Tekla Structures Basic Training

Tekla Structures 10.0

March 30, 2004

Contents

17 Analysis and Design

In this lesson   This lesson describes the basic functions of Analysis and Design. We will learn how to create and manage loads and load groups, how to make load combinations and create analysis model, and view the analysis results in STAAD*pro*2003 program and in Tekla Structures. We will also learn how to optimize the steel profiles using the analysis results.

17.1 Analysis Model

In this exercise we will build an analysis model and run the analysis.

Analysis model build-up can be divided into 5 steps:

1. Create load groups and define loads. See more in Tekla Structures help: Analysis > Loads > Grouping loads
2. Apply point, line and area loads to model. See more in Tekla Structures help: Analysis > Loads > Distributing loads
3. Define Restraints for each member in the model by creating supports and defining member connectivity to intermediate nodes. See more in Tekla Structures help: Modeling > Parts > Member end conditions
4. Select members and create a new analysis model. See more in Tekla Structures help: Analysis > Analysis and design > Following through structural analysis
5. Define Load combinations. See more in Tekla Structures help: Analysis > Analysis and design > Load combination

After these steps, you can run the analysis.

Create Load Groups and Define Loads

A load group is a set of loads that are treated alike in the load combination. You can group loads that are caused by the same action. This implies that the loads in a group have the common partial safety factors and the same action direction. They also act always at the same time and all together.

Every load has to belong to a load group. One load cannot be attached to several load groups, but a load group may have only one load on it. Loads in a load group can be of any load type (point, line, area, etc.), and you can include as many loads as you want in a load group.

Define load groups

To define load groups, select **Loads > Load groups** on the **Properties** pull-down menu. In the appearing **Load group properties** dialog box you can add new load groups, modify and delete existing load groups, name load groups, and define the load group type and direction.

1. Push **Add…** button in the **Load group properties** dialog box to create a new load group…
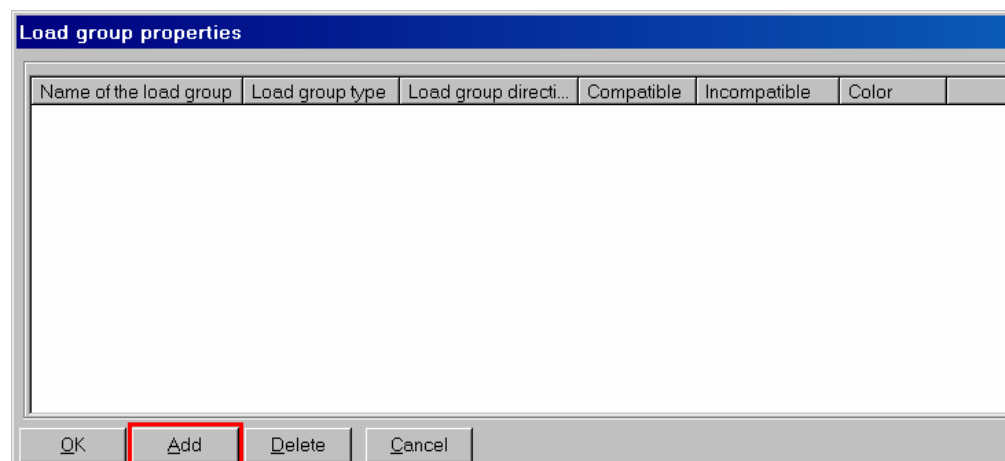
…a new load group named "Group name1" appears.

2. Change load group name to **Live1.** Load group type can remain as **Live load / housing** and direction can remain as **Z**.

⚠ Every load group must have a **unique name**, which can be used in filtering and for expressing

3. Push **Add…** to create another new load group.

4. Change load group name to **Live2.**

5. Change load group type to **Live load / housing,** direction can remain as **Z**.

⚠ The **type** of a load group is the type of the action that causes the loads in the load group. Actions, and thereby also load group types, are design code specific. The **direction** of a load group is the global direction of the action that causes the loads.

6. Change load color by picking new color from pull-down list.

7. Push **Add…** to create another new load group.

8. Change load group name to **Wind left.**

9. Change load group type to **Wind load**

10. Change load direction to **X.**

11. Change load color by picking new color from pull-down list.

12. Push **Add…** to create another new load group.

13. Change load group name to **Wind right.**

14. Change load group type to **Wind load**

15. Change load direction to **X.**

16. Change load color by picking new color from pull-down list.

Load group properties

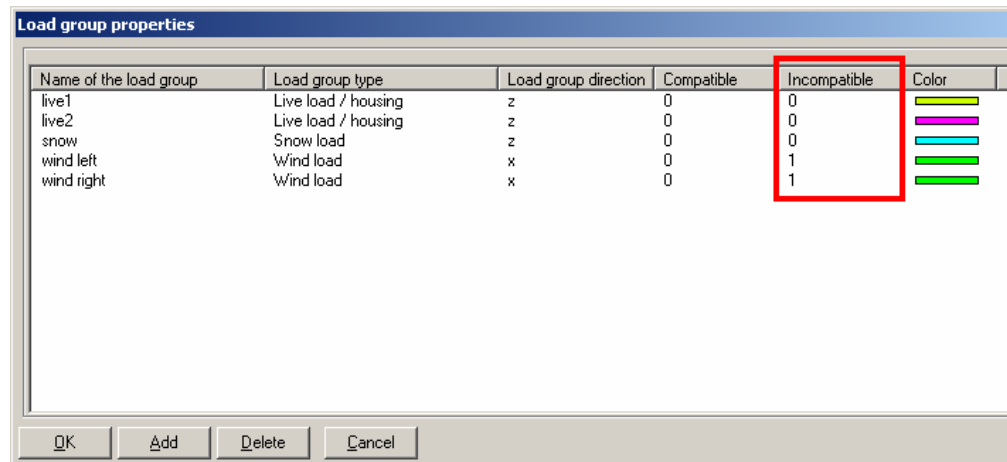| Name of the load group	Load group type	Load group direction	Compatible	Incompatible	Color
live1	Live load / housing	z	0	0	▬
live2	Live load / housing	z	0	0	▬
snow	Snow load	z	0	0	▬
wind left	Wind load	x	0	1	▬
wind right	Wind load	x	0	1	▬

| OK | Add | Delete | Cancel |

17. Push **Add…** to create another new load group.

18. Change load group name to **Snow.**

19. Change load group type to **Snow load**

20. Change load direction to **X.**

21. Change load color by picking new color from pull-down list.

Modify load groups

You have now defined load groups and you have two different winds, which cannot affect at the same time. Therefore you need to modify them and set **Incompatible** number for wind loads. Use same incompatible number for both to set them incompatible.

1. Change **Incompatible** number of **Wind left** to **1**.

2. Change **Incompatible** number of **Wind right** to **1**.

> ⚠️ The wind loading from left is incompatible with the wind loading from right. In load combination, only one of the load groups within an incompatible grouping is taken into consideration at a time.

Create Loads

After modeling the physical structures by creating parts and defining load groups you can add loads to the model.

Loads are modeled as objects and you can copy, move, edit or delete them like any other object in the model. Loads in a structure can be defined as point load, line load, area load with uniform or variable intensity and temperature load. You can model loads to be independent of the structural parts, or they can be attached to parts. Tekla Structures also generates the self-weigh of the structure and use it as uniformly distributed member loads in analysis.

See more in Tekla Structures help: <u>Analysis > Loads > Load types and properties > Load types</u>

Create Point load

We will now create a suspension load using point load type.
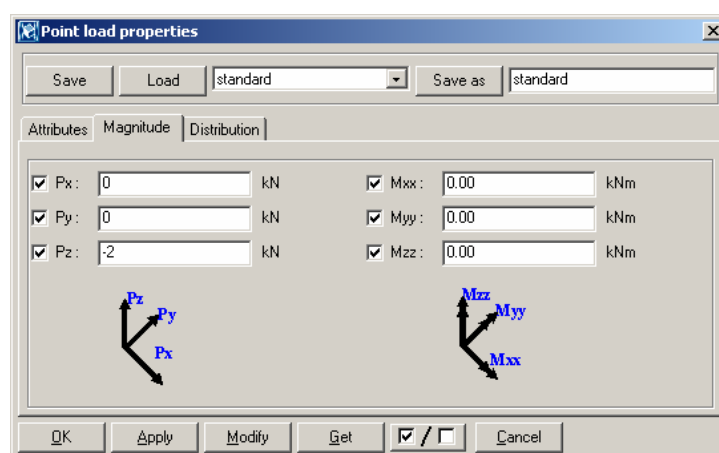
Define point load properties

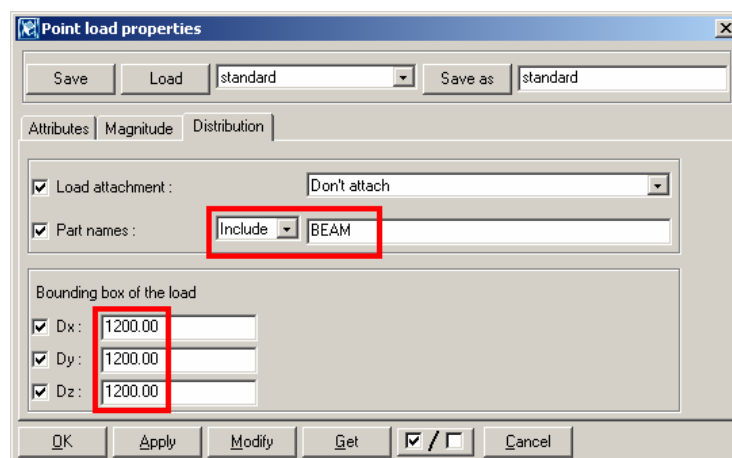1. Doubleclick **Create point load** icon. **Point load properties** dialog opens.

↓

2. Select **Snow** from Group name list. The list has all the load groups, which you have created and defined.
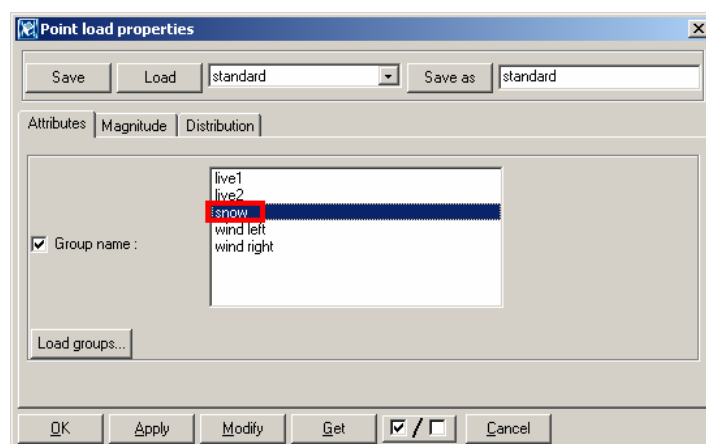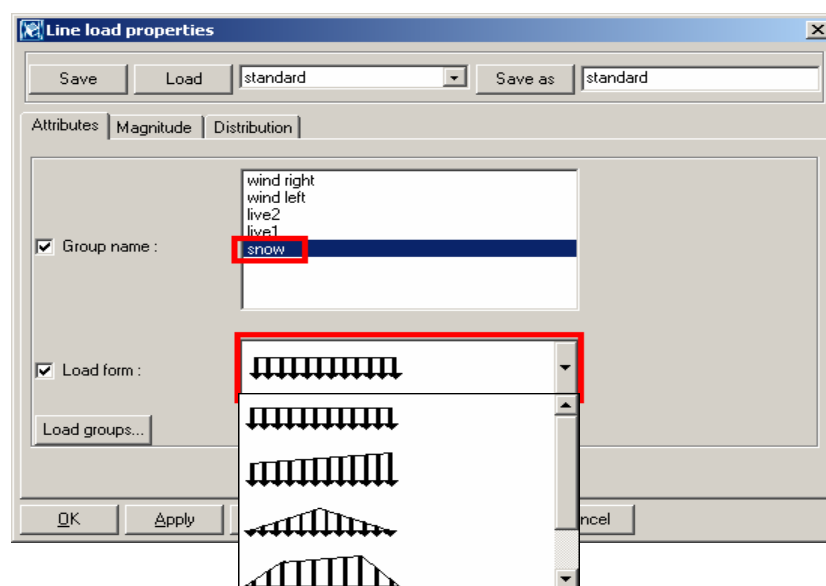

3. Open **Magnitude** tab on the Point load properties dialog.

4. Define point load magnitude to **–2** kN in Pz field.

5. Open **Distribution** tab on the Point load properties dialog.

6. Define load attachement. In Part names select **Include** and write **BEAM** to the name field. See more in Tekla Structures help: Analysis > Loads > Distributing loads > Attaching loads to parts or locations

7. Set bounding box of the load to **1200** in each direction. See more in Tekla Structures help: Analysis > Loads > Distributing loads > Applying loads to parts


8. Click **OK**.

Create point load	Now create a point load and copy it to a beam on grid line **B**, between lines **2** and **3**. Here the point load is representing suspension load.

1. Open the 3D view on level.

2. Zoom in to a top corner on grid lines **A'** and **2'**

3. Pick **Create point load** icon.

4. Point position to the middle of the column top. A point load object is created.
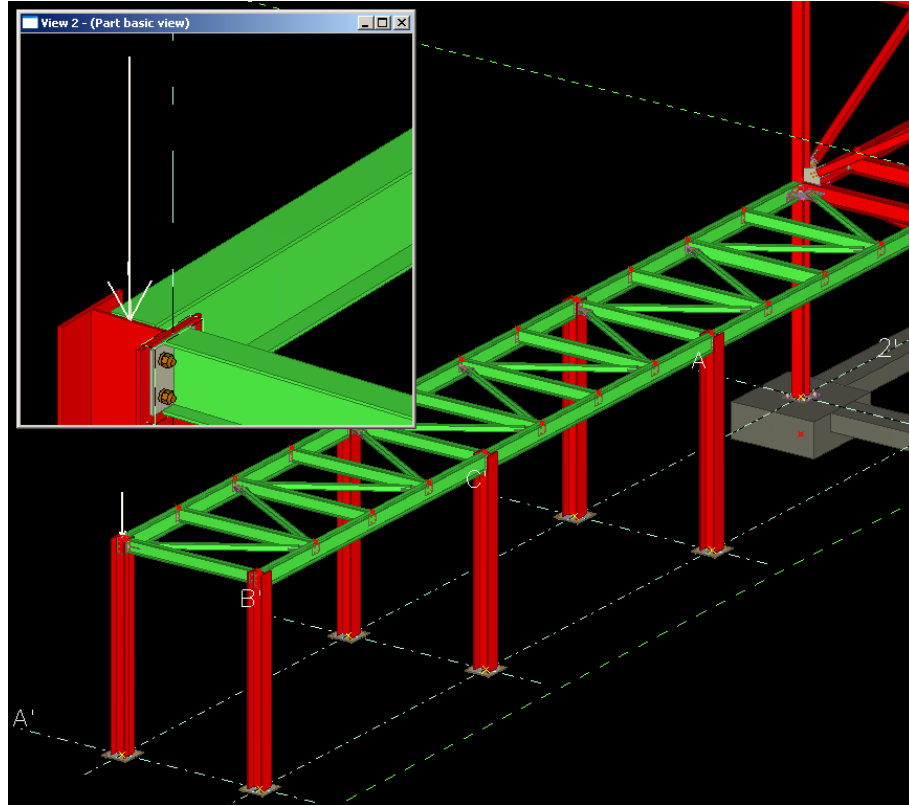
5. Select **Interrupt** from rightclick menu to end the command.

