



# Tekla Structures

## Analysis Guide



Product version 21.0  
March 2015

©2015 Tekla Corporation

# Contents

<b>1</b>	<b>Getting started with analysis.....</b>	<b>7</b>
1.1	What is an analysis model.....	7
	Analysis model objects.....	9
1.2	About analysis applications.....	12
1.3	Linking Tekla Structures with an analysis application.....	12
1.4	Structural analysis workflow in Tekla Structures.....	13
<b>2</b>	<b>Creating and grouping loads.....</b>	<b>15</b>
2.1	Load types.....	15
2.2	Setting the load modeling code.....	17
	Using non-standard load combination factors.....	17
2.3	Grouping loads.....	18
	Creating and modifying a load group.....	19
	Setting the current load group.....	19
	Load group compatibility.....	20
	Deleting a load group.....	21
2.4	Creating loads.....	21
	Defining the properties of a load.....	22
	Load magnitude.....	23
	Load form.....	24
	Creating a point load.....	24
	Creating a line load.....	25
	Creating an area load.....	26
	Creating a uniform load.....	26
	Creating a temperature load or a strain.....	27
	Creating wind loads.....	28
	Wind load examples.....	28
<b>3</b>	<b>Distributing and modifying loads.....</b>	<b>31</b>
3.1	Attaching loads to parts or locations.....	31
3.2	Applying loads to parts.....	32
	Defining load-bearing parts by name.....	32
	Defining load-bearing parts by selection filter.....	33
	Bounding box of a load.....	34
3.3	Changing the loaded length or area of a load.....	34
3.4	Modifying the distribution of a load.....	35
3.5	Modifying the location or layout of a load.....	37
3.6	Moving a load end or corner using handles.....	39
<b>4</b>	<b>Working with loads and load groups.....</b>	<b>40</b>

4.1	Scaling loads in model views.....	40
4.2	Checking loads and load groups.....	41
	Checking a load.....	41
	Finding out to which load group a load belongs.....	42
	Finding out which loads belong to a load group.....	43
	Checking loads using reports.....	43
4.3	Moving loads to another load group.....	44
4.4	Exporting load groups.....	45
4.5	Importing load groups.....	45
5	Creating analysis models.....	47
5.1	Defining basic properties for an analysis model.....	47
5.2	Including objects in analysis models.....	48
	Analysis model creation method.....	49
	Analysis model filter.....	49
5.3	Selecting the analysis application.....	50
5.4	Creating an analysis model.....	50
	Creating an analysis model of entire physical and load models.....	51
	Creating an analysis model for specific parts and loads.....	51
	Creating a modal analysis model.....	52
6	Modifying analysis models.....	53
6.1	Checking objects included in an analysis model.....	53
6.2	Modifying the properties of an analysis model.....	54
	Changing the creation method of an analysis model.....	54
	Defining the axis settings of an analysis model.....	55
	Defining seismic loads for an analysis model.....	56
	Defining modal masses for an analysis model.....	56
	Defining the design properties of an analysis model.....	58
	Defining analysis model rules.....	58
	Adding an analysis model rule.....	58
	Organizing analysis model rules.....	60
	Testing analysis model rules.....	60
	Deleting analysis model rules.....	61
	Saving analysis model rules.....	62
6.3	Adding objects to an analysis model.....	62
6.4	Removing objects from an analysis model.....	63
6.5	Creating an analysis node.....	63
	Analysis node colors.....	64
6.6	Creating a rigid link.....	65
6.7	Merging analysis nodes.....	65
6.8	Copying an analysis model.....	67
6.9	Deleting an analysis model.....	67
7	Modifying analysis parts.....	68
7.1	Defining and modifying analysis part properties.....	68
	Modifying the properties of an analysis part.....	69

<b>7.2</b>	<b>Defining support conditions.....</b>	<b>70</b>
	Defining the support conditions of a part end.....	71
	Defining the support conditions of a plate.....	72
	Support condition symbols.....	72
<b>7.3</b>	<b>Defining design properties for analysis parts.....</b>	<b>74</b>
	Omitting analysis parts from design.....	75
	Defining the buckling lengths of a column.....	75
	Kmode options.....	76
<b>7.4</b>	<b>Defining the location of analysis parts.....</b>	<b>77</b>
	Defining or modifying the axis location of an analysis part.....	78
	Defining offsets for an analysis part.....	79
	Resetting the editing of analysis parts.....	79
<b>7.5</b>	<b>Copying an analysis part.....</b>	<b>80</b>
<b>7.6</b>	<b>Deleting an analysis part.....</b>	<b>80</b>
<b>8</b>	<b>Combining loads.....</b>	<b>82</b>
<b>8.1</b>	<b>About load combinations.....</b>	<b>82</b>
<b>8.2</b>	<b>Creating load combinations automatically.....</b>	<b>83</b>
<b>8.3</b>	<b>Creating a load combination.....</b>	<b>84</b>
<b>8.4</b>	<b>Modifying a load combination.....</b>	<b>85</b>
<b>8.5</b>	<b>Copying load combinations between analysis models.....</b>	<b>85</b>
	Saving load combinations for later use.....	86
	Copying load combinations from another analysis model.....	86
<b>8.6</b>	<b>Deleting load combinations.....</b>	<b>87</b>
<b>9</b>	<b>Working with analysis and design models.....</b>	<b>88</b>
<b>9.1</b>	<b>Checking warnings about an analysis model.....</b>	<b>88</b>
<b>9.2</b>	<b>Exporting an analysis model.....</b>	<b>89</b>
<b>9.3</b>	<b>Merging Tekla Structures analysis models with analysis applications.....</b>	<b>90</b>
	Merging Tekla Structures analysis models with SAP2000.....	90
	Merging a Tekla Structures analysis model with a model in SAP2000.....	91
	Resetting merged analysis models.....	92
<b>9.4</b>	<b>Saving analysis results.....</b>	<b>92</b>
	Saving analysis results as user-defined attributes of parts.....	93
<b>9.5</b>	<b>Viewing the analysis results of a part.....</b>	<b>93</b>
<b>9.6</b>	<b>Showing analysis class in model views.....</b>	<b>94</b>
<b>9.7</b>	<b>Showing analysis bar, member, and node numbers.....</b>	<b>94</b>
<b>9.8</b>	<b>Showing the utilization ratio of parts.....</b>	<b>95</b>
<b>10</b>	<b>Analysis and design settings.....</b>	<b>97</b>
<b>10.1</b>	<b>Load group properties.....</b>	<b>97</b>
<b>10.2</b>	<b>Load properties.....</b>	<b>98</b>
	Point load properties.....	99
	Line load properties.....	99
	Area load properties.....	100
	Uniform load properties.....	101
	Temperature load properties.....	101

	Wind load properties.....	102
	Load panel settings.....	103
<b>10.3</b>	<b>Load combination properties.....</b>	<b>105</b>
	Load modeling code options.....	105
	Load combination factors.....	105
	Load combination types.....	106
<b>10.4</b>	<b>Analysis model properties.....</b>	<b>107</b>
<b>10.5</b>	<b>Analysis part properties.....</b>	<b>113</b>
	Analysis class options and colors.....	123
	Analysis axis options.....	125
<b>10.6</b>	<b>Analysis node properties.....</b>	<b>126</b>
<b>10.7</b>	<b>Analysis rigid link properties.....</b>	<b>128</b>
<b>10.8</b>	<b>Analysis bar position properties.....</b>	<b>129</b>
<b>10.9</b>	<b>Analysis area position properties.....</b>	<b>130</b>
<b>10.10</b>	<b>Analysis area edge properties.....</b>	<b>130</b>
<b>11</b>	<b>Disclaimer.....</b>	<b>132</b>



# 1 Getting started with analysis

This section explains some basic concepts and procedures you need to know to get started with structural analysis in Tekla Structures.

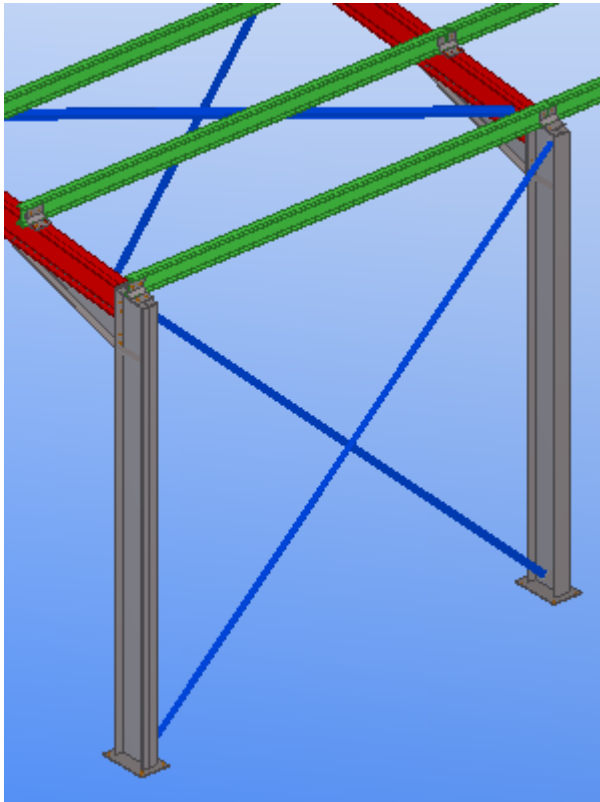
Click the links below to find out more:

- [What is an analysis model on page 7](#)
- [About analysis applications on page 12](#)
- [Linking Tekla Structures with an analysis application on page 12](#)
- [Structural analysis workflow in Tekla Structures on page 13](#)

## 1.1 What is an analysis model

When you use Tekla Structures to model, analyze, and design structures, you will become familiar with the following concepts:

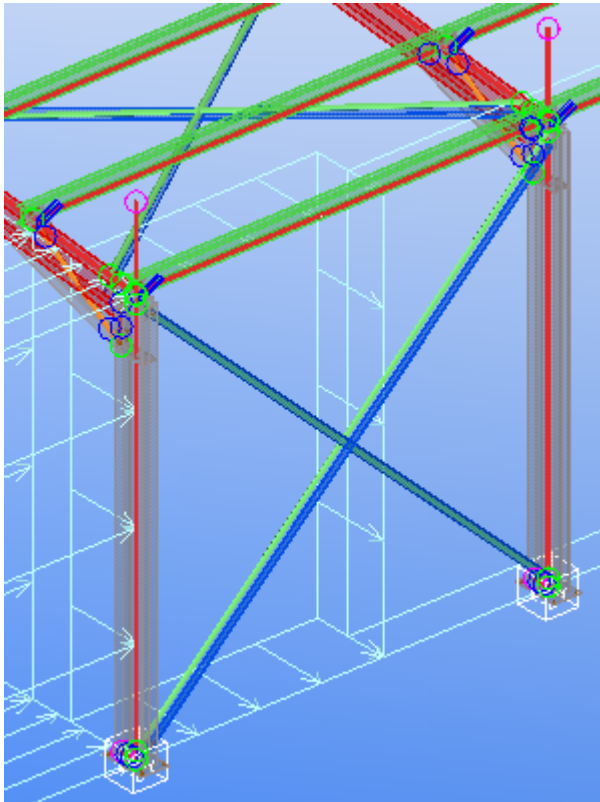
A *physical model* is a structural 3D model that includes the parts you create using Tekla Structures, and information related to them. Each part in the physical model will exist in the completed structure.



The *load model* contains information about loads and load groups that act on physical model parts. It also contains information about the building code that Tekla Structures uses in the load combination process.

An *analysis model* is a structural model that is created from a physical model. It is used for analyzing structural behavior and load bearing, and for design.





When you create an analysis model, Tekla Structures generates the following analysis objects and includes them in the analysis model:

- Analysis parts, bars, members, and areas of the physical parts
- Analysis nodes
- Support conditions for nodes
- Rigid links between the analysis parts and nodes
- Loads to analysis parts

The analysis model also includes load combinations.

**See also** [Analysis model objects on page 9](#)

What is a 3D model

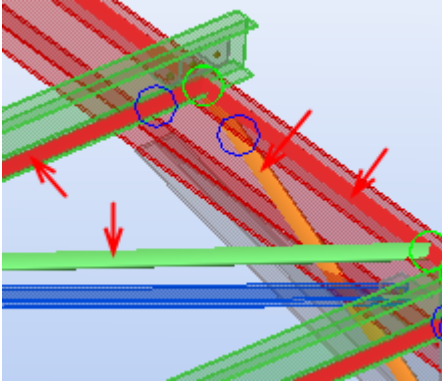
Creating parts

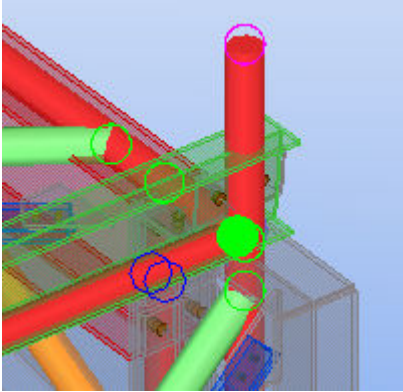
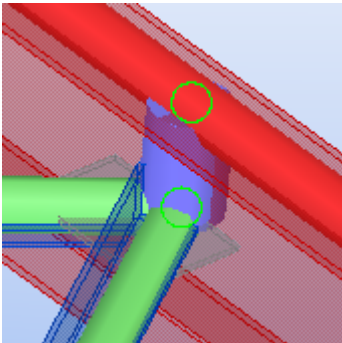
[Creating loads on page 21](#)

[Creating analysis models on page 47](#)

## Analysis model objects

Analysis model objects are model objects that Tekla Structures creates from physical model objects or on the basis of analysis part connectivity into an analysis model.

Object	Description
<p>Analysis part</p> 	<p>A representation of a physical part in an analysis model.</p> <p>In different analysis models, a physical part is represented by different analysis parts.</p>
<p>Analysis bar</p>	<p>An analysis object that Tekla Structures creates from a physical part (beam, column, or brace) or from a part segment.</p> <p>Tekla Structures creates more than one analysis bar from a physical part if:</p> <ul style="list-style-type: none"> <li>• The part is a polybeam</li> <li>• The part cross section changes non-linearly</li> </ul> <p>An analysis bar consists of one or more analysis members.</p>
<p>Analysis member</p>	<p>An analysis object that Tekla Structures creates between two nodes.</p> <p>Tekla Structures creates more than one analysis member from an analysis bar if the bar intersects with other bars and needs to be split.</p> <p>Every physical part that you include in an analysis model produces one or more analysis members. A single physical part produces several analysis members if the physical part intersects with other physical parts. Tekla Structures splits the physical part at the intersection points of the analysis axes. For example, a physical model beam that supports two other beams is split into three analysis members between nodes.</p>
<p>Analysis area</p>	<p>An analysis object that represents a plate, slab, or panel in an analysis model.</p>
<p>Analysis element</p>	<p>An analysis object that the analysis application creates from an analysis area.</p> <p>The analysis application creates an element mesh that includes several analysis elements.</p>

Object	Description
<p>Analysis node</p> 	<p>An analysis object that Tekla Structures creates at a defined point in an analysis model on the basis of analysis part connectivity.</p> <p>Tekla Structures creates analysis nodes at:</p> <ul style="list-style-type: none"> <li>• The ends of members</li> <li>• The intersection points of analysis axes</li> <li>• The corners of elements</li> </ul>
<p>Rigid link</p> 	<p>An analysis object that connects two analysis nodes so that they do not move in relation to each other.</p> <p>Rigid links have the following properties in Tekla Structures analysis models:</p> <ul style="list-style-type: none"> <li>• Profile = PL300.0*300.0</li> <li>• Material = RigidlinkMaterial</li> <li>• Density = 0.0</li> <li>• Modulus of elasticity = <math>100 \cdot 10^9 \text{ N/m}^2</math></li> <li>• Poisson's ratio = 0.30</li> <li>• Thermal dilatation coefficient = 0.0 1/K</li> </ul> <p>The analysis application that you use may model rigid links by dedicated rigid link objects.</p>
<p>Rigid diaphragm</p>	<p>An analysis object that connects more than two analysis nodes that move with exactly the same rotation and translation.</p>

Some analysis applications work on analysis members whereas others work on analysis bars. This also affects how analysis models are shown in Tekla Structures model views. Either member numbers or bar numbers are shown.

**See also** [Creating an analysis node on page 63](#)

[Creating a rigid link on page 65](#)

[Modifying analysis parts on page 68](#)

[Including objects in analysis models on page 48](#)

[Showing analysis bar, member, and node numbers on page 94](#)

## 1.2 About analysis applications

An *analysis application* is an external analysis and design software that you use with Tekla Structures to analyze and design structures.

The analysis application calculates the forces, moments, and stresses on the structures. It also calculates the displacements, deflections, rotations, and warping of objects under various loading conditions.

Tekla Structures links with a number of analysis applications and also supports export with them in several formats. The analysis application in which you run structural analysis uses data from the Tekla Structures analysis models to generate analysis results.

To analyze Tekla Structures analysis models with an analysis application, you need to install a direct link between Tekla Structures and the analysis application.

**See also** [Linking Tekla Structures with an analysis application on page 12](#)

[Selecting the analysis application on page 50](#)

Analysis and design systems

## 1.3 Linking Tekla Structures with an analysis application

To use an external analysis application with Tekla Structures analysis models, you need to install a direct link between Tekla Structures and the analysis application.

Before you start, ensure that you have:

- Access to the Tekla User Assistance service
- Access to the Tekla Extranet
- Administrator rights to your computer

To link Tekla Structures with an analysis application:

1. Log in to your computer as an administrator.
2. Install Tekla Structures if you do not already have it installed.
3. Install the analysis application if you do not already have it installed.
4. Log in to the [Tekla User Assistance](#) service and browse for the link installation instructions in **Support Articles --> Analysis and Design**.
5. Click an appropriate article, for example, **Linking Tekla Structures with SAP2000**.
6. Follow the instructions in the support article to download the link for the analysis application from the Tekla Extranet.
7. If needed, install the IFC and CIS/2 formats as advised in the support article.



If you need to uninstall and reinstall Tekla Structures and/or the analysis application for some reason, you will also need to reinstall the link after installing Tekla Structures and/or the analysis application.

---

See also

[About analysis applications on page 12](#)

## 1.4 Structural analysis workflow in Tekla Structures

Here is one example of the steps you may need to take when you analyze structures using Tekla Structures. Depending on your project, some of the steps may not be needed, some may be repeated or carried out in a different order.

Before you start, create the main load-bearing parts that you need to analyze. There is no need to detail or create connections at this stage. If you have a detailed model, or more parts in the physical model than you need to analyze, you can exclude these parts from the analysis.

To carry out a structural analysis in Tekla Structures:

1. Set the load modeling code.
2. Create load groups.
3. Create loads.
4. Define the basic analysis model properties.
5. If you do not want to create an analysis model of the entire physical and load models, define which objects to include in the analysis model.
6. Select the analysis application.
7. Create a new analysis model using the default analysis model properties.
8. Check the analysis model and analysis parts in a Tekla Structures model view.
9. Export the analysis model to the analysis application so as to run a test in the analysis application.
10. If needed, modify the analysis model or analysis parts or their properties. For example, you can:
  - Define the support conditions for analysis parts, and for connections if you have them.
  - Define other analysis properties for individual analysis parts.
  - Define design properties.
  - Add, move, and merge analysis nodes.
  - Create rigid links.

- Add or remove parts and/or loads.
11. If needed, create alternative or sub-analysis models.
  12. Create load combinations.
  13. Export the analysis model to the analysis application and run the analysis.
  14. If needed, add special loads and other required settings in the analysis application.
  15. If needed, use the analysis application to postprocess the analysis model or analysis results. For example, you can change part profiles.
  16. Import the analysis results to Tekla Structures, examine them, and use them, for example, in connection design.
  17. If the analysis results required changes in part profiles, import the changes to Tekla Structures.

**See also** [Setting the load modeling code on page 17](#)

[Creating and modifying a load group on page 19](#)

[Creating loads on page 21](#)

[Defining basic properties for an analysis model on page 47](#)

[Including objects in analysis models on page 48](#)

[Selecting the analysis application on page 50](#)

[Creating an analysis model on page 50](#)

[Checking objects included in an analysis model on page 53](#)

[Modifying analysis models on page 53](#)

[Modifying analysis parts on page 68](#)

[Defining support conditions on page 70](#)

[Combining loads on page 82](#)

[Exporting an analysis model on page 89](#)

[Saving analysis results on page 92](#)

[Viewing the analysis results of a part on page 93](#)

# 2 Creating and grouping loads

This section explains how to create and group loads in Tekla Structures.

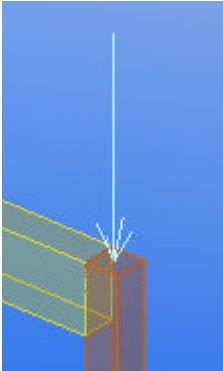
Click the links below to find out more:

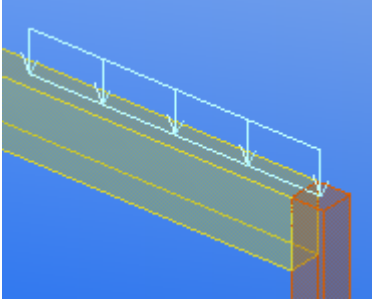
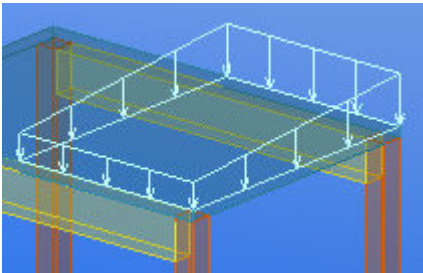
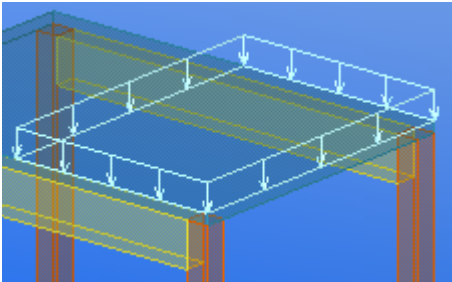
- [Load types on page 15](#)
- [Setting the load modeling code on page 17](#)
- [Grouping loads on page 18](#)
- [Creating loads on page 21](#)

## 2.1 Load types

You can create different types of loads in a Tekla Structures model.

Tekla Structures includes the following load types:

Load type	Description
<div>Point load</div>  A 3D diagram showing a vertical blue arrow pointing downwards at the end of a horizontal structural member. The member is represented by a yellow rectangular prism with a blue outline. The background is a solid blue color.	A concentrated force or bending moment that can be attached to a part.

Load type	Description
<p>Line load</p> 	<p>A linearly-distributed force or torsion. By default it runs from a point to another point. You can also create a line load with offsets from the points. A line load can be attached to a part. Its magnitude can vary linearly across the loaded length.</p>
<p>Area load</p> 	<p>A linearly-distributed force bounded by a triangle or quadrangle. You do not have to bind the boundary of the area to parts.</p>
<p>Uniform load</p> 	<p>A uniformly-distributed force bounded by a polygon. You do not have to bind the polygon to parts. Uniform loads can have openings.</p>
<p>Wind load</p>	<p>Area loads defined by pressure factors, along the height of and on all sides of a building.</p>
<p>Temperature load</p>	<ul style="list-style-type: none"> <li>• A uniform change in temperature that is applied to specified parts and that causes axial elongation in parts.</li> <li>• A temperature difference between two surfaces of a part that causes the part to bend.</li> </ul>
<p>Strain</p>	<p>An initial axial elongation or shrinkage of a part.</p>

To ensure that load analysis is correct, use area and uniform loads for loads on floors. For example, when the layout of beams changes, Tekla Structures recalculates the loads to the beams. It will not do this if you use point or line loads on individual beams. Tekla Structures also distributes area and uniform loads automatically if they act on parts that have openings.



See also [Creating loads on page 21](#)  
[Load properties on page 98](#)

## 2.2 Setting the load modeling code

Load modeling code settings determine the building code, safety factors, and load group types that Tekla Structures uses in the load combination process.



You should not need to change these settings during the project. If you change the settings, you will also need to change the load group types and check the load combinations.

---

To set the load modeling code and to use the standard building code specific load combination factors:

1. Click **Tools --> Options --> Options... --> Load modeling** to open the **Options** dialog box.
2. On the **Current code** tab, select a code from the **Load modeling code** list.
3. Check the load combination factors on the appropriate tab.
4. If you use the Eurocode, enter the reliability class factor and select the formula to be used on the **Eurocode** tab.
5. Click **OK**.

See also [Load modeling code options on page 105](#)  
[Load combination factors on page 105](#)  
[Using non-standard load combination factors on page 17](#)

### Using non-standard load combination factors

If necessary, you can change the values of building code specific load combination factors and create your own settings to be used in the load combination process.



You should not need to change these settings during the project. If you change the settings, you will also need to change the load group types and check the load combinations.

---

To use non-standard, user-defined load combination factors:

1. Click **Tools --> Options --> Options... --> Load modeling** to open the **Options** dialog box.

2. On the **Current code** tab, select a code from the **Load modeling code** list that is the most appropriate to your needs.
3. Change the load combination factors on the appropriate tab.
4. Save the settings using a new name.

- a. Enter a name in the box next to the **Save as** button.
- b. Click **Save as**.

Tekla Structures saves the settings in the `\attributes` folder under the current model folder with the file name extension `.opt`.

To later use the saved settings, select the name of the settings file from the **Load** list, and then click **Load**.

5. Click **OK**.

**See also** [Load combination factors on page 105](#)  
[Setting the load modeling code on page 17](#)

## 2.3 Grouping loads

Each load in a Tekla Structures model has to belong to a *load group*. A load group is a set of loads and loadings that are caused by the same action and to which you want to refer collectively. Loads that belong to the same load group are treated alike during the load combination process.

Tekla Structures assumes that all loads in a load group:

- Have the same partial safety and other combination factors
- Have the same action direction
- Occur at the same time and all together

You can include as many loads as you like in a load group, of any load type.

You need to create load groups because Tekla Structures creates load combinations on the basis of load groups. We recommend that you define the load groups before you create loads.


**See also** [Creating and modifying a load group on page 19](#)  
[Setting the current load group on page 19](#)  
[Load group compatibility on page 20](#)  
[Deleting a load group on page 21](#)  
[Load group properties on page 97](#)  
[Working with loads and load groups on page 40](#)  
[Combining loads on page 82](#)

## Creating and modifying a load group

You can create a load group by adding a new group or by modifying the default load group. You can modify any existing load group in the same way as the default load group.

Before you start, ensure that you have the appropriate load modeling code selected in **Tools --> Options --> Options... --> Load modeling --> Current code**.

