STABLAB 2013 USER MANUAL

CONTENT

1 GENERAL DESCRIPTION

1.1 INSTALLING THE SOFTWARE

In the followings we give those hardware and software requirements the *StabLab* software cannot be run without, or their lack may result in a slow operation:

System requirements:

- Processor: Intel® Core™ i5 CPU 2+ GHz or equivalent AMD or better
- Operating-system: 32-bit Microsoft Windows XP, 32-bit and 64-bit Windows Vista ,32-bit and 64-bit Windows 7, 32-bit and 64-bit Windows 8
- Memory: 2GB RAM minimum, 4GB or greater recommended
- HDD: 100MB of available hard-disk space for installation
- Video-card: 256MB of video memory or greater
- Constant Internet connection

You will need to have administrative or power user rights for the installation. Without administrative rights you cannot install the hard lock driver and those **.dll** files, which are indispensable for the system. You will have to set the language of setup. The StabLab will use this language for the first time you start, but you may switch to another language later. During installation the instructions of the setup shall be followed step by step. The installing program will copy the elements of the software into the Directory you have specified, then it will position the Menu of the program onto the selected place of the „START”-menu. Finally the install shield will place the starting-icon onto the desktop. The last step of the setup is the hard lock driver installation. This has no progress signal, so you will have to wait while it finishes.

As a default both 32 and 64 bit Windows operating systems are able to provide 2 GB RAM for the 32 bit applications, like StabLab is today. When the computer has more than 2 GB RAM physically, than with the following changes in the configuration of the boot.ini file of the Windows operating system approximately 3 GB will be accessible for StabLab.

Change the configuration for Windows Vista and Windows 7 operating systems:

To extend the available memory run the following from the command prompt:

bcdedit /set increaseuserva 3072

to switch back, run the following from the command prompt:

bcdedit /deletevalue increaseuserva

The configuration steps for the following operation systems:

- Windows XP Professional
- Windows Server 2003
- Windows Server 2003, Enterprise Edition
- Windows Server 2003, Datacenter Edition
- Windows 2000 Advanced Server
- Windows 2000 Datacenter Server
- Windows NT Server 4.0, Enterprise Edition:

backup the original boot.ini file and then put the ***"/3GB"*** switch in your **boot.ini**!

here is the sample boot.ini file with 3GB switch.

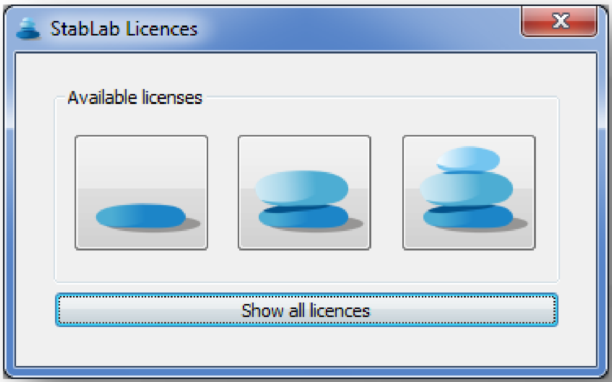
[boot loader]
timeout=30
default=multi(0)disk(0)rdisk(0)partition(2)\WINNT
[operating systems]
multi(0)disk(0)rdisk(0)partition(2)\WINNT="????"
/3GB

The "????" in the above line represents the name of the operating system, you have to add the "/3GB" text at the end of that line.

1.2 Login process

After starting *StabLab* login dialog will be appeared. To login the username or email and password have to be entered, which were given during the registration process on the www.stablab.net website.

After successful login the process the StabLab License dialog will appear.



The three big buttons show the available *StabLab* licenses. Click on one of the available buttons and the selected license will be started.

If Show all licenses button is clicked all the available licenses will be appeared in a table. The first and the second column of the table show the version of the licenses; third column shows username of the user who checked out the license; in the action column two types of button can be appeared: with button license can be checked out, with button license can be released.

After the license selection the main window of StabLab start automatically.

1.2.1 Web protection status

Icon on the left corner of the status bar shows the status of the web connection. Color of the icon means the following:

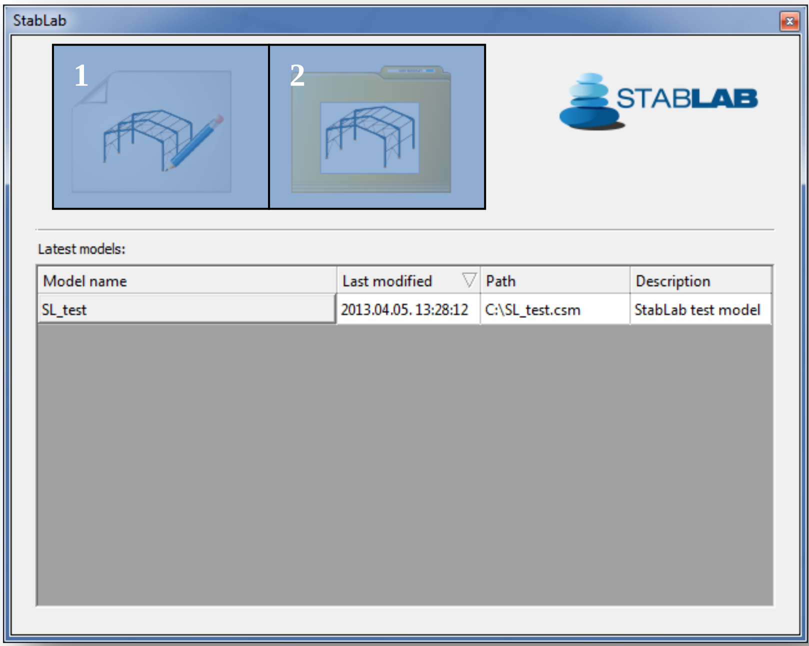
- connection is fine. StabLab connected to the license server properly
- connection is pending. Connection is terminated recently. In case the connection is not restored within two minutes, the status turns to lost connection
- connection is lost. All of the modeling and analysis functions are locked, but the model can be saved

1.3 The main window

The main window consists of six separate parts containing different functionalities. The graphical window is the area for the 3D structural modeling; the menu contains some important commands; the tabs from left to right lead the engineer through the steps of structural analysis; the side bar contains functions of grids, views, most commonly used transformations and selections; the status bar makes the drawing phases easy and the object and parameter tables at the right gives always sophisticated information about the model, making fast modifications possible.

1.3.1 The startup window

After login there is a startup dialog window which allows creating and opening models easily. The latest models can be opened without browsing folders. They are sorted by the last modification date as a default setting but it can also be sorted by name or by model path.



The first big icon (#1) is for creating a new model, the second (#2) is for open model from folder.

1.3.2 The graphical window

The structural model appears always in the graphical window. There are no other window opening options; however there are lots of viewing possibilities in this single window. The graphical window helps the modeling by the global coordinate system (*GCS*) and a moveable, rotatable and size adjustable grid, which is the main area for drawing. The coordinate system at the left bottom corner denotes always the unchangeable *GCS*; the origin of the user coordinate system (*UCS*) takes place at the middle point of the grid which is always the plane “XY” of the *UCS*. The following moving possibilities and quick view settings should be used during the structural model manipulation:

- **Move:**
  push and hold down the middle mouse button or use the four arrow buttons on the keyboard to move the model on the screen
- **Rotate:**
  hold down the ***Alt*** key and the left mouse button. The center of rotation is always adapting the actual model view

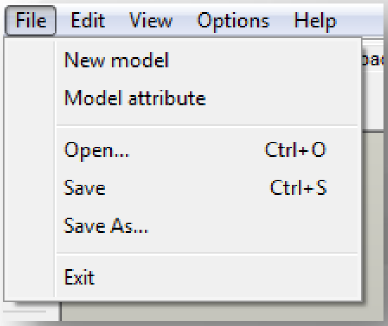
- **Scaling:**
  spin the middle mouse button forth and back or use the + and - buttons on the keyboard or hold down the ***ALT*** key and the right mouse button
- **Zoom window:**
  window drawing by the left mouse button, while pressing ***SHIFT* + *ALT*** keys
- **Hotkeys for vie:**
  - **Ctrl+1**: Switch to **top view**
  - **Ctrl+2**: Switch to **front view**
  - **Ctrl+3**: Switch to **side view**
  - **Ctrl+4**: Switch to **axonometric view**
  - **Ctrl+5**: Perpendicular to actual raster plane
  - **Ctrl+0**: Quick **zoom all**

1.3.3 THE MENU

In *StabLab* the menu does not contain many commands since the main functionality is placed to the structured tabs and side bars and the applicability and modeling efficiency does not really need to duplicate the functions. However five important function groups appear here:

the ***FILE*** handling, some ***EDIT*** options (undo-redo), ***VIEW*** and diagnostics, ***OPTIONS*** for settings (for saving, updating, selecting language, and for model diagnostics) and ***HELP***.

