Block C`
	Separators • backslash • dash • slash	Optional items that you can place between items.

Icon	Item	Further information
	• space • times	

Note that:

- When you create objects of a certain type, the default references from the active settings set are applied.
- You can modify the syntax of the reference format at any time. Objects created after the changes will adopt the new format.
- Once objects have been created, you can edit their references on an object-by-object basis, so that they can be further individualized.
- For members, the object reference does not include the group reference, geometric shape, and section size as part of the reference descriptor. Instead, options are provided in the **Scene Content** settings and the GA drawing's control to show the information.

Modify reference formats and texts in an existing project

1. On the **Home** tab, click **Model Settings**.

   The **Model Settings** dialog box opens.
2. Go to **References**.
3. Adjust the way in which the references are applied.
4. Click **OK**.

Modify the reference format syntax of an object type

1. Go to **References** --> **Formats** .
2. Click the **...** button on the right side of the desired reference format.
3. Do one of the following:

To	Do this
Add an extra item to the reference format	• Click **Add...** and select the desired item. The selected item appears at the end of the reference format.
Re-order the items in the reference format	a. Select the item you want to move. b. Drag the item to reposition it.
Add custom text to the reference format	a. Click **Add...** and select **Custom Text**. b. Drag the new component to the desired position. c. Click the **Custom Text** component to edit it. d. Type the desired text in the box. e. Click **Set**.

Remove an item from the reference format	a.	Select the item you want to remove.
	b.	Drag the item outside the **Edit Reference Format** dialog box.

4. Click **OK**.

Change the text used for the materials and characteristics in the reference format

1. Go to **References** --> **Texts** .
2. According to your needs, go to the **Characteristics** or **Materials** tab.
3. Modify the desired text according to your needs.
4. Click **OK**.

Renumber members

You can use the **Renumber** command to simultaneously renumber all member types in the model whose reference format includes the count item.

Member types are initially numbered in the order in which they are created. Renumbering makes the members easier to find in the model and on drawings.

By default, the **Renumber** command works from the lowest plane or level upwards. The count starts at 1 and continues sequentially.

NOTE To change the renumbering direction and the starting value, go to **Model Settings** --> **References** --> **General** .

1. On the **Structure** tab of **Project Workspace**, right-click the **Members** branch.
2. In the context menu, select **Renumber**.

   Tekla Structural Designer automatically renumbers all members in the model that include a count in their reference format.

Renumber slabs

You can use the **Renumber** command to simultaneously renumber all slab items in the model.

Slab items are initially numbered in the order in which they are created. Renumbering makes them easier to find in the model and on drawings.

By default, the **Renumber** command works from the lowest plane or level upwards. The count starts at 1 and continues sequentially.

NOTE To change the renumbering direction and the starting value, go to **Model Settings** --> **References** --> **General** .

1. On the **Structure** tab of **Project Workspace**, right-click the **Slabs** branch.
2. In the context menu, select **Renumber**.

   Tekla Structural Designer automatically renumbers all slab items in the number.

Adjust the default references to be applied to new projects

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select a suitable settings set in the list, and click **>> Active** to make it active.
4. Go to **References**.
5. Adjust the way in which the references are applied to each of the object types in your projects.
6. Click **OK**.
7. To start a new project using the new references, on the **Home** tab, click **New**.

12.2 Manage settings sets

The first time Tekla Structural Designer is run after installation you are required to select a country or region. Based on this selection, Tekla Structural Designer creates an initial settings set (the 'active' set), containing defaults for design codes, units, and sections, and settings for drawings, schedules, and reports.

You can edit the settings in this set and also create new sets, but only one set can be designated the 'active' settings set.

Each new project takes a copy of the 'active' settings set to form its own independent model settings.

Model settings can then be edited as required without affecting the active settings set.

Add a new settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.

3. Click **Add Copy**.
4. In the **Settings** dialog box, modify the settings for the new settings sets.
5. Click **OK** to save the settings to the new settings set.

Import a settings set for a different region

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Click **Import...**
4. Select the region for which you want to import the settings.
5. Click **OK**.

NOTE To use these settings in your new projects, you will need to make this the **active set**.

Edit the content of a settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the settings set you want to edit.
4. In the **Settings** dialog box, modify the settings according to your needs. For example:
   - Modify the font used to display results in the **Results Viewer**.
   - Modify the appearance of reports.
   - Set the units that you want to use, and set the desired precision for the units.
   - Set the design codes that you want.
   - Set default section sizes for each of the member types.
   - Set if confirmations are required for specific actions.
   - Set element references.
   - Modify the appearance of schedules.
   - Modify drawing types and styles.
   - Modify the colors used in 2D and 3D views.

5. Click **OK** to save the settings to the selected settings set.

Change the active settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the settings set required from the drop list.
4. To make the set a default set for new projects, click **>> Active**.
5. Click **OK**.

Delete a settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the set you want to delete.
4. Click **Remove**.

Load settings from the active settings set to the current project

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Ensure that the required settings set is active. If it is not, click **>> Active**.
4. Click **OK**.
5. Open the dialog that contains the settings to be updated, either:

   - On the **Home** ribbon, click **Model Settings**
   - On the **Home** ribbon, click **Embodied Carbon Factors**
   - Or, on the **Analyze** ribbon, click **Settings**

- Or, on the **Design** ribbon, click **Settings**
- Or, on the **Draw** ribbon, click **Settings**

6. Click **Load...**

   A confirmation dialog appears.
7. To update the model settings to match the active settings set, click **Yes**.

Save settings from the current project to the active settings set

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Ensure that the settings set is active. If it is not, click **>> Active**.
4. Click **OK**.
5. Open the dialog that contains the settings to be saved.
   - On the **Home** ribbon, click **Model Settings**
   - Or, on the **Home** ribbon, click **Embodied Carbon Factors**
   - Or, on the **Analyze** ribbon, click **Settings**
   - Or, on the **Design** ribbon, click **Settings**
   - Or, on the **Draw** ribbon, click **Settings**
6. Click **Save...**

   A confirmation dialog appears.
7. To update the active settings set to match the current settings, click **Yes**.

Copy a settings set from one computer to another

In order to apply consistent settings on a company wide basis, the xml file of the appropriate settings set should be manually copied to other computers.

1. On the computer containing the settings set, go to the **Home** ribbon and click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Click **Open Folder**.
4. Select the xml file of the settings set, and manually transfer it to the **Settings** folder on each destination computer.
5. To view and use the new settings set, restart Tekla Structural Designer on the destination computers.

See also

Settings set settings (page 1160)

12.3 Manage materials

Material databases contain an extensive range of sections, materials, reinforcement, decking, and connectors for each head code and country. The initial data in the material databases is protected, so that standard items cannot be accidentally edited or deleted. However, you can add user data from other sources or suppliers, if needed. All user data is fully editable. You can view the protected data and manage the user data from the Materials dialog (page 1201).

Embodied carbon factors are saved separately and are fully editable. You can add and manage these factors from the Embodied Carbon Factors dialog (page 1185).

- Add, modify and delete user-defined sections (page 890)
- Manage design section orders (page 892)
- Add simple connection resistances to the database (page 895)
- Add material properties from the model to a material database (page 902)
- Add materials for a head code (page 902)
- Timber property assumptions (page 907)
- Upgrade material databases (page 906)
- Add and manage embodied carbon factors (page 907)

Add, modify and delete user-defined sections

A user defined section can take the form of a custom, or a compound section, a compound section being one that comprises of 2 chords or more connected by battens or lattice or welded.

See also

The Sections dialog box (page 1208)

Add a user-defined custom or compound section to the material database

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to **Sections**, and click **Manage Sections**.

   The **Sections** dialog box opens.
3. In the country list, select the database country.
4. In the **Page** pane on the left, select the desired geometry.

   NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.

5. Click **Add...**
6. Enter values for each of the requested variables.
7. Click **OK**.

The new section size is now displayed in the **Item** pane.

Modify a user-defined custom or compound section in the material database

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to **Sections**, and click **Manage Sections**.

   The **Sections** dialog box opens.
3. In the country list, select the database country.
4. In the **Page** pane on the left, select the desired geometry.

   NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.

5. In the **Item** pane on the right, select the desired section size.

6. Click **Edit...**
7. Modify the section properties according to your needs.
8. Click **OK**.

Delete a user-defined custom or compound section from the database

NOTE Only user-defined sections, marked with *, can be deleted.

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to **Sections**, and click **Manage Sections**.

   The **Sections** dialog box opens.
3. In the country list, select the database country.
4. In the **Page** pane on the left, select the desired geometry.

   NOTE If you cannot find the desired shape, ensure that the **Geometry filter** does not filter it out.
5. In the **Item** pane on the right, select the desired section size.
6. Click **Delete**.

   A confirmation dialog appears.
7. To delete the section, click **Yes**.

Manage design section orders

A design section order is a list of sections you wish to consider for design when the Autodesign property is checked.

The design process commences by starting with the first section in the chosen order file. Any section that fails any of the design conditions is rejected and the design process is then repeated for the next available section in the list.

On completion of the design process, the first satisfactory section from the section designation list is assigned to the member.

View the list of sections in a design section order

You can view the list of sections in a design section order, by following the steps below.

1. Edit the properties of the member.
2. Click the **Design section order** drop list and select **<New\Edit...>**

3. Choose a section order from the available list and then click **Edit...**

The sections contained within the chosen order file appear in the **Sections in use** list on the right of the page.

Specify that a section in the list should not be considered for design

You can control which sections in a design order list are considered by following the procedure below.

Only checked sections within the list are considered during the design process. Uncheck a section and it will no longer be considered.

WARNING	Limiting the choice of sections by unchecking a section within an order file is a global change that affects ALL projects, (not just the currently open one). It is typically used to eliminate unavailable or non-preferred sections from the design process. If design requirements for an individual member require section sizes to be constrained, (due to, for example depth restrictions), then the choice of sections should be limited instead by using Size Constraints, (as these only affect the current member).

Sort the listed sections by a different property

You can sort a design section order list by following the steps below.

While viewing the list of sections:

1. Click **Criteria...** to open a dialog for selecting the sort critera.
2. Select a property from the **Available Criteria** list and click **Add**.
3. Choose the order for sorting (Ascending, or Descending).
4. Add further criteria as required.
5. Click **OK** to close the dialog.
6. Click **Sort** to re-order by the chosen criteria.
7. Having sorted, if you don't want to subsequently move individual sections up or down the list, check **Keep sorted** to de-activate **Move Up** and **Move Down**.

NOTE	Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

Specify that a section is non-preferred

Some sections might be more expensive or difficult to obtain; you might therefore want other sections to be chosen in preference to them, (whilst still keeping them available).

You can achieve this by moving the "non-preferred" sections further down the design order list.

To move a section up or down the list:

1. If **Keep sorted** is checked, you must uncheck it in order to activate **Move Up** and **Move Down**.
2. Highlight the section in the **Sections in use** list and then click **Move Down** or **Move Down** to promote or demote it.

NOTE Changing the order of sections within an order file is a global change that affects ALL projects, (not just the currently open one).

Reset a design section order back to the original default

If you have made changes to a design order list, the following steps you through resetting the order file to the original default values.

1. Edit the properties of the member.
2. Click the Design section order drop list and select **<New\Edit...>**
3. In the Select a Section Order dialog, highlight the section order that you want to reset.
4. Click **Reset**

The highlighted design section order is reset to its default settings.

NOTE The Reset button is only displayed for the pre-installed section orders. (User defined section orders can be deleted but not reset.)

Create a new Design section order

If you want to create a completely new design section order you can do so as follows:

1. Edit the properties of a member.
2. Click the **Design section order** drop list and select **<New\Edit...>**
3. In the **Select a Section Order** dialog, click **Add...**
4. Enter a unique name for the new design section order.
5. Select the **Country** and the **Section Group** required.
6. Either click **Add All** to add all the available sections, or highlight just the sections you require and click **Add Selected**.

   NOTE When adding selected sections you can use the **Top**, **Bottom**, **Above Selected**, **Below Selected** options to specify where they appear in the list. However, these options are ineffective if you have a sort criteria specified and the **Keep sorted** box is checked.

7. Sort individual sections in the **Sections in use** list, using **Move Up** or **Move Down**, as required.
8. When the list of sections in use is as you want it, click **OK**

The new design section order appears on the list of available section orders.

Add simple connection resistances to the database

The materials database contains pre-defined connection types with pre-defined resistances for steel beams to Eurocode[1] and US[2] head codes.

In addition, you can specify user-defined connection types and user-defined resistances for any head code and save them to the section database. Once defined, these can then be used to Check simple connection resistance (page 635) across all projects.

[1] *The Eurocode resistances are derived from SCI publication P358 and make use of UK NA values for partial safety factors, regardless of the Eurocode Country setting.*

[2] *The US resistances are derived from AISC publication Steel Construction Manual 14th Ed and are only given when US Customary units are selected. Separate resistances are given for LRFD and ASD.*

Pre-defined connection types and resistances

For Eurocodes:

With the Member Type set to Simple Beam, three *pre-defined* connection types are available for S355 and S275 grades:

- Fin Plates,
- Full Depth End Plates
- Partial Depth End Plates

These have connection resistance values defined for UB and UKB section beams. The bolts considered are all size M20, of property class 8.8, and ordinary or flowdrill type. The plates considered are all S275 grade. Fin Plates and Partial Depth End Plates have resistances defined for beams with 0, 1 or 2 notches, while the resistances for Full Depth End Plates assume no notches. Fin Plates have resistances for 1 or 2 vertical lines of bolts.

For US (ACI/AISC) and US Customary units:

With the Member Type set to Simple Beam, and the steel grade set to 'Any' one *pre-defined* connection type is available:

- Single Plate

Single Plates have connection resistance values defined for W section beams of any Fy value. The bolts considered are sizes 3/4 in, 7/8 in and 1 in, from Group A, with thread condition N, in standard holes. The plates considered are all of Fy 36 ksi, with thickness ranging from 1/4 in to 9/16 in. Single Plates have resistances defined for beams with no coping and top flange coping only. All resistances assume 1 vertical line of bolts only.

In addition, two further Simple Beam connection types are available for the 50 ksi grade:

- All-Bolted Dbl Angle,
- Shear End-Plate

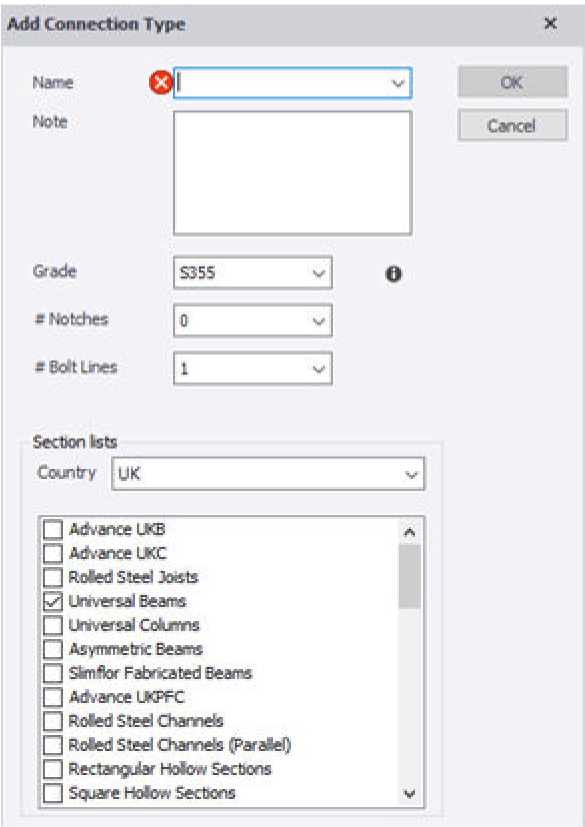
These have connection resistance values defined for W section beams, and for Single Plates with W section beams of F_y 50 ksi. The bolts considered are sizes 3/4 in, 7/8 in and 1 in, from Group A, with thread condition N, in standard holes. The plates and angles considered are all of F_y 36 ksi, with thickness ranging from 1/4 in to 9/16 in. All-Bolted Double Angles have resistances defined for beams with no coping and top flange coping only, while the resistances for Shear End-Plates assume no coping. All resistances assume 1 vertical line of bolts only.

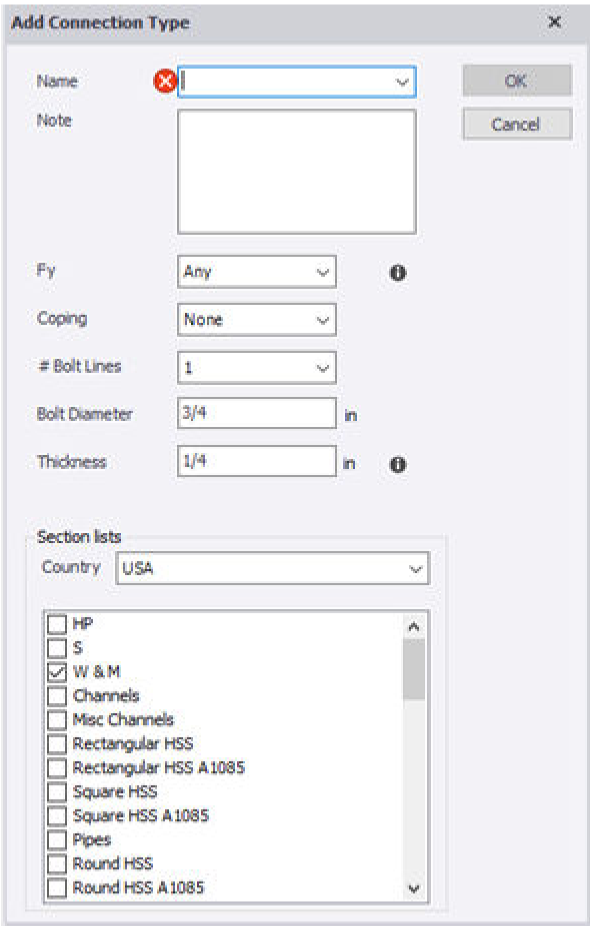
NOTE The pre-defined connection types and resistances can be reviewed in the Connection Resistance dialog (page 1174).

Add user-defined connection types

1. Open the Connection Resistance dialog (page 1174)
2. Under **Connection Types**, click **Add...**

   An **Add Connection Type** dialog appropriate to the current headcode is displayed.





3. In the Name box enter a name for the Connection Type.

   If you want to create a Connection Type with a number of variations of Grade, Notches or Bolt Lines then it is important to enter exactly the same Name each time - e.g. the database will recognise “CT1” and “CT 1” as separate Types simply because of the space in the second name. This has an effect when viewing results in tabular data or reports, where Tekla Structural Designer carries out an optimisation process to find the first passing resistance based on the name of the Connection Type and the number of Bolt Lines and Bolt Rows assigned to that name

   NOTE It is possible to use the same Name as one of the pre-defined Types and it will be recognised in the optimisation process as essentially being the same Connection Type

4. In the Note box enter any descriptive text you would like to see displayed in the dialog’s Info box.

5. Select a Grade or, with US codes, an F_y value.

   Note that *subgrades* like S355 J0 are ignored e.g. a Connection Type with S355 Grade assigned will have its resistances applied to beams or braces of Grade S355 J0, S355 J2, etc.

6. Select # Notches and # Bolt Lines or, with US codes, select Coping, # Bolt Lines, Bolt Diameter and Thickness (of plate or angle leg).
7. Select the section lists that you will want to define resistances for (this list can be added to later on if required, in the Edit Connection Type dialog).
8. Click **OK**

   NOTE **OK** is not active until a Name has been defined. After clicking OK 'Hide undefined values' becomes automatically un-ticked and resistance values can be entered against the required sections. If no resistance values are assigned at this stage for at least one section in a selected Section List, then that List will become de-selected and have to be re-selected later on.

You are now able to **add user-defined connection resistances to the database** against the required sections for this connection type.

Edit user-defined connection types

1. Open the Connection Resistance dialog (page 1174)
2. Under **Connection Types**, click **Edit...**

   An **Edit Connection Type** dialog appropriate to the current headcode is displayed.

3. Edit the values as required.
4. Click **OK**

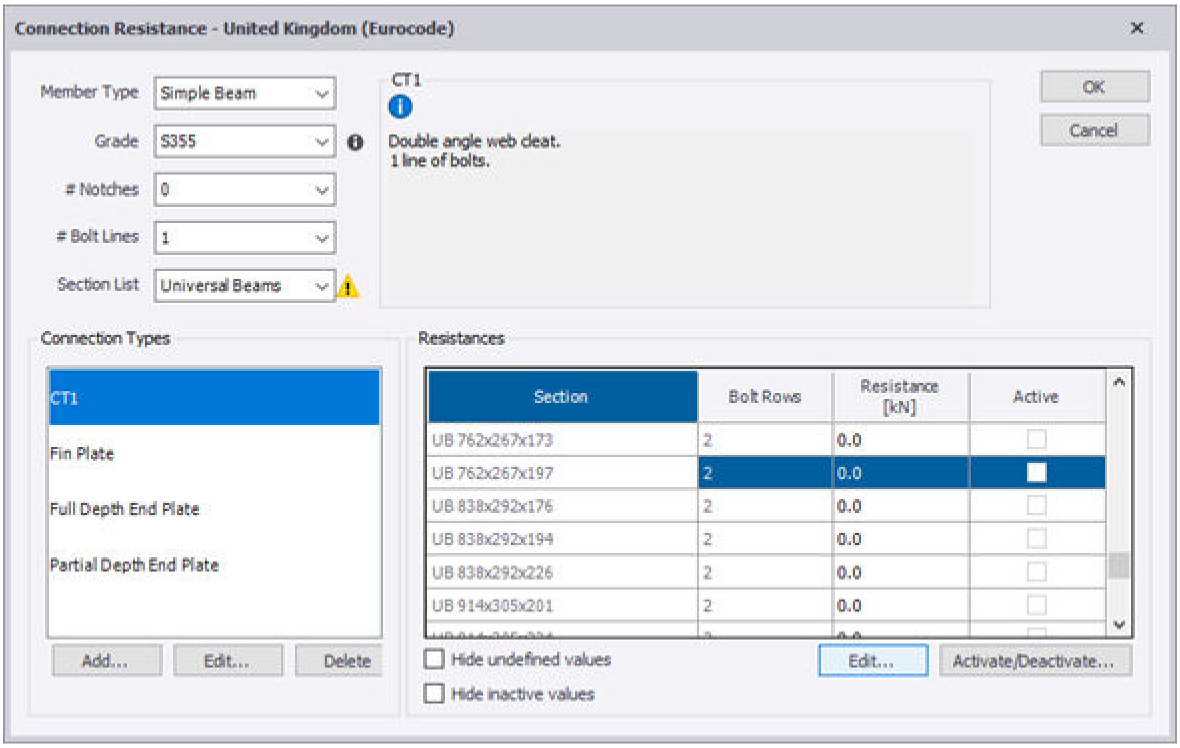
Add user-defined connection resistances

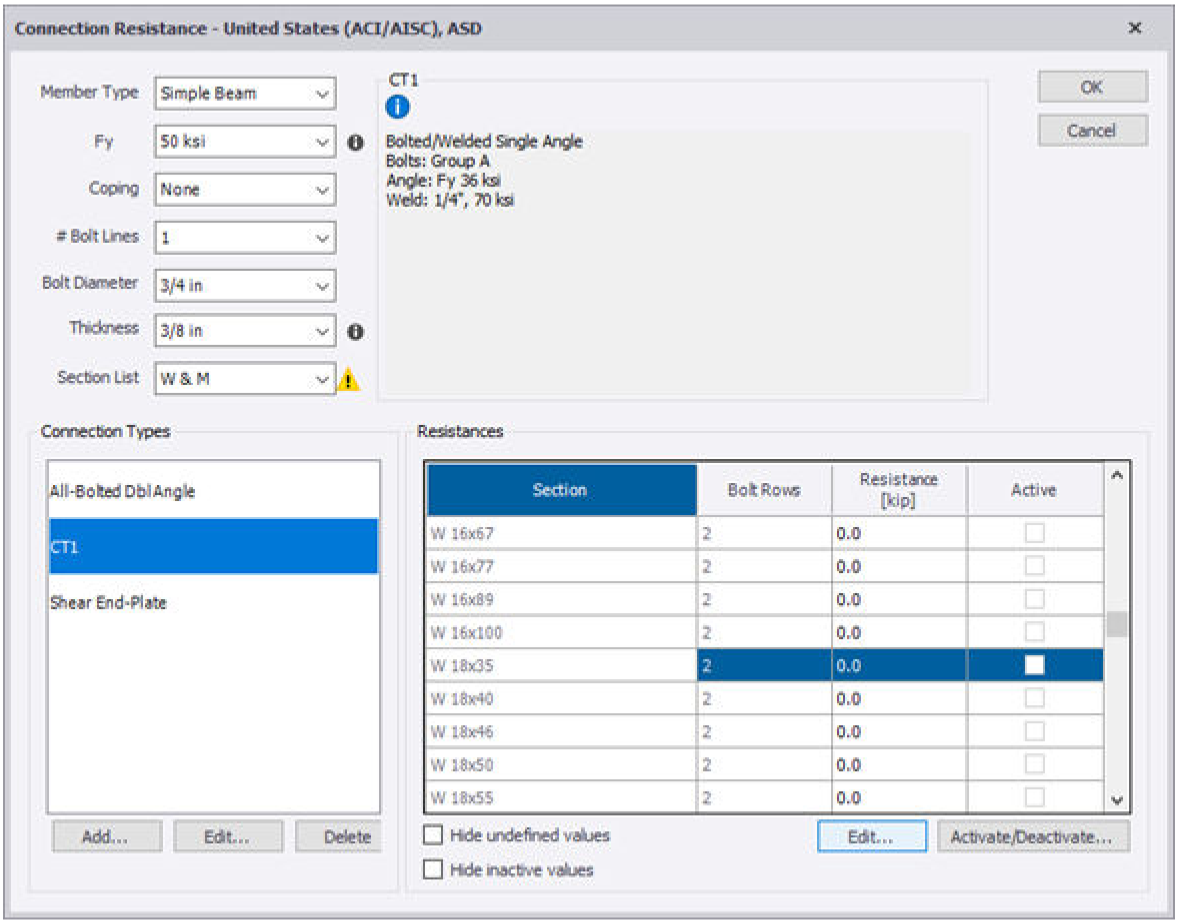
User-defined connection resistances can be added to the database for existing connection types as follows:

1. If it is not already displayed, open the Connection Resistance dialog (page 1174)
2. If necessary, use the filters at the top of the dialog until the required connection is displayed in the Connection Types list.
3. Select the connection in the Connection Types list. The sections are listed in the Resistances table, along with the bolt rows and resistance values for the selected connection type.

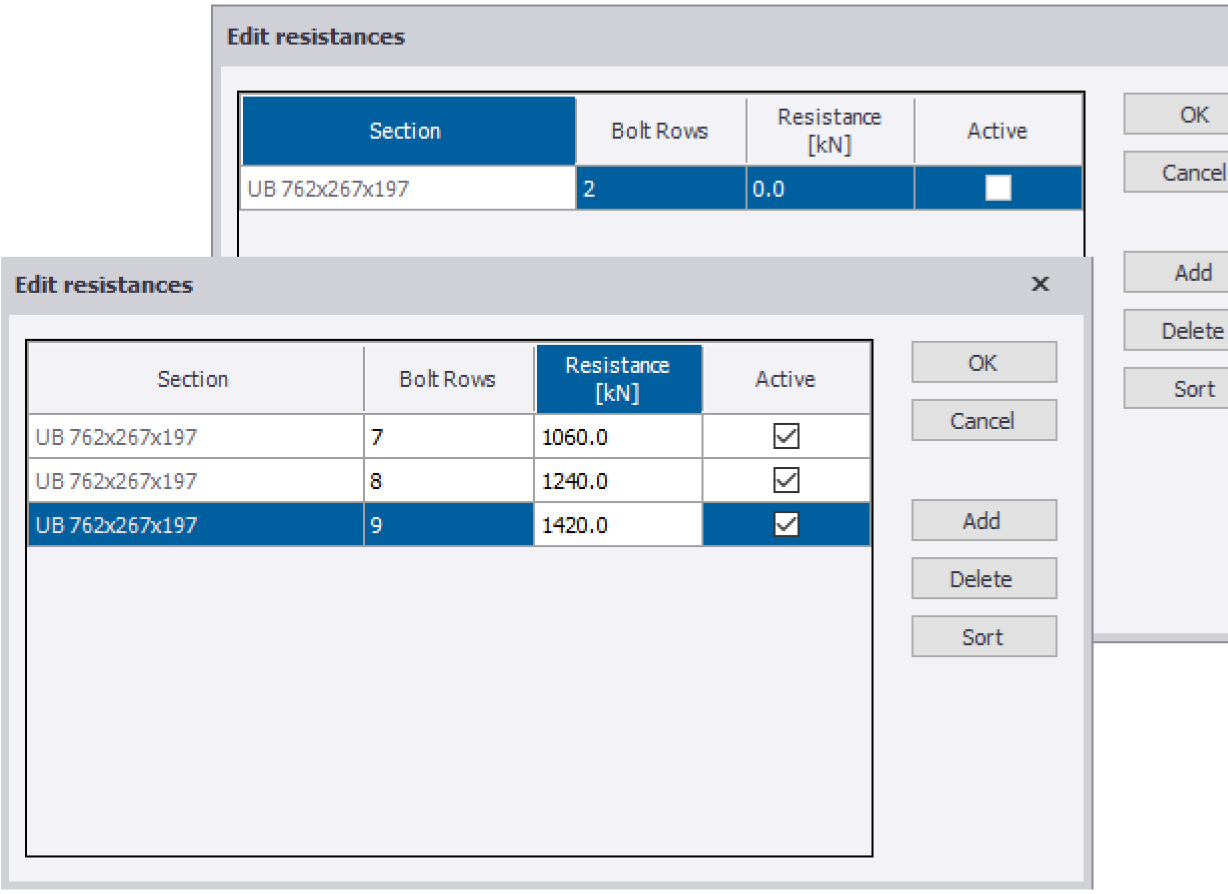
   NOTE If no sections are listed uncheck **Hide undefined values**.

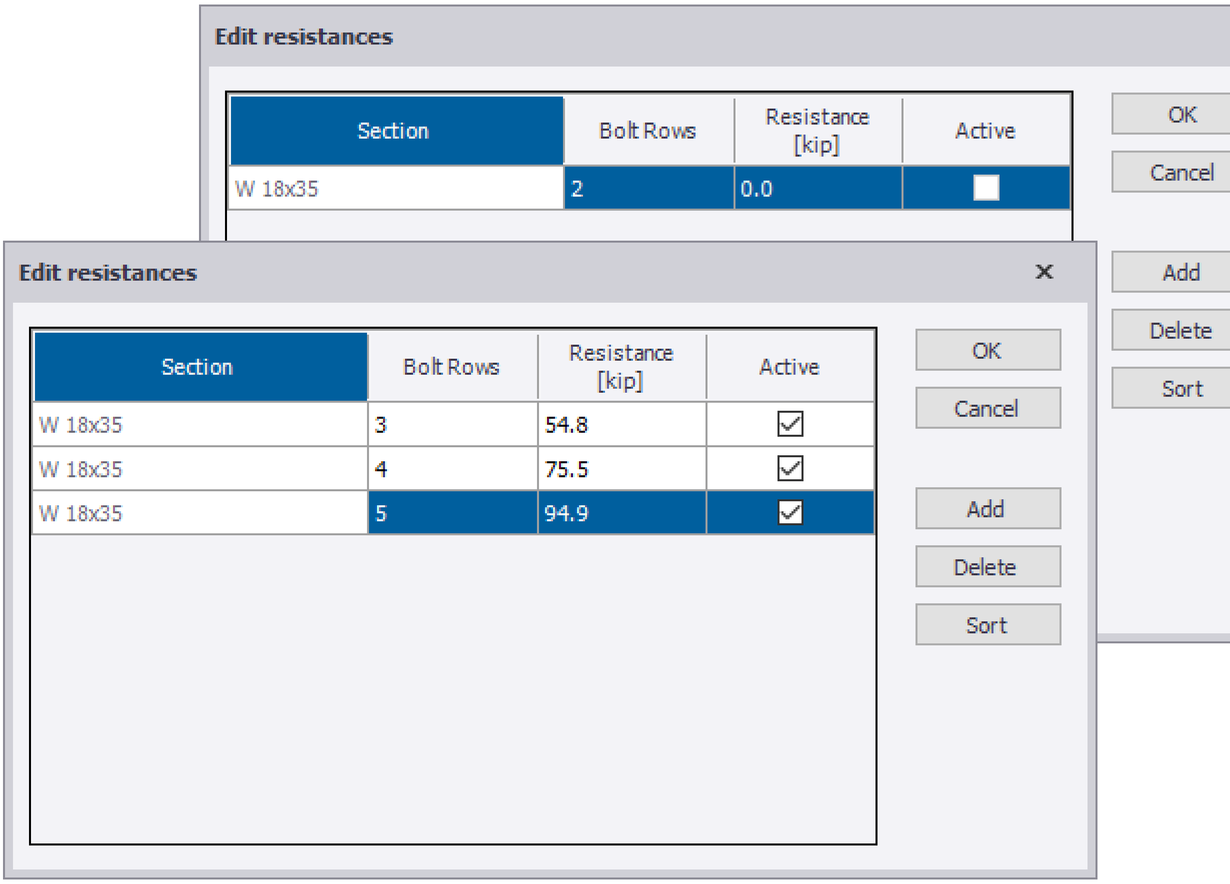
4. Select the section type in the Section List.





5. Resistance values can be entered directly in the table. Alternatively, select a particular section in the table and click **Edit...**. This displays the Edit resistances dialog which allows you to adjust the bolt rows count or to **Add** multiples of the same section size with more than one bolt row count. The bolt row counts can be entered in any order and then use **Sort** to number from lowest to highest. Use the Edit resistances dialog also to **Delete** data rows if no longer required in your database.


Edit resistances
Section
Bolt Rows
Resistance [kN]
Active
UB 762x267x197
2
0.0
OK
Cancel
Add
Delete
Sort
Edit resistances
Section
Bolt Rows
Resistance [kN]
Active
UB 762x267x197
7
1060.0
UB 762x267x197
8
1240.0
UB 762x267x197
9
1420.0
OK
Cancel
Add
Delete
Sort


Edit resistances
Section
Bolt Rows
Resistance [kip]
Active
W 18x35
2
0.0
OK
Cancel
Add
Delete
Sort
Edit resistances
Section
Bolt Rows
Resistance [kip]
Active
W 18x35
3
54.8
W 18x35
4
75.5
W 18x35
5
94.9
OK
Cancel
Add
Delete
Sort

NOTE Connection resistances are made active as the values are entered.

Related video

Predefined connection resistance database for Eurocode and AISC

Add material properties from the model to a material database

When you define a member, its material properties are initially read from the appropriate material database. However, subsequently, the material properties are held with the member itself. This means that you can open and run the model on another computer, even if it does not have a matching material in its database.

When a mismatch between the model's material data and the material database arises, you are not required to add the missing properties to the database. It may however be beneficial to do so if you anticipate that you will need to re-use the property in question in new models.

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.

2. Go to **Model**.
3. Select the desired objects in the **Material data objects in model** list.
4. Click **Add to Database**.

Add materials for a head code

Engineers may need to design using local sections, materials, and reinforcement. These may not be listed in the materials databases for the current head code. However, you can add materials to the material database manually, if necessary.

WARNING If you add data to a material database, you are responsible for both its accuracy and applicability to the selected design had code.

Add a material grade for a head code

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Material** page.
3. Select the required head code.
4. Select the required material type.
5. If the required grade is not listed for the selected head code, click **Add...**

   The **Add Grade** dialog opens.
6. Type in the grade properties.
7. Click **OK** to return the **Materials** dialog box

   The new grade is shown in the list of available grades.
8. To make the new grade the default grade for the material in the selected head code, select the grade and click **>> Default**.

Add a reinforcement class for a head code

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.
2. Go to the **Reinforcement** page.
3. Select the required head code.
4. Select the country to assign the reinforcement to.
5. Select the required type and rib type.
6. If the required class of reinforcement is not listed for the selected head code, click **Add...**

   The **New Reinforcement Class** dialog opens.
7. Type in the reinforcement class properties.
8. Click **OK** to return the **Materials** dialog box

   The new class is shown in the list of available grades.
9. To make the new class the default class for the combination of the selected head code and country, select the class and click **>> Default**.

Add new reinforcement sizes

When you have created a new reinforcement class, the class does not contain any specified bar sizes. Therefore, you must add the sizes you intend to use.

1. On the **Reinforcement** page, click **Add...** on the right side of the **Available sizes** list.

   The **New Reinforcement Bar Size** dialog opens.
2. Type in the properties of the first bar size.
3. Repeat steps 1 and 2 for all bar sizes that you require.
4. When you are finished, click **Close**.

Specify the bar size range to be applied in auto design

After the new reinforcement has been added to the materials database, you can specify the minimum and maximum bar sizes that should apply to the different concrete members when they are auto-designed.

1. On the **Design** tab, click **Settings**.
2. Go to the **Concrete** page.
3. Locate the reinforcement of each of the following:
   - **Beam**
   - **Column**
   - **Wall**
   - **Slab** --> **Slab on Beams**
   - **Slab** --> **Flat Slab**
   - **Punching Shear**
   - **Foundations** --> **Pad Base**
   - **Foundations** --> **Pile Cap**
   - **Foundations** --> **Mat Foundations**
4. For each of the above, select the country to which the new reinforcement is registered.
5. For each of the above, select the minimum and maximum bar sizes that apply to auto design.

The new design options now apply to the current project. If you want them to apply to new projects as well, save the changes back to the active set.

Change default design sections for a different head code

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to the **Section Defaults** page.
3. Select the settings set that you want to modify.
4. In the table on the **Section Defaults** page, click the currently displayed section for a member to change it.

   The **Select Section** dialog box opens.
5. Select the required country.
6. Select the shape in the **Page** pane on the left.
7. Select the section size in the **Item** pane on the right.
8. Click **Select**.

   The default section has now been updated.
9. Repeat steps 4–8 for all the sections that you want to update.

The new default sections will be applied to new models, provided that the settings set that you modified is the active settings set.

Change default design section orders for a head code

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to the **Section Order Defaults** page.
3. Select the settings set that you want to modify.
4. In the table on the page, click the currently displayed section for a member to change it.

   The **Select a Section Order** dialog box opens.
5. Select the required country.

   NOTE If there are no section orders listed for the selected country, they do not yet exist for the current head code. However, you can still create section orders can still be created by using the **Import...** feature.
6. Highlight an available section order in the list and click **Select**.

   The section order has now been updated.
7. Repeat the steps 4–6 for other members that you want to change.

The new section order will be applied to new models, provided that the settings set that you modified is the active settings set.

Create new section orders for a head code

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to the **Section Order Defaults** page.
3. Select the settings set that you want to modify.
4. In the table on the page, click the currently displayed section for a member to change it.

   The **Select a Section Order** dialog box opens.
5. Click **Export...** and save the section order as an XML file.
6. Close the **Select a Section Order** dialog box.
7. On the **Section Order Defaults** page, select the settings set that was originally being modified.
8. In the table, click the section order that you want to change.
9. Select the required country.
10. Click **Import...** and select the XML file that you just created.
11. Select the imported file by clicking **Select**.
12. Repeat steps 4–11 for other member types as needed.

Upgrade material databases

The installed material databases each have their own database version. You can see the version number separately on each page of the **Materials** dialog box.

The original data in each of the material databases is system data, and it cannot be edited. The version number relates specifically to this system data.

From time to time, new system data may become available through an update. When this happens, an **Upgrade** button is displayed next to the current database version on the relevant page of the **Materials** dialog box. You then have the choice to either upgrade the database or retain the old version.

NOTE Updating the database version does not cause you to lose any user data you may have added to the old version, because the user data is automatically copied back in to the new database.

In certain circumstances, an inconsistency can arise between the user data in a model and the installed databases. The inconsistencies can be identified and resolved on the **Model** page of the **Materials** dialog box, either by updating the model data, or by updating the material databases.

1. On the **Home** ribbon, click **Materials**.

   The **Materials** dialog box opens.

2. Click the **Upgrade** button next to the current database version number.

Timber property assumptions

The engineer shall verify that the properties of the selected grades are those that are characteristic of the material, are obtained from up-to-date manufacturer's data and are applicable to the analysis model. **Analysis Modification Factors** must also be confirmed to be correct.

Head Code Eurocode

The following assumptions have been made:

- Loading will be perpendicular to the wide faces of the laminations (bending about the major axis).
- Moisture content of the sections will be less than 15% - analysis modification factors may need to be applied if this is not the case.
- The glulam grade being used complies with the characteristic properties outlined in BS EN 14080:2013.
- The engineer has verified that the $E_{analysis}$ value of the member is suitable for the particular application.
- The section and strength class are compatible.

Head Code ACI/AISC

The following assumptions have been made:

- The moisture content of the structural composite lumber or glulam section is less than 16% and the temperature remains below 100°F - analysis modification factors may need to be applied if this is not the case.
- The glulam section will be loaded perpendicular to the wide faces of the laminations (bending about the x-x axis).
- The glulam grade being used complies with the characteristic properties outlined in AWC NDS supplement 2018.
- The engineer has verified that the $E_{analysis}$ value of the member is suitable for the particular application.
- The section and strength class are compatible.

Add and manage embodied carbon factors

To enable embodied carbon to be calculated in Tekla Structural Designer default embodied carbon factor (ECF) values have to be set up and saved. You can set up a local set of factors which apply to the current model only and a global set of factors which serve as the defaults for new models.

Set up a global set of factors

When setting up standard embodied carbon factors you should add them directly to a global settings set so that they are available for all new models.

This can be done as follows:

	1. On the **Home** toolbar, click **Settings** 2. On the **Embodied Carbon** page of the dialog, click **Edit...**
	The Embodied Carbon Factors dialog (page 1185) is displayed from where you can manage the global set of factors for the active setting set.

When you have finished, click **OK** to close the dialog and **OK** to close **Settings**

Set up and edit the local set of factors

The local set of factors are initially inherited from the global set. Often you might want to customize these by making specific factors inactive. You might also want to add project specific factors, for example cladding for walls and roofs.

These operations should be performed using the Embodied Carbon Factors dialog (page 1185), which is accessed as follows:

	On the **Home** toolbar, click **Embodied Carbon Factors** The Embodied Carbon Factors dialog (page 1185) is displayed from where you can manage the local set of factors.
	NOTE Changes to the local set do not get applied to new models unless they are saved to a global settings set (page 889).

When you have finished, click **OK** to close the dialog.

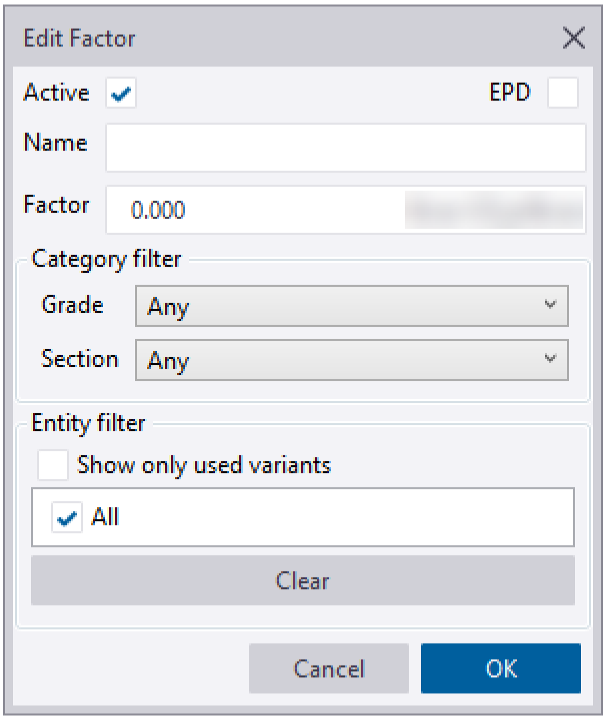
Add factor

1. Open the Embodied Carbon Factors dialog (page 1185)
2. In the **Category filter** select the category appropriate to the factor you are about to add.

   TIP If you intend to enter several factors with similar characteristics you can specify additional Category/Entity filters in advance, before clicking **Add factor**. This way you don't have to respecify the filters for each factor.

3. Click **Add factor**

   The **Add factor** dialog is displayed with any Category/Entity filters that you specified preselected.



4. If the factor being added is from manufacturers data and has an Environmental Product Declaration (EPD), check the EPD box.
5. In the Name box enter a description for the factor that can be used to identify it uniquely.
6. Enter the factor.
7. Confirm that the Category/Entity filters are set as you require.

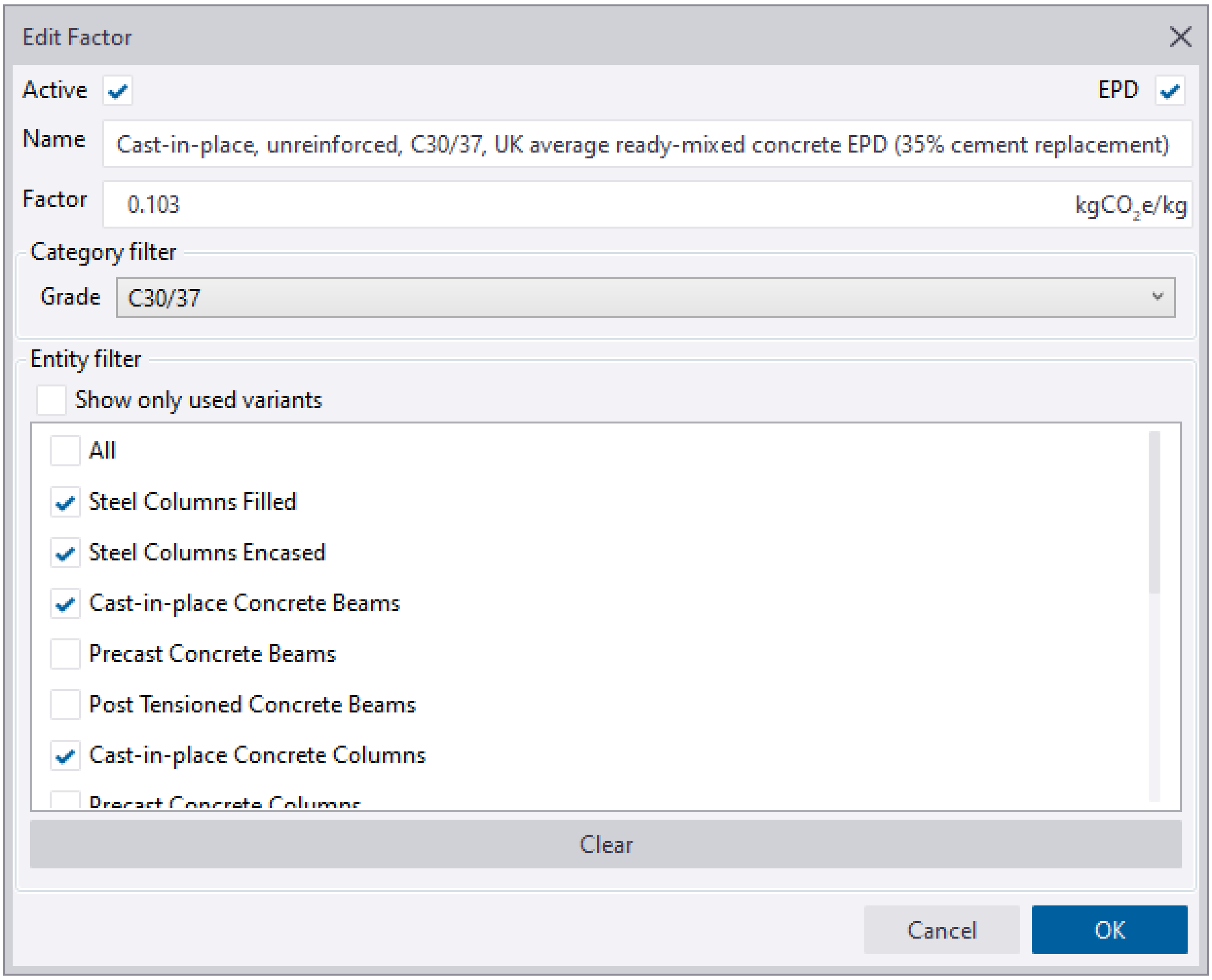
8. Click **OK** The factor is added to the list.

You can now continue to add further factors, or if finished, click **OK** to close the dialog.

Edit factor

1. Open the Embodied Carbon Factors dialog (page 1185)
2. In the **Category filter** select the category appropriate to the factor you want to edit.
3. Select the factor to be edited from the list.
4. Click **Edit factor**

   The **Edit factor** dialog is displayed.



5. Edit the values and filters as required.
6. Click **OK**

Remove factor

1. Open the Embodied Carbon Factors dialog (page 1185)
2. In the **Category filter** select the category appropriate to the factor you want to remove.
3. Select the factor to be removed from the list.
4. Click **Remove factor**

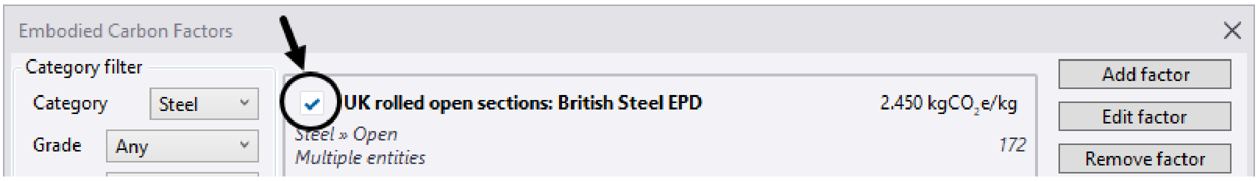
   The factor is removed from the list.
5. Click **OK**

Reorder the list of factors

1. Open the Embodied Carbon Factors dialog (page 1185)
2. In the **Category filter** select the category appropriate to the factors you want to reorder.
3. Select a factor to be move and drag it to move it up or down the list.
4. Continue moving factors as required, or click **OK** when done.

Set a factor as active or inactive

1. Open the Embodied Carbon Factors dialog (page 1185)
2. In the **Category filter** select the category appropriate to the factors you want to activate/deactivate.
3. Check the box to the left of a factor to make it active, or uncheck to make inactive.



4. Click **OK** when done.

Export factors to a spreadsheet

1. Open the Embodied Carbon Factors dialog (page 1185)
2. Click **Export** and choose to export either all factors, or active factors.

   A spreadsheet opens listing the factors.

12.4 Manage properties and property sets

When you create a new entity, default properties for it are displayed in the **Properties** window, these can be edited as required before the entity is placed.

Provided the properties are saved to a **property set**, they can be applied to similar entities elsewhere in the current model.

Property sets can also be transferred between models.

In this section we show you how to:

- Save properties to and recall properties from property sets (page 912)
- Apply property sets to existing entities (page 913)
- Review where property sets have been applied (page 914)
- Transfer property sets between models (page 915)
- Delete property sets (page 915)

Related information

See also

Modify the properties of entities or model objects (page 168)

Related video

Property Sets - tutorial

Save properties to and recall properties from property sets

Saving properties to property sets allows you to re-use them elsewhere.

Save properties from the Properties window to a new property set

NOTE You can only save properties in the **Properties** window to a property set when there are no entities selected. This ensures that each property in the property set has an unique entry.

1. From the **Model**, **Design**, or **Foundations** ribbon tab, click the entity type whose properties you want to save as a property set.

   The list on the top of the **Properties** window now reads **<unsaved set>**.

2. Specify the properties according to your needs.
3. Click the **Save...** button on the top right corner of the **Properties** window.

   The **Add property set** dialog box opens.

4. Name the property set.
5. Click **OK**.

Save the properties of an existing entity to a named property set

1. Hover the mouse pointer over the desired entity until it becomes highlighted.
2. Right-click the entity.
3. In the context menu select **Create property set...** (and if the entity is a beam or column or wall, select the span or stack required).

   The **Add property set** dialog box opens.
4. Name the property set.
5. Click **OK**.

Recall a previously saved property set to the Properties Window

Provided that you have previously saved a property set, you can recall it again later in the **Properties** window, if it is applicable to the current command.

1. From the **Model**, **Design**, or **Foundations** ribbon tab, click the desired entity type.

   The properties applicable to the selected entity type are displayed in the **Properties** window.
2. Click the list on the top of the **Properties** window.
3. In the list, select the desired property set.

Apply property sets to existing entities

You can apply a property set to existing entities in a **Structural View**, but if you want to make a lot of changes property sets can be applied more easily from a **Review View**.

Apply a property set to an individual entity in a Structural View

1. Hover the mouse pointer over the entity until it becomes highlighted.
2. Right-click the entity.
3. In the context menu, select **Apply Property Set...**

   The **Select property set to apply** dialog box opens.
4. Select the desired property set.

5. Click **OK**.

The property set is only applied to the individual entity that you right-click, even if multiple entities were selected. To apply a property set to multiple entities, see the following topic.

Apply a property set to multiple members in a Structural View

1. Select the entities that you want to update.
2. Ensure that the list on top of the **Properties** window shows the correct entity type.
3. On the top right corner of the **Properties** window, click **Apply...**
4. Select the desired property set.
5. Click **OK**.

The property set is now applied to all of the selected entities, provided that they are of the same type.