To create or modify a load group:

1. Click **Analysis --> Loads --> Load Groups...** or .
2. In the **Load Groups** dialog box, do one of the following:
  - Click **Add** to create a new load group.
  - Select the default load group from the list to modify it.
  - Select an existing load group from the list to modify it.
3. Click the load group name to modify it.
4. Click the load group type and select a type from the list.
5. Click the load group direction to modify it.
6. To indicate compatibility with existing load groups:
  - a. In the **Compatible** column, enter the number you have used for the load groups that are compatible with this load group.
  - b. In the **Incompatible** column, enter the number you have used for the load groups that are incompatible with this load group.
7. Click the load group color and select a color from the list.

Tekla Structures uses this color when it shows the loads of this load group in the model views.
8. Click **OK** to close the dialog box.

**See also** [Load group properties on page 97](#)

[Setting the current load group on page 19](#)

[Load group compatibility on page 20](#)

[Deleting a load group on page 21](#)

[Working with loads and load groups on page 40](#)


[Setting the load modeling code on page 17](#)

## Setting the current load group

You can define one of the load groups as current. Tekla Structures adds all new loads you create in the current load group.

Before you start, create at least one load group.

To set the current load group:

1. Click **Analysis --> Loads --> Load Groups...** or .
2. In the **Load Groups** dialog box:
  - a. Select a load group.
  - b. Click **Set current**.
  - c. Click **OK** to close the dialog box.

You can also use the **Load and Analysis** toolbar to set or change the current load group:



See also [Creating and modifying a load group on page 19](#)  
[Load group properties on page 97](#)

## Load group compatibility

When Tekla Structures creates load combinations for structural analysis, it follows the building code you select in **Tools --> Options --> Options... --> Load modeling**.

To accurately combine loads which have the same load group type, you need to use compatibility indicators (numbers) to identify which load groups:

- Can occur at the same time (are compatible)
- Exclude each other (are incompatible)

Compatible load groups can act together or separately. They can actually be one single loading, for example, a live loading that needs to be split in parts that act on different spans of a continuous beam. Tekla Structures then includes none, one, several, or all of the compatible load groups in a load combination.

Incompatible load groups always exclude each other. They cannot occur at the same time. For example, a wind loading from the x direction is incompatible with a wind loading from the y direction. In load combinations Tekla Structures only takes into account one load group in an incompatible grouping at a time.

Tekla Structures automatically applies basic compatibility facts, such as self-weight being compatible with all other loads, or live loads being compatible with wind load.

Tekla Structures does not combine loads in the x direction with those in the y direction.

Compatibility indicators are all 0 by default. This indicates that Tekla Structures combines the load groups as defined in the building code.

**See also** [Load group properties on page 97](#)  
[Creating and modifying a load group on page 19](#)  
[Combining loads on page 82](#)  
[Setting the load modeling code on page 17](#)

## Deleting a load group


You can delete one or several load groups at a time.



When you delete a load group, Tekla Structures also deletes all the loads in the load group.

If you try to delete the only load group, Tekla Structures will warn you. At least one load group must exist.

To delete a load group:

1. Click **Analysis --> Loads --> Load Groups...** or .
2. In the **Load Groups** dialog box:
  - a. Select the load group you want to delete.  
To select multiple load groups, hold down the **Ctrl** or **Shift** key.
  - b. Click **Delete**.
3. If there are loads in any of the deleted load groups, Tekla Structures displays a warning dialog box.  
Do one of the following:
  - Click **Cancel** to **not** delete the load group and the loads in the load group.
  - Click **Delete** to delete the load group and the loads in the load group.

**See also** [Grouping loads on page 18](#)  
[Creating and modifying a load group on page 19](#)  
[Working with loads and load groups on page 40](#)  
[Load group properties on page 97](#)

## 2.4 Creating loads

When you create loads, you have two choices: you can set the properties of a load before you create it, or you can modify the properties after you have created a load.



You cannot attach a load to a part after you have created the load.

You can detach a load from a part after you have created the load.

---



To create loads perpendicular to sloped parts, you can shift the work plane.

---

Before you start creating loads, define the load groups and set the current load group.

**See also** [Defining the properties of a load on page 22](#)

[Creating a point load on page 24](#)

[Creating a line load on page 25](#)

[Creating an area load on page 25](#)

[Creating a uniform load on page 26](#)

[Creating a temperature load or a strain on page 27](#)

[Creating wind loads on page 28](#)

[Distributing and modifying loads on page 31](#)

[Working with loads and load groups on page 40](#)

[Grouping loads on page 18](#)

[Combining loads on page 82](#)

## Defining the properties of a load

Before you create a load, it is a good idea to define or check the load properties.

To define the properties of a load:

1. Click **Analysis** --> **Properties** --> **Loads** , and then click a relevant load type.

For example, click **Area Load** to define area load properties.

2. In the load properties dialog box:

- a. Enter or modify the properties.

- Select a load group.
- Define the load magnitude, and the load form if needed.
- Attach the load to a part or to a position.

You cannot attach a load to a part after you have created the load.

You can detach a load from a part after you have created the load.

- Define the load-bearing parts.
  - If needed, adjust the loaded length or area.
  - If needed, modify the load distribution on the **Load panel** tab.
- b. Click **OK** to save the properties.

Tekla Structures uses these properties when you create new loads of this type.

See also [Load properties on page 98](#)

[Load magnitude on page 23](#)

[Load form on page 24](#)

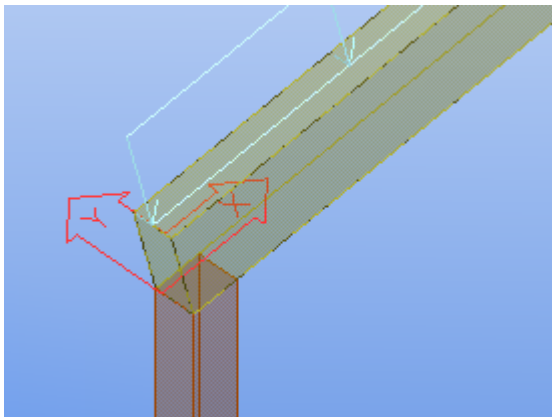
[Distributing and modifying loads on page 31](#)

[Grouping loads on page 18](#)

### ***Load magnitude***

Load magnitude can occur in x, y, and z directions. The coordinate system is the same as the current work plane. Positive coordinates indicate a positive load direction.

For example, when you create loads perpendicular to sloped parts, shifting the work plane helps you to place loads accurately.



Some load types can have several magnitude values. For example, the magnitude of line loads may vary along the loaded length.

In the load properties dialog boxes, the following letters denote different magnitude types:

- **P** is for a force acting on a position, along a line, or across an area.
- **M** is for bending moments acting on a position or along a line.
- **T** is for torsional moments acting along a line.

The units depend on the settings in **Tools --> Options --> Options --> Units and decimals**.





In the load properties dialog boxes, the numbering of the magnitude values relates to the order in which you pick points when you create loads.

See also [Load properties on page 98](#)



### **Load form**

Distributed loads (line and area loads) can have different load forms.

The load form of a line load defines how the load magnitude varies along the loaded length.  
The options are:

Option	Description
	The load magnitude is uniform across the loaded length.
	The load has different magnitudes at the ends of the loaded length. The magnitude changes linearly between the ends.
	The load magnitude changes linearly, from zero at the ends of the loaded length, to a fixed value in the middle of the loaded length.
	The load magnitude changes linearly, from zero at one end of the loaded length, through two (different) values, back to zero at the other end.

The load form of an area load defines the shape of the loaded area. It can be:

Option	Description
	Quadrangular
	Triangular

See also [Line load properties on page 99](#)  
[Area load properties on page 100](#)

### **Creating a point load**

You can create a concentrated force or a bending moment acting on a position.

Before you start, shift the work plane if you need to create a load perpendicular to a sloped part.

To create a point load:



1. Click **Analysis --> Properties --> Loads --> Point Load...** .
2. In the **Point Load Properties** dialog box:
  - a. Enter or modify the load properties.
  - b. On the **Distribution** tab, select whether you want to attach the load to a part.
  - c. Click **OK** to save the changes.
3. Click **Analysis --> Loads --> Create Point Load** .
4. If you selected to attach the load to a part, select the part.
5. Pick the position of the load.

**See also** [Point load properties on page 99](#)  
[Defining the properties of a load on page 22](#)  
[Attaching loads to parts or locations on page 31](#)

## Creating a line load

You can create a linearly-distributed force or torsion between two points you pick.

Before you start, shift the work plane if you need to create a load perpendicular to a sloped part.

To create a line load:

1. Click **Analysis --> Properties --> Loads --> Line Load...** .
2. In the **Line Load Properties** dialog box:
  - a. Enter or modify the load properties.
  - b. On the **Distribution** tab, select whether you want to attach the load to a part.
  - c. Click **OK** to save the changes.
3. Click **Analysis --> Loads --> Create Line Load** .
4. If you selected to attach the load to a part, select the part.
5. Pick the start point of the load.
6. Pick the end point of the load.

**See also** [Line load properties on page 99](#)  
[Defining the properties of a load on page 22](#)  
[Attaching loads to parts or locations on page 31](#)

## Creating an area load

Area loads affect triangular or quadrangular areas. If you select the triangular load form, the points you pick define the loaded area. To create a quadrangular load form, pick three points, and Tekla Structures automatically determines the fourth corner point.

Before you start, shift the work plane if you need to create a load perpendicular to a sloped part.

To create an area load:

1. Click **Analysis --> Properties --> Loads --> Area Load...**
2. In the **Area Load Properties** dialog box:
  - a. Enter or modify the load properties.
  - b. On the **Distribution** tab, select whether you want to attach the load to a part.
  - c. Click **OK** to save the changes.
3. Click **Analysis --> Loads --> Create Area Load**.
4. If you selected to attach the load to a part, select the part.
5. Pick three corner points for the load.

See also [Area load properties on page 100](#)

[Defining the properties of a load on page 22](#)

[Attaching loads to parts or locations on page 31](#)

## Creating a uniform load

Uniform load is an area load distributed uniformly on a polygonal area. The bounding polygon is defined by at least three corner points you pick. Uniform loads can have openings.

Before you start, shift the work plane if you need to create a load perpendicular to a sloped part.

To create a uniform load:

1. Click **Analysis --> Properties --> Loads --> Uniform Load...**
2. In the **Uniform Load Properties** dialog box:
  - a. Enter or modify the load properties.
  - b. On the **Distribution** tab, select whether you want to attach the load to a part.
  - c. Click **OK** to save the changes.
3. Click **Analysis --> Loads --> Create Uniform Load**.
4. If you selected to attach the load to a part, select the part.
5. Pick three corner points for the load.

6. If needed, pick more corner points.
7. Pick the first point again.
8. If you want to create an opening:
  - a. Pick the corner points of the opening.
  - b. Pick the first point of the opening again.
9. Click the middle mouse button to finish picking.

**See also** [Uniform load properties on page 101](#)

[Defining the properties of a load on page 22](#)

[Attaching loads to parts or locations on page 31](#)

## Creating a temperature load or a strain

You can model a temperature change in a part, or a temperature difference between two part surfaces, or a strain.

To create a temperature load or a strain:

1. Click **Analysis --> Properties --> Loads --> Temperature Load...**
2. In the **Temperature Load Properties** dialog box:
  - a. Enter or modify the load properties.
  - b. On the **Magnitude** tab, do one of the following:
    - Use the **Temperature difference** section to define a temperature load.  
If you want to apply a temperature load to an entire structure, enter the load in the **Temperature change for axial elongation** box.
    - Use the **Strain** section to define a strain.
  - c. On the **Distribution** tab, select whether you want to attach the load to a part.  
If you want to apply a temperature load to an entire structure, adjust the bounding box to surround all the beams and columns in the structure.
  - d. Click **OK** to save the changes.
3. Click **Analysis --> Loads --> Create Temperature Load**.
4. If you selected to attach the load to a part, select the part.
5. Pick the start point of the load.
6. Pick the end point of the load.

**See also** [Temperature load properties on page 101](#)

[Defining the properties of a load on page 22](#)

[Attaching loads to parts or locations on page 31](#)

## Creating wind loads

You can model the effects of wind on a building.

To create wind loads:

1. Click **Analysis --> Properties --> Loads --> Wind Load...** .
2. In the **Wind Load Generator (28)** dialog box:
  - a. Enter or modify the load properties.
  - b. Click **OK** to save the changes.
3. Click **Analysis --> Loads --> Create Wind Load** .
4. Pick points to indicate the shape of the building on the bottom level.
5. Click the middle mouse button to finish.



Tekla Structures does the following automatically:

- Creates area loads to model the effects of wind
- Includes wind loads in load combinations
- Distributes wind loads if they act on plates, slabs, or panels that have openings

---

To select or modify wind loads:



- Use the **Select components** switch  for all loads created as a group.
- Use the **Select objects in components** switch  for individual loads in a group.

---

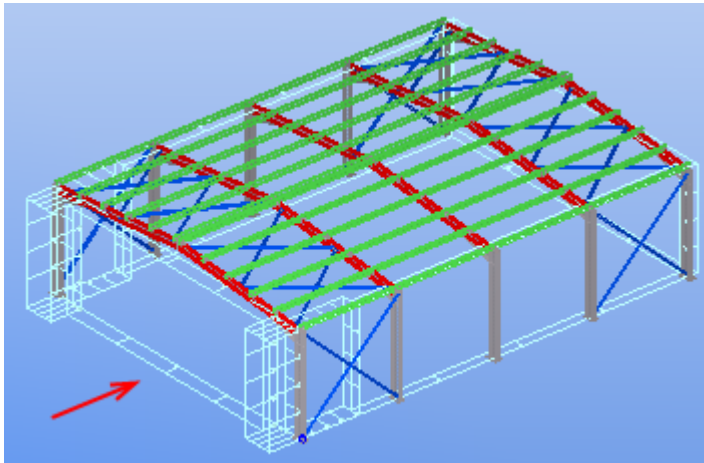
**See also** [Wind load properties on page 102](#)

[Wind load examples on page 28](#)

### *Wind load examples*

Here are examples on how you can use **Wind Load Generator (28)** to create wind loads.

**Example 1** In this example, there are concentrated wind loads at the corners of a building.



The loads induced by the wind in the global x direction are multiplied by 3 at both corners of wall 1 (windward wall), and at the other corner of walls 2 and 4 (side walls). The zone widths are defined by using dimensions.

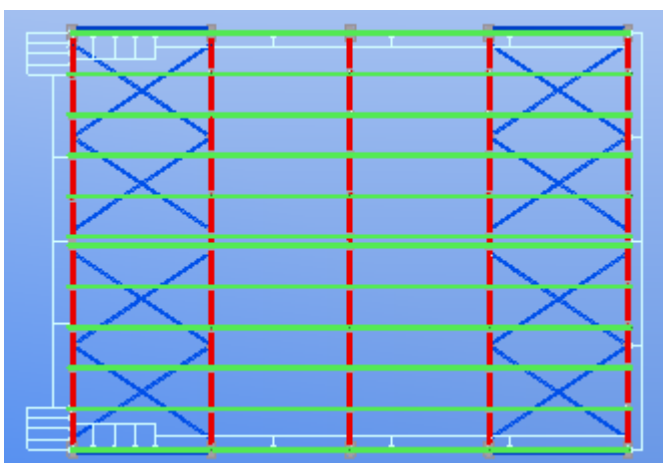
Tekla Structures x64 Wind Load Generator (28)

Save Load standard Save as standard Help...

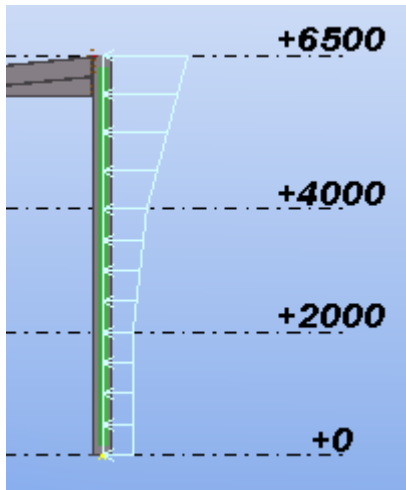
General Exposure factors Z-profile Global X Global Y Global -X Global -Y

	Zone widths	Load factors
wall 1	<input checked="" type="checkbox"/> 1500.00 12000.00 1500.00	<input checked="" type="checkbox"/> 3 1 3
wall 2	<input checked="" type="checkbox"/> 3000.00 17000.00	<input checked="" type="checkbox"/> 3 1
wall 3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
wall 4	<input checked="" type="checkbox"/> 17000.00 3000.00	<input checked="" type="checkbox"/> 1 3

The walls are numbered according to the picking order of the building shape. In this example, points were picked clockwise starting from the bottom left corner of the building.



**Example 2** In this example, wind loads vary along the height of the building.



The z profile is defined in terms of pressure factors.

The screenshot shows the Tekla Structures x64 Wind Load Generator (28) dialog box. The Z-profile tab is selected, showing the following settings:

Height	Load factor
<input checked="" type="checkbox"/> 20000.00	<input checked="" type="checkbox"/> 5.00
<input checked="" type="checkbox"/> 6500.00	<input checked="" type="checkbox"/> 5.00
<input checked="" type="checkbox"/> 4000.00	<input checked="" type="checkbox"/> 2.00
<input checked="" type="checkbox"/> 2000.00	<input checked="" type="checkbox"/> 1.00
<input checked="" type="checkbox"/> 0.00	<input checked="" type="checkbox"/> 1.00

See also [Creating wind loads on page 28](#)  
[Wind load properties on page 102](#)

# 3 Distributing and modifying loads

This section explains how Tekla Structures distributes loads to parts and how you can modify loads and load distribution.

Click the links below to find out more:

- [Attaching loads to parts or locations on page 31](#)
- [Applying loads to parts on page 31](#)
- [Changing the loaded length or area of a load on page 34](#)
- [Modifying the distribution of a load on page 35](#)
- [Modifying the location or layout of a load on page 37](#)
- [Moving a load end or corner using handles on page 39](#)

## 3.1 Attaching loads to parts or locations

You can attach loads to parts or locations for modeling purposes.

Attaching a load to a part binds the load and the part together in the model. If the part is moved, copied, deleted, etc., it affects the load. For example, you can attach a prestressing load to a part, so that the load moves with the part, and disappears if the part is deleted.

If you do not attach a load to a part, Tekla Structures fixes the load to the positions you pick when you create the load.



You cannot attach a load to a part after you have created the load.

You can detach a load from a part after you have created the load.

---

See also [Applying loads to parts on page 31](#)

## 3.2 Applying loads to parts

To apply loads in a structural analysis model, Tekla Structures searches for parts in the areas that you specify. For each load, you can define the load-bearing parts by name or selection filter, and the search area (the bounding box of the load).

Click the links below to find out more:

- [Defining load-bearing parts by name on page 32](#)
- [Defining load-bearing parts by selection filter on page 33](#)
- [Bounding box of a load on page 34](#)
- [Changing the loaded length or area of a load on page 34](#)
- [Modifying the distribution of a load on page 35](#)

### Defining load-bearing parts by name

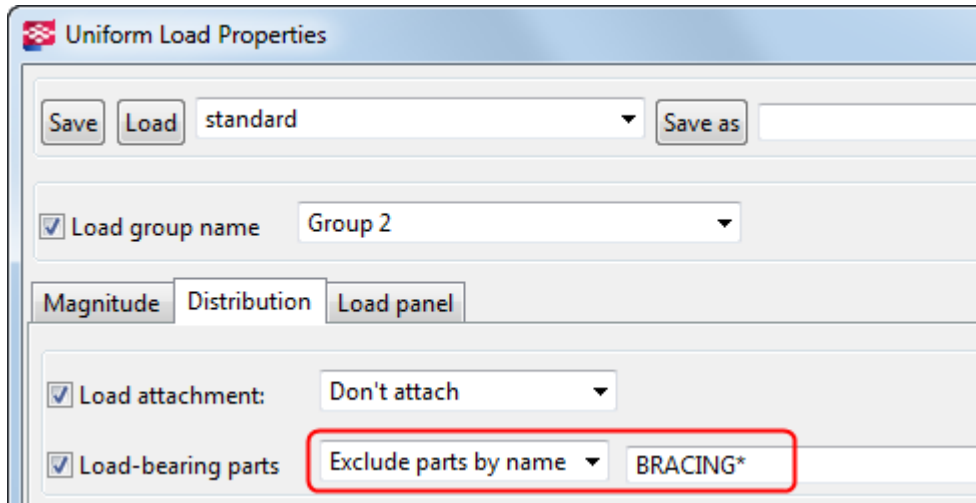
You can list the parts that carry a load or the parts that do not carry a load.

To define the load-bearing parts of a load by name:

1. Double-click the load that you want to distribute to parts.  
The load properties dialog box opens.
2. On the **Distribution** tab:
  - a. In the **Load-bearing parts** list, select one of the following:
    - **Include parts by name** to define the parts that carry the load.
    - **Exclude parts by name** to define the parts that do not carry the load.
  - b. Enter the part names.  
You can use wildcards when listing the part names.
3. Click **Modify** to save the change.

**Example** In this example, braces do not carry this uniform load:





See also [Defining load-bearing parts by selection filter on page 33](#)

## Defining load-bearing parts by selection filter

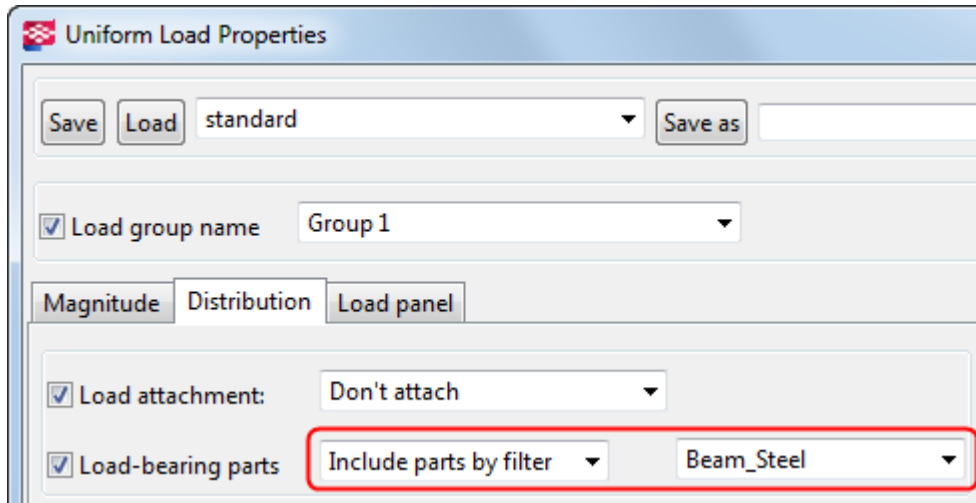
You can define the load-bearing parts by using selection filters.

Before you start, check if there is a selection filter available that suits your needs. If not, create one.

To define the load-bearing parts of a load by selection filter:

1. Double-click the load that you want to distribute to parts.  
The load properties dialog box opens.
2. On the **Distribution** tab:
  - a. In the **Load-bearing parts** list, select one of the following:
    - **Include parts by filter** to define the parts that carry the load.
    - **Exclude parts by filter** to define the parts that do not carry the load.
  - b. Select the selection filter in the second list.
3. Click **Modify** to save the changes.

**Example** In this example, parts that match the **Beam\_Steel** filter carry this uniform load:



See also

[Defining load-bearing parts by name on page 32](#)

## Bounding box of a load

A *bounding box* is the volume around a load where Tekla Structures searches for load-bearing parts.

In addition to selection filters or part name filters, you can use a load's bounding box to search for the parts that carry the load.

Each load has its own bounding box. You can define the dimensions of a bounding box in the x, y, and z directions of the current work plane. The dimensions are measured from the reference point, line, or area of the load.

Offset distances from the reference line or area do not affect the size of the bounding box.

See also [Defining load-bearing parts by name on page 32](#)

[Defining load-bearing parts by selection filter on page 33](#)

[Changing the loaded length or area of a load on page 34](#)

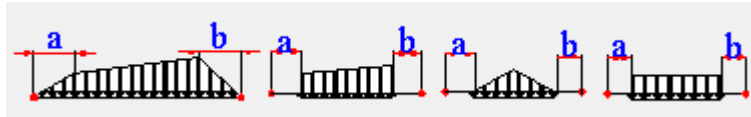
## 3.3 Changing the loaded length or area of a load

If a line, area, or uniform load affects a length or an area that is difficult to select in the model, select a length or an area close to it. Then define offset distances from the load reference points to set the length or area. You can shorten, lengthen, or divide the loaded

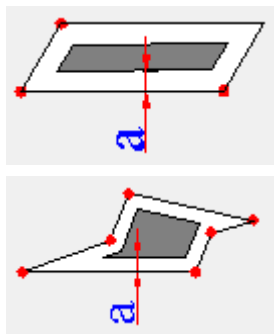
length, and enlarge or reduce the loaded area. Offset distances only apply to the outer edges of loads, not the openings in uniform loads.

To define offset distances for a load:

1. Double-click a load to open its properties dialog box.
2. On the **Distribution** tab, enter the distance values in the **Distances** boxes:
  - To shorten or divide the length of a line load, enter positive values for **a** and/or **b**.
  - To lengthen a line load, enter negative values for **a** and/or **b**.



- To enlarge an area load or a uniform load, enter a positive value for **a**.
- To reduce an area load or a uniform load, enter a negative value for **a**.



3. Click **Modify** to save the changes.

See also [Modifying the location or layout of a load on page 37](#)  
[Moving a load end or corner using handles on page 39](#)

### 3.4 Modifying the distribution of a load

You can modify the way Tekla Structures distributes loads.

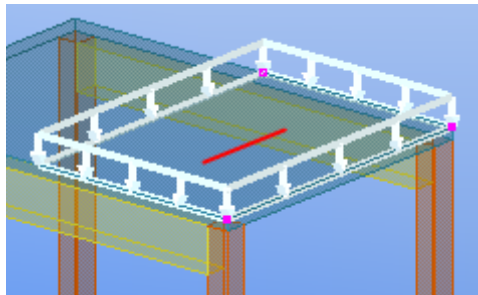
To change the distribution of a load:

1. Double-click a load to open its properties dialog box.
2. Go to the **Load panel** tab.
3. In the **Spanning** list, select whether to distribute the load in one or two directions.
4. If you set **Spanning** to **Single**, define the primary axis direction. If you set **Spanning** to **Double**, you need to define the primary axis direction to be able to manually define the primary axis weight.

Do one of the following:

- To align the primary axis direction with a part, click **Parallel to part...** or **Perpendicular to part...**, and then select the part in the model.
- To distribute the load in the global x, y, or z direction, enter 1 in the corresponding **Primary axis direction** box.
- To distribute the load between several global directions, enter the components of the direction vector in the relevant **Primary axis direction** boxes.

To check the primary axis direction of a selected load in a model view, click **Show direction on selected loads**. Tekla Structures indicates the primary direction using a red line.