Copy point load   We will next copy the point load.

1. Select the point load you just created.

2. Rightclick and select **Copy > Translate** from the list

3. Fill the Copy translate dialog fields: -500 in dY-direction, Number of copies 5.

4. Press **Copy**. Tekla Structures copies the point load 5 times to given direction.
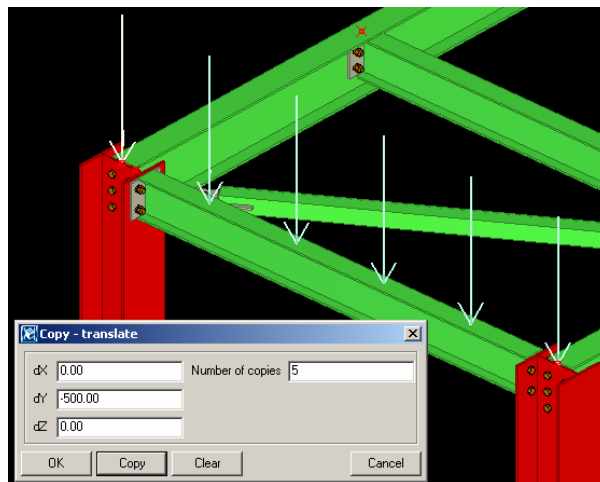
5. Select all point loads.

6. Now copy the load 1500 mms in x-direction **Copy > Translate command**, number of copies is 11.
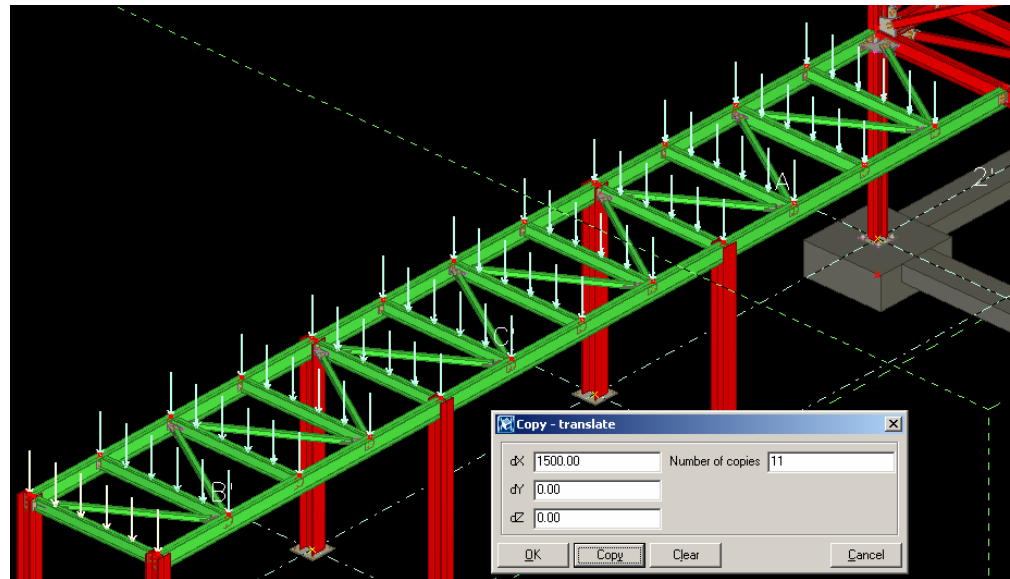
Create Line load

We will now create additional snow loads using line load type.

Define line load properties

1. Doubleclick **Create line load** icon.

2. Select **Snow** from Group name list.

3. Select uniformly distributed load as a load form.

4. Open **Magnitude** tab on the Line load properties dialog.

7. Set magnitude to **-12** kN/m in P1z field.

8. Open **Distribution** tab on the Line load properties dialog.

9. Define load attachement. In Part names select **Include** and write **BEAM** to the name field. See more in Tekla Structures help: Analysis > Loads > Distributing loads > Attaching loads to parts or locations

10. Set bounding box of the load to **1200** in each direction. See more in Tekla Structures help: Analysis > Loads > Distributing loads > Applying loads to parts
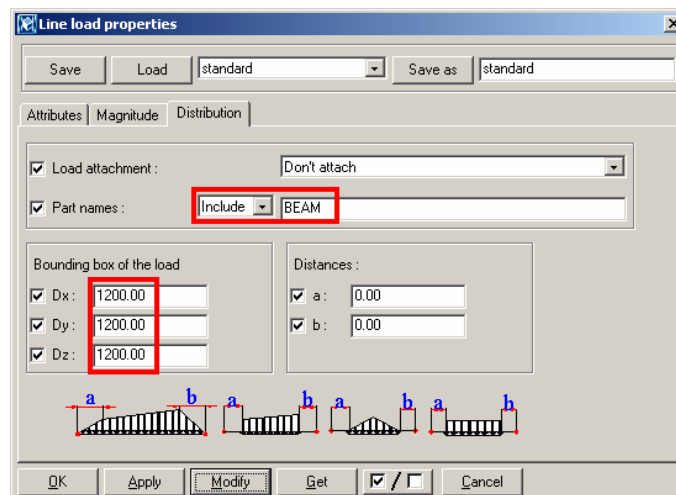
11. Click **OK**.

Create line loads

Now create line loads on top of the roof on grid line **1.** Here the line load represents snow load.

1. Open a view named **GRID 1**

2. Zoom in to upper beams.

3. Pick **Create line load** icon.

4. Point start position to the intersection of column top and grid line **A**.

5. Point end position to the intersection of column top and grid line **B**.

6. Repeat for another line load: point start position to the intersection of beam top and grid line **B**.

7. Point end position to the intersection of column top and grid line **C**.
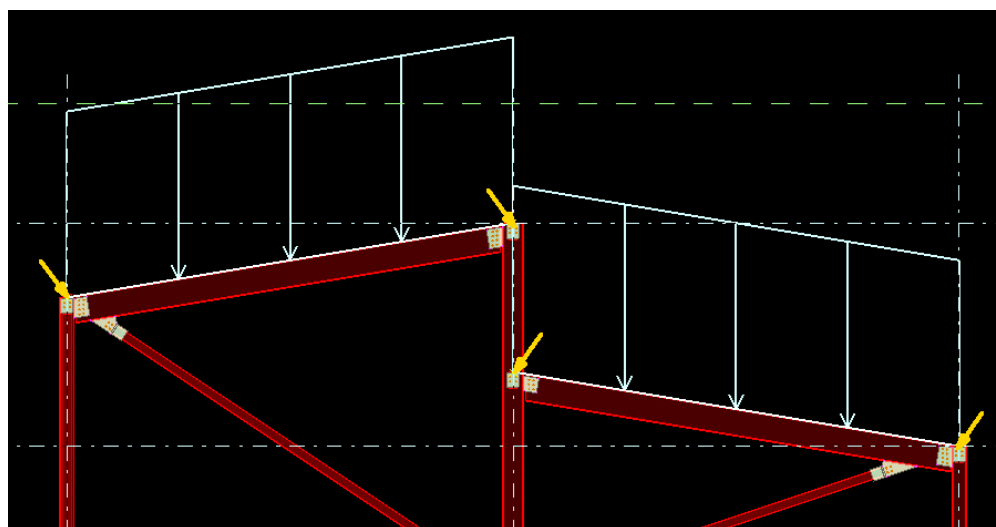
8. Select **Interrupt** from rightclick menu to end the command.

Copy line load	We will next copy the line loads.

1. Select the line loads you just created.

2. Rightclick and select **Copy > Translate** from the list

3. Fill the Copy translate dialog fields: 6000 in dY-direction, Number of copies 3.

4. Press **Copy**. Tekla Structures copies the line loads.
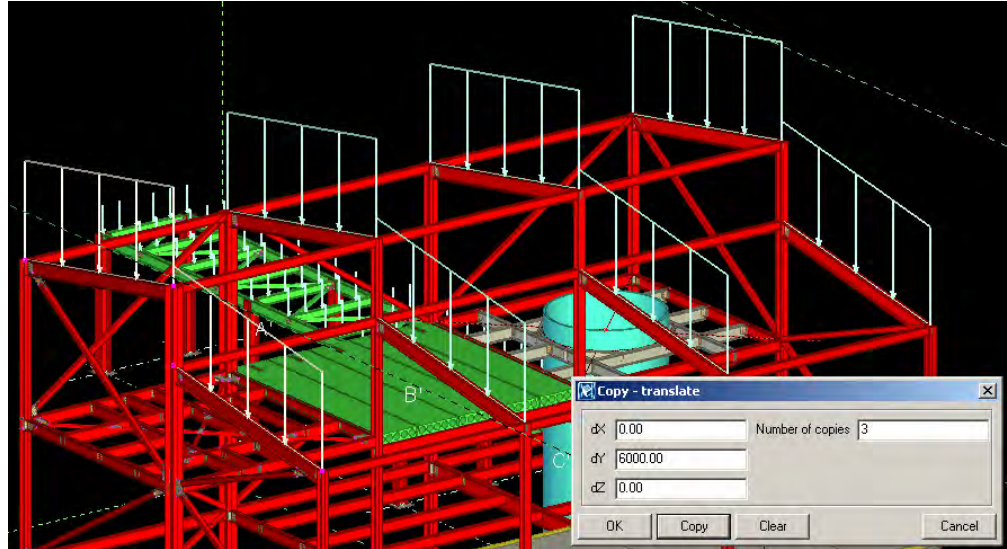
| *Define new line load properties*	We will next create wind loads using the line load.

1. Doubleclick **Create line load** icon.

2. Select **Wind right** from Group name list.

3. Select load form, which has different magnitudes at the ends of the loaded length.
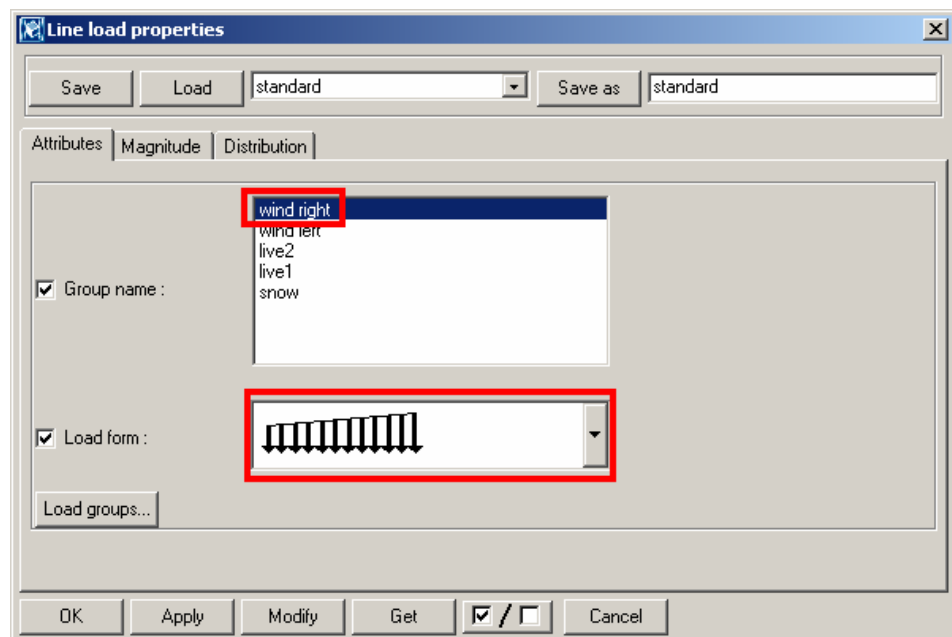
4. Open **Magnitude** tab on the Line load properties dialog.

5. Fill P1x **-2.5** kN/m and P2x **-3.0** kN/m
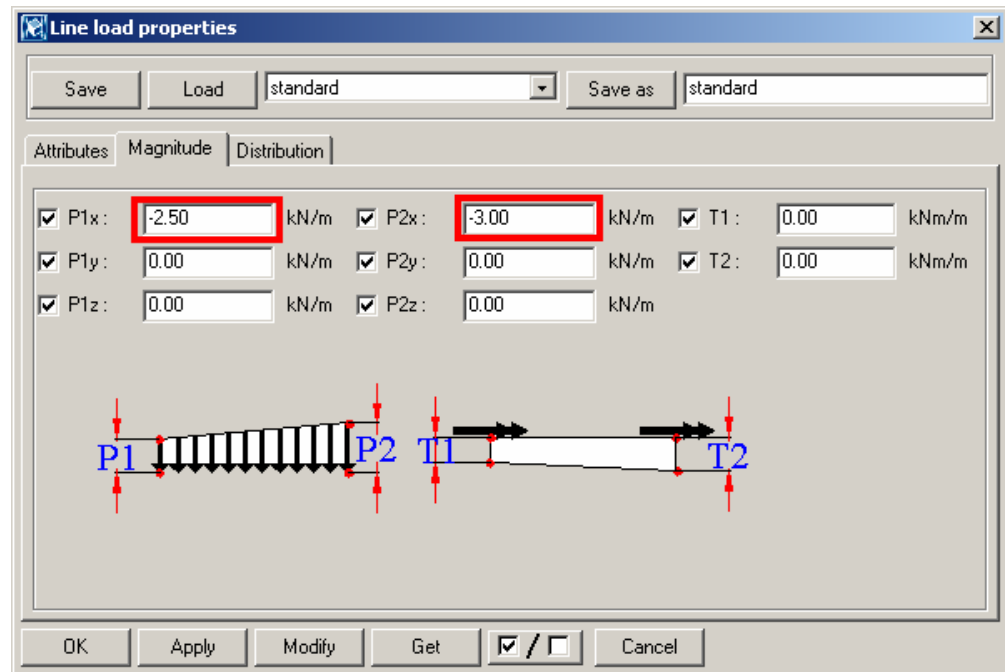
6. Open **Distribution** tab page the Line load properties dialog.
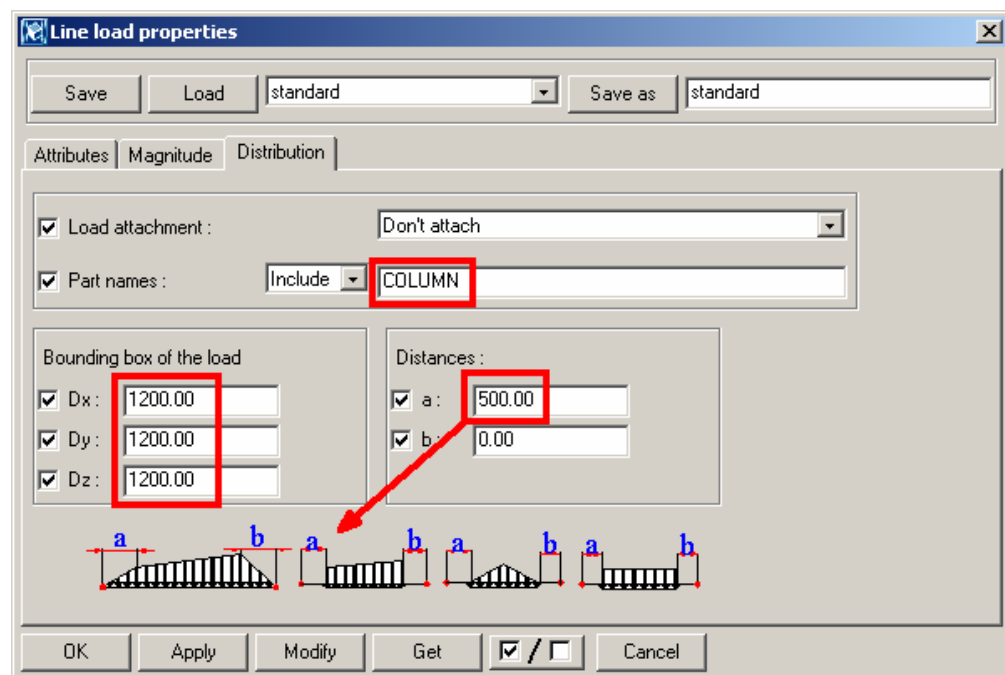
7. Write **COLUMN** into Part names field.