Apply a property set in a Review View

1. Open a **Review View**
2. On the **Review** tab, click **Property Sets**.
3. In the **Properties** window, do all of the following:
   a. Set **Review/Update** to **Apply property set**.
   b. Select the entity type.

      NOTE Only those entity types that already have property sets saved will be listed.

   c. If the **Member** entity type was selected, you will also need to select the characteristic.
   d. Select the property set that you want to apply.
4. In the **Review View**, click the entities to which you want to apply the property set.

Review where property sets have been applied

To graphically review where property sets have been applied, see the following instructions.

1. Open a **Review View**
2. On the **Review** tab, click **Property Sets**.
3. In the **Properties** window, set **Review/Update** as **Review All**.

4. Still in the **Properties** window, select the entity type that you want to review.

   NOTE Only those entity types that already have property sets saved will be listed.

   Entities of the selected type are color-coded to indicate where property sets have been applied.

Transfer property sets between models

Once property sets have been saved, they can be exported from the current model and then imported into another Tekla Structural Designer model.

Export property sets

1. On the **Home** tab, click **Manage Property Sets**.

   The **Manage Property Sets** dialog opens.
2. Select the property sets that you want to export.
3. Click **Export...**
4. Save the properties as a .tsps file.

Import property sets

1. On the **Home** tab, click **Manage Property Sets**.

   The **Manage Property Sets** dialog opens.
2. Select the property sets that you want to import.
3. Click **Import...**
4. Select the .tsps file that contains the required property sets.
5. Click **Open**.

   The imported property sets are now listed in the **Manage Property Sets** dialog.

Delete property sets

1. On the **Home** tab, click **Manage Property Sets**.

   The **Manage Property Sets** dialog opens.

2. Select the property sets that you want to delete.
3. Click **Delete**.
4. Click **OK**.

12.5 Create and manage user-defined attributes

You can define user-defined attributes (UDAs) to save miscellaneous data to files, individual members and panels.

User-defined attributes are flexible, and you can use them for a variety of purposes. For example, you can:

- Apply descriptive labels, such as construction phases.
- Record paint specifications.
- Attach office documents, pictures, or other associated files.
- Link to design files from other applications.
- Filter material lists and member design reports for specific attributes.

UDA definitions and values

Example UDA definitions are included in the default settings sets. You can easily modify the definitions according to your needs.

(page 1045) allows you modify the definitions that apply to the current model. From there you can also set UDAs to be **linked** so that the same UDA value gets assigned to all duplicate levels.

You assign specific UDA values to an entity via the **Properties** window. Once a UDA value exists, it can also be assigned to or removed from members graphically by using the **Review View**. For further details, see: Attach UDA values to members and panels (page 918)

Attaching files as attributes

When a file is attached as an attribute, you can embed it within the Tekla Structural Designer file. When embedded, the attached file is included when the model is transferred to another computer.

NOTE The embedded file only gets attached to the Tekla Structural Designer file when the model is saved using **Save** or **Save As**.

This means that the embedded files are not attached when you use **Save Model Only**. Similarly, if you have to revert to an autosaved version of the model, the embedded files will not be attached.

Transferring attributes

Attributes are transferred when models are exported to Tekla Structures and Revit. In the current release, attributes are not yet included for Tekla Structural Designer drawings.

See also

User-defined attribute settings (page 1045)

Create attribute definitions (page 917)

Attach UDA values to members and panels (page 918)

Apply attribute filters to material lists and reports (page 920)

Create attribute definitions

You can create and modify attribute definitions according to your needs. The attributes are defined either in the **Model Settings** dialog box or in the **Settings** dialog box, depending on whether you want the to the attributes be applied to the current model or new models.

Create attribute definitions in the current model

1. On the **Home** tab, click **Model Settings**.

   The **Model Settings** dialog opens.
2. Go to the **User Defined Attributes** page.
3. Click **Add**.
4. Define the following properties:
   - Name
   - Type
   - Source

- Values (value choices that apply when the source is value list)

NOTE The order in which UDAs are listed in the replicates the order in which the UDAs are listed in the **Model Settings** dialog.

Use the **Move Up** and **Move Down** buttons to reorder the UDAs.

Create attribute definitions for new models

1. On the **Home** ribbon, click **Settings**.

   The **Settings** dialog box opens.
2. Go to **Settings Sets**.
3. Select the settings set to which you want the attributes to apply.
4. Go to the **User Defined Attributes** page.
5. Define the following properties:
   - Name
   - Type
   - Source
   - Values (value choices that apply when the source is value list)

NOTE The order in which UDAs are listed in the replicates the order in which the UDAs are listed in the **Model Settings** dialog.

Use the **Move Up** and **Move Down** buttons to reorder the UDAs.

Attach UDA values to members and panels

You can attach office documents, pictures, or other files to the members and panels of your model. When you attach files to members and panels, they are embedded in the file. The embedded files will also be included in the model if the model is transferred to another computer.

See also

Create attribute definitions (page 917)

Attach a UDA value using the Properties Window

1. Select the members or panels to which you want to attach attributes.

NOTE If the selection contains different member types, select the first member type in the list at the top of the **Properties** window.

2. In the **Properties** window, under **UDA**, define the value for each attribute that you want to attach.
3. If the attribute is a file, select the **Embedded** option to save the file inside the model.

   Only embedded files are automatically transferred when the model is copied to another computer.

   TIP For multi-span beams and multi-stack columns or walls, you can attach separate UDAs to individual spans or stacks.

   ---
4. Repeat steps 1-3 for additional member types.

Attach an existing UDA value in the Review View

1. On the **Review** tab, click **UDA**.
2. In the **Properties** window, set **[M]ode** as **Set On** or **Toggle**.
3. In the **Properties** window, select the attribute.
4. In the **Properties** window, select a value for the selected attribute.

   NOTE You can use some additional settings in the **Properties** window to control which members and panels are displayed in the **Review View**.

   - Under **Filter**, you can use the **Show** and **Entity type** to select which attribute values and entities are displayed.
   - Select the **Color for other values** box to display members that have an other value of the selected attribute.

   ---
5. Click the desired members to add the UDA value.

   TIP To remove the UDA value:

   - If the **[M]ode** is set to **Set On**, change it to **Set Off**.
   - If the **[M]ode** is set to **Toggle**, click the selected members again.

Graphically review the attached UDA values

1. On the **Review** tab, click **UDA**.
2. In the **Properties** window, set **[M]ode** as **Review**.
3. In the **Properties** window, select the attribute that you want to review.

   The members or panels are color-coded to represent the different values of the selected attribute.

Open a file that has been attached as a UDA

1. On the **Review** tab, click **UDA**.
2. In the **Properties** window, set **[M]ode** as **Review**.
3. In the **Properties** window, select the attribute that you want to review.
4. Click the member or panel to which the file is attached.

   Provided that the extension has been associated with an appropriate application, the application opens and displays the file.

Apply attribute filters to material lists and reports

User-defined attributes (UDAs) allow you to filter material lists and member design reports for specific attributes. For more detailed information, see the following paragraphs.

See also

Create attribute definitions (page 917)

Attach UDA values to members and panels (page 918)

Material list settings (page 1039)

Apply an attribute filter to material list review data

1. On the **Review** tab, click **Tabular Data**.
2. In the **View Type** list, select **Material List**.
3. In the **Filter** list, select **Selected UDAs**.

   The **Select filter items** dialog box opens.
4. Select the required attribute values.
5. Click **OK**.

Apply an attribute filter to a report

1. On the **Report** tab, click **Model Report...**

   The **Report Contents** dialog box opens.
2. In the **Available Styles** list, select the report.
3. In the **Report Structure** list, right-click the chapter or sub-heading that you want to filter.
4. In the context menu, select **Model Filter** --> **Edit\New...**

   The **Select filter** dialog box opens.

5. Click **Add**.
6. In the **Filter properties** list, select **Selected UDAS**.
7. In the **Selected items** list, select the attribute values.
8. Click **OK**.

12.6 Manage sub structures

Sub structures are saved user-defined selection groups of any objects which can be of any size and mix of object types, materials etc. They can be viewed in isolation and used as filters for many areas of program operation. Using sub structures has multiple productivity benefits throughout the Tekla Structural Designer workflow, from model organization and editing through to results review, design and output.

NOTE In Tekla Structural Designer the term *sub structure* is specifically used to refer to a collection of elements, either above or below ground level. It is **not** being used to refer to that part of the structure below ground level (e.g. basements and foundations).

Sub structure characteristics

The basic characteristics of Tekla Structural Designer sub structures are:

- Elements can be a part of more than one sub structure.
- Not every element has to be in a sub structure.
- Deleting a sub structure does not delete the elements within it.
- Sub structures are distinct from, and should not be confused with sub models (page 488).
- Sub structures themselves are never analyzed.
- Sub structure groups are simply collections of other sub structures.

Create a sub structure

1. **To create a sub structure from a Structural View**

1. In the scene view, click or box around the members that you wan to include in the sub structure.
2. Right click and from the context menu select **Add to Sub Structure...**
3. In the left hand pane of the dialog, all the selected objects are listed with on/off check boxes

a. Uncheck any objects that you don't want to be included in the sub structure.

4. In the left hand pane of the dialog, choose an existing or new substructure, and if new, enter the sub structure name.
5. Click **OK**
2. **To create a sub structure from the Project Workspace**
6. In the **Project Workspace** open the **Structure** tree.
7. Expand the **Sub Structures** branch.
8. Right-click the **Sub Structures** sub branch, then in the context menu, select **Create Sub Structure**.
9. Review the sub structure properties in the **Properties** window.
10. In the **Name** field, type the name of the sub structure.
11. Select the color of the sub structure.
12. In the 3D view, click or box around the members that you want to include in the sub structure.
13. Press **[Esc]** when complete.
3. **To create a sub structure from a Review View**
14. On the status bar at the bottom of the window, click **Review View**.
15. On the **Review** tab, click **Sub Structures**.
16. Go to the **Properties** window.
17. Set **Review/Update** to **Update Selected**.
18. Set **Update Sub Structure** to **- - New - -**.
19. In the **Name** field, type the name of the sub structure.
20. Select the color of the sub structure.
21. In the review view, click or box around the members that you want to include in the sub structure.
22. Press **[Esc]** when complete.

Edit a sub structure

1. **To edit a sub structure from the Project Workspace**
1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch, and then the **Sub Structures** sub branch within it.

3. Right-click the sub structure that you want to add to, or remove from.
4. in the context menu, select **Edit**.
5. Skip to step 7 below.
2. **To edit a sub structure from a Review View**
6. On the status bar at the bottom of the window, click **Review View**.
7. On the **Review** tab, click **Sub Structures**.
8. Go to the **Properties** window.
9. Set **Review/Update** to **Update Selected**.
10. In **Update Sub Structure**, select the sub structure that you want to modify.
11. In **Selection Mode**, select how you want to modify the structure.
12. In the review view, click or box around the members that you want to modify.

Delete a sub structure

NOTE Deleting a sub structure only removes the association between the selected group of members - the members themselves are not deleted.

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch, and then the **Sub Structures** sub branch within it.
3. Right-click the sub structure that you want to delete.
4. in the context menu, select **Delete**.

Rename a sub structure

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch, and then the **Sub Structures** sub branch within it.
3. Right-click the sub structure that you want to rename.
4. in the context menu, select **Rename**.
5. Enter the new name as required.

Review sub structures

1. On the status bar at the bottom of the window, click **Review View**.
2. On the **Review** tab, click **Sub Structures**.
3. Go to the **Properties** window.
4. Set **Review/Update** to **Review All**.

   Tekla Structural Designer displays each sub structure in a different color.

Create a sub structure group

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch.
3. Right-click the **Sub Structure Groups** sub branch.
4. In the context menu, select **Create Sub Structure Group**.
5. Select the sub structures to be included in the sub structure group.
6. In the **Properties** window.
   a. In the **User name** field, type the name of the sub structure group.
   b. Select the color of the sub structure group.

Open a 3D view of a sub structure

1. In the **Project Workspace** open the **Structure** tree.
2. Expand the **Sub Structures** branch.
3. Double-click the sub structure.

   Tekla Structural Designer opens a 3D view of the selected sub structure.

   TIP To open a solver view of the sub structure, right-click the sub structure, and select **Open solver view**.

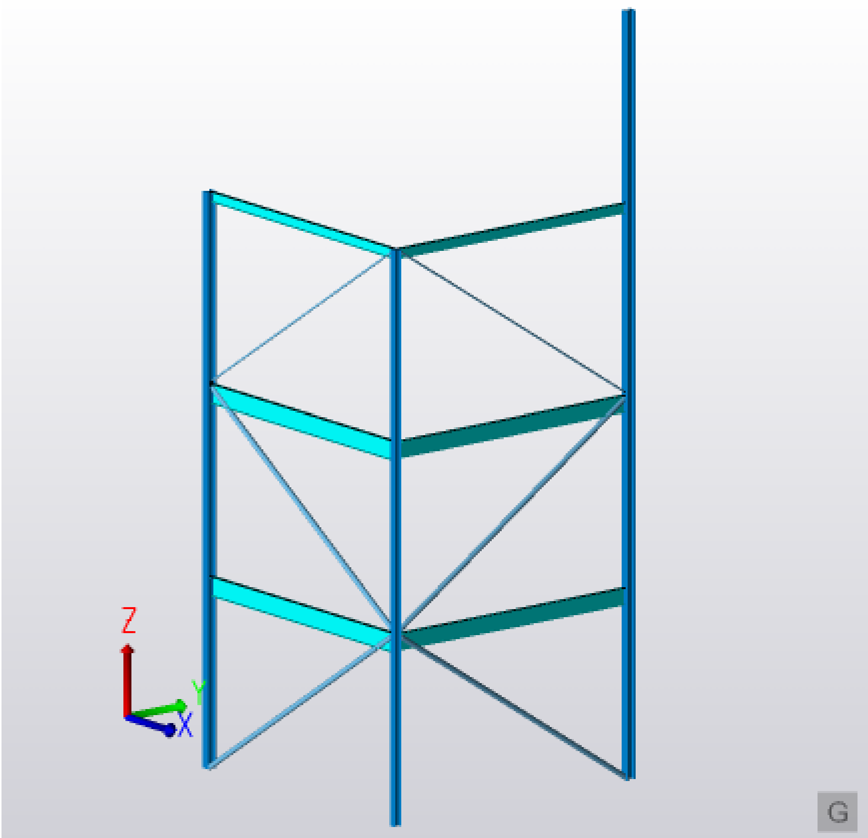
4. Once the view has opened, a (Ghosted) button is displayed in the bottom right corner. Clicking this button toggles the display to either show just the sub structure, or the sub structure with a 'Ghosted On' view of the rest of the model.

Use Ghosted to see the view in the context of the whole model

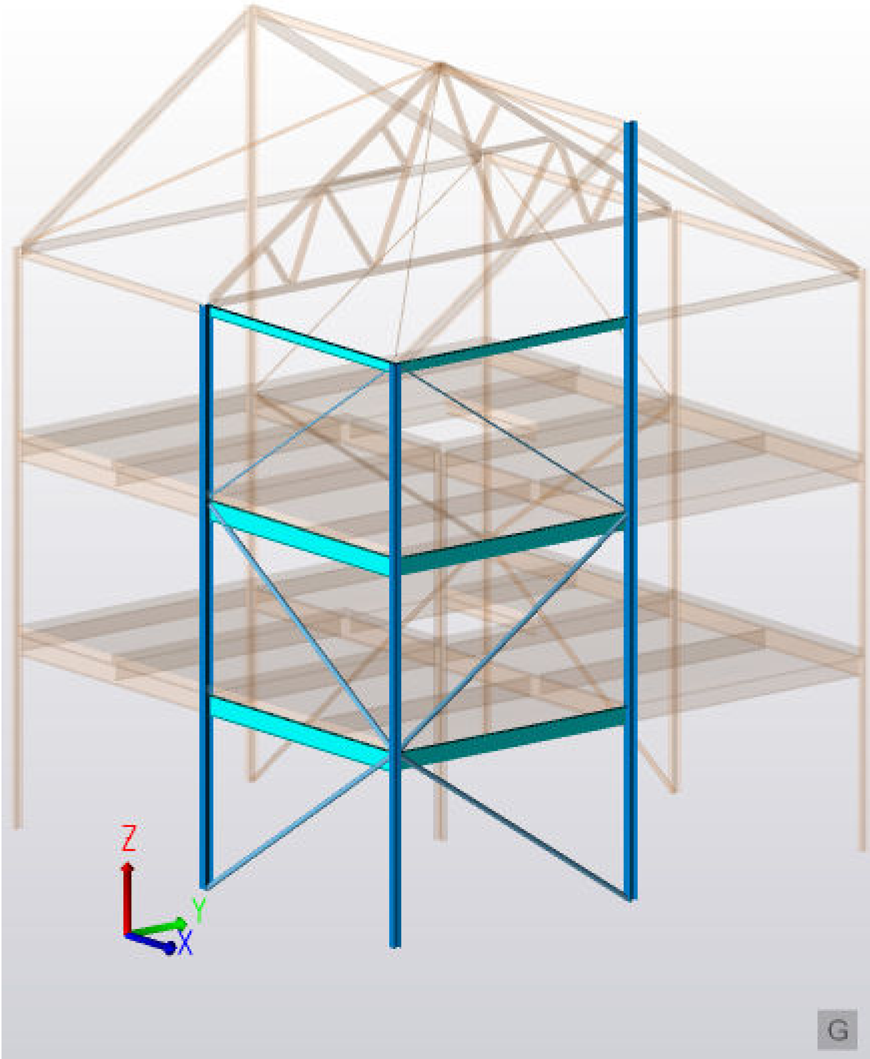
You can toggle **Ghosted** on and off using the button.

NOTE The **Ghosted** button is displayed in in the bottom corner of Sub Structure and Sub-model Views. It is also displayed in Level, Frame, and Slope Views when they have been toggled into 3D (via the 2D/3D toggle button).

- **Ghosted off**: the view is displayed in isolation.



- **Ghosted on**: the view is displayed in the context of the whole structure.



Related video

Ghosted Structure view

Related information

Related videos

How do I use Sub structures?

Sub Structure enhancements

See also

Check all members and walls in a sub structure (page 609)

Design all slab items in a sub structure (page 625)

Check all slab items in a sub structure (page 624)

12.7 Working with large models

Although bigger models will typically need more RAM, you should note that it is the volume of model data and results data created during analysis and design that generates the demand for RAM, so by controlling the volume of data you are able to influence the speed of solution.

Your modelling and design choices can affect the volume of data produced; some of the more significant of these choices are discussed below:

Don't mesh concrete slabs in 3D Analysis

Meshing is not necessary in 3D Analysis for traditional Beam and Slab models as these can use FE decomposition instead.

It is also not necessarily required for Flat Slab models, however, it does become required if you have a transfer slab. It is not critical unless the slab is part of the lateral resistance system.

By default the Mesh 2-way slabs in 3D analysis option that controls this is not checked at any level; you should only check it at a given level when you have good reason to do so.

Using coarser shell meshing in large models

You should review and consider adjusting the mesh parameters as the defaults can often be conservative.

- Using coarser mesh parameters has no impact on sway or modal frequencies.
- If you are not concentrating on slab design you can use very coarse meshes.

Ultimately it is your responsibility to be comfortable with the level of mesh refinement applied to the model. However we would definitely recommend using a courser mesh during design development and then perhaps consider refining a bit more at final design stage.

Using coarser semi-rigid diaphragm meshing in large models

If you have employed semi-rigid diaphragms and are experiencing performance issues you should review and consider adjusting the semi-rigid mesh parameters.

It has been observed that using refined semi-rigid meshes generally has negligible impact on results and only has the effect of slowing down analysis and increasing memory requirements. It is suggested that the default mesh parameters may be more refined that is actually necessary.

Limit the number of loadcases and combinations

You can control the number of combinations created when running the Combination Generator.

In particular you should consider limiting the number of wind loadcases and combinations.

Also:

- Don't add wind loading during the initial design development.
- Don't activate pattern loadcases and combinations until you need to, probably only at the final design stage.

Alternative design approach for large models

For large models, rather than running Design All you may be able to save time by running Analyse All instead and then run a selective design, such as:

- Design by level
- Design by frame
- Design by group
- Design by sub-structure
- Design by member

Effective use of Auto Design

Although for the first design run you might choose to use select bars starting from Minima, on subsequent runs it is generally more efficient to use select bars starting from Current; this will run a check on the current steel provision and if inadequate, it will automatically re-select new steel bars to pass the design.

Check Design can also be very effective - you can turn off the Autodesign and then manually deal with any fails.

Design members for FE chasedown analysis results

This is set in the design options for concrete and by default it is checked on for beams, columns and walls. However in a traditional Beam and Slab model it may not actually be necessary; it is generally not critical unless you have some unusual transfer level challenge. In large models you should therefore consider unchecking it.

Re-design columns (or beams) using previous analysis results

If you change the size of a member you can try out its design without being forced to re-analyse.

Basically you can make any edit you want that does not change the number of stacks in a column, (or spans in a beam); although the analysis results will be marked as out of date, you can still do a design for the changed member based on the old analysis results.

So using the list below as examples design can still be done in all the cases noted:

1. Changing column (or beam) size but retaining shape - designs ok (but see 3 and 4).

2. Changing column (or beam) shape - designs ok (but see 3 and 4).
3. Making column (or beam) smaller so that previously attached members no longer attach - this changes number of stacks - design beyond scope.
4. Making column (or beam) larger so that previously un-attached members now attach - this changes number of stacks (spans) - design beyond scope.
5. Adding / editing / deleting beams that attach to a column - design remains possible up to the point that it affects number of stacks - OK
6. Adding / editing / deleting flat slabs that attach to a column - design remains possible up to the point that it affects number of stacks - OK
7. Adding / deleting stacks or levels (or editing level properties) - Adjusting Levels designs ok because number of stacks is the same. If you add or remove stacks then design is beyond scope.

Model organisation

Tekla Structural Designer has a number of features for organising the model than can each be used to increase efficiency:

- Grouping - one design is applied to all members in the group.
- Sub-structures - allow you to focus on specific areas of interest.
- Duplicate levels - generally save modelling time and reduce the volume of data.

When using duplicate levels, you can achieve further efficiency by designing slabs for a fine mesh at one level only, and then check the slabs at duplicates of the level using a courser mesh result.

NOTE Because meshing parameters are "sub model", rather than "level" based, to achieve this you would set coarse mesh parameters in the structure settings but then override them for an individual sub model.

Our general advice for duplicate levels is:

- For preliminary design set a coarse mesh for entire structure
- For final design where there are a lot of duplicate levels
  - possibly refine the mesh used for the entire structure a little
  - but for each set of duplicate levels, select one and adjust the relevant sub-model parameters to get a finer mesh.

Model complexity

Do not model every little architectural detail - especially not things like small holes in slabs and walls.

Design Options

Take control (get it right at the beginning!)

13 Tekla Structural Designer reference

This section contains reference information mainly on settings, options, and sign conventions, but with with various other miscellaneous reference topics included also.

13.1 Properties

This section describes properties of the main object types.

Structure geometric/analytical properties:

Member properties:

- Slab item properties (page 982)
- Foundation mat properties (page 988)
- Pad base strip base and pile cap properties (page 993)

Industrial object properties:

- Line ancillary properties (page 1001)
- Area ancillary properties (page 1004)
- Equipment properties (page 1006)

Other object properties:

- Bearing wall properties (page 1008)
- Shear only wall properties (page 1011)
- Wall Panel Properties (page 1013)
- Roof Panel Properties (page 1015)
- Slab/Mat overhang properties (page 993)
- Support properties (page 1017)
- Analysis Element properties (page 1020)
- Base plate properties (page 1022)
- Patch properties (page 1024)
- Punching check properties (page 1027)
- Result strip properties (page 1031)

Structure Properties

Use the **Structure** properties to view or modify the basic properties of the structure.

Property	Description
Building Direction Rotation	Defines the building direction relative to the Global Axis System The default (0 degrees) aligns the building direction 1 arrow with the global X axis and the direction 2 arrow with the global Y axis. Entering a positive value rotates the Building Direction arrows clockwise about positive Global Z, a negative value rotates anti-clockwise. The limiting values are +45 degrees and -45 degrees. (If you enter larger values they will be capped at these limits). The building direction arrows are always at 90 degs to each other.
Show Building Arrows	When shown, building direction arrows are displayed in all 2D and 3D Views.

Property	Description
Building Direction Labels	The labels to be used for the building direction arrows. The options are: • Dir 1/2 • Dir H/V • Dir X/Y
Consequence class	Defines k_{FI} used in strength combinations. • CC3 - k_{FI} = 1.1 • CC2 - k_{FI} = 1.0 • CC1 - k_{FI} = 0.9 **NOTE** Applies to **Finland (Eurocode)** head code only
Reliability class	Defines γ_d used in strength combinations. • RC3 - γ_{FI} = 1.0 • RC2 - γ_{FI} = 0.91 • RC1 - γ_{FI} = 0.83 **NOTE** Applies to **Sweden (Eurocode)** head code only
Shell Mesh Size	Defines the shell mesh size for two way spanning slabs. **NOTE** To optimize solution time consider using a coarser mesh during design development before switching to a more refined mesh at the final design stage.
Shell Uniformity Factor	Defines the shell mesh uniformity for two way spanning slabs. (100% = maximum uniformity).
Semi-Rigid Mesh Size	Defines the mesh size for roof panels, and slabs when modeled as semi-rigid diaphragms.
Semi-Rigid Uniformity Factor	Defines the semi-rigid mesh uniformity for roof panels, and slabs when modeled as semi-rigid diaphragms.
Semi-Rigid Mesh Type	Defines the semi-rigid mesh type for roof panels, and slabs when modeled as semi-rigid diaphragms The options are: • QuadDominant • QuadOnly

Property	Description
	• Triangular
Wall Mesh Horizontal Size	Defines the horizontal mesh size for all meshed walls in the model - but can be overridden in individual wall properties.
Wall Mesh Vertical Size	Defines the vertical mesh size for all meshed walls in the model - but can be overridden in individual wall properties.
Wall Mesh Type	Defines the mesh type for all meshed walls in the model, (but can be overridden in individual wall properties): The options are: • Quad only • Tri only • Quad dominant

Level Properties

Use the **Level** properties to view or modify the basic properties of a level.

General	
Level	The height of the construction level above the base level
Floor	By setting a construction level to be a Floor you are indicating that it is a major level in the building. Floor levels are used to determine items such as your inter story height and positions from which column splices are laid out. If a level is not set to be a floor then no live load reductions will be accounted for in the beams at that level, or in the columns supporting that level. There can certainly be a number of levels that are clearly floor levels, but there could be many others that are not. For example you create intermediate levels in order to define: • half landing levels and stairs, • K Bracing - you require a construction level for the intermediate bracing connection points, • steps in the building floor levels. Where you define a level which is clearly not a floor, then you should not check the floor box.

General	
Check for drift	If this option is unselected, the sway/drift, seismic drift and wind drift check results for all column stacks and wall panels at the specific level are excluded from the tabular review data. In addition, the height of structure considered in the Overall Wind Drift check (page 679) is determined according to how this option has been set - the height used being the distance between highest level and the lowest level in the model that have **Check for drift** selected. **NOTE** This option is only displayed if the **Floor** option described above is selected.
Type	The level type can be: • T.O.S = Top of Steel • S.S.L = Structural Slab Level • T.O.F = Top of Foundation **NOTE** Slabs are modeled above the level when it is set to T.O.S or T.O.F but below the level when it is set to S.S.L
Short Name	Each construction level should be given a unique reference. Typically this might be a storey number, 1, 2 , 3 etc.
Long Name	Each construction level can also be given a name to further assist identification. 'First Floor', or 'Mezzanine' etc.
Name	Automatically generated from the short and long name. By default this will be used as the name in the **Structure** tree
Mesh 2-way Slabs in 3D Analysis	• On = 2-way slabs are meshed in the 3D building analysis and grillage chasedown analysis in addition to the FE chasedown analysis. • Off = 2-way slabs are only meshed in the FE chasedown analysis. **NOTE** The Sub Model in which the Level is contained determines the mesh parameters that are applied.
Include in Export	Determine whether Level should be included in CXL or IFC export. • Floor Only - include only if defined as a Floor. • Yes - always include. • No - never include. **NOTE** The lowest Level will always be exported.

General	
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in plane views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified level • Off = Grid & construction lines are always off for the specified level, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting **NOTE** This setting does **not** control the display of **Architectural Grids** which have a separate checkbox in Scene Content.
Decomposition> Keep solver model	Controls whether you will be able to review the solver model used for load decomposition between each analysis run: • On = the solver model is retained, you might choose to use this setting if you need to investigate any validation warnings that relate to load decomposition. • Off (default) = the solver model is discarded immediately after analysis has been performed.

Decomposition	
Keep solver model	Controls whether you will be able to review the solver model used for load decomposition between each analysis run: • On = the solver model is retained, you might choose to use this setting if you need to investigate any validation warnings that relate to load decomposition. • Off (default) = the solver model is discarded immediately after analysis has been performed.

Frame Properties

Use the **Frame** properties to view or modify the basic properties of a frame.

Property	Description
Name	Automatically generated, but can be replaced by **User name** if required.
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in plane views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified frame • Off = Grid & construction lines are always off for the specified frame irrespective of the Scene Content **Grid & Construction Lines** checkbox setting **NOTE** This setting does **not** control the display of **Architectural Grids** which have a separate checkbox in Scene Content.
Visible	Controls whether the frame view can be opened or not.

Slope Properties

Use the **Slope** properties to view or modify the basic properties of a frame.

Property	Description
Name	Automatically generated, but can be replaced by **User name** if required.
Show grids in plane view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in plane views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off • Off = Grid & construction lines are always off, irrespective of the Scene Content **Grid & Construction Lines** checkbox setting

Property	Description
Show grids in 3D view	Controls whether the **Grid & Construction Lines** checkbox in Scene Content has any effect in 3D views: • On = The Scene Content **Grid & Construction Lines** checkbox switches grid & construction lines on and off for the specified slope • Off = Grid & construction lines are always off for the specified slope irrespective of the Scene Content **Grid & Construction Lines** checkbox setting **NOTE** This setting does **not** control the display of **Architectural Grids** which have a separate checkbox in Scene Content.
Visible	Controls whether the slope view can be opened or not.

Sub Model Properties

Use the **Sub Models** properties to view or modify the selected sub model.

Property	Description
Override model's	Select this check box in order to override the Structure meshing properties in the current sub model.
Shell Mesh Size	Defines the shell mesh size for two way spanning slabs in the sub model.
Shell Uniformity Factor	Defines the shell mesh uniformity factor for two way spanning slabs in the sub model.
Slab Mesh Type	Defines the shell mesh type used in the sub model. These options are: • Quad only • Tri only • Quad dominant
Semi-Rigid Mesh Size	Defines the semi-rigid mesh size when slabs are modeled as semi-rigid diaphragms in the sub model.
Semi-Rigid Uniformity Factor	Defines the semi-rigid uniformity factor when slabs are modeled as semi-rigid diaphragms in the sub model.

Property	Description
Semi-Rigid Mesh Type	Defines the semi-rigid mesh type when slabs are modeled as semi-rigid diaphragms in the sub model. The options are: • Quad only • Tri only • Quad dominant

Beam properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See: ,
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Beam
Active	Clearing this option makes single span beams inactive in the analysis. See: Inactive members (page 306) **NOTE** Only displayed for single span members
Material type	• Steel • Concrete • Timber • General • Cold formed • Cold rolled
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 978)
Autodesign	See: ,

General	
Design section order (steel only)	The design order file from which a section size will be selected. **NOTE** Only displayed for Autodesign For details of managing order files, see: Manage design section orders (page 892)
Select bars starting from (concrete only)	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement. See: ,
Gravity only	Controls whether the member is defined for gravity combinations only, or gravity plus lateral: • On = designed for gravity combinations only • Off = designed for gravity and lateral combinations See:
Rotation	Rotation of the member about its local x axis. The default (Degrees0) aligns the major properties with the global Z axis, (provided that the member has not been specifically defined within an incline plane).
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 231)
Major snap level, Minor snap level (not concrete)	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset (not concrete)	Used to offset the member from the snap point in the major and minor axis.
Allow automatic join end 1,	When this check box is selected - the end in question will be automatically joined to a suitable connecting concrete beam end during design process or when the 'Beam Lines'

General	
end 2 (concrete only)	command is run, (providing the Beam Lines limiting criteria specified in Model Settings are met.)

All spans	
Section	The section size
Concrete type, Grade/Class	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Linearity	• Straight • Curved Major • Curved Minor
Chord height	This property is only displayed when 'Linearity' is curved major or curved minor. It is the perpendicular distance from the mid point of the chord baseline to the curve itself.
Maximum facet error	This property is only displayed when 'Linearity' is curved major or curved minor. It controls number of straight line elements that replace the curved member in the solver model. See: Analysis Model settings (page 1040)
Top flange cont. rest. (steel and composite beams only)	Define if the top flange is continuously restrained.
Alignment (concrete only)	
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Releases	
Free end 1, end 2	When this check box is selected - defines a cantilever end.
Fixity end 1, end 2	• Moment • Pin • Fully fixed See: Beam releases (page 949)

All spans	
Axial load release end 1, end 2	Check one end only to define an axial release.
Torsional load release end 1, end 2	Check one end only to define a torsional release.
My stiffness end 1, end 2	This property controls end fixity in the Major direction. It is only displayed when 'Fixity' is Fully fixed, or Moment. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness y end 1, end 2	When 'My stiffness' is set to Spring linear, this property allows you to specify the major direction stiffness in terms of a linear spring value. When 'My stiffness' is set to Partially fixed, this property allows you to specify the major direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Mz stiffness end 1, end 2	This property controls end fixity in the Minor direction. It is only displayed when 'Fixity' is Fully fixed. The choices are: • Fixed (default) • Spring linear • Partially fixed
Stiffness z end 1, end 2	When 'Mz stiffness' is set to Spring linear, this property allows you to specify the minor direction stiffness in terms of a linear spring value. When 'Mz stiffness' is set to Partially fixed, this property allows you to specify the minor direction stiffness as a percentage of a fully fixed connection. (% of 4EI/L).
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See:
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**

All spans	
	NOTE Default Cf factors are taken from the document ‘Wind Loads For Petrochemical And Other Industrial Facilities’ published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Load reductions	
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See:
Reduce imposed loads by (All other Head Codes)	This property is particularly applicable to the design of transfer beams. Although the percentage of imposed load reduction is not determined automatically for beams, this property allows you to specify the percentage manually. It can be applied to all, or individual spans. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced The reduced results are used in concrete beam design. See:
Deflection limits (steel only)	
Apply span \factor	With this option checked, the limit can be defined as a Relative span/factor.
Apply abs. limit	With this option checked, the limit can be defined as absolut value.
Limit for immediate live load deflection,	These options control how the deflection is calculated.

All spans	
Limit for total deflection affecting sensitive finishes Calculate total deflection at design time Calculate deflection after installation of finishes (ACI only)	
Torsion (steel only)	
Check for torsion, Apply rotational limit	Used to specify if the member should be checked for torsion, and also to apply a rotational limit if required. See:
Camber (steel only)	
Apply camber	Used to specify a camber to the beam if required. See:
Natural frequency (non-composite and composite beams only)	
Calculate natural frequency	For composite beams this is fixed to on (for EC and BS headcodes only) and is utilized during autodesign. The calculated natural frequency is displayed in the results viewer.
Check natural frequency against minimum	For composite beams this is fixed to on (for EC and BS headcodes only). For non-composite beams it can be used in conjunction with the 'Calculate natural frequency' property to impose an optional 2-step control, ‘calculate’ and ‘check’, which allows for the natural frequency value to be calculated (and displayed in Results Viewer) with or without the check. While the US headcode has no requirement for this check, it can still be requested if required. When performed, a simple (design model) approach is taken based on uniform loading and pin supports. This fairly simple calculation is provided to the designer for information only. The calculation can be too coarse particularly for long span beams and does not consider the response side of the behavior i.e. the reaction of the building occupants to any particular limiting value for the floor system under consideration. In such cases the

<table>
<tr><th colspan="2">All spans</th></tr>
<tr><td></td><td>designer has the option to perform a 1st Order Modal Analysis.
See:
•
•</td></tr>
<tr><td>Minimum natural frequency</td><td>The minimum value against which the natural frequency is checked (default 4Hz).</td></tr>
<tr><td>Include self weight (beam & slab)
Include other dead loads
Include live loads (US)
Include imposed loads (other headcodes)</td><td>The engineer can specify the percentages of each of these loads to be included in the calculation of the maximum static instantaneous deflection, δ</td></tr>
<tr><td>Factor for increased dynamic stiffness of the concrete flange</td><td>For composite beams this factor is applied to the beam's short term modular ratio</td></tr>
<tr><td>Effective width
Calculate effective width</td><td>For composite beams the effective width can either be entered directly or calculated from the geometry.
See:
•
•
•</td></tr>
<tr><td colspan="2">Size constraints (steel only)</td></tr>
<tr><td>Max depth,
Min depth,
Max width,
Min width</td><td>Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have. For instance for a composite beam you may want to ensure a minimum flange width of 150mm (6in). If so you would simply enter this value as the Minimum width, and Tekla Structural Designer would not consider sections with flanges less than this width for the design of this beam.
See:</td></tr>
</table>

All spans	
Apply max span/depth ratio Max span/ depth ratio	After setting a max span/depth ratio you can check the 'Apply' button for it to be considered by auto-design. During design, only sections which satisfy the maximum ratio limit will be selected. The setting can also be reviewed and/ or copied via Review View > Show/Alter State. See:
Instability factor (steel only)	
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Analysis & design control (concrete only)	
Structure supporting sensitive finishes (ACI and Eurocode only)	Any beam that supports or is attached to partitions or other constructions likely be damaged by large deflections should be identified as such by selecting this property.
	ACI - The deflection method applied to the beam depends on this setting as follows: • beams not required to support sensitive finishes adopt the simplified method. • beams required to support sensitive finishes adopt the rigorous method.
	Eurocode - The f2 parameter used in the deflection check depends on this parameter as follows: • When selected: f2 is calculated as MIN[1,7/L_{eff}] • When unselected: f2 will be taken as 1.0.
Increase reinforcement if deflection check fails (Eurocode BS and IS only)	Select in order to increase the reinforcement during the auto-design process if the deflection check fails.
Permissible increase in reinforcement	Specify the max percentage increase in reinforcement that is allowed in order to satisfy the deflection check.
Consider flanges	Select in order to consider flanges in the concrete beam design calculations - once checked additional fields are displayed for specifying an allowance for openings. Flange dimensions can only be calculated by editing the beam once it has been positioned and slabs have been defined. (In this case a ‘Calculate flanges’ button is also displayed, this can be clicked in order to automatically

All spans	
	calculate the flange dimensions based on the adjoining slabs.) See:
Include flanges in analysis	**NOTE** This property is only displayed when the ‘Consider flanges’ option has been selected. Select this check box to use flanged beam properties when the analysis is performed. See:
Isolated beam (ACI only)	**NOTE** This property is only displayed when the ‘Consider flanges’ option has been selected. Select this check box in order to apply ACI 318 clause 8.12.4. When the check is performed, if the flange geometry does not meet the requirements the flanges are ignored. See:
User defined flange (left/right)	**NOTE** This property is only displayed when ‘Include flanges in analysis’ has been checked. If you clear this check box the flange depth and the effective flange width are determined automatically. If you select the check box, two new properties are displayed for defining ‘Flange width’ and ‘Flange depth’. See:
Ignore lateral instability (Eurocode only)	This option allows you to ignore lateral instability for slender spans to EC2 clause 5.9(1). • When selected: the slender span check is excluded from design • When unselected (default): the slender span check is included.
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption. See:
Design parameters (concrete only)	

All spans	
Nominal cover beam top edge, bottom edge, section side, beam ends	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface. Different values of nominal cover can be specified to the beam edges, sides and ends.
Seismic	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 613)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Fire proofing	
Check for fire resistance	**NOTE** This property is only displayed for non-composite, simply supported rolled steel beams to Eurocodes. On • When the check is requested, the additional properties shown below are required.

Fire proofing	
	See: Off • no check is performed
Protected	On • When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See: Fire proofing (page 950) • If 'Check for fire resistance' is also selected, the check is performed using the 'protected' time interval for critical temperature iteration, as specified in Design Settings Off • If 'Check for fire resistance' is also selected, the check is performed using the 'unprotected' time interval for critical temperature iteration, as specified in Design Settings
Fire check (steel only, Eurocode only)	
Load reduction factor	0.65, or user input value.
Required time of fire exposure	R15, R30, R60, R90, or R120
Exposure	Exposed on three, or four sides

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Reinforcement (concrete beams only)	
Rib type - vertical, Class - longitudinal	Specifies the longitudinal reinforcement properties
Rib type - link, Class - link	Specifies the link properties.

Reinforcement (concrete beams only)	
Top longitudinal bar pattern, Bottom longitudinal bar pattern	Choose from standard patterns (which can be setup in Design Options) to control the top and bottom bar arrangement when the beam is auto-designed. See:

Span 1, 2, 3 etc.	
	In a multi-span beam properties can be entered for a specific span, over-riding those defined at the **All spans** level.

Beam releases

Releases at the two ends of a beam span can be set as follows:

- **Pin** - Pinned to the support or supporting member. This means pinned about the major and minor axes of the section but fixed torsionally.
- **Fully fixed** - with the following major (My), and minor (Mz) **stiffness** sub-options to allow definition of partial fixity in each direction if required:
  - **Fixed** - Encastré, all degrees of freedom fixed.
  - **Spring linear** - allows you to specify a linear spring stiffness value.
  - **Partially fixed** - allows you to specify partial fixity as a percentage of a fully fixed connection (%4EI/L)
- **Moment (pin Mz)** - Major axis moment connection, and pinned about the minor axis.
- **Moment (pin My)** - Minor axis moment connection, and pinned about the major axis.
- **Fully fixed (free end)** - Denotes a cantilever end. It is achieved by selecting Free end.

  (In a single span beam this box can only be checked if the opposite end is fully fixed.)
- **Continuous** - This setting is automatically applied when a continuous beam is created and effectively creates a non-editable fully fixed connection between the spans of the continuous member. The connection can only be edited by splitting the beam.

In addition to the above release options you are also able to apply a torsional release at either end by checking the appropriate box. Similarly an axial release can be applied to beams of all materials apart from concrete.

NOTE **Moment (pin My)** this (unusual) release type is not available in the Properties Window and can only be specified as follows:

1. Right-click the beam to display the context menu.
2. Choose Edit
3. From the Beam Property Dialog open the Releases page.
4. Check the Mz and uncheck the My degree of freedom at the desired end as required.

Fire proofing

Fire proofing is the application of fire resistance material on the surface of structural members in the form of:

- protective paints
- thin/thick film coating
- sprays
- fire boards

In Tekla Structural Designer fire proofing can be applied as a surface coat to members, defined by a thickness and density.

NOTE Fire proofing can be applied to all 1D member types apart from analysis elements, DELTABEAM, and FABSEC beams.

The additional self weight of fire protection material is then added to the "Self-weight - excluding slabs" loadcase.

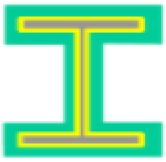
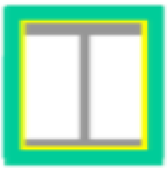
NOTE If **Calc Automatically** is unselected for the "Self-weight - excluding slabs" in the Loading dialog, neither the member self weight, or the fire proofing self weight are calculated.

How to apply fire proofing

Fireproofing is applied to a member by selecting ‘Protected’ under the 'Fire proofing' heading in the member properties. It can also be applied graphically using the Fire proofing attribute (page 737) in Show/Alter state.

Fire protection material details

Property	Description
Exposure	• Exposed on three sides (Applicable for beams only) • Exposed on all sides
Shape	Contour encasement

Property	Description
	 Hollow encasement 
Thickness	Thickness of fire proofing material
Density	Density of fire proofing material
Selfweight	Calculated self weight of fire proofing material per unit length.

Brace properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See:
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Brace
Active	Clearing this option makes the brace inactive in the analysis. See: Inactive members (page 306)
Material type	• Steel • Timber • General • Cold formed

General	
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 978)
Autodesign	See: ,
Design section order (steel only)	The design order file from which a section size will be selected. **NOTE** Only displayed for Autodesign For details of managing order files, see: Manage design section orders (page 892)
Rotation	Rotation of the member about its local x axis.
Alignment	
Global offset end 1, end 2	Can be used to model a physical offset with respect to the global axes at one or both ends of the member, (exceptions apply). See: Member global offsets (page 231)
Major snap level, Minor snap level	Defines the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.

All spans	
Section	The section size
Grade	The material grade
Connection	• Bolted • Welded
Compression only, Tension only	Specify if the brace is compression only, or tension only. **NOTE** Tension only and Compression only members are non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements

All spans	
Relaxation factor	Only displayed if the brace is tension only (default 0.0). Any value entered is not used unless you also check the ‘Use relaxation factors for tension only elements’ box in Analysis Options. **NOTE** It is highly unlikely that you would ever need to set a specific release factor for an individual brace. See 1st order non-linear settings in: Analysis Settings (page 1050)
Releases	
Fixity end 1, end 2	• Pinned
Torsional load release end 1, end 2	Check one end only to define a torsional release.
Vertical load release end 1, end 2	A vertical load release can be applied to the end of a V or A type brace pair so that they don’t prop other members against gravity loads, (you are prevented from releasing single braces, or other brace pairs in this way).
Include force in eccentricity moment end 1, end 2	Eccentricity moments in steel and precast columns as a result of beam connection eccentricities do not also consider the brace connection eccentricities unless this property is checked for the appropriate end of the brace. • When checked ON the brace axial force is resolved into a vertical force, multiplied by the connection eccentricity, to obtain a connection eccentricity moment. • When checked OFF the brace axial force does not contribute to the total connection eccentricity moment. See:

All spans	
	• Steel column connection eccentricity moments - overview • Precast column connection eccentricity moments - overview
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See:
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** **NOTE** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Load reductions	
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See:
Reduce imposed loads by (All other Head Codes)	Although the percentage of imposed load reduction is not determined automatically, this property allows

All spans	
	you to specify the percentage manually. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced See:
Compression	
Effectve length factor y-y, z-z	
Tension	
Net area	Specified as an effective net area or a percentage value.
Size constraints (steel only)	
Max depth, Min depth, Max width, Min width	Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have.
Seismic	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 613)
Apply (to check)	On

Utilization ratio	
	• When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Fireproofing	
Protected	When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See: Fire proofing (page 950)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Column properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Group	The name of the group to which the member belongs. See: ,
Top Level	Specifies the top level for the column.
Base Level	Specifies the bottom level for the column.
Plane	Indicates the Frame when the column has been placed in a Frame View.
Characteristic	Column
Material type	• Steel • Concrete

General	
	• Timber • General • Cold formed • Cold rolled
Construction, Fabrication	The available construction and fabrication options depend on the characteristic and material type selected, see Member characteristic, construction and fabrication properties (page 978)
Autodesign	See: ,
Design section order (steel only)	The design order file from which a section size will be selected. **NOTE** Only displayed for Autodesign For details of managing order files, see: Manage design section orders (page 892)
Select bars starting from (concrete only)	This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is 'on'. It applies to both longitudinal bars and links. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size. • Current - the auto design commences from the current bar arrangement. See: ,
Gravity only	Controls whether the member is defined for gravity combinations only, or gravity plus lateral: • On = designed for gravity combinations only • Off = designed for gravity and lateral combinations See:
Rotation	Rotation of the column about its local x axis. **NOTE** For a vertical column: • 0° - column local y aligns with global X • 90° - column local y aligns with global Y
Assume extra floors supported	Enter the number of extra floors supported.
Simple column (steel only)	See:

All stacks	
Section	The section size
Concrete type, Grade/Class (Concrete columns only)	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Alignment (concrete only)	
Major snap level, Minor snap level,	Define the major and minor alignment of the member relative to the insertion point.
Major offset, Minor offset	Used to offset the member from the snap point in the major and minor axis.
Face A cont. rest., Face C cont. rest. (steel only)	Indicates continuous restraint to Face A and/or Face C. See:
Releases	
Fixity Top, Fixity Bottom	• Pinned • Fixed • Moment (pin My) • Moment (pin Mz) See: Column releases (page 965)
Axial load release top	An axial (Fx) release is only allowed at the top of the column.
Torsional load release top, Torsional load release bottom	Check one end only to define a torsional release.
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. See:
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** **NOTE** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.

All stacks	
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Size constraints (steel only)	
Max depth, Min depth, Max width, Min width	Size Constraints are only applicable when Autodesign is checked. They allow you to ensure that the sections that Tekla Structural Designer proposes match any particular size constraints you may have.
Instability factor (steel only)	
Prevent out of plane instability	Define if out of plane stability is prevented. See:
Analysis & design control (concrete only)	
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/drift sensitivity calculations are also influenced by this assumption. See:
Apply rigid zones	Unless cleared, rigid zones are automatically created at the connection between the column and any connecting beams.
Design parameters (Eurocode only)	

All stacks	
Permanent load ratio option Relative humidity (RH) Age of loading	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. Age of loading is the age at which loading is applied. See: Design parameters (Eurocode only) (page 963)
Confinement reinforcement (concrete only)	
Provide support regions	• Checked - confinement reinforcement is designed separately in three regions. • Cleared - the same confinement reinforcement is designed for the whole stack.
Slenderness (concrete only)	
Direction 1, Direction 2	• Braced • Bracing
Direction 1 effective length factor, Direction 2 effective length factor	• Calculated • User input value
Stiffness (concrete only)	
Use slab for calculation upper, lower	For the unrestrained length calculation: • if a slab exists at a panel end, it can be ignored by unchecking the relevant box. • If no slab exists at that end, the setting is redundant - in which case the program considers the setting at the remote end of the next panel instead. See: Stiffness
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can

All stacks	
	override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all columns are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular column stacks from these calculations to avoid spurious results associated with very small stack lengths. You can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Stacks' to exclude the entire column, or you can exclude a particular stack by clearing the check box located under that stack only.
Nominal cover	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where relevant) and the nearest concrete surface.
Seismic	

All stacks	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
Shear demand/shear capacity ratio in Dir 1 Shear demand/shear capacity ratio in Dir 2 (ASCE7 code only)	When working to the ASCE7, the engineer can directly define the shear demand / capacity ratio (beta) in each direction. The default value of 1.0 could be over-conservative. The setting is applicable to all years of the ASCE7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 613)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Fireproofing	
Protected	When members are set as 'Protected', self-weight of the members is increased by the weight of fireproofing. See: Fire proofing (page 950)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Reinforcement	
Rib type - vertical, Class - vertical	Specifies the vertical reinforcement properties
Rib type - link/ confinement, Class - link/ confinement	Specifies the link/confinement properties.

Stack 1, 2, 3 etc.	
	In a multi-stack column properties can be entered for a specific stack, over-riding those defined at the **All stacks** level.
Splice, Splice offset	Used to indicate a splice at the bottom of a stack, and the offset above the floor level. See:

Level 1, 2, 3 etc.	
Load reductions	
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See:
Count the floor as being supported (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the live/imposed load reductions.
Eccentricity (steel only)	
Apply face A eccentricity, etc.	See:

Design parameters (Eurocode only)

Located under the Design parameters heading in the column properties, the following parameters relating to shrinkage and creep can be specified for individual members.

NOTE The Design Parameters described below are only applicable when the Head Code is set to Eurocode.

Permanent Load Ratio

The permanent load ratio is used in the equation for determining the service stress in the reinforcement, which is in turn used in table 7.3N to determine the maximum allowable centre to centre bar spacing. It is also used to calculate the effective creep ratio which appears in the column slenderness ratio calculations.

It is defined as the ratio of quasi-permanent load to design ultimate load.

i.e. SLS/ULS = (1.0Gk + y 2Qk) / (factored Gk + factored Qk*IL reduction)

If Qk is taken as 0 then:

SLS/ULS = (1 / 1.25) = 0.8

Hence, setting the permanent load ratio to 0.8 should provide a conservative upper bound for all cases.

When determining this ratio more precisely, consideration should be given to the amount of IL reduction specified, for example (assuming Gk = Qk and y 2 = 0.3):

For 50%IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5*0.5) = 0.65

For no IL reduction,

SLS/ULS = (1 + 0.3) / (1.25 + 1.5) = 0.47

NOTE The program defaults to a permanent load ratio of 0.65 for all members - you are advised to consider if this is appropriate and adjust as necessary.

Relative Humidity

Typical input range 20 to 100%

Age of Loading

This is the age at which load is first applied to the member.

The Age of Loading should include adjustments necessary to allow for cement type and temperature as defined in EC2 Annex B.

NOTE The program defaults the Age of Loading to 14 days for all members - you are advised to consider if this is appropriate and adjust as necessary.

Column releases

Releases in columns can be set as follows:

Moment Releases

- **Pinned** - Pinned to the adjacent stack and connecting members. This means pinned about the major and minor axes of the section but fixed torsionally.
- **Fixed** - fully fixed
- **Moment (pin My)** - Pinned about the y axis only
- **Moment (pin Mz)** - Pinned about the z axis only.

NOTE While **Moment (pin My)** and **Moment (pin Mz)** releases are less usual, for continuous columns composed of two or more stacks, they have been provided in response to customer feedback, to allow any/all column stacks to be pinned in one direction whilst the other direction remains fixed.