5. In the **Automatic primary axis weight** list, select whether Tekla Structures automatically weights the primary direction in load distribution.

If you select **No**, enter a value in the **Weight** box.

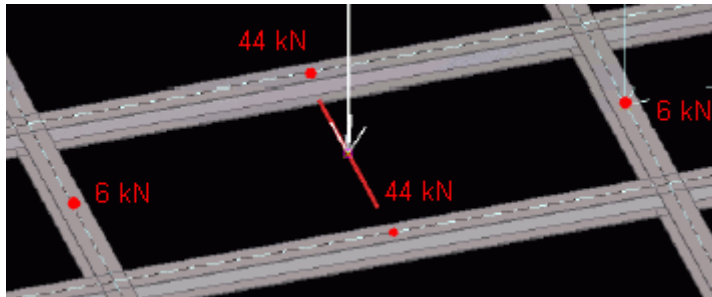
6. In the **Load dispersion angle** box, define the angle by which the load is projected onto the surrounding parts.
7. In the **Use continuous structure load distribution** list of a uniform load, define the distribution of support reactions in the first and last spans of continuous slabs.
  - Select **Yes** for the 3/8 and 5/8 distribution.



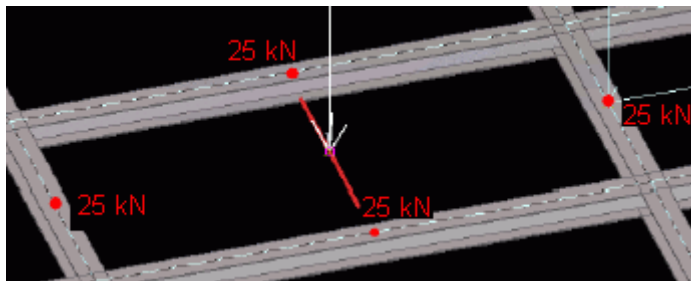
- Select **No** for the 1/2 and 1/2 distribution.
8. Click **Modify** to save the changes.

**Example** When using double spanning, automatic primary axis weight and the weight value affect the proportions of the load which is applied to the primary axis and to the perpendicular axis.

- If **Automatic primary axis weight** is **Yes**, the proportions will be in proportion to the third power of the span lengths in these two directions. This means that the shorter the span, the bigger the proportion of the load. The **Weight** value does not matter.



- If **Automatic primary axis weight** is **No**, the given **Weight** value (0.50 in this example) is used to divide the load.




See also [Load panel settings on page 103](#)  
[Distributing and modifying loads on page 31](#)

### 3.5 Modifying the location or layout of a load

You can modify the location or layout of loads using direct modification.

Before you start:





- Ensure that the **Direct modification** switch  is active.
- Select the load.

Tekla Structures displays the handles and dimensions that you can use to modify the load. When you right-click a handle, Tekla Structures displays a toolbar with more functions. The available functions depend on the type of the load you are modifying.



When you drag a handle, hold down the **Shift** key to use the snap switches. By default, the snap switches are off to make it easier to drag the handle to any location.

To modify the location or layout of a load:


To	Do this	Available for
Set a load reference point to move in one or two directions	<ol style="list-style-type: none"> <li>1. Right-click the handle in the load reference point.</li> <li>2. Click  to define whether the handle can move in one or two directions.</li> </ol> <p>You can also press <b>Tab</b> when you have the handle selected.</p>	Point loads, line loads, area loads, temperature loads, wind loads
Move a point load or a load end or corner	Drag the handle in the load reference point to a new location.	All loads
Move a line load or a load edge	Drag a line handle to a new location.	Line loads, area loads, uniform loads, temperature loads, wind loads
Show or hide diagonal dimensions	<ol style="list-style-type: none"> <li>1. Right-click a handle.</li> <li>2. Click .</li> </ol>	Line loads, area loads, uniform loads, temperature loads, wind loads
Change a dimension	<p>Drag a dimension arrowhead to a new location, or:</p> <ol style="list-style-type: none"> <li>1. Select the dimension arrowhead which you want to move.</li> </ol> <p>To change the dimension at both ends, select both arrowheads.</p> <ol style="list-style-type: none"> <li>2. Using the keyboard, enter the value with which you want the dimension to change.</li> </ol> <p>To start with the negative sign (-), use the numeric keypad.</p> <p>To enter an absolute value for the dimension, first enter \$, then the value.</p> <ol style="list-style-type: none"> <li>3. Press <b>Enter</b>, or click <b>OK</b> in the <b>Enter a Numeric Location</b> dialog box.</li> </ol>	Line loads, area loads, uniform loads, temperature loads, wind loads
Show or hide the midpoint handles of a uniform load	<ol style="list-style-type: none"> <li>1. Right-click a handle.</li> <li>2. Click .</li> </ol>	Uniform loads
Add corner points to a uniform load	Drag a midpoint handle  to a new location.	Uniform loads
Remove points from a uniform load	<ol style="list-style-type: none"> <li>1. Select one or more reference points.</li> <li>2. Press <b>Delete</b>.</li> </ol>	Uniform loads

See also [Moving a load end or corner using handles on page 39](#)

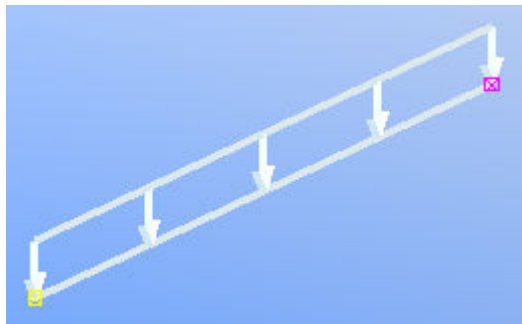
### 3.6 Moving a load end or corner using handles

Tekla Structures indicates the ends and corners of loads with handles. You can use these handles to move load ends and corners when you do not want to use direct modification.

To move a load end or corner:

1. Ensure that the **Direct modification** switch  is **not** active.
2. Select a load to display its handles.

When you select a load, the handles are magenta. For line loads, the handle of the first end is yellow.



3. Click the handle you want to move.  
Tekla Structures highlights the handle.
4. Move the handle like any other object in Tekla Structures.  
If you have **Drag and Drop** on, just drag the handle to a new position.

See also [Modifying the location or layout of a load on page 37](#)

# 4 Working with loads and load groups

This section explains how to work with loads and load groups.

Click the links below to find out more:

- [Scaling loads in model views on page 40](#)
- [Checking loads and load groups on page 41](#)
- [Moving loads to another load group on page 44](#)
- [Exporting load groups on page 44](#)
- [Importing load groups on page 45](#)

## 4.1 Scaling loads in model views

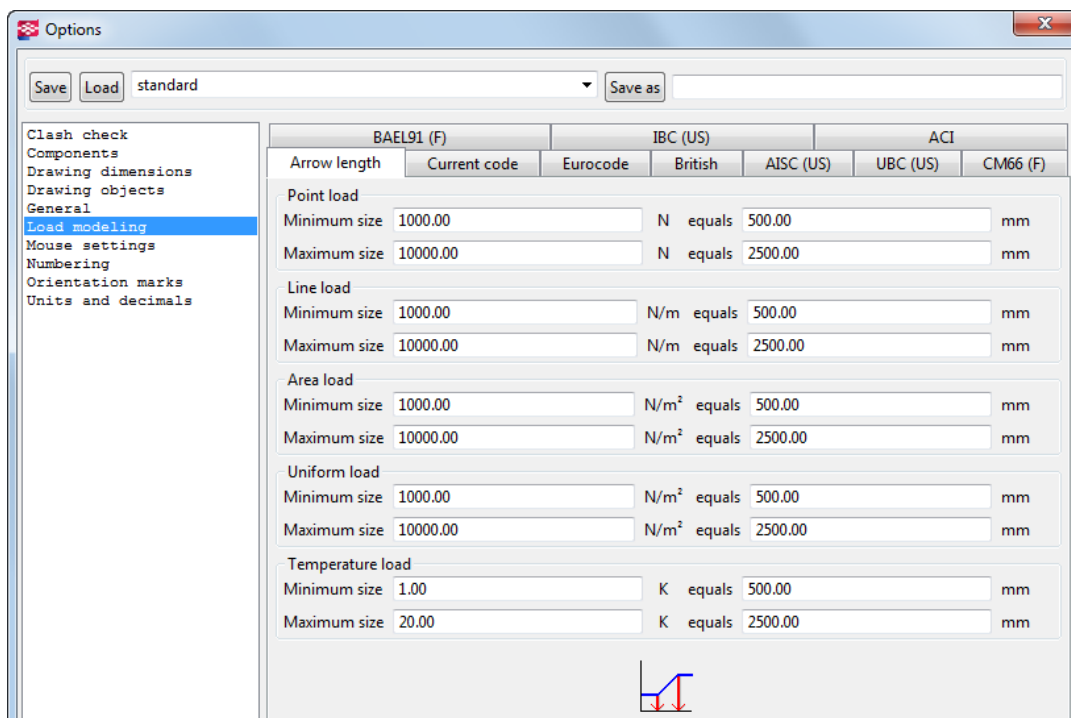
You can have Tekla Structures scale loads when you are modeling. This ensures that loads are not too small to see, or so large that they hide the structure.

To scale loads in model views:

1. Click **Tools** --> **Options** --> **Options...** --> **Load modeling** .
2. On the **Arrow length** tab, enter the minimum and maximum sizes for load types.
3. Click **OK**.

**Example** Define that point loads with magnitude of 1 kN or less are 500 mm high in the model, and that point loads with magnitude of 10 kN or more are 2500 mm high. Tekla Structures linearly scales all point loads that have magnitudes between 1 kN and 10 kN between 500 mm and 2500 mm.





The units depend on the settings in **Tools --> Options --> Options --> Units and decimals**.

See also [Working with loads and load groups on page 40](#)

## 4.2 Checking loads and load groups

You can use several methods to check loads and load groups.

Click the links below to find out more:

- [Checking a load on page 41](#)
- [Finding out to which load group a load belongs on page 42](#)
- [Finding out which loads belong to a load group on page 43](#)
- [Checking loads using reports on page 43](#)

### Checking a load

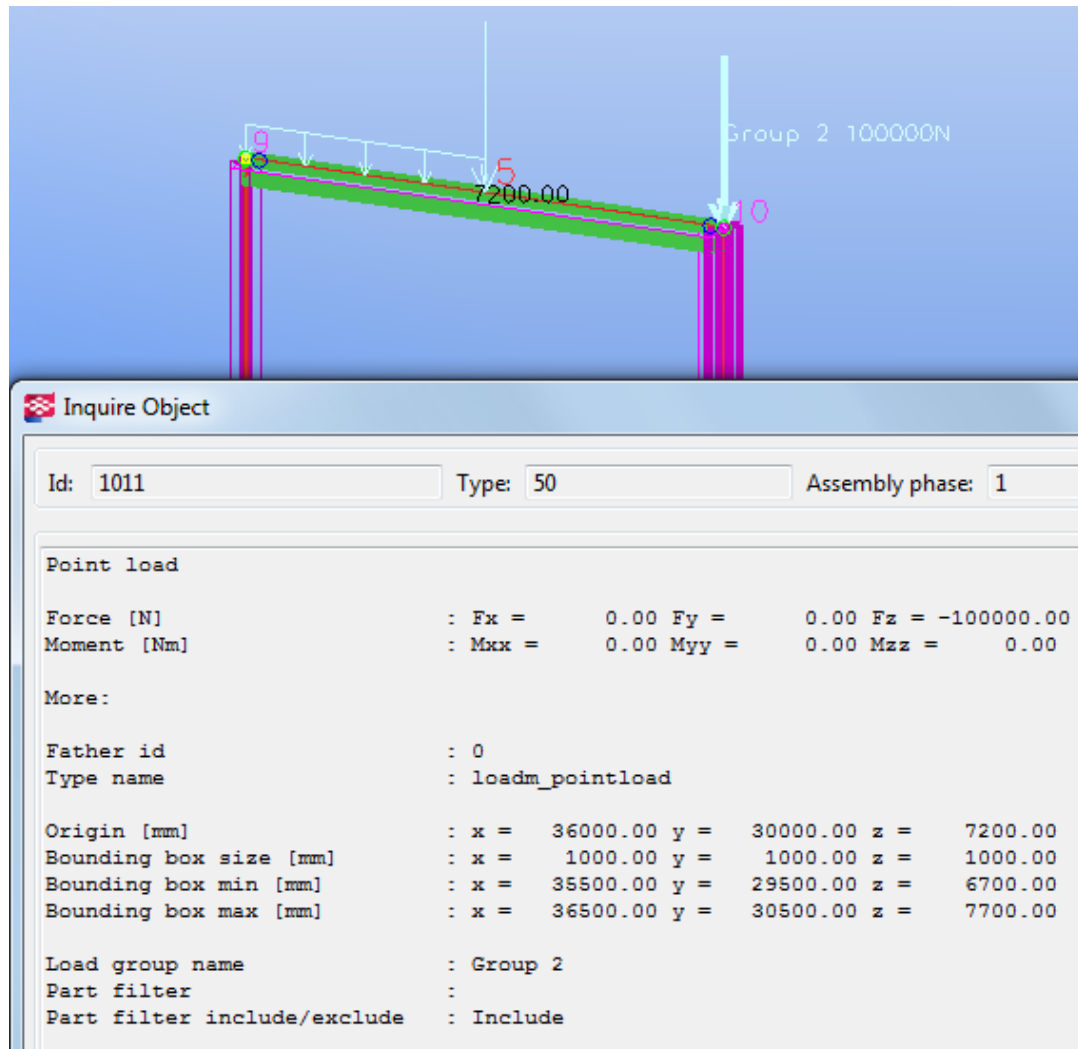
You can check the load group and the magnitude of a load and show them in a model view. Tekla Structures also shows more information about the load in the **Inquire Object** dialog box and highlights the parts that carry the load.

To check a load:

1. In a model view, select a load.

2. Right-click and select **Inquire** from the pop-up menu.

Tekla Structures shows the load group and the magnitude in the model view and highlights the parts that carry the load. The **Inquire Object** dialog box also opens and shows more information about the load.



See also [Finding out to which load group a load belongs on page 42](#)  
[Finding out which loads belong to a load group on page 43](#)  
[Checking loads using reports on page 43](#)

## Finding out to which load group a load belongs

You can check to which load groups selected loads belong.

To find out to which load group a load belongs:

1. Click **Analysis --> Loads --> Load Groups...** .

2. Select a load in the model.  
To select multiple loads, hold down the **Ctrl** or **Shift** key.
3. In the **Load Groups** dialog box, click **Load groups by loads**.  
Tekla Structures highlights the load group in the dialog box.

**See also** [Checking a load on page 41](#)

[Finding out which loads belong to a load group on page 43](#)

[Checking loads using reports on page 43](#)

## Finding out which loads belong to a load group

You can check which loads belong to a selected load group.

To find out which loads belong to a load group:

1. Click **Analysis --> Loads --> Load Groups...** .
2. In the **Load Groups** dialog box:
  - a. Select a load group from the list.
  - b. Click **Loads by load groups**.

Tekla Structures highlights the loads of the load group in the model.

**See also** [Finding out to which load group a load belongs on page 42](#)

[Checking loads using reports on page 43](#)

## Checking loads using reports

You can create reports of loads and load groups, and use them to check load and load group information.

When you select a row that contains an ID number in a load report, Tekla Structures highlights and selects the corresponding load in the model.

Tekla Structures includes the following standard report templates for loads and load groups:

- L\_Loaded\_Part
- L\_Loadgroups
- L\_Loadgroups\_and\_loads
- L\_Loads
- L\_Part\_Loads

**Example** This example report uses the `L_Loadgroups_and_loads` template:


ENGINEERS LOADGROUP AND LOAD REPORT		Page: 1			
Tekla Structures					
Contract No: 1		Contract Name: Tekla Corporation			
		Date: 22.03.2013			
** PLEASE NOTE THIS REPORT DOES NOT CONSIDER APPLIED MOMENTS **					
		Result.X	Result.Y	Result.Z	
LOAD GROUP NAME = DefaultGroup		LOADGROUP TYPE = Permanent load			
LOAD GROUP NAME = Wind load in X		LOADGROUP TYPE = Wind load			
Id:19084	Area load	44999	0	0	
Id:19086	Area load	119999	0	0	
Id:19088	Area load	45000	0	0	
Id:19089	Area load	0	45000	0	
Id:19092	Area load	0	84978	0	
Id:19095	Area load	-75000	0	0	
Id:19097	Area load	0	-85000	0	
Id:19098	Area load	0	-44935	0	
TOTAL FOR LOADGROUP		Wind load in X direc	134998	43	0

See also [Checking loads and load groups on page 41](#)

## 4.3 Moving loads to another load group

You can change the load group of a load, or move several loads at the same time to another load group.

To move loads to another load group, do one of the following:

To	Do this
Change the load group of a load	<ol style="list-style-type: none"> <li>1. Double-click a load in the model.</li> <li>2. In the load properties dialog box: <ol style="list-style-type: none"> <li>a. Select a new load group in the <b>Load group name</b> list.</li> <li>b. Click <b>Modify</b>.</li> </ol> </li> </ol>
Move loads to another load group	<ol style="list-style-type: none"> <li>1. Select the loads in the model.</li> <li>2. Click <b>Analysis --&gt; Loads --&gt; Load Groups...</b> or .</li> <li>3. In the <b>Load Groups</b> dialog box: <ol style="list-style-type: none"> <li>a. Select a load group.</li> <li>b. Click <b>Change load group</b>.</li> </ol> </li> </ol>

See also [Grouping loads on page 18](#)


[Working with loads and load groups on page 40](#)

## 4.4 Exporting load groups

You can export load groups to a file and then use them in another Tekla Structures model.

Before you start, ensure that you have created the relevant load groups.

To export load groups:

1. Click **Analysis --> Loads --> Load Groups...** or .
2. In the **Load Groups** dialog box:
  - a. Select the load group or groups to export.  
To select multiple load groups, hold down the **Ctrl** or **Shift** key.
  - b. Click **Export...**
3. In the **Export Load Groups** dialog box:
  - a. Browse for the folder to which you want to save the load group file.
  - b. Enter a name for the file in the **Selection** box.
  - c. Click **OK**.

The file name extension of a load group file is `.lgr`.

See also [Importing load groups on page 45](#)


[Grouping loads on page 18](#)

## 4.5 Importing load groups

You can import load groups from another Tekla Structures model if they have been exported to a file.

Before you start, ensure that you have the relevant load groups exported to a file.

To import load groups:

1. Click **Analysis --> Loads --> Load Groups...** or .
2. In the **Load Groups** dialog box, click **Import...**
3. In the **Import Load Groups** dialog box:
  - a. Browse for the folder where the load group file is.
  - b. Select the load group file (`.lgr`) to import.
  - c. Click **OK**.

**See also** [Exporting load groups on page 44](#)  
[Grouping loads on page 18](#)

# 5 Creating analysis models

This section explains how to create analysis models in Tekla Structures.

Click the links below to find out more:

- [Defining basic properties for an analysis model on page 47](#)
- [Including objects in analysis models on page 48](#)
- [Selecting the analysis application on page 50](#)
- [Creating an analysis model on page 50](#)

## 5.1 Defining basic properties for an analysis model

Before you create an analysis model, first define the basic analysis model properties, such as analysis model name, creation method, and analysis method.

To define properties for an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, click **New....**
3. In the **Analysis Model Properties** dialog box:
  - a. Enter a unique name for the analysis model.  
For example, you can use a name that describes the portion of the physical model you want to analyze.
  - b. Select the creation method for the analysis model.  
Whichever method you choose, you can easily add and remove objects later.  
If you select **By selected parts and loads** or **Floor model by selected parts and loads**, select the parts and loads in the physical model.
  - c. To make the analysis model more accurate, select a filter from the **Secondary member filter** list.
  - d. On the **Analysis** tab, change the analysis method if you need to run a non-linear analysis.

- e. If needed, define other analysis model properties.
- f. Click **OK**.

**See also** [Analysis model properties on page 107](#)  
[Including objects in analysis models on page 48](#)  
[Selecting the analysis application on page 50](#)  
[Creating an analysis model on page 50](#)

## 5.2 Including objects in analysis models

You can define which objects to include in an analysis model. Tekla Structures includes or ignores some objects automatically.

The following factors affect which objects Tekla Structures includes in analysis models:

- Which objects you select, add, remove, or ignore
- Analysis model creation method
- Analysis model filter

Tekla Structures ignores the following objects in the analysis, even if you have included them in an analysis model:

- Parts and loads that are filtered out
- Component objects, such as minor parts, bolts, and reinforcing bars
- Parts whose analysis class is **Ignore**
- Parts whose analysis part has been deleted

The following components set the analysis properties of the parts they create, so these parts **are included** in analysis models:

- **Shed (S57)**
- **Building (S58) and (S91)**
- **Slab generation (61) and (62)**
- **Truss (S78)**

**See also** [Analysis model creation method on page 49](#)  
[Analysis model filter on page 49](#)  
[Adding objects to an analysis model on page 62](#)  
[Removing objects from an analysis model on page 63](#)  
[Deleting an analysis part on page 80](#)  
[Analysis class options and colors on page 122](#)



## Analysis model creation method

You can define which objects to include in an analysis model by selecting the creation method for the analysis model.

The available analysis model creation methods are:

Option	Description
<b>Full model</b>	Includes all main parts and loads, except for parts whose analysis class is <b>Ignore</b> . Tekla Structures automatically adds physical objects to the analysis model when they are created.
<b>By selected parts and loads</b>	Only includes selected parts and loads, and parts created by components.  To later add or remove parts and loads, use the following buttons in the <b>Analysis &amp; Design Models</b> dialog box: <ul style="list-style-type: none"><li>• <b>Add selected parts</b></li><li>• <b>Remove selected parts</b></li></ul>
<b>Floor model by selected parts and loads</b>	Only includes selected columns, slabs, floor beams, and loads. Tekla Structures replaces columns in the physical model with supports.

See also [Analysis model filter on page 49](#)

[Creating an analysis model on page 50](#)

[Adding objects to an analysis model on page 62](#)

[Removing objects from an analysis model on page 63](#)

[Changing the creation method of an analysis model on page 54](#)

## Analysis model filter

You can use an analysis model filter to select parts to include in or exclude from an analysis model.

The analysis model filter works in a similar way to the selection filter, but Tekla Structures saves the settings with the analysis model properties. This means that you can check the criteria you used to select objects.

Tekla Structures automatically adds the new objects you create in the physical model to the analysis model if they fulfill the criteria in the analysis model filter.



Use the analysis model filter to filter out non-structural parts, such as railings, from the analysis model.

---

See also [Including objects in analysis models on page 48](#)

## 5.3 Selecting the analysis application

You can use various analysis applications with Tekla Structures. You can also transfer analysis data in several formats.

Before you start, ensure that you have the relevant analysis application, or applications, and Tekla Structures installed on your computer. You also have to install a direct link between Tekla Structures and each analysis application. If you do not have these installed, you will not be able to see and select any analysis application options.

To define which analysis application, or format, to use for an analysis model:

1. In Tekla Structures, click **Analysis** --> **Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, do one of the following:
  - To set the analysis application for a new analysis model, click **New....**
  - To change the analysis application of an existing analysis model, select the analysis model, and then click **Properties....**
3. In the **Analysis Model Properties** dialog box:
  - a. Select the analysis application from the **Analysis application** list on the **Analysis model** tab.
  - b. Click **OK**.

See also [About analysis applications on page 12](#)  
[Linking Tekla Structures with an analysis application on page 12](#)

## 5.4 Creating an analysis model

There are several methods to create an analysis model in Tekla Structures.

Click the links below to find out more:

- [Creating an analysis model of entire physical and load models on page 50](#)
- [Creating an analysis model for specific parts and loads on page 51](#)
- [Creating a modal analysis model on page 52](#)
- [Copying an analysis model on page 66](#)

## Creating an analysis model of entire physical and load models

You can create an analysis model that includes all parts and loads you have in a physical model.

Before you start, create the parts and loads you want to include in the analysis model.

To create an analysis model of an entire physical model and a load model:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, click **New....**
3. In the **Analysis Model Properties** dialog box:
  - a. Define the basic analysis model properties.
  - b. In the **Creation method** list on the **Analysis model** tab, select **Full model**.
  - c. If needed, enter or modify the remaining analysis model properties.
  - d. Click **OK**.

Tekla Structures automatically adds physical objects to the **Full model** analysis models when the objects are created.

**See also** [Defining basic properties for an analysis model on page 47](#)

[Analysis model properties on page 107](#)

[Modifying analysis models on page 53](#)

[Changing the creation method of an analysis model on page 54](#)

## Creating an analysis model for specific parts and loads

You can create an analysis model that includes the selected parts and loads.

Before you start, create the parts and loads you want to include in the analysis model.

To create an analysis model for specific parts and loads:

1. Select the objects you want to include in the analysis model.
2. Click **Analysis --> Analysis & Design Models...** .
3. In the **Analysis & Design Models** dialog box, click **New....**
4. In the **Analysis Model Properties** dialog box:
  - a. Define the basic analysis model properties.
  - b. In the **Creation method** list on the **Analysis model** tab, select **By selected parts and loads** or **Floor model by selected parts and loads**.
  - c. If needed, enter or modify the remaining analysis model properties.
  - d. Click **OK**.

**Limitations** If you create an analysis model for selected objects and then use a filter to leave out more objects, you cannot revert to the original objects, even if you remove the filtering.

**See also** [Defining basic properties for an analysis model on page 47](#)

[Analysis model creation method on page 49](#)

[Analysis model properties on page 107](#)

[Adding objects to an analysis model on page 62](#)

[Removing objects from an analysis model on page 63](#)

[Changing the creation method of an analysis model on page 54](#)

## Creating a modal analysis model

You can create modal analysis models of Tekla Structures models. In modal analysis models, resonant frequency and the associated pattern of structural deformation called mode shapes are determined, instead of performing stress analysis.

Before you start, create the parts you want to include in the analysis model.

To create a modal analysis model:

1. If you want to create an analysis model for specific parts, select them in the model.
2. Click **Analysis --> Analysis & Design Models...**
3. In the **Analysis & Design Models** dialog box, click **New...**
4. In the **Analysis Model Properties** dialog box:
  - a. Define the basic analysis model properties.
  - b. On the **Analysis** tab, select **Yes** in the **Modal analysis model** list.
  - c. Click **OK**.

**See also** [Defining modal masses for an analysis model on page 56](#)

[Defining basic properties for an analysis model on page 47](#)

[Analysis model properties on page 107](#)

# 6 Modifying analysis models

This section explains how to modify analysis models and how to work with analysis model objects.