8. Set bounding box to be **1200** mms in each direction.

9. Set offset value **500** mms to the start (**a** filed)

10. Press OK to close the dialog.

Create new line loads

1. In 3D view, zoom in to the concrete column on **1** and **C** grid lines' intersection.

2. Pick **Create line load** icon.

3. Point start position to the intersection of column bottom and grid line **1**.

4. Point end position to the intersection of column top.
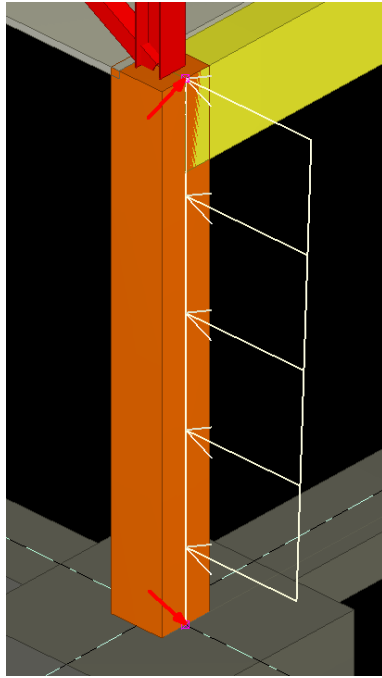
Tekla Structures creates a line load, which has different magnitudes at the ends and which starts 500 mms from the definition point.

Next define new values for a new line load: use same values as previously, but change P1x to be **-3.0** kN/m and P2x to be **-3.5** kN/m and remove the offset from starting point.

Then Create a line load to the steel column on **1** and **C** grid lines' intersection, on top of the concrete column.

Copy line load   We will next copy the line loads.

1. Select the wind line loads you just created.

2. Rightclick and select **Copy > Translate** from the list

3. Fill the Copy translate dialog fields: 6000 in dY-direction, Number of copies 3.

4. Press **Copy**. Tekla Structures copies the line loads.
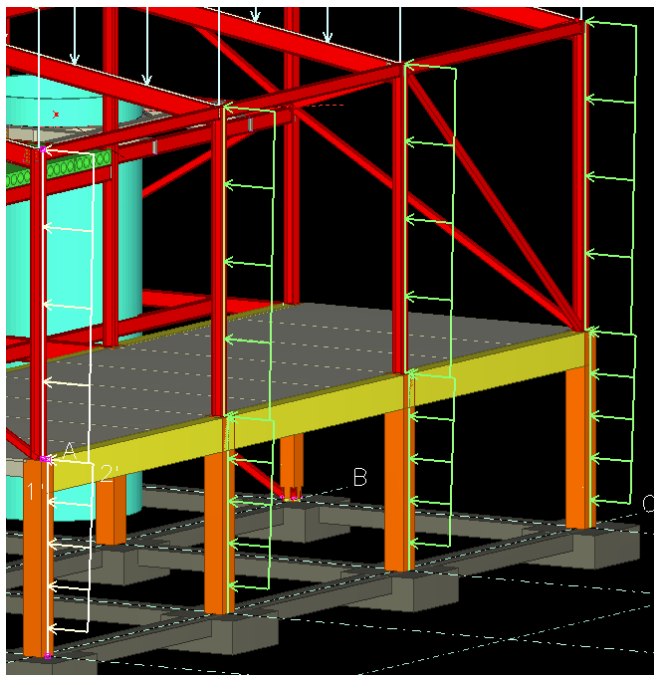
Next create Wind left line loads to the opposite side, on grid line A steel columns using same load form, P1x **2.5** kN/m and P2x **-3.5** kN/m magnitude values, and 500 mms offset in starting point.
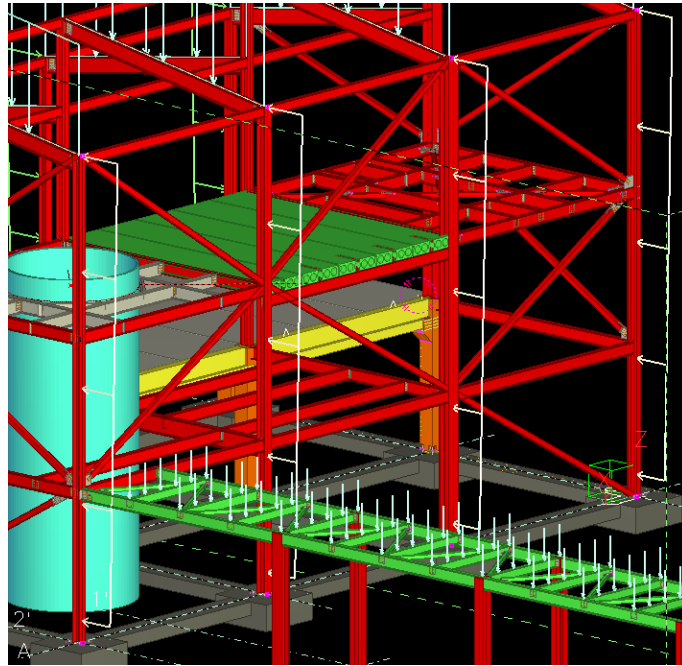
Create Area load

We will now create live loads using area load type.

*Define area
load properties*

1.   Doubleclick **Create area load** icon.

2.   Select **live1** from Group name list.

3.   Select **quadrangular** load as a load form.
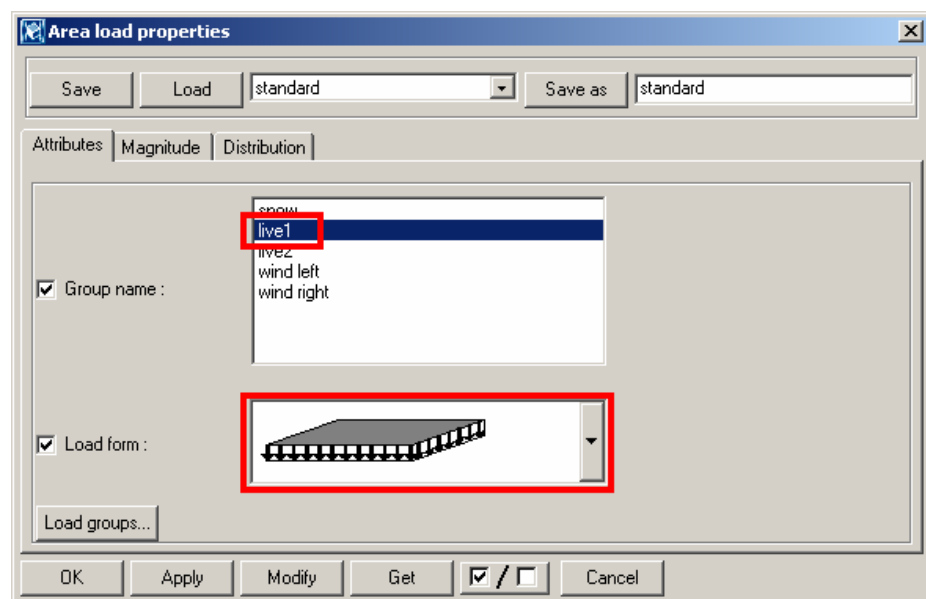
4.   Open **Magnitude** tab on the Area load properties dialog.

5.   Set magnitude to **-8.00** kN/m² in P1z, P2z and P3z fields.
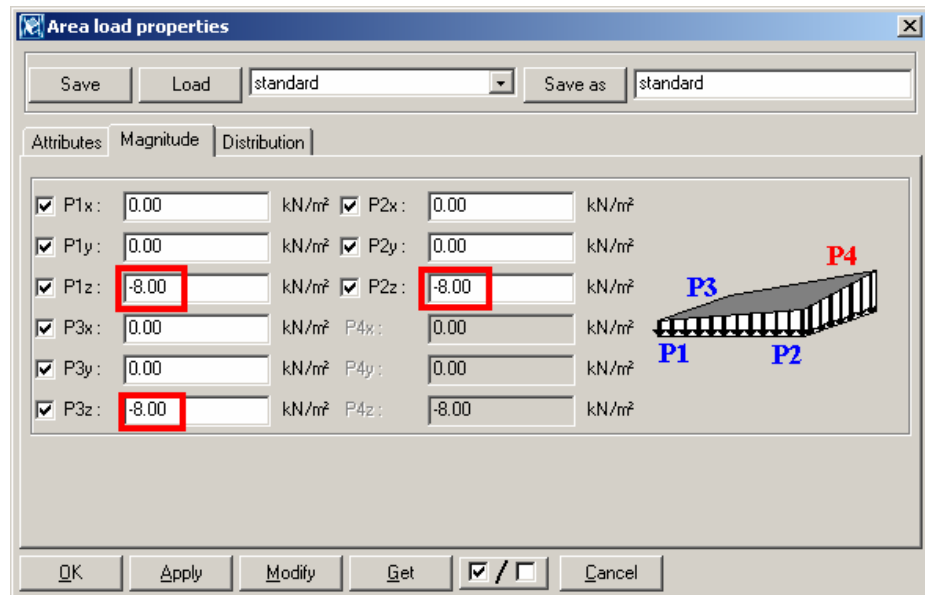
6. Open **Distribution** tab on the Area load properties dialog.

7. Set bounding box of the load to **1200** in each direction. Set off set from sides to – 300 mms. See more in Tekla Structures help: <u>Analysis > Loads > Distributing loads > Applying loads to parts</u>
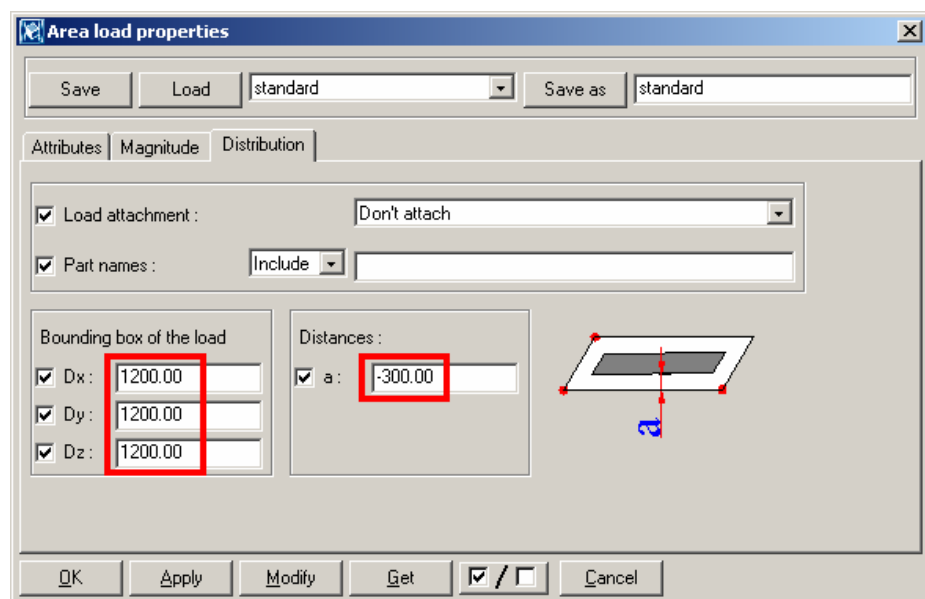
8. Click **OK**.

Create area load: Live load1

We will next define live 1 load by 3 points on top of slab in level +3500.

1. Open the **Plan +3500** view. Check that you work on **view plane**.

2. Zoom in to the intersection of grid lines **C** and **4**.

3. Pick **Create area load** icon.

4. Point start position to grid line intersection.

5. Point second position to another grid line intersection

6. Point end position again to grid line intersection.
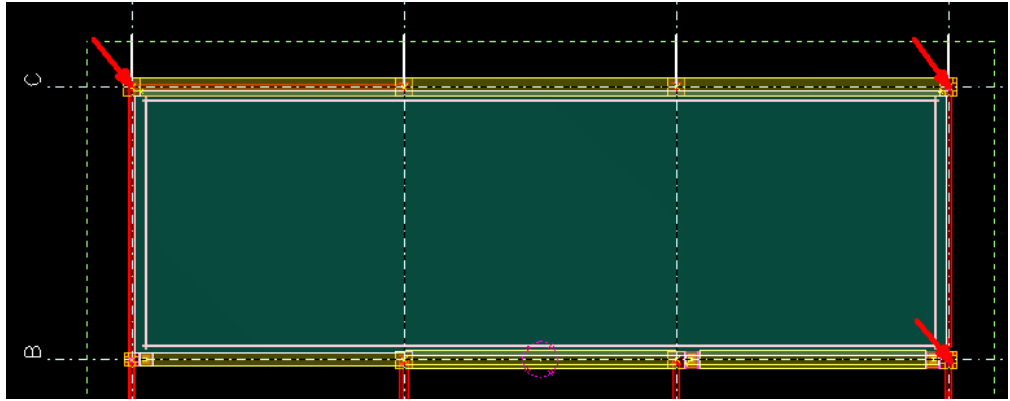
Create area load: Live load2

We will next define and create live2 load to level +7000. Define first the load parameters:

1. Doubleclick **Create area load** icon.

2. Select **Live2** from Group name list.

3. Select **quadrangular** load as a load form.

4. Open **Magnitude** tab on the Area load properties dialog.

5. Set magnitude to -**6** kN/m² in all **Pz** fields.

6. Open **Distribution** tab page.

7. Set bounding box of the load to **1200** in each direction.

8. Set offset to -100 mms

9. Click **OK**.

Now create the area load **Live2**:
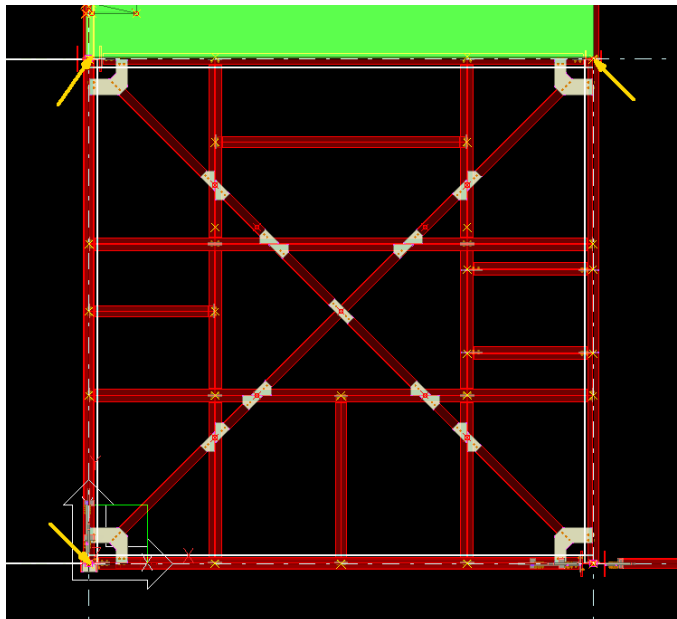
1. Open view **Plane +7000**.