When such a release is applied, a Warning status is issued in the Column Property dialog. This is intentional and does not prevent analysis or design - as the Warning tooltip states, it is to prompt the engineer to consider the connection details at this location in the column.

Translational releases

- Fy and Fz releases are prevented in all stacks i.e. no ‘roller’ connections.
- Fx (axial) release allowed at the top of the topmost stack only.
- Mx (torsional release allowed in all stacks.

Concrete meshed and mid-pier wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	• Meshed Shear Wall • Mid-pier Shear Wall **NOTE** Mid-pier shear walls must be rectangular in a vertical plane. Meshed shear walls can be vertical or sloping.

<table>
<tr><th colspan="2">General</th></tr>
<tr><td>Fabrication</td><td>Choice of:
• Cast-in-place
• Precast
NOTE Design of precast members is beyond scope in the current release</td></tr>
<tr><td>Autodesign</td><td>• Cleared - the specified reinforcement will be checked during the design process.
• Checked - reinforcement will be designed during the design process.
See:</td></tr>
<tr><td>Select bars starting from</td><td>This option controls the starting point for auto-design procedures and is therefore only displayed if Automatic design is ‘on’. It applies to both longitudinal bars and links.
• Minima (default) - removes the current arrangement and begins with the minimum allowed bar size.
• Current - the auto design commences from the current bar arrangement.
See:</td></tr>
<tr><td>Assume extra floors supported</td><td>Enter the number of extra floors supported.</td></tr>
<tr><td>Rotation
(mid-pier only)</td><td>• 0° - wall spans horizontally
• 90° - wall spans vertically</td></tr>
<tr><td>AutomaticGenerate Support</td><td>• Cleared - a support will only be created if the Generate support property is checked.
• Checked - a support will only be created if no members/ slabs capable of providing support exist under the wall.</td></tr>
<tr><td>Generate support</td><td>• Cleared - no support is created under the wall.
• Checked - a support is created under the wall.
NOTE When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall.
NOTE When a support is created, its degrees of freedom are as specified in the ‘Wall support’ area of the wall properties.</td></tr>
</table>

General	
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See:
Plane	Indicates the grid along which the wall is placed.

All panels	
Concrete type	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Grade	The concrete grades that are available here are set from the Materials button on the Home ribbon.
Thickness	The thickness of the wall.
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/ drift sensitivity calculations are also influenced by this assumption.
End 1 extension	The amount the wall is extended or trimmed back from end 1. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 239)

All panels	
End 2 extension	The amount the wall is extended or trimmed back from end 2. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 239)
Reinforcement Layers	Number of layers of reinforcement to be used in the wall: • 1 • 2
Releases	
Minor Top	• Fixed • Pinned • Continuous (incoming members pinned) **NOTE** The 'Continuous' option is only available for FE meshed walls.
Minor Bottom	• Fixed • Pinned • Continuous (incoming members pinned) **NOTE** The 'Continuous' option is only available for FE meshed walls.
Analysis & design control	
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/

All panels	
	drift sensitivity calculations are also influenced by this assumption. See:
Design parameters	
Permanent load ratio option Relative humidity (RH) Age of loading	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary. Age of loading is the age at which loading is applied. See: Design parameters (Eurocode only) (page 963)
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all walls are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small panel lengths. You can either clear the check box located under 'All Panels' to exclude the entire wall, or you can exclude a particular panel by clearing the check box located under that panel only.

All panels	
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Panels' to exclude the entire wall, or you can exclude a particular stack by clearing the check box located under that panel only.
Confinement reinforcement	
Provide support regions	• Checked - confinement reinforcement is designed separately in three regions. • Cleared - the same confinement reinforcement is designed for the whole stack.
Slenderness	
Major (Minor)	• Braced • Bracing
Effective length factor direction Major (Minor)	• Calculated • User input value
Stiffness	
Use slab for calculation (upper major/ minor, lower major/minor)	For the unrestrained length calculation: • if a slab exists at a panel end, it can be ignored by unchecking the relevant box. • If no slab exists at that end, the setting is redundant - in which case the program considers the setting at the remote end of the next panel instead. See:
Nominal cover	The nominal concrete cover is the distance between the surface of the reinforcement closest to the nearest concrete surface (including links and surface reinforcement where

All panels	
	relevant) and the nearest concrete surface.
Seismic	
In a seismic force resisting system	If this is the case, select the check box, and then specify the SFRS direction and type. **NOTE** Design of members in seismic force resisting systems is only supported for the ACI/AISC Head Code in the current release.
Shear demand/shear capacity ratio in Dir 1 Shear demand/shear capacity ratio in Dir 2 (ASCE7 code only)	When working to the ASCE7, the engineer can directly define the shear demand / capacity ratio (beta) in each direction. The default value of 1.0 could be over-conservative. The setting is applicable to all years of the ASCE7 code and has the following default and limits: Default = 1; Min = 0.001; Max = 10.

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 613)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Wall support	
Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear

Reinforcement	
Include end zones	• Cleared - the wall is designed without end zones of reinforcement • Checked - the wall is designed with end zones of reinforcement
Wall zone	
Form, type class	Specifies the wall zone reinforcement properties
End zone	
Form, type class	Specifies the end zone reinforcement properties (if end zones have been requested).

Panel 1, 2, 3 etc.	
	In a multi-stack wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Count the floor as being supported	
Top level (Interediate levels)	If checked, the floor will be treated as supported when calculating the live/imposed load reductions.

Count the floor as being supported	
Base level (Head Code Eurocode, BS or IS)	

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

Meshing (meshed walls only)	
Override Model's	Select this check box to override the default wall mesh size that is specified in the Structure properties.
Wall Mesh Horizontal Size	Used to override the default wall horizontal mesh size (1.000m).
Wall Mesh Vertical Size	Used to override the default wall vertical mesh size (1.000m).
Wall Mesh Type	Available mesh types: • Quad only • Tri only • Quad dominant See: How meshed walls are represented in solver models (page 577)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Concrete core properties

General	
Name	The automatically generated name for the core.

General	
User Name	Can be used to override the automatically generated name if required.
Rotation	Defines the axis system for reporting the core results: • Dir 1/2 - Main Building Directions • Principal 1/2 - Major and minor local axis • Angle - in Global Coordinate System

Level 1, Level 2 etc.	
Centroid above, Centroid below	Every core will have at least 2 core levels, with a core section for each level. The centroids of the core sections are reported above and below each level.

General wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	• Meshed Shear Wall
Material type	• General
Assume extra floors supported	Enter the number of extra floors supported.
AutomaticGenerate Support	• Cleared - a support will only be created if the **Generate support** property is checked. • Checked - a support will only be created if no members/ slabs capable of providing support exist under the wall.
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-

General	
	pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
Plane	Indicates the grid along which the wall is placed.

All panels	
Grade	The grades that are available for General materials are set from the Materials button on the Home ribbon.
Thickness	The thickness of the wall.
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
Assume cracked	Assuming concrete sections are cracked has a direct affect on the analysis; smaller **Modification Factors** are applied to cracked sections causing an increase in deflection. Indirectly the design can also be affected because the sway/ drift sensitivity calculations are also influenced by this assumption.
End 1 extension	The amount the wall is extended or trimmed back from end 1. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 239)

All panels	
End 2 extension	The amount the wall is extended or trimmed back from end 2. • A positive extension extends the wall length beyond its insertion point. • A negative extension trims the wall back from the insertion point. See: Specify extensions and releases (page 239)
Releases	
Minor Top	• Fixed • Pinned • Continuous (incoming members pinned)
Minor Bottom	• Fixed • Pinned • Continuous (incoming members pinned)
Sway and drift checks	
Merge with stack below	Columns are divided into stacks and walls are divided into panels at floor levels where members or slabs connect to the column/wall. For the purposes of Sway/Drift, Wind Drift and Seismic Drift checks only you can override the default stack/panel divisions by merging a stack/panel with a lower stack/panel. The length of the combined stack/panel is then used in the checks.
Sway/Seismic drift checks	By default all stacks of all walls are taken into account in the calculation to determine the sway sensitivity of the building. The results of this calculation being accessed from the Review toolbar. This parameter provides a facility to exclude particular wall panels from these calculations to avoid spurious results associated with very small panel lengths. You can either clear the check box located under 'All Panels' to exclude the entire wall, or

All panels	
	you can exclude a particular panel by clearing the check box located under that panel only.
Wind drift check, Wind drift ratio	Wind drift is automatically checked against the specified limiting ratio, (which can be set differently for different columns). The results of this calculation are accessed from the Review toolbar. If you don't want this check to be performed you can either clear the check box located under 'All Panels' to exclude the entire wall, or you can exclude a particular stack by clearing the check box located under that panel only.

Panel 1, 2, 3 etc.	
	In a multi-stack wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Count the floor as being supported	
Top level (Interediate levels) Base level (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the live/imposed load reductions.

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

Meshing	
Override Model's	Select this check box to override the default wall mesh size that is specified in the Structure properties.
Wall Mesh Horizontal Size	Used to override the default wall horizontal mesh size (1.000m).
Wall Mesh Vertical Size	Used to override the default wall vertical mesh size (1.000m).

Meshing	
Wall Mesh Type	Available mesh types: • Quad only • Tri only • Quad dominant See: How meshed walls are represented in solver models (page 577)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Member characteristic, construction and fabrication properties

The following table lists the available permutations of characteristic, construction and fabrication for each material type.

Material	Characteristic	Construction	Fabrication	Notes
Concrete	Beam	Concrete beam	Cast-in-place	
			Post tensioned	Not designed
			Precast	Not designed
		Coupling beam	Cast-in-place	Coupling beams are the only beams considered by the Assisted core function. If you set the Construction as a Coupling beam when part of a core, you have made an engineering decision that you are satisfied the section is sufficiently stiff to provide rigid section behavior - Tekla Structural Designer does not check for this.

Material	Characteristic	Construction	Fabrication	Notes
	Column	Concrete column	Cast-in-place	
			Precast	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed
Steel	Beam	Non-composite beam	Rolled	
			Plated	
			Westok cellular	
			Westok plated	
			Fabsec	Not designed
		Composite beam	Rolled	
			Plated	
			Westok cellular	Not designed
			Westok plated	Not designed
			Fabsec	Not designed
	Column	Non-composite column	Rolled	
			Plated	
			Concrete filled	Not designed
			Concrete encased	Not designed
		Composite column	Rolled	Not designed
			Plated	Not designed
	Brace	Steel brace	Rolled	
	Steel joist	Steel joist	Steel joist	Standard, Special or Girder types
	Gable post	Steel gable post	Rolled	
	Parapet post	Steel Parapet post	Rolled	
	Truss member top	Steel Truss member top	Rolled	
	Truss member bottom	Steel Truss member bottom	Rolled	

Material	Characteristic	Construction	Fabrication	Notes
	Truss member side	Steel Truss member side	Rolled	
	Truss member internal	Steel Truss member internal	Rolled	
	Analysis Element	Not applicable	Not applicable	Not designed
Cold formed	Beam	Cold formed beam member	Cold formed	
	Column	Cold formed column member	Cold formed	
	Brace	Cold formed brace member	Cold formed	
	Gable post	Steel gable post	Cold formed	
	Parapet post	Steel Parapet post	Cold formed	
	Truss member top	Steel Truss member top	Cold formed	
	Truss member bottom	Steel Truss member bottom	Cold formed	
	Truss member side	Steel Truss member side	Cold formed	
	Truss member internal	Steel Truss member internal	Cold formed	
	Analysis Element	Not applicable	Not applicable	Not designed
Cold rolled	Beam	Cold rolled beam member	Cold rolled	Not designed

Material	Characteristic	Construction	Fabrication	Notes
	Column	Cold rolled column member	Cold rolled	Not designed
	Eaves beam	Eaves beam	Cold rolled	Not designed
	Purlin	Purlin	Cold rolled	Not designed
	Rail	Rail	Cold rolled	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed
Timber	Beam	Timber beam	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Column	Timber column	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Brace	Timber brace	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Gable post	Timber gable post	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Truss member top	Timber Truss member top	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Truss member bottom	Timber Truss member bottom	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed

Material	Characteristic	Construction	Fabrication	Notes
	Truss member side	Timber Truss member side	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Truss member internal	Timber Truss member internal	Timber	Not designed
			Glulam	Not designed
			Structural composite lumber	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed
General	Beam	General material beam	Not applicable	Not designed
	Column	General material column	Not applicable	Not designed
	Brace	General material brace	Not applicable	Not designed
	Analysis Element	Not applicable	Not applicable	Not designed

Slab item properties

General	
Name	The automatically generated name for the slab item.
User Name	Can be used to override the automatically generated name if required.
Rotation angle (one-way slabs) or X direction (two way slabs)	This property is used for the following where appropriate: • Span direction for 1-way load decomposition • To determine the 2D solver element local axes in the solver model • Bar direction for Slab on Beam and Flat Slabs. Different angles can be specified for different panels within the slab.

General	
	NOTE When SpanDirection is switched on in Scene Content, the rotation angle/X direction is represented by a blue arrow. See: and Modify slab/panel span direction (page 274)
Include in diaphragm	If this option is cleared, the slab item does not participate in diaphragm action. All nodes linked to the mat item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. **NOTE** Vertical offsets are not structurally significant
Auto-design	For panels in Auto-design mode, As,prov is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For panels not in *Autodesign* mode, the result will be a pass or fail.
Select bars starting from	This option controls the starting point for auto-design procedures.
Plane	Indicates the level at which the slab is placed.

Slab general	
Name	The automatically generated name for the mat item.
User Name	Can be used to override the automatically generated name if required.
Slab type	• Precast • Steel deck • Timber deck • Composite slab • Foundation mat
Deck type	The deck type depends on the slab type as follows: • Precast concrete planks: Precast, • Steel plate: Steel deck, • Timber: Timber deck,

Slab general	
	• Profiled metal decking: Composite slab, • Reinforced concrete: Slab on beams, Flat slab, Foundation mat • Post tension: Slab on beams, Flat slab, Foundation mat
Decomposition	Decomposition choices depend on the slab type as follows: • One-way only: Precast, Timber deck, Composite slab • One-way or two way: Steel deck, Slab on beams • Two-way only: Flat slab, Foundation mat

Slab parameters	
Slab properties(general)	
Overall depth	Specifies the slab thickness.
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • Rigid • Semi-rigid • None
Slab properties (concrete slabs)	
Concrete type	• Normal • Lightweight
Grade/Concrete class	Specifies the concrete grade.
Concrete aggregate type	Specifies the aggregate type.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density
Long term elastic modulus = Ecm divided by... composite slab and precast only)	Factor by which the short term elastic modulus is divided to obtain the long term modulus .
Ponding allowance	If required, choose either value or percentage and then specify actual ponding allowance accordingly.

Slab parameters	
option, Ponding allowance value (composite slab only)	
Apply ponding to construction stage (composite slab only)	• On = ponding allowance applied at construction stage
Apply ponding to composite stage (composite slab only)	• On = ponding allowance applied at composite stage
Decking properties (where applicable)	
Country, Manufacturer, Reference, Gauge	Defines properties of decking (where applicable).
Reinforcement in slab for crack control or fire requirements (composite slab, and precast with topping specified only)	
Type, Rib type, Mesh type Class, Bar size	Defines reinforcement for crack control or fire requirements
Topping details(precast only)	
Topping	• None • Non-structural • Structural
Depth	Depth of topping when topping other than 'None' is selected.
Design parameters (Head Code Eurocode)	
Permanent load ratio	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width	• 0.2 • 0.3 • 0.4
Deflection parameters	

Slab parameters	
Restraint constant (ACI only)	See:
Restraint type (Eurocode only)	See:
Cement class (Eurocode only)	See:
Material properties	
Modulus of elasticity, Poisson's ratio, Shear moduls, Coefficient of thermal expansion	Properties specific to timber and steel decks.

Design parameters	
Adjustment ratio direction X ,Adjustment ratio direction Y (slab on beam only)	These factors are applied to the enclosing lengths in X and Y in order to manually adjust the X and Y direction span in the span-effective depth check. See Slab on beam idealized panels
Enclosing length X, Enclosing length Y (slab on beam only)	The automatically calculated span length in the X direction, and Y direction. See Slab on beam idealized panels
Adjusted length X, Adjusted length X (slab on beam only)	The adjusted span lengths in the X and Y directions. See Slab on beam idealized panels
Edge category start X, Edge category start Y (slab on beam only)	The assumed support condition at the start of the span in the X direction, and Y direction. See Slab on beam idealized panels

Design parameters	
Edge category end X, Edge category end Y (slab on beam only)	The assumed support condition at the end of the span in the X direction, and Y direction. See Slab on beam idealized panels
Average stiffness ratio (Slab on beam ACI/AISC only)	In the current release of Tekla Structural Designerr the average stiffness ratio is a user defined value. (Default value = 1.0). The way in which the minimum thickness is calculated directly depends on this value.

Live/Imposed load reduction	
Reduce live/ imposed loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. **NOTE** For mats, the bearing pressures are reduced as well.

Reinforcement	
	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to “none” if required). Select/clear Outside layer in X direction as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)

Utilization ratio	
Apply (to autodesign)	On

Utilization ratio	
	• When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 613)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

See also

Foundation mat properties (page 988)

Foundation mat properties

General	
Name	The automatically generated name for the mat item.

General	
User Name	Can be used to override the automatically generated name if required.
Foundation type	Mat
Rotation angle	Specifies the orientation of reinforcement. Different angles can be specified for different panels within the mat. See:
Include in diaphragm	If this option is cleared, the mat item does not participate in diaphragm action. All nodes linked to the mat item will be able to displace independently of the diaphragm.
Override slab depth	By default all panels in a slab adopt the same depth. Checking this option allows the selected panel to have a different depth and vertical offset.
Vertical offset	Only available when 'Override slab depth' is checked. A positive offset raises the slab panel surface, a negative offset drops it. **NOTE** Vertical offsets are not structurally significant
Auto-design	For panels in Auto-design mode, As,prov is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. For panels not in *Autodesign* mode, the result will be a pass or fail.
Plane	Indicates the level at which the mat is placed.

Slab general	
Name	The parent foundation mat name.
User Name	Can be used to override the automatically generated name if required.
Slab type	***Foundation mat***
Deck type	• Reinforced concrete • Post tension **NOTE** Design of post tensioned slabs is beyond scope in the current release.
Decomposition	Two-way

Slab parameters	
Slab properties	
Overall depth	Specifies the slab thickness.
Concrete type	• Normal • Lightweight
Grade/Concrete class	Specifies the concrete grade.
Concrete aggregate type	Specifies the aggregate type.
Concrete density class	For normal weight concrete only, specifies the density class.
Dry density	Specifies the dry concrete density
Wet density	Specifies the wet concrete density
Diaphragm option	Sets the default diaphragm action for all slab items within the parent slab. • Rigid • Semi-rigid • None
Design parameters (Head Code Eurocode)	
Permanent load ratio	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width	• 0.2 • 0.3 • 0.4
Deflection parameters	
Restraint constant	See:
Restraint type (Eurocode only)	See:
Cement class (Eurocode only)	See:
Soil parameters	
Use Ground Bearing Springs	Select to specify that the mat is ground bearing.
Allowable Bearing Pressure	The allowable bearing pressure is only required when the 'Use Ground Bearing Springs' option is selected.

Slab parameters	
Ground Stiffness The following properties are only displayed when the 'Use Ground Bearing Springs' option is selected.	
Type	• Spring Linear • Spring Non-Linear
Stiffness	Only displayed when the Type is set to 'Spring Linear'.
Stiffness -ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Tension limit -ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Stiffness +ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Compression limit +ve	Only displayed when the Type is set to 'Spring Non-Linear'.
Horizontal Support	• Fixed • Free • Spring
% of vertical spring stiffness	This field is displayed when the Horizontal Support is set to 'Spring'.

Live/Imposed load reduction	
Reduce live/imposed loads by	This property is applicable for the design of transfer slabs and mats. Although the percentage of live load reduction is not determined automatically, this property allows you to specify the percentage manually. It can be applied to an individual slab item - you don't have to apply a single value throughout a slab or mat (this can be important if you have discrete transfer panels within a large slab area). • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced • The reduced results are used in slab design. **NOTE** For mats, the bearing pressures are reduced as well.

Reinforcement	
	Slabs panels can potentially have 4 layers of background reinforcement, (however any of the layers/directions can be set to “none” if required). Select/clear Outside layer in X direction as appropriate to indicate if the outside layer is in the X or Y direction. (The outside layer can be set differently for the top and bottom bars if required.)

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 613)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

All edges, Edge 1, Edge 2 etc.	
Linear	Uncheck in order to specify curved edges.
Curvature	Defines the amount of curvature to all edges. Only displayed when Linear is cleared.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Slab/Mat overhang properties

General	
LengthofBeam	• When checked, the overhang is created by a single click on a supporting beam along the the slab edge. • When unchecked, the overhang is created by clicking two points along the slab edge.
EdgeParallel	• When checked, the overhang is created with a straight edge. • When unchecked, the overhang is created with a curved edge.
Curvature	Specifies the overhang curvature, (when EdgeParallel is unchecked).
Tapered	• When checked, the overhang varies in width. (Width1 to Width2) • When unchecked, the overhang is a constant width (Width1).
Width1	The width at end 1 of the overhang.
Width2	The width at end 2 of the overhang, (when tapered).

Pad base strip base and pile cap properties

Property	Description
General	
Foundation Type	Isolated Pad Base (not editable). Isolated Pile Cap (not editable).
Auto-design depth	• When Autodesign is selected an iterative procedure is used to determine the depth. If the shear design fails the depth is increased until either the check passes, or the maximum depth specified in Design Options has been reached. • When Autodesign is not selected (i.e. check mode), the existing depth is retained and Tekla Structural Designer determines if it is sufficient.

Property	Description
	NOTE In Design Options - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed. **NOTE** When Design Member on the right-click context menu is used to design an individual base, the depth will be checked or designed according to the auto-design depth setting.
Auto-design size	• When Autodesign is selected an iterative procedure is used to determine the base size. If the bearing check fails the size is increased until either the check passes, or the maximum side length specified in Design Options has been reached. • When Autodesign is not selected (i.e. check mode), the existing size is retained and Tekla Structural Designer determines if it is sufficient. **NOTE** In Design Options - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their depth auto-designed. **NOTE** When Design Member on the right-click context menu is used to design an individual base, the size will be checked

Property	Description
	or designed according to the Auto-design size setting.
Auto-design piles (Pile cap only)	Autodesign piles automatically determines the quantity and size of piles. • Cleared - the specified number of piles at their specified positions will be checked during the design process. • Checked - the number of piles and their positions under the pile cap will automatically determined during the design process. When this check box is selected, the actual procedure used will depend on the Pile auto-design method that has been specified in Design Options > Concrete Foundations > Isolated Foundations Piles The **Minimise number of piles** method commences by first selecting the pile with the smallest load capacity in the Pile Catalogue. Pile positioning is attempted using the user defined pile arrangements. If one of the limitations is exceeded the pile with the next smallest load capacity in the catalogue is selected and the process starts again. This is repeated until the pile loading check passes. The **Minimise pile capacity** method commences by first selecting the pile with the smallest load capacity in the Pile Catalogue. Pile positioning is attempted using the user defined pile arrangements. If one of the limitations is exceeded one pile is added to the pile group and the process starts again. This is repeated until the pile loading check passes.
Select size/depth starting from	This option only appears when either Autodesign depth or Autodesign size

Property	Description
	is selected. It sets the autodesign start point for the depth, or size, or both. The options are: • Minima (default) • Current Selecting 'Minima' removes the current depth/size and begins with the minimum allowed depth/size specified in Design Options. For both options the auto-design depth increment is that specified in Design Options> Foundations Isolated Foundations> Foundation Size
Autodesign reinforcement	This setting applies to top and bottom reinforcement, but reinforcement in either location can still be set to none. If enabled, auto-design of reinforcement occurs after the base size and depth have been established. • When Autodesign reinforcement is enabled an iterative procedure is used to determine the reinforcement. If the bending design fails the reinforcement size gets increased and spacing gets decreased until these checks pass. Iterations continue until either a satisfactory bar configuration has been achieved, or the maximum bar size and minimum spacing specified in Design Options has been reached. At this point the depth gets increased and the above procedure is repeated. **NOTE** If the smallest bar at largest spacing is insufficient then the next bar size at max spacing is used. If the largest bar is reached the spacing gets reduced and the smallest bar is used. Then the bar

Property	Description
	size gets increased until the largest bar size reached. • When Autodesign reinforcement is disabled (i.e. check mode), the existing reinforcement provision is retained and Tekla Structural Designer determines if it is sufficient. **NOTE** In Design Options - if Isolated Foundations have been set to be designed using groups, then if at least one base in the group is set to have its depth auto-designed the whole group will have their reinforcement auto-designed. **NOTE** When Design Member on the right-click context menu is used to design an individual base the reinforcement is always designed (irrespective of the Autodesign reinforcement setting).
Select bars starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Autodesign reinforcement is selected. • Minima (default) - removes the current arrangement and begins with the minimum allowed bar size specified in Design Options. • Current - the auto design commences from the current bar arrangement.
Foundation Parameters	

Property	Description
Eccentricity in Y direction	Specifies the eccentricity of the base in the Y direction. (For strip bases this is along the wall)
Eccentricity in X direction	Specifies the eccentricity of the base in the X direction. (For strip bases this is across the wall)
Rotation angle	Specifies the rotation of the base: • Pad base: relative to that of the support (which is controlled by the rotation angle specified for column). • Strip base: the angle of the base about global Z.
Shape Pad base and pile cap only)	Specifies the base shape in plan: • Square • Rectangular
Length in Y direction (Pad base and pile cap only)	Specifies the size of the pad in the Y direction.
Length in X direction (Pad base and pile cap only)	Specifies the size of the pad in the X direction.
Width (Strip base only)	Specifies the width of the base
Depth	Specifies the depth of the base.
Concrete class	The concrete grade. The concrete grades that are available are set from the Materials button on the Home ribbon.
Use loaded area (Pad base only)	Selecting this option allows you to specify a user override of the punching shear area - defined using the Loaded Area Breadth and Loaded Area Depth properties. This feature can be used in both pad bases and pile caps to, for example, consider unmodeled base plates or pedestals.
Sliding in Y prevented / Sliding in X prevented (Pad base only)	If sliding is prevented by some other means (e.g. if attached to a floor slab in one or other direction) then this

Property	Description
	can be indicated by selecting the appropriate direction. **TIP** If the direction is not clear make the base non-rectangular so that length in X and hence X-direction is clearer. Selecting this option sets the shear force for the particular direction to zero. This affects the sliding check, but actually affects all checks because there is no additional overturning moment developed due to shear at top of base.
Overturning about X prevented / Overturning about Y prevented (Pad base only)	This option only appears when the Sliding in Y prevented / Sliding in X preventedoption is selected. Selecting this option indicates that overturning is also prevented by some other means for the selected direction, which sets the support moment for that direction to zero. This affects all checks because there is no overturning moment considered.
Permanent load ratio option (Eurocode only)	You are required to supply a value for the permanent load ratio parameter. A default of 0.65 has been assumed, but you are advised to consider if this is appropriate and adjust as necessary.
Maximum crack width	• 0.2 • 0.3 • 0.4
Piles (Pile cap only)	
Pile type	Pile types that have previously been specified in the Pile Catalogue are available for selection. You can choose <New...> to add additional types.
User defined arrangement	Specifies the eccentricity of the pile cap in the X direction.

Property	Description
Pile arrangement	Opens the Pile arrangement dialog for specifying the number of piles, and spacing, pile type and principal direction.
Number of piles	Specifies the number of piles.
Principal direction	Specifies the principal direction.
Pile spacing	The dimension between pile centers.
Shape	For three piles only, you can choose to specify either a triangular. or rectangular pile cap.
Reinforcement	
Type	• Mesh • Loose Bars • None
Rib type (Head Code Eurocode, BS or IS)	• Plain • Type 1 • Type 2
Rib type (Head Code ACI)	• Plain • Deformed
Bar type	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size, spacing, Mesh type etc.	The actual reinforcement provided in each of the layers is indicated here.
Top, Bottom, Side cover	Nominal cover to reinforcement.
Soil Parameters	
Soil unit weight	Soil unit weight
Characteristic friction angle	Characteristic friction angle
Presumed bearing resistance (EC Head Code)	The presumed bearing resistance is only displayed and used when the ‘presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)’ is selected in Design Options.
Surcharge	
Soil Surcharge depth Other permanent Permanent surcharge load	Drop-list controls allow each surcharge load to be assigned to a specific loadcase. The partial safety factors defined in the design combinations for these

Property	Description
Other variable Variable surcharge load	cases are then used for these loads in all design calculations. The assigned loadcases are initially set to “None” and must be user-selected. Where no appropriate Dead or Live loadcase exist in the model and so no loadcase can be assigned, zero partial factor values are used in the calculations.
Design shear strength of soil angle (ACI & BS Head Codes)	Design shear strength of soil angle
Allowable bearing capacity (ACI & BS Head Codes)	Allowable bearing capacity
Bearing capacity A1 - STR (EC Head Code)	Allowable bearing capacity A1. Only displayed and used when the ‘presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)’ is unselected in Design Options.
Bearing capacity A2 - GEO (EC Head Code)	Allowable bearing capacity A2. Only displayed and used when the ‘presumed bearing capacity method (EN 1997 - 1 cl. 6.5.2.4)’ is unselected in Design Options.
UDA	A customizable list of the attributes that can be applied to individual members and panels.

Line ancillary properties

General	
Name	The automatically generated name for the ancillary.
User Name	Can be used to override the automatically generated name if required.
Width, W	Width of ancillary
Type	The following types are available: • Walkways/catwalks[1] • Ladders with cage • Ladders without cage • Access platforms[1]

General	
	• Operating platforms (storage laydown)[1] • Operating platforms (standard)[1] • Steel stair flight/landing[2] • Concrete stair flight/landing[3] • Timber stair flight/landing[3] • Lines of pipework[4] • Lines of cable tray[5] • Other [1] These types expose the **Guardrail** flag below. [2] This type exposes the **Measuring** property and **Guardrail + channel height, H1** value below. [3] These types expose the **Guardrail height, H1** value below. [4] This type exposes the **Category** and **Diameter of largest pipe** values below. [5] This type exposes the **Category** and **Height of Cable Tray** values below.
Default Load Values	If this option is selected, load values for the selected type (with or without guardrail) are obtained from **Model Settings > Loading**, if unselected, load values can be entered directly.
Dead Load	Load to be created in the **Ancillary Dead** loadcase
Live (Imposed) Load	Load to be created in the **Ancillary Live (Imposed)** loadcase
Measuring (Live/Imposed)	For stairs choose whether the live/imposed load is a vertically projected load, or measured along the element.
Live (Imposed) Load loadcase	If working to Eurocodes, some ancillaries may require different Ψ and ϕ factors in which case a new loadcase should be manually added with the desired factors. To create the load in the new loadcase select it here.
Empty Load	Load to be created in the **Pipework Empty**, or **Cable Tray Empty**loadcase
Operating Content Load	Load to be created in the **Pipework Empty**, or **Cable Tray Empty**loadcase
Operating Content Load loadcase	If working to Eurocodes, some cable trays may require different Ψ and ϕ factors in which case a new operating

General	
	content loadcase should be manually added with the desired factors. To create the load in the new loadcase select it here.
Testing Content Load	Load to be created in the **Pipework Testing Content** loadcase
Category	For cable trays and pipework, the default load values from **Model Settings > Loading** are those for the selected category.
Guardrail	If this option is selected, the default load values from **Model Settings > Loading** are those for the selected type **with** guardrail.
Guardrail height, H1	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Guardrail + channel height, H1	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Height of cable tray, H	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Diameter of largest pipe	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Height, H	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. For more information, see:
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** **NOTE** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	

General	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Reverse	For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking this flag.

See also

Ancillaries (page 285)

Area ancillary properties

General	
Name	The automatically generated name for the ancillary.
User Name	Can be used to override the automatically generated name if required.
Rotation	Defines the span direction.
Type	The following types are available: • Walkways/catwalks[1] • Ladders with cage • Ladders without cage • Access platforms[1] • Operating platforms (storage laydown)[1] • Operating platforms (standard)[1] • Steel stair flight/landing[2] • Concrete stair flight/landing[3] • Timber stair flight/landing[3] • Lines of pipework[4] • Lines of cable tray[5] • Other [1] These types expose the **Guardrail** flag below. [2] This type exposes the **Measuring** property and **Guardrail + channel height, H1** value below.

General	
	[3] These types expose the **Guardrail height, H1** value below. [4] This type exposes the **Category** and **Diameter of largest pipe** values below. [5] This type exposes the **Category** and **Height of Cable Tray** values below.
Default Load Values	If this option is selected, load values for the selected type (with or without guardrail) are obtained from **Model Settings > Loading**, if unselected, load values can be entered directly.
Dead Load	Load to be created in the **Ancillary Dead** loadcase
Live (Imposed) Load	Load to be created in the **Ancillary Live (Imposed)** loadcase
Measuring (Live/Imposed)	For stairs choose whether the live/imposed load is a vertically projected load, or measured along the element.
Live (Imposed) Load loadcase	If working to Eurocodes, some ancillaries may require different Ψ and ϕ factors in which case a new loadcase should be manually added with the desired factors. To create the load in the new loadcase select it here.
Empty Load	Load to be created in the **Pipework Empty**, or **Cable Tray Empty**loadcase
Operating Content Load	Load to be created in the **Pipework Empty**, or **Cable Tray Empty**loadcase
Operating Content Load loadcase	If working to Eurocodes, some cable trays may require different Ψ and ϕ factors in which case a new operating content loadcase should be manually added with the desired factors. To create the load in the new loadcase select it here.
Testing Content Load	Load to be created in the **Pipework Testing Content** loadcase
Category	For cable trays and pipework, the default load values from **Model Settings > Loading** are those for the selected category.
Guardrail	If this option is selected, the default load values from **Model Settings > Loading** are those for the selected type **with** guardrail.
Guardrail height, H1	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Guardrail + channel height, H1	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.

General	
Height of cable tray, H	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Diameter of largest pipe	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Height, H	The value specified here is used in the calculation of wind loads on open structures seen by the ancillary.
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. For more information, see:
Shape factor, Cf	The default shape factor varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading** **NOTE** Default Cf factors are taken from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE.
Effective area XY	
Formula	The default effective area formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Factor	The factor used in the above formula can be edited if required.
Effective area XZ	
Formula	The default area per length used in this formula varies according to the entity type and is taken from **Model Settings > Loading > Wind Loading**
Area per length	The area per length used in the above formula can be edited if required.
Reverse	For lines of pipework/cable trays the height is displayed and it can be visually reversed by checking this flag.

See also

Ancillaries (page 285)

Equipment properties

Properties	
Name	The automatically generated name for the equipment.

<table>
<tr><th colspan="3">Properties</th></tr>
<tr><td>User Name</td><td colspan="2">Can be used to override the automatically generated name if required.</td></tr>
<tr><td>Type</td><td colspan="2">The following types are available:
• Sphere
• Vertical Cylinder
• Horizontal Cylinder
• Cuboid</td></tr>
<tr><td>Diameter, D</td><td>Diameter of sphere or cylinder.
Used to determine the C_f value.</td><td rowspan="7">These values are also used in the calculation of wind loads on open structures seen by the equipment.
For more information, see:</td></tr>
<tr><td>Length, L</td><td>Length of cuboid or cylinder.
Used to determine the C_f value.</td></tr>
<tr><td>Breadth, B</td><td>Breadth of cuboid.
Used to determine the C_f value.</td></tr>
<tr><td>Height, H</td><td>Height of cuboid.
Used to determine the C_f value.</td></tr>
<tr><td>Base position, z</td><td>Entering a non-zero value raises/lowers the equipment above/below the loading area.</td></tr>
<tr><td>Angle to Global X</td><td>Can be used to rotate the equipment relative to global X</td></tr>
<tr><td>CoG offset</td><td>Can be used to offset the equipment relative to the global or local axes</td></tr>
<tr><td colspan="3">Empty Load</td></tr>
<tr><td>Apply Load</td><td colspan="2">Load can either be applied at CoG or at Supports.
This load is created in the Equipment Empty loadcase.</td></tr>
<tr><td>Load</td><td colspan="2">Enter a single value if at CoG, or individual values at each support.</td></tr>
<tr><td colspan="3">Operating Content Load</td></tr>
<tr><td>Apply Load</td><td colspan="2">Load can either be applied at CoG or at Supports.
This load is created in the Equipment Operating Content loadcase.</td></tr>
<tr><td>Load</td><td colspan="2">Enter a single value if at CoG, or individual values at each support.</td></tr>
<tr><td colspan="3">Testing Content Load</td></tr>
<tr><td>Apply Load</td><td colspan="2">Load can either be applied at CoG or at Supports.
This load is created in the Equipment Testing Content loadcase.</td></tr>
</table>

Properties	
Load	Enter a single value if at CoG, or individual values at each support.
Wind loading	
Apply open structure wind load	Select this flag if you want open structure wind loads to be calculated. For more information, see:
Surface	For vertical and horizontal cylinders, specify the surface as either, moderately rough, rough, or very rough. For cylinders the surface and L/D ratio are used to determine the Cf value
End	For horizontal cylinders, specify the end as either flat, or rounded. This is used to determine the Cf value, see ASCE7 Fig 6-21

See also

Equipment (page 295)

Bearing wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	Bearing Wall **NOTE** Bearing walls must be rectangular in a vertical plane.
Material Type	Choice of: • Concrete • Timber • General
Assume extra floors supported	Enter the number of extra floors supported.

General	
AutomaticGenerate Support	• Cleared - a support will only be created if the **Generate support** property is checked. • Checked - a support will only be created if no members/ slabs capable of providing support exist under the wall.
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
Plane	Indicates the grid along which the wall is placed.

All panels	
Concrete type	While you can apply both normal and lightweight concrete, wall design using lightweight concrete is currently beyond scope.
Grade	The grades available here are set from the Materials button on the Home ribbon. **NOTE** The grade is only required to specify the correct density to be used in the wall self weight calculation.
Thickness	The thickness of the wall. **NOTE** The thickness is only required for the wall self weight calculation.
Alignment	Alignment of the wall: • Front • Back • Middle • User

All panels	
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.

Wall support	
Angles (Fx/Fy/Fz, Mx/My/Mz)	Used to specify the translational and rotational degrees of freedom in which the support acts: • Fixed - indicates the support is fixed in the specified direction. • Free - indicates the support is free to move, or has a stiffness applied in the specified direction.
Translational stiffnesses (x/y/z)	Used to specify the translational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear
Rotational stiffnesses (x/y/z)	Used to specify the rotational stiffness applied in a direction that is not fixed: • Release • Spring Linear • Spring Non-linear

Panel 1, 2, 3 etc.	
	In a multi-stack wall properties can be entered for a specific panel, over-riding those defined at the **All panels** level.

Count the floor as being supported	
Top level (Interediate levels) Base level (Head Code Eurocode, BS or IS)	If checked, the floor will be treated as supported when calculating the live/imposed load reductions.

Restrained	
	Used to indicate at which levels the wall is restrained. **NOTE** Only levels with a connecting member are listed.

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Shear only wall properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Top Level	Specifies the top level for the wall.
Base Level	Specifies the bottom level for the wall.
Wall Type	Shear Only Wall **NOTE** Shear only walls must be rectangular in a vertical plane.
Material Type	Choice of: • Concrete • Timber • General
AutomaticGenerate Support	• Cleared - a support will only be created if the **Generate support** property is checked. • Checked - a support will only be created if no members/slabs capable of providing support exist under the wall.
Generate support	• Cleared - no support is created under the wall. • Checked - a support is created under the wall. **NOTE** When a support is created, a line support is formed under a meshed wall, a point support under a mid-

General	
	pier wall, and a series of point supports under a bearing wall. **NOTE** When a support is created, its degrees of freedom are as specified in the 'Wall support' area of the wall properties.
Plane	Indicates the grid along which the wall is placed.

All panels	
Grade	The grades available here are set from the Materials button on the Home ribbon. **NOTE** The grade is only required to specify the correct density to be used in the wall self weight calculation.
Thickness	The thickness of the wall. **NOTE** The thickness is only required for the wall self weight calculation.
Alignment	Alignment of the wall: • Front • Back • Middle • User
Alignment offset	When the alignment is set to User it can be adjusted by specifying an exact offset.
Spring stiffness	The stiffness to be used in the analysis. See: How shear only walls are represented in solver models (page 584)

UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Wall Panel Properties

Use the **Wall Panel** properties to view or modify the properties of a wall panel.

The **Wall Panel** command is used to create a wall panel. Once created, the panel properties can then be viewed or modified in the Properties Window:

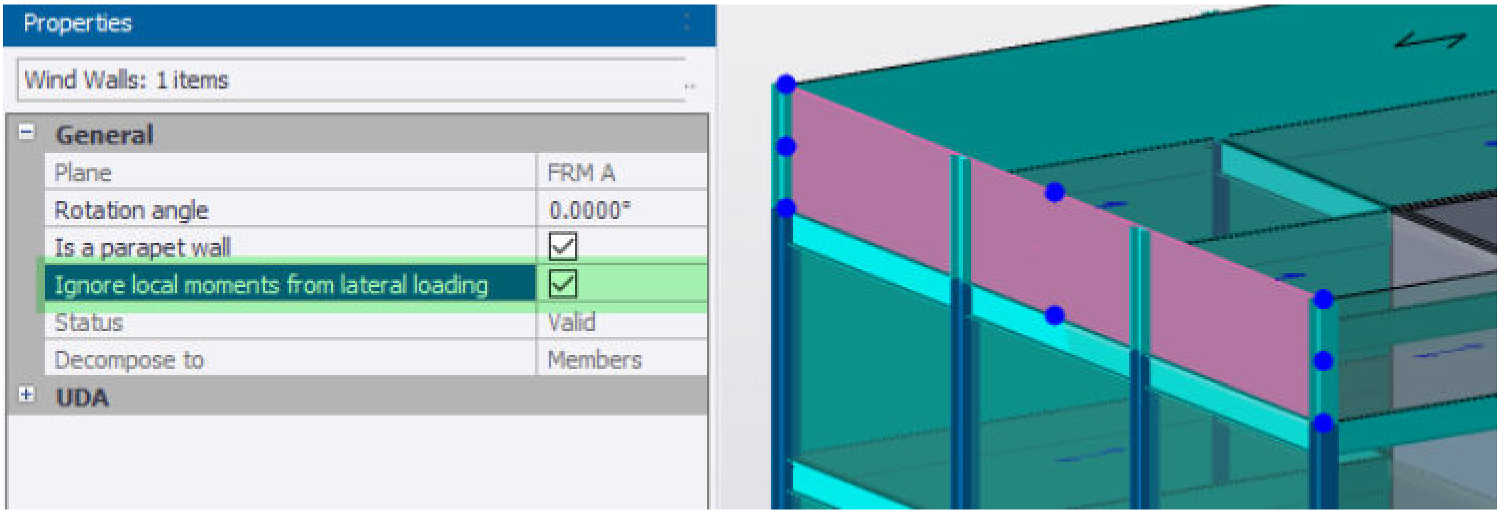
Property	Description
General	
Plane	Describes the plane in which the panel was placed.
Rotation angle	Describes the panel span direction as an angle, 0° is horizontal and 90° is vertical. See: and Modify slab/panel span direction (page 274)
Is a parapet wall	Check to indicate the panel is to be treated as a parapet in the wind analysis. See: Parapet wall panel load decomposition (page 1014)
Ignore local moments from lareral loading	This option is omly available when the “Is a parapet wall” setting is selected. When checked, only a lateral point load is applied to the top of a supporting column. See: Parapet wall panel load decomposition (page 1014)
Gap	Where the funneling gap to the adjacent building is not consistent due to the shapes of the buildings it is up to you to decide whether to specify the average or worst-case gap. The default gap is 1000 m which effectively give no funneling. A zero gap value explicitly means ignore funneling, for example where this building and the adjacent one are sheltered by upwind buildings.
Solidity	If you indicate that the wall panel is a parapet, then you also need to indicate the Solidity of the parapet. (Walls that are not parapets automatically adopt a solidarity of 1.0).
Status	Indicates whether the panel is valid or not.

Property	Description
Decompose to	Loads can be set to be decomposed to: • Members • Nodes (Default) • Rigid Diaphragms **NOTE** The decompose to nodes option only applies to those loads generated by the wind wizard. It does not apply to those loads applied directly to panels.
[+] UDA	A customizable list of the attributes that can be applied to individual members and panels.

Parapet wall panel load decomposition

For the special case of parapet wall panels spanning horizontally to parapet posts supported by columns, the panel load is decomposed as a lateral point load and point moment - which would be produced by a UDL on the post - to the top of the supporting column.

In some circumstances the engineer may wish only the lateral load but not a moment to be applied. This is catered for using the option “Ignore local moments from lateral loading” which is available in the wind wall properties only when the “Is a parapet wall” setting is checked on. When the “Ignore local moments from lateral loading” setting is checked on, only a lateral point load is applied to the top of a supporting column. No moment is applied.



NOTE

- The special decomposition of point loads - with or without a point moment - to supporting columns only occurs when all the following are true; the wind panel spans horizontally to parapet posts, the “Is a parapet wall” setting is checked on for the wall panel, the parapet post is supported by a column.
- In both the following cases a UDL will be decomposed to the members onto which the parapet wind panel directly spans;
  - The member supporting the parapet post is NOT a column.
  - The member onto which the parapet wind panel spans is not a parapet post (e.g. is an extended column or vertical beam).

Roof Panel Properties

Use the **Roof Panel** properties to view or modify the properties of a wall panel.

The **Roof Panel** command is used to create a roof panel. Once created, the panel properties can then be viewed or modified in the Properties Window:

Property	Description
General	
Plane	Describes the level at which the panel was placed.
Rotation angle	This value determines the following where appropriate: • Span direction for 1-way load decomposition

Property	Description
	• Orientation of semi-rigid 2D elements in the Solver Model See: and Modify slab/panel span direction (page 274)
Include in diaphragm	The options are: • Checked The roof panel is meshed to form a diaphragm of semi-rigid 2D elements. • Unchecked (Default) No diaphragm is defined on the roof panel.
Thickness	Roof panel thickness **NOTE** Only displayed if "Include in diaphragm" is checked
Youngs Modulus	Youngs Modulus **NOTE** Only displayed if "Include in diaphragm" is checked
Shear Modulus	Shear Modulus **NOTE** Only displayed if "Include in diaphragm" is checked
Temperature coefficient	Temperature coefficient. **NOTE** Only displayed if "Include in diaphragm" is checked
Divide Stiffness by	Used to adjust the roof panel stiffness. **NOTE** Only displayed if "Include in diaphragm" is checked
RoofType	The options are: • Default (Default) • Flat • Monopitch • Duopitch • Hip Gable • Hip Main • Mansard **NOTE** The default option assigns the roof panel as flat (if the slope < 5 deg) or Monopitch (if the slope is > 5 deg).

Property	Description
	You can more accurately specify the roof panel by choosing the appropriate option from the list. **NOTE** If the RoofType is changed after the wind model has already been established, you will need run  **Update Zones** to reinstate the zoning.
[+] UDA	Customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Support properties

Use the Support properties to view or modify the properties of a support.

When a support is first created, it's properties are taken as those displayed in the **Properties** window at that time.

You can either set the properties prior to placing the support, or select a support in the model afterwards to review or modify its properties.

Property	Description
General	
Name	Automatically created name based on the grid point location. **NOTE** This property is only displayed when editing an existing support
User Name	You can enter a user name to replace the automatically created name if required. **NOTE** This property is only displayed when editing an existing support
Plane	Describes the level at which the support was placed. **NOTE** This property is only displayed when editing an existing support

<table>
<tr><th>Property</th><th>Description</th></tr>
<tr><td>3 Grid Points</td><td>The options are:
Checked
A user defined coordinate system is applied to the support. (After clicking where you want to create the support, the second click defines the x direction and the third click defines the y direction.)
Unchecked (Default)
Support properties are defined in accordance with the global coordinate system.
NOTE This property is only displayed when creating a new support</td></tr>
<tr><td>Fx, Fy, Fz</td><td>The translational degrees of freedom can be set as either Free, or Fixed in each direction.</td></tr>
<tr><td>Mx, My, Mz</td><td>The rotational degrees of freedom can be set as either Free, or Fixed in each direction.</td></tr>
<tr><td>Angles</td><td></td></tr>
<tr><td>Inclination, Azimuth and Rotation</td><td>When creating new supports, the angles are calculated automatically depending on the placement method (3 Grid Points Checked/Unchecked).
When editing existing supports, the angles can be edited in order to redefine the direction in which the support acts.</td></tr>
<tr><td>Translational stiffness x, y, and z</td><td></td></tr>
<tr><td>Type</td><td>In order to define a translational spring in a particular direction, the translational degree of freedom in the same direction must first be set to Free. The available types are:
Release - (i.e. zero translational stiffness)
Spring Linear
Spring Non-linear</td></tr>
<tr><td>Stiffness</td><td>The options are:
Spring Linear
A single stiffness value is entered, which acts in both the positive and negative directions.</td></tr>
</table>

Property	Description
	• Spring Non-linear Two stiffness values are entered, one to act in the positive direction and a second to act in the negative direction.
Fmax -ve and Fmax +ve	For non-linear springs you are also able to define the spring capacity in each direction. (Note that this must always be entered as a positive value, for both +ve and -ve directions).
Rotational stiffness x, y, z	
Type	In order to define a rotational spring in a particular direction, the rotational degree of freedom in the same direction must first be set to Free. The available options are: • Release - (i.e. zero rotational stiffness) • Spring Linear • Spring Non-linear • Nominally Pinned • Nominally Free
Stiffness	The options are: • Spring Linear A single stiffness value is entered, which acts in both the positive and negative directions. • Spring Non-linear Two stiffness values are entered, one to act in the positive direction and a second to act in the negative direction.
Stiffness Percentage	The options are: • Nominally Pinned - 10% (i.e. 10% * 4 EI/L) • Nominally Fixed - 100% (i.e. 100% * 4 EI/L) **NOTE** In the above equations, L is the length from the base of the column to the level of the next column stack that is denoted as a floor, or it is the distance to the top of the column if shorter.

Property	Description
Fmax -ve and Fmax +ve	For non-linear springs you are also able to define the spring capacity in each direction. (Note that this must always be entered as a positive value, for both +ve and -ve directions).

See also

Create supports (page 314)

Analysis Element properties

General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Plane	Indicates the level or frame within which the member is placed.
Characteristic	Analysis Element
Active	Clearing this option makes the Analysis Element inactive in the analysis. See: Inactive members (page 306)
Element type	• Beam (page 319) • Truss (page 319) • Tension only (page 319) • Compression only (page 319) • Linear axial spring (page 320) • Linear torsional spring (page 320) • Non-linear axial spring (page 320) • Non-linear torsional spring (page 320) • Link (page 320) **NOTE** Tension only elements, Compression only elements, and non-linear axil/torsional springs are all non-linear elements and therefore require non-linear analysis. If linear analysis is performed they will be treated as linear elements.
Material type	• Steel • Concrete

General	
	• Timber • General • Cold formed • Cold rolled
Rotation	Rotation of the member about its local x axis.

Span 1	
Grade	The material grade
Section properties	For the **Beam** element type only, the area and inertia properties about each axis are required. For the **Truss**, **Tension only**, **Compression only**, and **Link** element types, the area Ax is required. For the spring Element types, spring properties are required.
Spring stiffnesses	For linear and non-linear axial and torsional springs, the spring properties are defined here.
Releases	
	For the **Beam** element type only, the six degrees of freedom; 3 translational (Fx, Fy, Fz) and 3 rotational (Mx, My, Mz); can be set at each end. See: Beam releases (page 949) All other analysis element types are released in all degrees of freedom at both ends apart from axial (Fx).
Load reductions	
KLL (Head Code ACI/AISC)	Specify the KLL factor in accordance with Table 4-2 in ASCE7-05/ASCE7-10. See:
Reduce imposed loads by (All other Head Codes)	Although the percentage of imposed load reduction is not determined automatically, this property allows you to specify the percentage manually. • reducible loadcases are reduced • combinations incorporating reducible loadcases are reduced See:

UDA	
Name Finish Class	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

UDA	
Phase Note File	

Base plate properties

Property	Description
General	
Name	The automatically generated name.
User Name	Can be used to override the automatically generated name if required.
Plane	Indicates the level or frame within which the base plate is placed.
Base Plate	
Autosize	Indicates whether base plate should be autosized when column section changes
Length	Length of base plate. User defined only when **autosize** is off.
Width	Width of base plate. User defined only when **autosize** is off.
Thickness	Thickness of base plate. User defined only when **autosize** is off.
Grade	Grade of base plate.
Prepared for direct contact in bearing (Eurocode only)	Indicates whether the base plate should be considered to be prepared for direct contact in bearing.
Rod (AISC only)	
Rod Grade	Rod grade.
Rod Size	Rod size.
Embedded depth	Rod embedded depth.
Friction Coefficient	Coefficient of friction between the base plate and the grout layer
Horizontal Shear Transfer	• Friction Alone • Shear on Bolts Alone • Friction and Shear on Bolts • Bearing on Shear Lug

Property	Description
	• Friction and Bearing on Shear Lug
Rod Layout Type	• Symmetric 4 rods • Symmetric
Edge Distance	Distance of bolt from edge of plate
End Distance	Distance of rod from end of plate
Bolts (Eurocode only)	
Bolt Grade	Bolt grade.
Bolt Size	Bolt size.
Bolt Length	Bolt length.
Projection above base plate	Length the bolt projects above the base plate.
Friction Coefficient	Coefficient of friction between the base plate and the grout layer
Horizontal Shear Transfer	• Friction Alone • Shear on Bolts Alone • Friction and Shear on Bolts
Bolt Layout Type	• Symmetric 4 Bolts • Symmetric • Asymmetric
Edge Distance	Distance of bolt from edge of plate
End Distance	Distance of bolt from end of plate
Anchor Plate (Eurocode only)	
Type	• Individual • Combined
Width	Anchor plate width.
Thickness	Anchor plate thickness.
Length	Anchor plate length.
Edge Distance	Anchor plate edge distance, applicable when **Type** is Combined.
Welds	
Type	• Fillet • Butt
Leg Length	Leg length of Fillet weld.
Full Profile	Indicates whether weld is full or partial.