Click the links below to find out more:

- [Checking objects included in an analysis model on page 53](#)
- [Modifying the properties of an analysis model on page 54](#)
- [Adding objects to an analysis model on page 62](#)
- [Removing objects from an analysis model on page 63](#)
- [Creating an analysis node on page 63](#)
- [Creating a rigid link on page 65](#)
- [Merging analysis nodes on page 65](#)
- [Copying an analysis model on page 66](#)
- [Deleting an analysis model on page 67](#)

## 6.1 Checking objects included in an analysis model

To check which parts and loads are included in an analysis model:

1. Click **Analysis --> Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box:
  - a. Select an analysis model.
  - b. Click **Select objects**.

Tekla Structures highlights and selects the parts and loads in the physical model.

To remove the highlighting, click the view background.

**See also** [Including objects in analysis models on page 48](#)

[Adding objects to an analysis model on page 62](#)

[Removing objects from an analysis model on page 63](#)

## 6.2 Modifying the properties of an analysis model

To modify the properties of an analysis model:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model to modify.
  - b. Click **Properties...**
3. In the **Analysis Model Properties** dialog box:
  - a. Modify the properties.
  - b. If you change the secondary member filter, select the **Reapply to all parts** check box to apply the change to all parts in the analysis model.  
  
If you do not select the **Reapply to all parts** check box, Tekla Structures will use the new filter only for the new parts in the analysis model.
  - c. Click **OK** to save the changes.

**See also** [Changing the creation method of an analysis model on page 54](#)

[Defining the axis settings of an analysis model on page 55](#)

[Defining seismic loads for an analysis model on page 56](#)

[Defining modal masses for an analysis model on page 56](#)

[Defining the design properties of an analysis model on page 57](#)

[Defining analysis model rules on page 58](#)

[Analysis model properties on page 107](#)

### Changing the creation method of an analysis model

You can change the creation method of existing analysis models.

If you change the creation method of an analysis model to **Full model**, Tekla Structures automatically adds all parts and loads in the physical model to the analysis model.

To change the creation method of an analysis model:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:

- a. Select the analysis model to modify.
  - b. Click **Properties....**
3. In the **Analysis Model Properties** dialog box:
  - a. On the **Analysis model** tab, select the required option from the **Creation method** list.
  - b. Click **OK** to save the analysis model properties.

**Example** To change the analysis model creation method from **Full model** to **Selected parts and loads**:

1. Copy an analysis model that has been created using the **Full model** creation method.
2. Change the creation method of the copied analysis model to **By selected parts and loads**.
3. Remove the unwanted parts and loads from the analysis model.

**See also** [Analysis model creation method on page 49](#)

[Copying an analysis model on page 66](#)

[Removing objects from an analysis model on page 63](#)

[Adding objects to an analysis model on page 62](#)

## Defining the axis settings of an analysis model

You can define and modify the analysis axis settings of an entire analysis model so that the settings apply to all parts in the analysis model.

To define the axis settings of an analysis model:

1. Click **Analysis --> Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box, do one of the following:
  - To define the axis settings for a new analysis model, click **New....**
  - To modify the axis settings of an existing analysis model, select the analysis model, and then click **Properties....**
3. In the **Analysis Model Properties** dialog box:
  - a. Click **More settings...** on the **Analysis model** tab.
  - b. In the **Member axis location** list, select an option.  
 If you select **Model default**, Tekla Structures uses the axis properties of individual analysis parts.
  - c. Click **OK**.

**See also** [Defining or modifying the axis location of an analysis part on page 78](#)

[Defining the location of analysis parts on page 77](#)

## Defining seismic loads for an analysis model

You can define additional lateral seismic loads for analysis models. The seismic loads are created in the x and y directions according to several building codes using a static equivalent approach.

Before you start, ensure that you have the appropriate load modeling code selected in **Tools --> Options --> Options... --> Load modeling --> Current code**.

To define seismic loads for an analysis model:

1. Click **Analysis --> Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box, do one of the following:
  - To create a new seismic analysis model, click **New...**
  - To modify an existing analysis model, select the analysis model, and then click **Properties...**

The **Analysis Model Properties** dialog box opens.

3. On the **Seismic** tab:
  - a. In the **Type** list, select the building code to be used in the seismic analysis to generate seismic loads.
  - b. Define the seismic properties.
4. On the **Seismic masses** tab, define the loads and load groups to be included in the seismic analysis:
  - a. To include the self-weight of parts, select the **Include self weight as seismic mass** check box.
  - b. If needed, click **Copy modal analysis masses** to include the same load groups in the seismic analysis as in the modal analysis.
  - c. To move the appropriate load groups to the **Included load groups** table, select them and use the arrow buttons.
  - d. For each load group in the **Included load groups** table, enter a load factor.
5. Click **OK**.

See also [Analysis model properties on page 107](#)

## Defining modal masses for an analysis model

You can perform a modal analysis instead of a stress analysis. In the modal analysis, resonant frequencies and the associated patterns of structural deformation called mode shapes are



determined. For the modal analysis, you can define modal masses to be used instead of static load combinations.

To define modal masses for an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, do one of the following:
  - To create a new modal analysis model, click **New....**
  - To modify an existing analysis model, select the analysis model, and then click **Properties....**

The **Analysis Model Properties** dialog box opens.

3. On the **Analysis** tab, select **Yes** from the **Modal analysis model** list.

This forces Tekla Structures to ignore static load combinations.

4. On the **Modal analysis** tab, define the modal analysis properties and the load groups to be included as masses in the modal analysis:
  - a. Enter the count of modes to calculate.
  - b. Enter the maximum frequency to calculate.
  - c. Select the appropriate **Include self weight** check boxes to indicate the directions for which Tekla Structures includes the self-weight of parts in the modal analysis.
  - d. If suitable, click **Copy seismic masses** to include the same load groups in the modal analysis as in the seismic analysis.
  - e. To move the appropriate load groups to the **Included load groups** table, select them and use the arrow buttons.
  - f. For each load group in the **Included load groups** table, enter a load factor and set the mass direction.

In the **Mass direction** column, select either:

- **XYZ** to include the load in all three directions.
- **Model default** to include the load only in the direction of the load.

5. Click **OK**.

**See also** [Creating a modal analysis model on page 52](#)

[Analysis model properties on page 107](#)

## Defining the design properties of an analysis model

You can define and modify the design properties of an entire analysis model so that the properties apply to all parts in the analysis model.

To define or modify the design properties of an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, do one of the following:
  - To define the design properties for a new analysis model, click **New...**
  - To modify the design properties of an existing analysis model, select the analysis model, and then click **Properties...**
3. In the **Analysis Model Properties** dialog box:
  - a. Go to a **Design** tab.  
There are separate **Design** tabs for steel, concrete, and timber.
  - b. Select the design code and design method for the material.
  - c. If needed, modify the design properties.  
Click an entry in the **Value** column, and then enter a value or select an option.
  - d. Click **OK**.

See also [Defining design properties for analysis parts on page 74](#)

[Analysis model properties on page 107](#)

## Defining analysis model rules

You can create analysis model rules to define how Tekla Structures handles individual parts when it creates analysis models, and how parts are connected with each other.

Click the links below to find out more:

- [Adding an analysis model rule on page 58](#)
- [Organizing analysis model rules on page 59](#)
- [Testing analysis model rules on page 60](#)
- [Deleting analysis model rules on page 61](#)
- [Saving analysis model rules on page 62](#)

### *Adding an analysis model rule*

To add a rule for an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...** .

2. In the **Analysis & Design Models** dialog box:
  - a. Select an analysis model.
  - b. Click **Properties...**
3. In the **Analysis Model Properties** dialog box:
  - a. Click **More settings...** on the **Analysis model** tab.
  - b. Click **Analysis model rules...**
4. In the **Analysis Model Rules** dialog box:
  - a. Click **Add** to define how two groups of parts are connected with each other in the analysis.
  - b. In the **Selection filter 1** column, select a filter to define the first part group.  
If you need to create a new selection filter that suits your needs, click **Selection filter...**
  - c. In the **Selection filter 2** column, select a filter to define the second part group.
  - d. If you want to prevent connections between the part groups, select **Disabled** in the **Status** column.
  - e. In the **Linkage** column, select one of the following options:
    - (blank): Merges nodes or creates a rigid link.
    - **Merge**: Always merges nodes when parts matching the first selection filter connect with parts matching the second selection filter.
    - **Rigid link**: Creates a rigid link when parts matching the first selection filter connect with parts matching the second selection filter.
    - **Rigid link, moment release at node 1**: Creates a rigid link and a moment release at the nodes of parts matching the first selection filter.
    - **Rigid link, moment release at node 2**: Creates a rigid link and a moment release at the nodes of parts matching the second selection filter.
    - **Rigid link, moment release at both nodes**: Creates a rigid link and moment releases at the nodes of parts matching the first and the second selection filter.
  - f. Click **OK** to save the rules.
5. In the **Analysis Model Properties** dialog box, click **OK** to save the rules as properties of the current analysis model.

**See also** [Organizing analysis model rules on page 59](#)

[Testing analysis model rules on page 60](#)

[Deleting analysis model rules on page 61](#)

[Saving analysis model rules on page 62](#)

### ***Organizing analysis model rules***

You can change the order of the analysis model rules that you have created for an analysis model. The last rule in the **Analysis Model Rules** dialog box overrides the previous ones.

To change the order of the rules in an analysis model:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model whose rules you want to organize.
  - b. Click **Properties...**
3. In the **Analysis Model Properties** dialog box:
  - a. Click **More settings...** on the **Analysis model** tab.
  - b. Click **Analysis model rules...**
4. In the **Analysis Model Rules** dialog box:
  - a. Select a rule.
  - b. To move the rule up in the list, click **Move up**.  
To move the rule down in the list, click **Move down**.
  - c. Click **OK** to save the changes.
5. In the **Analysis Model Properties** dialog box, click **OK** to save the rules as properties of the current analysis model.

**See also** [Adding an analysis model rule on page 58](#)

[Testing analysis model rules on page 60](#)

[Deleting analysis model rules on page 61](#)

[Saving analysis model rules on page 62](#)

### ***Testing analysis model rules***

You can test the analysis model rules that you have created on the selected parts before you bring the rules into use.

To test analysis model rules on the selected parts in an analysis model:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model whose rules you want to test.
  - b. Click **Properties...**
3. In the **Analysis Model Properties** dialog box:
  - a. Click **More settings...** on the **Analysis model** tab.
  - b. Click **Analysis model rules...**

4. In the model, select the parts on which you want to test the rules.
5. In the **Analysis Model Rules** dialog box:
  - a. Click **Test selected parts**.  
 Tekla Structures opens the **Analysis model rules test** report that lists the IDs of the selected parts, matching selection filters, and the results of using the rules.
  - b. If needed, modify or reorganize the rules and test again.
  - c. When the rules work as you desired, click **OK** to save the rules.
6. In the **Analysis Model Properties** dialog box, click **OK** to save the rules as properties of the current analysis model.

**See also** [Adding an analysis model rule on page 58](#)  
[Organizing analysis model rules on page 59](#)  
[Deleting analysis model rules on page 61](#)  
[Saving analysis model rules on page 62](#)

### ***Deleting analysis model rules***

You can delete selected analysis model rules from an analysis model.

To delete one or more analysis model rules from an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model.
  - b. Click **Properties...**
3. In the **Analysis Model Properties** dialog box:
  - a. Click **More settings...** on the **Analysis model** tab.
  - b. Click **Analysis model rules....**
4. In the **Analysis Model Rules** dialog box:
  - a. Select the rule or rules to delete.  
 To select multiple rules, hold down the **Ctrl** or **Shift** key.
  - b. Click **Remove**.
  - c. Click **OK** to save the changes.
5. In the **Analysis Model Properties** dialog box, click **OK**.

**See also** [Adding an analysis model rule on page 58](#)  
[Organizing analysis model rules on page 59](#)

[Testing analysis model rules on page 60](#)

[Saving analysis model rules on page 62](#)

### ***Saving analysis model rules***

You can save analysis model rules for later use in the same or another analysis model.

Before you start, organize the analysis model rules.

To save the rules of an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model whose rules you want to save.
  - b. Click **Properties...**
3. In the **Analysis Model Properties** dialog box:
  - a. Click **More settings...** on the **Analysis model** tab.
  - b. Click **Analysis model rules...**
4. In the **Analysis Model Rules** dialog box:
  - a. If needed, save the rules for later use:

Enter a unique name in the box next to the **Save as** button, and then click **Save as**.

Tekla Structures saves the rules file in the `\attributes` folder under the current model folder.

The file name extension of an analysis model rules file is `.adrules`.
  - b. Click **OK**.
5. In the **Analysis Model Properties** dialog box, click **OK** to save the rules as properties of the current analysis model.

**See also** [Adding an analysis model rule on page 58](#)

[Organizing analysis model rules on page 59](#)

[Testing analysis model rules on page 60](#)

[Deleting analysis model rules on page 61](#)

## **6.3 Adding objects to an analysis model**

You can modify existing analysis models by adding parts and loads to them.

To add objects to an analysis model:

1. In the physical model, select the parts and loads to add.

2. Click **Analysis --> Analysis & Design Models...** .
3. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model to modify.
  - b. Click **Add selected**.

Tekla Structures adds the selected objects to the selected analysis model.

**See also** [Checking objects included in an analysis model on page 53](#)

[Removing objects from an analysis model on page 63](#)

[Copying an analysis part on page 80](#)

[Creating an analysis node on page 63](#)

[Creating a rigid link on page 65](#)

## 6.4 Removing objects from an analysis model

You can modify existing analysis models by removing parts and loads from them.

To remove objects from an analysis model:

1. In the physical model, select the parts and loads to remove.
2. Click **Analysis --> Analysis & Design Models...** .
3. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model to modify.
  - b. Click **Remove selected**.

Tekla Structures removes the selected objects from the selected analysis model.

**See also** [Checking objects included in an analysis model on page 53](#)

[Adding objects to an analysis model on page 62](#)


[Deleting an analysis part on page 80](#)

## 6.5 Creating an analysis node

You can create nodes on analysis parts. The analysis nodes you manually add are not moved with the analysis part if you move the analysis part.

To create a node in an analysis model:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, select the analysis model to which you want to add the node.

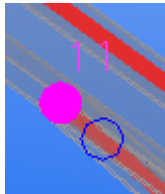

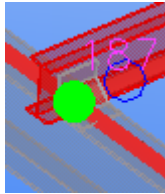
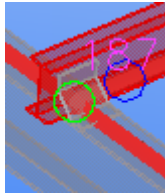
3. Click **Analysis --> Create Node** or .
4. Pick the location where you want to add the node.

**See also** [Analysis model objects on page 9](#)  
[Analysis node properties on page 126](#)  
[Analysis node colors on page 64](#)  
[Merging analysis nodes on page 65](#)

## Analysis node colors

Analysis nodes can have different colors in analysis models.

The color of an analysis node shows the status of the connectivity of the node and whether the node has been selected.

Node color	Connectivity status	Selection	Example
Magenta	Disconnected	Selected	
Magenta	Disconnected	Not selected	
Green	Connected	Selected	
Green	Connected	Not selected	

**See also** [Creating an analysis node on page 63](#)  
[Analysis node properties on page 126](#)




[Analysis model objects on page 9](#)

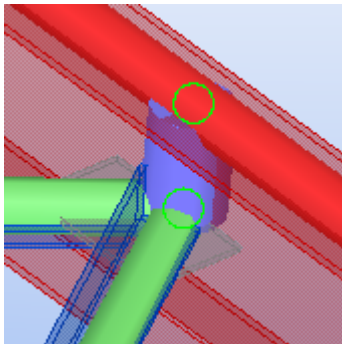
[Merging analysis nodes on page 65](#)

## 6.6 Creating a rigid link

You can create rigid links between analysis nodes.

To create a rigid link in an analysis model:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, select the analysis model to which you want to add the rigid link.
3. Click **Analysis --> Create Rigid Link** or  .
4. Pick the start point for the rigid link.
5. Pick the end point for the rigid link.



**See also** [Analysis model objects on page 9](#)

[Analysis rigid link properties on page 127](#)

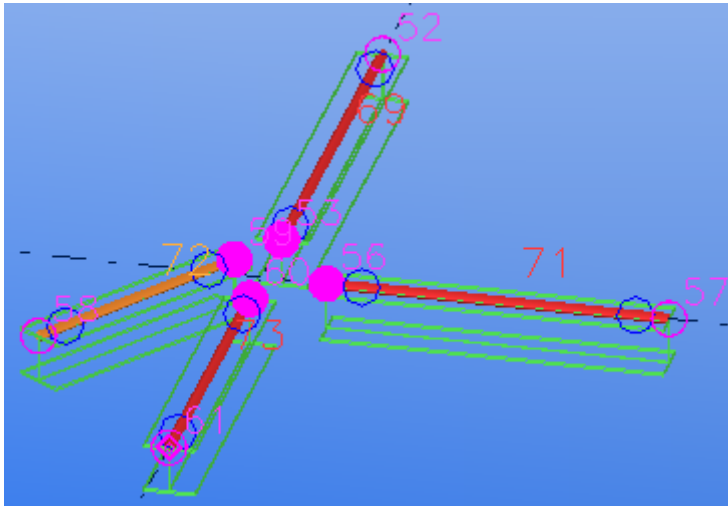
[Creating an analysis node on page 63](#)


## 6.7 Merging analysis nodes

You can merge analysis nodes that are located close to each other into a single node.

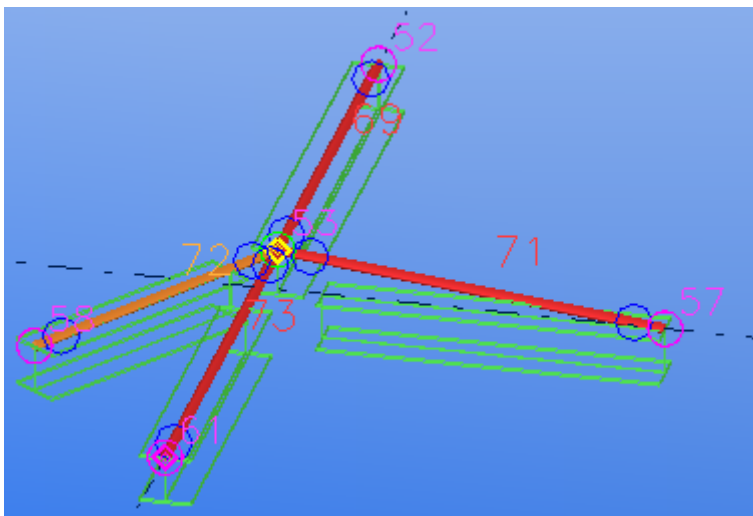
To merge nodes in an analysis model:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, select the analysis model in which you want to merge nodes.
3. Select the nodes you want to merge.



4. Click **Analysis --> Merge Selected Nodes** or .
5. If you are merging nodes on analysis parts that have **Keep axis position** set to **Yes**, Tekla Structures prompts you to change it to **No**. To accept the change, click **Set keep axis as No**.
6. Pick the location to which you want the nodes to be merged.

Tekla Structures merges the nodes into a single node and extends the analysis parts accordingly.



**See also** [Creating an analysis node on page 63](#)  
[Analysis node properties on page 126](#)  
[Analysis node colors on page 64](#)

## 6.8 Copying an analysis model

You can create copies of existing analysis models. You can then use the copies, for example, to create multiple calculations with different settings.

To copy an analysis model:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model to copy.
  - b. Click **Copy**.

Tekla Structures adds the new analysis model to the list with the name **<original model name> - Copy**.

**See also** [Modifying analysis models on page 53](#)  
[Modifying analysis parts on page 68](#)

## 6.9 Deleting an analysis model

You can delete unnecessary analysis models.

To delete an analysis model:

1. Click **Analysis --> Analysis & Design Models** .
2. In the **Analysis & Design Model** dialog box:
  - a. Select an analysis model.
  - b. Click **Delete**.
3. Click **Yes** to confirm.

**See also** [Modifying analysis models on page 53](#)  
[Creating analysis models on page 47](#)

# 7 Modifying analysis parts

This section explains how to modify analysis parts and their properties.

Click the links below to find out more:

- [Defining and modifying analysis part properties on page 68](#)
- [Defining support conditions on page 70](#)
- [Defining design properties for analysis parts on page 74](#)
- [Defining the location of analysis parts on page 77](#)
- [Copying an analysis part on page 80](#)
- [Deleting an analysis part on page 80](#)

## 7.1 Defining and modifying analysis part properties

You can view, define, or modify analysis part properties before or after creating analysis models. You can define analysis part properties independently from analysis models, or modify them according to an analysis model. Analysis parts can have different properties in different analysis models.

You can define analysis properties for parts before you create analysis models. Tekla Structures applies the analysis part properties when the parts are added to an analysis model. You can also modify analysis part properties after you have created analysis models.

If you view the analysis properties of a part before you have modified the properties or created any analysis models, Tekla Structures displays the analysis properties according to the part type. For example, all steel beams first have identical analysis properties. These settings are called *current analysis properties*.

If you modify the analysis properties of a part before you create analysis models, Tekla Structures saves the modified settings as the default analysis properties of the part in the `AnalysisPartDefaults.db6` file under the current model folder. These *default analysis properties* override the current analysis properties and will be used when you add the part to an analysis model.

When you create analysis models and then view the analysis properties of a part, Tekla Structures displays the properties according to the selected analysis model. If you do not

have an analysis model selected in the **Analysis & Design Models** dialog box, Tekla Structures displays the current analysis properties for unchanged parts and the default analysis properties for modified parts.

See also [Modifying the properties of an analysis part on page 69](#)

## Modifying the properties of an analysis part

You can view, define, and modify the properties of analysis parts using the analysis part properties dialog box.

To access the properties of an analysis part, do one of the following:

To	Do this
Define or modify the current analysis properties of a part type independently from analysis models	<ol style="list-style-type: none"> <li>1. Click <b>Analysis --&gt; Properties --&gt; Steel Parts</b> or <b>Concrete Parts</b>, and then click a relevant part type.</li> <li>2. In the analysis properties dialog box: <ol style="list-style-type: none"> <li>a. Modify the properties.</li> <li>b. Click <b>Apply</b> or <b>OK</b> to save the changes as the current analysis properties of the part type.</li> </ol> </li> </ol> <p>Tekla Structures will use these current analysis properties for new parts of this type that you create in the model.</p>
Define or modify the default analysis properties of a part independently from analysis models	<ol style="list-style-type: none"> <li>1. Ensure that you do not have an analysis model selected in the <b>Analysis &amp; Design Models</b> dialog box.</li> <li>2. In the physical model, select a part.</li> <li>3. Right-click and select <b>Analysis Properties</b> from the pop-up menu.</li> <li>4. In the part's analysis properties dialog box: <ol style="list-style-type: none"> <li>a. Modify the properties.</li> <li>b. Click <b>Modify</b> to save the changes as the default analysis properties of the part in the <code>AnalysisPartDefaults.db6</code> file.</li> </ol> </li> </ol> <p>Tekla Structures will use these default analysis properties instead of current analysis properties for this part when you add it to an analysis model.</p>
View the analysis properties of a part independently from analysis models	<ol style="list-style-type: none"> <li>1. Ensure that you do not have an analysis model selected in the <b>Analysis &amp; Design Models</b> dialog box.</li> <li>2. In the physical model, select a part.</li> </ol>

To	Do this
	<ol style="list-style-type: none"> <li>Right-click and select <b>Analysis Properties</b> from the pop-up menu.  If you have already previously modified the analysis properties of this part, Tekla Structures displays these default analysis properties in the part's analysis properties dialog box (for example, <b>Beam Analysis Properties</b>).  If you have not modified the analysis properties of this part, Tekla Structures displays the current analysis properties in the part's analysis properties dialog box (for example, <b>Beam Analysis Properties – Current properties</b>).</li> <li>In the part's analysis properties dialog box: <ol style="list-style-type: none"> <li>View the properties.</li> <li>Click <b>Cancel</b> to close the dialog box.</li> </ol> </li> </ol>
View or modify the properties of an analysis part in an analysis model	<ol style="list-style-type: none"> <li>Click <b>Analysis --&gt; Analysis &amp; Design Models...</b> .</li> <li>In the <b>Analysis &amp; Design Models</b> dialog box, select an analysis model (for example, AnalysisModel3).</li> <li>In the physical model, select a part.</li> <li>Right-click and select <b>Analysis Properties</b> from the pop-up menu.</li> <li>In the part's analysis properties dialog box (for example, <b>Beam Analysis Properties – AnalysisModel3</b>), do one of the following: <ul style="list-style-type: none"> <li>View the properties, and then click <b>Cancel</b> to close the dialog box.</li> <li>Modify the properties, and then click <b>Modify</b> to save the changes.</li> </ul> </li> </ol>

See also [Analysis part properties on page 113](#)

[Defining and modifying analysis part properties on page 68](#)

[Modifying analysis parts on page 68](#)

## 7.2 Defining support conditions

In structural analysis, the stresses and deflections of a part depend on how it is supported by, or connected to, other parts. You normally use restraints or springs to model connections. These determine how analysis parts move, deflect, warp, and deform in relation to each other, or to nodes.

Part ends and nodes have degrees of freedom (DOF) in three directions. The displacement of a part end can be free or fixed, and the rotation can be pinned or fixed. If the degree of

connectivity is between free, or pinned, and fixed, use springs with different elastic constants to model them.

Tekla Structures uses analysis part, connection, or detail properties to determine how to connect parts in the analysis model.

The analysis part properties determine the degrees of freedom for each end of a part. The first end of a part has a yellow handle, the second end has a magenta handle.

**See also** [Defining the support conditions of a part end on page 71](#)

[Defining the support conditions of a plate on page 72](#)

[Support condition symbols on page 72](#)

## Defining the support conditions of a part end

Before you start, in the **Analysis & Design Models** dialog box, select the analysis model in which you want to define the support conditions.

To define the support conditions of a part end:

1. Select a part.
2. Right-click and select **Analysis Properties** from the pop-up menu.
3. In the part's analysis properties dialog box:
  - To define the support conditions for the start of the part (yellow handle), click **Start releases**.
  - To define the support conditions for the end of the part (magenta handle), click **End releases**.
4. In the **Start** or **End** list, select an option.
5. If needed for a supported part end, define the rotation.
6. If needed, modify the translational and rotational degrees of freedom.
7. If you selected **Spring** for any of the degrees of freedom, enter the spring constant.  
The units depend on the settings in **Tools --> Options --> Options --> Units and decimals**.
8. If you selected **Partial release** for any of the rotational degrees of freedom, specify the degree of connectivity.  
Enter a value between 0 (fixed) and 1 (pinned).
9. Click **Modify**.

**See also** [Defining the support conditions of a plate on page 72](#)

[Support condition symbols on page 72](#)

[Analysis part properties on page 113](#)

[Defining and modifying analysis part properties on page 68](#)

### Defining the support conditions of a plate

You can define support conditions for contour plates, concrete slabs, and concrete panels. Tekla Structures creates supports for the bottom edge of a panel, for all edge nodes of a slab or a plate, or for all nodes of a beam. For panels, the bottom edge can be inclined.