In ***FILE*** menu:

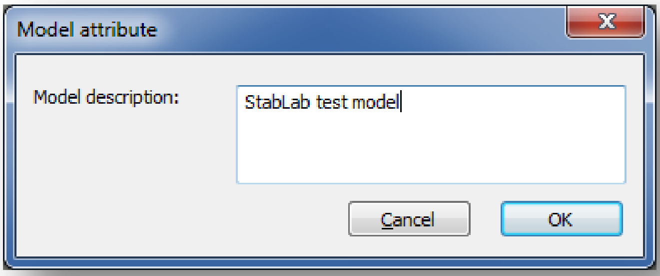


New model

New model can be created.

Model attribute

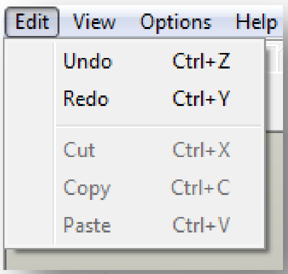
Model description can be changed. Model description can be entered at first by creating a new model.



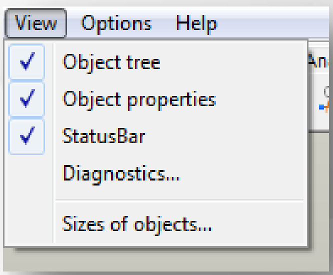
Open, Save, Save As

The functions are according to their names.

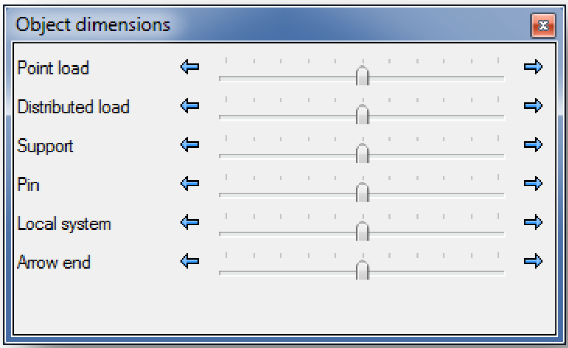
In ***EDIT*** menu the common functions can be found:



In ***VIEW*** menu:

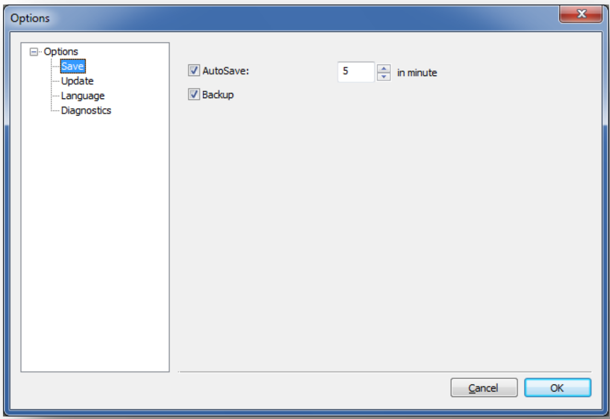


With the first four functions the visibility of the dockable windows can be switched. With the ***SIZES OF OBJECTS…*** function, the size of the objects can be changed.



In ***OPTIONS*** menu, the following settings can be found: ***SAVE***, ***UPDATE***, ***LANGUAGE***, and ***DIAGNOSTICS***.

Save



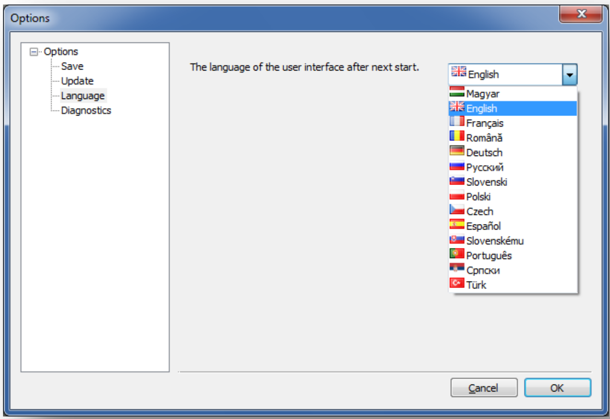
If **Autosave** is clicked, the program automatically performs a save periodically in accordance with the adjusted number of minutes. If **backup** is clicked, StabLab creates a backup file after manual save is performed. The backup model file can be used as a normal model by removing the **.bak** extension.

Update

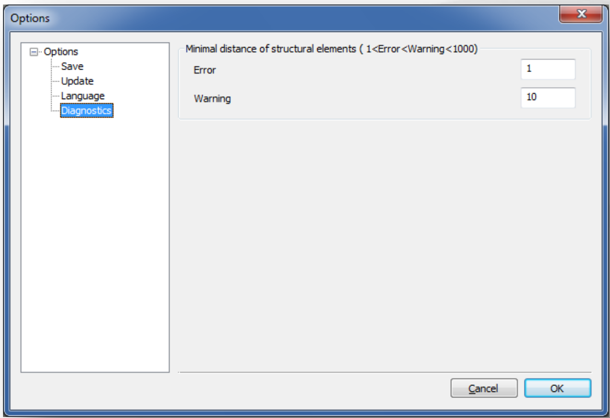
StabLab looks for an available new version on the web at every startup. It can be turned off. The check for a new version can be performed manually by clicking on the **Search for updates now icon**.

Language

The language of the user interface can be set here. StabLab has to be restarted after changing.



Diagnostics

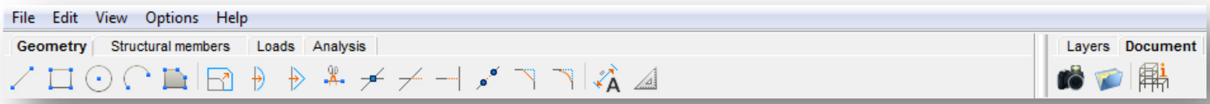


Before the analysis, the program checks the possible modeling mistakes.

There are two different levels to worn the user for the possible modeling mistakes:

- **Error:** In case of the distance between the endpoints of two members is more than 0 but less than the adjusted value, the program sends an error message, and shows with red sign the relevant members in the diagnostics window and not perform the analysis.
- **Warning:** In case of the distance between the endpoints of two members is more than the defined error level distance, but less than the adjusted value here, the program sends a warning message, and shows with yellow sign the relevant members in the diagnostics window.

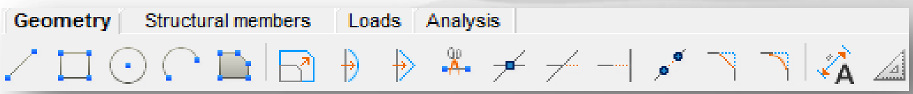
1.3.4 The tabs



The tabs contain the systematically collected functions of modeling and analysis, leading the engineers through the logical steps of the structural analysis.

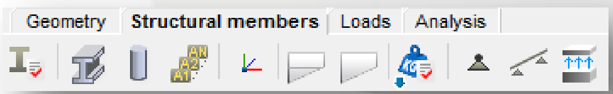
By approaching any of the icons with the cursor, the short name of the icon will be appeared.

Geometry tab



Contains all the important CAD drawing, modification functions, dimensioning and measuring tools.

Structural members tab

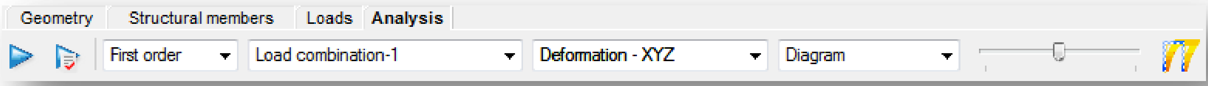


The functions related to cross-sections, structural columns, beams, supports and link elements are collected on this tab.

LOADS TAB

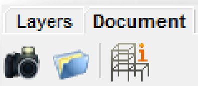
All types of loading including load cases, load groups, load combinations and unique loads placed on the structure can be created on the ***LOADS*** tab.