Property	Description
Partial Profile	Length of partial weld, applicable when **Full Profile**is turned off and is only applicable to weld associated with the web.
Concrete base	
Length	Length of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Width	Width of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Depth	Depth of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Concrete	Material grade of concrete foundation associated with base plate. If no concrete base exists this value is user defined.
Grout Space	Grout space between base plate and concrete base.
Geometric enhancement parameter, α (Eurocode only)	Geometric enhancement parameter (EN 1992-1-1 Cl. 6.7) automatically calculated from base plate and concrete base dimensions.
UDA	
Name Finish Class Phase Note File	A customizable list of the attributes that can be applied to individual members and panels. See: Create and manage user-defined attributes (page 916)

Patch properties

General	
Name	The automatically generated name for the patch.
User Name	Can be used to override the automatically generated name if required.
Type	• Column • Beam • Wall
Lx	Specifies the size of the patch in the X direction.
Ly	Specifies the size of the patch in the Y direction.

<table>
<tr><th colspan="2">General</th></tr>
<tr><td>Associated Slab Panel</td><td>Where a patch sits over more than one panel then an automatic choice is made, but you are able to override this if required by selecting one of the alternative panels from a drop-down list.</td></tr>
<tr><td>Align to Panel Reinforcement</td><td>When this check box is selected, the calculated strip reinforcement is aligned with the background reinforcement in the Associated Slab Panel.</td></tr>
<tr><td>Local X Angle</td><td>The angle of the X axis reinforcement is only editable ifAlign to Panel Reinforcement is cleared.</td></tr>
<tr><td>Surface</td><td>Specifies the reinforcement to be associated with and designed by the patch:
Top or Bottom (cannot be both).</td></tr>
<tr><td>Autodesign</td><td>This setting applies to all strips in both directions, but reinforcement in either direction can still be set to none - see below.
Cleared - the specified reinforcement will be checked during the design process.
Checked - reinforcement will be designed during the design process.</td></tr>
<tr><td>Select bars starting from</td><td>This option controls the starting point for auto-design procedures.
Minima (default) - removes the current arrangement and begins with the minimum allowed bar size.
Current - the auto design commences from the current bar arrangement.</td></tr>
<tr><td>Consider Strips</td><td>This setting controls which strips are to be designed by the patch.
X
Y
X and Y</td></tr>
<tr><td>Consider patch surface moments only</td><td>Off (default)
patch reinforcement is designed in the surface of the slab at which the patch has been specified AND the slab reinforcement is also checked in the opposite surface.
On
patch reinforcement is designed in the surface of the slab at which the patch has been specified, but the slab reinforcement in the opposite surface is NOT checked.</td></tr>
</table>

General	
	NOTE The only situation when you might need to activate this option is if the check of slab reinforcement in the opposite surface fails but another patch exists in the same location at that face.(When the other patch is designed additional reinforcement will be provided at that surface, so this check is not required for the original patch).

Strips in X, Strips in Y	
Center, Left and Right Strip	
Width	The left and right strip widths can be specified independently: the center strip width is recalculated accordingly and cannot be edited. (By default the center strip covers half the panel, so that the left and right strips each cover a quarter of the panel.)
Design Force	• Average (of all the FE nodal values within the strip). • Maximum (of all the FE nodal values within the strip).
Reinforcement	
Combine with Panel Reinforcement	When this check box is selected, the calculated strip reinforcement takes into account any existing panel reinforcement in the Associated Slab Panel that is in the same alignment as the strip.
Cover as Panel	When this check box is selected, the cover is set to be the same as that in the Associated Slab Panel.
Outer Bar Direction as Panel	When this check box is selected, the outer bar direction is set to be the same as that in the Associated Slab Panel. When cleared, the outer bar direction can be set in X or Y.

Reinforcement	
Combine with Panel Reinforcement	When this check box is selected, the calculated strip reinforcement takes into account any existing panel reinforcement in the Associated Slab Panel that is in the same alignment as the strip.
Cover as Panel	When this check box is selected, the cover is set to be the same as that in the Associated Slab Panel.
Outer Bar Direction as Panel	When this check box is selected, the outer bar direction is set to be the same as that in the Associated Slab Panel. When cleared, the outer bar direction can be set in X or Y.

Reinforcement	
Reinforcement	This setting is used to specify whether bars or mesh are to be used in each direction. • Mesh • Bars XY • Bars X • Bars Y • None (If Mesh is selected an extra setting then allows you to specify if main bars are in X or Y.)

Reinforcement in X, Reinforcement in Y, or Mesh	
Bar Size, spacing, Mesh type etc.	The actual reinforcement provided in each of the strips is indicated here.

Punching check properties

General	
Tension Reinforcement	This setting identifies the slab reinforcement to be used in the punching check calculation. • Top • Bottom
Center	The check location (not editable).
Column Drop	Indicates if the check considers a Column Drop (not editable).
Beta - User limit (Head Code Eurocode)	When this check box is selected, a minimum value of Beta = 1.15 is applied to all internal columns.
User factor for Vt (Head Code BS)	When this check box is selected, the user factor for Vt is applied.
u0 - user reduction	Can be used to manually specify a reduction in the length of the u0 perimeter to account for undefined openings.
u1 - user reduction	Can be used to manually specify a reduction in the length of the u1 perimeter to account for undefined openings.
Check Status and Ratio	Indicates the status of the checks for each calculated perimeter and the overall check ratio.

Loaded Perimeter	
Length	Indicates the length of the u0 Loaded Perimeter (not editable).
Reduced length	The reduced length of u0 after accounting for openings.
BEquiv, DEquiv, BBound, BBound, Bounding Perimeter	Refer to the Concrete Design Reference Guide for the current Head Code for the appropriate definition of these terms.
d Effective Depth	Indicates the average effective depth to the tension reinforcement (not editable). d = (dy + dz) / 2 where dy and dz are the effective depths in the two orthogonal directions. There is a value of d for top steel and a different value for bottom steel. Note this definition changes in the presence of a drop panel. This information is only available if the reinforcement is known in each direction.
Slab Override	When an override is applied the slabs in each direction can be de-activated in the check. In this way the Loaded Perimeter Position can be edited. **NOTE** In the typical case of punching checks around a column, the slab ‘y’ & ‘z’, ‘positive’ & ‘negative’ are defined by the local axis system of the column. This can be displayed by displaying the Local Axes for 1D Elements in Scene Content.

Control Perimeter	
Length	Indicates the length of the u1 Control Perimeter (not editable).
Reduced length	The reduced length of u1 after accounting for openings.
[-] Design Input (Head Code Eurocode and ACI)	
Use Reinforcement	Select this check box in order to apply a default punching reinforcement arrangement that can then either be

Control Perimeter	
	checked, or used as the starting reinforcement for an auto-design. You would only choose to clear the check box when specifying a new check if you want to perform an auto-design but starting from Minima.
Reinforcement type	In the current release only Stud reinforcement is considered.
Arrangement type	The options are: • Orthogonal (default) • Circular
Auto-design	When run in Auto-design mode, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded.
Select reinforcement starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Auto-design is ‘on’. • Minima - removes the current arrangement and begins with the minimum allowed bar size. • Current - auto-design commences from the current bar arrangement.
Rib type	Specifies the reinforcement rib type.
Grade	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size	The reinforcement bar sizes that are available here are set from the Materials button on the Home ribbon.
Spacing	Defines the spacing between bars along each rail.
Spacing from column face	Defines the spacing of the first bar in each rail from the column face.
Stud rails spacing in Y direction	Spacing between rails in the local Y direction.
Stud rails spacing in Z direction	Spacing between rails in the local Z direction.
Number of diagonal stud rails on one corner	The number of stud rails adjacent to each corner of the column. (This property is only displayed when the ‘Arrangement Type’ is Circular)
Number of studs per	The number of stud rails adjacent to the column face in the local Y direction.

Control Perimeter	
column face - Y direction	
Number of studs per column face - Z direction	The number of stud rails adjacent to the column face in the local Z direction.
Number of studs per rail	Number of studs on each rail.

Utilization ratio	
Apply (to autodesign)	On • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than the ratio limit. Off • When an Autodesign is performed, the design will be incremented to achieve a utilization ratio less than 1.0. See: Apply user defined utilization ratios (page 613)
Apply (to check)	On • When a Check is performed, the check will pass provided the utilization ratio is less than the ratio limit. Off • When a Check is performed, the check will pass provided the utilization ratio is less than 1.0.
Ratio limit	The utilization ratio against which the autodesign or check is performed (when applied above).

Design Input (Eurocode and ACI only)	
Use Reinforcement	Select this check box in order to apply a default punching reinforcement arrangement that can then either be checked, or used as the starting reinforcement for an auto-design. You would only choose to clear the check box when specifying a new check if you want to perform an auto-design but starting from Minima.
Reinforcement type	In the current release only Stud reinforcement is considered.
Arrangement type	The options are: • Orthogonal (default) • Circular

Design Input (Eurocode and ACI only)	
Auto-design	When run in Auto-design mode, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded.
Select reinforcement starting from	This option controls the starting point for auto-design procedures and is therefore only displayed if Auto-design is 'on'. • Minima - removes the current arrangement and begins with the minimum allowed bar size. • Current - auto-design commences from the current bar arrangement.
Rib type	Specifies the reinforcement rib type.
Grade	The reinforcement grades that are available here are set from the Materials button on the Home ribbon.
Bar size	The reinforcement bar sizes that are available here are set from the Materials button on the Home ribbon.
Spacing	Defines the spacing between bars along each rail.
Spacing from column face	Defines the spacing of the first bar in each rail from the column face.
Stud rails spacing in Y direction	Spacing between rails in the local Y direction.
Stud rails spacing in Z direction	Spacing between rails in the local Z direction.
Number of diagonal stud rails on one corner	The number of stud rails adjacent to each corner of the column. (This property is only displayed when the 'Arrangement Type' is Circular)
Number of studs per column face - Y direction	The number of stud rails adjacent to the column face in the local Y direction.
Number of studs per column face - Z direction	The number of stud rails adjacent to the column face in the local Z direction.
Number of studs per rail	Number of studs on each rail.

Result strip properties

General	
Name	The automatically generated name for the strip.
User Name	Can be used to override the automatically generated name if required.
Start Width	The total strip width at the first point picked when creating the strip.
End Width	The total strip width at the second point picked when creating the strip.
Result Type	Determines how the strip result is calculated: • Average • Centreline • Maximum
Number of Stations	The number of stations per metre along the strip.
Number of Points	The number of points per metre across the strip at each station.

13.2 Settings

Settings for the current project

Several dialogs are provided for controlling the various settings and options in the current project:

- Model Settings dialog (page 1207)
- Analysis Settings dialog (page 1173)
- Design Settings dialog (page 1179)
- Slab Deflection Settings dialog (page 1225)
- Drawing Settings dialog (page 1180)

Settings for future projects

A single dialog is provided for defining all settings and options that are to be applied to new projects:

- Settings dialog (page 1209)

Model Settings

Many (but not all) of the settings for the current project are accessed from the Model Settings dialog (page 1207) on the **Home** tab.

- Design code settings (page 1033)
- Unit settings (page 1034)
- Object reference settings (page 1035)
- Loading settings (page 1037)
- Grouping model settings (page 1038)
- Material list settings (page 1039)
- Beam lines settings (page 1039)
- Analysis Model settings (page 1040)
- Validation settings (page 1042)
- Live/imposed load reduction settings (page 1043)
- Global Imperfections settings (page 1044)
- User-defined attribute settings (page 1045)
- Graphics view settings (page 1047)
- Structural BIM settings (page 1047)

See also

Analysis Settings (page 1050)

Design Settings (page 1064)

Slab deflection settings (page 1122)

Drawing settings (page 1126)

Design code settings

The **Design Codes** page allows you to specify the head code and the subsequent design codes that are applied:

- To the **current** project - when accessed from the Model Settings (page 1032)
- To **new** projects - when accessed from the Settings dialog (page 1209)

Item	Description
Select the head code list	Allows you to select the head code that you want to apply. When you select a head code in the list, the

Item	Description
	Design Codes table automatically updates. **WARNING** If you change the head code in an existing project, the following will occur in the model: • Some materials, steel sections, studs, decks and reinforcement may require re-selecting in the model to make them consistent with the new head code/unit system. • Wind loading (if any) and wall/roof panel properties will be deleted. The wind wizard will need rerunning, wall/roof panel properties need resetting and the wind loadcases will need recreating. • Seismic loading (if any) will be deleted. The seismic wizard will need re-running. • All combinations will be deleted.
Design Codes table	Displays the available action codes and resistance codes, which are dependent on the selected head code. The lists within the **Design Codes** table can be used to select between available alternatives.

See also

Define and modify head codes and design codes (page 878)

Unit settings

The **Units** page allows you to specify the units, format and precision that are applied

- To the **current** project - when accessed from the Model Settings (page 1032)
- To **new** projects - when accessed from the Settings dialog (page 1209)

Item	Description
System list	Allows you to determine whether metric or US customary units are used.
Table of quantities	Displays each quantity, its current unit and its precision. You can modify quantities by selecting them in the table.
Settings list	Displays and allows you to select the units for the currently selected quantity.
Precision list	Displays and allows you to define the precision of the currently selected quantity.
Use for values lower than	Allow you to define the values for which Tekla Structural Designer applies exponential formats.
or greater than or equal to	

See also

Define and modify units (page 879)

Object reference settings

The **References** page and its subpages allow you to adjust different object reference settings:

- To the **current** project - when accessed from the Model Settings (page 1032)
- To **new** projects - when accessed from the Settings dialog (page 1209)

Item	Description
General subpage	

Item	Description
Initial value in levels	Allows you to specify the start number (such as 1, 100, 1000) at each construction level for object references that contain the **Count** item.
Renumbering Direction	Allows you to specify renumbering directions that control how the member numbering is applied when you use the **Renumber** command.
Ignore letters I & O	Allows you to ignore the letters I and O in grid line names.
Initial number	Allows you to specify the initial number applied to the first grid lines.
Initial letter	Allows you to specify the initial letter applied to the first grid lines.
Naming style list	Allows you to select a desired naming style for grid lines.
Groups subpage **NOTE** This subpage is only displayed in the **Settings** dialog box not in the **Model Settings** dialog box.	
Sub-group Name	Allows you to specify the text used to designate the group labelling. This text forms the stem of the Design Group and Detailing Group names that are displayed on the Groups tab of the Project Workspace. These names are shown in the output reports and drawings when grouped design has been applied.
Formats subpage	
Format table	Lists each object type showing its current reference format and allows you to customize the formats. **NOTE** To customize a object reference format, click the click the **...** button in the **Edit** column
Texts subpage	
Characteristics table	Allows you to specify the text used to designate the characteristic for object

Item	Description
	references that contain the **Characteristic** item.
Materials table	Allows you to specify the text used to designate the material for object references that contain the **Material** item.

See also

Manage object references (page 881)

Loading settings

The **Loading** page and its subpages allow you to adjust certain loading settings:

- In the **current** project - when accessed from the Model Settings (page 1032)
- In **new** projects - when accessed from the Settings dialog (page 1209)

General

Item	Description
Use load patterning for steel beams	Allows you to select whether Tekla Structural Designer applies load patterns for steel beams.
Use patterning of eccentricity moments for steel columns	Allows you to select whether Tekla Structural Designer considers patterned eccentricity moments in the design of steel columns.
Pipework Operating & Testing Content Loadcase Type	Allows you to select whether the ancillary loadcases created for lines of pipework are considered as Dead or Live (Imposed).
Equipment Operating & Testing Content Loadcase Type	Allows you to select whether the equipment loadcases created for operating and testing are considered as Dead or Live (Imposed).

Line/Area Ancillary Loading

Item	Description
Ancillary Type	Ancillary Type.
Dead/Empty Load	The default Dead/Empty Load factors applied for each type.
Live (Imposed)/Content Load	The default Live (Imposed)/Content Load factors applied for each type.
Testing Content Load	The default Testing Content Load factors applied for each type.

Wind Loading

Item	Description
Entity Type	The loadcase names that will be created when specific ancillary types have been specified.
Shape factor C_f	The default C_f factors come from the document 'Wind Loads For Petrochemical And Other Industrial Facilities' published by ASCE
Effective Area$_{xy\ ,\ xz}$	The effective area of entity in xy and xz planes respectively.
Factor$_{xy\ ,\ xz}$	Factor applied to the effective area of entity in xy and xz planes respectively.
Area Per Length$_{xz}$	The area per length.

Grouping model settings

The **Grouping** page in the **Model Settings** dialog box allows you to control the tolerance applied when members are grouped in the current project.

Item	Description
Maximum edge length variation	Allows for a tolerance to be applied to the automatic grouping. A member can only be included in an existing group if its span length is within the specified tolerance of the group's (average) span length.
Maximum length variation	Allows for a length tolerance to be applied to the automatic grouping of trusses. A truss can only be included in an existing group if its span length is within the specified tolerance of the group's (average) span length.

Item	Description
Maximum height variation	Allows for a height tolerance to be applied to the automatic grouping of trusses. A truss can only be included in an existing group if its height is within the specified tolerance of the group's (average) height.

Material list settings

The **Material List** page allows you to specify settings affecting material lists:

- To the **current** project - when accessed from the Model Settings (page 1032)
- To **new** projects - when accessed from the Settings dialog (page 1209)

Item	Description
Ignore openings with area less than	Allows you to specify the size of opening that can be considered small enough to be ignored when determining the quantity of necessary slab reinforcement.

See also

Apply attribute filters to material lists and reports (page 920)

Beam lines settings

The **Beam Lines** page allows you to control the parameters used for continuous concrete beam formation:

- In the **current** project - when accessed from the Model Settings (page 1032)
- In **new** projects - when accessed from the Settings dialog (page 1209)

NOTE The parameters are only used to control the automatic concrete beam joining that occurs during the design process or when the **Beam Lines** command is run. The parameters are not considered when members are joined manually using the **Join** command.

Item	Description
Join pinned beam end	Allows you to control whether joining should occur or not if a pin is defined at the end the last span of the first beam or the start of the first span of the second beam, the fixity at the end in question changing from pinned to continuous once joined. If the beam is subsequently re-split at the same location, the pin gets reinstated.
Limiting join angle in plan	Specifies the limiting angle in plan to be applied for joining beams. Only beams meeting in plan at an angle less than the specified value can be joined. Where two beams start at the end of the first, the one that has the minimum angle is the one that gets connected.
Limiting join angle in elevation	Specifies the limiting angle in elevation to be applied for joining beams. Only beams meeting in elevation at an angle less than the specified value can be joined. Where two beams start at the end of the first, the one that has the minimum angle is the one that gets connected.
Minimum section overlap	Allows you to apply a tolerance when joining beams if they do not fully overlap in section. You can use it to prevent joining if there is very little physical overlap between the beam cross sections.

See also

Automatically join all concrete beams (page 331)

Analysis Model settings

On the **Analysis Model** page you can adjust certain rigid zone, curved beam, and concrete column centerline settings:

- In the **current** project - when accessed from the Model Settings (page 1032)
- In **new** projects - when accessed from the Settings dialog (page 1209)

Rigid Zones

Design codes allow engineers to assume parts of concrete beams or columns are rigid, leading to more efficient designs.

NOTE Rigid zones should not be confused with rigid offsets, which are used to ensure that the analysis model is properly connected. You can have rigid offsets in the model even if rigid zones are turned off.

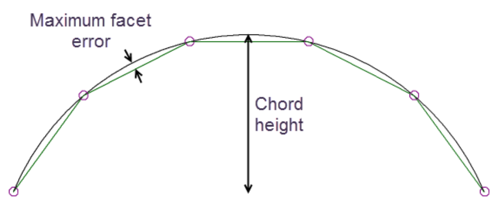
Item	Description
Percentage of rigidity	Allows you to specify the extent of the rigid zone created. The option has an effect only if rigid zones are applied (see below).
Rigid zones not applied	Allows you to switch rigid zones on or off. The option affects where releases are applied in the analysis model and where members start and end for design. When rigid zones are not applied, the design model corresponds to the analysis model, whereas when they are applied, the design model is defined between the ends of the rigid zones.

NOTE There is a significant difference between **Rigid zones not applied** and rigid zones applied with 0% rigidity. The total elastic length of a member will be the same in the two models, but the position of releases and start or end of design members will be different.

Curved Beams

In the solver model, curved beams are replaced by a series of straight line solver elements. The number of solver elements used being controlled by the **Maximum facet error** that has been specified.

Item	Description
Maximum facet error	For any given chord height, reducing this value increases the number of

Item	Description
	solver elements used to represent the curve. 

Column Centerlines

For concrete columns connected to a two-way slab, the column perimeter shape is cut out of the slab 2D element mesh. Stiff analysis 'rigid link' elements then connect the column centerline (where the column 1D elements are located) to the 2D mesh nodes. In some circumstances - for example for L-section columns of certain proportions - very short link elements could result, which could cause validation errors or warnings and analysis solution problems.

The following settings are provided in order to move the column centerline a small distance where needed to avoid these problems.

Item	Description
Automatically move column centerlines when the smallest rigid link is less than the error limit for 1D element length	This option is initially on by default. The error limit used in this check is specified in Validation settings (page 1042)
Automatically move column centerlines when the smallest rigid link is less than the warning limit for 1D element length	This option is initially off by default. The warning limit used in this check is specified in Validation settings (page 1042)

Validation settings

The **Validation** page of the **Model Settings** dialog box allows you to control the checks that are applied when the current model is validated.

Item	Description
Error limit for length	Allows you to control when an error is displayed when very short analysis elements are detected.
Warning limit for length	Allows you to control when a warning is displayed when very short analysis elements are detected.

Item	Description
Error limit for quality	Allows you to control when an error is displayed when poor quality 2D elements are detected. **NOTE** 2D element quality depends on two things: skew and aspect ratio. 0% is bad quality: a squashed triangle tends towards bad quality, whereas 100% is perfect quality, so an equilateral triangle is perfect quality.
Warning limit for quality	Allows you to control when a warning is displayed when poor quality 2D elements are detected.
Check for validation warnings	Allows you to specify which model, analysis, and design validation checks are performed. If a box is cleared, the validation check is not performed.

Live/imposed load reduction settings

The **Live load reduction** (US headcode) or **Imposed load reduction** (other head codes) page in the **Model Settings** dialog box allows you to control the live/imposed load reductions.

US headcode

Item	Description
Limit on live load reductions	Specify the limit on reductions applied to beams. Specify the limit on reductions applied to column stacks and wall panels for the number of floors carried. These apply to all loadcases of type **Live Load**
Limit on live load reductions	Specify the limit on reductions applied to beams columns and walls. These apply to all loadcases of type **Roof Live Load**

Other headcodes

Item	Description
Number of floors carried	The number of floors carried by the column or wall.
Reduction percentage	The reduction that is applied to the column stack or wall panel for the number of floors carried.

See also

Activate reductions in live or imposed loadcases (page 341)

Global Imperfections settings

This page in the **Model Settings** dialog box is used to control the percentages of load used for the global imperfection calculations.

The percentages are material independent, which means that the same percentages are used for concrete and steel. The percentages vary depending on the height of the structure and the number of columns in each direction.

EHF Global Imperfections (Eurocode)

Item	Description
Height of the structure	Allows you to specify the effective height of the structure to be used in the EHF calculations.
Set Default	Sets the height of the structure to the highest construction level.
Number of columns in X direction **Number of columns in Y direction**	In the Eurocode Cl 5.2(5) the calculation of the reduction factor a_m depends on the number of contributing members, m. Valid input for m in the X and Y directions is any whole number from 1 to 1000. The default value is 1 which results in a_m = 1.000. If a value of 1000 is entered then a_m would reduce to 0.707.
Global initial sway imperfections	Displays the calculated alpha values and phi percentages for the above input. The override checkboxes can be used to enter your own percentages in

Item	Description
	place of the calculated values if required.

NL Global Imperfections (US head code)

Item	Description
Global initial sway imperfections Direction Dir 1, Dir 2	Displays the calculated percentage imperfections to be applied in each direction. The override checkboxes can be used to enter your own percentages in place of the calculated values if required.

NHF Global Imperfections (BS, IS & AS head codes)

Item	Description
Global initial sway imperfections Direction Dir 1, Dir 2	Displays the calculated percentage imperfections to be applied in each direction. The override checkboxes can be used to enter your own percentages in place of the calculated values if required.

See also

Display notional forces and seismic equivalent lateral forces (page 504)

User-defined attribute settings

The **User Defined Attributes** page allows you to control the attributes that are available:

- In the **current** project - when accessed from the Model Settings (page 1032)
- In **new** projects - when accessed from the Settings dialog (page 1209)

You can add new attribute definitions, delete them, set a type for each attribute, and adjust the acceptable values.

Item	Description
Name	Displays and allows you to modify the name of the attribute that will be displayed in the **UDA** section of the **Properties** window.
Type list	Allows you to specify a type for the option. The options are text, number, and file.
Source list	Allows you to restrict the allowable input of the attribute, if necessary. The two options are: • **Custom Value**: Does not restrict the allowable input. • **Value List**: Restricts the allowable input for an attribute to a pre-set list.
Values	Allows you to specify the allowable attribute values when the source is set to **Value List**.
Linked	This setting has no effect if all levels in the model are unique. If the model contains existing duplicate levels it works as follows: • for UDAs that are linked, a single UDA value is assigned to the entity on the source and the duplicate level. • for UDAs that are not linked, a different UDA value can be assigned to an entity on a duplicate level. **NOTE** In the Construction Levels dialog, when a level is first set to be duplicate of a source level: • initially UDA values from the source level are copied to the duplicate regardless of the **Linked** setting • only from that point on does the linked/not linked functionality apply.

Item	Description
	• i.e. when set to not linked, the UDA at each level becomes independently editable.
Add	Creates a blank row that allows you to define a new attribute.
Delete	Deletes the currently selected attribute from the table.
Move Up	Move the currently selected attributes up or down in the table and in the **Properties** window.
Move Down	

See also

Create and manage user-defined attributes (page 916)

Graphics view settings

The **Graphics View Settings** page in the **Model Settings** dialog box allows you to control the display of miscellaneous items in 2D and 3D views in the current project.

Item	Description
Do not display values of story shear below	Allows you to limit the values of story shear for a new model. This way, you can easily ignore the small values of story shear that might otherwise detract you from the more important story shear values. The story shear values that are less than the limiting value are not displayed in the results view.
Show full pile length	Allows you to select whether Tekla Structural Designer displays the full length of piles or a shorter pile length that you can specify.
2x scale for steel columns	Allows you to double the scale of steel columns in 2D views in order to simplify viewing the columns and their orientation.

Structural BIM settings

The **Structural BIM** page and its subpages allow you to control the structural BIM import and export processes:

- In the **current** project - when accessed from the Model Settings (page 1032)
- In **new** projects - when accessed from the Settings dialog (page 1209)

Button, command, or option	Description
Export --> **Continuous Objects** subpage	
Separate objects for each stack	Allows you to select which columns are exported as separate objects. **NOTE** Tekla Structural Designer organizes rebar information by stacks and spans. If you do not export concrete columns as separate objects, no rebar information is exported.
Separate objects for each span	Allows you to select which beams or other members are exported as separate objects. **NOTE** Tekla Structural Designer organizes rebar information by stacks and spans. If you do not export concrete beams as separate objects, no rebar information is exported.
Separate objects for each panel	Allows you to select which wall types are exported as separate objects. **NOTE** Tekla Structural Designer organizes rebar information by stacks and spans. If you do not export shear walls as separate objects, no rebar information is exported.
Export --> **Reinforcement Information** subpage	
Concrete columns	Allows you to select whether bar marks are included in the exported

Button, command, or option	Description
	concrete column reinforcement information.
Concrete beams	Allows you to select whether bar marks, link leg count, and region length percentage are included in the exported concrete beam reinforcement information.
Export --> **Mappings** subpage	
Materials --> **Reset mappings for Structural BIM export**	Resets the mappings for Structural BIM export of materials to the default settings.
Decking --> **Reset mappings for Structural BIM export**	Resets the mappings for Structural BIM export of decking to the default settings.
Import --> **Settings** subpage	
Concrete beams --> **Default "Allow automatic join"**	Allows you to control whether new imported concrete beams are automatically joined or not at the end of the import. **NOTE** If the BIM model contains continuous beams as one piece, you might want to clear the option. After the import, you can change the properties back, if necessary.
Steel columns --> **Ignore splice offsets in physical member positions**	Allows you to select whether splice offsets are ignored when importing physical member positions.
Import --> **Mappings** subpage	
Materials --> **Reset mappings for 'TEL' File import**	Resets the mappings for TEL file import of materials to the default settings.
Materials --> **Reset mappings for Structural BIM import**	Resets the mappings for Structural BIM import of materials to the default settings.
Decking --> **Reset mappings for Structural BIM import**	Resets the mappings for Structural BIM import of decking to the default settings.

See also

Import a project from a Structural BIM Import file (page 109)

Analysis Settings

The **Analysis Settings** page and its subpages allow you to adjust the options applied to the different analyses:

- In the **current** project - when accessed from the Analysis Settings dialog (page 1173) on the **Analyze** tab.
- In **new** projects - when accessed from the Settings dialog (page 1209) on the **Home** tab.

1st order non-linear settings

Button, command, or option	Description
Maximum number of iterations	The number of iterations to perform. Default = 100.
Tolerance	At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.0001.
Relative	Default: ON.
Relaxation Factors	Relaxation factors control an amount of nominal compression tension-only braces can undergo while remaining active during analysis iteration. This improves convergence for rare problematic models/ loading situations in which otherwise most or all braces may experience compression and become inactive, causing instability and preventing solution. When converged tension-only braces will still have only either zero or tension force. The process is entirely automatic by default and it is anticipated the factors will not require manual editing for most circumstances. • **Use relaxation factors for tension only elements**: Default = cleared • **Minimum relaxation factor**: Default = 0.1

Button, command, or option	Description
	• **Maximum relaxation factor**: Default = 0.5 If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the above minimum and maximum values) may produce convergence. **NOTE** Relaxation factors are not activated by default, as it is anticipated they will not be required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree.

2nd order non-linear settings

Button, command, or option	Description
Maximum number of iterations	The number of iterations to perform. Default = 100.
Tolerance	At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.0001.
Relative	Default: ON.
Relaxation Factors	Relaxation factors control an amount of nominal compression tension-only braces can undergo while remaining active during analysis iteration. This improves convergence for rare problematic models/ loading situations in which otherwise most or all braces may experience compression and become inactive, causing instability and preventing solution. When converged tension-only braces will still have only either zero or tension force. The process is

Button, command, or option	Description
	entirely automatic by default and it is anticipated the factors will not require manual editing for most circumstances. • **Use relaxation factors for tension only elements**: Default = cleared • **Minimum relaxation factor**: Default = 0.1 • **Maximum relaxation factor**: Default = 0.5 If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the above minimum and maximum values) may produce convergence. **NOTE** Relaxation factors are not activated by default, as it is anticipated they will not be required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree.

1st order modal settings

Button, command, or option	Description
Extraction Method	Allows you to select the appropriate extraction method for your model. The options are: • **Automatic** (default option): Initially uses the **Subspace** option to find the lowest modes. If the criteria (either mass or number of modes) is not fulfilled, **FEAST** is then automatically used to find higher modes until the stopping criteria is fulfilled.

Button, command, or option	Description
	• **Jacobi**: An iterative transformation method used to calculate all eigenvalues and eigenvectors. Suitable for small models, but not for medium to large models. • **Subspace**: An iterative simultaneous vector method to calculate the smallest eigenvalues and the corresponding eigenvectors. Suitable for quickly finding the lowest frequencies in medium to large models. • **FEAST**: uses the FEAST algorithm to effectively calculate all the eigenvalues within a specific range. Suitable for any size structure. For more information, see: http://www.ecs.umass.edu/~polizzi/feast/.
Mass Model	Allows you to select how the mass model of the analysis is displayed. The options are: • **Consistent** • **Lumped**
Mass reporting	For users that wish to determine a center of mass for each floor, the **Simple** mass option is available to simplify the calculation. A single value of mass for each node is listed in the Tabular Data for both the "Active Masses by Node" & "Total Masses by Node" view types. The simple mass is calculated by averaging the mass values in the active directions. **NOTE** It is recommended that the simple mass option is used only when the lumped mass model is used.
Stopping Criteria	Stopping criteria prevent analysis continuing forever. If either of the criteria (**Maximum number of modes** or **Stopping Frequency**) are

Button, command, or option	Description
	met, the analysis will not look for any more modes.
Modes	Allows you to select one of the following options: • **Automatic number of modes**: If the option is selected, you must specify the mass participation required in each direction. You can optionally specify an initial number of modes that should be close to the actual number required in order to speed up the analysis process. • **Total number of modes**: If **Automatic number of modes** is cleared, you can specify the total number of modes required. The default value is 10.
Jacobi	Allows you to adjust the following settings: • **Maximum number of sweeps**: A sweep is a transformation of every off-diagonal in the global matrices. The option allows you to set the maximum number of sweeps allowed. • **Sweep tolerance**: At the end of each sweep, values are checked against the previous sweep results. If the difference is less than the tolerance specified here, the result is converged, and the analysis is complete.
Subspace	Allows you to adjust the following settings: • **Maximum number of iterations**: The number of iterations to perform. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance specified here,

Button, command, or option	Description
	the result is converged, and the analysis is complete.
FEAST	Allows you to adjust the following settings: • **Initial search range**: Specifies the initial range of values FEAST will search for eigenvalues in. • **Overestimation multiplier**: Specifies the initial guess for the subspace dimension within each range; an overestimate of the predicted number of modes in the range. • **Maximum modes in range**: The maximum number of modes in the range. If more modes are found in a range, the range is split into several smaller ranges. • **Minimum search range**: When a range is smaller than the value specified here, it will no longer be split, even if the maximum number of modes is greater than that allowed.

2nd order buckling settings

Button, command, or option	Description
Maximum number of iterations	The number of iterations to perform. Default = 1000.
Tolerance	At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.00001.
Maximum number of sweeps	A sweep is a transformation of every off-diagonal in the global matrices. The option allows you to set the maximum number of sweeps allowed. Default = 50.

Button, command, or option	Description
Sweep tolerance	At the end of each sweep, values are checked against the previous sweeps results. If the difference is less than the tolerance determined here the result is converged, and the analysis is complete. Default = 1.0E-12.
Total number of modes	Allows you to specify the total number of modes required. Default = 10.
Show negative buckling factors	Default = OFF.
Extraction Method	Allows you to select the appropriate extraction method for your model. The options are: • **Jacobi**: An iterative transformation method used to calculate all eigenvalues and eigenvectors. Suitable for small models, but not for medium to large models. • **Subspace**: An iterative simultaneous vector method to calculate the smallest eigenvalues and the corresponding eigenvectors. Suitable for quickly finding the lowest frequencies in medium to large models. • **Automatic** (default option): Tekla Structural Designer determines the most appropriate extraction method for the structure.

1st order seismic settings

Button, command, or option	Description
Extraction Method	Allows you to select the appropriate extraction method for your model. The options are: • **Automatic** (default option): Initially uses the **Subspace** option to find the lowest modes. If the criteria (either mass, or number of modes) is fulfilled, **FEAST** is then automatically used to find higher

Button, command, or option	Description
	modes until the stopping criteria is fulfilled. • **Jacobi**: An iterative transformation method used to calculate all eigenvalues and eigenvectors. Suitable for small models, but not for medium to large models. • **Subspace**: An iterative simultaneous vector method to calculate the smallest eigenvalues and the corresponding eigenvectors. Suitable for quickly finding the lowest frequencies in medium to large models. • **FEAST**: uses the FEAST algorithm to effectively calculate all the eigenvalues within a specific range. Suitable for any size structure. For more information, see: http://www.ecs.umass.edu/~polizzi/feast/.
Mass Model	Allows you to select how the mass model of the analysis is displayed. The options are: • **Consistent** • **Lumped**
Mass reporting	For users that wish to determine a center of mass for each floor, the **Simple** mass option is available to simplify the calculation. A single value of mass for each node is listed in the Tabular Data for both the "Active Masses by Node" & "Total Masses by Node" view types. The simple mass is calculated by averaging the mass values in the active directions. **NOTE** It is recommended that the simple mass option is used only when the lumped mass model is used.

Button, command, or option	Description
Stopping Criteria	Stopping criteria prevent analysis continuing forever. If either of the criteria (**Maximum number of modes** or **Stopping Frequency**) are met, the analysis will not look for any more modes.
Modes	Allows you to modify the following options: • **Initial number of modes**: In order to speed up the analysis process, you can specify an initial number of modes you expect to be required to achieve the required participation. The value specified here should be close to the actual number required because if you enter too few or too many modes, the analysis may take longer. • **Mass participation for RSA**: Allows you to specify the mass participation required in each direction. If this isn't achieved before the stopping criteria apply, the RSA analysis will still be performed, but a warning will be issued. • **Min. Mass participation for RSA**: If the minimum participation isn't achieved before the stopping criteria apply, the RSA analysis is not performed. **NOTE** The options in **Stopping Criteria** overrule both the number of modes and mass percentage.
Jacobi	Allows you to adjust the following settings: • **Maximum number of sweeps**: A sweep is a transformation of every off-diagonal in the global matrices. The option allows you to set the

Button, command, or option	Description
	maximum number of sweeps allowed. • **Sweep tolerance**: At the end of each sweep, values are checked against the previous sweeps results. If the difference is less than the tolerance determined here the result is converged, and the analysis is complete.
Subspace	Allows you to adjust the following settings: • **Maximum number of iterations**: The number of iterations to perform. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete.
FEAST	Allows you to adjust the following settings: • **Initial search range**: Specifies the initial range of values FEAST will search for eigenvalues in. • **Overestimation multiplier**: Specifies the initial guess for the subspace dimension within each range; an overestimate of the predicted number of modes in the range. • **Maximum modes in range**: The maximum number of modes in the range. If more modes are found in a range, the range is split into several smaller ranges. • **Minimum search range**: When a range is smaller than the value specified here, it will no longer be split, even if the maximum number of modes is greater than that allowed.

Button, command, or option	Description
Modal Combination Method	To determine the representative maximum response of interest for a loadcase, the relevant values for each relevant mode are combined by using the modal combination method specified here. Note that once modes have been combined, the relative signs are lost. The options are: • **Complete Quadratic Combination (CQC)**: Suitable for models where modes are closely spaced or well spaced. • **Square Root of Summation of Squares (SRSS)**: Suitable only for models where modes are well spaced.

Iterative cracked section analysis settings

Button, command, or option	Description
Global Convergence Criteria	Allows you to modify the following options: • **Maximum number of iterations**: The number of iterations to perform. Default = 200. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.00100. • **Relative**: Default: ON. **TIP** To resolve global convergence failures: • Increase the value in **Maximum number of iterations**. • Decrease the value in **Tolerance**.

<table>
<tr><th>Button, command, or option</th><th>Description</th></tr>
<tr><td>Local Convergence Criteria</td><td>Allows you to modify the following options:
• Maximum number of iterations: The number of iterations to perform. Default = 500.
• Tolerance: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.000001.
• Smoothing Parameter: A property only available for the Eurocode head code. Default = 0.005.

TIP To resolve local convergence failures:
• Increase the value in Maximum number of iterations.
• Decrease the value in Tolerance.
• If you are using the Eurocode head code, increase the value in Smoothing Parameter.</td></tr>
</table>

Modification factors

Different factors can be applied for each of the different materials in the model in order to adjust the following properties:

- E: Young's modulus
- G: Shear modulus
- $I_{torsion}$: Section inertia about local X
- I_{major}: Section inertia about local Y
- $I_{minor\ -}$ Section inertia about local Z
- Area: Section area in compression or tension

- $A_{\text{parallel to minor}}$: Section shear area in local Y
- $A_{\text{parallel to major}}$ - Section shear area in local Z
- t: shell thickness (applicable to concrete only)

The factors also vary according to the member types, (and in the case of concrete members whether they are cracked or not).

For concrete members in particular, design codes can require that analysis stiffness adjustment factors are applied since the appropriate properties to use in analysis are load and time dependent.

For various other investigations it is also possible that you will want to apply an adjustment to material properties. One suggested example is the assessment of structures subject to corrosion. Another classic requirement in this regard relates to torsion, it is common engineering practice to assume that if it will work without assuming any torsional strength, then torsion can be ignored.

Although default modification factors for each material are provided in the settings sets to reflect the design code being worked to, you should check that these are appropriate for your particular analysis model.

NOTE If you make changes to any of the modification factors, analysis must be repeated.

Meshing settings

The meshing options can be adjusted in order to fine-tune the meshes that are produced in slabs and walls. The default settings are generally appropriate, but they could potentially be adjusted if 2D element quality errors are being created during validation.

Composite steel beams settings

Button, command, or option	Description
Inertia used for loading analysis list	Allows you to specify the inertia to be used in the global analysis of the model for a composite steel beam: • Steel beam (default option) • Long term composite • Short term composite **NOTE** • If the model still fails to converge when using relaxation factors, expanding the allowable range (by changing the

Button, command, or option	Description
	above minimum and maximum values) may produce convergence. • Relaxation factors are not activated by default as it is anticipated they will not be required for the majority of models. In addition, since the solution process is necessarily more complex, their use can increase the analysis time to a degree. The inertia has different effects on different head codes: • US head code: Long and short term composite inertia only apply to rolled symmetric I sections without web openings. For all other sections, the standard steel beam inertia is used regardless of the analysis option selected. • EC head code: Long and short term composite inertia only apply to: • Symmetric I rolled sections without web openings • I plated sections without web openings • Westok plated sections without web openings • Fabsec sections without web openings For all other sections, the standard steel beam inertia is used regardless

Button, command, or option	Description
	of the analysis option selected. • BS head code: Long and short term composite inertia only apply to: • Symmetric I rolled sections without web openings • Asymmetric I rolled sections without web openings • I plated sections without web openings • Westok plated sections without web openings • Fabsec sections without web openings For all other sections, the standard steel beam inertia is used regardless of the analysis option selected. • AUS and IS head codes: Design of composite beams is not currently supported for the head codes.

Design Settings

The **Design Settings** sub-pages allow you to control the design options applied:

- to the **current** project - when accessed from the Design Settings dialog (page 1179) on the **Design** tab.
- to **new** projects - when accessed from the Settings dialog (page 1209) on the **Home** tab.

Design Settings contains the following sub-pages:

- Analysis (page 1066)

Design Settings - General and Analysis

General design settings

Button, command, or option	Description
Fix column nodes horizontally	Allows you to automatically apply lateral translational fixed supports to column nodes for the 3D building analysis performed during the gravity design processes. The option is useful at the early design stage when lateral systems have not been established. The automatic lateral supports can remove unwanted lateral displacements that may prevent the analysis from completing and produce unrealistic forces. **NOTE** The option does not apply to: • The column nodes that are in a rigid floor diaphragm • An analysis run as a separate process • The **Analyze All (Static)** command

Button, command, or option	Description
	• The 3D building analysis performed during any **Design (Static)** command

Analysis settings

Button, command, or option	Description
Analysis	Allows you to select if a first or second order 3D building analysis is performed. **TIP** For steel structures in particular you should consider running a first-order analysis for the initial gravity design before switching to a second-order analysis for the final design. In addition, **Solve lateral loadcases in isolation if non-linear analysis is required during design** allows you to try to solver lateral loadcases in isolation if non-linear analysis is required. However, in non-linear analysis, lateral loadcases often fail to solve or take a long time to converge when considered in isolation, especially for compression only foundation mats. Therefore, we recommend that you leave the option cleared. If you need to see the results for the individual lateral loadcases, you can either run the analysis directly on the **Analyze** tab, or select the option if you know that the analysis does not fail.
Stability coefficient tolerance United States head code (ACI/AISC) head code only	**Stack height /** ratio allows you to adjust the value of deflection that can safely be ignored. Selecting the option prevents any small deflections from causing erroneously high stability

Button, command, or option	Description
	coefficients in stability coefficient calculations. **NOTE** If the second order drift is less than the tolerance specified in **Stack height /**, the stability coefficient value is returned as N/A with a note that states that the drift is small enough to be ignored.
Reduced stiffness factor United States head code (ACI/AISC) head code only	This factor is only exposed once the analysis is set to second-order. It is applied to the stiffness (EI and EA) of all steel members in addition to any settings made on Analysis Options, Modification Factors. For correct design to the AISC Specification using the DAM, it should be set to 0.8. **NOTE** As an alternative to setting the analysis to first-order to explore the reason for any second order analysis failure, it is possible to alter this factor. If you set it to a value of say 10, this will stiffen both the Modulus of elasticity (E) and the shear modulus of elasticity (G) by a factor of 10 in the second order analysis. Although the results will not be able to be used for a valid design, it should now be possible to run the analysis to see which member might fail a design and hence be the cause of the analysis instability. This factor can then be reduced towards 0.8 for further investigation.

Design Settings - Steel > General (Eurocode only)

General

NOTE Partial safety factors are only applicable to EC3 and relevant National Annexes.

Button, command, or option	Description
Partial safety factors	
Use user defined partial safety factors	When **Use user defined partial safety factors** is selected, the options to define partial safety factors are enabled. Otherwise, the corresponding country defaults will be used. **NOTE** Default for this setting is unselected.
Resistance of cross-sections, γ_{M0}	Specify a user-defined value for this partial safety factor as required.
Resistance of members to instability, γ_{M1}	Specify a user-defined value for this partial safety factor as required.
Resistance of cross-sections to tension, γ_{M2}(EC3 - Part 1-1)	Specify a user-defined value for this partial safety factor as required.
Resistance of joints, γ_{M2}(EC3 - Part 1-8)	Specify a user-defined value for this partial safety factor as required.

Design Settings - Steel > Composite Beams

Composite beams

Button, command, or option	Description
Tolerance of rectilinearity	The calculation of the effective width of composite beams is only performed if they lie within the tolerance on rectilinearity adjusted here. The default tolerance is 15 degrees. At greater angles, you will be prompted to type the effective width manually.

Button, command, or option	Description
Update effective width prior to design check	Allows you to determine the effective width each time auto or check design is run. **NOTE** Unless this box is selected, you will be unable to manually override the effective width on the Floor construction and Natural frequency pages of the beam properties.

Design Settings - Steel > Steel Joists

Steel joists

NOTE Steel joists are only applicable in the US.

Button, command, or option	Description
Uniform load tolerance	If the calculated percentage load tolerance for steel joists is less than this value, the load is classified as Uniform.
Equivalent load tolerance	If the calculated percentage load tolerance for steel joists is greater than this value, the load is classified as Non-uniform. **NOTE** If 0% equivalent load tolerance is specified, only pure uniform loading is accepted.
Maximum sum of concentrated loads	Allows you to specify a user-defined maximum sum of concentrated loads that limits K, LH and DLH joists. If any concentrated loads exceed the specified maximum sum, the relevant load combination is classified as Non-uniform.
Maximum eccentricity of zero shear	In the equivalent uniform load method, the position of the point of

Button, command, or option	Description
	zero shear relative to the center span point of the joist is determined. The option allows you to specify the maximum eccentricity of zero shear value in the equivalent uniform load method. If the position is located outside the value, the load is classified as Non-uniform.
Deflection increase to allow for shear	Deflections due to live load are calculated using the following effective inertia value: $I_{effective} = I_{gross} / (1 + R)$, where R = the deflection increase to allow for shear effects (defaulted to 15%) and applied to KCS only. The option allows you to adjust the value of R, (the deflection increase to allow for shear, percentage).

Design Settings - Concrete > Cast-in-place

General

Buttons, commands, or options	Description
Limitation on concrete cylinder strength for shear/torsion design (Eurocode only)	
Normal weight concrete limit **Lightweight concrete limit**	For shear/torsion design the cylinder strength and its use in all derived mechanical properties is limited (but 'overridable') for specific national annexes. Default national annex values of these limits for normal weight/lightweight respectively: • UK, Eire, Singapore, Malaysia: 50/50 • Norway: 65/55

Buttons, commands, or options	Description
	• EU, Finland, Sweden: 90/90 (no limit stated) **NOTE** Changing the headcode/settings set does NOT update these values automatically.
Reinforcement anchorage length parameters (used in the calculation of the ultimate bond stress, from which anchorage lengths are determined)	
Plain Bars Bond Quality Modifier	Default value 0.5.
Deformed Bars Bond Quality Modifier	Default value 0.8. Only applies to the Eurocode head code.
Type-1 Bars Bond Quality Modifier	Default value 1.0. Only applies to the Eurocode head code.
Type-2 Bars Bond Quality Modifier	Default value is 1.0. Only applies to the ACI headcode.

Beam

. . . . Reinforcement Settings

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Longitudinal bars	Allows you to adjust the following settings applied to longitudinal bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the longitudinal bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the longitudinal bar design process. • **Minimum side bar size**: Sets the minimum allowable side bar size that can be used in the longitudinal bar design process. • **Minimum top steel clear spacing**: Sets the minimum allowable top steel clear spacing that can be used in the longitudinal bar design process.

Buttons, commands, or options	Description
	• **Minimum bottom steel clear spacing**: Sets the minimum allowable bottom steel clear spacing that can be used in the longitudinal bar design process. • **Maximum tension steel spacing**: Sets the maximum allowable tension steel spacing that can be used in the longitudinal bar design process. • **Maximum compression steel spacing**: Sets the maximum allowable compression steel spacing that can be used in the longitudinal bar design process. • **Use single bars when beam width <=**: Single bars are only permitted in beams whose width is less than the width specified. • **Steel overstrength factor**: Sets the value that accounts for the reinforcement steel yielding overstrength in seismic design. Only applies to the ACI headcode.
Short span maximum length	Spans smaller than the value set here are treated as short spans. Support bars of short spans are merged with the span bars.

. . . . Detailing Settings

Buttons, commands, or options	Description
Use same size bars in multilayer arrangements	Allows you to use the same bar size in each layer.
Use same number of bars in each layer	Allows you to use the same number of bars in each layer.
Cut length rounding increment	Allows you to specify a value to which cut lengths are rounded.
Merge identical longitudinal bars if appropriate	Allows you to merge bars of the same size, number, and position, provided that the total length of the merged bar does not exceed the max allowable bar length. **TIP** The max bar length can be verified on the **Reinforcement** page of the

Buttons, commands, or options	Description
	Materials dialog box. To do so, click an available bar size and then, click **View...** **NOTE** Top bar patterns 1 and 2 do not have any bars that you could merge.
Extend top longitudinal support bars symmetrically	Extends the support bars symmetrically to both spans based on the larger effective span length; but only if the spans vary by less than the percentage specified.
Extend top longitudinal support bars by anchorage length	Adds anchorage lengths to the calculated extension lengths.
Min anchorage length {value} x dia	Allows you to control the minimum anchorage length as a multiple of bar diameter. **NOTE** The option is only available for the Eurocode headcode.
Min anchorage length of plain bars {value} x dia	Allows you to control the minimum anchorage length as a multiple of bar diameter when the rib type is plain. **NOTE** The option is only available for the ACI headcode.
Extend top span bars to end support	Extends the top span bars of the first and last spans to the end supports.
Use 'U' bars at end support if appropriate	Allows you to replace the top and bottom bars at the end support region with 'U' bars under certain conditions. The bars that are joined or merged to create the 'U' bars depend on the top and bottom patterns selected for the beam. The anchorage lengths for the resulting 'U' bars are taken as the lengths required for the pair of bars that made the 'U' bar.
Select same bar size in support region and in the span	Allows you to use the same bar size in the support and span regions.

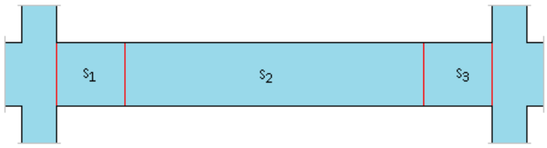
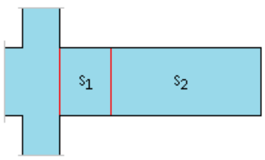
Buttons, commands, or options	Description
Select symmetrical stirrup in support region	Allows you to use the same stirrup arrangement (bar size and spacing) in both supports.