Before you start, in the **Analysis & Design Models** dialog box, select the analysis model in which you want to define the support conditions.

To create supports for a plate:

1. Select a plate.
2. Right-click and select **Analysis Properties** from the pop-up menu.
3. In the plate's analysis properties dialog box:
  - a. On the **Area attributes** tab, select an option in the **Supported** list:
    - **No**: No supports are created.
    - **Simply (translations)**: Only translations are fixed.
    - **Fully**: Both translations and rotations are fixed.
  - b. Click **Modify**.

**See also** [Defining the support conditions of a part end on page 71](#)

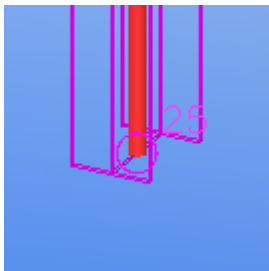
[Support condition symbols on page 72](#)

[Analysis part properties on page 113](#)

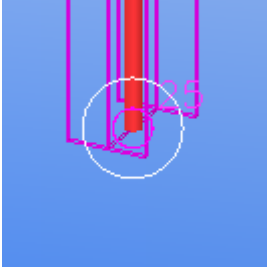
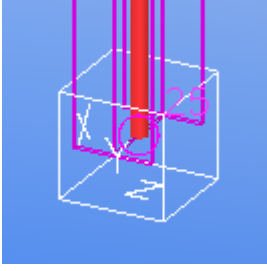
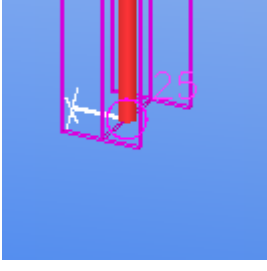
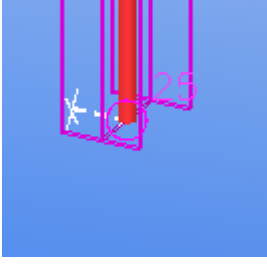
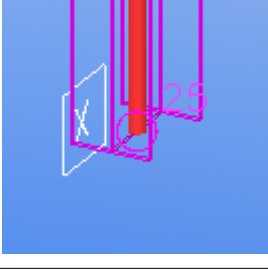
[Defining and modifying analysis part properties on page 68](#)

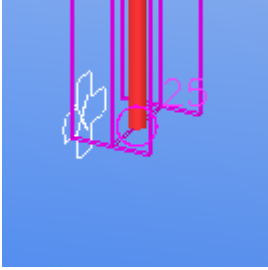
### Support condition symbols

Tekla Structures displays symbols for nodes that indicate the support conditions of a node.

Symbol	Support condition
	No supports



Symbol	Support condition
	Pinned connection
	Fixed connection
	Translational direction fixed
	Translational direction spring
	Rotational fixed

Symbol	Support condition
	Rotational spring

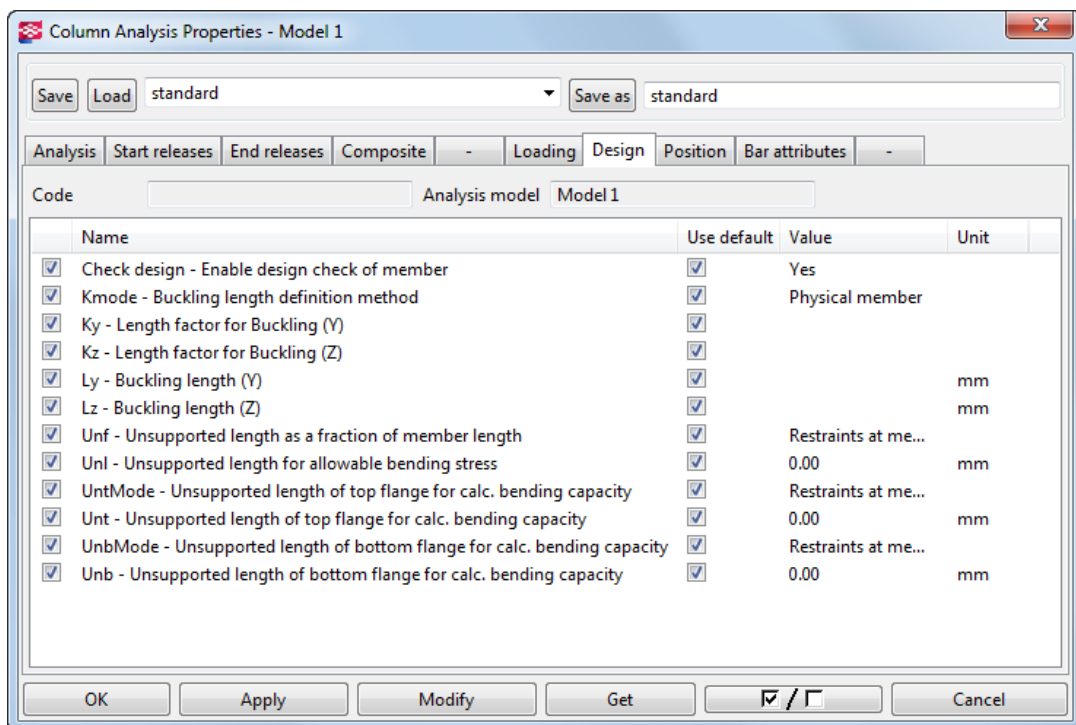
If you do not want to show the support condition symbols in model views, set the advanced option `XS_AD_SUPPORT_VISUALIZATION` to `FALSE` in **Tools --> Options --> Advanced Options...** --> **Analysis & Design** .

See also [Defining support conditions on page 70](#)

### 7.3 Defining design properties for analysis parts

You can define design properties for individual analysis parts. Design properties are properties which can vary according to the design code and the material of the part (for example, design settings, factors, and limits).

The properties you see when you first open the **Design** tab in an analysis part properties dialog box are the properties that apply to the entire analysis model you have selected in the **Analysis & Design Models** dialog box.



You can modify the design properties of specific analysis parts using the appropriate analysis part properties dialog boxes. When you change a value or select an option in the **Value** column, the check box in the **Use default** column is cleared indicating that the analysis model properties are not in use for this particular analysis part and design property.

**Example** If an analysis model contains parts with different material grades, define the most common material grade using the analysis model properties. Then change the material grade of specific parts in the analysis part properties.

**See also** [Omitting analysis parts from design on page 75](#)  
[Defining the buckling lengths of a column on page 75](#)  
[Defining the design properties of an analysis model on page 57](#)  
[Analysis part properties on page 113](#)

## Omitting analysis parts from design

You can omit individual analysis parts from the design check during the analysis.

Before you start, in the **Analysis & Design Models** dialog box, select the analysis model in which you want to modify the analysis part properties.

To omit an analysis part from the design check:

1. In the physical model, select a part.
2. Right-click and select **Analysis Properties** from the pop-up menu.
3. In the part's analysis properties dialog box:
  - a. Go to the **Design** tab.
  - b. In the **Value** column, select **No** for **Check design – Enable design check of member**.
  - c. Click **Modify**.

**See also** [Defining design properties for analysis parts on page 74](#)  
[Defining and modifying analysis part properties on page 68](#)

## Defining the buckling lengths of a column

You can define buckling lengths for columns and column segments. Column segments represent the building levels. Tekla Structures automatically divides columns into segments at the point where a support in the buckling direction exists, or where the column profile changes.

Effective buckling length is  $K \cdot L$ , where  $K$  is the length factor and  $L$  is the buckling length.

A column can have different buckling lengths in different analysis models.

Before you start, in the **Analysis & Design Models** dialog box, select the analysis model in which you want to define the buckling lengths.

To define the effective buckling lengths of a column:

1. Select the column.
2. Right-click and select **Analysis Properties** from the pop-up menu.
3. In the column's analysis properties dialog box:
  - a. Go to the **Design** tab and the **Value** column.
  - b. Select an option for **Kmode**.
  - c. Enter one or more values for **K – Length factor for buckling** in the y and/or z direction.

The number of values you can enter depends on the option you selected for **Kmode**.

To enter multiple values, enter a value for each column segment starting from the lowest segment, and use spaces to separate the values. You can also use multiplication to repeat factors, for example, 3\*2.00.

<input checked="" type="checkbox"/> Kmode - Buckling length definition method	<input type="checkbox"/> Column segment, multiple values
<input checked="" type="checkbox"/> Ky - Length factor for Buckling (Y)	<input type="checkbox"/> 1.00 1.50 2.00
<input checked="" type="checkbox"/> Kz - Length factor for Buckling (Z)	<input type="checkbox"/> 1.00 1.50 2.00

- d. Enter one or more values for **L – Buckling length** in the y and/or z direction.
  - To automatically calculate length values, leave the fields blank.
  - To override one or more length values, enter values in the relevant buckling length fields. The number of values you need to enter depends on the option you selected for **Kmode**. You can use multiplication to repeat buckling lengths, for example, 3\*4000.
- e. Click **Modify**.

See also [Kmode options on page 76](#)

[Defining and modifying analysis part properties on page 68](#)

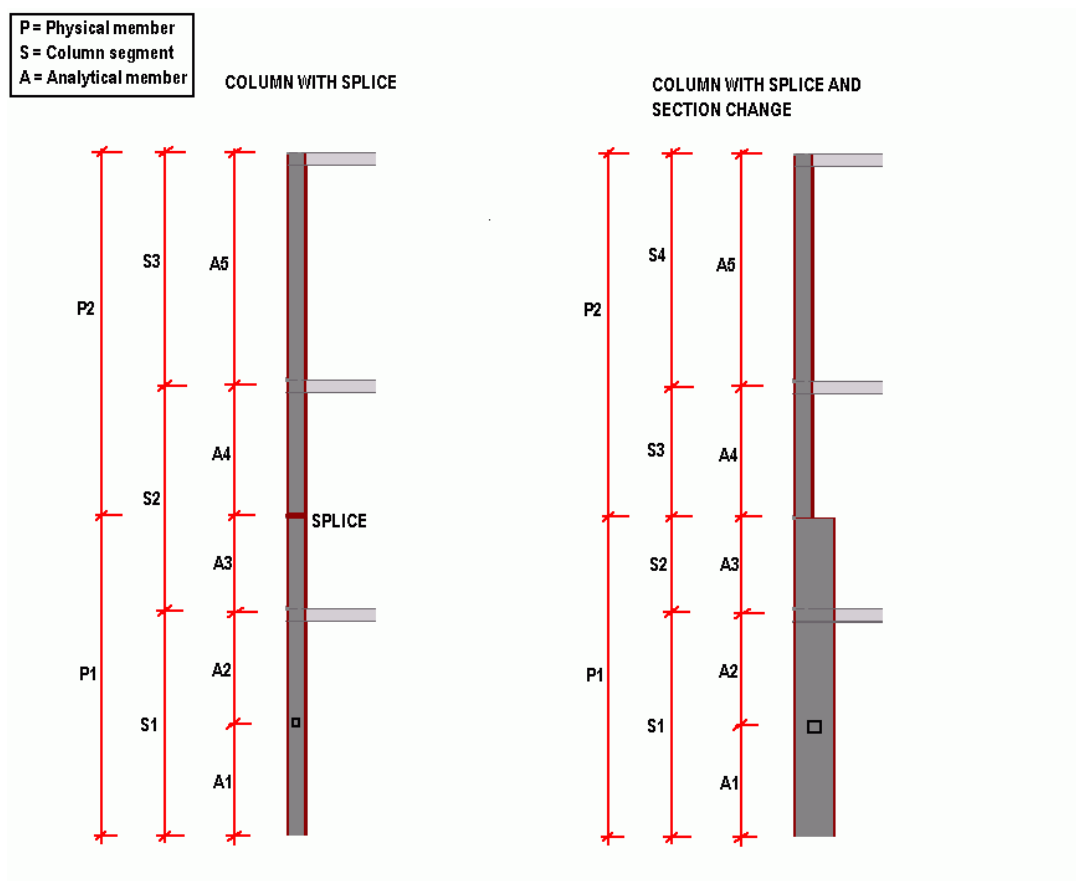
### ***Kmode options***

Use the **Kmode** options to define how Tekla Structures calculates the buckling lengths of columns.

The options are:

Option	Description
<b>Physical member</b>	L is the length of the column.
<b>Column segment</b>	L is the length of one column segment.
<b>Column segment, multiple values</b>	L is the length of one column segment with user-defined factors and lengths for each column segment.

Option	Description
Analytical member	L is the length of the member in the analysis model.
Analytical member, multiple values	L is the length of the member in the analysis model with user-defined factors and lengths for each member.



See also [Defining the buckling lengths of a column on page 75](#)

## 7.4 Defining the location of analysis parts

You can define and modify the analysis axis location of individual parts in an analysis model, or you can use the axis settings of the analysis model that apply to all parts in the analysis model.

You can also define offsets for analysis parts and use handles to move analysis parts.

If you move an analysis part handle, you can view the offsets in the following dialog boxes:

- **Analysis Bar Position Properties**
- **Analysis Area Position Properties**
- **Analysis Area Edge Properties**

If you move a physical part or an analysis part, these handle offsets will be reset. The **Reset Editing of Selected Parts** command also resets the changes you have made using the analysis part handles.

**See also** [Defining or modifying the axis location of an analysis part on page 78](#)

[Defining offsets for an analysis part on page 79](#)

[Resetting the editing of analysis parts on page 79](#)

[Analysis bar position properties on page 129](#)

[Analysis area position properties on page 129](#)

[Analysis area edge properties on page 130](#)

[Analysis part properties on page 113](#)

[Defining the axis settings of an analysis model on page 55](#)

## Defining or modifying the axis location of an analysis part

You can define and modify the analysis axis location of individual parts. The analysis axis defines the location of an analysis part in relation to the corresponding physical part. For example, the analysis part can be located on the neutral axis or the reference line of the physical part.

Before you start:

- In the **Analysis & Design Models** dialog box, select the analysis model in which you want to modify the analysis part properties.
- For the selected analysis model, ensure that **Member axis location** is **Model default** in the **Analysis Model Properties** dialog box.

To define or modify the axis location of an analysis part:

1. In the physical model, select a part.
2. Right-click and select **Analysis Properties** from the pop-up menu.
3. In the part's analysis properties dialog box:
  - a. Go to the **Position** tab.
  - b. In the **Axis** list, select an option.
  - c. In the **Keep axis position** list, define whether the part's analysis axis can move, and in which direction when the part is connected with other parts.
  - d. If needed, use the **Axis modifier** boxes to define whether the axis is bound to global coordinates, to the nearest grid line, or neither.
  - e. Click **Modify**.

**See also** [Defining offsets for an analysis part on page 79](#)

[Analysis part properties on page 113](#)

[Defining and modifying analysis part properties on page 68](#)

[Defining the axis settings of an analysis model on page 55](#)

## Defining offsets for an analysis part

You can define offsets for an analysis part. Offsets move the analysis part in relation to the default location of the analysis axis.

Before you start, in the **Analysis & Design Models** dialog box, select the analysis model in which you want to define offsets.

To define offsets for an analysis part:

1. In the physical model, select a part.
2. Right-click and select **Analysis Properties** from the pop-up menu.
3. In the part's analysis properties dialog box:
  - a. Go to the **Position** tab.
  - b. In the **Offset** boxes, define the offset of the analysis part from the physical part's analysis axis in the global x, y, and z directions.

These values change if you move the analysis part in the model.

These values do not reset if you move the physical part.
  - c. In the **Longitudinal offset mode** list, select whether the longitudinal end offsets **Dx** of the physical part are taken into account.

End offsets determine where Tekla Structures creates the end nodes of the analysis part.
  - d. Click **Modify**.

**See also** [Defining or modifying the axis location of an analysis part on page 78](#)

[Analysis part properties on page 113](#)

## Resetting the editing of analysis parts

If you have changed the location of analysis parts using handles, you can reset the selected analysis parts to the default analysis settings.

To reset parts to the default analysis settings in an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box, select the analysis model in which you want to reset parts.
3. Select the parts to reset.

4. Click **Analysis** --> **Reset Editing Of Selected Parts** or .

See also [Defining the location of analysis parts on page 77](#)

[Modifying analysis parts on page 68](#)

## 7.5 Copying an analysis part

You can create copies of existing analysis parts together with the applied properties and node offsets.

For example, you can use copying to apply analysis settings to multiple repeated frames. First apply the correct analysis settings to one frame. Then copy the settings to other similar frames.

To copy an analysis part:

1. Click **Analysis** --> **Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box, select the analysis model that includes the part you want to copy and uses the analysis part properties you want to use.
3. In the physical model, select the part to copy.
4. Click **Edit** --> **Copy**, or right-click and select **Copy** from the pop-up menu.
5. Pick the origin for the copying.
6. Pick one or more destination points.

If there is an identical physical part at a destination point, Tekla Structures creates an analysis part with settings identical to the original.

If there already was an analysis part at a destination point, Tekla Structures modifies the analysis part.

If a physical part at the destination point is not yet included in the analysis model, Tekla Structures adds the part to the analysis model.

7. To stop copying, click **Edit** --> **Interrupt**, or right-click and select **Interrupt** from the pop-up menu.

See also [Modifying analysis parts on page 68](#)

## 7.6 Deleting an analysis part

You can remove parts from analysis models by deleting analysis parts.

If the analysis model creation method is **Full model** and you delete an analysis part, Tekla Structures ignores the part in the analysis. If the analysis model creation method is **By**



**selected parts and loads** or **Floor model by selected parts and loads**, and you delete an analysis part, Tekla Structures removes the part from the analysis model.

To delete an analysis part:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, select the analysis model that includes the part you want to delete.
3. Select the analysis part to delete.
4. Do one of the following:
  - Click **Edit --> Delete**
  - Right-click and select **Delete** from the pop-up menu.
  - Press **Delete**.



To undo the **Delete** command:

- For **Full model** analysis models, change the analysis class of the deleted part from **Ignore** to the original setting.
- For other analysis models, add the deleted part again to the analysis model.

---

**See also** [Removing objects from an analysis model on page 63](#)

[Modifying analysis models on page 53](#)

[Analysis model creation method on page 49](#)

# 8 Combining loads

This section explains the load combination process in Tekla Structures.

Load combination is a process in which some simultaneously acting load groups are multiplied by their partial safety factors and combined with each other according to specific rules.

Load combination rules are specific to a design process and are defined in building or design codes. One of the most typical design processes is the limit state design.

Load combination properties define how Tekla Structures combines loads. The following properties control the load combination process:

- [Load modeling code options on page 105](#)
- [Load combination factors on page 105](#)
- [Load combination types on page 106](#)
- [Load group compatibility on page 20](#)

Click the links below to find out more:

- [About load combinations on page 82](#)
- [Creating load combinations automatically on page 83](#)
- [Creating a load combination on page 84](#)
- [Modifying a load combination on page 85](#)
- [Copying load combinations between analysis models on page 85](#)
- [Deleting load combinations on page 87](#)

## 8.1 About load combinations

A load combination is a set of load groups that is created in the load combination process. Each load combination represents a real loading situation, which means that permanent load should always be included in each load combination.

Each load combination must have a unique name. Use names that describe the loading situation.

Each load combination has an ID. This is an incremental number, based on the order in which load combinations are created in the analysis model.

You can have Tekla Structures automatically create load combinations, or you can create and modify them manually.

**See also** [Creating load combinations automatically on page 83](#)

[Creating a load combination on page 84](#)

[Modifying a load combination on page 85](#)

[Copying load combinations between analysis models on page 85](#)

[Deleting load combinations on page 87](#)

## 8.2 Creating load combinations automatically

You can have Tekla Structures automatically generate load combinations according to a building code.

Before you start, ensure that you have the appropriate load modeling code selected in **Tools --> Options --> Options... --> Load modeling --> Current code**.

To automatically create load combinations for an analysis model:

1. Click **Analysis --> Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box:
  - a. Select an analysis model.
  - b. Click **Load combinations....**
3. In the **Load Combinations** dialog box, click **Generate....**
4. In the **Load Combination Generation** dialog box:
  - a. If needed, check the load combination factors.  
Click **Options...**, and then do one of the following:
    - View the factors. Then click **Cancel** to close the dialog box.
    - Modify the factors. Then click **OK** to save the changes.
  - b. Select the check boxes against the combinations you want to create.
  - c. To automatically include the self-weight of parts in load combinations, select the **Include self weight** check box.
  - d. (This step only applies to the Eurocode.) If needed, select the **Minimum permanent load with lateral loads only** check box. This reduces the amount of load combinations when only minimum permanent loading needs to be considered in lateral loading situations.

- e. Click **OK** to create the load combinations.

If the analysis model has imperfection loads, Tekla Structures automatically creates load combinations with both the positive and negative directions (x and -x, or y and -y).

5. In the **Load Combination Generation** dialog box, click **OK** to save the load combinations.

**See also** [Setting the load modeling code on page 17](#)

[Load combination factors on page 105](#)

[Load combination types on page 106](#)

[Creating a load combination on page 84](#)

[Modifying a load combination on page 85](#)

[Deleting load combinations on page 87](#)

## 8.3 Creating a load combination

If needed, you can create load combinations one by one.

Before you start, ensure that you have the appropriate load modeling code selected in **Tools --> Options --> Options... --> Load modeling --> Current code**.

To create a load combination for an analysis model:

1. Click **Analysis --> Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box:
  - a. Select an analysis model.
  - b. Click **Load combinations....**
3. In the **Load Combinations** dialog box, click **New....**
4. In the **Load Combination** dialog box:
  - a. Select a load combination type from the **Type** list.
  - b. Enter a unique name for the load combination.
  - c. Use the arrow buttons to move load groups between the **Load groups** list and the **Load combination** table.
  - d. If needed, modify the signs (+ or -) and combination factors in the **Load combination** table by clicking a value.
  - e. Click **Apply** to create the load combination.
  - f. If needed, repeat steps a–e to create more load combinations.
  - g. Click **OK** to create the last load combination and close the dialog box.
5. In the **Load Combinations** dialog box, click **OK** to save the load combinations.

**See also** [Setting the load modeling code on page 17](#)  
[Load combination types on page 106](#)  
[Load combination factors on page 105](#)  
[Creating load combinations automatically on page 83](#)  
[Modifying a load combination on page 85](#)  
[Deleting load combinations on page 87](#)

## 8.4 Modifying a load combination

You can modify load combinations by changing the load combination name and factors.

To modify a load combination of an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box:
  - a. Select an analysis model.
  - b. Click **Load combinations...**
3. In the **Load Combinations** dialog box:
  - a. To change the name of a load combination, select it and enter a new name.
  - b. To change a load combination factor, select it and enter a new value.
  - c. Click **OK** to save the changes.

**Limitations** You cannot change the load combination type or ID, or add or remove load groups after you have created the load combination.

**See also** [Creating load combinations automatically on page 83](#)  
[Creating a load combination on page 84](#)  
[Copying load combinations between analysis models on page 85](#)  
[Deleting load combinations on page 87](#)

## 8.5 Copying load combinations between analysis models

You can copy load combinations between analysis models within a physical model. You can also copy between physical models if they have the same environment and load groups.

First you need to save the load combinations that you want to copy to a `.lco` file. If you want to make the load combinations available in another physical model, you need to copy

the `.lco` file to the `\attributes` folder of the destination model, or to the project or firm folder. Then you can load the load combinations to another analysis model.

Click the links below to find out more:

- [Saving load combinations for later use on page 86](#)
- [Copying load combinations from another analysis model on page 86](#)

## Saving load combinations for later use

You can save the load combinations of an analysis model for later use in other analysis models.

To save the load combinations of an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select an analysis model.
  - b. Click **Load combinations....**
3. In the **Load Combinations** dialog box:
  - a. Enter a name for the saved load combinations in the box next to **Save as**.
  - b. Click **Save as**.  
Tekla Structures saves the load combinations as a `.lco` file in the `\attributes` folder under the current model folder.
4. Click **OK** to close the dialog boxes.

**See also** [Copying load combinations from another analysis model on page 86](#)

## Copying load combinations from another analysis model

You can copy load combinations from another analysis model that has the same load groups and environment.

To copy load combinations from an analysis model to another:

1. Ensure that the load combinations you want to copy have been saved in a `.lco` file.
2. Check that the `.lco` file is located in the `\attributes` folder under the current model folder, or in the project or firm folder. If not, copy the `.lco` file.
3. If you are copying load combinations between two physical models, open the model to copy to. If you are copying within a physical model, reopen the model.
4. Click **Analysis** --> **Analysis & Design Models...** .
5. In the **Analysis & Design Models** dialog box:

- a. Select the analysis model to copy to.
  - b. Click **Load combinations....**
6. In the **Load Combinations** dialog box:
  - a. Select a load combinations file (.lco) from the list next to **Load**.
  - b. Click **Load**.
7. Click **OK** to close the dialog boxes.

**See also** [Saving load combinations for later use on page 86](#)

## 8.6 Deleting load combinations

You can delete load combinations one by one, or several selected or all load combinations of an analysis model at once.

To delete load combinations of an analysis model:

1. Click **Analysis --> Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model whose load combinations you want to delete.
  - b. Click **Load combinations....**
3. In the **Load Combinations** dialog box, do one of the following:
  - Select the load combination to delete, and then click **Remove**.
  - Hold down the **Ctrl** or **Shift** key and select the load combinations to delete. Then click **Remove**.
  - To delete all load combinations, click **Remove all**.
4. Click **OK** to close the dialog boxes.

**See also** [Modifying a load combination on page 85](#)

[Creating load combinations automatically on page 83](#)

[Creating a load combination on page 84](#)

# 9 Working with analysis and design models

This section explains how to export, merge, and view analysis and design models and how to save and view analysis results.

Click the links below to find out more:

- [Checking warnings about an analysis model on page 88](#)
- [Exporting an analysis model on page 89](#)
- [Merging Tekla Structures analysis models with analysis applications on page 90](#)
- [Saving analysis results on page 92](#)
- [Viewing the analysis results of a part on page 93](#)
- [Showing analysis class in model views on page 94](#)
- [Showing analysis bar, member, and node numbers on page 94](#)
- [Showing the utilization ratio of parts on page 95](#)

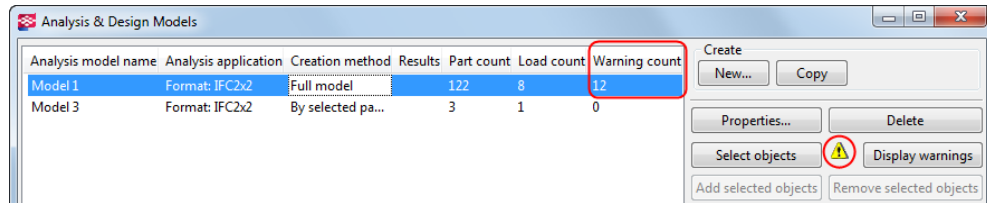
## 9.1 Checking warnings about an analysis model

If there were problems in creating an analysis model, Tekla Structures shows a warning sign in the **Analysis & Design Models** dialog box when you select the analysis model.