ANALYSIS TAB



The structural analysis types can be set and executed, the results can be viewed and labeled in various forms.

DOCUMENT TAB



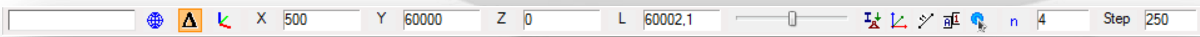
The functions on the ***DOCUMENT*** tab allow of the creation of the snapshot, and contain a model information tool.

1.3.5 The side bar

- *Save model*
- *Undo/Redo*
- *Settings for the coordinate system, and snapping grids*
- *Top view*
- *Front view*
- *Side view*
- *Isometric view*
- *Perpendicular to raster view*
- *Line view visualization of the model*
- *Wireframe visualization of the model*
- *Hidden line view visualization of the model*
- *Solid view visualization of the model*
- *Move point and edge*
- *Move/Copy the selected objects*
- *Mirror the selected objects*
- *Rotate the selected objects*
- *Select all objects*
- *Deselect all objects*
- *Invert selection*
- *Select by properties*
- *Fit view (Ctrl+0)*

The side bar contains the mostly used functions for modeling.

1.3.6 The status bar



The first field is a progress bar, shows the progress of the actions.

The next 3 icons on the status bar are for specifying the interpretation of the coordinates during drawing/modeling actions, while values are entered manually after pressing the appropriate letter (“X, Y, Z, L” for coordinate axis or length into a direction, or “a, b, L” for polar coordinates). The user could influence the interpretation of the manually entered values with the following settings:

Global	**User**
Absolute	**Relative**
Descartes	**Polar**

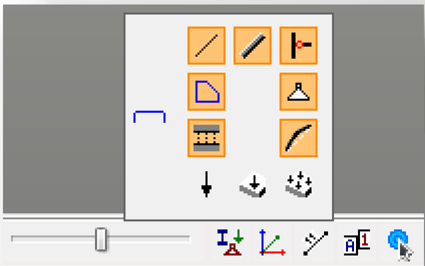
By moving or clicking the slider with the right mouse button, the size-representation of the objects can be changed. The next four icons allow the sophisticated visibility adjustment. Approaching any of these icons with the cursor a group of graphic symbols will appear ordered into a matrix shape.

Clicking the first icon on each of these four visibility setting matrix the scope of the settings will be changed between global (valid for all tabs), or valid only for the current tab .

Setting this option on any of the 4 visibility matrix, the selected scope will change on all the other three accordingly.

Changing this scope will also change the visibility between the previously adjusted global and the current tab visibility settings.

Visibility options of graphic symbols



Visibility of lines, created with line, circle and arc function of Geometry menu.

Visibility of the structural members created with beam or column function in Structural members menu. Switching these members non visible their centerlines might still be visible accordingly to the status of line visibility settings .

Visibility of pins (end releases).

Visibility of the 2 dimensional shapes. Switching these 2 dimensional shapes non visible their surrounding lines might still be visible accordingly to the status of line visibility settings .

Visibility of supports.

Visibility of link elements.

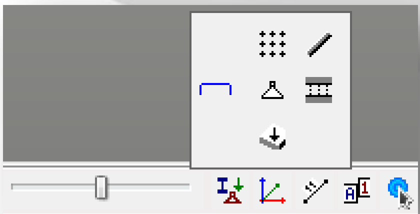
Visibility of initial bow imperfection.

Visibility of point, line and surface loads.

Visibility of load transfer surfaces. Switching these load transfer surfaces non visible their surrounding lines might still be visible accordingly to the setting of visibility of the lines .

Visibility of distributed surface loads.

Visibility options of the grid and the local coordinate system



Visibility of the grid

Visibility of the axes of the local coordinate systems of structural members (beams and columns)

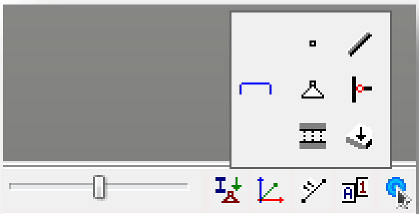
Visibility of the axes of local coordinate systems of supports.

Visibility of the axes of local coordinate systems of link elements.

Visibility of the axes of the local coordinate systems of the load transfer surfaces.

When the visibility of the object is off, the axes of the object also will not be visible!

Visibility options of object names



Visibility of the number of finite elements. (Visible only on the **Analysis** tab!)

Visibility of the name of bar elements.

Visibility of the name of supports.

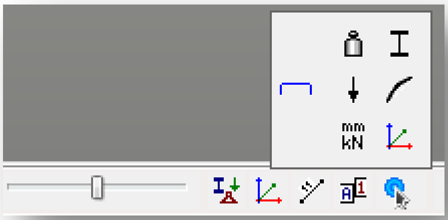
Visibility of the name of pins (end releases).

Visibility of the name of link elements.

Visibility of the name of the load transfer surfaces.

When the visibility of the object is off, the name of the object also will not be visible!

Visibility options of labels



Show material grades.

Show shape names.

Show the load intensity.

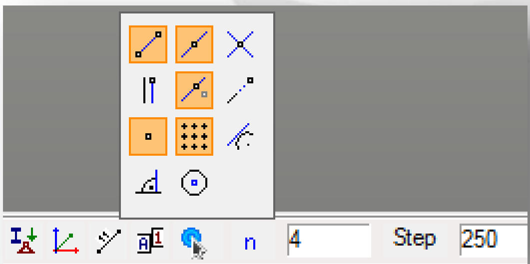
Show initial bow imperfection.

Show the units of quantities.

Show labels of the local coordinate axes.

Action point sets

The **Action point sets** offer a wide range of setting the point snapping functions.



Snapping the end points of graphical (lines, arcs) and structural (beams, columns) objects

Switch divide function ON / OFF

- ON: In case of Divide snap mode is ON, a new field will appear on the status bar. There are 3 options for the divide snap point specification. The user can select by clicking on the icon before the numeric field:

- % 25, clicking the n icon, the percentage will be calculated to a length of the approached element, and from the approached end this length will be measured by the snapping points. Usually there is a rest distance at the end of the element
- d 1000, clicking to the % icon, a distance can be specified. This length will be measured by the snapping points from the approached end of the object. Usually there is a rest distance at the end of the element
- n 4, clicking to the d icon, the number of division can be specified. This snapping points show the endpoint of the subdivided length. There is no rest distance at the end of the element
- OFF: The field of division will disappear from the bottom status bar

Snapping the intersection points of graphical (lines, circles, arcs) and structural (beams, columns) objects

Snapping the parallel point to a linear object

Snapping the nearest element point to an object

Snapping the lengthened point of a linear object. The system is showing the actual distance of the snapping point in [mm] from the endpoint of a linear element.

Switches On/OFF the snap point of the raster.

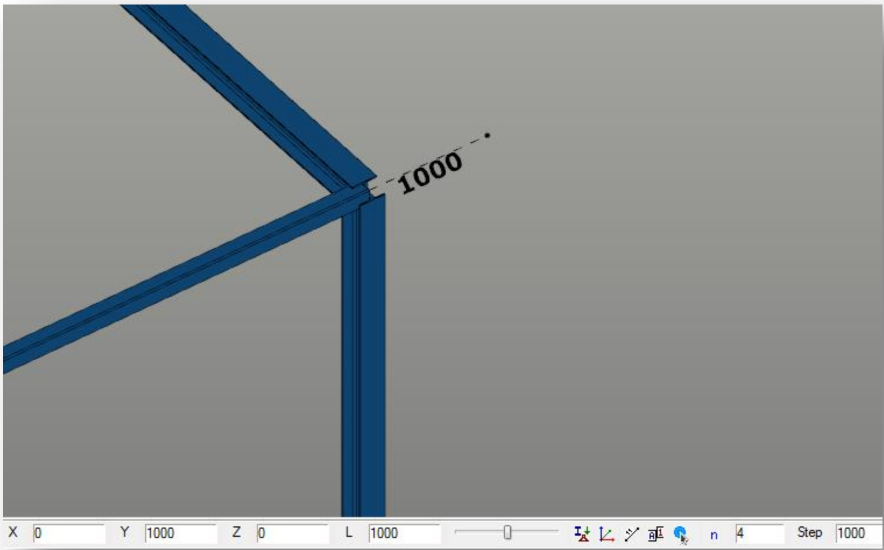
Snapping point to a tangent of an Arc / Circle from one point

Snapping point to a perpendicular from one point to an object (Line or Arc/Circle)

Snapping to Center point of Arc / Circle.