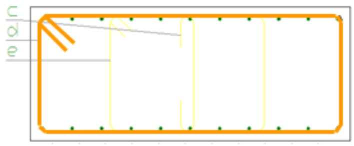
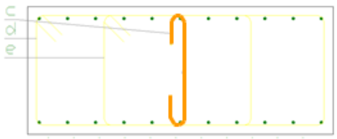
. . . . Top / Bottom Longitudinal Bar Pattern

Buttons, commands, or options	Description
Longitudinal Bar Pattern	Allows you to select a pattern to be viewed and modified.
Longitudinal Default Pattern	Allows you to select the pattern that is applied to new beams when they are first created. **NOTE** The **Longitudinal Default Pattern** option cannot be used to change the pattern applied to existing beams. Instead, you should modify the beam properties.
Continuous Span + Cantilever (Backspan) tab	Allows you to define how the selected longitudinal bar pattern is applied to continuous spans and cantilever backspans. An interactive diagram updates to match the changes you make. • Bars: Selected checkboxes that are greyed out are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your needs. • Region fields: The regions in which bars are applied are defined as a percentage of the span length. • Bar lapping: Select whether you want to lap the bottom bars in the middle of the support region (inside support) or at the face of the support region (outside support).
Single Span tab	Allows you to define how the selected longitudinal bar pattern is applied to single spans. An interactive diagram is also displayed and updates to match the changes you make.
Cantilever	Allows you to define how the selected longitudinal bar pattern is applied to

Buttons, commands, or options	Description
	cantilevers. An interactive diagram updates to match the changes you make.

. . . . *Stirrup (Link) Settings*

Buttons, commands, or options	Description
Normal	When considering shear, the design shear checks are performed in each of 3 regions S_1, S_2, and S_3 as shown below. In each region, the maximum vertical shear from all load combinations is determined and this maximum value used to determine the shear reinforcement required in that region.  • **Region S1**, **Region S2**, **Region S3**: The regions are defined as fixed proportions of the clear span of the beam. By defining the extent of the S_1 region, the other regions are determined automatically. • **Stirrup Type**: Select either open or closed stirrups in the list.
Cantilever	In cantilevers, the design shear checks are performed in 2 regions S_1 and S_2 as shown below. 
Minimum bar size	Sets the minimum allowable bar size that can be used in the design process.
Maximum bar size	Sets the maximum allowable bar size that can be used in the design process.
Minimum spacing	Sets the minimum allowable stirrup spacing that can be used in the design process.

Buttons, commands, or options	Description
Maximum spacing	Sets the maximum allowable stirrup spacing that can be used in the design process.
Spacing increment	The designed stirrup spacings are multiples of this value.
Maximum stirrup leg spacing across beam	Allows you to determine if single stirrups, double stirrups, or more are required, depending on the width of the beam.
Use single outside stirrup	Allows you to use a single outside link with additional links added as required (as per stirrup d shown dotted below). 
Accept single leg internal stirrup	Allows the use of single leg internal links (as per stirrup c shown dotted below). 
Optimize stirrup design regions where possible	In this case in the central region S_2, shear reinforcement is provided to meet the minimum of the code requirement or user preference whilst in regions S_1 and S_3, designed shear reinforcement is required. The position and length of region S_2 is determined from considerations of the shear resistance of the concrete cross-section and this then enables the lengths of regions S_1 and S_3 to be determined.

. . . . General Parameters

Buttons, commands, or options	Description
Partial fixity coefficient, β	A coefficient applied to the maximum positive moment in the beam span (excluding support positions) to set a user-defined minimum design moment

Buttons, commands, or options	Description
	for beam support regions. Allowable range is 0.0 to 1.0. Default value is 0.25 **NOTE** The option is only available for the ACI headcode.
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.
Long term deflection period	Allowable range is 3 to 60 months. Default value is 60 months. **NOTE** The option is only available for the ACI headcode.
Time at which brittle finishes are introduced	Allowance range is 1 to 6 months. Default value is 1 month. **NOTE** The option is only available for the ACI headcode.
Design Beams for FE Chasedown analysis results	Allows you to specify that the beams are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.

Buttons, commands, or options	Description
Tolerance on rectilinearity	The calculation of the effective width is only carried out for concrete beams if they lie within the tolerance on rectilinearity set here. The default tolerance is 15 degrees. At greater angles you will be prompted to enter the effective width manually.

Column

. . . . Reinforcement Layout

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Vertical bars	Allows you to adjust the following settings applied to vertical bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the vertical bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the vertical bar design process. • **Minimum clear spacing**: Sets the minimum allowable clear bar spacing that can be used in the vertical bar design process. • **Maximum center spacing**: Sets the maximum value for bar spacing when a column is set to auto design mode. The exact spacing may be reduced to meet the requirements of the design if the design fails with the bars at the maximum spacing. • **Spacing increment**: Sets the value whose multiples the designed bar spacings are. • **Minimum reinforcement ratio**: Sets the minimum allowable area of reinforcement as a ratio of the concrete section area.

Buttons, commands, or options	Description
	• **Maximum reinforcement ratio**: Sets the maximum allowable area of reinforcement as a ratio of the concrete section area. • **Steel overstrength factor**: Sets the value that accounts for the reinforcement steel yielding overstrength in seismic design.
Tie bars	Allows you to adjust the following settings applied to tie bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the tie bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the tie bar design process. • **Minimum clear spacing**: Sets the minimum allowable clear bar spacing that can be used in the tie bar design process. • **Maximum center spacing**: Sets the maximum value for bar spacing when a column is set to auto design mode. The exact spacing may be reduced to meet the requirements of the design if the design fails with the bars at the maximum spacing. • **Spacing increment**: Sets the value whose multiples the designed bar spacings are.
Multiple Layers	For circular column sections only, if autodesign requires a 2nd layer of reinforcement you can control the **layer spacing**. Autodesign will use a larger spacing if needed - and it is checked in the check results.
Aspect ratio change point	Allows you to specify a value that determines which rectangular sections are considered to have a low aspect ratio and which are considered to have a high aspect ratio.

Buttons, commands, or options	Description
Use double ties for low aspect ratio	Allows you to use double ties where applicable instead of single ties for sections with a low aspect ratio.
Use double ties for high aspect ratio	Allows you to use double ties where applicable instead of single ties for sections with a high aspect ratio.
Use triple ties for low aspect ratio	Allows you to use triple ties where applicable instead of double or single ties for sections with a low aspect ratio.
Use cross ties for low aspect ratio	Allows you to use cross ties where applicable instead of triple, double or single ties for sections with a low aspect ratio.

. . . . Detailing Settings

Buttons, commands, or options	Description
Cut length rounding increment	Allows you to specify a value to which cut lengths are rounded.
Kicker dimension	Allows you to specify the height of column kicker cast above the slab level.
Assumed foundation depth for starter bars	Allows you to control the length of starter bars at the base of the column in drawings.
Join identical bars where possible	Allows you to merge bars together only if the bar size and layout in the current and adjacent stack are identical and proving that the max bar length is not exceeded.
Number of span region ties required to use separate regions	Allows you to specify the minimum number of span region ties that are necessary for a span region to be indicated on the detail, and in consequence, standardize the tie spacing on the detail.
Provide ties through full foundation depth	Allows you to draw ties through the full depth of the foundation.
Foundation penetration depth of ties	Allows you to specify the penetration depth of ties into the foundation. **NOTE** The option is only available in the If the **Provide ties through full foundation depth** option is cleared.

Buttons, commands, or options	Description
Provide ties through floor depth for internal walls	For walls restrained by flat slabs: ties are always provided through the floor depth irrespective of this setting. For walls restrained by beam and slab: ties are always provided through the beam depth for edge walls but are only provided for internal walls when the option is selected. When the option is cleared, ties are provided up to the soffit of the shallowest beam depth.
Tie bar type	Allows you to select the tie bar type that you want to use.

. . . . General Parameters

Buttons, commands, or options	Description
Bar sizes no smaller than stack above	Allows you to ensure that bar sizes do not reduce in lower stacks.
Match bar position to stack above	Allows you to have the starting arrangement for longitudinal bars match the arrangement of the bars in the stack above if the section geometry matches.
Increase main bar size preferentially	Allows you to have the corner bars increased in preference to the intermediate bars. All bars start off at the same size (unless the initial bar size is driven by the current arrangement or the stack above), but when the check fails, the corner bars will be increased in size if all bar sizes are the same. Otherwise, the intermediate bars will be increased in size. This means that when the final design is produced, either all bars will be the same size or the corner bars will be one size larger than the intermediate bars. Alternatively, if you require all bar sizes to be the same size, clear the option. This way, all bar sizes are increased together.
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the

Buttons, commands, or options	Description
	anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.
Design Columns for FE Chasedown analysis results	Allows you to specify that the columns are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.

Wall

. . . . Reinforcement Layout

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Vertical bars	Allows you to adjust the following settings applied to vertical bars: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the vertical bar design process. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the vertical bar design process. • **Minimum clear spacing**: Sets the minimum allowable clear bar spacing that can be used in the design process.

Buttons, commands, or options	Description
	• **Maximum center spacing**: Sets the maximum value for bar spacing when a wall is set to auto design mode. The exact spacing may be reduced to meet the requirements of the design if the design fails with the bars at the maximum spacing. • **Spacing increment**: Sets the value whose multiples the designed bar spacings are. • **Minimum reinforcement ratio**: Sets the minimum allowable area of reinforcement as a ratio of the concrete section area. • **Maximum reinforcement ratio**: Sets the maximum allowable area of reinforcement as a ratio of the concrete section area.
Vertical bars in end zone	Allows you to adjust the following settings applied to vertical bars in the end zone area: • **Minimum bar size**: Sets the minimum allowable bar size that can be used in the vertical bar design process in the end zone area. • **Maximum bar size**: Sets the maximum allowable bar size that can be used in the vertical bar design process in the end zone area. • **Minimum reinforcement ratio**: Sets the minimum allowable area of reinforcement as a ratio of the end zone area. • **Maximum reinforcement ratio**: Sets the maximum allowable area of reinforcement as a ratio of the end zone area.
Horizontal bars	Allows you to set the minimum allowable bar size that can be used in the design process of horizontal bars and the minimum allowable area of reinforcement as a ratio of the end zone area.
Tie/horizontal bars	Allows you to set the minimum and maximum allowable bar sizes that can be

Buttons, commands, or options	Description
	used in the tie/horizontal bar design process and specify the value whose multiples the designed bar spacings are.
Tie/confinement bars	Allows you to set the minimum and maximum allowable bar sizes that can be used in the tie/confinement bar design process.
Confinement bars in end zone	Allows you to set the minimum and maximum allowable bar sizes that can be used in the confinement bar design process.
Substitute loose bars if mesh inadequate	Allows you to use additional loose bars in the end zones when the mesh is inadequate.

*. . . . **Detailing Settings***

Buttons, commands, or options	Description
Cut length rounding increment	Allows you to specify a value to which cut lengths are rounded.
Kicker dimension	Allows you to specify the height of column kicker cast above the slab level.
Assumed foundation depth for starter bars	Allows you to control the length of starter bars at the base of the wall in drawings.
Join identical bars where possible	Allows you to merge bars together only if the bar size and layout in the current and adjacent stack are identical and proving that the max bar length is not exceeded.
Number of span region ties required to use separate regions	Allows you to specify the minimum number of span region ties that are necessary for a span region to be indicated on the detail, and in consequence, standardize the tie spacing on the detail.
Provide ties through full foundation depth	Allows you to draw ties through the full depth of the foundation.
Foundation penetration depth of ties	Allows you to specify the penetration depth of ties into the foundation. **NOTE** The option is only available in the If the **Provide ties through full foundation depth** option is cleared.

Buttons, commands, or options	Description
Provide ties through floor depth for internal walls	For walls restrained by flat slabs: ties are always provided through the floor depth irrespective of this setting. For walls restrained by beam and slab: ties are always provided through the beam depth for edge walls but are only provided for internal walls when the option is selected. When the option is cleared, ties are provided up to the soffit of the shallowest beam depth.
Tie bar type	Allow you to select the tie bar type that you want to use.
Tie bar type in end zones	

. . . . *General Parameters*

Buttons, commands, or options	Description
Bar sizes no smaller than stack above	Allows you to ensure that bar sizes do not reduce in lower stacks.
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.
Design Walls for FE Chasedown analysis results	Allows you to specify that the walls are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.

Slab

. . . . Reinforcement Layout

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Minimum loose bar size	Allows you to set the minimum allowable loose bar size that can be used in the design process.
Maximum loose bar size	Allows you to set the maximum allowable loose bar size that can be used in the design process.
Minimum clear spacing	Allows you to set the minimum allowable clear bar spacing that can be used in the design process.
Minimum spacing (slab auto-design)	Allows you to set the minimum allowable clear bar spacing used in the slab item auto design process. As the option only applies to slab items, it allows panel design to more readily select bars at a wider spacing, while allowing patch design to more readily pass by reducing minimum spacing check failures in the patches.
Maximum principal bar spacing	Allows you to set the maximum allowable principal bar spacing both as a distance and as a function of the slab depth.
Maximum secondary bar spacing	Allows you to set the maximum allowable secondary bar spacing that can be used in the design process both as a distance and as a function of the slab depth.
Bar spacing increment	Allows you to set a value whose multiples the bar spacings are.
Maximum bar length	Allows you to set the maximum length of a bar.
Bar length increment	Allows you to set a value whose multiples the bar lengths are.
Use mesh where possible	Select to use mesh where possible in bars.
Make bob for top steel of cantilevers	Allows you to apply a bob to the end of the top steel in cantilevers.
Auto selection of outer bars	Allows you to automatically select the bars in the outer layer.

. . . . *Patches*

Buttons, commands, or options	Description
Use mesh where possible	Select to use mesh where possible in patches.
Patch Size	Allows you to set the default patch size to be used.
Center strip width	Allows you to set the width of the center strip in beam patches.
Edge strip width as proportion of patch width	Allows you to control how the patch width is split between edge and center strips in column patches.

. . . . *General Parameters*

Buttons, commands, or options	Description
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.
Allowance for additional detailing bars	Allows you to specify a percentage by which the total mass of bars for slabs or mats shown in material listing reports and in slab or mat detailing drawings is increased.

Punching Shear

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Reinforcement bars	Allows you to set the minimum and maximum allowable bar sizes that can be welded to the rail in the design process and set the maximum allowable diagonal spacing for radial layouts.
Spacing along rail	Allows you to set the minimum and maximum allowable bar spacing along each rail and specify a value whose multiples the bar spacings are.
Spacing to first reinforcement line	Allows you to set the minimum and maximum spacing from the column face to the first reinforcement line and define a value whose multiples the spacings from the column face are.
Auto-design	Allows you to select whether the auto design method is to minimize the bar size or minimize the number of rails and specify the minimum number of studs per face that will be used when auto-designing from minima.
Diagonal spacing between rails	Allows you to set the maximum diagonal spacing between the outermost studs in the last critical perimeter for circular arrangements.
Spacing between rails in Y direction	Allows you to set the minimum and maximum allowable rail spacing in the Y direction and specify a value whose multiples the bar spacings in the Y direction are.
Spacing between rails in Z direction	Allows you to set the minimum and maximum allowable rail spacing in the Z direction and specify a value whose multiples the rail spacings in the Z direction are.
Spacing from last line to outer perimeter	Allows you to set the maximum spacing from the last reinforcement line to the outer perimeter.

Foundations - General

Buttons, commands, or options	Description
Minimum distance from column/wall face to base/cap edge	Allows you to specify the minimum distance from the column or wall face to the base edge or cap edge.
Use presumed bearing capacity method for pad bases and mats (EN 1997-1 cl.6.5.2.4)	If this option is selected the bearing pressure is checked for serviceability combinations using the presumed bearing resistance specified in the pad base properties (no ‘STR’ and ‘GEO’ capacities). **NOTE** The option is only available for the Eurocode head code.

Isolated Foundations

. . . . Reinforcement Layout

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Minimum loose bar size	Allows you to set the minimum allowable loose bar size that can be used in the design process.
Maximum loose bar size	Allows you to set the maximum allowable loose bar size that can be used in the design process.
Minimum clear spacing	Allows you to set the minimum allowable clear bar spacing that can be used in the design process.
Maximum principal bar spacing	Allows you to set the maximum allowable principal bar spacing both as a distance and as a function of the slab depth.
Maximum secondary bar spacing	Allows you to set the maximum allowable secondary bar spacing that can be used in the design process both as a distance and as a function of the slab depth.
Bar spacing increment	Allows you to set a value whose multiples the bar spacings are.
Use mesh where possible	Select to use mesh where possible in bars.

. . . . Foundation Size

Buttons, commands, or options	Description
Minimum depth	Allows you to set the minimum allowable foundation depth.
Maximum depth	Allows you to set the maximum allowable foundation depth.
Auto-design depth increment	Allows you to set the depth increment.
Minimum side length	Allows you to set the minimum allowable side length.
Maximum side length / strip width	Allows you to set the maximum side length under columns as a distance.
Maximum side length ratio under columns	Allows you to set the maximum side length ratio under columns.
Rounding increment for footing dimensions	Allows you to specify a value to which the overall footing size is rounded.
Default 3-pile cap shape	Allows you to define whether the default shape is triangular or rectangular when 3 piles are used.

. . . . General Parameters

Buttons, commands, or options	Description
Maximum Bond Quality Coefficient	A value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	A value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$ **NOTE** The option is only available for the Eurocode head code.

Buttons, commands, or options	Description
(Pad Base tab) **Consider passive resistance of soil for sliding check**	Allows you to control whether the passive resistance of the soil is included in the check of Sliding Resistance.
(Pad Base tab) **Pad base shear reactions at pinned bases**	**Ignorable percentage of passive pressure** This allows you to ignore small shear forces at pinned bases. The default of 1% is typically a negligible load level which would commonly be ignored. If you do not want to ignore any shear forces, set the value to zero.
(Pad Base tab) **FOS for sliding and uplift checks**	Allows you to edit the design safety factor for these checks. **NOTE** The option is only available for the ACI/AISC head codes.
(Pile Cap tab) **Tolerance for pile position variation**	Allows you to specify the allowable deviation value of the pile position from its original plan position. The value is used in the design moment calculations.

. . . . Piles

Buttons, commands, or options	Description
Minimum number of piles for pile caps under columns	Minimum and maximum numbers of piles that apply when a pile cap under a column has been set to auto design the piles.
Maximum number of piles for pile caps under columns	
Minimum spacing option list	Allows you to select whether the minimum spacing check uses a set minimum spacing value or a multiple of the pile width or pile circumference.
Minimum spacing of piles	Allows you to set the minimum spacing value that you can select in the **Minimum spacing option** list.
Minimum pile edge distance	Allows you to set the minimum distance from the pile face to edge of foundation.
Pile auto-design method list	Allows you to select to use the smallest number of high capacity piles or more piles of lowest capacity when a pile cap has been set to auto design.

Buttons, commands, or options	Description
Limit maximum critical section size	When selected, allows you to specify a maximum critical section size as a multiple of the pile dimension.
Use pile capacity for pile punching and shear checks	When selected, allows you to use pile capacity for pile punching and punching shear checks. When unselected, the pile load is used.
Check piles for lateral load	When selected, a pile lateral capacity check is performed.

Mat Foundations

. . . . Reinforcement Layout

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Minimum loose bar size	Allows you to set the minimum allowable loose bar size that can be used in the design process.
Maximum loose bar size	Allows you to set the maximum allowable loose bar size that can be used in the design process.
Minimum clear spacing	Allows you to set the minimum allowable clear bar spacing that can be used in the design process.
Minimum spacing (slab auto-design)	Allows you to set the minimum allowable clear bar spacing used in the panel auto design process. As the option only applies to slab items, it allows panel design to more readily select bars at a wider spacing, while allowing patch design to more readily pass by reducing minimum spacing check failures in the patches.
Maximum principal bar spacing	Allows you to set the maximum allowable principal bar spacing both as a distance and as a function of the slab depth.
Maximum secondary bar spacing	Allows you to set the maximum allowable secondary bar spacing that can be used in the design process both as a distance and as a function of the slab depth.

Buttons, commands, or options	Description
Bar spacing increment	Allows you to set a value whose multiples the bar spacings are.
Maximum bar length	Allows you to set the maximum length of bar.
Bar length increment	Allows you to set a value whose multiples the bar lengths are.
Auto selection of outer bars	Allows you to automatically select the bars in the outer layer.
Use mesh where possible	Select to use mesh where possible in bars.

. . . . Patches

Buttons, commands, or options	Description
Use mesh where possible	Select to use mesh where possible in patches.
Patch Size	Allows you to set the default patch size to be used.
Center strip width	Allows you to set the width of the center strip in beam patches.
Edge strip width as proportion of patch width	Allows you to control how the patch width is split between edge and center strips in column patches.

. . . . General Parameters

Buttons, commands, or options	Description
Maximum Bond Quality Coefficient	Allows you to specify a value used in the calculation of the ultimate bond stress from which the anchorage lengths are determined. The allowable range is 0.5 to 1.0. **NOTE** The option is only available for the Eurocode headcode.
Maximum nominal aggregate size	Allows you to specify a value for the calculation to determine the minimum clear horizontal distance between individual parallel bars.
Allowance for deviation	Allows you to specify a value is used in the calculation to determine the limiting nominal concrete cover $c_{nom, lim}$. The full calculation is: $c_{nom, lim} = c_{min} + \Delta c_{dev}$

Buttons, commands, or options	Description
	NOTE The option is only available for the Eurocode headcode.

. . . . Piles

Buttons, commands, or options	Description
Use pile capacity for punching check	When selected, allows you to use pile capacity for the pile punching check. When unselected, the pile load is used.
Check piles for lateral load	When selected, a pile lateral capacity check is performed.

Design Settings - Concrete > Precast

Beam

. . . . Reinforcement Settings

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Reinforcement settings	Allows you to adjust the **Reinforcement Class** and **nominal cover** settings applied to beam reinforcement:

. . . . Top / Bottom Longitudinal Bar Pattern

Buttons, commands, or options	Description
Longitudinal Bar Pattern	Allows you to select a pattern to be viewed and modified.
Longitudinal Default Pattern	Allows you to select the pattern that is applied to new beams when they are first created. **NOTE** The **Longitudinal Default Pattern** option cannot be used to change the pattern applied to existing beams. Instead, you should modify the beam properties.
Continuous Span tab	Allows you to define how the selected longitudinal bar pattern is applied to

Buttons, commands, or options	Description
	continuous spans. An interactive diagram updates to match the changes you make. • Bars: Selected checkboxes that are greyed out are mandatory for the selected pattern. Other bars can optionally be included in the pattern according to your needs. • Region fields: The regions in which bars are applied are defined as a percentage of the span length. **NOTE** The Top and Bottom Region fields dictate how many **design sections** appear in the Tedds Calculation - a different design section is required at each interface between regions.
Pinned Span tab	Allows you to define how the selected longitudinal bar pattern is applied to single pinned spans. An interactive diagram is also displayed and updates to match the changes you make.
Cantilever	Allows you to define how the selected longitudinal bar pattern is applied to cantilevers. An interactive diagram updates to match the changes you make.

. . . . Stirrup (Link) Settings

Buttons, commands, or options	Description
Normal	When considering shear, the design shear checks are performed in each of 3 regions S_1, S_2, and S_3. In each region, the maximum vertical shear from all load combinations is determined and this maximum value used to determine the shear reinforcement required in that region. • **Region S1**, **Region S2**, **Region S3**: The regions are defined as fixed proportions of the span of the beam. By defining the extent of the S_1 region, the other regions are determined automatically.

Buttons, commands, or options	Description
Cantilever	In cantilevers, the design shear checks are performed in 2 regions S_1 and S_2.

*. . . . **General Parameters***

Buttons, commands, or options	Description
Design Beams for FE Chasedown analysis results	Allows you to specify that the beams are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.
Design precast beams for lifting forces	Allows you to specify that the beams are also to be designed for lifting forces, (the lifting check being optional in the Tedds calculation).

Column

*. . . . **Reinforcement Settings***

Buttons, commands, or options	Description
Country list	Allows you to specify the country used for the reinforcement information.
Reinforcement settings	Allows you to adjust the **Reinforcement Class** and **nominal cover** settings applied to column reinforcement:

*. . . . **General Parameters***

Buttons, commands, or options	Description
Design Columns for FE Chasedown analysis results	Allows you to specify that the columns are designed for the forces obtained from a previously performed FE chasedown analysis in addition to the forces obtained from the other analyses that have been performed.
Design precast column for lifting forces	Allows you to specify that the columns are also to be designed for a set of lifting forces, (the lifting check being optional in the Tedds calculation).

Design Settings - Design Forces

NOTE Values entered in the **Ignorable Forces Below** section:

- **are** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

Values entered in the **Minimum Design Forces** section:

- **are** considered when creating **member end force** reports/drawings and **foundation reaction** reports/drawings in Tekla Structural Designer
- **are** considered when performing **connection resistance checks** in Tekla Structural Designer
- **are** considered when designing **connections** in Tekla Connection Designer, or IDEA StatiCa
- **are not** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

Ignore forces below

NOTE Values entered in the **Ignorable Forces Below** section:

- **are** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

Design	
Ignore Forces Below	Allow you to specify appropriate negligible or nominal force levels to prevent 3D analysis from exposing small forces that are normally ignored in design. When the small forces in the 3D analysis are below the specified levels, they are ignored and the design in Tekla Structural Designerproceeds automatically.

Concrete beams (Eurocode only)

Button, command, or option	Description
Torsion force % of concrete resistance	Allows you to limit the maximum torsion force to a % of the concrete resistance.

Minimum design forces

NOTE Values entered in the **Minimum Design Forces** section:

- **are not** considered when designing **steel** and **cast-in-place concrete** members in Tekla Structural Designer
- **are not** considered when designing **timber** or **precast** members via Tekla Tedds

These settings provide flexibility to control minimum design values and rounding increments used in the following:

- Member end force reports/drawings
- Foundation reaction reports/drawings
- Connection resistance checks
- Export of connection forces to another application for design (Tekla Connection Designer or IDEA StatiCa)

Setting	Description
Brace Force Axial Force	Click the links to see examples of how the minimum entered here is used in: • Member end force reports/ drawings (page 1099) • Connection resistance checks (page 1108) • Exported connection forces (page 1110)
Beam End Forces Axial Force Minor Axis Shear Force Minor Axis Moment Major Axis Shear Force Major Axis Moment	Click the links to see examples of how the minimums and rounding increments entered here are used in: • Member end force reports/ drawings (page 1099) • Connection resistance checks (page 1108) • Exported connection forces (page 1110)
Rounding Increment for Force Rounding Increment for Moment	

Setting	Description
Foundation Reactions Axial Force Minor Axis Shear Force Minor Axis Moment Major Axis Shear Force Major Axis Moment	Click the links to see examples of how the minimums and rounding increments entered here are used in: • Foundation reaction reports/ drawings (page 1103) • Exported connection forces (page 1110)
Rounding Increment for Force Rounding Increment for Moment	

How Minimum Design Forces are used

The following topics illustrate how Minimum Design Forces are applied in different areas of Tekla Structural Designer.

Member end force reports/drawings

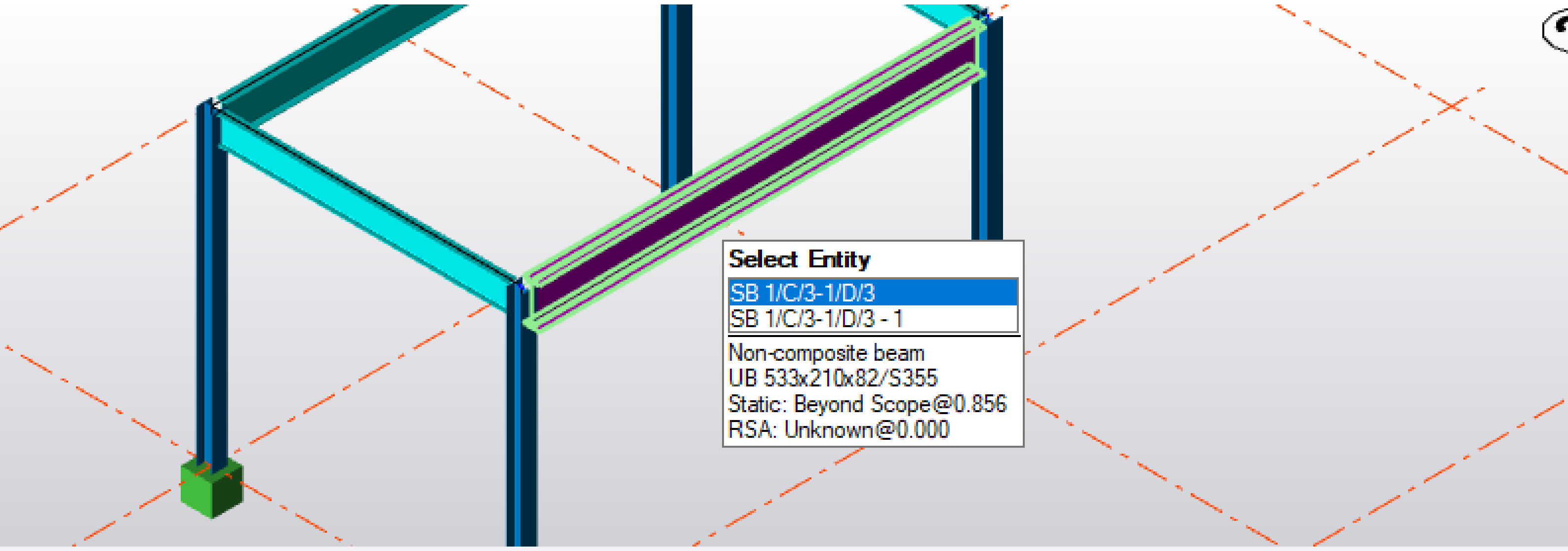
Tekla Structural Designer reports member end forces (Beam end forces/ Bracing forces) based on analysis results.

The 'Minimum Design Forces' inputs under **Brace Force/Beam End Forces** allow the user to specify minimum values for the reported forces, and also apply rounding increments to them.

NOTE **While minimum design values and rounding increments are applied to the member end force reports/drawings, they have no impact on steel and concrete member design in Tekla Structural Designer, or timber and precast design using Tekla Tedds.**

Consider a fixed beam with axial force + major/minor shear + torsion + major/ minor moment.

Forces/moments are reported in the Beam End Force report.

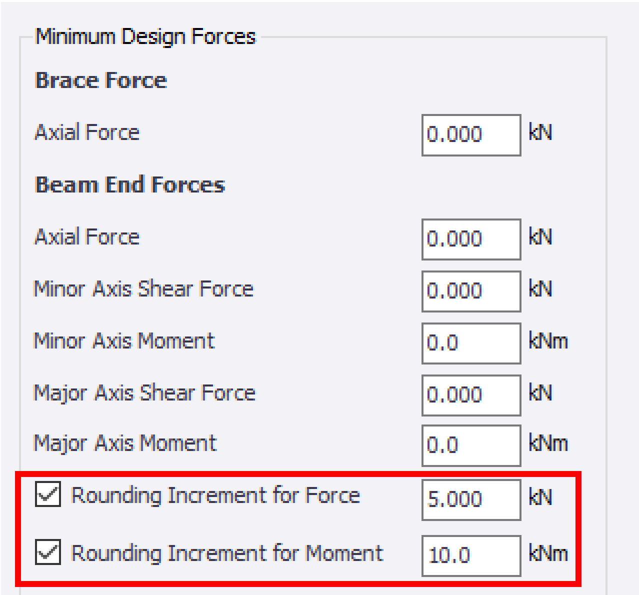


es

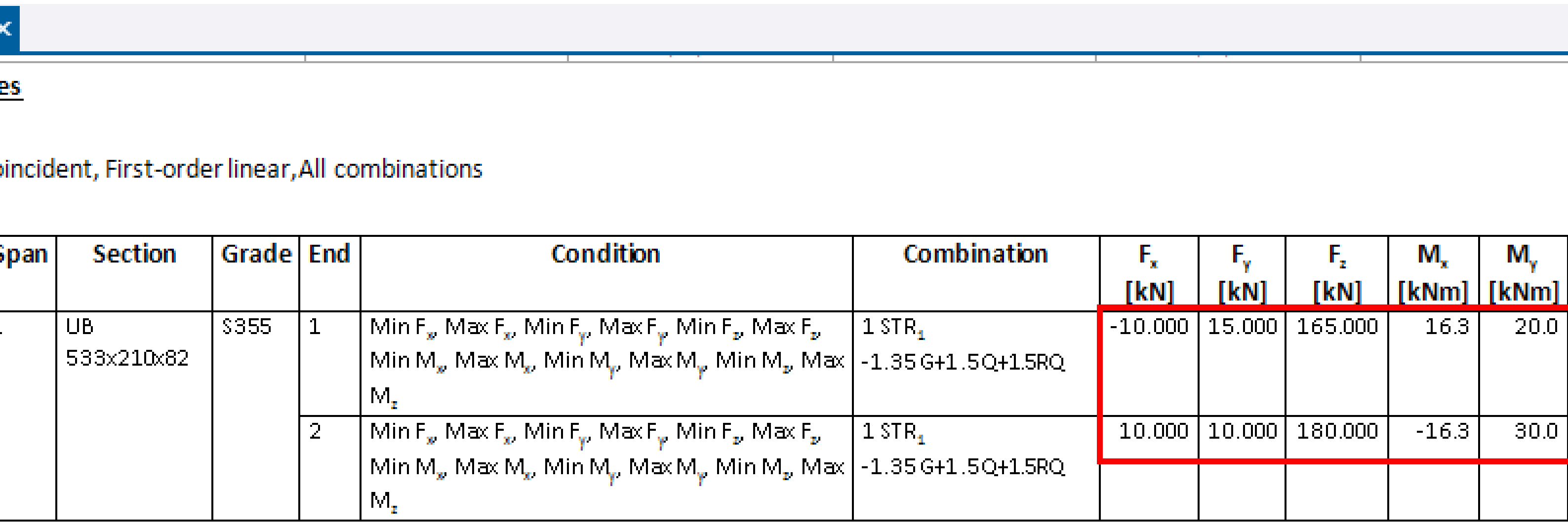
oincident, First-order linear, All combinations

Span	Section	Grade	End	Condition	Combination	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]
1	UB 533x210x82	S355	1	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	-6.224	10.438	163.878	16.3	10.4
			2	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	6.224	5.762	175.650	-16.3	24.9

The user wants the forces rounded to the nearest 5kN and moments to the nearest 10kNm so inputs 5kN in the 'Rounding Increment for Force' and 10kNm in the 'Rounding Increment for Moment' under **Beam End Forces**.



The Beam End Forces report displays rounded value of forces and moments.

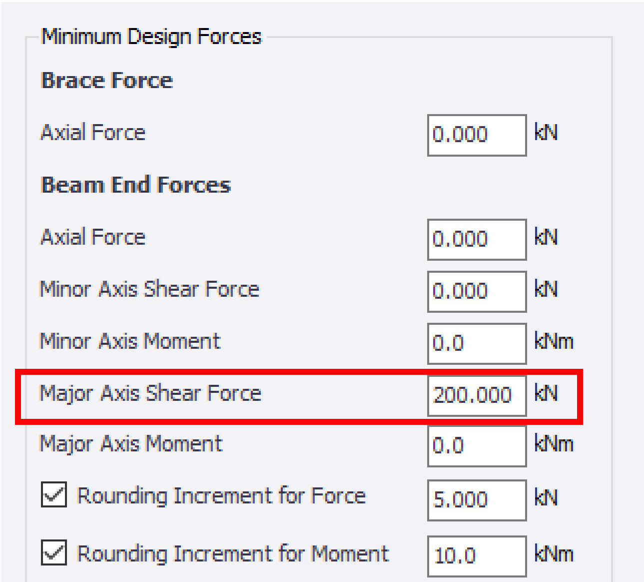

es

oincident, First-order linear, All combinations

Span	Section	Grade	End	Condition	Combination	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]
.	UB 533x210x82	S355	1	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	-10.000	15.000	165.000	16.3	20.0
			2	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	10.000	10.000	180.000	-16.3	30.0

NOTE The rounding increment for moment applies to major axis and minor axis moments, but does not apply to torsional moments (M_x).

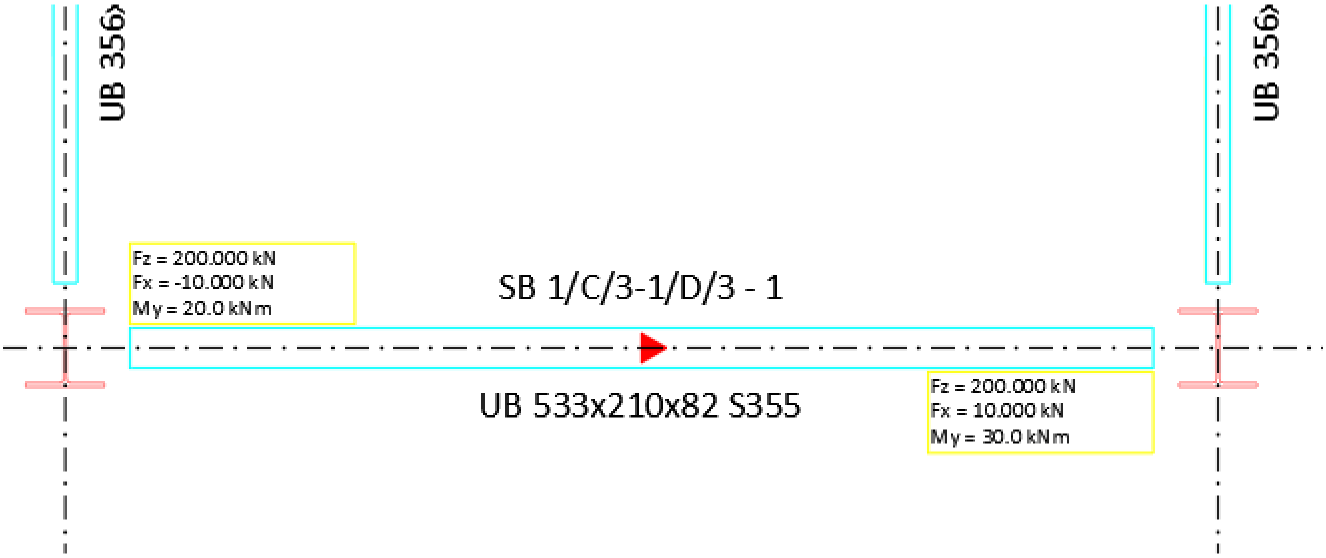
The user might want the forces reported to be not less than minimum values, for example they might enter 200kN in the 'Major Axis Shear Force' under '**Beam End Forces**.



This results in the report being updated as follows.

Span	Section	Grade	End	Condition	Combination	F_x [kN]	F_y [kN]	F_z [kN]	M_x [kNm]	M_y [kNm]
1	UB 533x210x82	S355	1	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	-10.000	15.000	200.000	16.3	20.0
			2	Min F_x, Max F_x, Min F_y, Max F_y, Min F_z, Max F_z, Min M_x, Max M_x, Min M_y, Max M_y, Min M_z, Max M_z	1 STR_1 -1.35G+1.5Q+1.5RQ	10.000	10.000	200.000	-16.3	30.0

The changes to the rounding increments and mimimum force are also reflected in the Beam End Forces drawing.



Foundation reaction reports/drawings

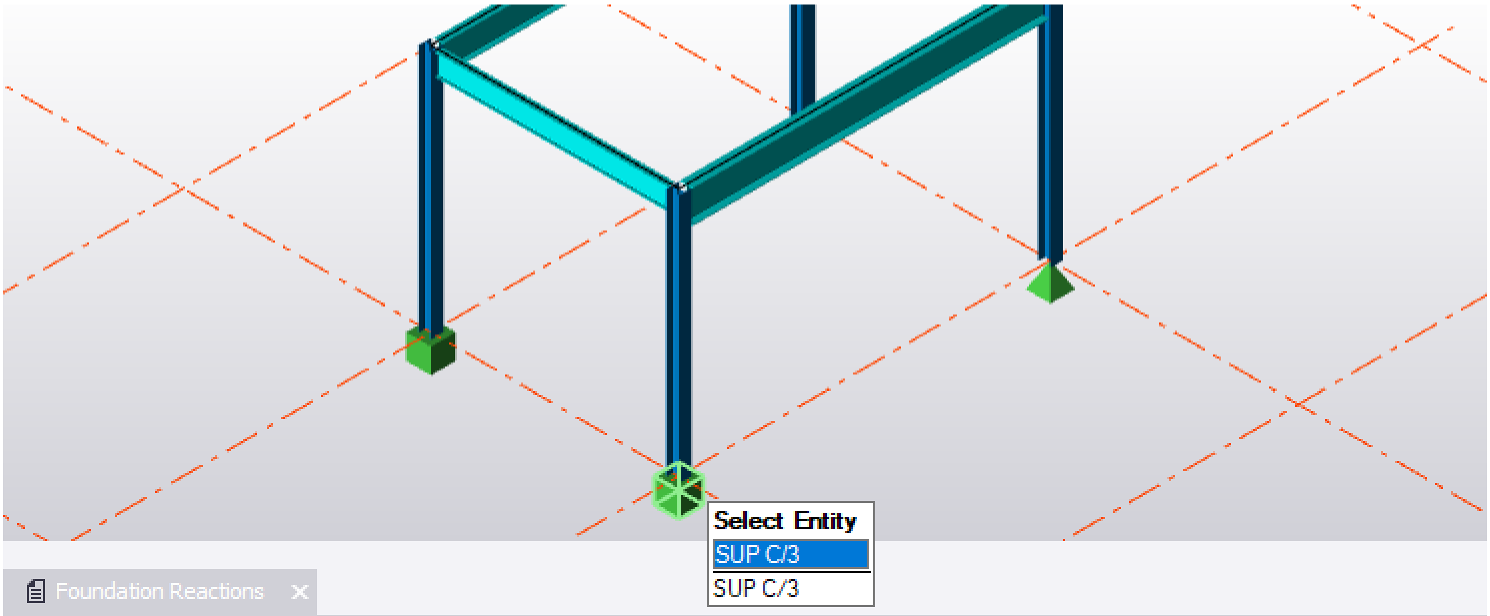
Tekla Structural Designer reports foundation reactions based on analysis results.

The ‘Minimum Design Forces’ inputs under **Foundation Reactions** allow the user to specify minimum values for the reactions, and also apply rounding increments to them.

NOTE Changed force/moment values are for reporting purposes only.

Consider a fixed end support.

Forces/moments are reported in the Foundation Reactions report.



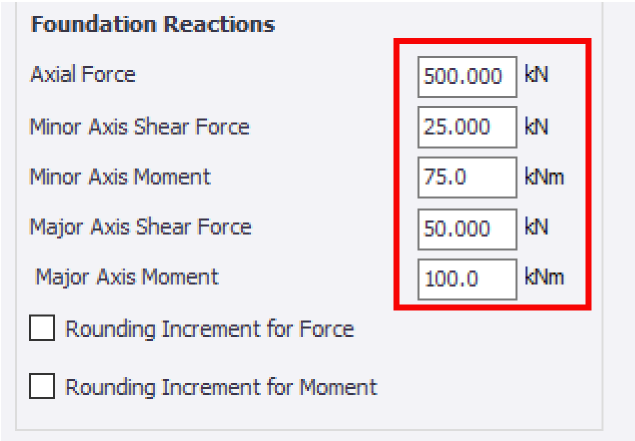
FoundationReactions

FoundationReactions, First-order linear, Strength Factors

Supports

Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
					F_{vert} [kN]	F_{major} [kN]	F_{minor} [kN]	M_{major} [kNm]	M_{minor} [kNm]	M_{tor} [kNm]
SUP A/1	0.0000	SC A/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	147.140	40.500	-40.500	0.0	0.0	0.0
SUP A/2	0.0000	SC A/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	75.824	0.000	0.000	0.0	0.0	0.0
SUP B/1	0.0000	SC B/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	148.911	0.000	0.000	0.0	0.0	0.0
SUP B/2	0.0000	SC B/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	104.596	0.000	0.000	0.0	0.0	0.0
SUP C/3	0.0000	SC C/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	278.195	3.468	-7.276	-30.2	-18.7	7.0
SUP C/4	0.0000	SC C/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	282.895	6.971	0.002	-34.6	0.0	0.0
SUP D/3	0.0000	SC D/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	289.967	4.071	-6.224	0.0	0.0	7.0
SUP D/4	0.0000	SC D/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	282.895	1.691	-0.002	0.0	0.0	0.0

The user wants these reported values to be not less than certain minimum values, so inputs the required minimums under **Foundation Reactions**.



The Foundation Reactions report shows minimum design forces/moments specified by the user.

Foundation Reactions

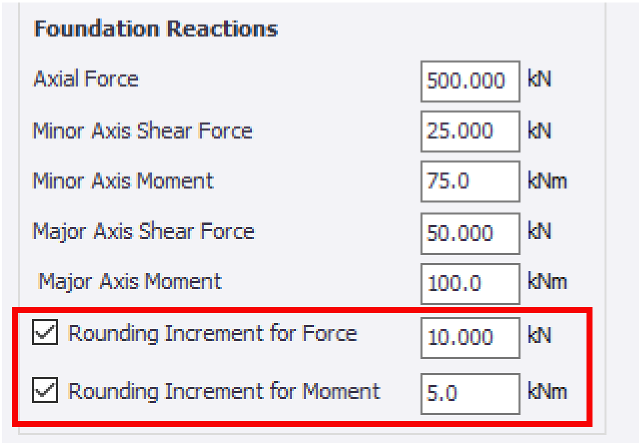
FoundationReactions

FoundationReactions, First-order linear, Strength Factors

Supports

Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions					
					F_{vert} [kN]	F_{major} [kN]	F_{minor} [kN]	M_{major} [kNm]	M_{minor} [kNm]	M_{tor} [kNm]
SUP A/1	0.0000	SC A/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-40.500	0.0	0.0	0.0
SUP A/2	0.0000	SC A/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/1	0.0000	SC B/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/2	0.0000	SC B/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	-25.000	0.0	0.0	0.0
SUP C/3	0.0000	SC C/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	-100.0	-75.0	7.0
SUP C/4	0.0000	SC C/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	25.000	-100.0	-75.0	0.0
SUP D/3	0.0000	SC D/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	7.0
SUP D/4	0.0000	SC D/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	0.0

The user also wants the forces rounded to the nearest 5kN and moments to the nearest 10kNm so inputs 5kN in the ‘Rounding Increment for Force’ and 10kNm in the ‘Rounding Increment for Moment’ under **Foundation Reactions**.



When the report is updated, any values exceeding the minimums are rounded up accordingly.

Foundation Reactions ✕

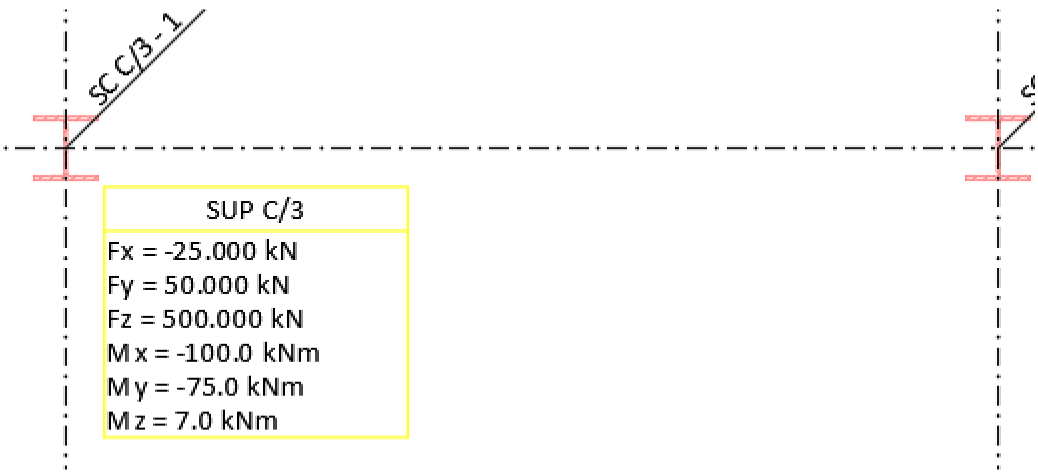
FoundationReactions

FoundationReactions, First-order linear, StrengthFactors

Supports

Support	Support rotation [°]	Column Ref.	Column rotation [°]	Combination	Reactions F_{vert} [kN]	F_{major} [kN]	F_{minor} [kN]	M_{major} [kNm]	M_{minor} [kNm]	M_{tor} [kNm]
SUP A/1	0.0000	SC A/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-50.000	0.0	0.0	0.0
SUP A/2	0.0000	SC A/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/1	0.0000	SC B/1 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	25.000	0.0	0.0	0.0
SUP B/2	0.0000	SC B/2 (UC 152x152x23)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	0.000	-25.000	0.0	0.0	0.0
SUP C/3	0.0000	SC C/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	-100.0	-75.0	7.0
SUP C/4	0.0000	SC C/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	25.000	-100.0	-75.0	0.0
SUP D/3	0.0000	SC D/3 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	7.0
SUP D/4	0.0000	SC D/4 (UC 203x203x46)	0.0000	1 STR_1 -1.35G+1.5Q+1.5RQ	500.000	50.000	-25.000	0.0	0.0	0.0

The changes are also reflected in the Foundation Reactions drawing, which now shows the same minimum values of design force and moment.



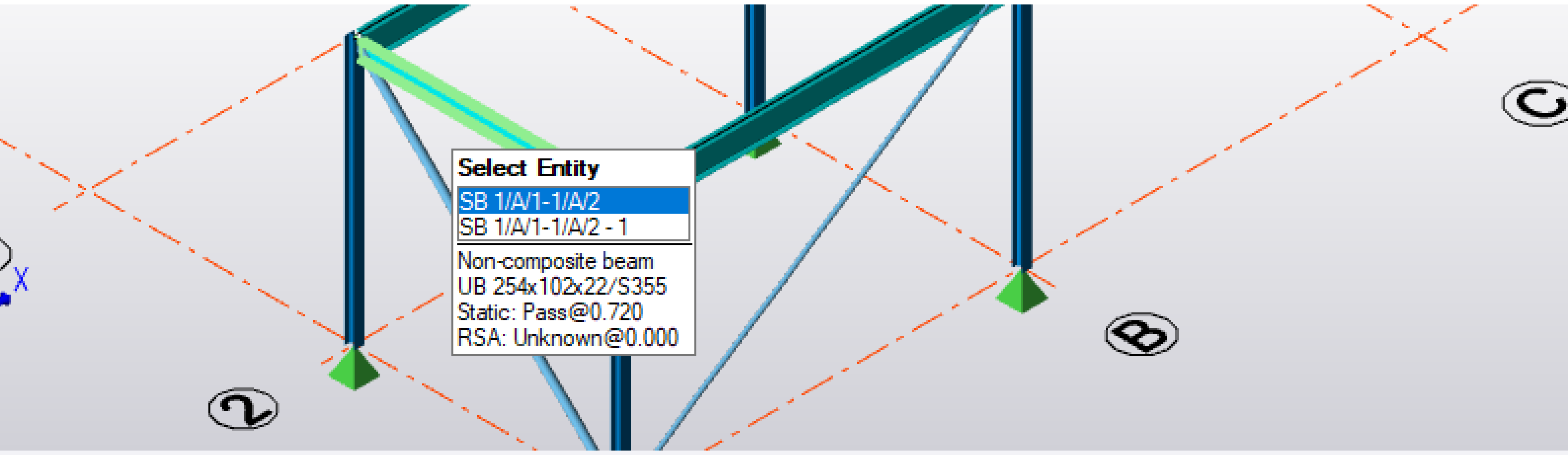
Connection resistance checks

Tekla Structural Designer performs connection resistance checks for simple beams and braces which use design forces from the analysis results.

The ‘Minimum Design Forces’ inputs under **Brace Force/Beam End Forces** allow the user to specify minimum values for these forces, and also apply rounding increments to them.

Consider a simple beam connection.

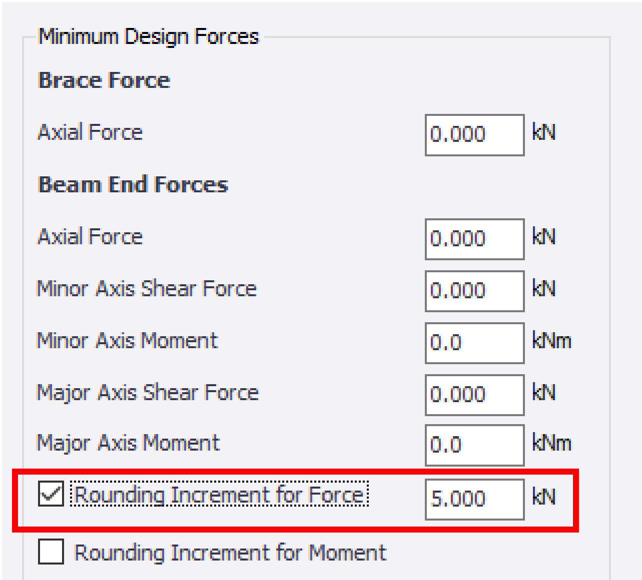
Review Data of Connection Resistance shows the following details.