To check warnings about an analysis model:

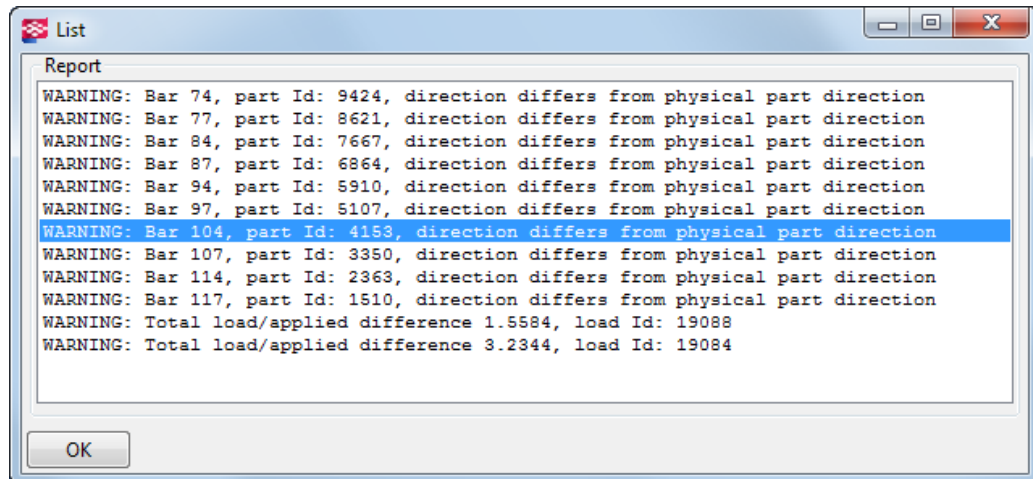
1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select an analysis model.
  - b. If a warning sign appears, click **Display warnings**.





3. In the warning dialog box, click **Details...** to find out more.

Tekla Structures displays a list of warnings:



- If you select a row with an object ID, Tekla Structures highlights the corresponding object in the physical model.
- If you right-click a row with an object ID, you can access the object's pop-up menu.

See also [Creating analysis models on page 47](#)

[Working with analysis and design models on page 88](#)

## 9.2 Exporting an analysis model

To run structural analysis on a Tekla Structures analysis model using an analysis application, you need to export the analysis model into a folder. By default, the export folder is the current model folder. If you have a direct link to an analysis application, and you export an analysis model from Tekla Structures using that particular analysis application, the analysis model is opened in the application.

To export an analysis model:

1. Click **Analysis** --> **Analysis & Design Models...**
2. If needed, define the export folder.
  - a. In the **Analysis & Design Models** dialog box, select the analysis model to export, and then click **Properties....**

- b. In the **Analysis Model Properties** dialog box, click **Browse for export folder** on the **Analysis model** tab.
  - c. In the **Browse For Folder** dialog box, browse for the export folder, and then click **OK**.
  - d. Click **OK** to save the export folder settings with the analysis model properties.
3. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model to export.
  - b. Click **Export**.

**See also** [Structural analysis workflow in Tekla Structures on page 13](#)  
[Working with analysis and design models on page 88](#)

### 9.3 Merging Tekla Structures analysis models with analysis applications

You can merge Tekla Structures analysis models with some external analysis applications. This means that you can make changes to Tekla Structures physical and analysis models even after you have exported them to an analysis application, and still keep the additions you have made to the exported models in the analysis application.

For example, you can create a Tekla Structures model, create an analysis model of it, export the analysis model to an analysis application, add special loads to the model in the analysis application, and then run the analysis. If you then need to make changes to the physical or analysis model in Tekla Structures, you can merge models with the analysis application. If you do not merge models and you re-export the changed Tekla Structures analysis model to the analysis application, you will lose the additions you have made to the model in the analysis application.

**See also** [Merging Tekla Structures analysis models with SAP2000 on page 90](#)  
[Merging a Tekla Structures analysis model with a model in SAP2000 on page 91](#)  
[Resetting merged analysis models on page 92](#)

#### Merging Tekla Structures analysis models with SAP2000

You can merge Tekla Structures analysis models with SAP2000.

By default, Tekla Structures and SAP2000 analysis models are not merged. This means that a new SAP2000 model is created always when you export a Tekla Structures analysis model to SAP2000.

If you choose to merge a Tekla Structures analysis model with SAP2000, the changes in the Tekla Structures physical or analysis model are merged to the model in SAP2000. Additional objects and definitions, such as parts, bars, loads, and load combinations, created in SAP2000 are retained in SAP2000. Additional objects created in SAP2000 cannot be imported to Tekla

Structures, but they are taken into account in the analysis. They affect the analysis results, which you can import to Tekla Structures.

When exported to SAP2000, the objects created in Tekla Structures will receive a prefix "\_" to their names. The prefix distinguishes the objects created in Tekla Structures from the objects created in SAP2000.

Additional loads created in SAP2000 will be added to the load combinations that are created in SAP2000. If you add additional loads to the load combinations that are created in Tekla Structures, the loads will be removed from these load combinations when you merge models and export a Tekla Structures analysis model to SAP2000.

Merging Tekla Structures and SAP2000 analysis models helps in retaining the existing analysis node and bar numbers in SAP2000.

- Existing node numbers are kept if the node coordinates stay the same.
- Existing bar numbers are kept if the start and end node numbers stay the same.
- Old node and bar numbers are not re-used.

**Limitations** Changes in the following properties in Tekla Structures are not updated in SAP2000 even if you merge models:

- The profile and material properties of parts if a profile or material name already exists in SAP2000
- Load combinations if the name of the load combination already exists in SAP2000

To keep the changes made in SAP2000 when you re-export a changed Tekla Structures analysis model, you can adjust the profile and material properties and the load combination type in SAP2000.

If you change the support condition settings in SAP2000 and then re-export a Tekla Structures analysis model, you will lose these changes.

**See also** [Merging a Tekla Structures analysis model with a model in SAP2000 on page 91](#)

## Merging a Tekla Structures analysis model with a model in SAP2000

You can merge Tekla Structures analysis models with models in SAP2000.

To merge a Tekla Structures analysis model with an existing model in SAP2000:

1. In Tekla Structures, click **Analysis --> Analysis & Design Models...**
2. In the **Analysis & Design Models** dialog box, do one of the following:
  - To merge an existing analysis model, select the analysis model, and then click **Properties...** to check and modify its properties.
  - To create a new analysis model and merge it, click **New....**
3. In the **Analysis Model Properties** dialog box:
  - a. In the **Analysis application** list, select **SAP2000**.

- b. Click **More settings...** on the **Analysis model** tab.
- c. In the **Model merging with analysis application** list, select **Enabled**.
- d. If you are merging a new analysis model, modify the other analysis model properties if needed.
- e. Click **OK** to save the analysis model properties.

Tekla Structures merges the models the next time you export the Tekla Structures analysis model to SAP2000 to run the analysis.

**See also** [Merging Tekla Structures analysis models with SAP2000 on page 90](#)  
[Resetting merged analysis models on page 92](#)

## Resetting merged analysis models

You can reset model merging between Tekla Structures and external analysis applications.

To reset the model merging of an analysis model:

1. In Tekla Structures, click **Analysis** --> **Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select the analysis model to reset.
  - b. Click **Properties...**
3. In the **Analysis Model Properties** dialog box:
  - a. Click **More settings...** on the **Analysis model** tab.
  - b. In the **Model merging with analysis application** list, select **Disabled**.
  - c. Click **OK** to save the analysis model properties.

**See also** [Merging a Tekla Structures analysis model with a model in SAP2000 on page 91](#)  
[Merging Tekla Structures analysis models with analysis applications on page 90](#)

## 9.4 Saving analysis results

When you save the analysis results and then save the physical model, Tekla Structures saves the results of all load combinations in a database, `analysis_results.db5`, in the current model folder.

If you do not want to create the analysis results database `analysis_results.db5`, set `XS_AD_RESULT_DATABASE_ENABLED` to `FALSE` in **Tools** --> **Options** --> **Advanced Options...** --> **Analysis & Design** .

Use the following advanced options in **Tools --> Options --> Advanced Options... --> Analysis & Design** to define the analysis member points whose results are saved in the database:

- XS\_AD\_MEMBER\_RESULT\_DIVISION\_COUNT
- XS\_AD\_MEMBER\_RESULT\_DISP\_DIVISION\_COUNT
- XS\_AD\_MEMBER\_RESULT\_MIN\_DISTANCE
- XS\_AD\_MEMBER\_RESULT\_GRID\_SIZE

See also [Saving analysis results as user-defined attributes of parts on page 93](#)

## Saving analysis results as user-defined attributes of parts

After running the analysis, you can save the maximum axial force, shear force, and bending moment at the part ends as user-defined attributes in the part properties. You can save the results for each part in an analysis model or for specific parts.

Before you start, run the analysis.

To save the results of an analysis model as user-defined attributes:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box:
  - a. Select an analysis model.
  - b. Do one of the following:
    - To save the results for each part in the analysis model, click **Get results**.
    - To save the results for specific parts, select the parts in the physical model, and then click **Get results for selected**.

See also [Viewing the analysis results of a part on page 93](#)

[Showing the utilization ratio of parts on page 95](#)

## 9.5 Viewing the analysis results of a part

You can view a part's analysis results using the user-defined attributes.

Before you start, ensure that you have saved the analysis results using the **Get results** or **Get results for selected** command on the correct analysis model.

To view the analysis results of a part in an analysis model:

1. Double-click a part in the physical model.

2. In the part properties dialog box, click **User-defined attributes...** on the **Attributes** tab.
3. In the user-defined attributes dialog box:
  - Go to the **End Conditions** tab to view the analysis results at the part ends.
  - Go to the **Analysis** tab to view the utilization ratio of a steel part or the required area of reinforcement in a concrete part.

To access the analysis results database, use the .NET interface or Tekla Structures's excel design interface.

**See also** [Saving analysis results as user-defined attributes of parts on page 93](#)  
[Saving analysis results on page 92](#)

## 9.6 Showing analysis class in model views

The analysis class defines how Tekla Structures handles individual parts in the analysis. You can show the analysis class of parts using different colors in the analysis model.

Before you start, create an object group that includes the parts whose analysis class you want to show.

To indicate the analysis class of parts in an object group using colors:

1. Click **Analysis --> Analysis & Design Models...** .
2. In the **Analysis & Design Models** dialog box, select an analysis model.
3. Click **View --> Representation --> Object Representation...** .
4. In the **Object Representation** dialog box:
  - a. Select an object group.
  - b. In the **Color** column, select **Color by analysis type** from the list.
  - c. Click **Modify**.

**See also** [Analysis class options and colors on page 122](#)

## 9.7 Showing analysis bar, member, and node numbers

You can show analysis bar, member, and analysis node numbers in model views.

Use the following advanced options in **Tools --> Options --> Advanced Options... --> Analysis & Design** to define which numbers are shown:

- XS\_AD\_MEMBER\_NUMBER\_VISUALIZATION
- XS\_AD\_NODE\_NUMBER\_VISUALIZATION

- XS\_AD\_NODE\_NUMBER\_BY\_Z

Some analysis applications work on analysis members whereas others work on analysis bars. This also affects how analysis models are shown in Tekla Structures model views. Either member numbers or bar numbers are shown.

See also

## 9.8 Showing the utilization ratio of parts

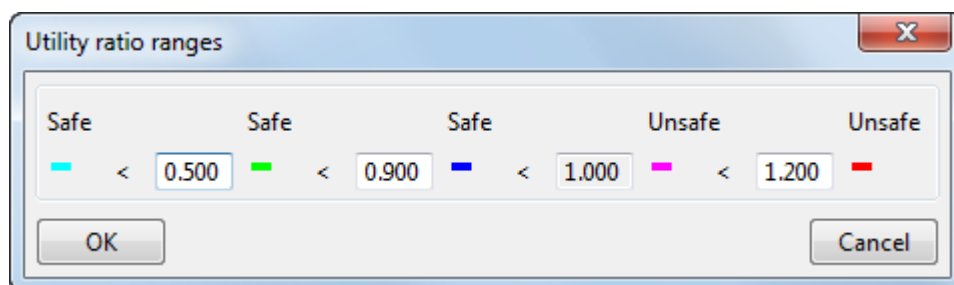
Once you have exported an analysis model to an analysis application and run the analysis, you can view the analysis results. To perform a visual check, you can use different colors to show the utilization ratio of steel parts in the physical model.

Before you start, ensure that you have saved the analysis results using the **Get results** or **Get results for selected** command on the correct analysis model.

To show the utilization ratio of the steel parts in an object group:

1. Create an object group that includes the parts whose utilization ratio you want to show.
2. Click **View --> Representation --> Object Representation...**
3. In the **Object Representation** dialog box:
  - a. Select the object group whose utilization ratios you want to show.
  - b. In the **Color** column, select **Color by analysis utility check...** from the list.
4. In the **Utility Ratio Ranges** dialog box:
  - a. Set the ranges of ratio for each of the colors that Tekla Structures uses to show safe and unsafe parts.
  - b. Click **OK**.
5. In the **Object Representation** dialog box, click **Modify**.

Tekla Structures shows the utilization ratio of the steel parts in the selected analysis model using the following colors:



See also [Exporting an analysis model on page 89](#)

[Saving analysis results as user-defined attributes of parts on page 93](#)

[Viewing the analysis results of a part on page 93](#)



# 10 Analysis and design settings

This section provides information about the various analysis and design settings you can modify in Tekla Structures.

Click the links below to find out more:

- [Load group properties on page 97](#)
- [Load properties on page 98](#)
- [Load combination properties on page 105](#)
- [Analysis model properties on page 107](#)
- [Analysis part properties on page 113](#)
- [Analysis node properties on page 126](#)
- [Analysis rigid link properties on page 127](#)
- [Analysis bar position properties on page 129](#)
- [Analysis area position properties on page 129](#)
- [Analysis area edge properties on page 130](#)

## 10.1 Load group properties

Use the **Load Groups** dialog box to view, define, and modify the load group properties and to work with load groups.

Option	Description
<b>Current</b>	<p>The @ symbol identifies the current load group.</p> <p>When you create loads in the model, Tekla Structures adds them to the current load group. You can only define one load group as current.</p> <p>To change the current load group, select a load group and click <b>Set current</b>.</p>

Option	Description
<b>Name</b>	<p>Unique name of the load group.</p> <p>Use load group names to define the visibility and selectability of loads. For example, you can select, modify, or hide loads based on their load group.</p>
<b>Type</b>	<p>The type of a load group is the type of action that causes the loads.</p> <p>Actions causing loads are building code specific. Most building codes use some or all of the following actions and load group types:</p> <ul style="list-style-type: none"> <li>• Permanent, dead, and/or prestressing loads</li> <li>• Live, imposed, traffic, and/or crane loads</li> <li>• Snow loads</li> <li>• Wind loads</li> <li>• Temperature loads</li> <li>• Accidental and/or earthquake loads</li> <li>• Imperfection loads</li> </ul>
<b>Direction</b>	<p>The direction of a load group is the global direction of the action that causes the loads. Individual loads in a load group retain their own magnitudes in the global or local x, y, and z directions.</p> <p>The load group direction affects which loads Tekla Structures combines in a load combination:</p> <ul style="list-style-type: none"> <li>• z direction groups are combined with both x and y direction groups.</li> <li>• x or y direction groups are <b>not</b> combined with each other.</li> </ul>
<b>Compatible</b>	A number that identifies all the load groups that are compatible with each other.
<b>Incompatible</b>	A number that identifies all the load groups that are incompatible with each other.
<b>Color</b>	The color that Tekla Structures uses to show the loads in the group.

See also [Grouping loads on page 18](#)  
[Working with loads and load groups on page 40](#)

## 10.2 Load properties

This section provides information about the properties of specific loads.

Use the load properties dialog boxes to view, define, and modify the load properties. Each load type has its own properties dialog box.

Click the links below to find out more:

- [Point load properties on page 99](#)
- [Line load properties on page 99](#)
- [Area load properties on page 100](#)
- [Uniform load properties on page 101](#)
- [Temperature load properties on page 101](#)
- [Wind load properties on page 102](#)
- [Load panel settings on page 103](#)

## Point load properties

Use the **Point Load Properties** dialog box to view and modify the properties of a point load or a bending moment. The file name extension of a point load properties file is `.lml`.

Option	Description
<b>Load group name</b>	The load group to which the load belongs.  To view load group properties or to create a new load group, click <b>Load groups...</b>
<b>Magnitude tab</b>	Load magnitudes in the x, y, and z directions of the work plane.
<b>Load attachment</b>	Indicates if the load is attached to a part.
<b>Load-bearing parts</b>	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
<b>Bounding box of the load</b>	Dimensions of the bounding box in the x, y, and z directions.
<b>Load panel tab</b>	See <a href="#">Load panel settings on page 103</a> .

See also [Creating a point load on page 24](#)  
[Defining the properties of a load on page 22](#)  
[Load magnitude on page 23](#)  
[Attaching loads to parts or locations on page 31](#)  
[Applying loads to parts on page 31](#)  
[Modifying the distribution of a load on page 35](#)

## Line load properties

Use the **Line Load Properties** dialog box to view and modify the properties of a line load or a torsional moment. The file name extension of a line load properties file is `.lml`.

Option	Description
<b>Load group name</b>	The load group to which the load belongs.  To view load group properties or to create a new load group, click <b>Load groups....</b>
<b>Magnitude tab</b>	Load magnitudes in the x, y, and z directions of the work plane.
<b>Load form</b>	Defines how the load magnitude varies along the loaded length.
<b>Load attachment</b>	Indicates if the load is attached to a part.
<b>Load-bearing parts</b>	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
<b>Bounding box of the load</b>	Dimensions of the bounding box in the x, y, and z directions.
<b>Distances</b>	Offsets from the load end points, used to shorten or lengthen the loaded length.  To shorten the loaded length, enter positive values for <b>a</b> and <b>b</b> . To lengthen the loaded length, enter negative values.
<b>Load panel tab</b>	See <a href="#">Load panel settings on page 103</a> .

See also [Creating a line load on page 25](#)  
[Defining the properties of a load on page 22](#)  
[Load magnitude on page 23](#)  
[Load form on page 24](#)  
[Distributing and modifying loads on page 31](#)

## Area load properties

Use the **Area Load Properties** dialog box to view and modify the properties of an area load.  
The file name extension of an area load properties file is `.lm3`.

Option	Description
<b>Load group name</b>	The load group to which the load belongs.  To view load group properties or to create a new load group, click <b>Load groups....</b>
<b>Magnitude tab</b>	Load magnitudes in the x, y, and z directions of the work plane.
<b>Load form</b>	Defines the shape of the loaded area.
<b>Load attachment</b>	Indicates if the load is attached to a part.
<b>Load-bearing parts</b>	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
<b>Bounding box of the load</b>	Dimensions of the bounding box in the x, y, and z directions.

Option	Description
<b>Distances</b>	Offset used to enlarge or reduce the loaded area. To enlarge the loaded area, enter a positive value for <b>a</b> . To reduce the loaded area, enter a negative value.
<b>Load panel</b> tab	See <a href="#">Load panel settings on page 103</a> .

See also [Creating an area load on page 25](#)  
[Defining the properties of a load on page 22](#)  
[Load magnitude on page 23](#)  
[Load form on page 24](#)  
[Distributing and modifying loads on page 31](#)

## Uniform load properties

Use the **Uniform Load Properties** dialog box to view and modify the properties of a uniform load. The file name extension of a uniform load properties file is `.lm4`.

Option	Description
<b>Load group name</b>	The load group to which the load belongs. To view load group properties or to create a new load group, click <b>Load groups...</b>
<b>Magnitude</b> tab	Load magnitudes in the x, y, and z directions of the work plane.
<b>Load attachment</b>	Indicates if the load is attached to a part.
<b>Load-bearing parts</b>	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
<b>Bounding box of the load</b>	Dimensions of the bounding box in the x, y, and z directions.
<b>Distances</b>	Offset used to enlarge or reduce the loaded area.
<b>Load panel</b> tab	See <a href="#">Load panel settings on page 103</a> .

See also [Creating a uniform load on page 26](#)  
[Defining the properties of a load on page 22](#)  
[Load magnitude on page 23](#)  
[Distributing and modifying loads on page 31](#)

## Temperature load properties

Use the **Temperature Load Properties** dialog box to view and modify the properties of a temperature load or a strain. The file name extension of a temperature load properties file is `.lm6`.

Option	Description
<b>Load group name</b>	The load group to which the load belongs.  To view load group properties or to create a new load group, click <b>Load groups...</b>
<b>Temperature change for axial elongation</b>	Temperature change in the part.
<b>Temperature differential from side to side</b>	Difference in temperature between the left side and the right side of a part.
<b>Temperature differential from top to bottom</b>	Difference in temperature between the top surface and the bottom surface of a part.
<b>Initial axial elongation</b>	Axial strain of a part.  A positive value indicates elongation, a negative value indicates shrinkage.
<b>Load attachment</b>	Indicates if the load is attached to a part.
<b>Load-bearing parts</b>	Parts to which the load is applied, or not applied, on the basis of part names or selection filters.
<b>Bounding box of the load</b>	Dimensions of the bounding box in the x, y, and z directions.

See also [Creating a temperature load or a strain on page 27](#)

[Defining the properties of a load on page 22](#)

[Applying loads to parts on page 31](#)

## Wind load properties

Use the **Wind Load Generator (28)** dialog box to view and modify the properties of wind loads.

Option	Description
<b>Wind load direction</b>	The main direction of the wind.  The options are: <ul style="list-style-type: none"> <li>• <b>Global X</b></li> <li>• <b>Global -X</b></li> <li>• <b>Global Y</b></li> <li>• <b>Global -Y</b></li> <li>• <b>Global X, -X, Y, -Y</b> (for all directions)</li> </ul>
<b>Nominal wind pressure</b>	The nominal value of wind pressure.
<b>Top level</b>	The highest level of the wind loads.
<b>Bottom level</b>	The lowest level of the wind loads.
<b>Ground level</b>	The level of the ground around the building.

Option	Description
<b>Part names</b>	Parts to which the load is applied, or not applied.
<b>Front</b>	The external exposure factors for the windward, leeward, and side walls.  A positive value indicates pressure, a negative value indicates suction.
<b>Left side</b>	
<b>Back</b>	
<b>Right side</b>	
<b>Internal</b>	The internal exposure factor.
<b>Z profile tab</b>	The distribution of wind load along the height of the building, in terms of pressure factors. Starts from the ground level.
<b>Global X, Global Y, Global -X, Global -Y tabs</b>	<p>A tab for each wind direction, where you can define zones for concentrated corner loads on each wall.</p> <p>Each zone is the height of the wall. Define the width of the zone using either dimensions or proportions. You can define up to five zones for each wall.</p> <p>Walls are numbered according to the order you pick points to indicate the shape of the building on the bottom level.</p>

See also [Creating wind loads on page 28](#)

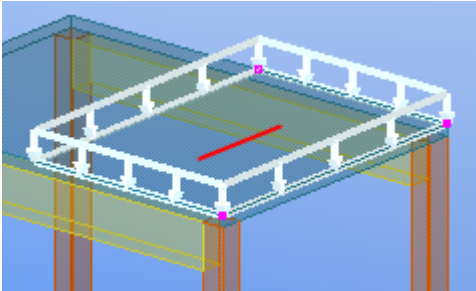
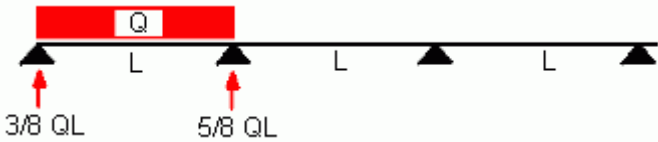
[Wind load examples on page 28](#)

[Defining load-bearing parts by name on page 32](#)

## Load panel settings

Use the options on the **Load panel** tab in a load properties dialog box to modify the way Tekla Structures distributes the load.

Option	Description
<b>Spanning</b>	<p>Defines the directions in which Tekla Structures distributes the load.</p> <ul style="list-style-type: none"> <li>• <b>Single</b> distributes the load only in the direction of the primary axis.</li> <li>• <b>Double</b> distributes the load along the primary and secondary axes.</li> </ul>
<b>Primary axis direction</b>	<p>Defines the direction of the primary axis using one of the following methods:</p> <ul style="list-style-type: none"> <li>• A value (1) in the <b>x</b>, <b>y</b>, or <b>z</b> box distributes the load in the corresponding global direction.</li> <li>• Values in multiple boxes distribute the load between the corresponding global directions. The values are the components of the direction vector.</li> </ul>

Option	Description
	<ul style="list-style-type: none"> <li>Clicking <b>Parallel to part...</b>, or <b>Perpendicular to part...</b>, and then selecting a part in the model aligns the primary axis direction with the part.</li> </ul> <p>If <b>Spanning</b> is <b>Double</b>, you need to define the primary axis direction to be able to manually define the primary axis weight.</p> <p>To check the primary axis direction of a selected load in a model view, click <b>Show direction on selected loads</b>. Tekla Structures indicates the primary direction using a red line.</p> 
<b>Automatic primary axis weight</b>	<p>Defines whether Tekla Structures automatically weights the directions in load distribution.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li><b>Yes:</b> Tekla Structures automatically calculates the load portions for the primary and secondary directions in proportion to the third power of the span lengths in these two directions. This means that the shorter the span, the bigger the proportion of the load.</li> <li><b>No:</b> You can enter the weight for the primary direction in the <b>Weight</b> box. Tekla Structures calculates the weight for the secondary direction by subtracting this value from 1.</li> </ul>
<b>Load dispersion angle</b>	<p>The angle by which the load is projected onto the surrounding parts.</p>
<b>Use continuous structure load distribution</b>	<p>Use for uniform loads on continuous slabs. Defines the distribution of support reactions in the first and last spans.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li><b>Yes:</b> The distribution of support reactions is <math>3/8</math> and <math>5/8</math>.</li> </ul>  <ul style="list-style-type: none"> <li><b>No:</b> The distribution of support reactions is <math>1/2</math> and <math>1/2</math>.</li> </ul>

See also [Modifying the distribution of a load on page 35](#)



## 10.3 Load combination properties

This section provides information about the settings that control the load combination process.