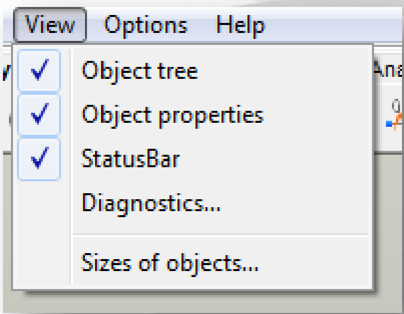
For snapping the center of linear elements, you have to use the appropriate settings of snap divided points, see above!

The last field of the Status bar is the STEP field. Here the given number in mm is the snapping distance towards the length direction of line and bar elements, when the lengthening snapping point is ON.



1.3.7 The windows of object tree, diagnostics results and object properties

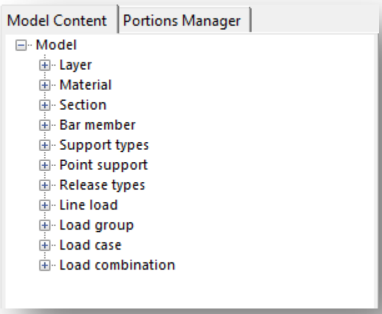
The visibility of these windows can be switched ON / OFF in the **VIEW** menu.



Object tree window

Object window has two tabs:

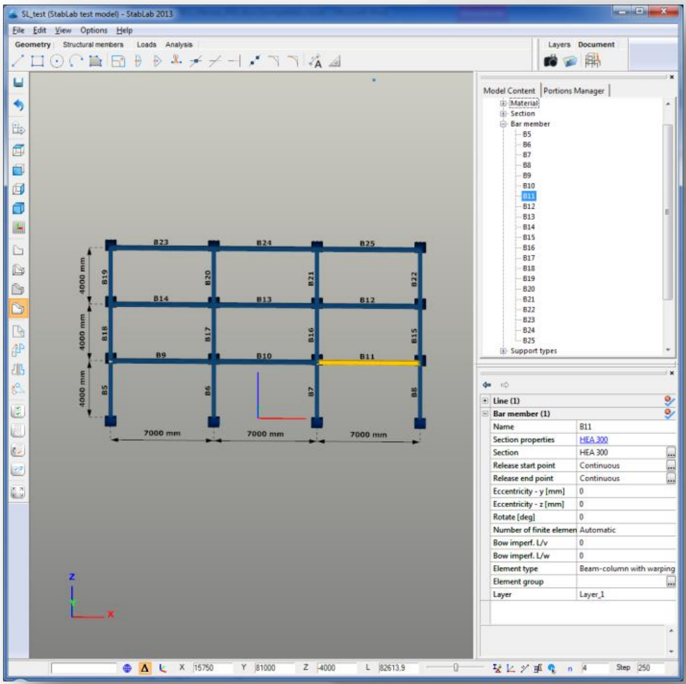
MODEL CONTENT TAB



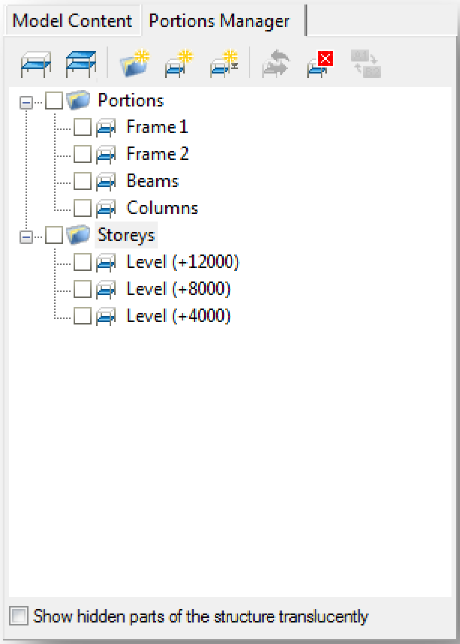
By defaults this tab contains all the predefined basic object types which are necessary for building up a structure.

While modeling, each new object (materials, sections, loads, members, supports etc.) will appear in the tree object structure in the appropriate group.

Selecting any of the objects, it will be highlighted (selected) in the model space. Multiple selections of objects are possible in this tree.



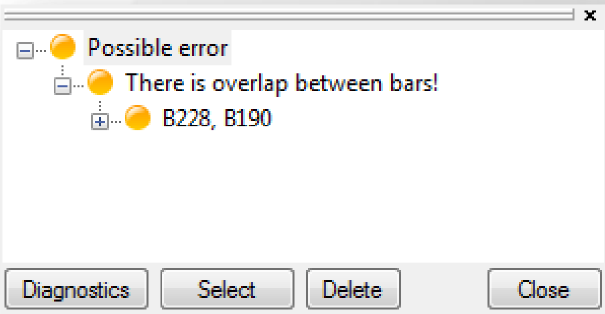
Portions Manager tab



Please find the description of this tab in the ***3.2 Model views***, and in the ***3.4 Portions manager*** chapters below!

Diagnostic window

In case of a geometrical or modeling mistakes (like overlapping two or more object, missing supports, etc..) are detected, the Diagnostic windows visibility will switch ON automatically, showing the name of problematic objects in a tree structure.



The first type of diagnostic results is the error messages appearing in red color.

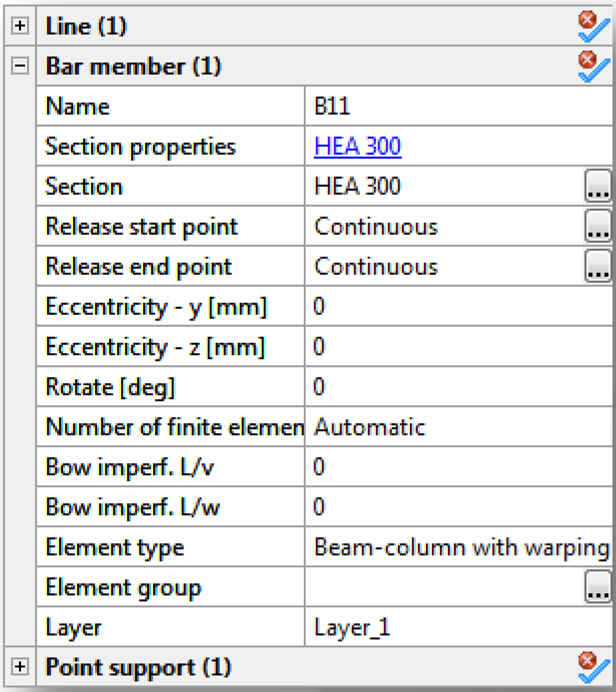
The second type of the diagnostic result is the possible errors in yellow color.

By clicking on any of the object name in tree structure, and pressing the ***SELECT*** button, the selected object will be highlighted in the model space. You can select more objects at the same time clicking their name while pressing the ***SHIFT*** or ***CTRL*** key.

The selected object can be erased by pressing ***DELETE*** button at the bottom of the Diagnostic window, or using the ***DELETE*** key on the keyboard.

Object properties window

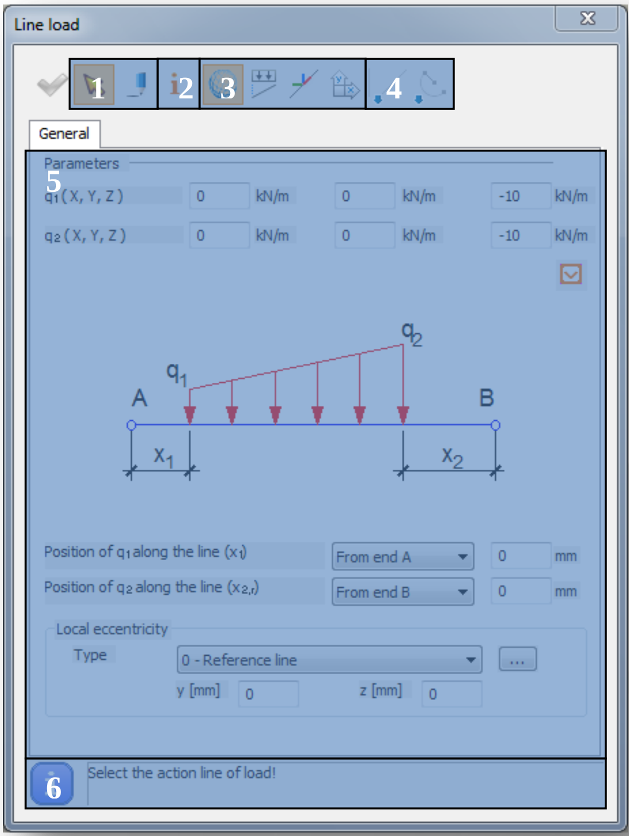
Selecting one object in the model space, all the relevant properties will appear in this window. The values of the parameters can be overviewed, and in most cases these parameters can be changed.