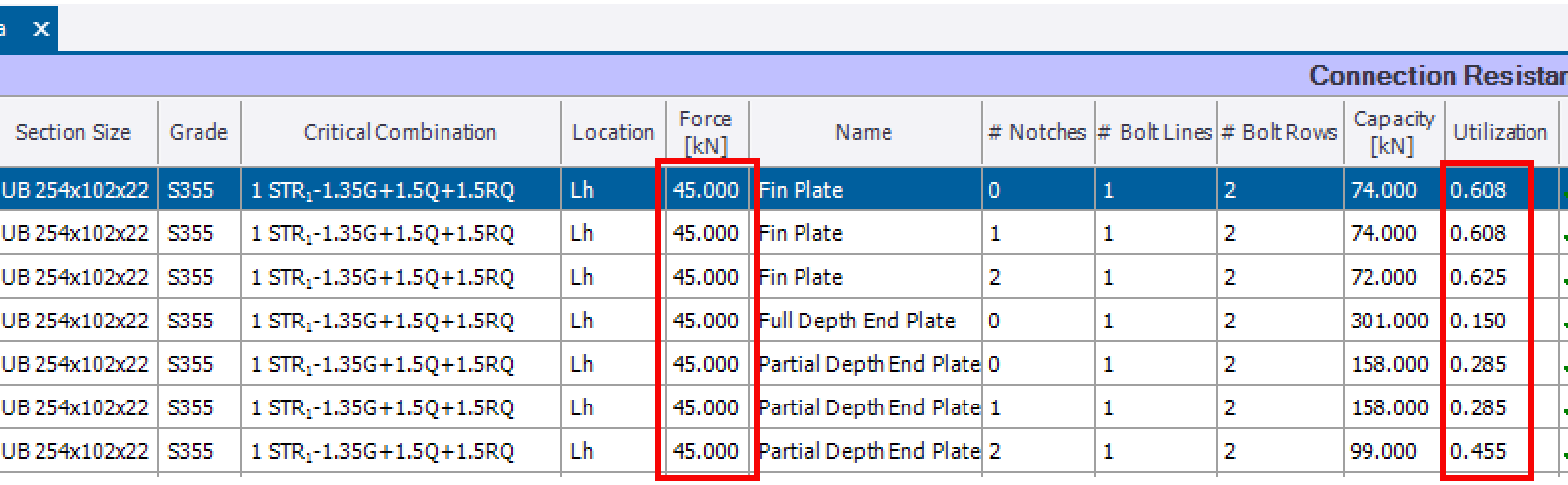
Connection Resistar

Section Size	Grade	Critical Combination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Fin Plate	0	1	2	74.000	0.575
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Fin Plate	1	1	2	74.000	0.575
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Fin Plate	2	1	2	72.000	0.591
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Full Depth End Plate	0	1	2	301.000	0.141
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Partial Depth End Plate	0	1	2	158.000	0.270
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Partial Depth End Plate	1	1	2	158.000	0.270
UB 254x102x22	S355	1 STR$_1$-1.35G+1.5Q+1.5RQ	Lh	42.582	Partial Depth End Plate	2	1	2	99.000	0.430

The user wants the forces rounded to the nearest 5kN so inputs 5kN in the ‘Rounding Increment for Force’ under **Beam End Forces**.



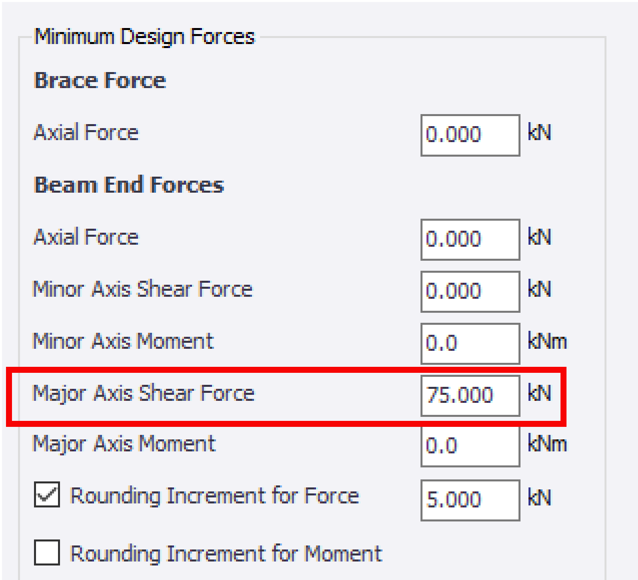
By closing and re-opening the Connection Resistance Review Data, the connection resistance check is updated and now uses the rounded value.



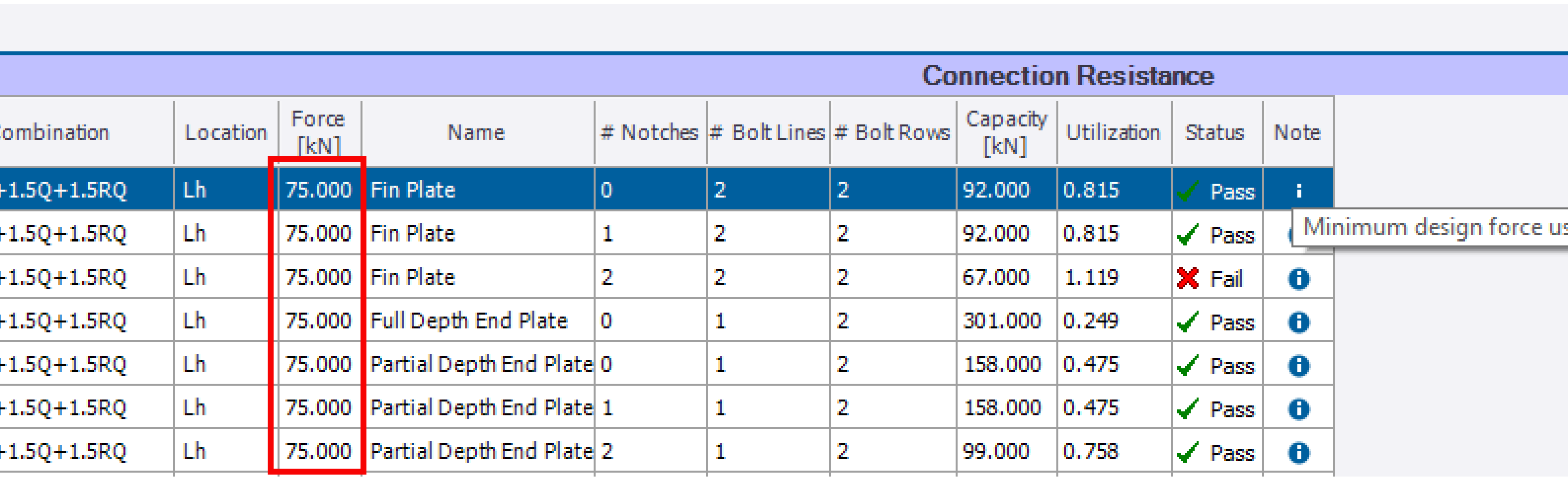
Connection Resistar

Section Size	Grade	Critical Combination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Fin Plate	0	1	2	74.000	0.608
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Fin Plate	1	1	2	74.000	0.608
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Fin Plate	2	1	2	72.000	0.625
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Full Depth End Plate	0	1	2	301.000	0.150
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Partial Depth End Plate	0	1	2	158.000	0.285
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Partial Depth End Plate	1	1	2	158.000	0.285
UB 254x102x22	S355	1 STR_1-1.35G+1.5Q+1.5RQ	Lh	45.000	Partial Depth End Plate	2	1	2	99.000	0.455

The user also wants a minimum design force of 75 kN for connection resistance check so inputs 75 kN in the ‘Major Axis Shear Force’ under **Beam End Forces**.



By closing and re-opening the Connection Resistance Review Data once again, the connection resistance check is updated to use the specified minimum design force.



Connection Resistance

ombination	Location	Force [kN]	Name	# Notches	# Bolt Lines	# Bolt Rows	Capacity [kN]	Utilization	Status	Note
+1.5Q+1.5RQ	Lh	75.000	Fin Plate	0	2	2	92.000	0.815	Pass	i
+1.5Q+1.5RQ	Lh	75.000	Fin Plate	1	2	2	92.000	0.815	Pass	
+1.5Q+1.5RQ	Lh	75.000	Fin Plate	2	2	2	67.000	1.119	Fail	
+1.5Q+1.5RQ	Lh	75.000	Full Depth End Plate	0	1	2	301.000	0.249	Pass	
+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	0	1	2	158.000	0.475	Pass	
+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	1	1	2	158.000	0.475	Pass	
+1.5Q+1.5RQ	Lh	75.000	Partial Depth End Plate	2	1	2	99.000	0.758	Pass	

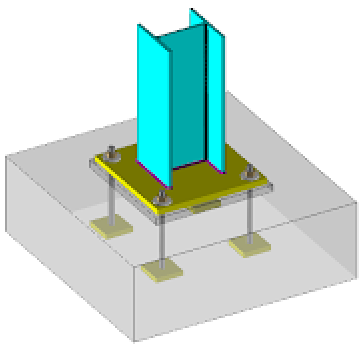
Because the minimum design force is being used this is indicated in the table under the 'Note' column.

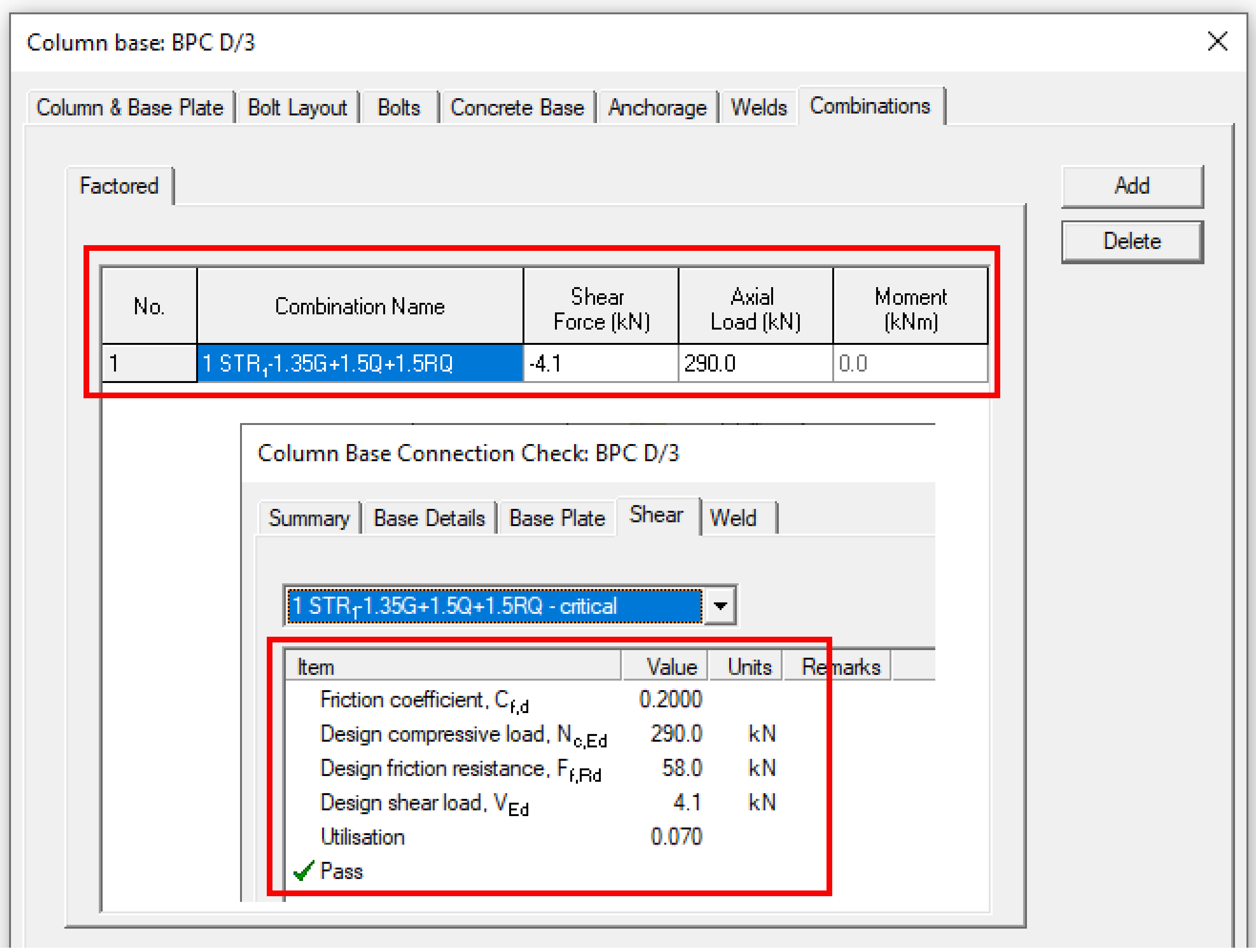
Exported connection forces

Forces exported to Tekla Connection Designer or IDEA StatiCa from Tekla Structural Designer are based on analysis results.

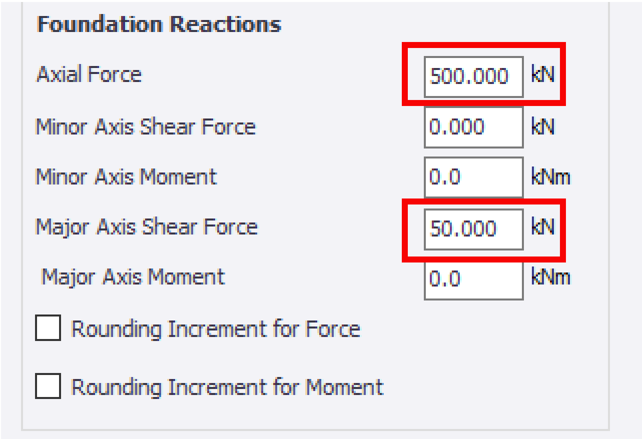
The ‘Minimum Design Forces’ inputs allow the user to specify minimum values for the exported forces, and also apply rounding increments to them.

Consider a base plate connection exported to Tekla Connection Designer and designed for axial load and major shear.





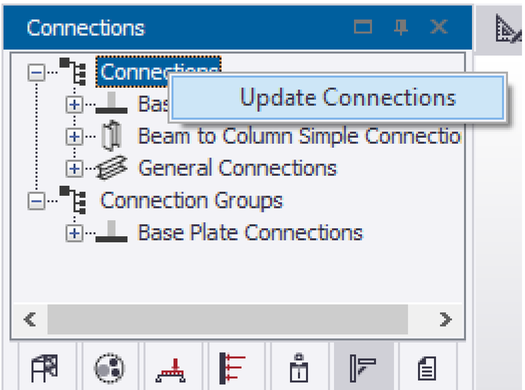
The user wants to design the base plate connection for a minimum shear of 50 kN and for a minimum axial load of 500 kN, so inputs these values under **Foundation Reactions**.



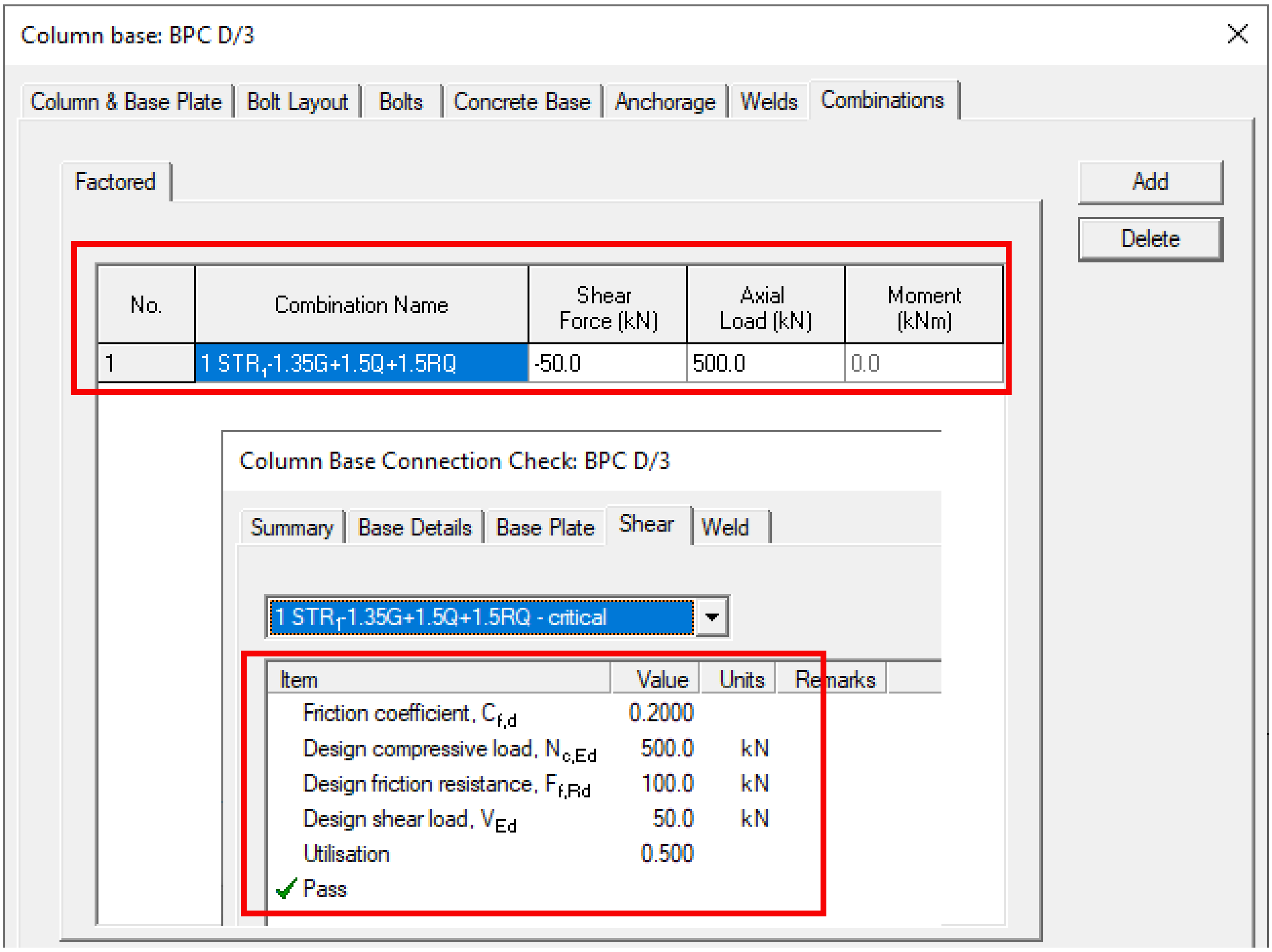
NOTE For other connection types the values under **Brace Forces/Beam End Forces** would be edited instead.

Rounding increments could also be applied if required. These would only apply to forces and moments exceeding the above minimums.

To apply the new minimums to the existing connections, it is necessary to click **Update Connections** from the Project Workspace.



The base plate connection is now designed for the minimum shear of 50 kN and for the minimum axial load of 500 kN specified by the user.



Design Settings - Design Groups and Autodesign

Design groups

Button, command, or option	Description
Members to design using groups	Allows you to select the member types for which you want to apply grouped design. When the members are grouped, only one member in the group is designed. This design is then copied to the remaining members in the group, so that they can be checked. Any failing member in the group is redesigned, and the process is repeated until all

Button, command, or option	Description
	members pass, or a design is not possible.

Autodesign settings

Reset Autodesign to off...	Allows you to control what happens to individual steel and concrete member auto design settings and isolated foundation auto design settings at the end of the design process. The options are:
	• **Always**: the auto design setting is automatically cleared at the end of the design process, so that each member is put into check mode.
	• **Never**: the autodesign settings is always retained as it is at the end of the design process.
	• **When check status is at worst**:
	    • **Pass**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass.
	    • **Warning**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass or Warning.
	    • **Fail**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, or Fail.
	    • **Invalid**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail or Invalid.

	• **Beyond Scope**: The auto design setting is only automatically cleared at the end of the design process for members with design status Pass, Warning, Fail, Invalid, or Beyond Scope. **TIP** The most practical use of **When check status is at worst** is to set it to **Pass** and start with all members in auto design mode. At the end of the first design run, passing members would be set to check mode, allowing you to focus on the remaining members still in auto design mode.

Design Settings - Design Warnings

Warnings used as guidance during the design of a structure to US seismic provisions are in effect by default. These can be de-activated as found useful from this page.

Design Warnings (AISC/ASC Head code only)

Button, command, or option	Description
Concrete Seismic Warnings	
Specified concrete grade is too low	• Compliance with ACI 318-08 and ACI 318-11 Section 21.1.4.2, “(..) f'_c, shall not be less than 3000 psi.“ • Applies to normal weight reinforced concrete columns and beams part of Special Moment Frames • Applies to normal weight reinforced concrete walls assigned as Special Reinforced Concrete Structural Walls • Prompts the designer to increase the concrete grade of the member
Specified longitudinal reinforcement grade too high	• Compliance with ACI 318-11 Section 21.1.5.2, “(..)

Button, command, or option	Description
	reinforcement resisting earth-quake induced flexure, axial force, or both, shall comply with ASTM A706, Grade 60.” • Applies to normal weight reinforced columns and beams part of Special Moment Frames • Applies to normal weight reinforced concrete walls assigned as Special Reinforced Concrete Structural Walls • Prompts the designer to use lower longitudinal reinforcement grades
Column at the end of the beam is not in the same seismic frame	• Used to validate if the frame of which the beam is part of is correctly set-up as a seismic frame • Applies to normal weight reinforced concrete beams part of a seismic force resisting system • Applies to both SRFS type and direction • Checks both ends of each beam span • Prompts the designer to set up a consistent SFRS type and direction between all members of the frame
Beam at the end of the column is not in the same seismic frame	• Used to validate if the frame of which the column is part off is correctly set-up as a seismic frame • Applies to normal weight reinforced concrete columns part of a seismic force resisting system • Checks the top of each column stack • Prompts the designer to set up a consistent SFRS type and direction between all members of the frame
Beam support region size is too small	• Compliance with ACI 318-08 and ACI 318-11 Section 21.3.4.2, “(..) hoops shall be provided over lengths not less than 2h measured

Button, command, or option	Description
	from the face of the supported member toward the midspan” • Applies to normal weight reinforced concrete beams part of Intermediate and Special Moment Frames • Comes into effect if the length of the end shear design regions is smaller than the requirement • Prompts the designer to review the shear design regions - Design options
High seismic to conventional demand on non-SFRS beams **High seismic to conventional demand ratio of non-SFRS columns** **High seismic to conventional demand of non-SFRS walls**	• Used to recognize when a member has a significant contribution to the building’s lateral force resistance but has not been assigned to the SFRS with the required ductility properties • Specifically apply to each member type • The highest non-seismic design ratio from bending and shear design is obtained • The highest seismic design ratio from bending and shear design is obtained • Comes into effect when the ratio between seismic and non-seismic values is higher than a given threshold (default = 1.0) • The maximum seismic/non-seismic ratio threshold that triggers the warning is set through the **Seismic to conventional minimum demand ratio** field described below • Prompts the designer to either include the member in the SFRS or review the structure layout to reduce the contribution of the member to the lateral force resistance

Button, command, or option	Description
Non-SFRS design requirements applicable in SDC D-F	• Compliance with ACI 318-08 and ACI 318-11 Sections 21.13.1, "Requirements of section 21.13 apply to members not designated as part of the seismic-force-resisting system in structures assigned to SDC D, E and F" • Applies to normal weight reinforced concrete beams, column and walls not included in the SFRS • Comes into effect in Seismic Design Category, SDC D or above • Prompts the designer to apply code prescribed requirements on top of current design
Steel Static Warnings	
Additional notional loads required when alpha Pr/Py > 0.5, C2.3	• Compliance with 7.3 (3) of AISC 360-05 and C2.3 of AISC 360-10 requires the stiffness of certain members to be reduced. When the ratio of the required axial compressive force (Pr) and the axial yield strength (Py = Fy Ag) is > 0.5 then this stiffness must be reduced further • As an alternative which is the approach adopted in Tekla Structural Designer, an additional notional load of 0.001 Yi can be applied instead • When the program identifies that the ratio of Pr/Py exceeds 0.5 a warning is given to alert the designer • If this additional notional load has been dealt with by the designer then this warning is no longer relevant and this option allows you to switch off the warning
Concrete Seismic	
Seismic to conventional minimum demand ratio	• Sets the maximum seismic/non-seismic ratio threshold, which is

Button, command, or option	Description
	used to trigger the **High seismic to conventional demand**warnings listed above • Default: 1.000

Design Settings - Sway & Drift Checks

Sway & drift check settings

Button, command, or option	Description
Merge short stacks	With this setting off the Sway/Drift, Wind Drift & Seismic Drift checks are performed for all column stacks and wall panels irrespective of their stack lengths. With this setting on you can specify a minimum stack length. **NOTE** The ‘Merge short stacks’ option does not apply to columns/walls where you have elected to merge stacks manually (via the column/wall properties). The following logic is applied to merging stacks when the setting is on: 1. Start at the topmost stack of the column/wall. 2. If the stack length is less than the merge limit then merge with the stack below. 3. Check again to see if then new length exceeds the merge limit, if not then merge again with the next stack below. 4. Repeat step 3 as required until the stack length exceeds the merge limit. 5. For subsequent stacks repeat steps 2 and 3 as required until the length exceeds the merge limit.

Button, command, or option	Description
	6. When the bottommost stack is reached, if this needs to be merged, merge with the stack above. **NOTE** If a single stack column or single panel wall length is less than the merge short stacks limit it is not considered for sway/drift, wind drift, seismic drift checks.
Check wind cases only	Allows you to set that the wind drift check performed on all columns only considers the effect of the wind loadcases in each wind combination. Clear the setting to consider the effects of all loadcases in wind combinations, such as drift induced by gravity loads. For more information, see: Consider wind cases only for the wind drift check (page 683)
Wind Drift Limit	The limit applied to all columns and walls. **NOTE** This can be overridden for specific members as required in the member properties. **NOTE** A re-analysis is required if the limit is changed. For more information, see: Set the wind drift limit (page 682)
Check for Resultant Wind Drift	The wind drift check can either be performed in two directions (Dir 1 & 2), or for the resultant wind drift direction only. For new models the default is to check for the resultant direction only. For more information, see: Choose resultant or directional wind drift checks (page 682)

Design Settings - Fire check

Fire check (Eurocode Head code only)

Button, command, or option	Description
Time interval for critical temperature iteration if 'Unprotected'	default 5 sec
Time interval for critical temperature iteration if 'Protected'	default 30 sec

Design Settings - Timber

The settings allow you to apply your own set of timber design defaults that broadly apply to all members. These are incorporated into Tedds calculations as default variables, which can then be edited within the calculation if required.

Timber (US Headcode)

Button, command, or option	Description
Service condition	NDS: Service condition • Dry (default) • Wet
Temperature range	• up to 100 degF (default) • 100 degF to 125 degF • 125 degF to 150 degF
Is section incised	• yes • no (default) The incisions made when applying a preservative treatment result in a loss of area and section modulus. When you indicate that the section is incised, appropriate incising factors are applied in the Tedds calculation.

Timber (Eurocode)

Button, command, or option	Description
Service class	• 1 (default)

Button, command, or option	Description
	• 2 • 3
Individual grade stamp	• yes • no (default) Depending on the National Annex being worked to, the partial factor for material properties can be reduced when timber has been individually marked.

Slab deflection settings

The **Slab Deflection** page and its subpages allow you to control the slab deflection defaults applied:

- In the **current** project - when accessed from the Slab Deflection Settings dialog (page 1225) on the **Slab Deflection** tab.
- In **new** projects - when accessed from the Settings dialog (page 1209) on the **Home** tab.

Slab deflection settings when accessed from the Slab Deflection Check Catalogue

Button, command, or option	Description
New Load Event Defaults subpage	
Start time offset	Allows you to define a default start time offset value for new event load start times. The value of a new event load start time is always the previous event start time + the default start time offset.
Construction load	Allows you to define a default value for new construction loads. **NOTE** The default construction load value depends on the head code that you are using.
New Check Defaults subpage	

Button, command, or option	Description
Deflection limit, L /	Allows you to define a default deflection limit to new checks added to the When a new check is added to the Slab Deflection Check Catalogue it initially defaults to the deflection limit set here.
Aging, Creep & Shrinkage subpage	
Allowance for shrinkage effects in total deflection	Allows you to define an amplification base factor for shrinkage. **NOTE** The value must be within the range from 0.1 to 0.9.
Aging Coefficient	Allows you to select between 2 creep/aging methods: **User defined** and **Automatic**. **NOTE** The **Aging Coefficient** option is only available for the US head code.
Modification Factors subpage	Allows you to adjust the properties used for the different element types in the iterative cracked section analysis.
Iteractive Cracked Section Analysis subpage	
Global Convergence Criteria	Allows you to modify the following options: • **Maximum number of iterations**: The number of iterations to perform. Default = 200. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.00100. • **Relative**: Default: ON. **TIP** To resolve global convergence failures:

Button, command, or option	Description
	• Increase the value in **Maximum number of iterations**. • Decrease the value in **Tolerance**.
Local Convergence Criteria	Allows you to modify the following options: • **Maximum number of iterations**: The number of iterations to perform. Default = 500. • **Tolerance**: At the end of each iteration, values are checked against the previous iteration results. If the difference is less than the tolerance determined here, the result is converged, and the analysis is complete. Default = 0.000001. • **Smoothing Parameter**: A property only available for the Eurocode head code. Default = 0.005. **TIP** To resolve local convergence failures: • Increase the value in **Maximum number of iterations**. • Decrease the value in **Tolerance**. • If you are using the Eurocode head code, increase the value in **Smoothing Parameter**.

Slab deflection settings in the Settings dialog

Button, command, or option	Description
New Load Event Defaults subpage	
Start time offset	Allows you to define a default start time offset value for new event load

Button, command, or option	Description
	start times. The value of a new event load start time is always the previous event start time + the default start time offset.
Construction load	Allows you to define a default value for new construction loads. **NOTE** The default construction load value depends on the head code that you are using.
New Check Defaults subpage	
Deflection limit, L /	Allows you to define a default deflection limit to new checks added to the When a new check is added to the Slab Deflection Check Catalogue it initially defaults to the deflection limit set here.
Aging, Creep & Shrinkage subpage	
Allowance for shrinkage effects in total deflection	Allows you to define an amplification base factor for shrinkage. **NOTE** The value must be within the range from 0.1 to 0.9.
Aging Coefficient	Allows you to select between 2 creep/aging methods: **User defined** and **Automatic**. **NOTE** The **Aging Coefficient** option is only available for the US head code.
Load Event Sequence subpage	Allows you to modify the global event sequence according to your needs. **TIP** The global event sequence provides the basis for the model event sequence that you can modify, whereas the model event sequence provides the basis for a submodel event sequence that you can modify.

Drawing settings

The **Drawings** page and its subpages allow you to adjust different drawing settings:

- In the **current** project - when accessed from the Drawing Settings dialog (page 1180) on the **Draw** tab.
- In **new** projects - when accessed from the Settings dialog (page 1209) on the **Home** tab.

Export preferences

NOTE The **Export Preferences** subpage is only available when drawing settings are accessed from the Settings dialog (page 1209)

Setting	Description
Drawing Variant list	Allows you to select the drawing category whose export preferences you want to modify.
Minimum Text Block Spacing	Allows you to define a minimum value between independent drawing blocks. **NOTE** If the value is too great, text labels can end up far from the objects to which they refer.
Available Scales list	Displays the existing drawing scales. **NOTE** You can add user-defined scales in the list by typing the scale in the **Scale** field and clicking the **Add** button, or delete them by selecting the scale and clicking the **Remove** button.
Scale	Allows you to create a user-defined drawing scale by typing it in the field.
Name	Allows you to specify a name for a user-defined drawing scale.

Layer configurations

Setting	Description
Drawing Variant list	Allows you to select the drawing category whose layer configurations you want to modify. **NOTE** Use the subpages of the **Layer Configurations** subpage to modify the required drawing type.
Available Configurations list	Displays the available layer configurations and allows you to select or modify them. You can use the following buttons to modify the available layer configurations: • **Add**: creates an empty layer configuration that you can modify in the **Active Configuration** section. • **Add copy...**: allows you to copy a drawing item to create a new layer configuration. • **Remove**: deletes the selected layer configuration.
Active Configuration	Displays the name and layers in the selected layer configuration.
Name	Displays and allows you to modify the name of the selected layer configuration.
Layers list	Displays and allows you to select or clear the layers that are included in the selected layer configuration. **TIP** If you access the drawing settings from the **Draw** tab and modify them, consider saving the changes to a settings set, so that you can use them in later projects as well.

Layer styles

Setting	Description
Drawing Variant list	Allows you to select the drawing category whose layer styles you want to modify. **NOTE** Use the subpages of the **Layer Styles** subpage to modify the required drawing type.
Available Styles list	Displays the available layer styles and allows you to select or modify them. You can use the following buttons to modify the available layer styles: • **Add**: creates an empty layer style that you can modify in the **Active Style** section. • **Add copy...**: allows you to copy a drawing item to create a new layer style. • **Remove**: deletes the selected layer style.
Active Style	Displays the name and layers in the selected layer style.
Name	Displays and allows you to modify the name of the selected layer style.
Apply to All...	This button opens a dialog for rapidly applying a single color, report color, line type, font, or font size to all layers in the active layer style.
Layers list	Displays and allows you modify the appearance of layers that are included in the active layer style.
Description	The layer description is fixed and can't be changed.
Name	The layer name as it will appear in the drawing can be edited if required.
Is Merged, **Merged with**	By checking the box you are able to merge two or more layers together. All objects in the merged layer are drawn in the **Merged with** layer, adopting its line type, font and font size.

Setting	Description
Color	The colors specified here are used for the different layers in the dxf drawing output
Report Color	**Member Reports** have the facility to include a member drawing along with the calculations and diagrams etc. When a member report is printed the drawings adopt the report colors that have been specified here.
Line Type	Different line types can be specified for the different drawing layers as required.
Font	Different fonts can be specified for the different drawing layers as required.
Font Size	Different font sizes can be specified for the different drawing layers as required.

Planar drawing options

. . . . General (all drawing variants)

Setting	Description
Hatching	Allows you to select how columns and walls are hatched in planar drawings. You can select or clear the following options: • **Show columns and walls above the level as hatched**: The columns and walls that continue above the current level are hatched. • **Show transfer columns and walls as cross hatched**: Transfer columns and walls are cross hatched.
Force and Moment Values	**NOTE** Applies to the following drawing variants: • Foundation Reactions • Beam End Forces

Setting	Description
	• Column Splice Loads Allows you to select which force and moment values are displayed in planar drawings. The options are: • **Strength factors**: Displays the strength factors in planar drawings. • **Service factors**: Displays the service factors in planar drawings. **Additionally factor by** allows you to specify a value by which the strength or service factors are factored in the selected drawing category.
Moment Values	**NOTE** Applies to the following drawing variants: • Foundation Reactions • Beam End Forces • Column Splice Loads Allows you to select which moment values are displayed in planar drawings. The options are: • **None**: Does not display any moment values. • **Only greater than**: Moment values are only displayed if their value is greater than the value specified here. • **All**: Displays all moment values, even if zero.
Shear Force Values	**NOTE** Foundation Reactions drawing variant only. Allows you to select which shear values are displayed in Foundation Reactions drawings. The options are: • **None**: Does not display any shear values.

Setting	Description
	• **Only greater than**: Shear values are only displayed if their value is greater than the value specified here. • **All**: Displays all shear values, even if zero.
Axial Force Values	**NOTE** Beam End Forces drawing variant only. Allows you to select which axial force values are displayed in planar drawings. The options are: • **None**: Does not display any axial force values. • **Only greater than**: Axial force values are only displayed if their value is greater than the value specified here. • **All**: Displays all axial force values, even if zero.
Display End Forces for	**NOTE** Beam End Forces drawing variant only. Allows you to select the members for which end forces are displayed.
Enveloped Reaction Values	**NOTE** Beam End Forces drawing variant only. **Include coincident forces**: when this option is selected, further options are presented allowing you to choose the max/min values and coincident forces to display when the beam end forces drawing is created for a loading envelope.

. . . . Beams (all drawing variants)

Setting	Description
Grouped Beam Labelling	Allow you to modify the labeling of beams when the beams have been

Setting	Description
	designed using groups. You can select or clear the following options: • **Use detail group name**: Select the option to use the detail group name in the beam label, or clear the option to use the design group name in the beam label. • **Include the beam name**: Select the option to include the beam name in the beam label for grouped beams.
Concrete Beam Labelling Position	Allows you to set the position of the beam label in relation to the beam in planar drawings. The options are: • **Above** • **Inside** • **Below**
Beam Mark Position	Allows you to adjust the appearance and position of the beam mark in relation to the brace in planar drawings. To display beam marks, select **Show beam mark**. The position options are: • **Above** • **Below**
Beam Attributes Position	Allows you to adjust the appearance and position of beam attributes in relation to the beam in planar drawings. To display beam attributes, select **Show beam attributes**. The position options are: • **Above** • **Below**
Beam Attributes	Allows you to select which beam attributes are displayed in planar drawings and how they are displayed. You can select or clear the following options: • **Show beam size in parentheses**: Places brackets around the beam size in the beam label.

Setting	Description
	• **Grade**: Displays the beam grade in planar drawings. • **Camber**: Displays the camber in steel beams. To modify the camber prefix, type the desired value in the **Camber prefix** field. • **Composite properties**: Displays composite beam properties. To change the separators inside which the number of studs is displayed, select the desired the **Stud separator** list. • **Transverse reinforcement**: Displays transverse reinforcement in planar drawings.

. . . . Braces (all drawing variants)

Setting	Description
Brace Mark Position	Allows you to adjust the appearance and position of the brace mark in relation to the brace in planar drawings. To display brace marks, select **Show brace mark**. The position options are: • **Above** • **Below**
Brace Attributes Position	Allows you to adjust the appearance and position of brace attributes in relation to the brace in planar drawings. To display brace attributes, select **Show brace attributes**. The position options are: • **Above** • **Below**
Brace Attributes	Allows you to select whether brace grades are displayed in planar drawings.

. . . . Columns (all drawing variants)

<table>
<tr><th>Setting</th><th>Description</th></tr>
<tr><td>Grouped Column Labelling</td><td>Allow you to modify the labeling of columns when the columns have been designed using groups. You can select or clear the following options:
Use detail group name: Select the option to use the detail group name in the column label, or clear the option to use the design group name in the label.
Include the column name: Select the option to include the column name in the column label for grouped columns.</td></tr>
<tr><td>Column Mark Position</td><td>Allows you to adjust the appearance and position of the column mark in relation to the brace in planar drawings. To display column marks, select Show column mark. The position options are:
Above
Below</td></tr>
<tr><td>Column Attributes Position</td><td>Allows you to adjust the appearance and position of the column attribute in relation to the brace in planar drawings. To display column attributes, select Show column attributes. The position options are:
For position in elevation:
Above
Below
For position in cross-section:
To the right of the mark
Below the mark</td></tr>
<tr><td>Column Attributes</td><td>Allows you to select which column attributes are displayed in planar drawings and how they are displayed. You can select or clear the following options:
Show column size in parentheses: Places brackets</td></tr>
</table>

Setting	Description
	around the column size in the column label. • **Grade**: Displays the column grade in planar drawings.
2x scale for steel columns	Allows you to double the scale of steel columns to simplify viewing the columns and their orientation.

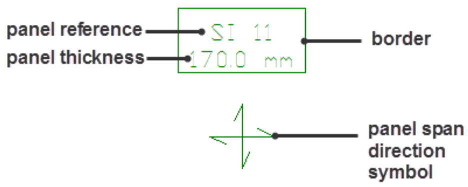
*. . . . **Walls (all drawing variants)***

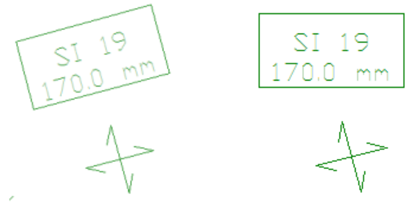
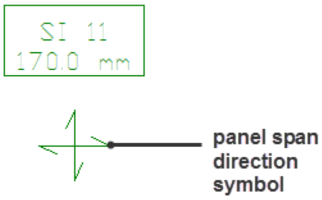
Setting	Description
Wall Labelling position	Allows you to set the position of the wall label in relation to the wall in planar drawings. The options are: • **Above** • **Inside** • **Below**
Show wall size in parentheses	Allows you to select whether brackets are placed around the wall size in the wall label.

Wall Reactions	Allows you to select whether or not to include distributed vertical reaction values of walls in the foundation reactions planar drawing. These are available in two different formats:
	• **Show distributed wall reactions table**: a table with the distributed wall reaction end values is included in the drawing,
	• **Show vertical reaction values**: values are added for each wall in the planar drawing,
	    • **Distributed (Maximum)**: Maximum absolute values of the vertical distributed reaction retaining sign,
	    • **Distributed (Average)**: Average value of the vertical distributed reaction along the length of the bottom analytical chord of the wall,

Setting	Description
	• **Distributed (Extremes)**: Extreme values at each end of the wall bottom analytical chord, • **Total**: Single integrated total vertical reaction of the wall.

. . . . Slabs/Mats (all drawing variants)

Setting	Description
Panel Labelling	Allows you to select which panel properties are included in panel labels and how panel labels are displayed in planar drawings.  You can select or clear the following options: • **Include panel reference**: Allows you to include the panel reference in the panel label. • **Include panel thickness**: Allows you to include the panel thickness in the panel label. • **Include surface offset (if non-zero)**: Allows you to include any surface offset that has been applied to the panel in the panel label. • **Include border around label**: Allows you to add a border around the label. • **Align label to panel reinforcement**: Allows you to select how the panel label is aligned to the panel span direction. Select the option to achieve the result displayed on the left, and clear the option to

Setting	Description
	achieve the result displayed on the right. 
Include panel span direction symbol	Allows you to select whether a direction symbol is displayed in the slab or mat geometry. 

. . . . Loads (Loading Plan drawing variant only)

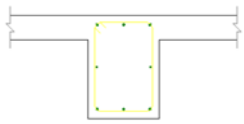
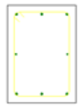
Setting	Description
Display size	Allows you to adjust the width and height of line, udl, and vdl loads and the size of point loads in planar drawings. The different options that you can adjust are: • **Width of line/UDL/VDL loads on plan**: Line, UDL, and VDL loads are drawn as hatched rectangles of fixed width when drawn in plan. The option allows you to adjust the width of the hatched rectangle. • **Height of line/UDL/VDL loads on plan**: Line, UDL, and VDL loads are drawn as hatched rectangles of fixed width when drawn in plan. The option allows you to adjust the height of the hatched rectangle. • **Point load marker size**: Allows you to adjust the size of point loads in planar drawings.

Setting	Description
Show dimensions for panel loads	Allows you to include the dimensions of panel loads in planar drawings.
Include dimensions to reference points	Allows you to include the dimensions from panel loads to their reference points in planar drawings.
Show dimensions for member loads	Allows you to include the dimensions of member loads in planar drawings.

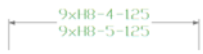
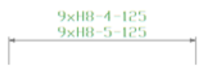
Member detail options

. . . . Beam Detail

Setting	Description
Content subpage	
Grouped Beams tab	
Show number of beams in group	If the beams in the model have been arranged into detailing groups, selecting the option causes the detailing number to be used as the member label instead of the beam reference. If detailing groups have not been used, the beam reference is always used as the member label.
Levels tab	
Show span levels	Allows you to display the span levels on the elevation.
Cross-sections tab	
Spans list	Allows you to select which cross-sections to display. You can decide to display no cross-sections, only the cross-sections for first spans of multi-span beams, or all spans.
Positions list	If cross-sections are displayed, the list allows you to select where they are positioned. You can decide to only position cross-sections for spans, for supports, or for supports and spans.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **None** • **Standard**

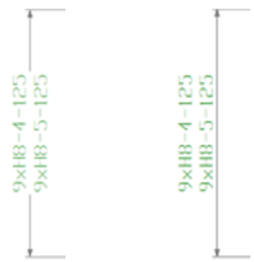
Setting	Description
	• **IStructE**
Display bar marks	If the cross-section annotation is set to **Standard** or **IStructE**, you can select the option to display bar marks in cross-section labels.
Show slab lines in section	Select the option to display slab lines in sections, as shown in the first image:  Clear the option to not display slab lines in sections, as shown in the second image: 
Bar Labels tab	
Show bar marks in elevation	Select the option to include bar marks in the bar labels on the elevation. Bar marks will also be displayed on the cross-sections, provided that the **Display bar marks** option is also checked. Clear the option to not display bar marks on cross-sections irrespective of the **Display bar marks** option.
Show steel bar layer information	Allows you to display steel bar layer information (B1, B2, T1, T2, and so on).
Dimensions tab	
Laps list	Allows you to select whether existing laps are dimensioned, not dimensioned, or the dimension is replaced by a standard label (TL).
Anchorage lengths list	Allows you to select whether any required anchorage lengths are dimensioned, not dimensioned, or the dimension is replaced by a standard label (AL).

Setting	Description
Axes list	Allows you to select whether axes are not displayed, are displayed above the beam with dimensions, or are displayed below the beam with dimensions.
Additional bottom span bar positioning dimensions	Allows you to dimension any existing optional second span bars from the face of the support.
Support region length	If different stirrup regions have been used along a beam span, the option allows you to dimension the length of the support regions on the elevation.
First and last stirrups	Allows you to add dimensions from the face of the supports to the first and last stirrups.
Support columns and clear spans	Allows you to add dimensions showing the width of each supporting column and the clear beam span between supports.
Beam section	Allows you to dimension the beam depth and width on the cross-section.
Slabs in beam sections	Allows you to dimension the slab depth on the cross-section.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in beam detail drawings.
Style subpage	
Beam Labels tab	
Print beam labels below the detail	Allows you to display the beam label centrally below each span. Clear the option to position the label immediately above each span.
Underline beam labels	Allows you to to underline the beam label on the elevation.
Label every span of multi-span beams	Allows you to display a label for each span in multi-span beams.
Cross-sections tab	
Section label style list	Allows you to select the label naming style that is applied to cross-sections.
Restart section labels in each beam line	Allows you to restart the section labels for each line when multiple

Setting	Description
	beam lines are displayed on the same drawing sheet.
Add beam name label as prefix to section labels	Allows you to prefix each section label with the beam name.
Place cross-sections underneath the elevation	Allows you to to position cross-sections under the elevation.
Longitudinal Bars tab	
Draw bobs as shifted	Allows you to apply a slight offset to bobbed bars in drawings, so that the bobbed bars do not overlap each other in drawings an therefore, the drawings are easier to read.
Display only a single side bar in detail	Allows you to draw only a single side bar is drawn full length when multiple side bars are required in each face. Clear the option to draw all side bars full length.
Stirrups tab	
Draw stirrup labels in line	Allows you to display stirrup labels in line on the elevation, as shown in the first image:  Clear the option to display stirrup labels above the line, as shown in the second image: 
Print stirrup labels inside beam	Allows you to display the stirrup labels inside the beam. Clear the option to display the stirrup labels below the beam.
Stirrup label distance from bottom edge [n] % of the beam depth	Allows you to control the vertical position of stirrup labels that are positioned inside the beam.
Dimensions tab	
Lap and anchorage rounding increment	Allows you to control the rounding increment of lap and anchorage dimensions.

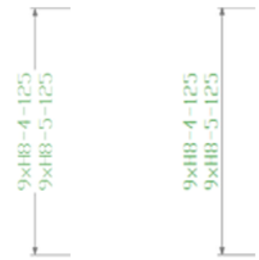
. . . . Column Detail

Setting	Description
Content subpage	
Grouped Columns tab	
Show number of columns in group	Allows you to display the number of columns in a column group.
Levels tab	
Show levels	Allows you to label the construction levels.
Cross-sections tab	
Show cross-sections	Allows you to display cross-sections through each stack.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **None** • **Standard** • **IStructE**
Display bar marks	If the cross-section annotation is set to **Standard** or **IStructE**, you can select the option to display bar marks in cross-section labels.
Bar Labels tab	
Show bar marks in elevation	Select the option to include bar marks in the bar labels on the elevation. Bar marks will also be displayed on the cross-sections, provided that the **Display bar marks** option is also checked. Clear the option to not display bar marks on cross-sections irrespective of the **Display bar marks** option.
Dimensions tab	
Laps list	Allows you to select whether existing laps are dimensioned, not dimensioned, or the dimension is replaced by a standard label (TL or CL).
Support region length	Allows you to dimension the support regions on the elevation.
Levels	Allows you to add dimensions between levels.

Setting	Description
Grid line offsets	Allows you to add dimensions from the grid to the column face on the elevation.
Connecting elements and clear heights	Allows you to add dimensions connecting elements and clear heights.
Column section	Allows you to add column dimensions to the column section.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in column detail drawings.
Style subpage	
Column Labels tab	
Underline column labels	Allows you to to underline the column label on the elevation.
Cross-sections tab	
Section label style list	Allows you to select the label naming style that is applied to cross-sections.
Restart section labels in each column line	Allows you to restart the section labels for each line when multiple column lines are displayed on the same drawing sheet.
Add column name label as prefix to section labels	Allows you to prefix each section label with the column name.
Ties tab	
Draw tie labels in line	Allows you to display tie labels in line on the elevation, as shown in the image on the left. Clear the option to display tie labels above the line, as shown in the image on the right. 
Dimensions tab	
Lap dimension rounding increment	Allows you to control the rounding increment of lap dimensions.

. . . . Wall Detail

Setting	Description
Content subpage	
Levels tab	
Show levels	Allows you to label the construction levels.
Cross-sections tab	
Show cross-sections	Allows you to display cross-sections through each stack.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **None** • **Standard** • **IStructE**
Display bar marks	If the cross-section annotation is set to **Standard** or **IStructE**, you can select the option to display bar marks in cross-section labels.
Bar Labels tab	
Show bar marks in elevation	Select the option to include bar marks in the bar labels on the elevation. Bar marks will also be displayed on the cross-sections, provided that the **Display bar marks** option is also checked. Clear the option to not display bar marks on cross-sections irrespective of the **Display bar marks** option.
Dimensions tab	
Laps list	Allows you to select whether existing laps are dimensioned, not dimensioned, or the dimension is replaced by a standard label (TL or CL).
Support region length	Allows you to dimension the support regions on the elevation.
Levels	Allows you to add dimensions between levels.
Grid line offsets	Allows you to add dimensions from the grid to the column face on the elevation.

Setting	Description
Wall section	Allows you to add wall dimensions to the wall section.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in wall detail drawings.
Style subpage	
Wall Labels tab	
Underline wall labels	Allows you to underline the wall label on the elevation.
Cross-sections tab	
Section label style list	Allows you to select the label naming style that is applied to cross-sections.
Add wall name label as prefix to section labels	Allows you to prefix each section label with the wall name.
Don't label typical panel bars	Allows you to not label typical bars in cross-sections. Clear the option to label every bar in cross-sections.
Horizontal Bars and Ties tab	
Draw labels in line	Allows you to display tie labels in line on the elevation, as shown in the image on the left. Clear the option to display tie labels above the line, as shown in the image on the right. 
Label bars inside panel	Allows you to label bars inside the wall panel.
Dimensions tab	
Lap dimension rounding increment	Allows you to control the rounding increment of lap dimensions.

Member schedule options

. . . . General

Setting	Description
NOTE The settings that you see on the **General** tab depend on which drawing category subpage you have selected in the left side pane.	
Item list	Allows you to select an item to view the text label that will applied to it in the concrete beam schedule drawing.
Text	Allows you to modify the text displayed in the beam schedule for the item selected in the **Item** list.
Size Column Format	Allows you to select whether to display the width of the beam or its height first in the size column.
Use single column for size	Allows you to display both the width and height in a single column.
Use single column for bottom bars	Allows you to display the bottom bars in a single column
Omit top middle bars column	Allows you to omit the top middle bars from the schedule drawing.
Use single column for stirrups	Allows you to display the stirrups in a single column.
Show design group name	Allows you to display only the design group name in the mark column.
Show grouped column number	Allows you to display column groups.
Include starter bars	Allows you to include starter bars in member schedule drawings.
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in member schedule drawings.
Bar annotation list	Allows you to select the cross-section annotation. The options are: • **Standard** • **IStructE**
Display bar marks	Allows you to display bar marks in cross-section labels.
Show outline of stack below	Allows you to display the outline of the column stack below.

Setting	Description
Dimension column section	Allows you to add column dimensions to the column section.
Dimension levels	Allows you to add dimensions between levels.
Show outline of panel below	Allows you to display the outline of the wall panel below.
Dimension wall section	Allows you to add wall dimensions to the wall section.

. . . . Bar Key

Setting	Description
Reference column	Lists every reference that can potentially appear in the bar bending details table in concrete beam schedule drawings. **TIP** Hover the mouse pointer over a reference to see the bar and its associated note.
	Allows you to use a custom name for the selected reference.
	Allows you to specify the custom name that replaces the original reference in the bar bending details table if the

Slab and mat layout options

. . . . General

Setting	Description
Layout subpage	
General tab	
Show columns and walls above the level as hatched	Allows you to hatch columns and walls that continue above the current level.
Show transfer columns and walls as cross hatched	Allows you to cross-hatch transfer columns and walls.

. . . . *Beams*

Setting	Description
Use detail group name	Allows you to use the detail group name in the beam label. Clear the option to use the design group name in the beam label instead.
Include the beam name	Allows you to include the beam name in the label for grouped beams.
Beam Labelling position	Allows you to set the position of the beam label in relation to the beam in slab and mat layout drawings. The options are: • **Above** • **Inside** • **Below**
Show beam mark	Allows you to display beam marks in slab and mat layout drawings. The position options for beam marks are: • **Above** • **Below**
Show beam attributes	Allows you to display beam attributes in slab and mat layout drawings. The position options for beam attributes are: • **Above** • **Below**
Show beam size in parentheses	Allows you to place brackets around the beam size in the beam label.
Grade	Allows you to display the beam grade in slab and mat layout drawings.