Click the links below to find out more:

- [Load modeling code options on page 105](#)
- [Load combination factors on page 105](#)
- [Load combination types on page 106](#)

### Load modeling code options

These are the load modeling codes available in Tekla Structures in **Tools --> Options --> Options...** --> **Load modeling --> Current code** :

Option	Description
<b>Eurocode</b>	European code
<b>British</b>	British code
<b>AISC (US)</b>	American Institute of Steel Construction, US code
<b>UBC (US)</b>	Uniform building code, US code
<b>CM66 (F)</b>	French code for steel structures
<b>BAEL91 (F)</b>	French code for concrete structures
<b>IBC (US)</b>	International building code, US code
<b>ACI</b>	American Concrete Institute's publication 318

Each of the available codes has a separate tab in the **Options** dialog box. The **Options** dialog box lists the partial safety factors in limit states and other combination factors for the code, based on load group types. For the Eurocode, you can also set the reliability class factor and the formula to be used in load combination.

See also [Setting the load modeling code on page 17](#)

[Load combination factors on page 105](#)

### Load combination factors

In the load combination process, Tekla Structures uses partial safety factors and, for example, reduction factors on load groups to create load combinations.

The *partial safety factors* needed in the limit state design are:

- Unfavorable partial safety factor in the ultimate limit state ( $\gamma_{sup}$ )
- Favorable partial safety factor in the ultimate limit state ( $\gamma_{inf}$ )

- Unfavorable partial safety factor in the serviceability limit state ( $\gamma_{sup}$ )
- Favorable partial safety factor in the serviceability limit state ( $\gamma_{inf}$ )

Depending on the codes you use, you may need to use other combination factors. For example, the Eurocode contains three *reduction factors* ( $\psi_0, \psi_1, \psi_2$ ). Reduction factors exclude the impractical effects of simultaneous loads.

You can use values for load combination factors that are building code specific or user-defined.

See also [Setting the load modeling code on page 17](#)  
[Using non-standard load combination factors on page 17](#)

## Load combination types

You can perform several load combination types which vary according to the building code you use.

Use the **Load Combination Generation** dialog box, or the **Load Combination** dialog box, to select the load combination types you want to create. The options are:

Combination type	Description	Applies to
<b>Ultimate limit state (ULS)</b>	Combines load groups that occur persistently and transiently. Uses the partial safety factors of the ultimate limit state when combining loads.	Eurocode, British, AISC (US)
<b>Serviceability limit state (SLS)</b>	Combines load groups that occur quasi-permanently. Uses the partial safety factors of the serviceability limit state when combining loads.	Eurocode, AISC (US)
<b>Serviceability limit state – Rare (SLS RC)</b>	Combines load groups that occur quasi-permanently and rarely. Uses the partial safety factors of the serviceability limit state when combining loads.	Eurocode
<b>Serviceability limit state – Quasi-permanent (SLS QP)</b>	Combines load groups that occur quasi-permanently. Uses the partial safety factors of the serviceability limit state when combining loads.	Eurocode
<b>Normal loads</b>	Combines load groups and uses factors according to the French codes CM66 or BAEL91.	CM66, BAEL91
<b>Extreme loads</b>		CM66
<b>Displacement loads</b>		CM66
<b>Accidental loads</b>		CM66, Eurocode
<b>Ultimate loads</b>		BAEL91
<b>Ultimate accidental loads</b>		BAEL91

Combination type	Description	Applies to
Earthquake loads	Combines load groups and uses factors according to the Eurocode.	Eurocode
Loads for public structures	Combines load groups according to the US IBC code (International Building Code).	IBC (US)
Loads for public structures with drifted snow		IBC (US)
Loads for non public structures		IBC (US)
Loads for non public structures with drifted snow		IBC (US)
Loads for public non concrete and masonry structures	Combines load groups according to the US UBC code (Uniform Building Code).	UBC (US)
Loads for public non concrete and masonry structures with drifted snow		UBC (US)
Loads for non concrete and masonry structures		UBC (US)
Loads for non concrete and masonry structures with drifted snow		UBC (US)
Loads for public concrete and masonry structures		UBC (US)
Loads for public concrete and masonry structures with drifted snow		UBC (US)
Loads for concrete and masonry structures		UBC (US)
Loads for concrete and masonry structures with drifted snow		UBC (US)
ACI Table 1 – ACI Table 8		ACI

See also [Combining loads on page 82](#)

## 10.4 Analysis model properties

Use the **Analysis Model Properties** dialog box to define, view, and modify the properties of an analysis model. These properties apply to all parts in an analysis model.

Analysis model  
tab

Option	Description
<b>Analysis model name</b>	<p>A unique name for the analysis model. User-definable.</p> <p>For example, you can use a name that describes the portion of the physical model that you want to analyze.</p> <p>To define the export folder for the analysis model, click <b>Browse for export folder</b>.</p>
<b>Creation method</b>	<p>Defines which objects are included in the analysis model.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Full model</b></li> <li>• <b>By selected parts and loads</b></li> <li>• <b>Floor model by selected parts and loads</b></li> </ul>
<b>Filter</b>	<p>Defines which objects to include in the analysis model, based on the list of available selection filters.</p>
<b>Secondary member filter</b>	<p>Defines which of the included objects are considered to be secondary analysis parts. The nodes of secondary analysis parts can move more freely than the ones of primary analysis parts.</p> <p>To have Tekla Structures automatically recognize, for example, skewed parts as secondary parts, select <b>Auto-detect secondary members</b>.</p> <p>If you change this setting for an existing analysis model, select the <b>Reapply to all parts</b> check box to apply the change to all parts in the analysis model. If you do not select the check box, Tekla Structures will use the new setting only for the new parts in the analysis model.</p>
<b>Analysis application</b>	<p>The analysis application or format used in the analysis of the analysis model.</p> <p>To use the same application or format by default for other new analysis models, select the <b>Set as the default</b> check box.</p>
<b>More settings...:</b> Click to show the following settings.	
<b>Use rigid links</b>	<p>Use to allow or prevent rigid links in the analysis model.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Enabled</b></li> </ul> <p>Rigid links are created if they are needed to connect analysis parts.</p> <ul style="list-style-type: none"> <li>• <b>Disabled, with keep axis: Default</b></li> </ul> <p>No rigid links are created. The <b>Keep axis position</b> settings of the analysis parts are not changed.</p>

Option	Description
	<ul style="list-style-type: none"> <li>• <b>Disabled, with keep axis: No</b></li> </ul> <p>No rigid links are created. The <b>Keep axis position</b> settings of the connected analysis parts are changed to <b>No</b>.</p>
<b>Default keep axis for secondary members</b>	<p>Defines whether the axis of the secondary analysis parts is free to move.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>No</b></li> <li>• <b>Yes – Ignore out-of-plane eccentricities</b></li> <li>• <b>Yes</b></li> <li>• <b>Yes – Keep end positions also</b></li> </ul>
<b>Analysis model rules...</b>	<p>Click to create rules to define how Tekla Structures handles individual parts in the analysis model, and how parts are connected with each other in the analysis.</p>
<b>Curved beams</b>	<p>Defines whether beams are analyzed as curved beam or as straight segments. Select either:</p> <ul style="list-style-type: none"> <li>• <b>Split into straight segments</b></li> <li>• <b>Use curved member</b></li> </ul> <p>Use the advanced option in <b>Tools --&gt; Options --&gt; Advanced Options... --&gt; Analysis &amp; Design</b> to define how closely straight segments follow the curved beam.</p>
<b>Consider twin profiles</b>	<p>Defines whether twin profiles are considered as one part (<b>Enabled</b>) or as two parts (<b>Disabled</b>) in the analysis.</p>
<b>Member axis location</b>	<p>Defines the location of each analysis part in relation to the corresponding physical part.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Neutral axis</b></li> </ul> <p>The neutral axis is the analysis axis for all parts. The location of the analysis axis changes if the profile of the part changes.</p> <ul style="list-style-type: none"> <li>• <b>Reference axis (eccentricity by neutral axis)</b></li> </ul> <p>The part reference line is the analysis axis for all parts. The location of the neutral axis defines axis eccentricity.</p> <ul style="list-style-type: none"> <li>• <b>Reference axis</b></li> </ul> <p>The part reference line is the analysis axis for all parts.</p>

Option	Description
	<ul style="list-style-type: none"> <li>• <b>Model default</b></li> </ul> <p>The analysis axis of each part is defined individually according to the analysis part properties.</p> <p>To define the axis location of specific parts, use the <b>Position</b> tab in the appropriate analysis part properties dialog box.</p> <p>If you select <b>Neutral axis</b>, Tekla Structures takes the part location and end offsets into account when it creates nodes. If you select either of the <b>Reference axis</b> options, Tekla Structures creates nodes at part reference points.</p>
<b>Member end release method by connection</b>	Defines whether the support conditions of parts ( <b>No</b> ) or connections ( <b>Yes</b> ) are used.
<b>Automatic update</b>	<p>Defines if the analysis model is updated according to the changes in the physical model.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Yes – Physical model changes are considered</b></li> <li>• <b>No – Physical model changes are ignored</b></li> </ul>
<b>Model merging with analysis application</b>	<p>Use when changes occur in the Tekla Structures physical or analysis model that has already been exported to the analysis application.</p> <p>Defines whether the changed analysis model is merged with the previously exported model in the analysis application.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Disabled</b></li> </ul> <p>Models are not merged. Additions made in the analysis application to the previously exported model are lost. A new model is created every time you export the analysis model to the analysis application.</p> <ul style="list-style-type: none"> <li>• <b>Enabled</b></li> </ul> <p>Models are merged. Additions made in the analysis application to the previously exported model are retained when you re-export the analysis model to the analysis application. The model in the analysis application is updated with the changes from Tekla Structures.</p>

#### Analysis tab

Option	Description
<b>Analysis method</b>	<p>Defines whether second order stresses are taken into consideration.</p> <p>The options are:</p>

Option	Description
	<ul style="list-style-type: none"> <li>• <b>1st order</b> Linear analysis method.</li> <li>• <b>P-Delta</b> A simplified second order analysis method. This method gives accurate results when deflections are small.</li> <li>• <b>Non-linear</b> Non-linear analysis method.</li> </ul>
<b>Maximum number of iterations</b>	Tekla Structures repeats second order iteration until it reaches one of these values.
<b>Accuracy of the iteration</b>	
<b>Modal analysis model</b>	Select <b>Yes</b> to create a modal analysis model and to use modal analysis properties instead of static load combinations.

**Job tab** Defines the job information in STAAD.Pro reports.

**Output tab** Defines the contents of the STAAD.Pro analysis results file.

**Seismic tab** Use the **Seismic** tab to define which building code to follow in the seismic analysis and the properties required by the seismic analysis. These properties vary depending on the code you select.

Option	Description
<b>Type</b>	<p>The building code to use to generate seismic loads.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>None</b>: Seismic analysis not run.</li> <li>• <b>UBC 1997</b>: Uniform Building Code 1997</li> <li>• <b>UBC 1994</b>: Uniform Building Code 1994</li> <li>• <b>IBC 2000</b>: International Building Code 2000</li> <li>• <b>IS 1893-2002</b>: Indian Standard. Criteria for Earthquake Resistant Design of Structures</li> <li>• <b>IBC 2003</b>: International Building Code 2003</li> <li>• <b>IBC 2006</b>: International Building Code 2006</li> <li>• <b>IBC 2006 (ZIP)</b>: International Building Code 2006, with an option to add a ZIP code in the properties</li> <li>• <b>IBC 2006 (Longitude/Latitude)</b>: International Building Code 2006, with an option to add longitude and latitude information in the properties</li> <li>• <b>AIJ</b>: Japanese code</li> <li>• <b>Response spectrum</b>: Response spectrum specification</li> </ul>

Option	Description
Seismic properties	Depending on the code you select, you can define various seismic properties.

**Seismic masses** The loads and load groups included in the seismic analysis.

**tab**

**Modal analysis** Use the **Modal analysis** tab to define the properties required by the modal analysis.

**tab**

Option	Description
<b>Count of modes</b>	The number of natural mode shapes in the structure.
<b>Max frequency</b>	The maximum natural resonant frequency of the structure.
<b>Modal analysis masses</b>	The loads and load groups included in the modal analysis.

**Design tabs** Use the **Design** tabs for steel, concrete, and timber to define the codes and methods to use in structural design. The design options available vary depending on the material.

Option	Description
<b>Design code</b>	Design codes for different materials. The design code options available vary depending on the analysis application you use.
<b>Design method</b>	The material-specific principle used to compare stresses and material capacities. The options are: <ul style="list-style-type: none"> <li>• <b>None</b> Tekla Structures only runs a structural analysis and creates data on stresses, forces, and displacements. Available for steel, concrete, and timber.</li> <li>• <b>Check design</b> Tekla Structures checks whether the structures fulfill the criteria in the design code (whether cross sections are adequate). Available for steel and timber.</li> <li>• <b>Calculate required area</b> Tekla Structures defines the required area of reinforcement. Available for concrete.</li> </ul>
<b>Design properties</b>	The design code and method specific design properties of the analysis model that apply to all parts in the analysis model. When you select a design code and method for a material, Tekla Structures lists the design properties in the lower part of the <b>Design</b> tab.



Option	Description
	<p>To change the value of a particular property, click on an entry in the <b>Value</b> column.</p> <p>The units depend on the settings in <b>Tools --&gt; Options --&gt; Options --&gt; Units and decimals</b> .</p> <p>To change the design properties of a specific part, use the <b>Design</b> tab in the appropriate analysis part properties dialog box.</p>

See also [Creating analysis models on page 47](#)

[Defining basic properties for an analysis model on page 47](#)

[Modifying the properties of an analysis model on page 54](#)

## 10.5 Analysis part properties

Use the options in a part's analysis properties dialog box (for example, **Beam Analysis Properties**) to define how Tekla Structures handles the part in the analysis. The settings you have available in the dialog box vary depending on the part type and analysis class. The table below lists all settings regardless of the part type and the analysis class.

**Analysis tab** Use the **Analysis** tab to define the analysis properties of a part.

Option	Description
<b>Class</b>	<p>Defines how the part is handled in the analysis.</p> <p>The selected <b>Class</b> defines which analysis properties are available. For example, plates have different properties from columns.</p>
<b>Filter</b> (Rigid diaphragm properties)	<p>Only available when the <b>Class</b> is <b>Contour plate – Rigid diaphragm</b> or <b>Slab – Rigid diaphragm</b>.</p> <p>Defines the filter used when filtering objects for a rigid diaphragm.</p> <p>Nodes that belong to a part matching the filter will be connected to the rigid diaphragm. For example, you can use <b>column_filter</b> to connect only column nodes to rigid diaphragms.</p>
<b>Built-up section mode</b>	<p>Indicates the role of the part in a built-up section that consists of a main part and one or more sub-parts. In the analysis, sub-parts are merged to the main part.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Automatic</b></li> <li>• <b>Not part of built-up section</b></li> </ul> <p>Disconnects the part from a built-up section.</p>


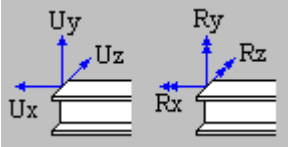
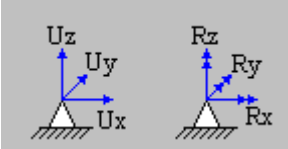
Option	Description
	<ul style="list-style-type: none"> <li>• <b>Main part of built-up section</b> Always use to define the main part of a built-up section.</li> <li>• <b>Sub-part of built-up section</b></li> <li>• <b>Beam sub-part of built-up section</b> Defines that the part is a part of the built-up section when the main part of the built-up section is a beam.</li> <li>• <b>Column sub-part of built-up section</b> Defines that the part is a part of the built-up section when the main part of the built-up section is a column.</li> </ul>
<b>Design group</b>	Defines to which design group the part belongs. Used in optimization.
<b>Automatic update</b>	<p>Defines if the analysis part is updated according to the changes in the physical model.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Yes – Physical model changes are considered</b></li> <li>• <b>No – Physical model changes are ignored</b></li> </ul>

**Start releases  
tab, End releases  
tab**

Use the **Start releases** and **End releases** tabs to define the support conditions and the degrees of freedom for the part ends.

The **Start releases** tab relates to the first part end (yellow handle), the **End releases** tab to the second part end (magenta handle).

Option	Description
<b>Start or End</b>	<p>Defines which of the predefined or user-defined combinations for support conditions is used for part start or end.</p> <p>These are the predefined options:</p> <p>They automatically set the support condition and degrees of freedom.</p> <p>You can modify a predefined combination to suit your needs. If you do that, Tekla Structures indicates it with this option:</p>

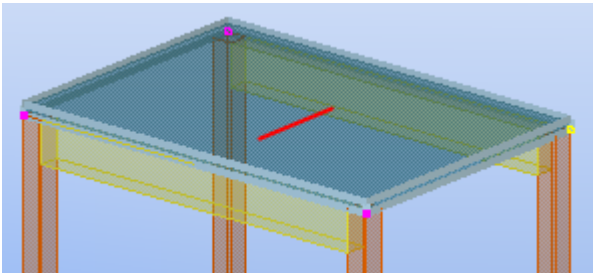
Option	Description
	
<b>Support condition</b>	<p>Defines the support condition.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Connected</b></li> </ul>  <p>Part end is connected to an intermediate analysis node (another part).</p> <p>Indicate degrees of freedom for the node.</p> <ul style="list-style-type: none"> <li>• <b>Supported</b></li> </ul>  <p>Part end is the ultimate support for a superstructure (for example, the foot of a column in a frame).</p> <p>Indicate degrees of freedom for the support.</p>
<b>Rotation</b>	<p>Only available if <b>Support condition</b> is <b>Supported</b>.</p> <p>Defines whether the support is rotated.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Not rotated</b></li> <li>• <b>Rotated</b></li> </ul> <p>If you select <b>Rotated</b>, you can define the rotation around the local x or y axis, or you can set the rotation by the current work plane by clicking <b>Set rotation by current work plane</b>.</p>
<b>Ux</b> <b>Uy</b> <b>Uz</b>	<p>Define the translational degrees of freedom (displacements) in the global x, y, and z directions.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Free</b></li> <li>• <b>Fixed</b></li> </ul>

Option	Description
	<ul style="list-style-type: none"> <li>• <b>Spring</b></li> </ul> <p>If you select <b>Spring</b>, enter the translational spring constant. The units depend on the settings in <b>Tools --&gt; Options --&gt; Options --&gt; Units and decimals</b>.</p>
<b>Rx</b> <b>Ry</b> <b>Rz</b>	<p>Define the rotational degrees of freedom (rotations) in the global x, y, and z directions.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Pinned</b></li> <li>• <b>Fixed</b></li> <li>• <b>Spring</b></li> <li>• <b>Partial release</b></li> </ul> <p>If you select <b>Spring</b>, enter the rotational spring constant. The units depend on the settings in <b>Tools --&gt; Options --&gt; Options --&gt; Units and decimals</b>.</p> <p>Use <b>Partial release</b> to specify if the degree of connectivity is between fixed and pinned. Enter a value between 0 (fixed) and 1 (pinned).</p>

**Composite tab** Use the **Composite** tab with STAAD.Pro to define the analysis properties of the slab in a composite beam.

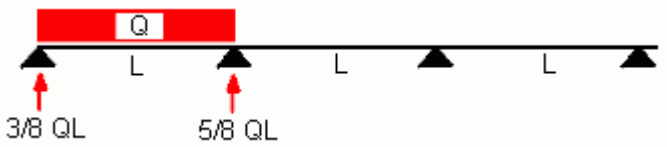
Option	Description
<b>Composite beam</b>	<p>Defines whether the composition is a:</p> <ul style="list-style-type: none"> <li>• <b>Non-composite beam</b></li> <li>• <b>Composite beam</b></li> <li>• <b>Automatic composite beam</b></li> </ul>
<b>Material</b>	Defines the material of the slab.
<b>Thickness</b>	Defines the thickness of the slab.
<b>Effective slab width</b>	<p>Defines if the effective slab width is calculated automatically or based on the values you enter.</p> <p>You can define different values for the left and right side of the beam.</p> <p>Automatic values are calculated in relation to the span length.</p>

**Spanning tab** Use the **Spanning** tab to define the analysis and load distribution properties of a one-way or two-way slab system.

Option	Description
<b>Spanning</b>	<p>Defines in which directions the part carries loads.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Single</b> spanning plate carries loads in the direction of the primary axis. Beams or columns parallel to the spanning direction are not connected to the part, and will not carry loads from the part.</li> <li>• <b>Double</b> spanning part carries loads along the primary and secondary axes. Beams or columns in both directions will carry loads from the part.</li> </ul>
<b>Primary axis direction</b>	<p>Defines the direction of the primary axis in one of the following ways:</p> <ul style="list-style-type: none"> <li>• Enter 1 in the box (<b>x</b>, <b>y</b>, or <b>z</b>) which is parallel to the primary axis direction.</li> <li>• Enter values in multiple boxes to define the components of a direction vector.</li> <li>• Click <b>Parallel to part...</b>, and then select a part in the model that is parallel to the direction.</li> <li>• Click <b>Perpendicular to part...</b>, and then select a part in the model that is perpendicular to the direction.</li> </ul> <p>To check the primary spanning direction of a selected part in a model view, click <b>Show direction on selected members</b>. Tekla Structures indicates the primary direction using a red line.</p> 

**Loading tab** Use the **Loading** tab to include a part as loads in analysis models.

Option	Description
<b>Generate self weight load</b>	<p>Analysis models include the part weight, for example a deck, as a load even if the part is not otherwise included in the analysis models.</p> <p>If the part is included in an analysis model, so is its self-weight. The option <b>No</b> works only with the analysis classes <b>Ignore</b> and <b>Rigid diaphragm</b>.</p>
List boxes for additional loads	Enter slab live load or additional self-weight (screed, services) using three additional loads with a load group name and

Option	Description
	magnitude. The directions of these loads follow the direction of the load group to which they belong.
<b>Part names</b>	Use this filter to ensure that the area load from the slab is transferred to the correct parts, for example, beams supporting the slab. Typically, you would enter the beam name as the filter value.
<b>Use continuous structure load distribution</b>	Use to assign most of the load to the middle supports on continuous structures. 

**Design tab** Use the **Design** tab in the analysis part properties dialog box to view and modify the design properties of an individual part in an analysis model. Design properties are properties which can vary, according to the design code and the material of the part (for example, design settings, factors, and limits).

**Position tab** Use the **Position** tab to define the location and offsets of an analysis part.

Option	Description
<b>Axis</b>	<p>Defines the location of the analysis part in relation to the corresponding physical part.</p> <p>The location of the analysis axis of a part defines where the part meets with other parts and where Tekla Structures creates nodes in analysis models.</p> <p>The options are:</p>

Option	Description
	<p> Neutral axis  Reference axis (eccentricity by neutral axis)  Reference axis  Top left  Top center  Top right  Middle left  Middle center  Middle right  Bottom left  Bottom center  Bottom right  Top plane  Middle plane  Bottom plane  Left plane  Right plane  Middle plane (of left/right) </p> <p>If you select <b>Neutral axis</b>, Tekla Structures takes the part location and end offsets into account when it creates nodes. If you select either of the <b>Reference axis</b> options, Tekla Structures creates nodes at part reference points.</p>
<b>Keep axis position</b>	<p>Defines whether the axis position is kept or changed according to changes in the physical model.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>No</b> The axis is free to move when snapping end positions to nearby objects. Use this option for secondary members.</li> <li>• <b>Partial – keep in major direction</b> The axis is free to move partially, but the member is not moved in the major (stronger) direction of the part profile.</li> <li>• <b>Partial – keep in minor direction</b> The axis is free to move partially, but the member is not moved in the minor (weaker) direction of the part profile.</li> <li>• <b>Yes</b> The axis is not moved, but the end positions can move along the axis (thus extending or shortening the member).</li> <li>• <b>Yes – Keep end positions also</b> The axis and the end positions of the member are not changed.</li> </ul>

Option	Description
<b>Connectivity</b>	<p>Defines whether the member snaps or connects with rigid links to other members.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Automatic</b> The member snaps or connects with rigid links to other members.</li> <li>• <b>Manual</b> The member does not snap or connect with rigid links to other members. Automatic connectivity to other members is created only if the member position matches the other member exactly.</li> </ul>
<b>Axis modifier X</b> <b>Axis modifier Y</b> <b>Axis modifier Z</b>	<p>Define whether the member location is bound to global coordinates, grid line, or neither.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>None</b> The member location is not bound.</li> <li>• <b>Fixed coordinate</b> The member location is bound to the coordinate you enter in the <b>X</b>, <b>Y</b>, or <b>Z</b> box.</li> <li>• <b>Nearest grid</b> The member is bound to the nearest grid line (the snap zone is 1000 mm).</li> </ul>
<b>Offset</b>	Use to move the analysis part in the global x, y, and z directions.
<b>Longitudinal offset mode</b>	<p>Defines whether the longitudinal end offsets <b>Dx</b> of the physical part are used from the <b>Position</b> tab of the part properties dialog box.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Offsets are not considered</b></li> <li>• <b>Only extensions are considered</b></li> <li>• <b>Offsets are always considered</b></li> </ul>

**Bar attributes tab** Use the **Bar attributes** tab in a frame object's (beam, column, or brace) analysis properties dialog box to define the properties of its analysis bars.

You can use the options on this tab when the analysis class of the analysis part is **Beam**, **Column**, or **Secondary**.



Option	Description
<b>Start offset</b> <b>End offset</b>	<p>Calculate offsets to account for longitudinal eccentricity at the member end (resulting in a bending moment).</p> <p>These offsets have no effect on the topology on the analysis model. The offset value is only passed as a member attribute to the analysis.</p>
<b>Replacement profile name</b>	<p>Select a profile from the Profile Catalog. You can use different analysis profiles at the start and end of parts if the analysis application you use supports it.</p> <p>To use different profiles at part ends, enter two profiles separated by a pipe character, for example: HEA120   HEA140</p> <p>If the part is a built-up section in an analysis model, the name of the built-up section can be entered here. Any name can be entered, but if the name matches an existing catalog profile name, the physical properties of the section will be the same as the catalog profile properties.</p>
<b>Curved beam mode</b>	<p>Defines whether a beam is analyzed as a curved beam or as straight segments.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Use model default</b></li> <li>• <b>Use curved member</b></li> <li>• <b>Split into straight segments</b></li> </ul> <p>If you select <b>Use model default</b>, Tekla Structures uses the option selected from the <b>Curved beams</b> list in the <b>Analysis Model Properties</b> dialog box.</p> <p>Use the advanced option in <b>Tools --&gt; Options --&gt; Advanced Options... --&gt; Analysis &amp; Design</b> to define how closely straight segments follow the curved beam.</p>
<b>No. of split nodes</b>	<p>Use to create additional nodes or analyze a beam as straight segments, for example, a curved beam.</p> <p>Enter the number of nodes.</p>
<b>Split distances</b>	<p>To define additional nodes in the member, enter distances from the part starting point to the node.</p> <p>Enter distances, separated by spaces, for example:</p> <p>1000 1500 3000</p>
<b>Bar start number</b>	Defines the start number for analysis bars.
<b>Member start number</b>	Defines the start number for analysis members.