2. Pick **Create area load** icon.

3. Point start position to the intersection of grid lines **A** and **1**.

4. Point second position to another grid line intersection.

5. Point end position again to grid line intersection.

Create Uniform load

We will next create a uniformly distributed load, which is a linearly distributed load/force that is inside a polygonal (distribution/loaded) area. We will also add an opening to the polygon.

1. Pick **Create uniform load** icon.

Define Uniform load properties

2. Select **Live1** from Group name list.

3. Open **Magnitude** tab on the Uniform load properties dialog.

4. Set magnitude to –**2.0** kN/m² in P1z field.
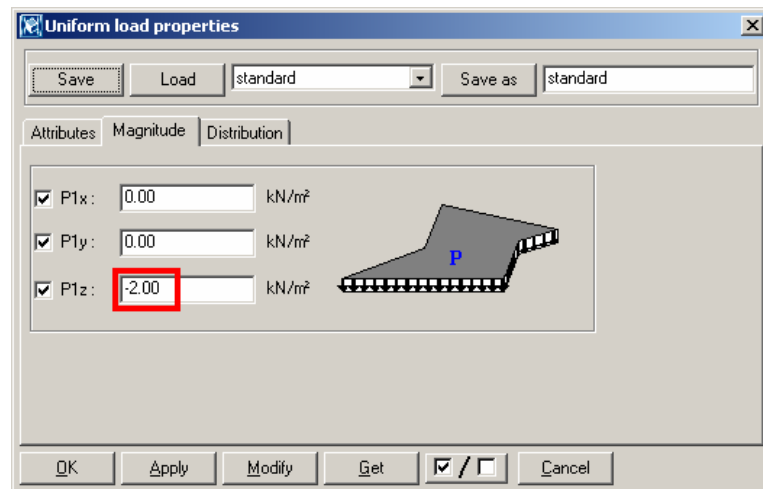
5. Open **Distribution** tab on the Uniform load properties dialog

6. Set bounding box to 1200 in each direction.

7. Set offset to -100 mms.
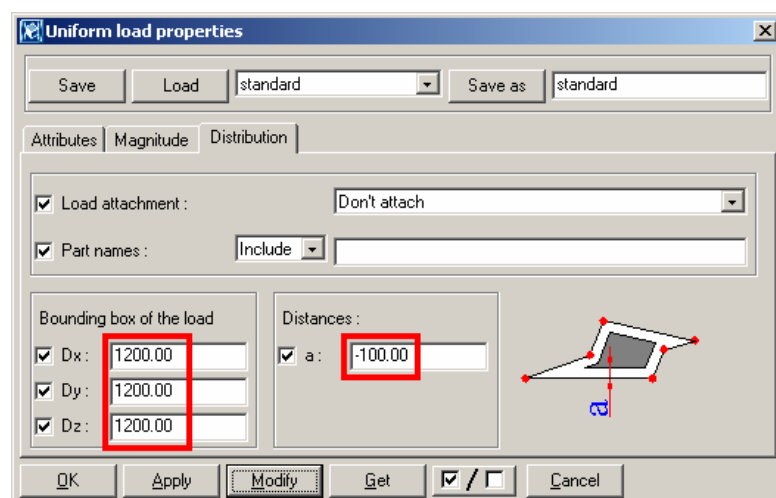
8. Close the dialog by pressing **OK**.

Create uniform load

Add uniform load to level +**7000** by picking polygon shape and opening shape, then executing command by middle mouse click.

1. Pick polygon shape (points 1 - 5) and close the outermost polygon to starting point (point 5)

2. Continue picking the points of the holes (points 6 - 14) and close the inner polygon (point 14). You can do this clockwise or counterclockwise.

3. Finish command with **middle button**.
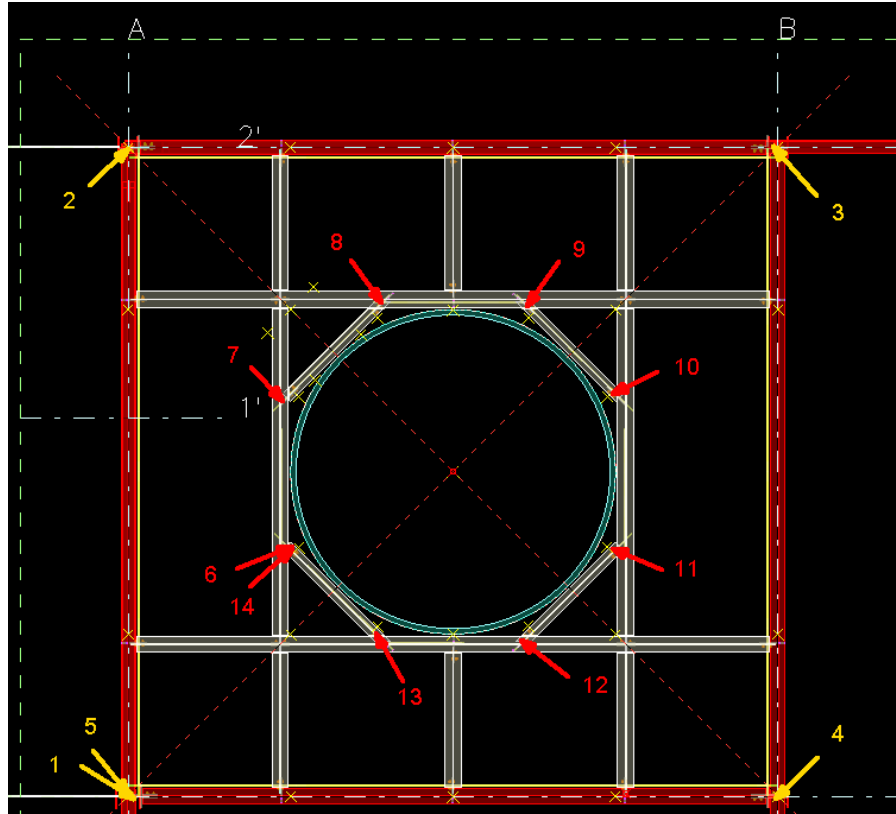
Tekla Structures creates an uniform load, which is cut around the silo.
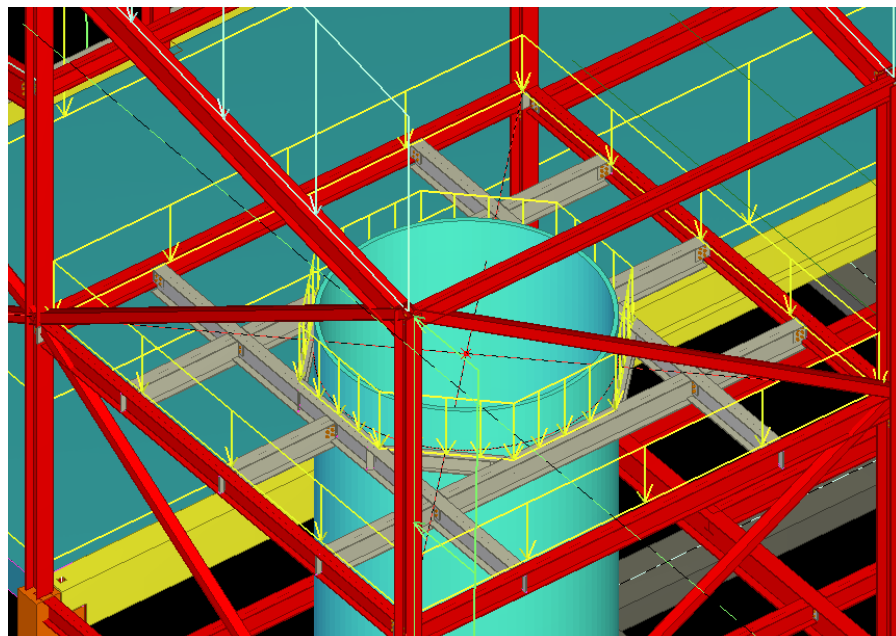
17.2 Restraints

To get a valid analysis model that can be solved with FEM analysis engine, we need to specify the supports and member end or edge releases. To define the correct values in the analysis model the information should be stored in model. See more in Tekla Structures help: Analysis > Analysis and Design > Member end connectivity

Define Restraints for Members

We will now define restraints first to beams having point and line loads.

1. Double click any beam and open **Beam properties** dialog box.
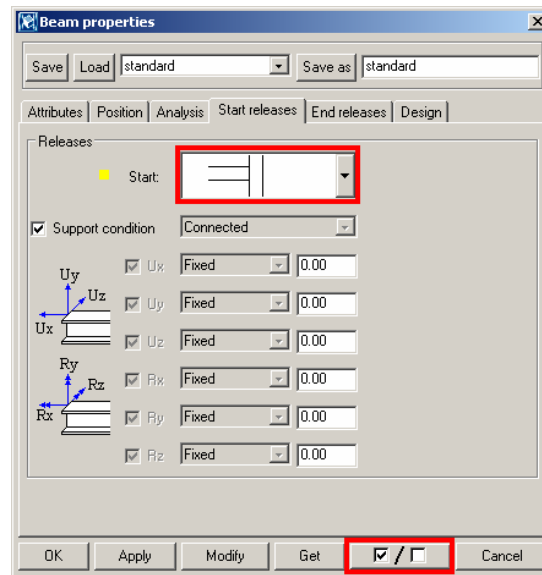
2. Open **Start releases** tab page

3. Set Support condition to "Connected"

4. Open **End releases** tab page

5. Set Support condition to "Connected"

6. Untick all dialog fields by pushing **On/off** button (between Get and Cancel buttons)

7. Tick Support condition on Start and End releases tab pages

8. Select all the beams, which have point, line or area loads.

9. Push **Modify**. Only support conditions are modified.

We will next define restraints to all columns.

1. Double click any column and open **Column properties** dialog box.

2. Open Start releases tab page

3. Set Support condition to "Supported"

4. Open End releases tab page

5. Set Support condition to "Connected"

6. Untick all dialog fields by pushing **On/off** button (between Get and Cancel buttons)

7. Tick Support condition on Start and End releases tab pages
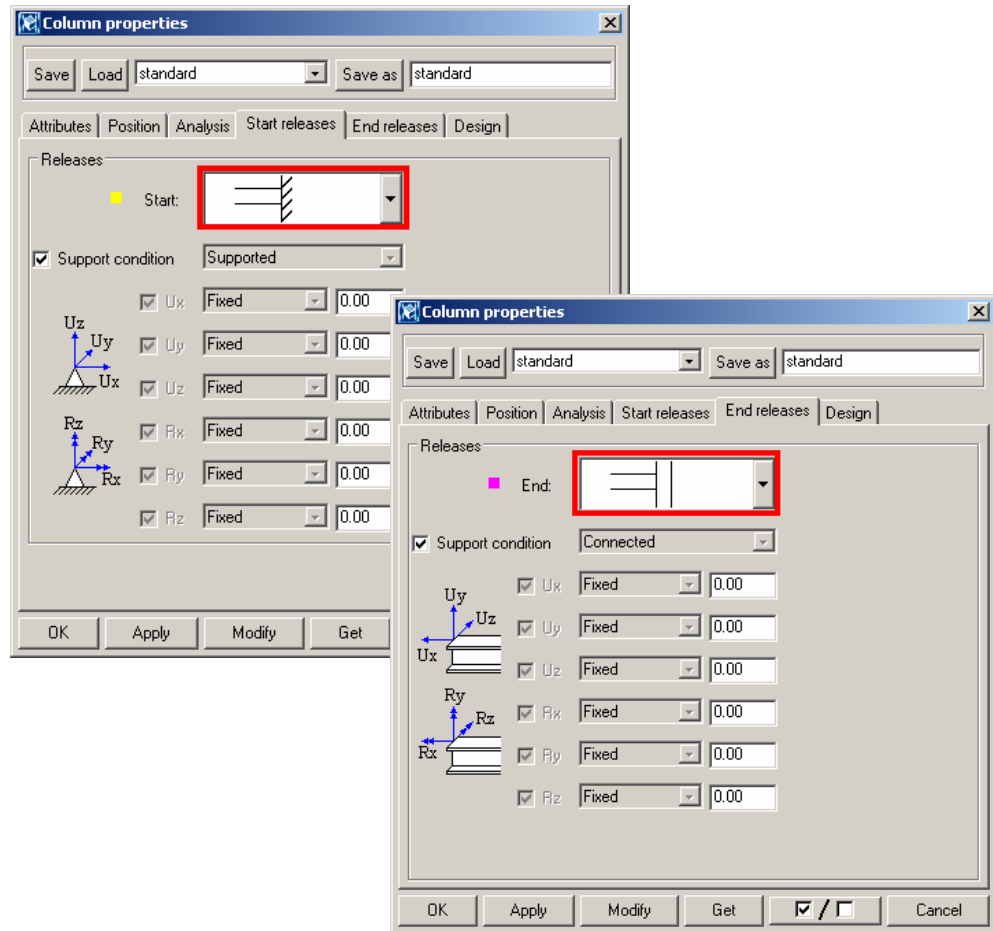
8. Select all the columns.

9. Push **Modify**. Only support conditions are modified.

We will next define restraints to bracings.

1. Double click any beam and open **Beam properties** dialog box.

2. Open **Start releases** tab page

3. Set Support condition to "Connected", which allows rotation.

4. Open **End releases** tab page

5. Set Support condition to "Connected", which allows rotation.

6. Untick all dialog fields by pushing **On/off** button (between Get and Cancel buttons)

7. Tick Support condition on Start and End releases tab pages

8. Open **Select filter** dialog

9. Write **BRACE** to Name field on Part tab page.

10. Close the Select filter dialog by pressing **OK**.

11. Area select all the bracings.

12. Press **Modify** on the **Beam properties** dialog box. Only support conditions are modified.

13. Select **Standard** from Select filter list to return to normal selection.

Then set the analysis conditions for the in situ slab.

1. Double click the slab and open **Slab properties** dialog box.

2. Open **Analysis** tab page
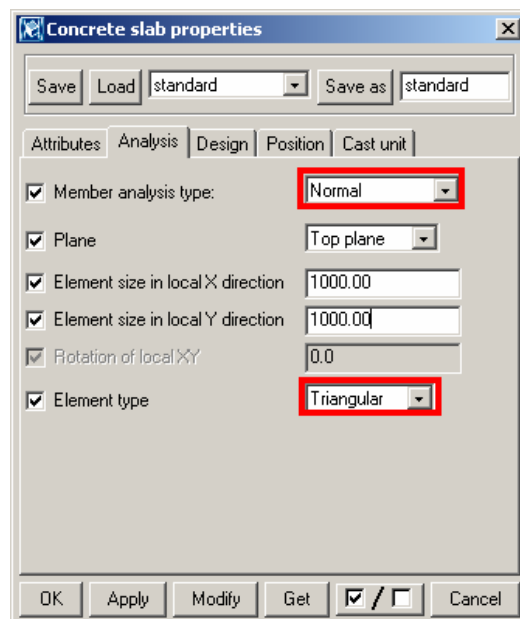
3. Set member analysis type to "Normal"

4. Select **Triangular** element type

5. Press **Modify**

6. Close the dialog by pressing **OK** or **Cancel**.

17.3 Create Analysis Model

The model Structures are analyzed by using the finite element method (FEM). All the input data required and the results generated in the structural analysis constitutes an analysis model. The analysis model consists of nodes, elements/members, loads, restraints, constraints, and other related information. When starting the structural analysis, Tekla Structures generates an analysis model of the corresponding physical and load models. This includes the splitting of physical parts into analysis members, node generation for the members, additional analysis member generation, support condition determination, load splitting, and load division for the analysis members. See more in Tekla Structures help: Analysis > Analysis and Design > Analysis model properties

Create a New Analysis Model

Before creating the analysis model, check that you have the correct load modeling code selected. From **Setup** pull down menu, select **Analysis load modeling…** Use the **Analysis load modeling dialog box** to determine the building code and safety factors Tekla Structures uses in load combination.