In the lower table all the selected objects (and sub-object) appear, and after expanding one object, all the parameters of it can be seen, and modified if possible. In case of multiple objects selection only the identical parameters appear, however the different ones can also be changed to identical.

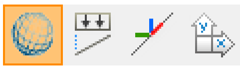
1.4 General structure of dialogue windows

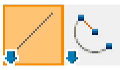
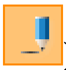
Several dialogue windows – especially the most important ones on the ***Structural members*** and ***Loads*** tabs – show same structure in order to make easy the orientation in the labyrinth of tools and functions. The usual parts of these dialogues are described below:



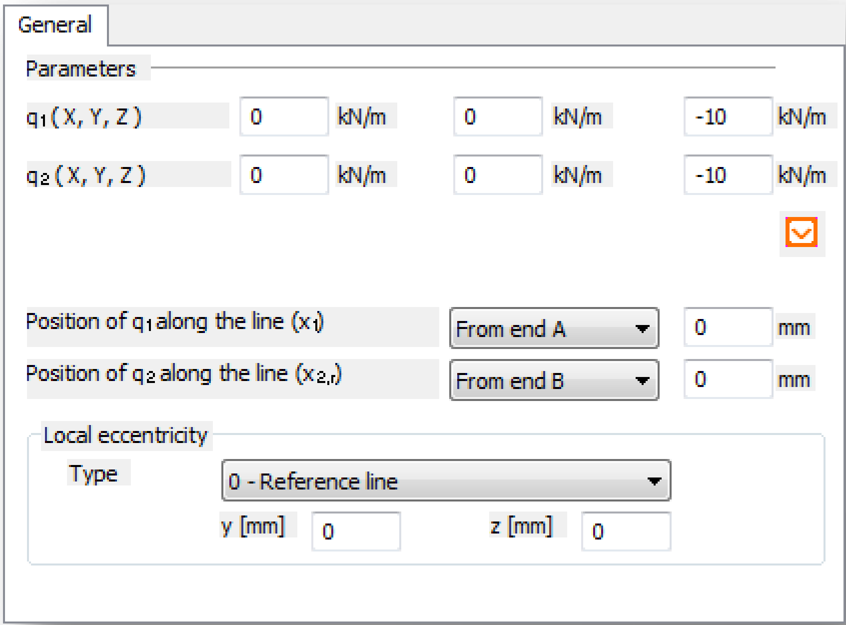
1.  the method of placement (assign to an element or draw from one point to another)

2.  extract data from a previously placed object. All the parameters are set to the same as for the selected object

3.  coordinate systems in which the directions are considered

4.  additional drawing functions if the placement is by drawing ()

The main parameters of the object (#5) are placed in the middle part of the dialog.



The information field (#6) gives always information about the next required step of the placement.

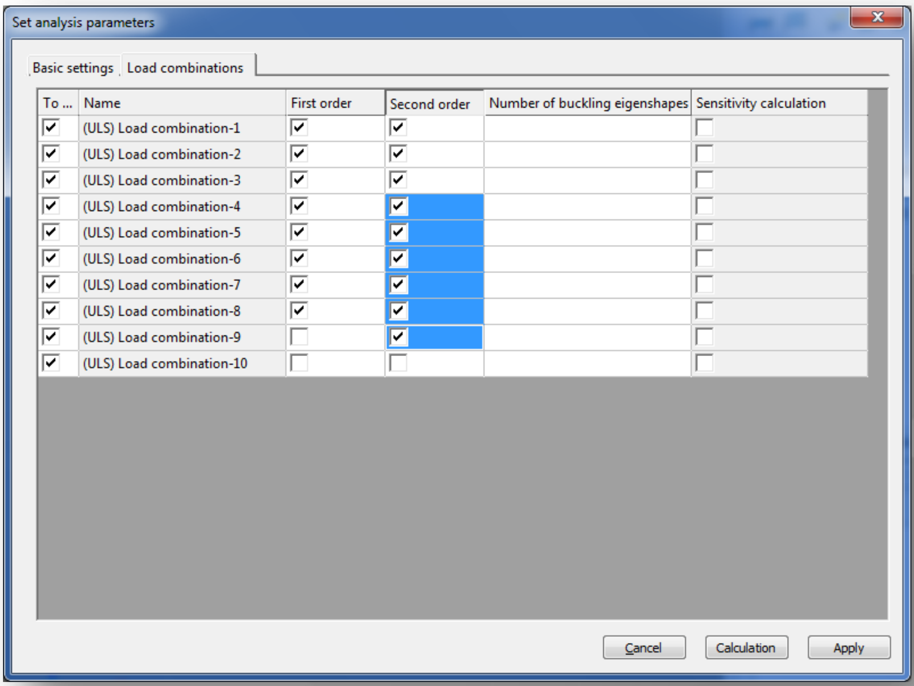
Every icon on these dialogues has a tooltip with the name of the function on it. The text will appear when the mouse approaches the icon

1.5 General functions for tables

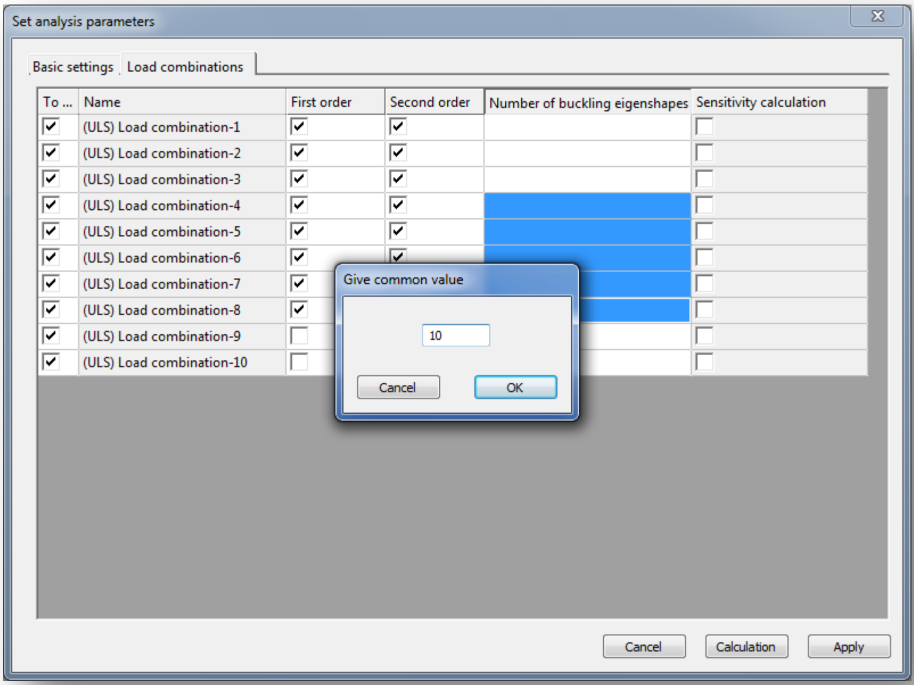
Basically there are two types of tables in *StabLab*: the input tables and the output tables. Since tables are usually used for working with large amount of data, both table types have special features making the data managing more comfortable.