. . . . *Braces*

Setting	Description
Show brace mark	Allows you to display brace marks in slab and mat layout drawings. The position options for brace marks are: • **Above** • **Below**

Setting	Description
Show brace attributes	Allows you to display brace attributes in slab and mat layout drawings. The position options for brace attributes are: • **Above** • **Below**
Brace Attributes	Allows you to select whether brace grades are displayed in slab and mat layout drawings.

. . . . Columns

Setting	Description
Use detail group name	Allows you to use the detail group name in the column label. Clear the option to use the design group name in the column label instead.
Include the column name	Allows you to include the column name in the label for grouped columns.
Show column mark	Allows you to display column marks in slab and mat layout drawings. The position options for column marks are: • **Above** • **Below**

Show column attributes	Allows you to display column attributes in slab and mat layout drawings. The position options for column attributes are:
	• For position in elevation:
	  • **Above**
	  • **Below**
	• For position in cross-section:
	  • **To the right of the mark**
	  • **Below the mark**

Show column size in parentheses	Allows you to place brackets around the column size in the column label.

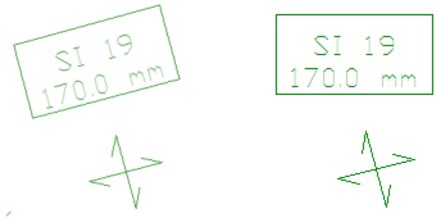
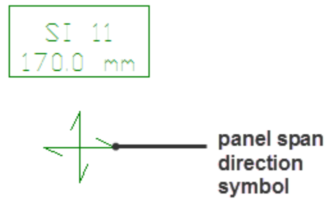
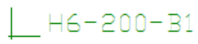
Setting	Description
Grade	Allows you to display the column grade in foundation layout drawings.
2x scale for steel columns	Allows you to double the scale of steel columns to simplify viewing the columns and their orientation.

. . . . Walls

Setting	Description
Wall Labelling position	Allows you to set the position of the wall label in relation to the wall in slab and mat layout drawings. The options are: • **Above** • **Inside** • **Below**
Wall Attributes	Allows you to select whether brackets are placed around the wall size in the wall label.

. . . . Slabs/Mats

Setting	Description
Include panel reference	Allows you to include the panel reference in the panel label.
Include panel thickness	Allows you to include the panel thickness in the panel label.
Include surface offset (if non-zero)	Allows you to include any surface offset that has been applied to the panel in the panel label.
Include border around label	Allows you to add a border around the label.
Align label to panel reinforcement	Allows you to select how the panel label is aligned to the panel span direction. Select the option to achieve the result displayed on the left, and

Setting	Description
	clear the option to achieve the result displayed on the right. 
Include panel span direction symbol	Allows you to select whether a direction symbol is displayed in the slab or mat geometry. 
Extend loose bar panel reinforcement lines across full panel	Allows you to draw bar panel reinforcement across the entire panel. Clear the option to display the loose bar reinforcement as shown in the following image: 
Always show main bar layer for rectangular mesh	If the mesh is not a square mesh, the main bars are normally put on the outer layer of the drawing, and no text is required in the drawing. However, if the bars are not on the outer layer of the drawing, they are indicated by adding B2 (if bottom mesh) or T2 (if top mesh) aligned to the main bar direction. **NOTE** If a square mesh (has the same size and spacing of bars in both directions) is applied, a square mesh symbol is used, and bars are shown

Setting	Description
	equally spaced in both directions. If the mesh is not a square mesh (does not have the same size and spacing of bars in both directions), a rectangular mesh symbol is used. Bars are shown in both directions, but with closer spacing for the more closely spaced bars in the mesh.
Anchorage rounding increment	Allows you to specify the rounding value applied to the anchorage length.

. . . . Patches

Setting	Description
Show patches with no reinforcement	Allows you to display patches that have no reinforcement specified in drawings.
Draw full anchorage lengths	Allows you to display bars with full anchorage lengths.
Draw curtailed (indicative) anchorage lengths = max()	Allows you to draw curtailed (indicative) anchorage lengths instead of full anchorage lengths and specify their size.

. . . . Punching Shear

Setting	Description
Show punching reinforcement details	Allows you to include a detail to the side of the layout showing the punching shear reinforcement provided.
Don't show area of steel requirement where rails have been designed	Allows you to only display the area of steel requirement for punching shear check items where rails are not provided (such as walls or column drops). Clear the option to display the area of steel requirement for all punching check items.

Setting	Description
Hide punching reinforcement on the main layout	Allows you to hide the punching reinforcement on the main layout. The reinforcement is still displayed in the punching check detail drawing, provided that **Show punching reinforcement details** is selected.

Slab and mat punching check detail options

Button, command, or option	Description
Content subpage	
Dimensions tab	
Show column to first stud spacing	Allows you to add a dimension from the column face to the first stud.
Show stud spacing	Allows you to add dimensions displaying the stud spacings along the rail.
Show rail spacing	Allows you to add dimensions displaying the rail spacings.
Include schematic showing stud dimensions	Allows you to add a schematic displaying the stud width and height above the rail.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in punching check detail drawings.
Style subpage	
Show column as hatched	Allows you to hatch the column.
Underline punching check label	Allows you to underline the punching check label in punching check detail drawings.

Foundation layout options

. . . . General

Setting	Description
Show columns and walls above the level as hatched	Allows you to hatch the columns and walls that continue above the current level.
Show transfer columns and walls as cross hatched	Allows you to cross-hatch transfer columns and walls.

Setting	Description
Show pile location and loading table	Click **Columns** to choose which columns of tabular data are to be included in the table on the drawing. **NOTE** Only the single most critical load condition is reported in the table for each pile. - could be tension or compression - could be a gravity, wind, or seismic combination

. . . . Beams

Setting	Description
Use detail group name	Allows you to use the detail group name in the beam label. Clear the option to use the design group name in the beam label instead.
Include the beam name	Allows you to include the beam name in the label for grouped beams.
Concrete Beam Labelling Position	Allow you to modify the labeling of beams when the beams have been designed using groups. You can select or clear the following options: • **Use detail group name**: Select the option to use the detail group name in the label, or clear the option to use the design group name in the label. • **Include the beam name**: Select the option to include the beam name in the label for grouped beams.
Show beam mark	Allows you to display beam marks in foundation layout drawings. The position options for beam marks are: • **Above** • **Below**
Show beam attributes	Allows you to display beam attributes in foundation layout drawings. The position options for beam attributes are: • **Above** • **Below**

Setting	Description
Beam Attributes	Allows you to select which beam attributes are displayed in foundation layout drawings and how they are displayed. You can select or clear the following options: • **Show beam size in parentheses**: Places brackets around the beam size in the beam label. • **Grade**: Displays the beam grade in planar drawings. • **Camber**: Displays the camber in steel beams. To modify the camber prefix, type the desired value in the **Camber prefix** field. • **Composite properties**: Displays composite beam properties. To change the separators inside which the number of studs is displayed, select the desired the **Stud separator** list. • **Transverse reinforcement**: Displays transverse reinforcement in foundation layout drawings.

. . . . Braces

Setting	Description
Show brace mark	Allows you to display brace marks in foundation layout drawings. The position options for brace marks are: • **Above** • **Below**
Show brace attributes	Allows you to display brace attributes in foundation layout drawings. The position options for brace attributes are: • **Above** • **Below**
Brace Attributes	Allows you to select whether brace grades are displayed in foundation layout drawings.

. . . . *Columns*

Setting	Description
Use detail group name	Allows you to use the detail group name in the column label. Clear the option to use the design group name in the column label instead.
Include the column name	Allows you to include the column name in the label for grouped columns.
Show column mark	Allows you to display column marks in foundation layout drawings. The position options for column marks are: • **Above** • **Below**

Show column attributes	Allows you to display column attributes in foundation layout drawings. The position options for column attributes are:
	• For position in elevation:
	  • **Above**
	  • **Below**
	• For position in cross-section:
	  • **To the right of the mark**
	  • **Below the mark**

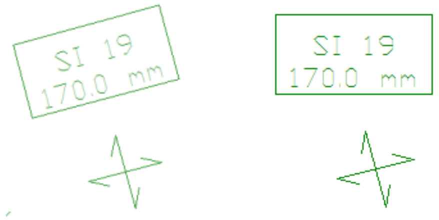
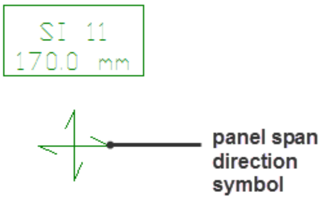
Show column size in parentheses	Allows you to place brackets around the column size in the column label.
Grade	Allows you to display the column grade in foundation layout drawings.
2x scale for steel columns	Allows you to double the scale of steel columns to simplify viewing the columns and their orientation.

. . . . *Walls*

Setting	Description
Wall Labelling position	Allows you to set the position of the wall label in relation to the wall in foundation layout drawings. The options are: • **Above** • **Inside** • **Below**

Setting	Description
Wall Attributes	Allows you to select whether brackets are placed around the wall size in the wall label.

. . . . *Slabs/Mats*

Setting	Description
Include panel reference	Allows you to include the panel reference in the panel label.
Include panel thickness	Allows you to include the panel thickness in the panel label.
Include surface offset (if non-zero)	Allows you to include any surface offset that has been applied to the panel in the panel label.
Include border around label	Allows you to add a border around the panel label.
Align label to panel reinforcement	Allows you to select how the panel label is aligned to the panel span direction. Select the option to achieve the result displayed on the left, and clear the option to achieve the result displayed on the right. 
Include panel span direction symbol	Allows you to select whether a direction symbol is displayed in the slab or mat geometry. 
Show pile location table	Allows you to include a table showing the pile locations in foundation layout drawings.
Show pile reference	Allows you to display the pile references in foundation layout drawings.

. . . . Isolated Foundations

Setting	Description
Show pile type name	Allows you to include the pile type names in the footing attributes displayed for each pile cap in foundation layout drawings.
Show foundation details	Allows you to include pad base or strip base and pile cap details, displaying the designed reinforcement, on the side of the foundation layout drawing.
Show reinforcement quantities table	Allows you to include the reinforcement quantity table for the pad bases and strip bases and pile caps displayed in foundation layout drawings.
Show pad/strip base schedule	Allows you to include the reinforcement schedule for the pad bases or strip bases in foundation layout drawings.
Show pile cap schedule	Allows you to include the reinforcement schedule for the pile caps in foundation layout drawings.
Show allowable pile capacity	Allows you to include the allowable pile capacity table for the piles under pile caps in foundation layout drawings.
Show overall dimensions	Allows you to include the overall dimensions of the pad bases or strip bases and pile caps in foundation layout drawings.
Show pile spacings	Allows you to include the pile cap pile spacing dimensions in foundation layout drawings.
Include the foundation name	Allows you to include the pad base or strip base or pile cap name in the footing mark for grouped footings in foundation layout drawings.
Labelling Position	Allows you to control where the footing mark and attributes are displayed for isolated foundations. The position options are: • **Above** • **Below**

Isolated Foundation detail options

. . . . Content

Setting	Description
Content subpage	

Setting	Description
Grouped Foundations tab	
Show number of foundations in group	Allows you to display the number of bases or pile caps in the group.
Cross-sections tab	
Positions list	Allows you to select how cross-section positions are displayed. You can select to not display cross-section positions at all, display a length-wise section or display cross-sections in both directions.
Pile Labels tab	
Show pile type name	Allows you to include the pile type name in the pile label.
Dimensions tab	
Show overall dimensions	Allows you to include the overall dimensions of the base/pile cap.
Show supported member dimensions	Allows you to include dimensions of the column or wall being supported.
Show pile spacings	Allows you to include dimensions from center to center of each pile for pile caps.
Quantities tab	
Show reinforcement quantities table	Allows you to include reinforcement quantity tables in isolated foundation detail drawings.

. . . . Style

Setting	Description
General tab	
Show supported member as hatched	Allows you to hatch columns and walls that are supported on the base pile cap in isolated foundation detail drawings.
Underline foundation label	Allows you to underline the foundation label on the detail in isolated foundation detail drawings.
Cross-sections tab	
Section label style list	Allows you to select the label style to be applied to cross-sections in isolated foundation detail drawings.
Restart labels in each foundation	Allows you to restart labeling from the beginning of each section when multiple foundations are included in the same drawing.

See also

Adjust and apply drawing settings (page 851)

Settings set settings

The **Settings Sets** page in the **Settings** dialog box controls the settings sets which are used to create the model/analysis/design settings in new projects, and which can also be used to replace the model/analysis/design settings in existing projects.

Button, command, or option	Description
Available settings sets list	Allows you to select a settings set whose content you can view and modify on the other pages of the **Settings** dialog box.
Add Copy	Creates a new settings set based on the one selected in the **Available settings sets** list. You can then customize the new settings set on the other pages of the **Settings** dialog box.
>> Active	Makes the settings set that is selected in the **Settings** dialog box the active settings set.
Import...	Allows you to import a settings set from another region. The selected settings set appears in the **Settings** dialog box.
Rename	Allows you to rename the settings set that is selected in the **Available settings sets** list.
Remove	Deletes the settings set that is selected in the **Available settings sets** list.
Open Folder	Opens the folder in which the existing settings sets are located.

See also

Manage settings sets (page 886)

General settings

The **General** page and its subpages in the **Settings** dialog box allow you to configure the language, captions, autosave settings, confirmation messages, and updates in Tekla Structural Designer.

Button, command, or option	Description
Language subpage	
Select the preferred language list	Allows you to select whether the terms in the interface and the output are in US English or UK English. **NOTE** The language cannot be configured differently in different settings sets.
Appearance page	
Show captions on top	Allows you to select whether you want to show captions on the top or the bottom of the **Project Workspace**.
Autorecover subpage	
Autosave	
Enabled	Allows you to select whether Tekla Structural Designer creates an automatic saves of models. Autosave may be useful, for example, when restarting Tekla Structural Designer after a crash. For more information, see: Work with autosave and backups (page 51)
Interval	Allows you to determine the interval at which Tekla Structural Designer creates automatic saves of models.
Backups	
Enabled	Allows you to select whether Tekla Structural Designer creates automatic backups of models. Backup may be useful, for example, if you want to revert to a previous version of the model. For more information, see: Work with autosave and backups (page 51)

Button, command, or option	Description
Interval	Allows you to determine the interval at which Tekla Structural Designer creates automatic backups of models.
Location	Defines the backup **Location** directory - the default is your Windows user profile Documents directory. Click the link to browse to your preferred location (both local and network locations are allowed).
Maximum number of backups per project	default = 10 (when the limit is reached the older backups start to be automatically deleted).
Confirmation subpage	
Confirm on list	Allows you to select the operations that you want to confirm each time.
Update Service subpage	
Enable Update Service	Allows you to select whether you want Tekla Structural Designer to notify you when new product updates or service packs can be installed, so that you can keep Tekla Structural Designer up to date.
Check for updates when starting Tekla Structural Designer	Allows you to select whether Tekla Structural Designer check if new updates are available each time it starts.
Check for non critical updates every [n] days	Allows you to determine the interval at which Tekla Structural Designer checks if new non-critical updates are available.
Yes, I am willing to participate anonymously	Allows you to select whether the Tekla Customer Experience Improvement Program can collect anonymous information on your hardware configuration and your use of Tekla products to improve the customer experience of Tekla products.

Results Viewer settings

The **Results Viewer** page and its subpages in the **Settings** dialog box allow you to configure the appearance of the on-screen results viewer for the settings set selected in the **Select the settings set to edit** list.

NOTE Unlike other settings, changes made to the active settings set on the **Results Viewer** page are instantly applied to the current session when you click **OK** to close the **Settings** dialog box.

Button, command, or option	Description
General subpage	
Viewer font --> **Change...**	Allows you to modify the font that is displayed on the left side pane of the results viewer.
Styles subpage	
Report Styles list	Allows you to select the style whose font, color, or other properties you want to modify. The options are: • **Calc Normal**: not used in the current version. • **Table**: controls all the text displayed on the right side pane of the results viewer except for the first row in any table. • **Table Heading**: controls the heading row, or first row, in any table.
Font --> **Change...**	Allows you to change the font used for the currently selected style.
Colors --> **Foreground** --> **Change...**	Allows you to change the foreground color of the selected style.
Colors --> **Background** --> **Clear**	Allows you to use a clear background for the selected style.
Colors --> **Foreground** --> **Change...**	Allows you to change the background color of the selected style.
Vertical alignment	Allows you to modify the vertical alignment of the selected style.
Indentation	Allows you to determine the indentation of the selected style.
Line spacing	Allows you to determine the line spacing of the selected style.

Structure default settings

The **Structure Defaults** page in the **Settings** dialog box allows you to configure miscellaneous structure default settings that are applied to the settings set selected in the **Select the settings set to edit** list.

Button, command, or option	Description
Construction Levels --> **Default Type**	Allows you to define the default type for the new constructions levels you create in the **Construction Levels** dialog box.
Grid & Construction Lines --> **Extension length**	Allows you to define the default extension length of grid and construction lines.
Pattern loadcases	Allows you to define whether load patterns are applied to loadcases.
Nominal Cover	Allows you to modify the nominal cover of different members. **NOTE** Unlike other settings, changes made to the nominal cover settings are instantly applied to new members.

Section default settings

The **Section Defaults** page in the **Settings** dialog box allows you to specify the default section size for each member type when a new member is create. The default section sizes are applied to the settings set selected in the **Select the settings set to edit** list.

To change the default section size for a member type, click the section size in the **Section** column.

Section order default settings

The **Section Order Defaults** page in the **Settings** dialog box allows you to modify the default section orders for different countries by clicking the desired order in the **Section Order** column.

Solver settings

The **Solver** page in the **Settings** dialog box allows you to specify the solver method that is applied to the settings set selected in the **Select the settings set to edit** list.

Button, command, or option	Description
Global Matrix Storage	Allows you to specify the global matrix storage method. The options are **Compressed Sparse Row** and **Skyline**. **NOTE** We recommend that you select the **Compressed Sparse Row** option because it generally reduces the analysis time, particularly in the case of finite element analysis.

Scene settings

The **Scene** page and its subpages in the **Settings** dialog box allow you to control the color and opacity of each object type in scene views when using the settings set selected in the **Select the settings set to edit** list.

NOTE Unlike other settings, changes made to the active settings set on the **Scene** page are instantly applied to the current session when you click **OK** to close the **Settings** dialog box.

Button, command, or option	Description
Graphics subpage	
Driver **Antialiasing** **Transparency**	**NOTE** The default settings on the **Graphics** subpage are the recommended settings. Do not adjust them unless the Tekla Support Team instructs you to do so.
Colors subpage	
Background color list	Allows you to select whether the table below displays and allows you to adjust the colors for a light or dark background.

Button, command, or option	Description
Gradient Background	Unselect this option to apply a uniform white or black background to each window, with no gradient applied.
Reset Colors	Resets all the colors to the default settings.
Color Name	Displays the interface item to which the color applies. **TIP** An arrow is displayed to the left of some of the items, this can be clicked on in order to set the colors for sub-items. **NOTE** The **Gradient Legend** colors are those used when reviewing embodied carbon.
Opacity	Allows you to set the opacity of each color.
Color	Allows you to set the color for each interface item according to your needs.
Fonts subpage	
Reset Fonts	Resets all the fonts to the default settings.
Font Name	Displays the interface item to which the font applies.
Font	Allows you to change the font used for the interface item.
Size	Allows you to adjust the size of the font.
Bold	Allow you to further format the font.
Italic	
Contours subpage	
Lower bound [%]	Displays the lower bound of each FE contour in the results view. **TIP** To adjust the size of the lower bound, modify **Size [%]**.

Button, command, or option	Description
Upper bound [%]	Displays the upper bound of each FE contour in the results view. **TIP** To adjust the size of the upper bound, modify **Size [%]**.
Size [%]	Allows you to adjust the size of individual contours according to your needs. Note that the sum of the sizes must equate to 100 %.
Color	Allows you to change the color of a contour.
Split	Divides the contour that is currently selected in the table into two contours that are each half the size of the original contour.
Delete	Deletes the contour that is currently selected in the table.
Reset	Resets the contours to the default configuration that consists of 10 evenly sized contours.
Utilization Ratios subpage	
Minimum value	Allows you to adjust the minimum value of a ratio band. **NOTE** The value of the highest ratio band can be increased above 1.0, if necessary.
Color	Allows you to change the color of a ratio band.
Add	Creates a new ratio band that you can modify to the bottom of the table.
Delete	Deletes the ratio band that is currently selected in the table.
Reset	Resets the ratio bands to the default configuration that consists of 5 evenly sized ratio bands.
Sort	Rearranges the ratio bands in order of the values (from highest to lowest).
View Settings subpage	
Do not display values of story shear below	Allows you to limit the values of story shear for a new model. This way, you

Button, command, or option	Description
	can easily ignore the small values of story shear that might otherwise detract you from the more important story shear values. The story shear values that are less than the limiting value are not displayed in the results view.
Show full pile length	Allows you to select whether Tekla Structural Designer displays the full length of piles or a shorter pile length that you can specify.
2 x scale for steel columns	Allows you to double the scale of steel columns in 2D views in order to simplify viewing the columns and their orientation.

Report settings

The **Report** page and its subpages in the **Settings** dialog box allow you to modify the appearance of reports.

NOTE Unlike other settings, changes made to the active settings set on the **Report** page are instantly applied to the current session when you click **OK** to close the **Settings** dialog box.

Button, command, or option	Description
Styles subpage	
Report Styles list	Allows you to select a style used in reports whose font, color, or other properties you want to modify.
Font --> **Change...**	Allows you to change the font used for the currently selected style.
Colors --> **Foreground** --> **Change...**	Allows you to change the foreground color of the selected style.
Colors --> **Background** --> **Clear**	Allows you to use a clear background for the selected style.
Colors --> **Background** --> **Change...**	Allows you to change the background color of the selected style.
Vertical alignment	Allows you to modify the vertical alignment of the selected style.
Indentation	Allows you to determine the indentation of the selected style.

Button, command, or option	Description
Line spacing	Allows you to determine the line spacing of the selected style.
Page Options subpage	
Page margins	Allows you to adjust the margins on report pages.
Margin frame --> **Draw page margin frame**	Allows you to select whether Tekla Structural Designer creates a margin frame to separate the report text from the margins.
Margin frame --> **Color** --> **Change...**	Allows you to change the color of the margin frame.
First page number	Allows you to determine the number from which the page numbers start.
Page number prefix	Allows you to specify a prefix to the page number.
Table Options subpage	
Border Style	Allows you to specify the table border style. You can select to have no table borders, a single line border, or a double line border.
Border Properties --> **Color** --> **Change...**	Allows you to change the table border color.
Border Properties --> **Width**	Allows you to adjust the table border width.
Table width --> **[] percent of page width**	Allows you to specify the width of all tables included in reports.
Use landscape orientation for wide tables	When enabled (default) the report page orientation will automatically change to landscape mode when the table header will not fit in portrait orientation. **NOTE** You can reduce the need for landscape orientation and number of pages by; reducing page margins (default values may be quite wide); increasing the allowable table width from 90 to 95% of the available width
Document Options subpage	

Button, command, or option	Description
Show document header	Allows you to include the header at the top of each report page.
Show border in document header	Draws a border around each field cell in the header.
Show document footer	Allows you to include the header at the top of each report page.
Show border in document footer	Draws a border around each field cell in the footer.
Show document field description in line with the value	Allows you to display the field descriptions in line with the field values in each cell. If you do not select the **Show document field description in line with the value** option, the field values are displayed on a new line.
Underline document field cell	Underlines the cell value of each field.
Image width --> [] percent of page width	Allows you to specify the width of all images included in reports.
Paragraphs --> Spacing	Allows you to specify the paragraph spacing in reports.
Start each item on new page	Allows you to start each report chapter on a new page.
Start each member report on new page	Allows you to start each member report on a new page.
Picture Fonts subpage	
Reset Fonts	Resets all the fonts to the default settings.
Font Name	Displays the report item to which the font applies.
Font	Allows you to change the font used for a report item.
Size	Allows you to adjust the size of the font.
Bold	Allow you to further format the font.
Italic	

See also

Adjust and apply report settings (page 834)

Embodied carbon settings

The **Embodied Carbon** page allows you to control the carbon factors applied:

- In the **current** project - when accessed from the Model Settings (page 1032)
- In **new** projects - when accessed from the Settings dialog (page 1209)

Embodied carbon settings when accessed from Model Settings

Button, command, or option	Description
Steel Connections	
Pin-ended allowance	Allows you to define the allowance for pin-ended connections in steel members as a percentage of the mass of the member.
Fixed-ended allowance	Allows you to define the allowance for fixed-ended connections in steel members as a percentage of the mass of the member.

Embodied carbon settings in the Settings dialog

Button, command, or option	Description
Carbon factors	
Edit	Opens the Embodied Carbon Factors dialog (page 1185) to allow you to define the embodied carbon factors that will be used in new models.
Steel Connections	
Fixed-ended allowance	Allows you to define the allowance for fixed-ended connections in steel members as a percentage of the mass of the member.
Pin-ended allowance	Allows you to define the allowance for pin-ended connections in steel members as a percentage of the mass of the member.

Performance settings

The **Performance** page in the **Settings** dialog box allow you to optimise analysis and design performance according to your preference and your PC's capabilities.

Button, command, or option	Description
General	To optimise performance choose one of the following: • **Conserve memory** (default) • **Favour speed** **NOTE** We recommend the 'Favour speed' option is used only on higher specification PC's with multi-core processors and the recommended amount of RAM.
Design	Options are provided to allow you to take advantage of multi-core processing in the design and chase-down phases. • **Use multi-core processors for design** • **Run chase-downs concurrently**

13.3 Dialogs

This section covers some of the important dialogs in Tekla Structural Designer and their different options.

Click the links below to find out more:

- Analysis Settings dialog (page 1173)
- Connection Resistance dialog (page 1174)
- Construction Levels dialog (page 1177)
- Design Settings dialog (page 1179)
- Drawing Settings dialog (page 1180)
- Edit Reinforcement dialog (page 1180)
- Embodied Carbon Factors dialog (page 1185)
-

Analysis Settings dialog

Summary

The **Analysis Settings** dialog and its subpages allow you to adjust the settings applied to the different analyses.

Location

On the **Analyze** tab, click **Settings**.

Content

Button, command or option	Description
1st Order Non-Linear	See 1st order non-linear settings (page 1050)
2nd Order Non-Linear	See 2nd order non-linear settings (page 1051)
1st Order Modal	See 1st order modal settings (page 1052)
2nd Order Buckling	See 2nd order buckling settings (page 1055)
1st Order Seismic	See 1st order seismic settings (page 1056)

Button, command or option	Description	
Iterative Cracked Section Analysis	See Iterative cracked section analysis settings (page 1060)	
Modification Factors	See Modification factors (page 1061)	
Meshing	See Meshing settings (page 1062)	
Composite Steel Beams	See Composite steel beams settings (page 1062)	
OK button	Apply the changes to the current project.	
Cancel button	Cancel the changes.	
Save button	Save the changes back to the active settings set for future use.	
Load button	Revert to the model settings specified in the active settings set.	

Connection Resistance dialog

The **Connection Resistance dialog** displays the pre-defined connection resistances for steel beams to Eurocode and US head codes. It can also be used to add or edit user-defined connection resistances.

To open the dialog:

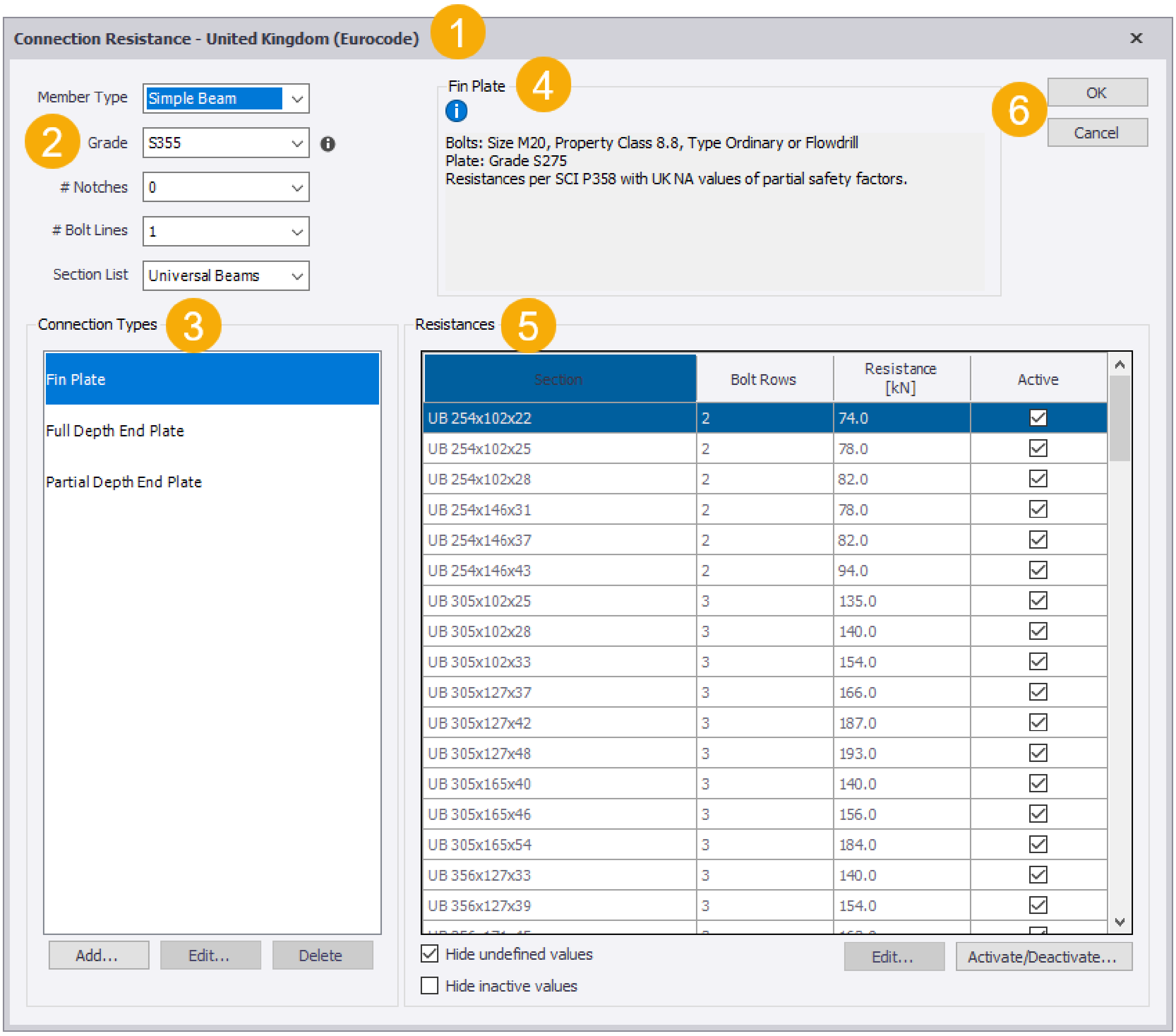
1. Click the **Home** tab.
2. Click  **Materials**.

   The **Materials** dialog is displayed.
3. On the **Sections** page, check that the required head code and material are displayed.

   NOTE Both steel and cold formed materials can have resistances defined, unless the head code is US, in which case resistances can be defined for steel only.

4. Click **Connection Resistance**

The dialog content is described below.



1. Title Bar

The title bar displays:

- The currently selected head code.
- For India this also displays LS or WS, and for US displays LRFD or ASD, according to the setting of the currently open model.

2. Filters

The filter settings determine what is displayed elsewhere in the viewer.

The Member Type filter has options for:

- Simple Beam
- Brace

3. Connection Types

Pre-defined and user-defined Connection Types are listed here according to the Filter settings.

- Click the **Add...** button to add a user-defined Connection Type (page 896).
- Click the **Edit...** button to edit a user-defined Connection Type.
- Click the **Delete** button to delete a user-defined Connection Type.

NOTE The **Edit...** and **Delete** buttons are only active when a *user-defined* Type is selected.

4. Info box

Displays information related to the currently selected Connection Type. For user-defined Connection Types the information displayed here can be added or edited via the Add... and Edit... buttons in the Connection Types area of the viewer.

5. Resistances

- Resistance values are listed in the window according to the Filter settings and the currently selected Connection Type. The resistance values (for user-defined Connection Types) can be edited directly.
- The 'Active' setting means that, when ticked, the particular resistance will be considered in the check. The Active status can be edited directly in this window for individual resistances but is more quickly edited using Activate/ Deactivate. All pre-defined resistances start out Active.
- If 'Hide undefined values' is un-ticked those resistances which have zero value currently assigned will be displayed along with the other sections. This box is ticked by default when opening the viewer.
- If 'Hide inactive values' is ticked those resistances which have 'Active' un-ticked will be hidden in this window. This box is un-ticked by default when opening the viewer.
- Use the Edit... button to edit the bolt row count or resistance or Active status of a user-defined Connection Type. Note that multiples of the same section size can be added through the Edit Resistances dialog, so that the same size with different bolt row counts and resistances can be defined.

  Click here for an example (page 898) showing the use of Edit... as described above.

- Use the Activate/Deactivate... button to change the Active setting for multiple sections. Set the filters in the Activate/Deactivate dialog, then set either Activate or Deactivate, and then Apply if you want to change further Active settings or Apply & Close to finish.

  Click here for a Eurocode example (page 636) or here for a US example (page 638) showing the use of Activate/Deactivate as described above.

6. OK and Cancel

Button	Description
OK	Commit all changes made in the viewer to the database.
Cancel	Closes the dialog box without saving changes.

See also:

Check simple connection resistance (page 635)

Construction Levels dialog

The **Construction Levels** dialog allows you to define the levels required in order to construct your model.

Choose from:

- On the **Model** tab, click  **Construction Levels**.
- In the **Structure** tree, double-click  **Levels**.

Content

Button, command or option	Description
Options	
Ref	Allows you to assign a unique reference to a construction level.
Name	Allows you to assign an optional name to further assist identification. For example, "First floor" or "Mezzanine".
Type	Allows you to select the Type from the drop down menu: • **T.O.S.** = Top of steel • **S.S.L.** = Structural slab level

Button, command or option	Description
	• **T.O.F.** = Top of foundation **WARNING** Slabs are modeled above the level when they are set to T.O.S or T.O.F but below the level when they are set to S.S.L
Level	Allows you to assign the height of the construction level above the base level.
Spacing	Allows you to assign the distance of this construction level to the one immediately below.
Source	Allows you to assign the level as Unique or the same as another level. Unique levels can be edited independently, whereas edits to indentical levels are applied at both levels simultaneously.
Slab Th.	Allows you to set the default thickness for any slab created on the level.
Floor	Allows you to assign the construction level as a major level in the building. Floor levels determine the number of sub models that are created for the chasedown analysis. Floor levels are also used to determine items such as your inter story height. There will certainly be a number of levels that are clearly floor levels, but there could be many others that are not. For example you create intermediate levels in order to define: • half landing levels and stairs • K bracing. You require a construction level for the intermediate bracing connection points • Steps in the building floor levels When you define a level that is clearly not a floor, then you should not check the option. If a floor exists only at some locations in the level (e.g. in a building with stepped floor levels), then you should check the option where applicable.
Buttons	
OK	Allows you to save any changes made.
Cancel	Allows you to discard any changes made.
Insert Above	Allows you to insert a new construction level above the currently highlighted level with the same spacing.
Insert Below	Allows you to insert a new construction level below the currently highlighted level with the same spacing.

Button, command or option	Description
New on Top	Allows you to insert multiple construction levels above the uppermost level with variable spacings **NOTE** 3*5 inserts three levels, each at 5m spacing 3,4,5 inserts three levels at spacings of 3,4 and 5m respectively.
New at the Bottom	Allows you to insert multiple construction levels below the lowest level with variable spacings
Delete	Allows you to delete the selected level.

Design Settings dialog

The **Design Settings** dialog applies design settings to the current project.

Location

On the **Design** tab, click **Settings**.

Content

Button, command or option	Description
Analysis	See Analysis (page 1066)
General	See General (page 1065)
Steel > General	Steel > General (Eurocode only) (page 1067)
Steel > Composite Beams	See Steel > Composite Beams (page 1068)
Steel > Steel Joists	See Steel > Steel Joists (page 1069)
Concrete > Cast-in-place	See Concrete > Cast-in-place (page 1070)
Concrete > Precast	See Concrete > Precast
Design Forces	See Design Forces (page 1096)
Design Groups	See Design Groups (page 1113)
Autodesign	See Autodesign (page 1114)
Design Warnings (AISC/ASC only)	See Design Warnings (page 1115)
Sway & Drift Checks	See Sway and Drift Checks (page 1119)
Fire check	See Fire check (page 1121)
OK button	Apply the changes to the current project.
Cancel button	Cancel the changes.

Button, command or option	Description
Save button	Save the changes back to the active settings set for future use.
Load button	Revert to the model settings specified in the active settings set.

Drawing Settings dialog

Location

On the **Draw** tab, click **Settings**.

Content

Button, command or option	Description
General > Default export directory	Specifies the default folder for creation of dxf files
Layer Configuration	See Layer configurations (page 1126)
Layer Styles	See Layer styles (page 1127)
Options > Planar Drawings	See Planar drawing options (page 1129)
Options > Member Details	See Member detail options (page 1138)
Options > Member Schedules	See Member schedule options (page 1145)
Options > Slabs and Mats	See Slab and mat layout options (page 1147)
Options > Foundations	See Foundation layout options (page 1153)
OK button	Apply the changes to the current project.
Cancel button	Cancel the changes.
Save button	Save the changes back to the active settings set for future use.
Load button	Revert to the model settings specified in the active settings set.

Edit Reinforcement dialog

Summary

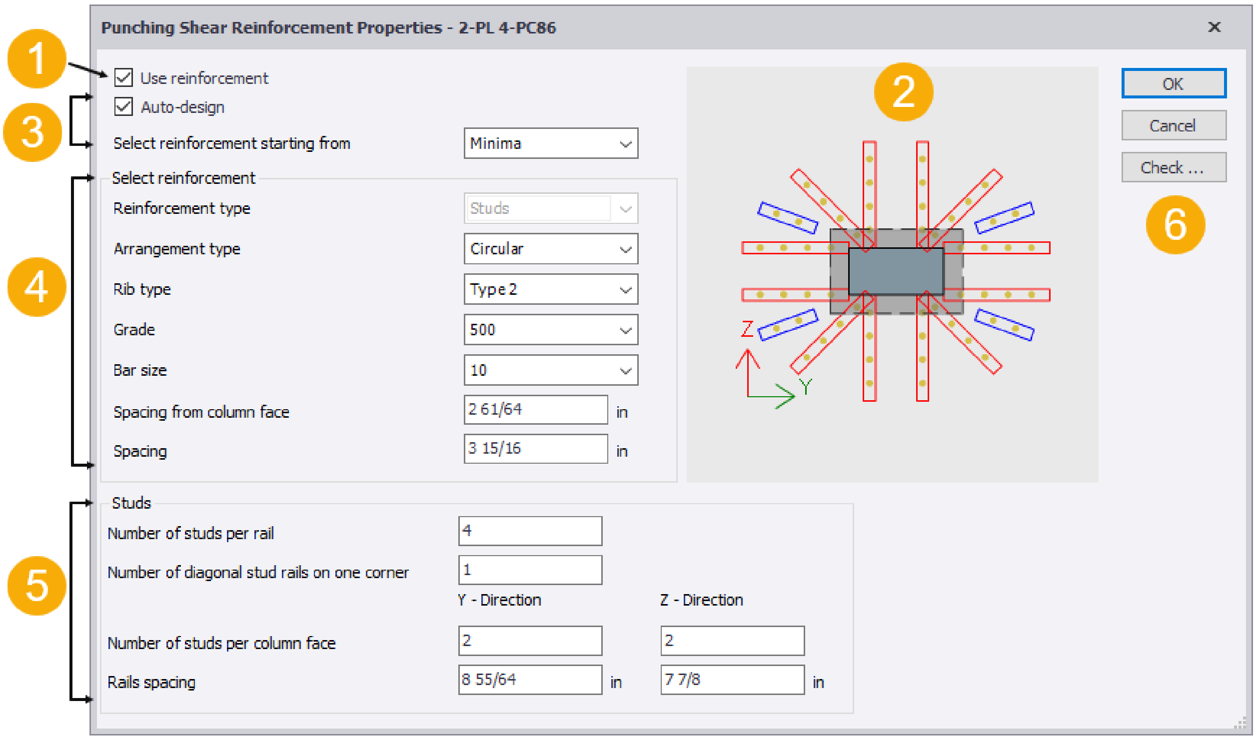
The **Edit Reinforcement** dialog box allows you to modify the reinforcement used in a punching shear check. The graphic in the dialog box previews the reinforcement and updates to match the changes that you make. The graphic also indicates other items that are specific to the check location.

Location

To display the dialog:

1. Right click on an existing punching check item.
2. In the context menu, select **Edit Reinforcement**.

Content



1. Use reinforcement

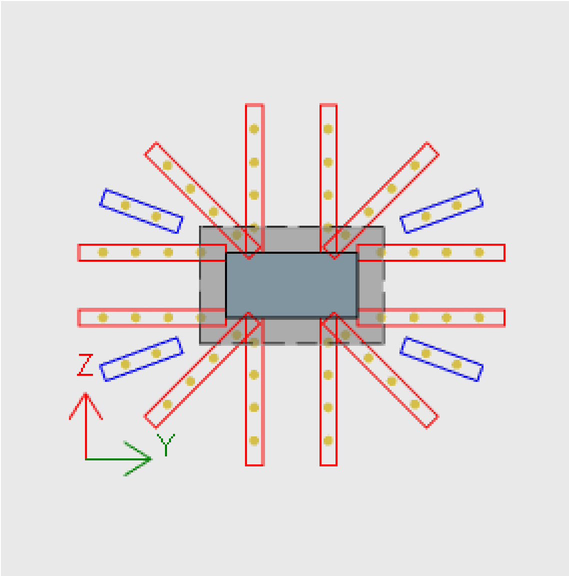
Selecting the **Use reinforcement** option allows you to apply a default punching reinforcement arrangement that can either be checked or used as the starting reinforcement for an auto design.

2. Preview graphic

The components of the preview graphic are as follows:

Stud rail reinforcement

Stud rail reinforcement is only shown if you have selected the **Use reinforcement** option.



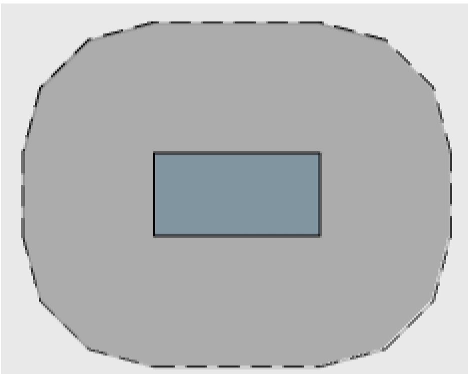
The red design rails signify rails considered in design.

The blue detailing rails are only considered for detailing purposes.

Perimeters

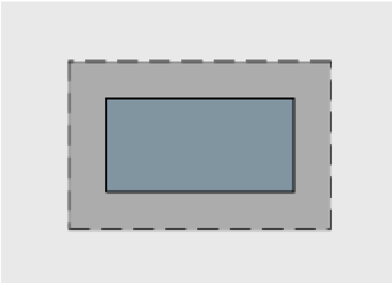
Perimeters are displayed as dashed lines around the column section and will have different shapes and positioning depending upon the head code being worked to.

Control Perimeter (EC)



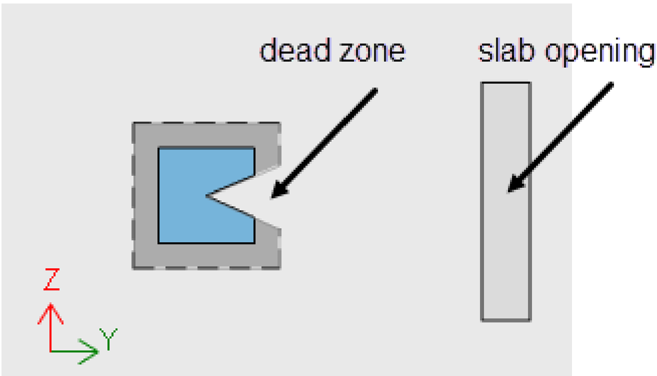
The control perimeter is the perimeter of the dark grey area as shown above. This will vary depending on existing slab edges and openings.

Critical Perimeter (ACI)



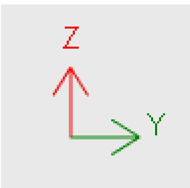
The critical perimeter is the perimeter of the dark grey area as shown above. This will vary depending on existing slab edges and openings.

Slab opening dead zones



Slab opening dead zones are displayed as interruptions in perimeters and reinforcement.

Punching shear check local axis



Using the YZ system orients the X axis upwards following the right-hand rule. This is also the local axis system for the column elements, so it is easier to relate.

Additional Perimeters

If reinforcement is found to be required, additional perimeters will be displayed representing positions beyond the critical/control perimeter where the reinforcement requirement is checked.

3. Auto-design

If you select the **Auto-design** option, the reinforcement is increased until either a pass is achieved or the limiting reinforcement parameter limits have been exceeded. You can then select the starting point in the **Select reinforcement starting from** list.

The **Select reinforcement starting from** list allows you to select the staring point for auto-design procedures. The options are:

- **Minima**: removes the current arrangement and starts the reinforcement with the minimum allowed bar size.
- **Current**: auto design starts from the current bar arrangement. The **Current** option is only available if you have selected **Use reinforcement** in the **Properties** window.

4. Select reinforcement parameters

Reinforcement type

In the current release, only stud reinforcement is available.

Arrangement type

Allows you to define whether the reinforcement arrangement is orthogonal or circular.

Rib type

Allows you to specify the reinforcement rib type.

Grade

The reinforcement grades that are available here are set in the **Materials** dialog box.

Bar size

The reinforcement bar sizes that are available here are in the **Materials** dialog box.

Spacing from column face

Defines the spacing of the first bar in each rail from the column face.

NOTE The option is only available if the **Use reinforcement** option has been selected.

Spacing

Defines the spacing between bars along each rail.

NOTE The option is only available if the **Use reinforcement** option has been selected.

5. Studs parameters

Number of studs per rail

Allows you to define the number of studs on each rail.

Number of diagonal stud rails on one corner

Allows you to define the number of stud rails adjacent to each corner of the column.

NOTE The option is only displayed when **Arrangement type** is set to **Circular**.

Number of studs per column face

Allows you to define the number of stud rails adjacent to the column face in the local y and z directions.

Rails spacing

Allows you to define the spacing between rails in the local y and z directions.

6. Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Check...	Opens the **Results** dialog box that displays the detailed results for the current design.

See also

Create punching shear checks (page 627)

Design and check punching shear (page 629)

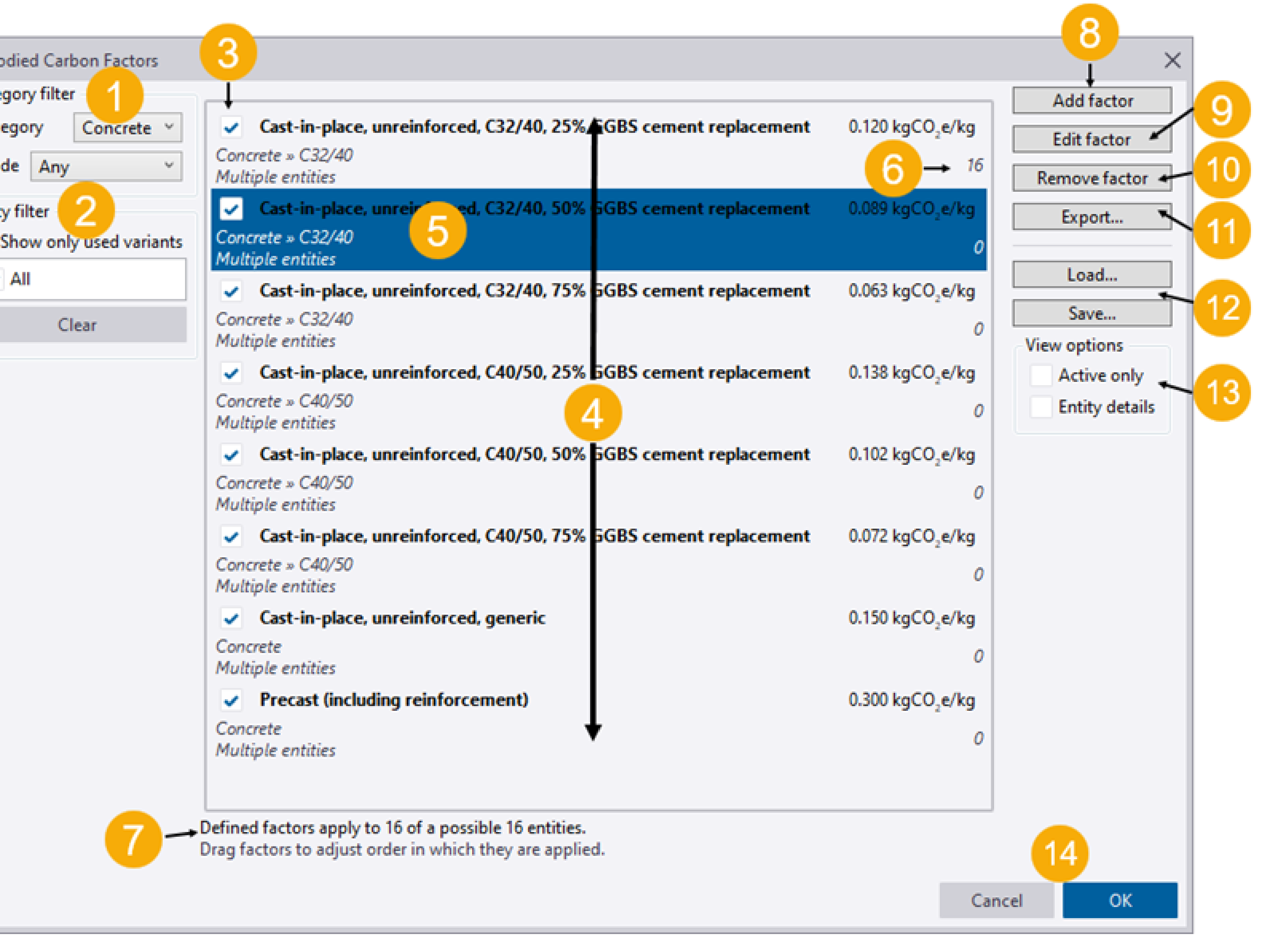
Embodied Carbon Factors dialog

The **Embodied Carbon Factors** dialog on the **Home** ribbon is used to add and to manage embodied carbon factors (ECFs). The order in which factors are arranged in the dialog determines the priority in which they are assigned to models.

Filters can be applied to make it easier to interrogate the list, these do not affect how the factors are subsequently applied to the model.

NOTE Any changes made in the dialog will only apply to the current model until the factors are saved to the active global settings set.

The dialog is shown below.



1. Category filter

The category filter contains a list of all item categories that it is possible to apply embodied carbon factors to.

Select a category in order to see the factors that have been previously defined in that category.

Some of these such as 'Cladding' are optional items and might not have any factors defined.

You cannot add new categories.

Additional filters are also provided which vary depending on the category selected. For example, when the Steel category is selected the filters are *Grade* and *Section*.

NOTE For the UK Settings Set, some initial defaults have been provided for the Concrete, Metal Deck, Reinforcement, Steel, and Timber categories.

2. Entity filter

Each factor is associated with one or more variants within a specified category.

As there are potentially hundreds of factors that could be defined in any one category, the **Entity filter** provides a means of filtering the list to make it easier to interrogate a specific variant.

The **Entity filter** defaults to displaying all factors applicable to any variant. If this results in a long list that requires a lot of scrolling, you might choose to filter the list as follows:

1. Unselect **All** in order to see a choice of variants that apply to the current category.
    - Click **Show only used variants** if you want to have the choice of variants reduced further to just those that exist in the currently open model.
2. Select one or more variants in order to filter the main list accordingly.
    - Click **Clear** if you want to clear the selected variants and return to the default **All** setting.

3. Active flag

If the **Active** flag is unselected, the factor will not be applied to model.

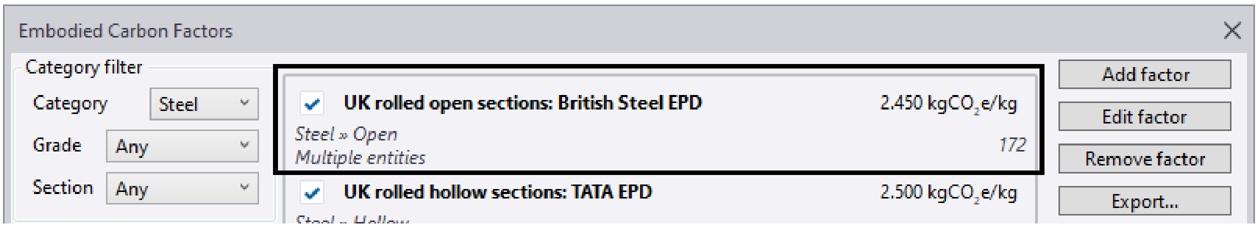
4. Filtered list of embodied carbon factors

The list is displayed in accordance with the **Category filter** and **Entity filter** criteria. It also takes account of the specified **View options**.