Define analysis model attributes

1. Select all loads and members except pad footings and strip footings.
2. In **Analysis** drop down menu select **New model…**
3. Give a name to the analysis model: Full model
4. Set creation method to "**By selected parts and loads**"
5. Set member axis to "**Reference axis**"

6. Set Node definition to "Use rigid links"

7. Open the **Analysis** tab page

8. Set analysis method to 1st order.

9. Open **Design-Steel** tab page

10. Select the "EC3" code from Desing code

11. Set design method to **Check design**.

12. Open **Design-Concrete** tab page

13. Select the "EC2" code from Desing code
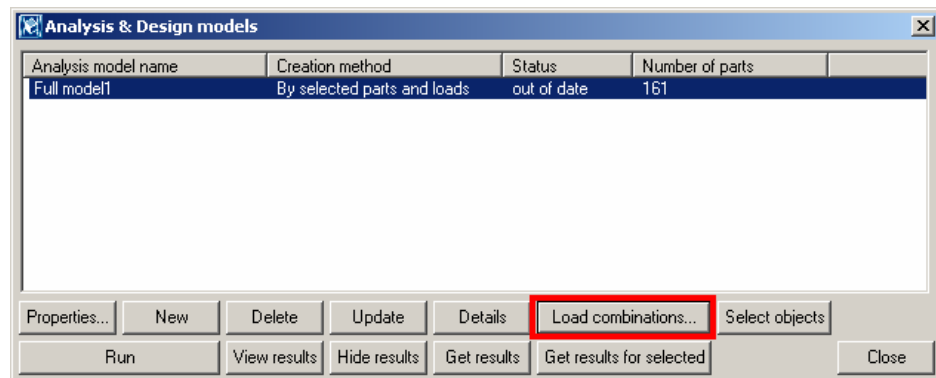
14. Set design method to **Calculate reguired area**.
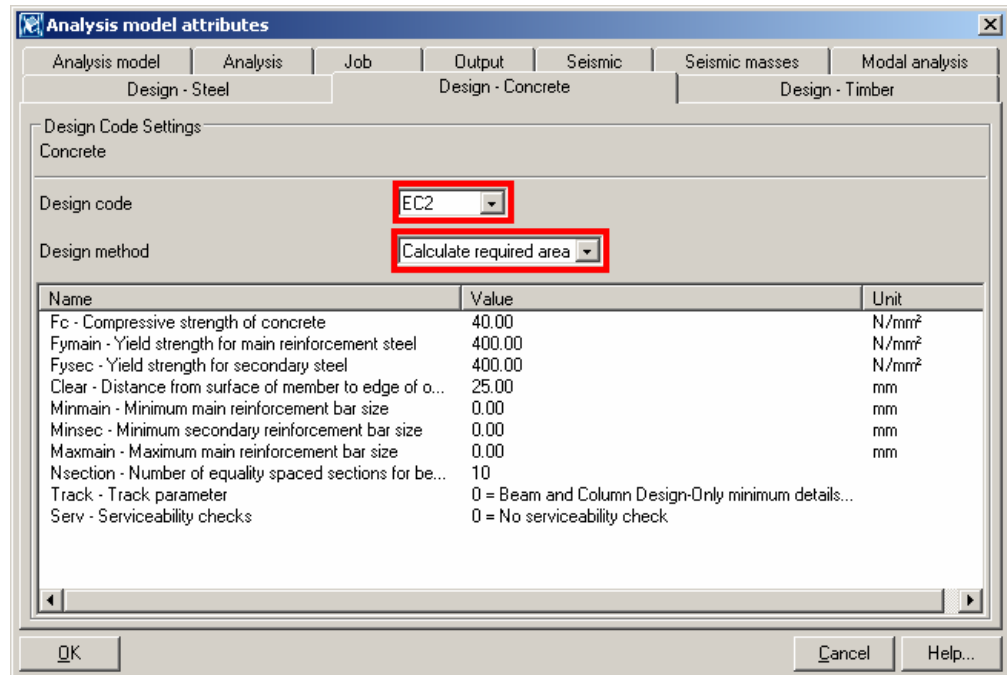
15. Press **OK**. Tekla Structures constructs the analysis model.

Define load combinations

Before running the analysis we need to define the load combination, which will be used in analysis.

1. Select the analysis model in **Analysis & Design models** dialog.

2. Press **Load combinations...**

3. Load combinations dialog opens. Press **Generate…** button to use automatic load combination generation.
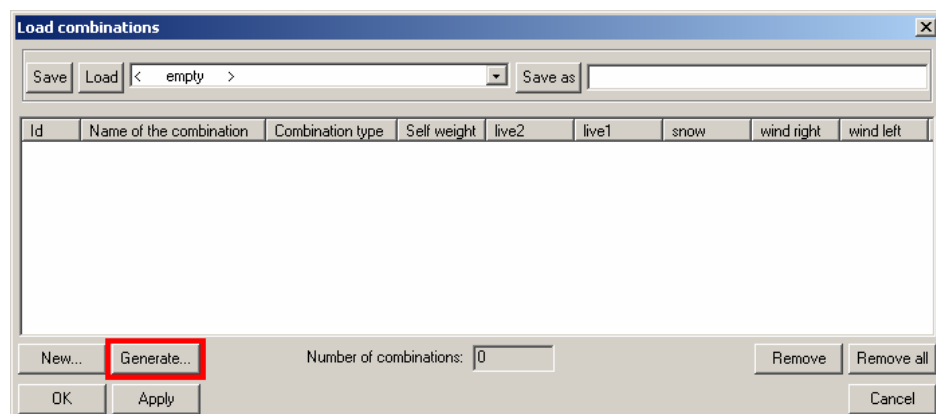
4. Tick all combination options.
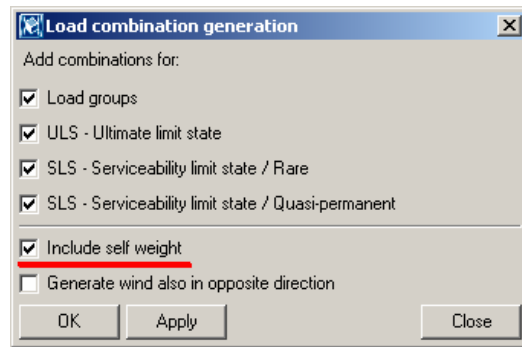
⚠️ Note that self weight can be included automatically.

5. Press **OK** to insert the combinations to Load combinations list.

Tekla Structures generates load combinations using code specific partial safety factors.

Id	Name of the combination	Combination type	Self weight	live2	live1	snow	wind right	wind left
1	LG1	LG	1.00x1.00					
2	LG2	LG		1.00x1.00				
3	LG3	LG			1.00x1.00			
4	LG4	LG				1.00x1.00		
5	LG5	LG					1.00x1.00	
6	LG6	LG						1.00x1.00
7	ULS7	ULS	1.00x1.35					
8	ULS8	ULS	1.00x1.00					
9	ULS9	ULS	1.00x1.35	1.00x1.50	0.70x1.50			
10	ULS10	ULS	1.00x1.35	0.70x1.50	1.00x1.50			
11	ULS11	ULS	1.00x1.00	1.00x1.50	0.70x1.50			
12	ULS12	ULS	1.00x1.00	0.70x1.50	1.00x1.50			
13	ULS13	ULS	1.00x1.35				1.00x1.50	
14	ULS14	ULS	1.00x1.35					1.00x1.50
15	ULS15	ULS	1.00x1.00				1.00x1.50	
16	ULS16	ULS	1.00x1.00					1.00x1.50
17	ULS17	ULS	1.00x1.35			1.00x1.50		
18	ULS18	ULS	1.00x1.00			1.00x1.50		
19	ULS19	ULS	1.00x1.35	1.00x1.50	0.70x1.50	0.60x1.50	0.60x1.50	
20	ULS20	ULS	1.00x1.35	1.00x1.50	0.70x1.50	0.60x1.50		0.60x1.50
21	ULS21	ULS	1.00x1.35	0.70x1.50	1.00x1.50	0.60x1.50	0.60x1.50	
22	ULS22	ULS	1.00x1.35	0.70x1.50	1.00x1.50	0.60x1.50		0.60x1.50
23	ULS23	ULS	1.00x1.00	1.00x1.50	0.70x1.50	0.60x1.50	0.60x1.50	
24	ULS24	ULS	1.00x1.00	1.00x1.50	0.70x1.50	0.60x1.50		0.60x1.50

6. Add a new combination, which has modified partial safety factors by pressing **New…** button.
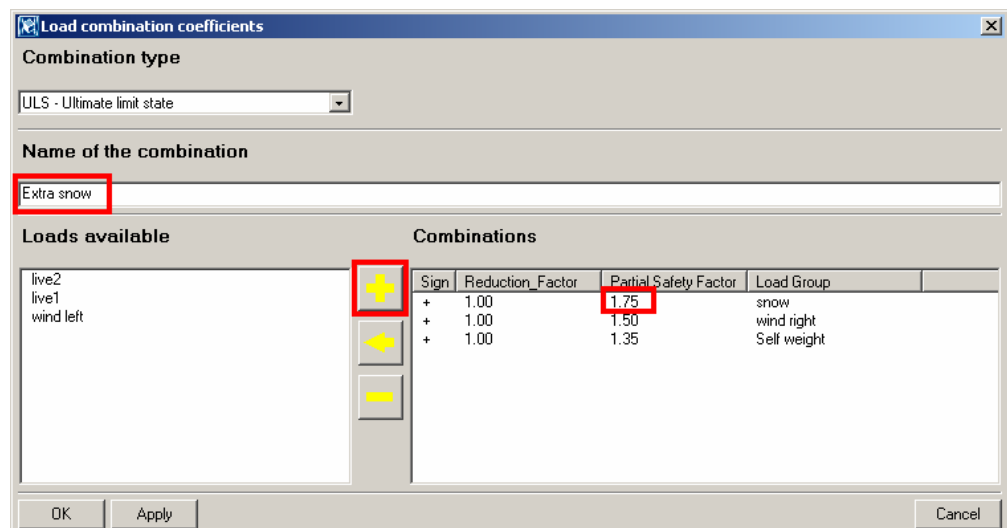
7. Select combination type to be "ULS-Ultimate limit state".

8. Enter name "Extra snow" for the load combination

9. Move load groups "Snow", "Wind right" and "Self weight" from Loads available list to Combinations by selecting the loads an pressing the + button.

10. Change the Partial safety factor of "Snow" from 1.50 to 1.75

11. Press **OK** to insert the load combinations to the Load combinations list.

12. Press **OK** to accept the load combinations and to insert them into the analysis model.

Analysis Model Calculation and Result Viewing

Run the calculation

Analysis model is now ready to be analyzed using STAAD*pro*2003.

1. Select the analysis model.

2. Press **Run** to start analysis.

3. **STAAD*pro*2003** analyses the model and writes an .anl file, which will be saved in **Model_folder \Analysis** directory. Press **Done** to close the dialog.
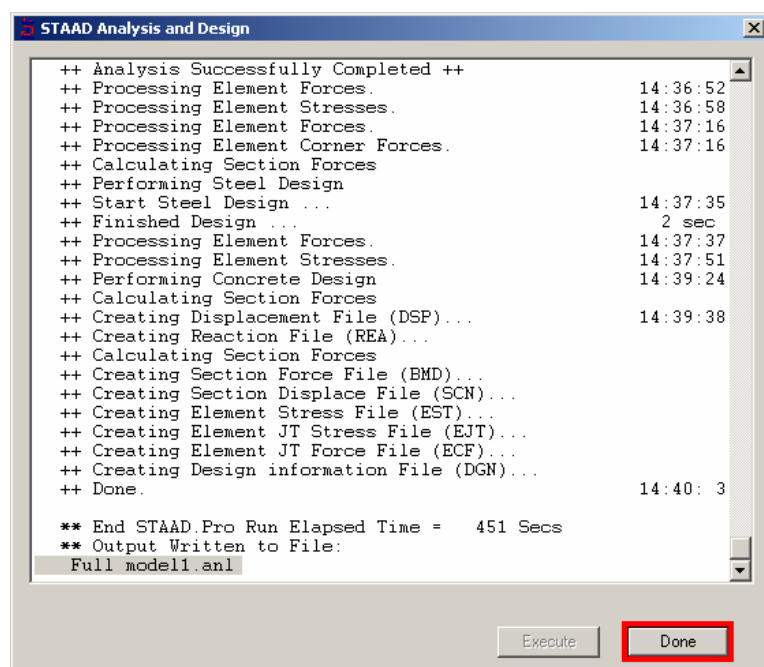
Analysis model has now been analyzed and the results can be seen in STAAD*pro*2003.

View results

Press **View results** button.

STAAD*pro*2003 program opens in post prosessing mode.

Check the Output File

During the analysis process, Tekla Structures creates an Output file. This file provides important information on whether the analysis was performed properly. For example, if Tekla Structures encounters an instability problem during the analysis process, it will be reported in the output file.