Input tables (used in the ***Load combinations*** and ***Analysis parameters*** dialogues) have the following common features:

- clicking on the heading cells the whole column below will be selected
- if the there is a checking option in the table the multiple checking is possible for the selected cells clicking on the right mouse button



- if the there is value entry option in the table cells multiple value entry is possible for the selected cells clicking on the right mouse button and defining the common value



1.6 HOT KEYS

The hot keys can significantly speed up the modeling work. In *StabLab* the following hot keys can be used:

- **CTRL+N: new model**
- **CTRL+O: open model**
- **CTRL+S: save model**
- **CTRL+Z: undo last action**
- **CTRL+Y: redo**
- **CTRL+A: select all**
- **CTRL+I: inverse selection**
- **CTRL+1: switch to XY view**
- **CTRL+2: switch to XZ view**
- **CTRL+3: switch to YZ view**
- **CTRL+4: switch to XYZ view**

- **CTRL+5: switches the view perpendicular to the raster**
- **ESC: unselect all or terminate (or abort) the last action**
- **Delete: delete selected objects**
- **F1: open Help system**
- **X: manual definition of X coordinate value**
- **Y: manual definition of Y coordinate value**
- **Z: manual definition of Z coordinate value**
- **a: manual definition of alpha polar coordinate value**
- **b: manual definition of beta polar coordinate value**
- **L: manual definition of length from the previous point in a defined direction**
- **R: switch between the global and last defined user coordinate system**
- **Middle mouse button: move model**
- **ALT+left mouse button: rotate model**
- **ALT+right mouse button: zooming model**
- **↑: move model up**
- **↓: move model down**
- **→: move model right**
- **←: move model left**
- **middle mouse button: scale model**
- **+: scale up model**
- **-: scale down model**
- **SHIFT+left mouse button: unselect**
- **SHIFT+ALT+left mouse button: window scale**

2 File handling

2.1 Basics

The file handling in *StabLab* follows the same usual and simple way of the MS Windows standard. The saved files contains all information about the model, it can be relocated to other folder and opened. Normally the **.slm** file extension is associated with the *StabLab*, these files can be opened by double click.

2.2 File types

The following file types are handled in *StabLab*:

.slm: the StabLabModel file, native binary file type. It can be opened with or without results. The result file is saved to a separate file with **.slr** (StabLabResult) extension, but this file cannot be opened solely. Open and save.

.slm~: the StabLabModel file created by autosave functionality. AutoSave settings can be edited in the ***Options*** menu.

.slm.bak: the StabLabModel backup file. Backup save settings can be edited in the ***Options*** menu. Backup file is created at every manual save and stores the previous saved version of the model. If necessary **.bak** extension can be deleted and backup model used as a normal model.

The following file types can be opened in StabLab:

.anf: StruCad text file type.

.csf: old *ConSteel* version text file type (ConSteel 2.x, 3.x).

.csm: *ConSteel* binary file type.

.asc: *Tekla Structures* ASCII file.

.dxf: *AutoCad* text file type.

.sc1: *BoCad* text file type.

.snf: *StruCad* text file type.

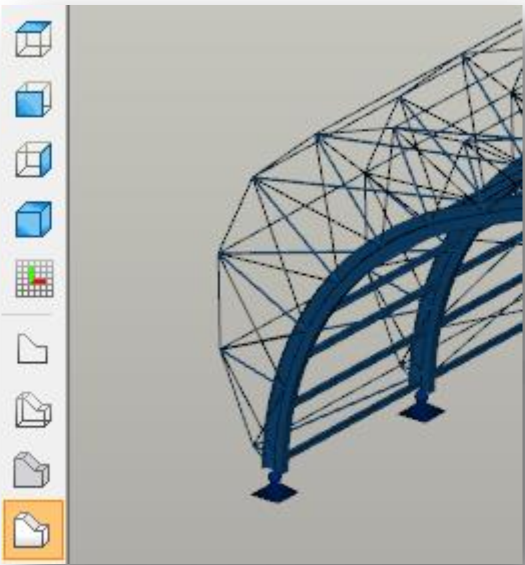
3 Model view

3.1 Basics

The transparent and multipurpose model views are very important to the engineer, since these views provide the first graphical model survey opportunity. Accordingly the first and most important objective of the model views is to feedback the user about the objects placed, yield a visual proof that the right objects are on the right place.

3.2 Model views

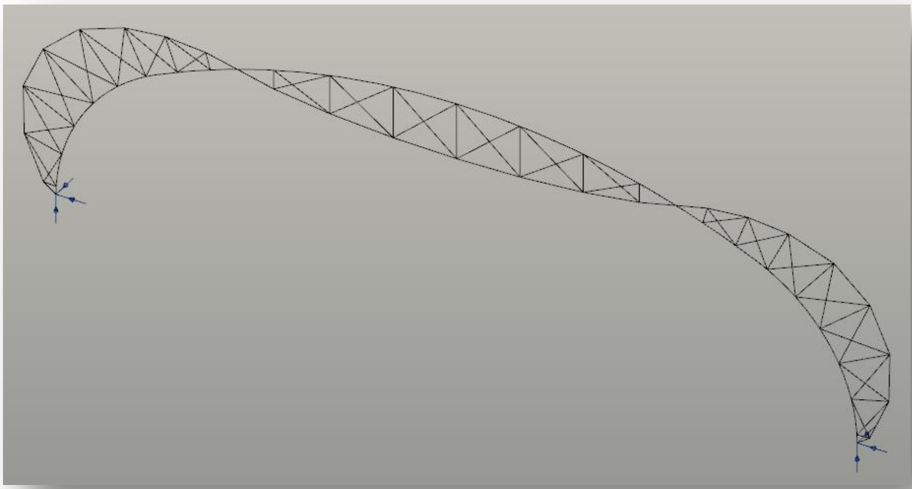
The model viewing options can be found on the left side bar.



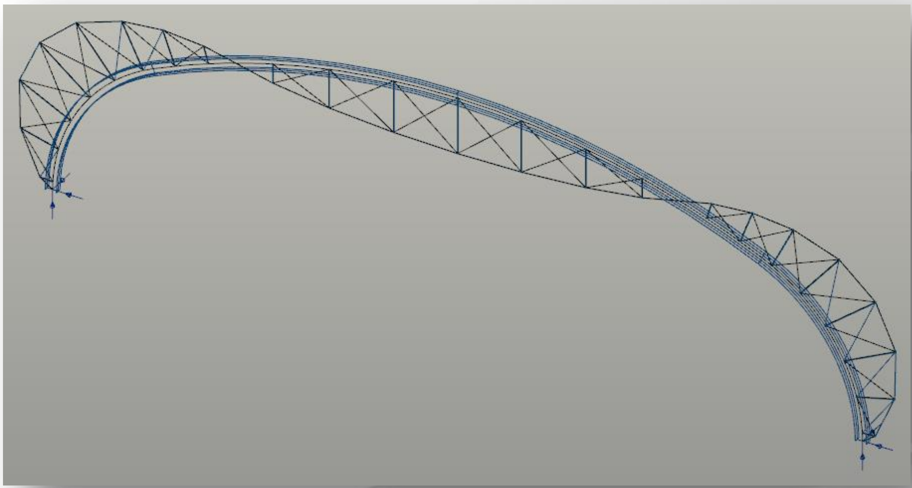
Beyond the usual model views (top view, front view, side view, axonometric view, perpendicular to raster view) there are fore visualizing options for the objects used:

- **line view**  : the simplest model view, the bars are represented by a single line, the surfaces by a two-dimensional figure without thickness, the supports are line types.

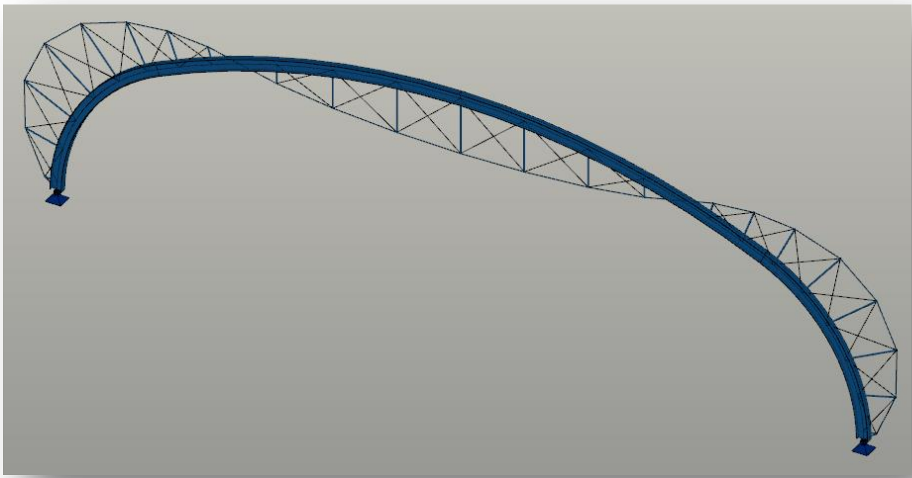
Recommended to use in the model building phases, since the clear visualization of the member snap points making the placement of supports and loads easier.