**Area attributes tab** Use the **Area attributes** tab in a plate's (contour plate, concrete slab, or concrete panel) analysis properties dialog box to define the properties of its analysis elements.

You can use the options on this tab when the analysis class of the analysis part is **Contour plate**, **Slab**, or **Wall**.

Option	Description
<b>Element type</b>	The shape of the elements.
<b>Rotation of local XY</b>	Defines the rotation of the local xy plane.
<b>Element size</b>	<p><b>x</b> and <b>y</b>: The approximate dimensions of the elements, in the local x and y direction of the plate. For triangular elements, the approximate dimensions of the bounding box around each element.</p> <p><b>Holes</b>: The approximate size of the elements around openings.</p>
<b>Area start number</b>	Defines the start number for the plate.
<b>Simple area (ignore cuts etc)</b>	Select <b>Yes</b> to create a simpler analysis model of the plate, where cuts and openings are not considered.
<b>Smallest hole size to consider</b>	<p>Use to ignore small openings in the plate in the analysis.</p> <p>Enter the size of the bounding box around the opening.</p>
<b>Supported</b>	<p>Use to define supports for a contour plate, concrete slab, or concrete panel.</p> <p>You can create supports for the bottom edge of a panel, for all edge nodes of a slab or plate, or for all nodes of a beam. For panels, the bottom edge can be inclined.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>No</b> No supports are created.</li> <li>• <b>Simply (translations)</b> Only translations are fixed.</li> <li>• <b>Fully</b> Both translations and rotations are fixed.</li> </ul>

**See also** [Analysis class options and colors on page 122](#)  
[Analysis axis options on page 125](#)  
[Modifying the properties of an analysis part on page 69](#)  
[Defining support conditions on page 70](#)  
[Defining design properties for analysis parts on page 74](#)  
[Defining the location of analysis parts on page 77](#)

## Analysis class options and colors

Use the options in the **Class** list on the **Analysis** tab in a part's analysis properties dialog box to define how Tekla Structures handles the part in the analysis.

The option you select in the **Class** list determines which tabs are available in the analysis part properties dialog box.

You can show the analysis class of parts using different colors in the analysis model.

The analysis application you use may not support all of the following options.

Option	Description	Color
<b>Beam</b>	Line object of two nodes. Part can take any load, including temperature.	Dark red
<b>Beam – Truss</b>	Part can only take axial forces, not bending or torsion moments, or shear forces. Usually used for braces.	Green
<b>Beam – Truss– Compression only</b>	Part can only take compressive axial forces, not moments or shear forces. If this part goes into tension, it is ignored in the analysis.	Yellow
<b>Beam – Truss– Tension only</b>	Part can only take tensile axial forces, not moments or shear forces. If this part goes into compression, it is ignored in the analysis.	Pink
<b>Beam – Ignore</b>	Part ignored in the analysis. Self-weight load is taken into account if you have set <b>Generate self weight load to Yes</b> on the <b>Loading</b> tab.	Part not shown in the model
<b>Column</b>	Vertical line object of two nodes. Modeled from bottom to top. Part can take any load, including temperature.	Dark red
<b>Column – Truss</b>	Part can only take axial forces, not bending or torsion moments, or shear forces. Usually used for braces.	Green
<b>Column – Truss– Compression only</b>	Part can only take compressive axial forces, not moments or shear forces. If this part goes into tension, it is ignored in the analysis.	Yellow
<b>Column – Truss– Tension only</b>	Part can only take tensile axial forces, not moments or shear forces. If this part goes into compression, it is ignored in the analysis.	Pink
<b>Column – Ignore</b>	Part ignored in the analysis. Self-weight load is taken into account if you have set <b>Generate self weight load to Yes</b> on the <b>Loading</b> tab.	Part not shown in the model
<b>Secondary</b>	Line object of two nodes. Part can take any load, including temperature. For parts whose analysis class is <b>Secondary, Keep axis position</b> is off by default. Secondary parts snap to nearest nodes instead of part end nodes.	Dark red

Option	Description	Color
<b>Secondary – Truss</b>	Part can only take axial forces, not bending or torsion moments, or shear forces. Usually used for braces.	Green
<b>Secondary – Truss-Compression only</b>	Part can only take compressive axial forces, not moments or shear forces. If this part goes into tension, it is ignored in the analysis.	Yellow
<b>Secondary – Truss-Tension only</b>	Part can only take tensile axial forces, not moments or shear forces. If this part goes into compression, it is ignored in the analysis.	Pink
<b>Secondary – Ignore</b>	Part ignored in the analysis. Self-weight load is taken into account if you have set <b>Generate self weight load to Yes</b> on the <b>Loading</b> tab.	Part not shown in the model
<b>Wall – Shell</b>	Part can take any load, except temperature.	Aqua
<b>Wall – Plate</b>	Same as <b>Wall – Shell</b> but plate elements are used in the analysis application.	Aqua
<b>Slab – Shell</b>	Part can take any load, except temperature.	Aqua
<b>Slab – Plate</b>	Same as <b>Slab – Shell</b> but plate, membrane, or mat foundation elements are used in the analysis application.	Aqua
<b>Slab – Membrane</b>		
<b>Slab – Mat foundation</b>		
<b>Slab – Rigid diaphragm</b>	Only applies to parts parallel to a global xy plane. <b>Filter:</b> Nodes that belong to a part matching the filter will be connected with rigid links which together affect displacement. For example, you can use <b>column_filter</b> to connect only column nodes to rigid diaphragms.	Lilac
<b>Slab – Ignore</b>	Part is ignored in the analysis. Self-weight load is taken into account if you have set <b>Generate self weight load to Yes</b> on the <b>Loading</b> tab.	Part not shown in the model
<b>Contour plate – Shell</b>	Part can take any load, except temperature.	Aqua
<b>Contour plate – Plate</b>	Same as <b>Contour plate – Shell</b> but plate or membrane elements are used in the analysis application.	Aqua
<b>Contour plate – Membrane</b>		Aqua
<b>Contour plate – Rigid diaphragm</b>	Only applies to parts parallel to a global xy plane. <b>Filter:</b> Nodes that belong to a part matching the filter will be connected with rigid links which together affect displacement. For example, you can use <b>column_filter</b> to connect only column nodes to rigid diaphragms.	Lilac

Option	Description	Color
<b>Contour plate – Ignore</b>	Part ignored in the analysis. Self-weight load is taken into account if you have set <b>Generate self weight load</b> to <b>Yes</b> on the <b>Loading</b> tab.	Part not shown in the model

See also [Showing analysis class in model views on page 94](#)

[Analysis part properties on page 113](#)

## Analysis axis options

Use the options in the **Axis** list on the **Position** tab in a part's analysis properties dialog box to define the location of the analysis part in relation to the physical part.

Option	Description	Use for
<b>Neutral axis</b>	The neutral axis is the analysis axis for this part. The location of the analysis axis changes if the profile of the part changes.	
<b>Reference axis (eccentricity by neutral axis)</b>	The part reference line is the analysis axis for this part. The location of the neutral axis defines the axis eccentricity.	
<b>Reference axis</b>	The part reference line is the analysis axis for this part.	
<b>Top left</b>	The analysis axis is located in the top left corner of the part.	Beam objects (beams, columns, braces)
<b>Top center</b>	The analysis axis is located in the top center point of the part cross section.	Beam objects
<b>Top right</b>	The analysis axis is located in the top right corner of the part.	Beam objects
<b>Middle left</b>	The analysis axis is located in the middle of the left side of the part.	Beam objects
<b>Middle center</b>	The analysis axis is located in the center point of the part cross section.	Beam objects
<b>Middle right</b>	The analysis axis is located in the middle of the right side of the part.	Beam objects
<b>Bottom left</b>	The analysis axis is located in the bottom left corner of the part.	Beam objects
<b>Bottom center</b>	The analysis axis is located in the bottom center point of the part cross section.	Beam objects
<b>Bottom right</b>	The analysis axis is located in the bottom right corner of the part.	Beam objects

Option	Description	Use for
<b>Top plane</b>	The analysis axis is bound to the top plane.	Plate objects (plates, slabs, panels)
<b>Middle plane</b>	The analysis axis is bound to the middle plane.	Plate objects
<b>Bottom plane</b>	The analysis axis is bound to the bottom plane.	Plate objects
<b>Left plane</b>	The analysis axis is bound to the left plane.	Plate objects
<b>Right plane</b>	The analysis axis is bound to the right plane.	Plate objects
<b>Middle plane (of left/right)</b>	The analysis axis is bound to the middle plane of left/right.	Plate objects

Tekla Structures uses the options above for each part when you select **Model default** from the **Member axis location** list in the **Analysis Model Properties** dialog box.

If you select **Neutral axis**, Tekla Structures takes the part location and end offsets into account when it creates nodes. If you select either of the **Reference axis** options, Tekla Structures creates nodes at part reference points.

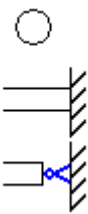

See also [Analysis part properties on page 113](#)  
[Analysis model properties on page 107](#)

## 10.6 Analysis node properties

Use the **Analysis node properties** dialog box to view and modify the properties of a node in an analysis model.

To access the dialog box, double-click an analysis node.

Option	Description
<b>Supports</b>	<p>Defines which support conditions are used for the node.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Get supports from part(s)</b> The support conditions of a corresponding part end are used for the node.</li> <li>• <b>User-defined node supports</b> You can define the support conditions for the node.</li> </ul> <p>If you select <b>User-defined node supports</b>, you can select one of the following options:</p>

Option	Description
	 <p>These options automatically set the degrees of freedom for the node.</p> <p>You can modify a predefined combination to suit your needs. If you do that, Tekla Structures indicates it with this option:</p> 
<b>Rotation</b>	<p>If you selected <b>User-defined node supports</b>, you can define the rotation of the node.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Not rotated</b></li> <li>• <b>Rotated</b></li> </ul> <p>If you select <b>Rotated</b>, you can define the rotation, or you can set the rotation by the current work plane by clicking <b>Set rotation by current work plane</b>.</p>
<b>Ux</b> <b>Uy</b> <b>Uz</b> <b>Rx</b> <b>Ry</b> <b>Rz</b>	<p>Define the translational (U) and rotational (R) degrees of freedom (displacements and rotations) of the node in the global x, y, and z directions.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Free</b></li> <li>• <b>Fixed</b></li> <li>• <b>Spring</b></li> </ul> <p>If you select <b>Spring</b>, enter the spring constant. The units depend on the settings in <b>Tools --&gt; Options --&gt; Options --&gt; Units and decimals</b>.</p>

See also [Creating an analysis node on page 63](#)

[Merging analysis nodes on page 65](#)

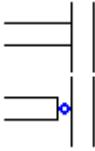

[Analysis model objects on page 9](#)

[Analysis node colors on page 64](#)

## 10.7 Analysis rigid link properties

Use the **Analysis rigid link properties** dialog box to view and modify the end conditions of a rigid link.

To access the dialog box, double-click a rigid link.

Option	Description
<b>Releases</b>	<p>Defines which releases are used for a rigid link start or end.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Automatic releases (by rules)</b></li> <li>• <b>User-defined releases</b></li> </ul>
<b>Start or End</b>	<p>Defines which of the predefined or user-defined combinations for releases is used for a rigid link start or end.</p> <p>These are the predefined options:</p>  <p>These options automatically set the degrees of freedom.</p> <p>You can modify a predefined combination to suit your needs. If you do that, Tekla Structures indicates it with this option:</p> 
<b>Ux</b> <b>Uy</b> <b>Uz</b>	<p>Define the translational degrees of freedom (displacements) in the global x, y, and z directions.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Free</b></li> <li>• <b>Fixed</b></li> <li>• <b>Spring</b></li> </ul> <p>If you select <b>Spring</b>, enter the translational spring constant. The units depend on the settings in <b>Tools --&gt; Options --&gt; Options --&gt; Units and decimals</b>.</p>
<b>Rx</b> <b>Ry</b> <b>Rz</b>	<p>Define the rotational degrees of freedom (rotations) in the global x, y, and z directions.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Pinned</b></li> <li>• <b>Fixed</b></li> <li>• <b>Spring</b></li> </ul>



Option	Description
	<ul style="list-style-type: none"> <li>• <b>Partial release</b></li> </ul> <p>If you select <b>Spring</b>, enter the rotational spring constant. The units depend on the settings in <b>Tools --&gt; Options --&gt; Options --&gt; Units and decimals</b>.</p> <p>Use <b>Partial release</b> to specify if the degree of connectivity is between fixed and pinned. Enter a value between 0 (fixed) and 1 (pinned).</p>
<b>Local Y direction</b>	<p>Defines the local y direction of the rigid link. The options are the global x, y, and z directions.</p> <p>The local x direction is always the direction of the rigid link.</p>

See also [Creating a rigid link on page 65](#)  
[Analysis model objects on page 9](#)

## 10.8 Analysis bar position properties

Use the **Analysis Bar Position Properties** dialog box to view and modify the position of an analysis bar.

To access the dialog box, select an analysis bar, and then double-click a handle at an end of the analysis bar.

Option	Description
<b>Offset mode</b>	Defines whether the automatic ( <b>Automatic offset</b> ) or user-defined ( <b>Manual offset</b> ) offset values are used for the analysis bar end.
<b>Offset</b>	Defines the offset values in the global x, y, and z directions.

See also [Defining the location of analysis parts on page 77](#)

## 10.9 Analysis area position properties

Use the **Analysis Area Position Properties** dialog box to view and modify the position of an analysis area.

To access the dialog box, select an analysis area, and then double-click a handle at an analysis area corner.

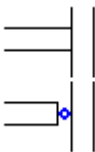

Option	Description
<b>Offset mode</b>	Defines whether the automatic ( <b>Automatic offset</b> ) or user-defined ( <b>Manual offset</b> ) offset values are used for the analysis bar end.
<b>Offset</b>	Defines the offset values in the global x, y, and z directions.

See also [Defining the location of analysis parts on page 77](#)

## 10.10 Analysis area edge properties

Use the **Analysis Area Edge Properties** dialog box to view and modify the position and connectivity of an analysis area edge.

To access the dialog box, select an analysis area, and then double-click a handle at the mid-point of an analysis area edge.

Option	Description
<b>Offset mode</b>	Defines whether the automatic ( <b>Automatic offset</b> ) or user-defined ( <b>Manual offset</b> ) offset values are used for the analysis bar end.
<b>Offset</b>	Defines the offset values in the global x, y, and z directions.
<b>Releases</b>	<p>Defines which of the predefined or user-defined combinations for releases is used for the analysis area edge.</p> <p>These are the predefined options:</p>  <p>These options automatically set the degrees of freedom.</p> <p>You can modify a predefined combination to suit your needs. If you do that, Tekla Structures indicates it with this option:</p> 

Option	Description
<b>Ux</b> <b>Uy</b> <b>Uz</b>	<p>Define the translational degrees of freedom (displacements) in the global x, y, and z directions.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Free</b></li> <li>• <b>Fixed</b></li> <li>• <b>Spring</b></li> </ul> <p>If you select <b>Spring</b>, enter the translational spring constant. The units depend on the settings in <b>Tools --&gt; Options --&gt; Options --&gt; Units and decimals</b>.</p>
<b>Rx</b> <b>Ry</b> <b>Rz</b>	<p>Define the rotational degrees of freedom (rotations) of a member end in the global x, y, and z directions.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li>• <b>Pinned</b></li> <li>• <b>Fixed</b></li> <li>• <b>Spring</b></li> <li>• <b>Partial release</b></li> </ul> <p>If you select <b>Spring</b>, enter the rotational spring constant. The units depend on the settings in <b>Tools --&gt; Options --&gt; Options --&gt; Units and decimals</b>.</p> <p>Use <b>Partial release</b> to specify if the degree of connectivity is between fixed and pinned. Enter a value between 0 (fixed) and 1 (pinned).</p>

See also [Defining the location of analysis parts on page 77](#)

# 1 1 Disclaimer

© 2015 Tekla Corporation and its licensors. All rights reserved.

This Software Manual has been developed for use with the referenced Software. Use of the Software, and use of this Software Manual are governed by a License Agreement. Among other provisions, the License Agreement sets certain warranties for the Software and this Manual, disclaims other warranties, limits recoverable damages, defines permitted uses of the Software, and determines whether you are an authorized user of the Software. All information set forth in this manual is provided with the warranty set forth in the License Agreement. Please refer to the License Agreement for important obligations and applicable limitations and restrictions on your rights. Tekla does not guarantee that the text is free of technical inaccuracies or typographical errors. Tekla reserves the right to make changes and additions to this manual due to changes in the software or otherwise.

In addition, this Software Manual is protected by copyright law and by international treaties. Unauthorized reproduction, display, modification, or distribution of this Manual, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the full extent permitted by law.

Tekla, Tekla Structures, Tekla BIMsight, BIMsight, Tedds, Solve, Fastrak and Orion are either registered trademarks or trademarks of Tekla Corporation in the European Union, the United States, and/or other countries. More about Tekla trademarks: <http://www.tekla.com/tekla-trademarks>. Trimble is a registered trademark or trademark of Trimble Navigation Limited in the European Union, in the United States and/or other countries. More about Trimble trademarks: <http://www.trimble.com/trademarks.aspx>. Other product and company names mentioned in this Manual are or may be trademarks of their respective owners. By referring to a third-party product or brand, Tekla does not intend to suggest an affiliation with or endorsement by such third party and disclaims any such affiliation or endorsement, except where otherwise expressly stated.

Portions of this software:

D-Cubed 2D DCM © 2010 Siemens Industry Software Limited. All rights reserved.

EPM toolkit © 1995-2004 EPM Technology a.s., Oslo, Norway. All rights reserved.

Open CASCADE Technology © 2001-2014 Open CASCADE SA. All rights reserved.

FLY SDK - CAD SDK © 2012 VisualIntegrity™. All rights reserved.

Teigha © 2003-2014 Open Design Alliance. All rights reserved.

PolyBoolean C++ Library © 2001-2012 Complex A5 Co. Ltd. All rights reserved.

FlexNet Copyright © 2014 Flexera Software LLC. All Rights Reserved.

This product contains proprietary and confidential technology, information and creative works owned by Flexera Software LLC and its licensors, if any. Any use, copying, publication, distribution, display, modification, or transmission of such technology in whole or in part in any form or by any means without the prior express written permission of Flexera Software LLC is strictly prohibited. Except where expressly provided by Flexera Software LLC in writing, possession of this technology shall not be construed to confer any license or rights under any Flexera Software LLC intellectual property rights, whether by estoppel, implication, or otherwise.

To see the third party licenses, go to Tekla Structures, click **Help** --> **About** and click the **Third party licenses** button.

The elements of the software described in this Manual are protected by several patents and possibly pending patent applications in the European Union and/or other countries. For more information go to page <http://www.tekla.com/tekla-patents>.



# Index

## A

adding		
analysis model rules.....	58	
objects to analysis model.....	62	
analysis & design.....	7	
analysis and design.....	7	
settings.....	97	
workflow.....	13	
analysis and design applications.....	12	
analysis and design models		
working with.....	88	
analysis and design systems.....	12	
analysis applications.....	12	
linking to Tekla Structures.....	12	
merging models.....	90	
selecting for analysis model.....	50	
analysis areas		
edge properties.....	130	
position properties.....	129	
analysis axis		
of analysis models.....	55	
of parts.....	77,78	
options for parts.....	125	
analysis bars.....	9	
position properties.....	129	
showing numbers.....	94	
analysis class.....	94,122	
analysis members.....	9	
showing numbers.....	94	
analysis model rules.....	58	
adding.....	58	
arranging.....	59	
creating.....	58	
deleting.....	61	
organizing.....	59	
saving.....	62	
testing.....	60	
analysis models.....	7	
adding objects.....	62	
adding rules.....	58	
arranging rules.....	59	
axis settings.....	55	
changing creation method.....	54	
checking objects.....	53	
copying.....	66	
creating.....	47,50,51	
creating by copying.....	66	
creating modal models.....	52	
creating rules.....	58	
creation method.....	49	
defining properties.....	47	
deleting.....	67	
exporting.....	89	
filtering objects.....	49	
including objects.....	48	
merging.....	90	
modifying.....	53	
modifying properties.....	54	
objects.....	9	
organizing rules.....	59	
properties.....	107	
removing objects.....	63	
resetting editing.....	79	
resetting model merging.....	92	
viewing results.....	95	
warnings.....	88	
working with.....	88	
analysis nodes.....	9	
colors.....	64	
creating.....	63	
merging.....	65	
properties.....	126	
showing numbers.....	94	
analysis parts.....	9	
axis location.....	77,78	
copying.....	80	
defining properties.....	68,69	
deleting.....	80	
modifying.....	68	
modifying properties.....	68,69	
offsets.....	79	

position.....	77
properties.....	113
resetting editing.....	79
viewing properties.....	69
analysis results	
saving.....	92
saving as user-defined attributes.....	93
viewing.....	93
analysis type.....	122
applying loads to parts.....	31
area loads.....	25
properties.....	100
attaching	
loads to parts.....	31
axis settings	
defining for analysis models.....	55
axis	
of analysis parts.....	78

## B

bounding box.....	34
buckling length.....	75
Kmode options.....	76

## C

check design.....	75
checking	
analysis models.....	53
load groups.....	41,42,43
loads.....	41,42,43
colors	
by analysis type.....	94,122
by analysis utility check.....	95
of analysis nodes.....	64
combining	
loads.....	82
compatibility of load groups.....	20
copying	
analysis models.....	66
analysis parts.....	80
load combinations.....	85,86
creating	
analysis model rules.....	58
analysis models.....	47,50,51
analysis models by copying.....	66

analysis nodes.....	63
area loads.....	25
line loads.....	25
load combinations.....	83,84
load groups.....	15,19
loads.....	15,21
modal analysis models.....	52
point loads.....	24
rigid links.....	65
strain.....	27
temperature loads.....	27
uniform loads.....	26
wind loads.....	28
creation method	
changing for analysis model.....	54
of analysis model.....	49

## D

defining	
analysis model properties.....	47
analysis part properties.....	68,69
design properties of analysis models.....	57
design properties of analysis parts.....	74
load groups.....	19
modal masses for analysis models.....	56
seismic loads for analysis models.....	56
deleting	
analysis model rules.....	61
analysis models.....	67
analysis parts.....	80
load combinations.....	87
load groups.....	21
design properties	
defining for analysis models.....	57
defining for analysis parts.....	74
design	
omitting parts.....	75
direct links.....	12
distances	
of loads.....	34
distributing loads.....	31

## E

effective buckling length.....	75
Kmode options.....	76



examples	
creating wind loads.....	28
exporting	
analysis models.....	89
load groups.....	44

## F

filtering	
analysis model objects.....	49
filters	
in analysis models.....	49

## G

grouping	
loads.....	15,18

## H

handles	
of loads.....	39

## I

importing	
load groups.....	45

## K

Kmode options.....	76
--------------------	----

## L

line loads.....	25
properties.....	99
linking	
Tekla Structures with analysis applications.....	12
load attachment.....	31
load combination process.....	82
using non-standard factors.....	17
load combination	
factors.....	105
properties.....	105

settings.....	105
types.....	106
load combinations.....	82
copying.....	85,86
creating.....	83,84
deleting.....	87
modifying.....	85
saving for later use.....	86
load forms.....	24
load groups.....	18
checking.....	41,42,43
compatibility.....	20
creating.....	15,19
defining.....	19
deleting.....	21
exporting.....	44
importing.....	45
modifying.....	19
moving loads to another group.....	44
properties.....	97
setting current.....	19
working with.....	40
load modeling code.....	17
options.....	105
load modeling	
non-standard combination factors.....	17
load models.....	7
load panel.....	35,103
load types.....	15
load-bearing parts.....	31
by name.....	32
by selection filter.....	33
loaded area.....	34
loaded length.....	34
loads	
applying.....	31
attaching.....	31
bounding box.....	34
changing length or area.....	34
changing load group.....	44
checking.....	41,42,43
combining.....	82
creating.....	15,21
defining properties.....	22
distribution.....	31
forms.....	24
grouping.....	15,18
load panel properties.....	103

magnitude.....	23
modal.....	56
modifying.....	31,34,39
modifying distribution.....	35
modifying location or layout.....	37
moving to another load group.....	44
properties.....	98
scaling in model views.....	40
seismic.....	56
types.....	15
working with.....	40

## M

member axis location.....	55,125
merging	
analysis models.....	90
analysis nodes.....	65
models with analysis applications.....	90
models with SAP2000.....	90,91
resetting.....	92
modal analysis.....	56
creating analysis models.....	52
modal masses.....	56
model merging.....	90
resetting.....	92
modifying	
analysis model properties.....	54
analysis models.....	53
analysis part properties.....	68,69
analysis parts.....	68
load combinations.....	85
load groups.....	19
load location or layout.....	37
loads.....	31
moving	
load ends or corners.....	39

## N

nodes, see analysis nodes.....	63
--------------------------------	----

## O

offsets	
of analysis parts.....	79

## P

partial safety factors.....	105
parts	
analysis properties.....	113
physical models.....	7
point loads.....	24
properties.....	99
position	
of analysis parts.....	77
properties	
analysis models.....	107
analysis parts.....	113
load combination.....	105
loads.....	98

## R

reduction factors.....	105
removing	
objects from analysis model.....	63
reports	
of loads.....	43
resetting	
editing of analysis parts.....	79
rigid diaphragms.....	9
rigid links.....	9
creating.....	65
properties.....	127
rules	
in analysis models.....	58

## S

SAP2000	
merging analysis models.....	90
merging models.....	91
saving	
analysis model rules.....	62
analysis results.....	92
analysis results as user-defined attributes.....	93
load combinations.....	86
scaling	
loads in model views.....	40
seismic analysis.....	56
seismic loads.....	56
seismic masses.....	56

setting	
current load group.....	19
load modeling code.....	17
settings	
analysis and design properties.....	97
analysis area edge properties.....	130
analysis area position properties.....	129
analysis bar position properties.....	129
analysis model properties.....	107
analysis node properties.....	126
analysis part properties.....	113
area load properties.....	100
line load properties.....	99
load combination properties.....	105
load group properties.....	97
load panel properties.....	103
load properties.....	98
point load properties.....	99
rigid link properties.....	127
temperature load properties.....	101
uniform load properties.....	101
wind load properties.....	102
showing	
analysis bar numbers.....	94
analysis member numbers.....	94
analysis node numbers.....	94
strain.....	27
support conditions.....	70
defining for part ends.....	71
defining for plates.....	72
symbols.....	72

## T

temperature loads.....	27
properties.....	101
testing	
analysis model rules.....	60

## U

uniform loads.....	26
properties.....	101
utilization ratio.....	95

## V

viewing	
analysis results.....	93

## W

warnings	
about analysis models.....	88
Wind Load Generator (28).....	28
properties.....	102
wind loads	
creating.....	28
examples.....	28
properties.....	102
workflow	
in analysis and design.....	13