1. Push the **Output file** icon to open the file.

2. Check if there are any warnings in list on the left side.

3. Select the warning in the list.

4. Find the cause of the warning from the text on the right. There is usually the node number, which you can locate in the STAAD- model.
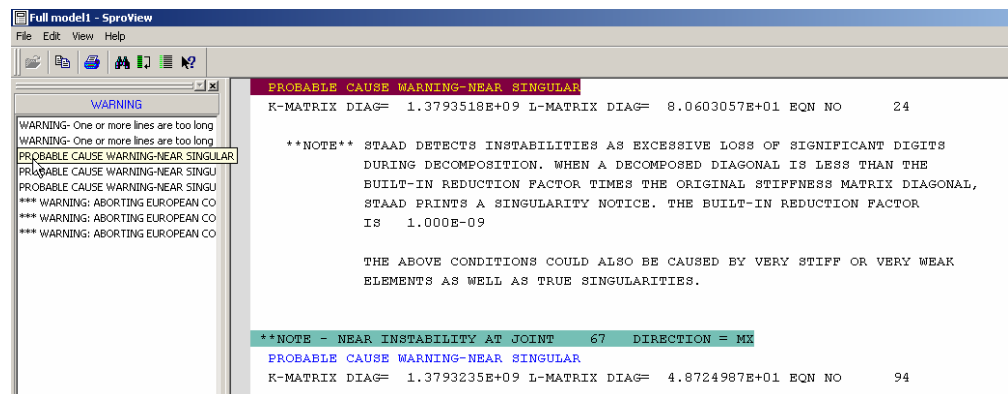
5. Find the node number in the Node displacements list. The node is highlighted in STAAD model and you can use zoom window to have a closer look. Shift + N shows the node numbers in the view.
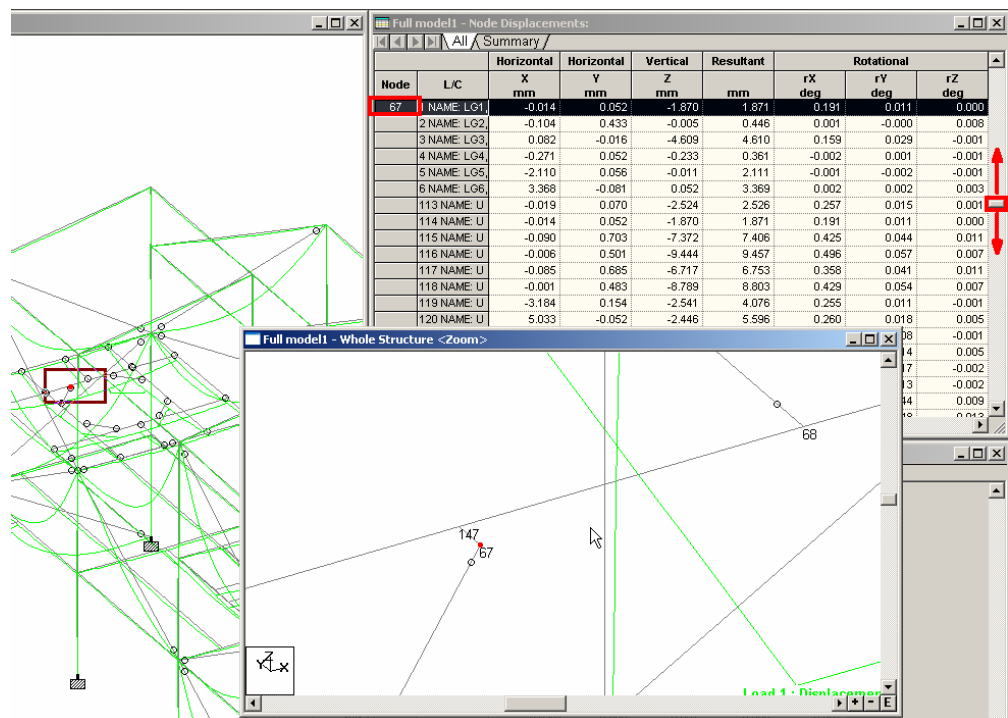
Check Results Both Graphically and Numerically in STAAD*pro*2003

Steel members

We can now check result windows in STAAD*pro*2003. Start by checking that the loads are correct.

Check loads
1. Open **Beam > Graphs** tab page on the left side of the result window.

2. Select corresponding load group LG2 from **View** toolbar

The loads in the selected group are displayed.
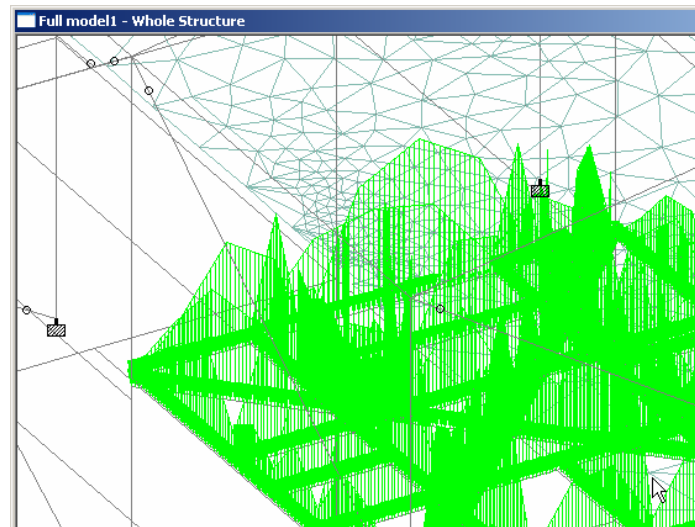
3. Pick **Load edit cursor** icon from Selection toolbar.

4. Doubleclick a load to see the properties
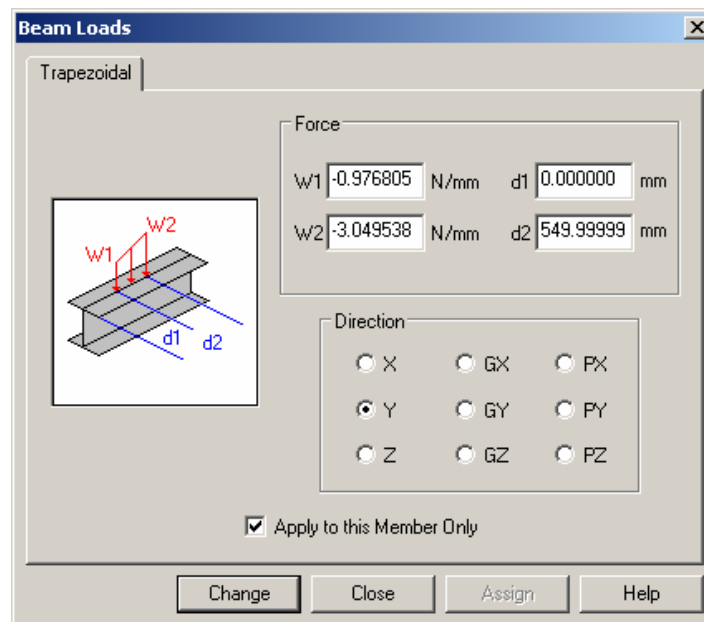
5. Check that load magnitude and direction are the same as you have set in Structures load model.

6. Select next load group from **View** toolbar to see the loads.

7. Doubleclick line load using **Load edit cursor** to see the properties.

8. Repeat for remaining load groups.

You can view the member end forces in graphical and tabular forms. Check Axial forces, Shear forces in Y and Z directions, Torsion and Bending moments in Y and Z directions. The scale factor for plotting the diagram may be changed using the menu option **Results > Scale > Scales tab.**
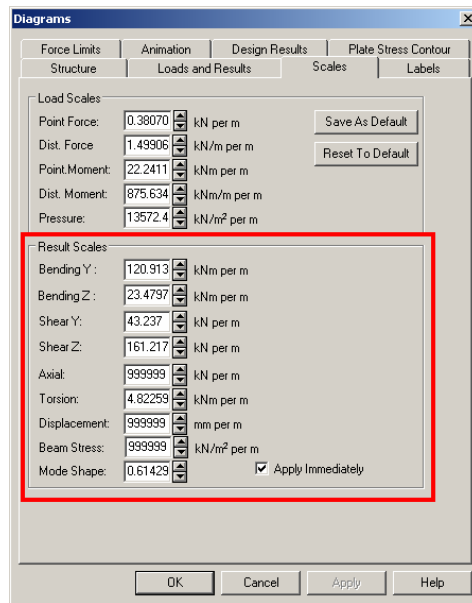
1. To view the forces, open **Beam > Forces** tab page on the left side of the result window.

2. Select member forces from **Results** toolbar, for example Bending Z and check visually the model if there are too big bending moments in different load combinations.

3. Select a single beam to see the forces in more detailed form in the list on the right.

4. **Steel detailing** tab shows the critical load combination for the selected beam.

5. On **Shear Bending** tab, select the critical load combination to see the details.
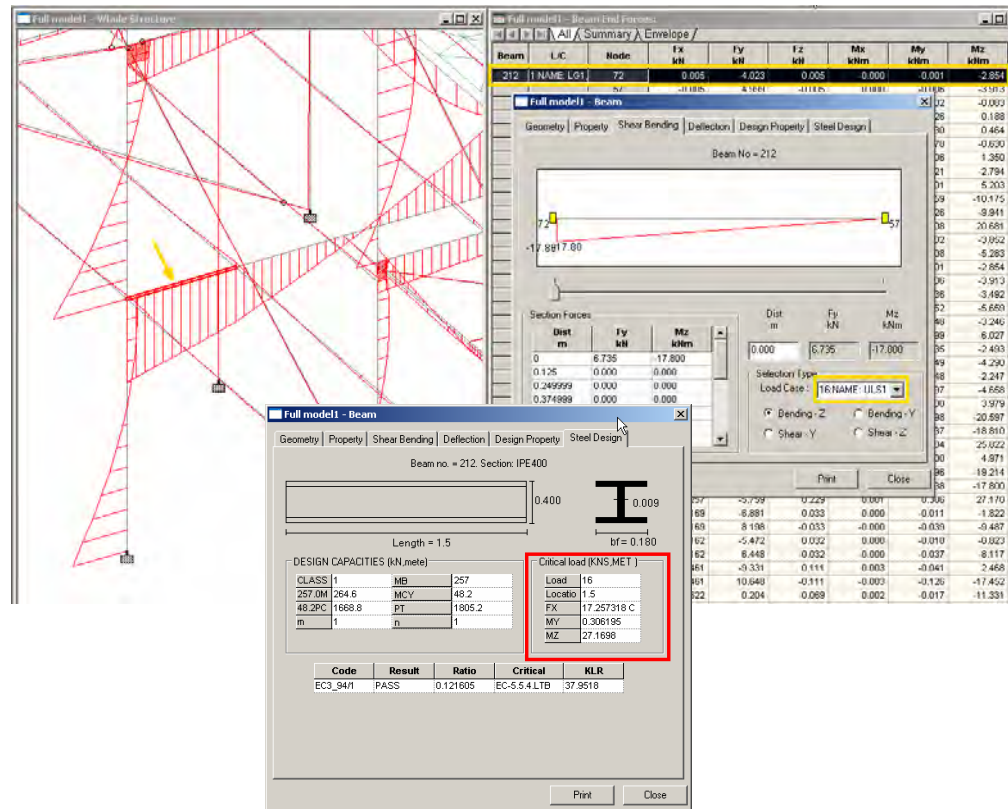
It is also possible to have more graphics of the forces.

1. To view detailed force graphics select **Beam > Graphs** tab page on the left side of the result window. STAAD*pro*2003 opens new graphical windows to view the Mz, Fy and Fx forces.

2. Select a beam from the model. The force values are displayed.
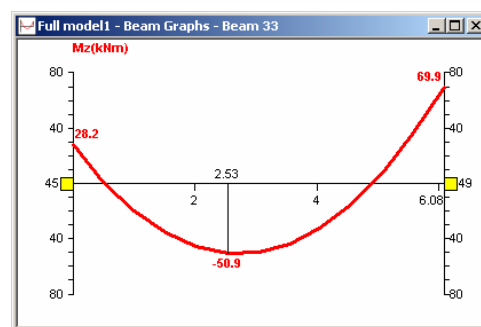
Check member stresses

Member stresses can be printed at specified intermediate sections as well as at the start and end joints.

1. To view the stresses, open **Beam > Stresses** tab page on the left side of the result window.

2. Select a beam from the model. STAAD*pro*2003 shows the stresses graphically and in tabular forms.

3. Move the handle in the **Select section plane** dialog to view stress values in wanted distance from the end of beam.

4. Tick the **Display Legend** and **Display Corner Stress** options to view them in the window.

5. Push **Add Stress to Table button** to save the stress values at given distance. The values can later be called into a report.
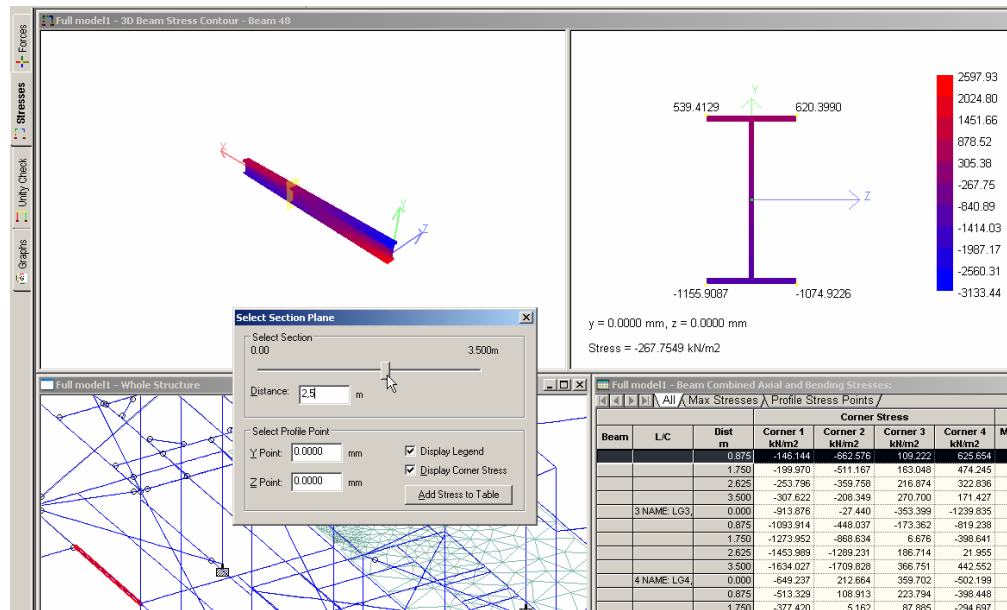

Utility Check

When **code checking** is selected, the program calculates and prints whether the members have passed the code or have failed; the critical condition of the code (any of the compression, tension, shear, etc.); the value of the ratio of the critical condition (overstressed for a value more than **1.0** or any other specified RATIO value); the governing load case, and the location (distance from the start of the member) of forces in the member where the critical condition occurs. The colors and limits for safe and failed members for plotting the diagram may be changed using the menu option **Results > Scale > Design Results tab.**

1. To view the Utility check results, open **Beam > Utility Check** tab page on the left side of the result window.
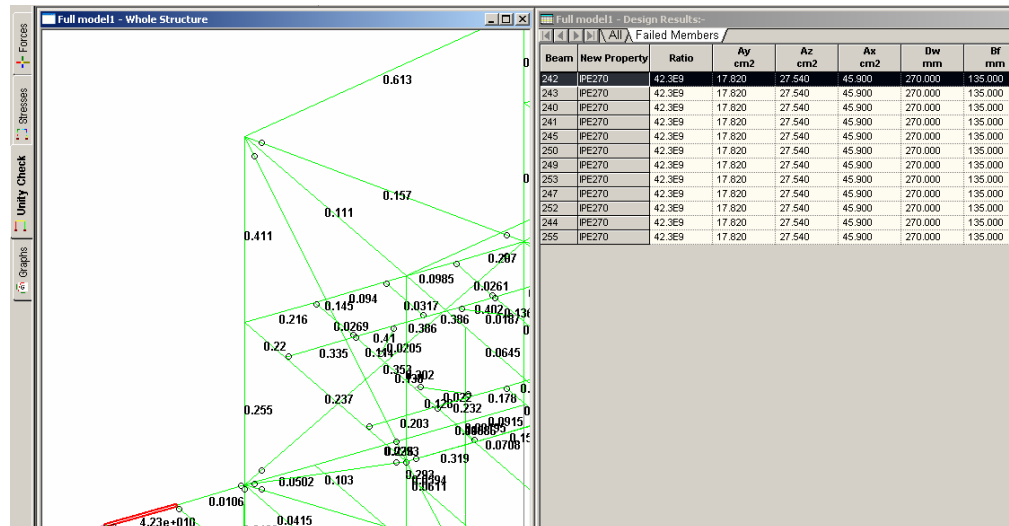
2. Select **Failed members** from the tabular data on the right.