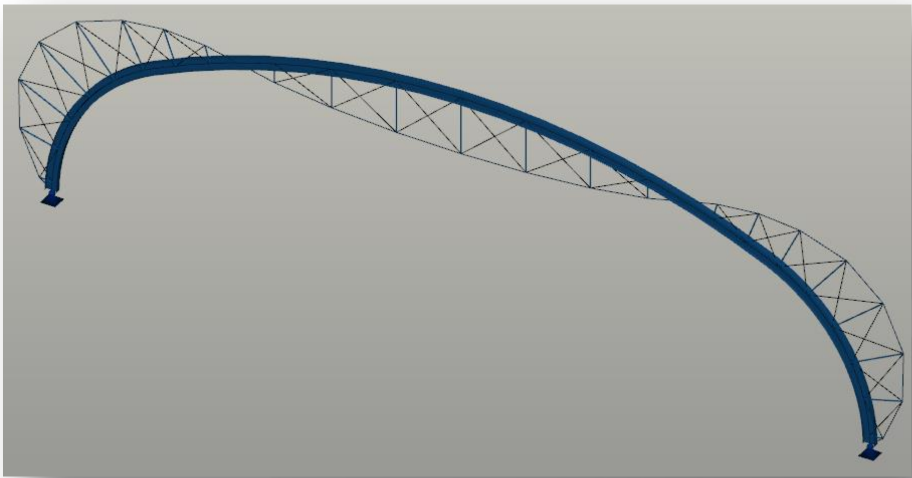
- **wireframe view** : on the wireframe view the lines of the cross sections and plate thickness appear, the supports are line types.



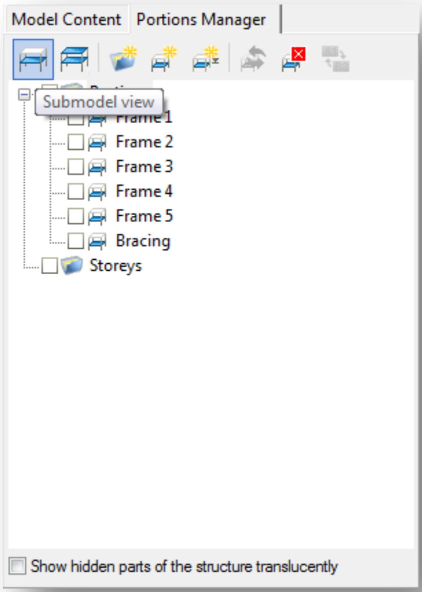
- **hidden line view** : the cross section and plate thickness appear in a solid form without shading and sparkling effects, the supports are solid types.



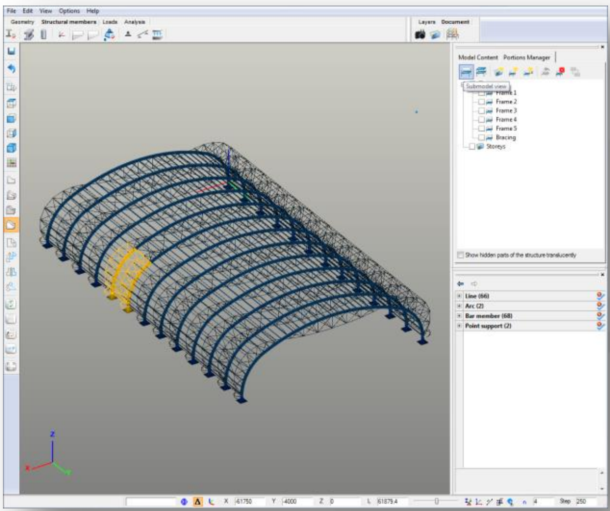
- **solid view** : the cross section and plate thickness appear in a solid form with shading and sparkling effects, the supports are solid types.

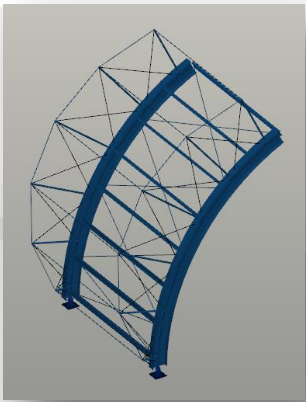


When the full model tends to be more complicated and parts of the model are in focus then the ***SUBMODEL VIEW*** should be applied.

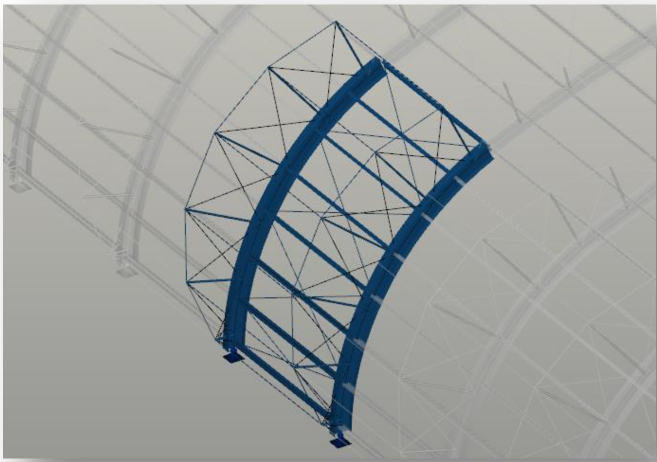


This option shows only the selected parts of the model,



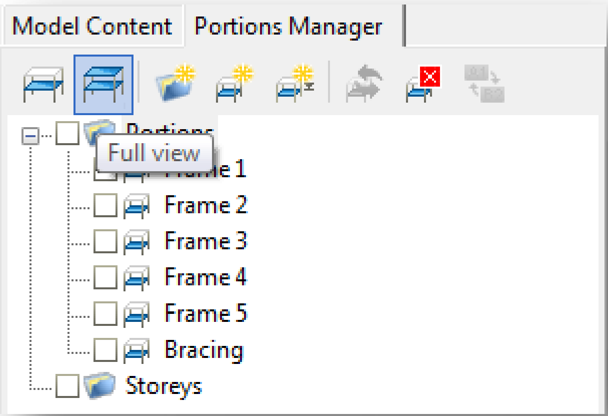


or checked the *Show hidden parts of the structure translucently* checkbox at the bottom of the ***PORTIONS MANAGER***.



From ***GEOMETRY*** to ***ANALYSIS*** in every tab the selected submodel will be visible.

To make the whole model visible again click the ***FULL VIEW*** icon.



3.3 SELECTION

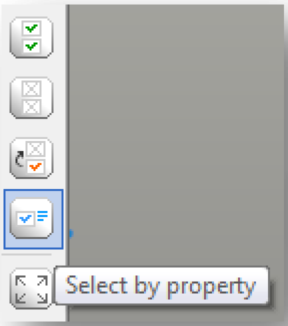
Objects can be selected on the graphical window, in the object tree, or by the additional selection options. As a general rule only the visible objects can be selected graphically, accordingly the same action can result in different selection on different tabs or model views. A selected object change to unselected by clicking on it while pressing the ***SHIFT*** button. The ***ESC*** button always unselects all objects.

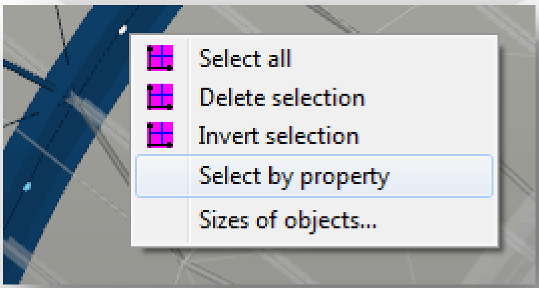
The basic selection modes and their operation are the followings:

- **simple graphical selection:** click on the object on the graphical window by the left mouse button. All objects will be selected which graphical symbol intersects the imaginary line set up perpendicular to the viewing (camera) plane at the clicking point. It means that the selection depends on the size of the graphical symbol of the objects and note that the covered objects will be selected as well!