The list can be reordered by selecting individual items and dragging them up or down. The order is very important because each entity in the model will adopt the first applicable factor it finds in the list.

The number of entities to which the factor currently applies is shown to the right of the item. When the list is reordered these numbers are instantly updated.

5. Embodied carbon factor item



When a factor is added its name and value appear in the list as shown.

Beneath the item to the left are its category and entity criteria, and on the next line, the entities to which it applies.

Beneath the item to the right the number of entities to which the factor currently applies is shown.

6. Number of entities the factor currently applies to

The number of entities to which a factor currently applies is shown to the right of the item. When the list is reordered this number is instantly updated.

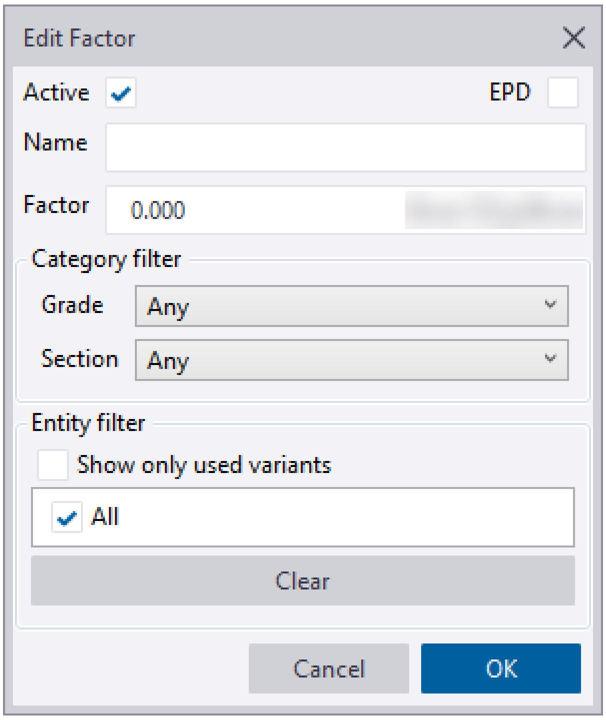
7. Defined factors applied to possible entities message

Located underneath the list of factors, this message indicates for the current filtering criteria, the number of entities that have a factor applied, and the total number of entities in the model meeting the criteria.

NOTE If the numbers don't match this indicates that the total embodied carbon will be under-reported.

8. Add factor

Add factor is used to add new factors within the category currently shown in the **Category filter**. Initially these would be available for the current model only.



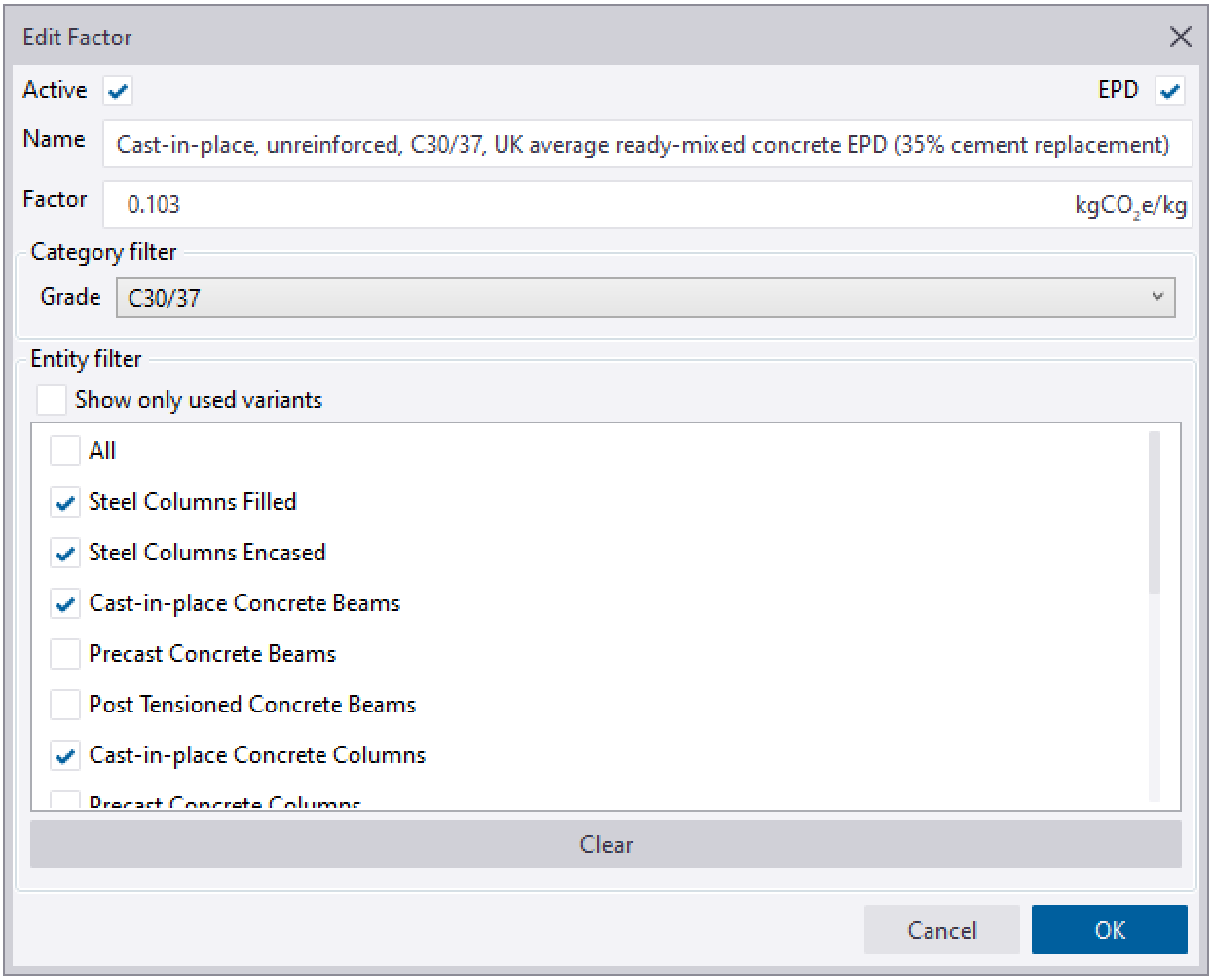
Item	Description
Active	• selected - the factor will initially be active when added • unselected - the factor will initially be inactive when added
EPD	This flag is used to indicate that the factor has come from an EPD. To make this clear it is suggested that you include 'EPD' in the name. **NOTE** Flagging a factor as from an EPD in this way does not make it uneditable. (Users might wish to update factors when EPDs get updated due to new manufacturing practices, and they might wish to change the name, or the filter determining which entities the factor applies to.)
Name	The name that you assign to a factor should be an exclusive identifier, so that it can be clearly differentiated from other factors that exist in the same category.
Factor	• defined in kgCO2/kg (metric units) • defined in lb av CO2 e/ib av (US customary units)

Item	Description
Category filter	The filters shown here will vary depending on the category. You can apply a filter in order to limit the range of applicability of the factor. **NOTE** The main category cannot be set from here. It has to be preselected in the **Category filter** on the **Embodied Carbon Factors dialog** before clicking **Add factor**.
Entity filter	You can apply an entity filter in order to further limit the range of applicability of the factor. The variants available in the entity filter depend on the applied category filter. Accepting the default setting **All** would allow the factor to be applied to all variants within the current category filter. Alternatively, unselect **All** in order to see a list of all variants to which the factor could potentially be applied. From this list you can then select one or more variants to which the factor will be exclusively applied. • Click **Clear** if you want to clear the selected variants and reselect. • Click **Show only used variants** to have the list only show those variants that have been used in the currently open model.
OK / Cancel	Click OK to add the factor to the list.

NOTE Once factors have been added, the updated list should be saved to the currently active global settings in order to make it available to new models also.

9. Edit factor

Edit factor is used to edit existing factors within the category currently shown in the **Category filter**



Item	Description
Active	• selected - the factor will initially be active after editing • unselected - the factor will initially be inactive after editing
EPD	This flag is used to indicate that the factor has come from an EPD. **NOTE** Flagging a factor as from an EPD in this way does not make it uneditable. (Users might wish to update factors when EPDs get updated due to new manufacturing practices, and they might wish to change the name, or the filter determining which entities the factor applies to.)
Name	The name that you assign to a factor should be an exclusive identifier, so that it can be clearly differentiated from other factors that exist in the same category.
Factor	• defined in kgCO2/kg (metric units), or

Item	Description
	• lb av CO2 e/ib av (US customary units)
Category filter	The filters shown here will vary depending on the category. You can apply a filter in order to limit the range of applicability of the factor.
Entity filter	You can apply an entity filter in order to further limit the range of applicability of the factor. The variants available in the entity filter depend on the applied category filter. Accepting the default setting **All** would allow the factor to be applied to all variants within the current category filter. Alternatively, unselect **All** in order to see a list of all variants to which the factor could potentially be applied. From this list you can then select one or more variants to which the factor will be exclusively applied. • Click **Clear** if you want to clear the selected variants and reselect. • Click **Show only used variants** to have the list only show those variants that have been used in the currently open model.
OK / Cancel	Click OK to add the factor to the list.

NOTE Initially any edits would apply to the current model only, but they can be saved to the currently active global settings in order to make them applicable to new models also.

10. Remove factor

If you select a factor in the main list and then click **Remove factor** it is deleted.

11. Export

Exports factors to a spreadsheet. You are given the choice to export:

- All factors
- Active factors
- Used factors

12. Load and Save

Button	Description
Load	Load carbon factors from the currently active global settings.

Button	Description
	WARNING Any existing factors will be discarded.
Save	Save carbon factors to the currently active global settings. **WARNING** Any existing factors in the currently active global settings will be discarded.

13. View options

View options apply to the display of the main list. When selected these apply as follows:

- Active only - only those factors with the Active flag selected are displayed in the main list.
- Entity details - the *multiple entities* text is expanded to list the entity details instead.

14. OK and Cancel

Button	Description
OK	Commit all changes made in the viewer to the database.
Cancel	Closes the dialog box without saving changes.

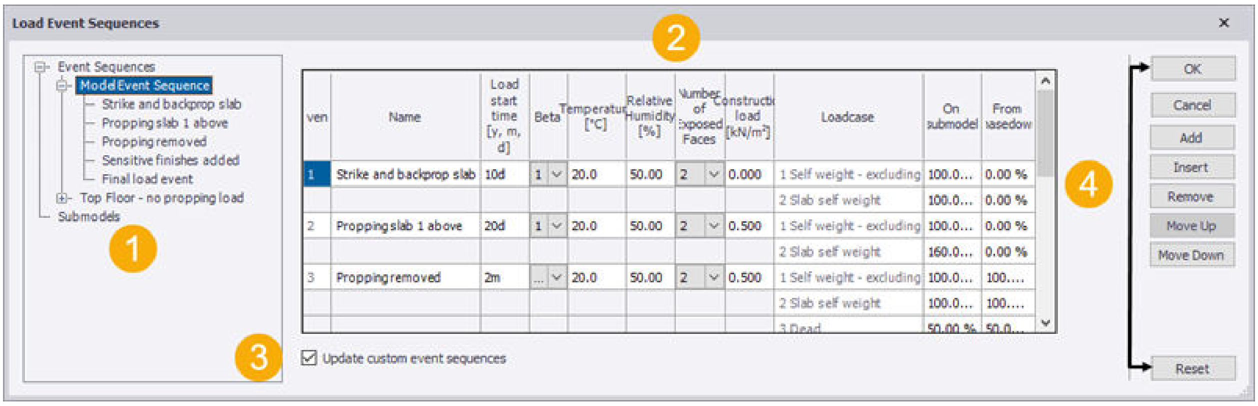
Load Event Sequences dialog

The **Load Event Sequences dialog** is used to define all the construction stage events that slabs in a sub model will go through starting from the day the slabs are cast.

To display the dialog:

- On the **Slab Deflection** toolbar, click **Event Sequences**

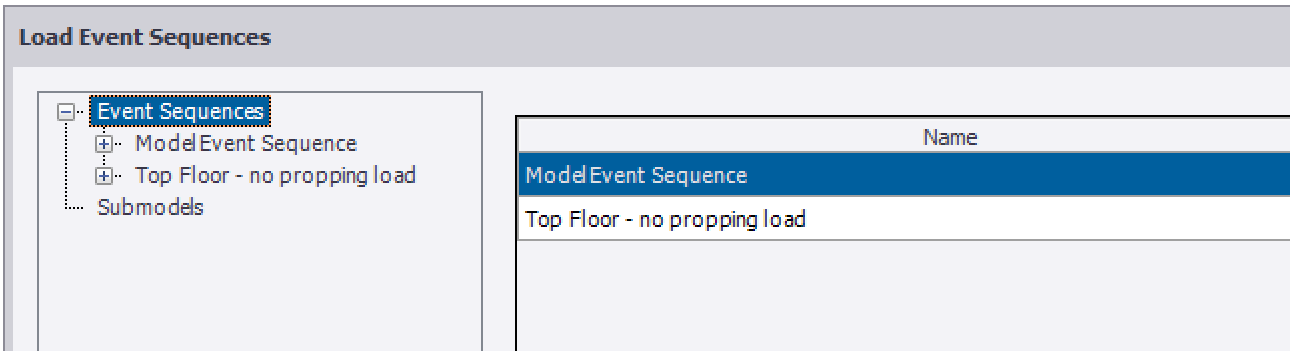
The dialog content is described below.



1. Event sequences and submodels pane

Event Sequences

Select **Event Sequences** to show a summary of the model event sequence and any custom event sequences that have been defined.

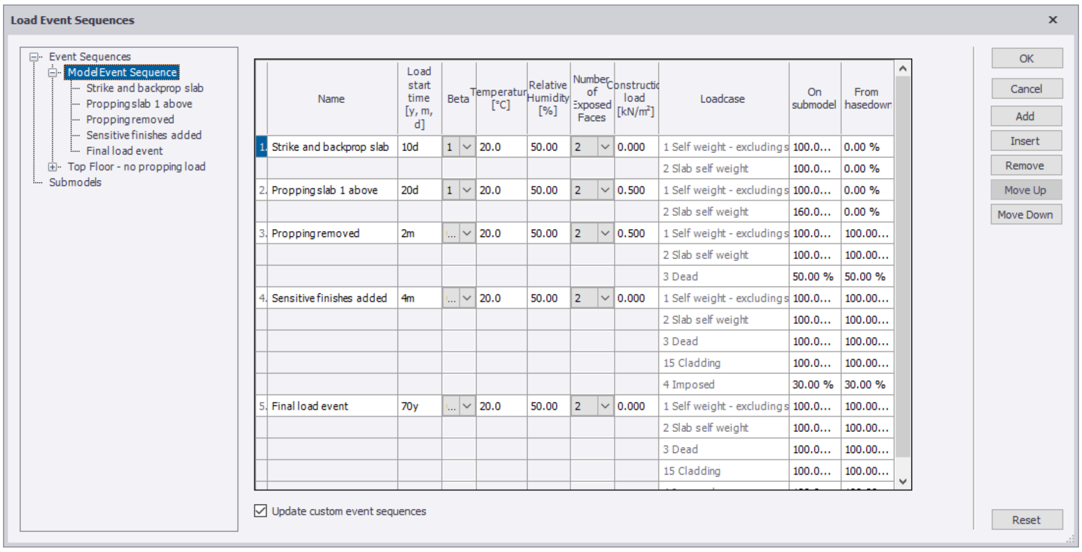


When this page is active you can:

- **Add** a Custom Load Event Sequence.
- **Remove** a selected Custom Load Event Sequence.
- Review which Event Sequences are used in submodels

Model Event Sequence

Select **Model Event Sequence** to display the various event sequence parameters for editing.

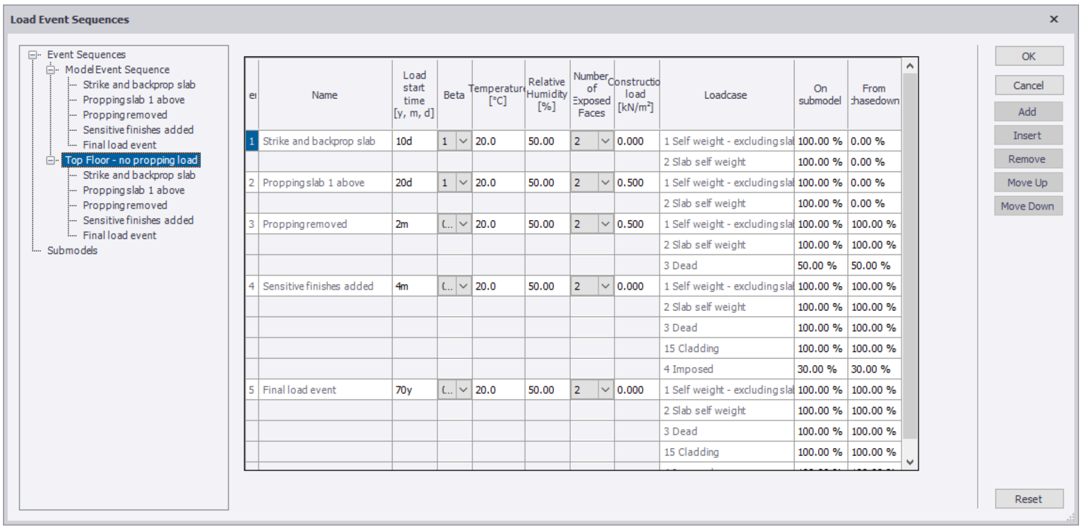


When this page is active you can:

- Review the existing Load Events and edit all the quantities (except included loadcases)
- **Add** an event to the end of the sequence
- **Insert** an event above the selected event in the sequence
- **Remove** events
- **Move Up** or **Move Down** events to re-order the sequence
- **Reset** the Model Event Sequence to the default Event Sequence in the active Settings Set

Custom Event Sequences

Pages for custom event sequences are only displayed if they have been added from the **Event Sequences** page.



When a custom event sequence page is active you can:

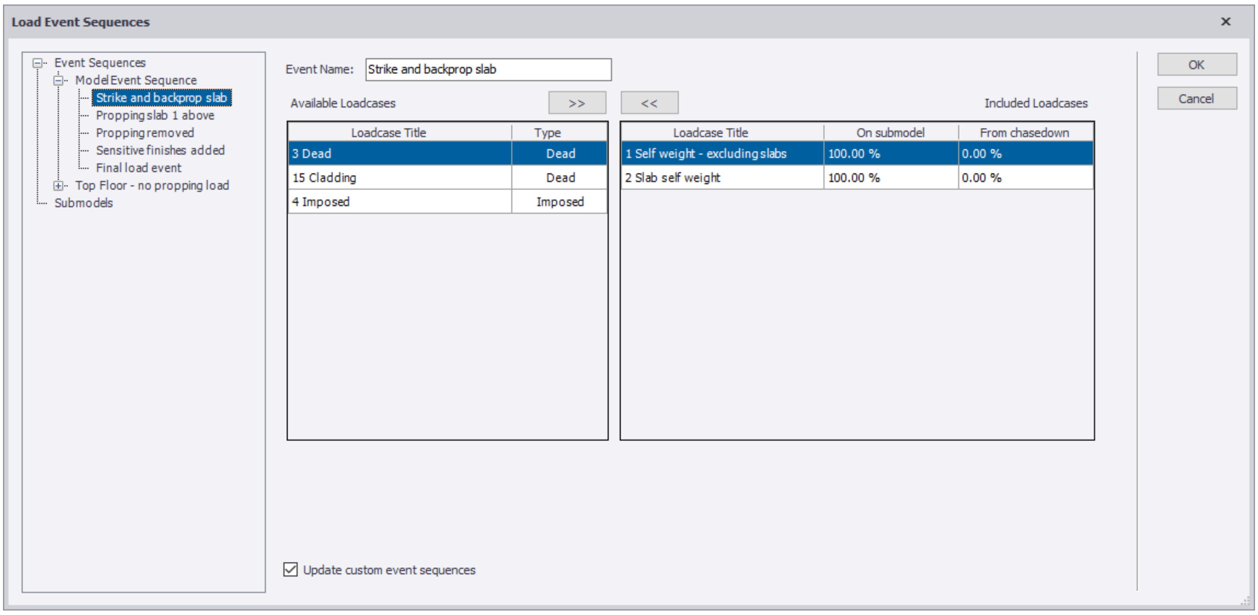
- Review the existing Load Events and edit all the quantities (except included loadcases)
- **Add** an event to the end of the sequence
- **Insert** an event above the selected event in the sequence
- **Remove** events
- **Move Up** or **Move Down** events to re-order the sequence
- **Reset** the custom event sequence to be the same as the Model Event Sequence

---

NOTE Model Event Sequences and Custom Event Sequences do not behave differently - they are both just Event Sequences.

Load Event sub-pages

Load Event sub-pages are displayed under the Model Event Sequence, and any custom Event Sequences pages. Each Load Event has a separate sub-page which is used to define the loadcases included in the Load Event.



The [>>] and [<<] buttons are used to add or remove loadcases from the Included Loadcases list.

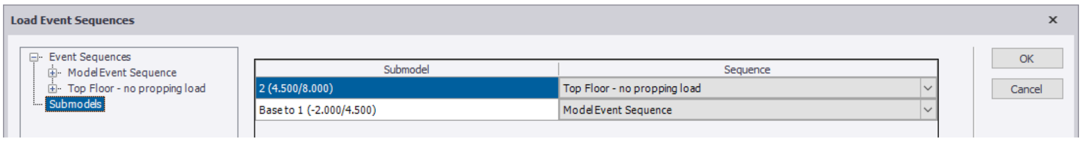
On submodel specifies the percentage of each included loadcase on the submodel.

From chasedown specifies the percentage of each included loadcase from the chasedown.

NOTE	For propping loadcases - “From Chasedown” should be 0%.

Submodels

Select **Submodels** to assign a custom event sequence to a submodel if required.



- With this row selected you can edit the section size and grade for all stacks simultaneously.

2a. Event sequence parameters table (Eurocode)

Event	Name	Load start time [y, m, d]	Beta	Temperature [°C]	Relative Humidity [%]	Number of Exposed Faces	Construction load [kN/m²]	Loadcase	On submodel	From chasedown
1	Strike and backprop slab	10d	1	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	100.00 %	0.00 %
2	Propping slab 1 above	20d	1	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	0.00 %
								2 Slab self weight	160.00 %	0.00 %
3	Propping removed	2m	0.5	20.0	50.00	2	0.500	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	50.00 %	50.00 %
4	Sensitive finishes added	4m	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	30.00 %	30.00 %
5	Final load event	70y	0.5	20.0	50.00	2	0.000	1 Self weight - excluding slabs	100.00 %	100.00 %
								2 Slab self weight	100.00 %	100.00 %
								3 Dead	100.00 %	100.00 %
								15 Cladding	100.00 %	100.00 %
								4 Imposed	100.00 %	100.00 %

The input parameters required for each event are:

- **Event Number** - Automatic
- **Event Name** - A user defined name to help explain the event
- **Load Start time** - The time at which the event takes place

- **Beta** - a coefficient to take account of the influence of the duration of loading. (See: EC2 Clause 7.4.3) This coefficient is used to account for the fact that tension stiffening effects reduce over time.

  Tension stiffening is the phenomenon that when the concrete is not fully cracked, there is still concrete in the tension zone that transfers some tensile forces, so the stiffness is greater than that of the fully cracked stiffness (and less than the uncracked stiffness). Since this effect reduces over time, to model loads with a longer duration, you want to model a lower stiffness, which means setting beta = 0.5.

  In Tekla Structural Designer, Beta defaults to 1.0 where the start event time is ≤ 30 days and 0.5 if >30 days, but may be changed for any event.

  Tension stiffening reduces over time because of increased cracking and bond failure between the steel reinforcement and concrete.

  These phenomena cannot be reversed, so reduced tension stiffening cannot be recovered.

  For this reason, if Beta is set to 0.5, and then in a later event set to 1.0, a warning flag is shown.
- **Temperature** - Used in the calculation of the Composite Modulus. The effective age of concrete is adjusted to account for the defined temperature.
- **Relative Humidity** - Used in the calculation of creep.
- **Number of Exposed Faces** - Used for the calculation of creep.
- **Construction load** - The construction load you wish to allow for at the chosen load event.
- **Loadcase** - You select the loadcases you wish to be included in the event.
- **On submodel** - The % of load to apply to the slab, applied directly to the sub-model.
- **From chasedown** - The % of load to apply to the slab, from the reactions established in analysis of the sub-models above.

2b. Event sequence parameters table (ACI)

Event	Name	Load start time [y, m, d]	Ultimate Creep Coefficient	Aging Coefficient	Number of Exposed Faces	Construction load [psf]	Loadcase	On submodel	From chasedown
1	Start event	10d	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
2	Finishes added	3m	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							4 Cladding	100.00 %	100.00 %
							3 Live	33.00 %	33.00 %
3	Final load event	5y	2.350	0.800	0	0.0	0 Self weight - excluding slabs	100.00 %	100.00 %
							1 Slab self weight	100.00 %	100.00 %
							2 Dead	100.00 %	100.00 %
							4 Cladding	100.00 %	100.00 %
							3 Live	100.00 %	100.00 %
4	Live load only	5y	2.350	0.800	0	0.0	3 Live	100.00 %	100.00 %

The input parameters required for each event are:

- **Event Number** - Automatic
- **Event Name** - A user defined name to help explain the event
- **Load Start time** - The time at which the event takes place
- **Ultimate Creep Coefficient**
- **Aging Coefficient**
- **Number of Exposed Faces** - Used for the calculation of creep.
- **Construction load** - The construction load you wish to allow for at the chosen load event.
- **Loadcase** - You select the loadcases you wish to be included in the event.
- **On submodel** - The % of load to apply to the slab, applied directly to the sub-model.
- **From chasedown** - The % of load to apply to the slab, from the reactions established in analysis of the sub-models above.

3. Update custom event sequences

If you select the **Update custom event sequences** option, changes made to the Event in the Model Event Sequence are also replicated in custom event sequences.

4. Buttons

Button	Description
OK	Saves the current reinforcement and closes the dialog box.
Cancel	Closes the dialog box without saving changes.
Add	Depends on the page selected in the Event sequences and submodels pane.
Insert	Depends on the page selected in the Event sequences and submodels pane.
Remove	Depends on the page selected in the Event sequences and submodels pane.
Move Up	Depends on the page selected in the Event sequences and submodels pane.
Move Down	Depends on the page selected in the Event sequences and submodels pane.
Reset	Depends on the page selected in the Event sequences and submodels pane.

Materials dialog

The **Materials** dialog box allows you to view material database properties and manage material databases. The databases contain an extensive range of sections, materials, reinforcement, decking and connectors for each head code and country.

NOTE Although the Materials dialog can be used to view the properties for any of the head codes, only the properties for the currently selected head code in **Model Settings** can be applied to the model.

To display the dialog:

1. Click the **Home** tab.
2. Click **Materials**.

The dialog content is described below.

Sections settings

The **Sections** page of the **Materials** dialog box allows you to view the available steel, cold formed, cold rolled, and timber sections for each head code. You can also add new user-defined sections.

Field or button	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Unit System	Allows you to select which units are used for sections.
Head Code	Allows you to select the head code whose material database properties you want to view.
Material	Allows you to select the material whose sections you want to view.
Manage Sections	Allows you to add, modify, and delete user-defined sections.
Manage Section Orders	Allows you to adjust section orders according to your needs.
Connection Resistance	Allows you to add connection resistance information for a section.
Steel Joists	Allows you to add steel joist information for a section.
Plate Dimensions	Allows you to adjust plate widths and thicknesses.

Material settings

The **Material** page of the **Materials** dialog box allows you to view properties of each grade of each material for each head code. You can also add new user-defined sections.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Head Code	Allows you to select the head code whose material database properties you want to view.
Material Type	Allows you to select the material type whose material database properties you want to view.

Button or field	Description
Add...	If necessary, allows you to add user-defined material grades to the **Available Grades** list.
View...	Allows you to view the properties of pre-defined material grades.
Delete	Allows you to permanently delete user-defined material grades.
>> Default	Allows you to make the currently highlighted material grade a default material grade, so that it appears in the **Default Grade** field.

Reinforcement settings

The **Reinforcement** page of the **Materials** dialog box allows you to view the properties of reinforcement classes and bar sizes.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Head Code	Allows you to select the head code whose material database properties you want to view.
Country	Allows you to select the country whose material database properties you want to view.
Type	Allows you to select the type whose material database properties you want to view.
Rib Type	Allows you to select the rib type whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined reinforcement classes or bar sizes to the appropriate list.
View...	Allows you to view the properties of pre-defined reinforcement classes or bar sizes.
Delete	Allows you to permanently delete user-defined reinforcement classes or bar sizes.

Button or field	Description
>> Default	Allows you to make the currently highlighted reinforcement class a default class, so that it appears in the **Default Class** field.

Decking settings

The **Decking** page of the **Materials** dialog box contains different sub pages: **Metal Decking** and **Precast Concrete Decking**. The **Metal Decking** page allows you to view the properties of different profiles and gauges, and the **Precast Concrete Decking** page allows you to view the properties of different precast concrete planks and depths. The sub pages contain the same commands.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Country	Allows you to select the country whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined profiles, gauges, planks, or depths to the appropriate list.
View...	Allows you to view the properties of pre-defined profiles, gauges, planks, or depths.
Delete	Allows you to permanently delete user-defined profiles, gauges, planks, or depths.
>> Default	Allows you to make the currently highlighted profile, gauge, plank, or depth a default option, so that it appears in the appropriate **Default** field.

Shear Connectors settings

The **Shear Connectors** page of the **Materials** dialog box allows you to view the properties of different connectors.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the material database to a newer version.
Head Code	Allows you to select the head code whose material database properties you want to view.
Country	Allows you to select the country whose material database properties you want to view.
Add...	If necessary, allows you to add user-defined sources or connectors to the appropriate list.
View...	Allows you to view the properties of pre-defined sources or connectors.
Delete	Allows you to permanently delete user-defined sources or connectors.
>> Def. Metal	Allows you to make the currently highlighted option the default option for metal, so that it appears in the **Default for Metal** field.
>> Def. Concrete	Allows you to make the currently highlighted option the default option for concrete, so that it appears in the **Default for Concrete** field.

Bolts (Rods in US) settings

The **Bolts**/**Rods** page of the **Materials** dialog box allows you to view the properties of bolt/rod classes and sizes.

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the bolts/rods database to a newer version.
Unit System	Allows you to select which units are used for bolts/rods.
Head Code	Allows you to select the head code whose material database properties you want to view.

Button or field	Description
Add...	If necessary, allows you to add user-defined bolts/rods.
Copy...	Allows you to copy an existing bolt/rod.
View...	Allows you to view the properties of the selected bolt/rod.
Delete	Allows you to permanently delete user-defined bolts/rods.
>> Default	Allows you to make the currently highlighted bolt a default, so that it appears in the Default Bolt/Rod field.

Welds settings

The **Welds** page of the **Materials** dialog box allows you to view the properties of welds leg lengths and electrodes.

NOTE Electrodes are US only

Button or field	Description
Upgrade	If the button is visible, you can click it to upgrade the welds database to a newer version.
Unit System	Allows you to select which units are used for welds.
Head Code	Allows you to select the head code whose material database properties you want to view.
Leg Lengths	Allows you to view the Leg Lengths currently in the welds database.
Defaults...	Allows you to set and view the default weld leg lengths
Reset	Allows you to reset the default weld leg lengths and delete all user defined welds.
Add	If necessary, allows you to add user-defined leg lengths to the database.
Electrodes	
Electrodes	Allows you to view the list of electrodes in the welds database.
Add	If necessary, allows you to add user-defined electrodes to the database.

Button or field	Description
View	Allows you to view the properties of pre-defined electrodes.
Delete	Allows you to permanently delete user-defined electrodes.
>> Default	Allows you to make the currently highlighted option the default electrode, so that it appears in the **Default Electrode** field.

Model settings

The **Model** page of the **Materials** dialog box allows you to both update the material databases with new properties from the model and update material properties in the model with new values from the material databases.

Button, field, or column	Description
In Database	Uses **?** to display if there are inconsistencies between the material data in the model and the material databases.
Add to Database	Allows you to update the material databases with values from the model.
Update from Database	Allows you to update the material properties in the model with values from the material databases.
Show only objects not saved in the database	Hides the model properties that are consistent with the material databases.

Model Settings dialog

The majority of settings for the current project are accessed from the **Model Settings** dialog.

Location

On the **Home** tab, click **Model Settings**.

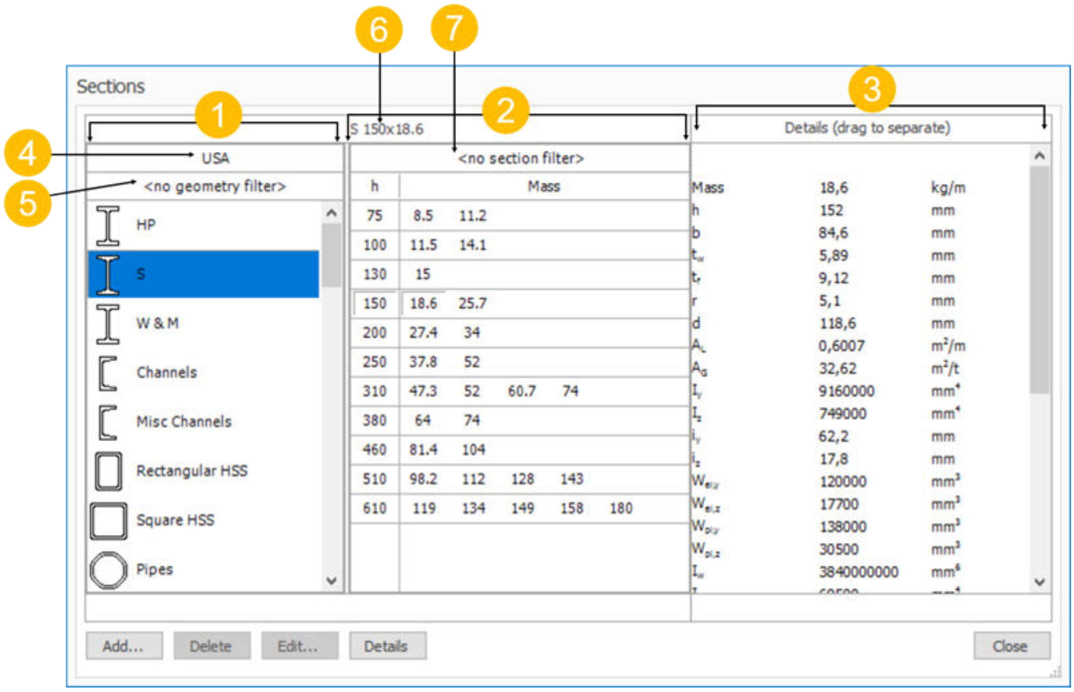
Content

Button, Command or Option	Description
Design Codes	See Design code settings (page 1033)
Units	See Unit settings (page 1034)
References	See Object reference settings (page 1035)
Loading	See Loading settings (page 1037)
Grouping	See Grouping model settings (page 1038)
Material List	See Material list settings (page 1039)
Beam Lines	See Beam lines settings (page 1039)
Analysis Model	See Analysis Model settings (page 1040)
Validation	See Validation settings (page 1042)
Live, or Imposed Load reductions	See Live/imposed load reduction settings (page 1043)
EHF, NL, or NHF (Global Imperfections)	See Global Imperfections settings (page 1044)
User Defined Attributes	See User-defined attribute settings (page 1045)
Graphics View Settings	See Graphics view settings (page 1047)
Structural BIM	See Structural BIM settings (page 1047)
OK button	Apply the changes to the current project.
Cancel button	Cancel the changes.
Save button	Save the changes back to the active settings set for future use.
Load button	Revert to the model settings specified in the active settings set.

Sections dialog

The **Sections** dialog box is used to select sections to apply to the model and also to manage the sections stored in the material database.

The dialog content is described below.



1. **Page** pane: views the different section geometries.
2. **Item** pane: views the different sizes of the selected section when you select a section geometry **Page** pane. Any user-defined sections are marked with *.
3. **Details** pane: views the details of the selected section when you click the **Details** button.
4. **Country** droplist: allows you to display the sections of each country.
5. **Geometry filter**: allows you to filter sections according to their geometry.
6. Selected section: displays the details of the currently selected section.
7. **Section filter**: allows you to filter sections according to their details.

You can also use the following buttons to manage the sections:

- **Add...**: allows you to add a user-defined section to the material database.
- **Delete**: allows you to delete a user-defined section.
- **Edit...**: allows you to modify the properties of a user-defined section.
- **Details**: opens the **Details** pane and displays the properties of the currently selected section.

Settings dialog

The **Settings** dialog allows you to manage defaults, collectively referred to as a settings set, that are used in future projects. In addition, you can manage general and display settings that are applied instantly to the current work session.

Location

On the **Home** tab, click **Settings**.

Content

List, Page, or Button	Description
Select the settings set to edit list, **Available settings sets** list	Allows you to select the settings set to view or modify.
Settings Sets	See Settings set settings (page 1160)
General	See General settings (page 1160)
Results Viewer	See Results Viewer settings (page 1162)
Report	See Report settings (page 1168)
Units	See Unit settings (page 1034)
Design Codes	See Design code settings (page 1033)
Design Settings	See Design Settings (page 1064)
Analysis Settings	See Analysis Settings (page 1050)
Loading	See Loading settings (page 1037)
Structure Defaults	See Structure default settings (page 1164)
Section Order Defaults	See Section order default settings (page 1164)
References	See Object reference settings (page 1035)
Drawings	See Drawing settings (page 1126)
Material List	See Material list settings (page 1039)
Beam Lines	See Beam lines settings (page 1039)
Analysis Model	See Analysis Model settings (page 1040)
Solver	See Solver settings (page 1164)
User Defined Attributes	See User-defined attribute settings (page 1045)
Scene	See Scene settings (page 1165)
Structural BIM	See Structural BIM settings (page 1047)
Slab Deflection	See Slab deflection settings (page 1122)
Embodied Carbon	See Embodied carbon settings (page 1170)
Performance	See Performance settings (page 1171)
OK button	Apply the changes.

List, Page, or Button	Description
Cancel button	Cancel the changes.

Slab Deflection Check Catalogue

The **Slab Deflection Check Catalogue** is used to define the deflection checks that are applied to check lines.

Location

On the **Slab Deflection** tab, click **Deflection Checks**.

Content

Each deflection check has a unique name and can either be defined as a total or instantaneous check for a specific event, or a differential check between load events. A deflection limit is set and you can specify if the check is to be applied to each new check line as it is defined.

Field or button	Description
Name	Allows you to modify the check name.
Type	Allows you to select the deflection check type, see Display slab deflection analysis results (page 818) .
Start Event	Allows you to select the start event for a differential check.
Event	Allows you to select the event to which the check applies.
Deflection Limit	Allows you to specify the deflection check limit.
Use in new Check Lines	Check the box only to be automatically apply the check to new check lines as they are created.
Add	Allows you to add a new row in the table for defining a new check.
Remove	Removes the selected check from the table.

Snow wizard (Eurocode)

Summary

You can use the **Snow wizard...** to define sufficient site information to calculate the snow loadcases.

Location

1. Ensure the head code is set as Eurocode with the required National Annex.
2. On the **Load** tab, click **Snow Load** --> **Snow wizard...**

Plain Eurocode, Ireland and Sweden National Annex

Button, command or option	Description	
Page 1 - Basic data (Plain Eurocode, Ireland and Sweden National Annex)	The following basic data is required:	
Exposure Coefficient	User defined value. (Default 1.0)	
Thermal Coefficient	User defined value. (Default 1.0)	
Coefficient for Exceptional Snow Loads	User defined value. (Default 1.0)	
Snow Density	Define the density of snow. (Default 2.0 kN/m^2)	
Snow Load	User defined value. (No default value)	
Next	Click to proceed to the next page of the **Snow wizard...**	
Page 2 - Snow loadcases (Plain Eurocode, Ireland and Sweden National Annex)	On this page you specify the design situations to be considered.	
Loadcase	Automatically generated wind loadcase title. The loadcase title can be manually renamed if required.	
Apply	Options are: • Checked If checked the loadcase will be created.	

Button, command or option	Description	
	• Unchecked If unchecked the loadcase will not be created. **NOTE** Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3) • Loads in these cases are generated automatically Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4) • You must create the loads in these cases manually **NOTE** The EC/NA recommends which loadcases to apply.	
Loadcase Type (Plain Eurocode and Ireland National Annex)	Each loadcase has a loadcase type which can be: • Snow • Snow > 1000m • Snow Drift Refer to EC 1991-1-3 Annex A, Table A1	
Loadcase Type (Sweden National Annex)	Each loadcase has a loadcase type which can be: • Snow • Snow S_k > 2 kN/m^2 • Snow S_k > 3 kN/m^2 • Snow Drift Refer to EC 1991-1-3 Annex A, Table A1.	
Number of wind directions to be considered for drifted snow	You can select between 1 and 4 separate loadcases to be generated for each 'Drifted' loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name). For example if you have checked the box to apply loadcase "Snow Load - Case	

Button, command or option	Description
	A - 2) Drifted" and selected 2 wind directions, the following loadcases will be created: • Snow Load - Case A - 2) Drifted 1 • Snow Load - Case A - 2) Drifted 2
Finish	Completes the **Snow wizard...** and sets up the loadcases based upon the input.

UK National Annex

Button, command or option	Description
Page 1 - Basic data (UK National Annex)	The following basic data is required:
Exposure Coefficient	User defined value. (Default 1.0)
Thermal Coefficient	User defined value. (Default 1.0)
Coefficient for Exceptional Snow Loads	User defined value. (Default 1.0)
Snow Density	Define the density of snow. (Default 2.0 kN/m^2)
Snow Load	
Zone Number, Z	Zone Number. (Default 2.0 kN/m^2)
Altitude, A	Altitude. (Default 1.0)
Characteristic Ground Snow Load, s_k	Characteristic ground snow load calculated as $s_k = (0.15 + (0.1 * Z + 0.05)) + ((A - 100)/525)$
Next	Click to proceed to the next page of the **Snow wizard...**
Page 2 - Snow loadcases (UK National Annex)	On this page you specify the design situations to be considered.

Button, command or option	Description	
Loadcase	Automatically generated wind loadcase title. The loadcase title can be manually renamed if required.	
Apply	Options are: • Checked If checked the loadcase will be created. • Unchecked If unchecked the loadcase will not be created. **NOTE** Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3) • Loads in these cases are generated automatically Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4) • You must create the loads in these cases manually **NOTE** The EC/NA recommends which loadcases to apply.	
Loadcase Type	Each loadcase has a loadcase type which can be: • Snow • Snow > 1000m • Snow Drift Refer to EC 1991-1-3 Annex A, Table A1.	
Number of wind directions to be considered for drifted snow	You can select between 1 and 4 separate loadcases to be generated for each 'Drifted' loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name). For example if you have checked the box to apply loadcase "Snow Load - Case A - 2) Drifted" and selected 2 wind	

Button, command or option	Description	
	directions, the following loadcases will be created: • Snow Load - Case A - 2) Drifted 1 • Snow Load - Case A - 2) Drifted 2	
Finish	Completes the **Snow wizard...** and sets up the loadcases based upon the input.	

Finland National Annex

Button, command or option	Description	
Page 1 - Basic data (Finland National Annex)	The following basic data is required:	
Topography	Options are: • Windswept • Normal (Default) • Sheltered	
Exposure Coefficient	Value determined from Topography.	
Thermal Coefficient	User defined value. (Default 1.0)	
Coefficient for Exceptional Snow Loads	User defined value. (Default 1.0)	
Snow Weight Density	Define the density of snow. (Default 2.0 kN/m^2)	
Characteristic Ground Snow Load	User defined value. (No default value)	
Next	Click to proceed to the next page of the **Snow wizard...**	
Page 2 - Snow loadcases (Finland National Annex)	On this page you specify the design situations to be considered.	

Button, command or option	Description	
Loadcase	Automatically generated wind loadcase title. The loadcase title can be manually renamed if required.	
Apply	Options are: • Checked If checked the loadcase will be created. • Unchecked If unchecked the loadcase will not be created. **NOTE** Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3) • Loads in these cases are generated automatically Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4) • You must create the loads in these cases manually **NOTE** The EC/NA recommends which loadcases to apply.	
Loadcase Type	Each loadcase has a loadcase type which can be: • Snow • Snow, S_k > 2.75 kN/m^2 • Snow Drift • Ice Refer to EC 1992-1-3 Annex A, Table A1.	
Number of wind directions to be considered for drifted snow	You can select between 1 and 4 separate loadcases to be generated for each 'Drifted' loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name). For example if you have checked the box to apply loadcase "Snow Load - Case	

Button, command or option	Description
	A - 2) Drifted" and selected 2 wind directions, the following loadcases will be created: • Snow Load - Case A - 2) Drifted 1 • Snow Load - Case A - 2) Drifted 2
Finish	Completes the **Snow wizard...** and sets up the loadcases based upon the input.

Norway National Annex

Button, command or option	Description
Page 1 - Basic data (Norway National Annex)	The following basic data is required:
Topography	Options are: • Windswept • Normal (Default) • Sheltered
Exposure Coefficient	Value determined from Topography.
Thermal Coefficient	User defined value. (Default 1.0)
Coefficient for Exceptional Snow Loads	User defined value. (Default 1.0)
Snow Weight Density	Define the density of snow. (Default 2.0 kN/m^2)
Altitude, H	User defined value.
Basic Reference Altitude	User defined value.
Basic Snow Load	User defined value.
????	User defined value.
????	User defined value.
Characteristic Ground Snow Load	Automatically determined from the above values.

Button, command or option	Description
Next	Click to proceed to the next page of the **Snow wizard...**
Page 2 - Snow loadcases (Norway National Annex)	On this page you specify the design situations to be considered.
Loadcase	Automatically generated wind loadcase title. The loadcase title can be manually renamed if required.
Apply	Options are: • Checked If checked the loadcase will be created. • Unchecked If unchecked the loadcase will not be created. **NOTE** Undrifted (Cases A-1, B1-1, B1-3, B2-1, B3-1, B3-3) • Loads in these cases are generated automatically Drifted Snow Load (Cases A-2, B1-2, B1-4, B2-2, B2-3, B3-2, B3-4) • You must create the loads in these cases manually **NOTE** The EC/NA recommends which loadcases to apply.
Loadcase Type	Each loadcase has a loadcase type which can be: • Snow • Snow > 1000m • Snow Drift Refer to EC 1992-1-3 Annex A, Table A1.
Number of wind directions to be	You can select between 1 and 4 separate loadcases to be generated for

Button, command or option	Description	
considered for drifted snow	each 'Drifted' loadcase (identified by....1, ...2, etc. appended to the drifted loadcase name). For example if you have checked the box to apply loadcase "Snow Load - Case A - 2) Drifted" and selected 2 wind directions, the following loadcases will be created: • Snow Load - Case A - 2) Drifted 1 • Snow Load - Case A - 2) Drifted 2	
Finish	Completes the **Snow wizard...** and sets up the loadcases based upon the input.	

Snow wizard (ASCE7)

Summary

You can use the **Snow wizard...** to define sufficient site information to calculate the snow loadcases.

Location

1. Ensure the snow loading code is set as ASCE7.
2. On the **Load** tab, click **Snow Load** --> **Snow wizard...**

Content

Button, command or option	Description	
Page 1 - Basic data (ASCE7 Snow wizard)	The following basic data is required:	
Ground snow load	(No default)	
Terrain Category	Options are: • B (urban, suburban, wooded) - Default • C (open terrain scattered obstructions),	

Button, command or option	Description
	• D (flat unobstructed areas), Above treeline, Alaska no trees
Exposure	Options are: • Fully exposed • Partially exposed
Exposure factor, C_e	C_e is determined from the following table: <table><tr><th></th><th>Fully Exposed</th><th>Fully Exposed</th><th>Sheltered</th></tr><tr><td>B</td><td>0.9</td><td>1</td><td>1.2</td></tr><tr><td>C</td><td>0.9</td><td>1</td><td>1.1</td></tr><tr><td>D</td><td>0.8</td><td>0.9</td><td>1</td></tr><tr><td>Above treeline</td><td>0.7</td><td>0.8</td><td>0</td></tr><tr><td>Alaska</td><td>0.7</td><td>0.8</td><td>0</td></tr></table>
Thermal Condition	Option are: • All except below • Structures kept just above freezing • Unheated open air structures • Structures kept below freezing • Continuously heated greenhouses
Thermal factor, C_t	C_t is determined from the following table: <table><tr><td>A</td><td>1</td></tr><tr><td>B</td><td>1.1</td></tr><tr><td>C</td><td>1.2</td></tr><tr><td>D</td><td>1.3</td></tr><tr><td>E</td><td>0.85</td></tr></table>

<table>
<tr><th>Button, command or option</th><th>Description</th></tr>
<tr><td>Risk Category</td><td>Option are:
• I
• II
• III
• IV</td></tr>
<tr><td>Snow Importance factor, I_s</td><td>I_s is determined from the following table:<table><tr><td>I</td><td>0.8</td></tr><tr><td>II</td><td>1</td></tr><tr><td>III</td><td>1.1</td></tr><tr><td>IV</td><td>1.2</td></tr></table></td></tr>
<tr><td>Flat Roof Snow load, , p_f</td><td>p_f is determined from the following:
$p_f = 0.7 * C_e * C_t * I_s * p_g$</td></tr>
<tr><td>Next</td><td>Takes you to the next page.</td></tr>
<tr><td>Page 2 - Snow loadcases (AISC Snow wizard)</td><td>This page is used to specify the loadcases as follows:</td></tr>
<tr><td>Minimum Snow Load</td><td>Option are:
• Yes
• No</td></tr>
<tr><td>Balanced Snow Load</td><td>Option are:
• Yes
• No</td></tr>
<tr><td>Unbalanced Snow Load</td><td>Option are:
• No
• 1
• 2
• 3
• 4
You can select between 1 and 4 cases to generate (1 Unbalanced..., 2 Unbalanced, etc.)</td></tr>
</table>

Button, command or option	Description
Draft Snow Load	Option are: • No • 1 • 2 • 3 • 4 You can select between 1 and 4 cases to generate (1 Unbalanced..., 2 Unbalanced, etc.)
Rain on Snow Surcharge	Option are: • Yes • No
Finish	Completes the **Snow wizard...** and sets up the loadcases based upon the input.

Sub Models dialog

The **Sub Models** dialog box allows you to split the structure into a continuous series of sub models, working from the top of the building down to, and including the foundations.

Location

In the **Structure** tree, double-click **Sub Models**.

Content

Button, Command or Option	Description
Options	
Level	Before any analysis has been performed, only two levels are displayed, one at a distance (2m) above the highest construction level and a second at the same distance below the lowest level. These cannot be changed. Hence, at this point, there is a single sub-model comprising the whole structure.

Button, Command or Option	Description	
Active	Only active levels act to divide the structure into sub-models. The top and bottom levels must always remain active, as there must always be at least one sub model. Once intermediate levels have been inserted, you can choose to inactivate them if required. When this occurs the sub-models immediately above and below the level in question are merged into a single sub-model.	
Auto Generate	If any adjustments to the cutting planes or cutting plane levels, or any of them are set as inactive you must also clear Auto-Generate. Otherwise the changes you have made will be lost the next time the sub models are generated.	
Buttons		
OK	Allows you to save any changes made.	
Cancel	Allows you to discard any changes made.	
Insert Above	Allows you to insert a new sub model between the selected level and the one above. It defaults to being located half way between the two, however this can be edited manually, provided it remains between the two reference levels.	
Insert Below	Allows you to insert a new sub model between the selected level and the one below. It defaults to being located half way between the two, however this can be edited manually, provided it remains between the two reference levels.	
Delete		
Generate	Allows you to auto generate default sub models or not • Checked Auto Generate creates default sub models for every level specified as a floor in the Construction Levels dialog. It is optional as default sub models are automatically generated for you when you run the analysis, provided the Auto	

Button, Command or Option	Description	
	Generate box on the dialog is checked. You may choose to do this if you want to review the sub models prior to the first run of the analysis. • Unchecked Does not Auto Generate sub models prior to the first run of the analysis. **NOTE** The Generate button can also be used to revert back to the default sub models at any time.	

See also

Manage sub models (page 488)

Create sub models (page 489)

Slab Deflection Settings dialog

The **Slab Deflection Settings** dialog and its subpages allow you to adjust the settings applied to the different analyses.

Location

On the **Slab Deflection** tab, click **Settings**.

Content

Button, command or option	Description
• New Load Event Defaults • New Check Defaults • Aging, Creep & Shrinkage • Modification Factors • Iterative Cracked Section Analysis	See: Slab deflection settings
OK button	Apply the changes to the current project.
Cancel button	Cancel the changes.
Save button	Save the changes back to the active settings set for future use.

Button, command or option	Description
Load button	Revert to the model settings specified in the active settings set.

Index

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

R

S

T

U

V

W

Z