3. Select a row from the Failed members list to activate the member in the model.

4. Double click the failed member to see more detailed data in the Beam dialog.
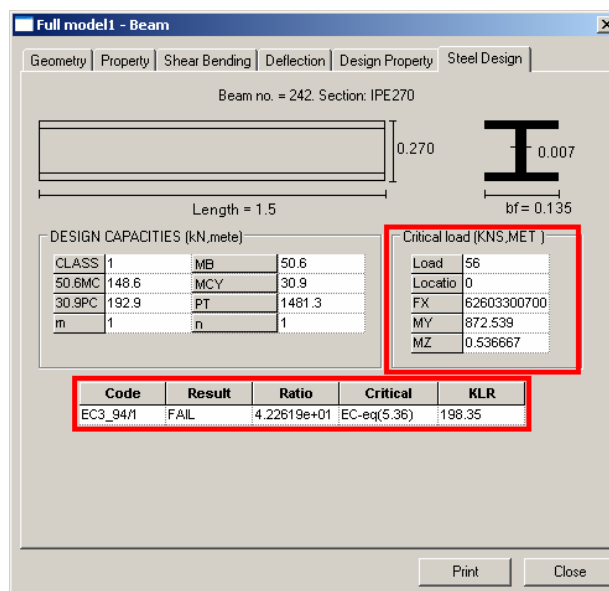
Creating Customized Reports, Plotting Result Diagrams

The **Report** menu options display various numerical results in tabular form. The results may be sorted in ascending or descending order. Each table appears as a separate window on screen and may be saved and included in a Custom Report.

1. Select the **Reports** tab page on the left side of the result window.

Report Setup dialog opens.

2. Select the **Items** tab page.
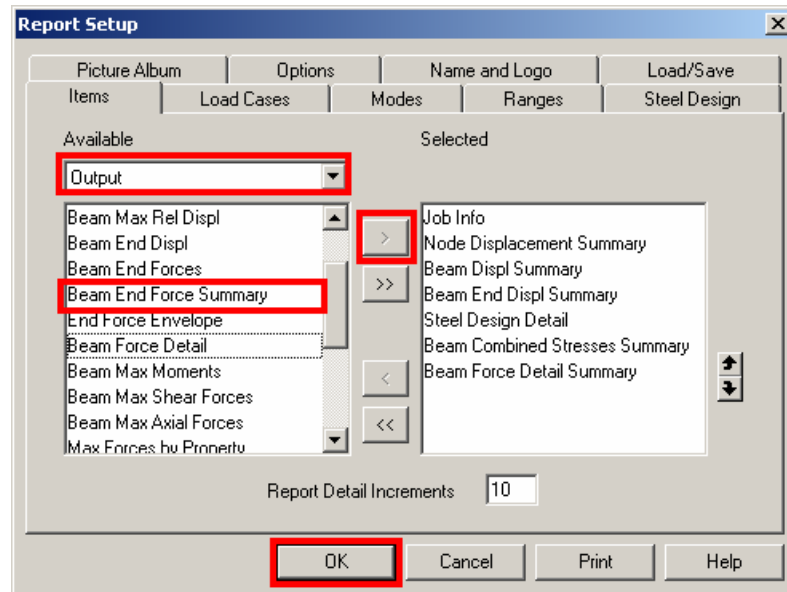
3. Select **Output** under Available section.

4. Select the results you want in the report under **Available** section and move them to the **Selected** section using the arrow button.

5. Open the **Load Cases** tab page to select the load cases, which you want to display in the report.

6. By default all load cases are selected. To remove a load case from the selected list pick the row and push the arrow button.
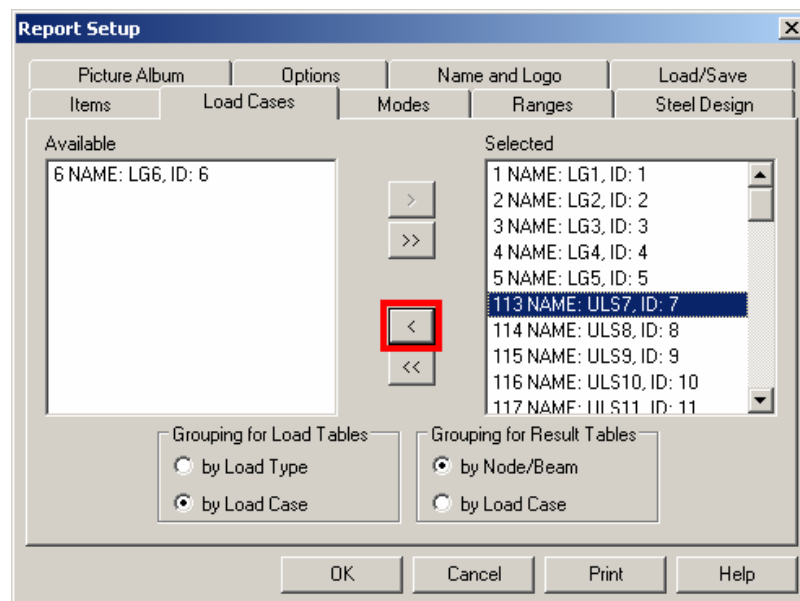
7. Then define if you want the results to be grouped by Load Case or by Node/Beam

8. To save the report settings open the **Load/Save** tab page.
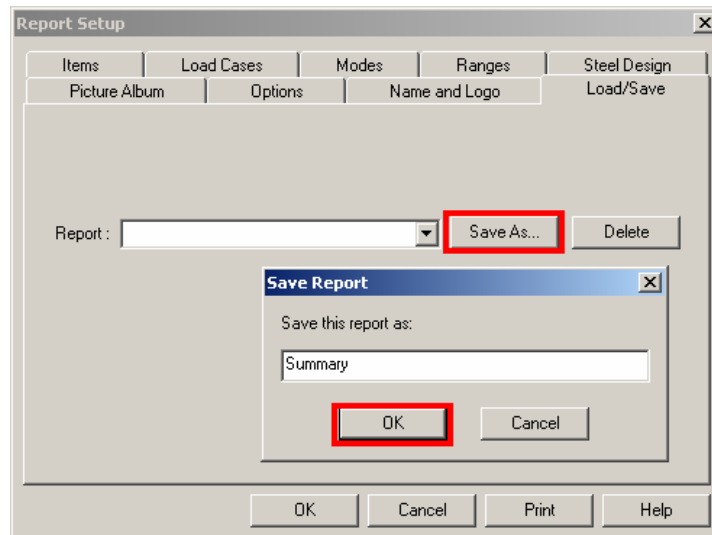
9. Push the **Save as...** button and write the name of the report into the dialog.

10. Close the **Save Report** dialog by pressing **OK**.

11. Close the **Report Setup** dialog by pressing **OK**

You can create additional reports from **Report** pull down menu. First we will create a joint displacement report.
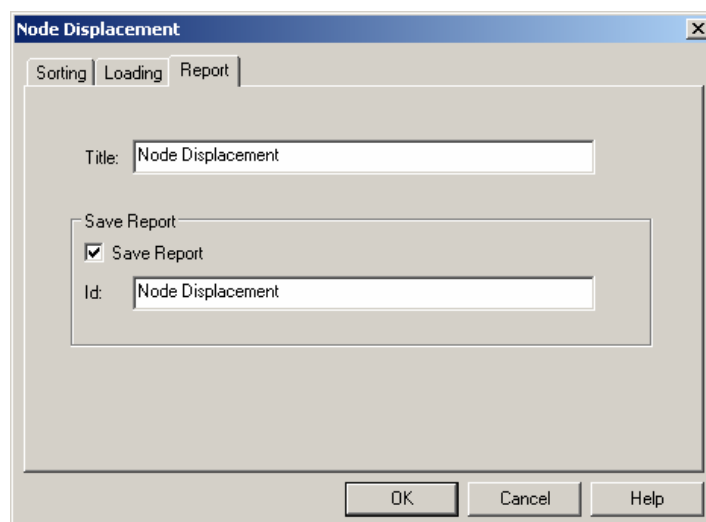
1. Select the nodes you want to the report in the model by using the **Nodes cursor**.

2. From **Report** pull down menu select **Joint displacement…**

3. In Sorting tab page select **Absolute Displacement** to sort by it.
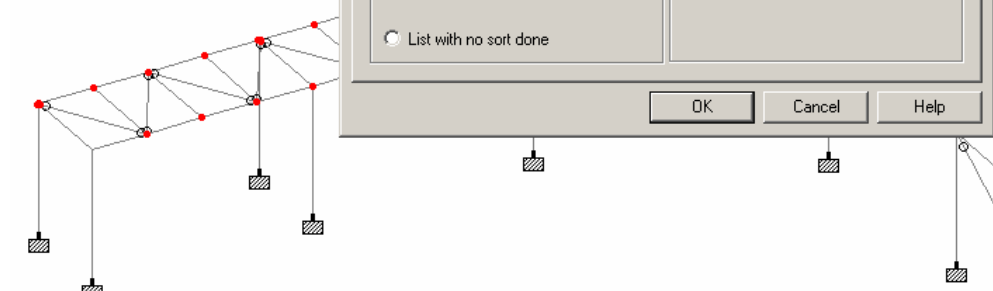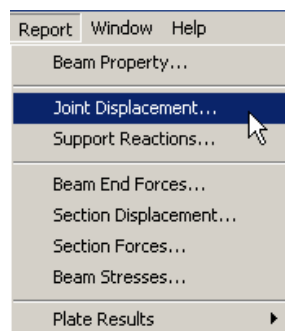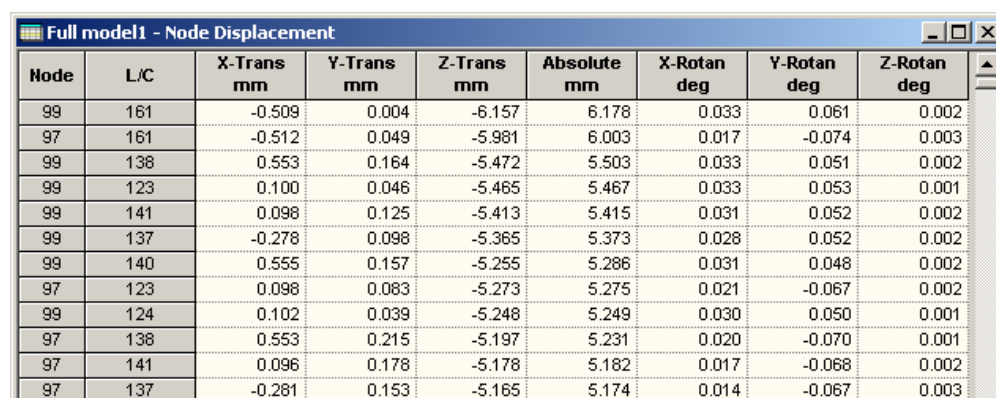
4. Select Report tab page

5. Write the name for your repot file.
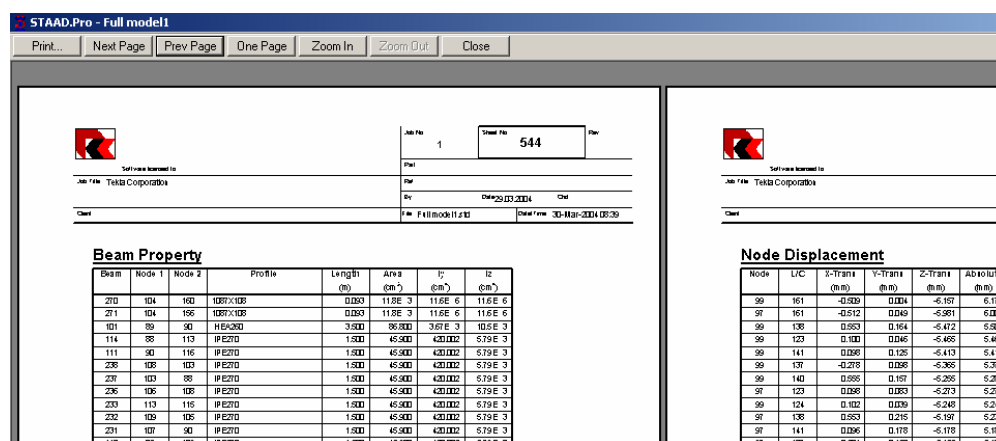
6. Tick **Save Report** to save the report.

7. Press **OK** button. The report file opens in tabular form.

To view the report file select the **Print preview report** command under **File** menu.

All existing reports are put together in one file. You can display one or two pages of the file at a time by picking the corresponding option on the toolbar. To exit the viewer, push the **Close** button.

The report file can be save as text file or MS Word file using **File > Export report** functionality.

Check Results Both Graphically and Numerically in Tekla Structures

The analysis log file can be viewed in Tekla Structures by clicking the **Display analysis log** icon, which opens the log in result window.



```
List
Report
Model name: C:\TeklaStructuresModels\hvac_magicad\TSDemoModel
Analysis model name: Full modell

++++++++++++++++++++++++++++++++++++

Initializing Analysis Component
Number of nodes generated = 163
Number. of members generated = 272
Number of primary load cases generated = 6
Number of combination load cases generated = 49
 **NOTE - NEAR INSTABILITY AT JOINT NodeId:    151   DIRECTION = MZ
  PROBABLE CAUSE WARNING-NEAR SINGULAR
  K-MATRIX DIAG=  1.3793518E+09 L-MATRIX DIAG=  8.0603057E+01 EQN NO    24
 **NOTE - NEAR INSTABILITY AT JOINT NodeId:     67   DIRECTION = MX
  PROBABLE CAUSE WARNING-NEAR SINGULAR
  K-MATRIX DIAG=  1.3793235E+09 L-MATRIX DIAG=  4.8724987E+01 EQN NO    94
 **NOTE - NEAR INSTABILITY AT JOINT NodeId:     66   DIRECTION = MZ
  PROBABLE CAUSE WARNING-NEAR SINGULAR
  K-MATRIX DIAG=  1.3793517E+09 L-MATRIX DIAG=  8.0603052E+01 EQN NO   150
*** WARNING: ABORTING EUROPEAN CODE DESIGN FOR MEMBER 141(Id: 133625)
  CHECKING SLENDER SECTION IS LIMITED TO I-SECTIONS
*** WARNING: ABORTING EUROPEAN CODE DESIGN FOR MEMBER 142(Id: 133666)
  CHECKING SLENDER SECTION IS LIMITED TO I-SECTIONS
*** WARNING: ABORTING EUROPEAN CODE DESIGN FOR MEMBER 255(Id: 133625)
  CHECKING SLENDER SECTION IS LIMITED TO I-SECTIONS
```

Utility Check   You can visualize the utility check results in Tekla Structures.

1. Select **Setup > Colors…**

2. Select **Analysis utility check** from the list.
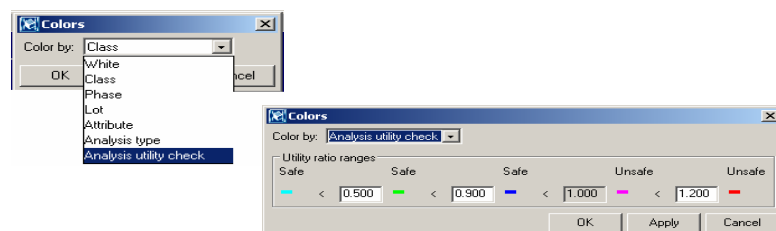
3. Press **OK** to change the member colors in the model. Note that you must have the analysis model selected in the **Analysis & Design models** dialog.
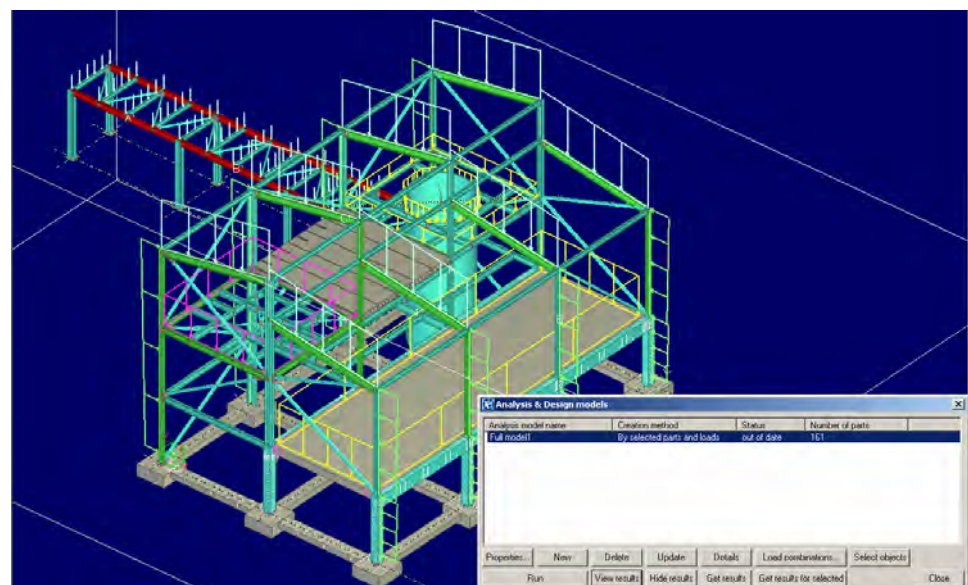
The utility check results can also be shown in reports.

1. To select the members of the analysis model and to view the analysis model structure in Tekla Structures, push **Select objects** button in the **Analysis & Design models** dialog. The analysis model frame is displayed as red bars inside the physical model.
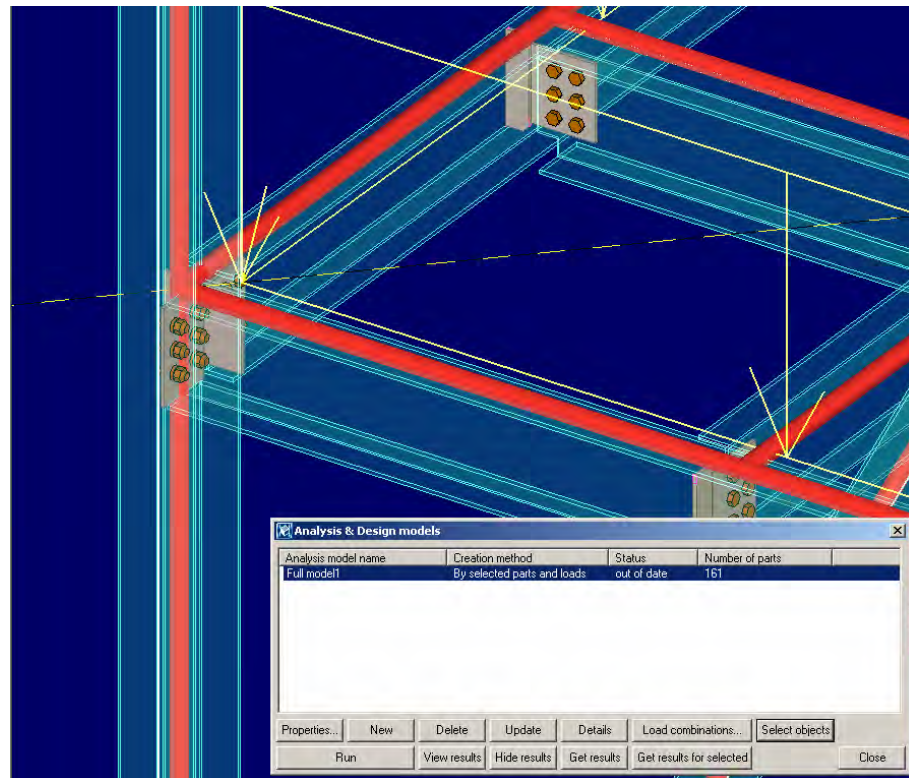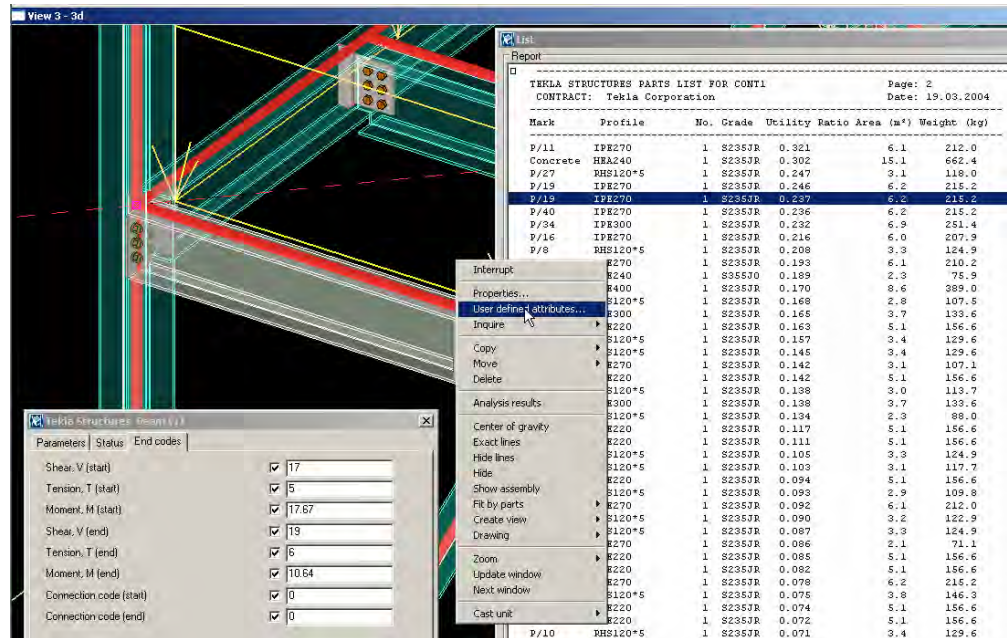

2. Push the **Report** icon

3. Select **part_utility_ratio_ID** report from the list.

4. Push the **Create from selected** button. Tekla Structures generates a report of the selected members.
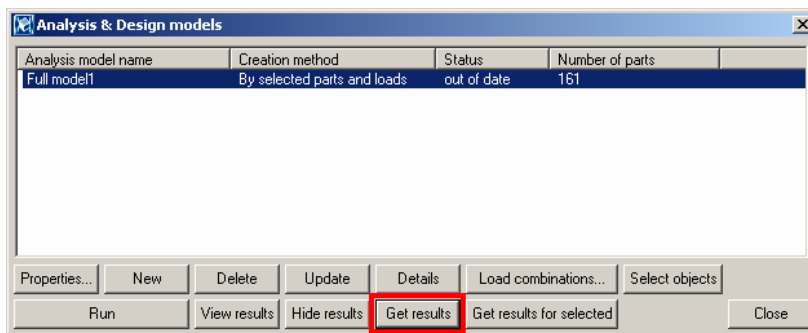

5. To select a part from the model, pick a line in the report.

6. Right click and select **User defined attributes…** from the menu.

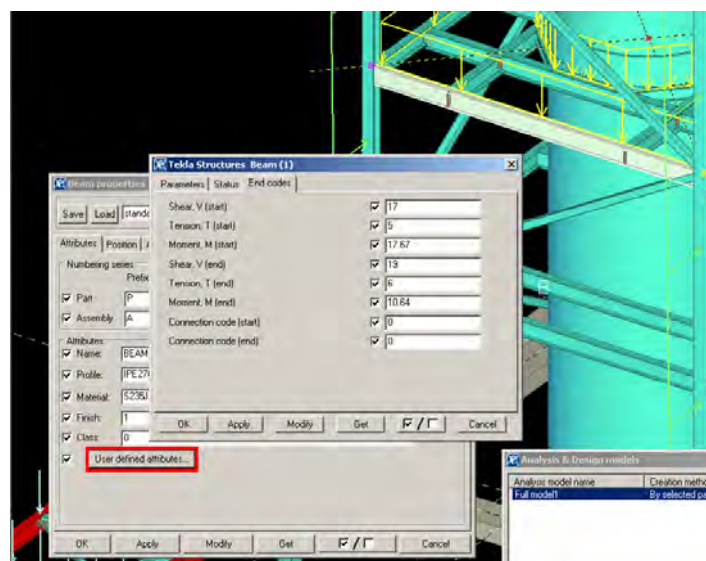7. Select **End codes** tab page to view forces of the selected part.

You can also call the analysis results for single members from STAAD*pro*2003 into Tekla Structures model.

1. Press Get results in the **Analysis & Design models** dialog.

2. Double click a beam to view the results in the Beam properties dialog.

3. Push **User defined attributes…** button.

4. Select **End codes** tab page.

# Optimize the Profiles in the Model According to Analysis Results

You can have Tekla Structures to check the profiles of the steel parts in the physical model and suggest the best profiles to use, based on the analysis results. See more in Tekla Structures Help: Analysis > Analysis and Design > Optimizing part size

Tekla Structures creates a **design group** of steel parts that have the same name and profile. Tekla Structures uses design groups when it searches for the optimal profiles for parts. It assigns the same profile to all parts in a design group.

1. To optimize part sizes, run the analysis and then click **Analysis > Optimize**....

2. The STAAD*pro*2003 runs the steel design and writes a Full model1_opt.anl file

3. Press the **Done** button on the dialog to close it. Optimization results dialog opens.
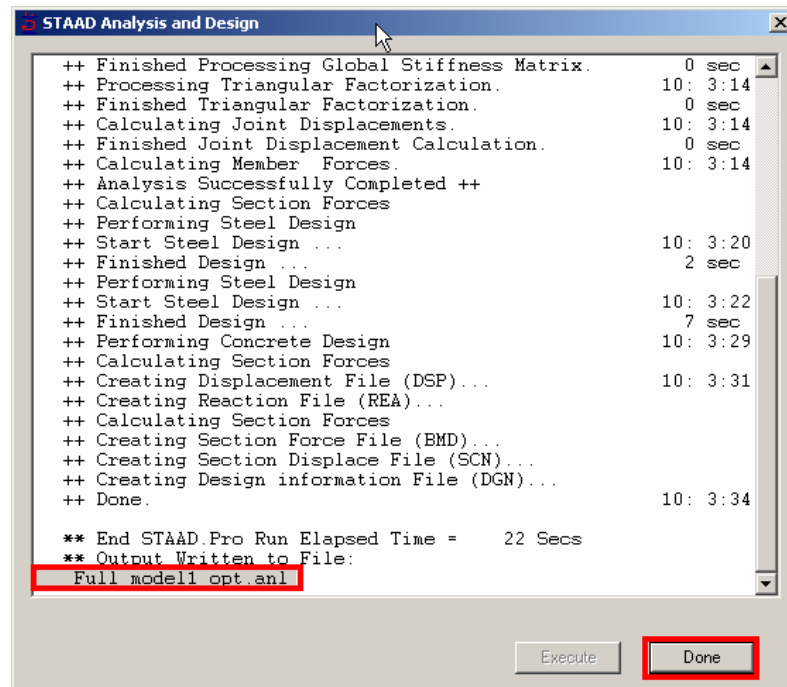
4. Select the BEAM-IPE270 design group on the dialog. All members, which have the same profile are selected in the model.
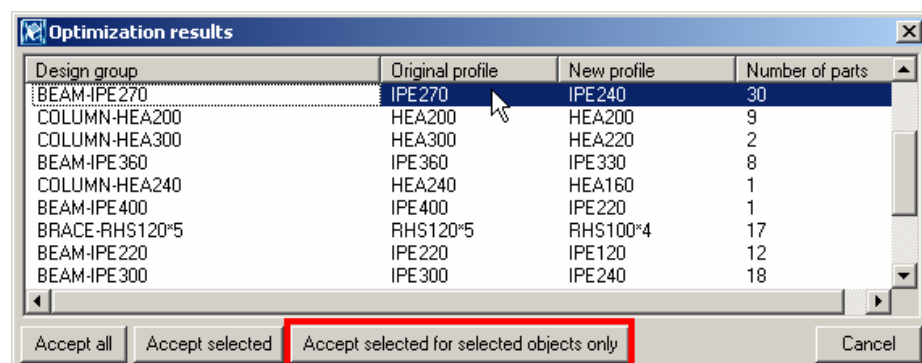

5. Using **Ctrl** – button, unselect the members, which you **DON'T** want to change. Leave only beams on **+7000 level** selected.

6. On Optimization results dialog, press **Accept selected for selected objects only** button. The selected profiles are changed.
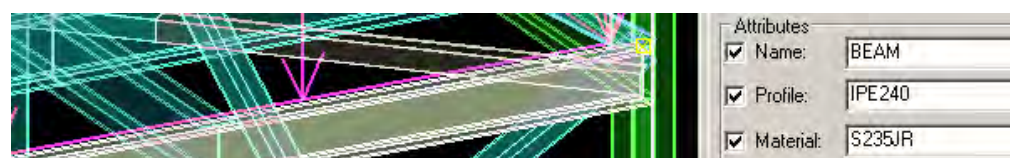
When the optimization is done, the analysis model needs to be updated. If you want to keep the old results select the analysis model on the Analysis & Design models dialog and click **Analysis > Freeze** results.

1. On Analysis & Design models dialog press **Update**.
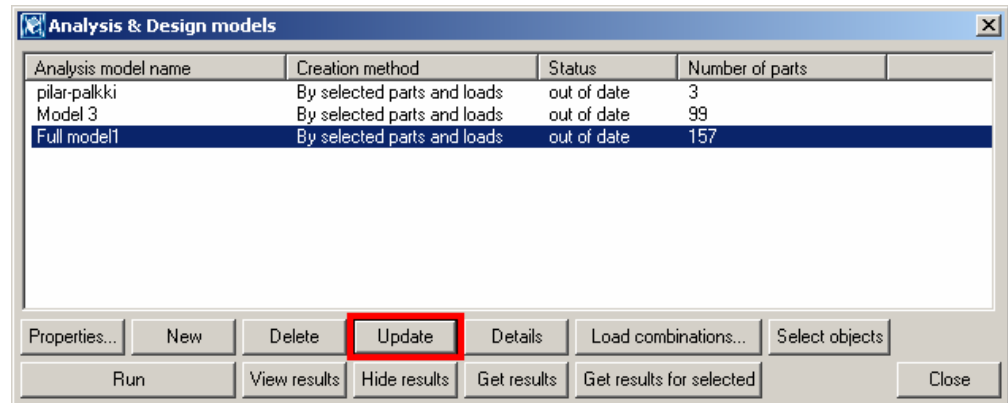
2. Then re-run the analysis by pressing the **Run** button.

3. Press **View results** button to open STAAD*pro*2003

4. Check the new results both in STAAD*pro*2003 and in Tekla Structures.