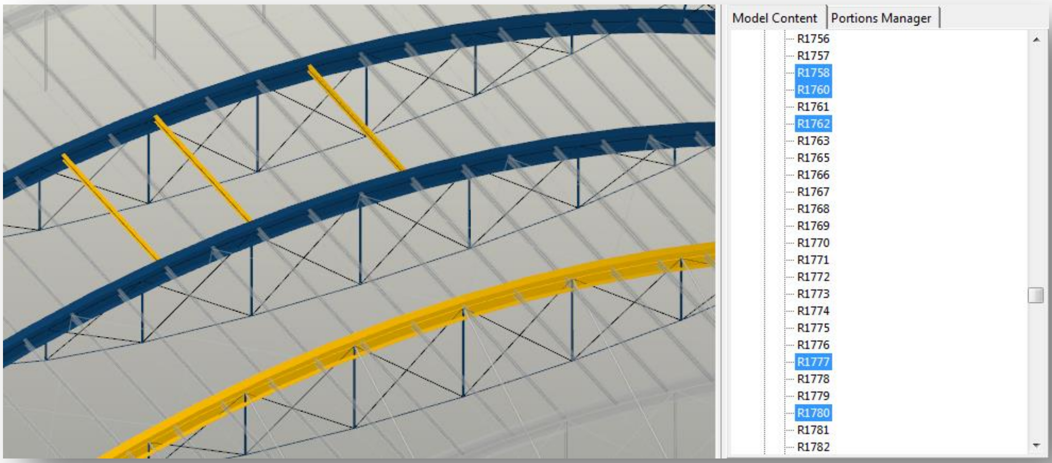
- **window selection:** holding down the left button and moving the mouse will create the window. It is completed when the mouse button is released. The actual size of the window is continually seen on the screen. Drawing the window from left to right, then only those members are selected that are within the window with their full size (inclusion window selection). Drawing the windows from right to left then every member is selected no matter with how small part is found in the rectangle (section and inclusion window selection).

- **selection by property:** this option can be reached on the left side bar, or by right mouse button click on the graphical window. The selection can be made by choosing the desired object and further narrowing is possible by sorting the appropriate parameters out.





- **object tree selection:** selecting objects by clicking on the elements of the Object Tree.



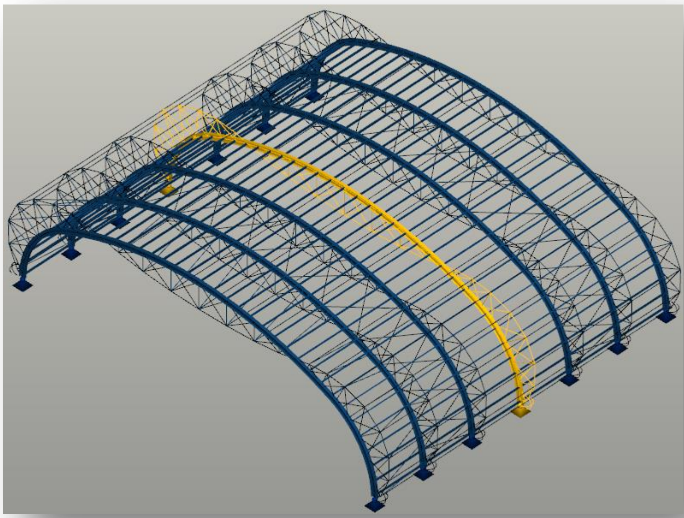
By clicking the object names in the object tree while holding down the ***CTRL*** or ***SHIFT*** key results multiple objects selection. To reselect any of the selected object: hold down the ***CTRL*** key and click on the object name again in the object tree.

The additional generalized selection options (select all, unselect all, inverse selection) can be found on the left side bar or they can be reached by right mouse button click on the graphical window.

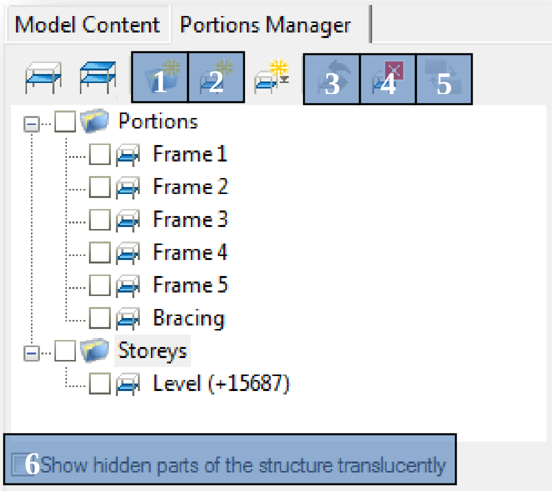
3.4 PORTIONS MANAGER

PORTIONS MANAGER is a great tool to manage different but related parts of the model ie. floors, columns, bracings, beams, etc. This function can be achieved on the ***PORTIONS MANAGER*** tab.

New portions can be added by clicking the ***NEW PORTION*** icon (#2). Before do so, select the parts of the model which will belong to the portion.



The portions can be arranged into folders. New folder can be created by licking the ***New folder*** icon (#1).

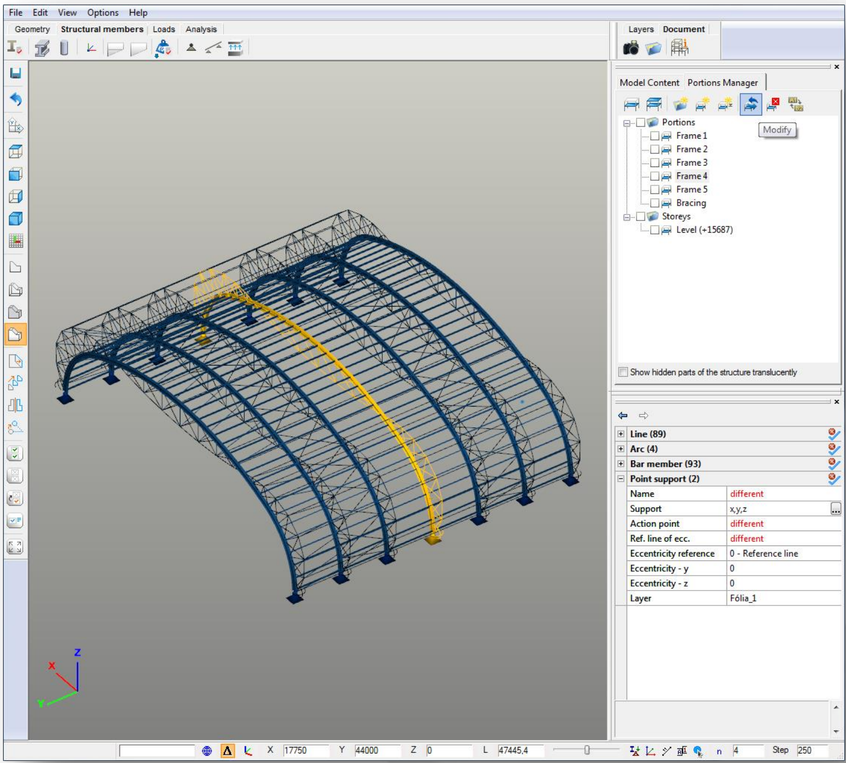


To turn on portion, check in the check-box in front of name of the portion. More portions can be turned on at the same time.

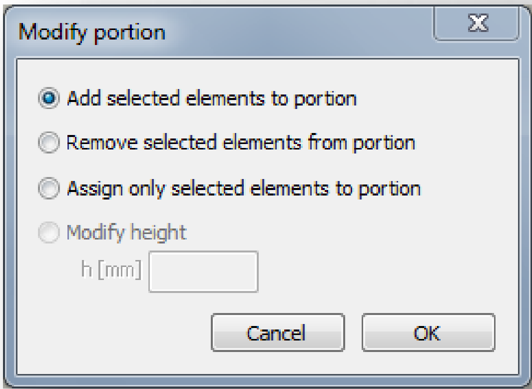
If the *Show hidden parts of the structure translucently* check box (#6) is clicked at bottom of the ***Portions manager*** tab, then the hidden parts of the model is also shown but those parts a translucent.

Any objects of the structural model can be part of more portions at the same time.

To modify a previously defined Portion, first select the relevant parts of the model which will be the subject of the modification, then click on the name of the part-model you are going to modify and finally click on the ***Modify*** icon (#3).



The ***Modify portion*** window will appear with the following options:



- **Add selected elements to portion:** By clicking the ***OK*** button, the selected members will be added to the active portion. When in the selection some member already belongs to the actually modified portion, then it will have no additional effect.
- **Remove selected elements from portion:** All those selected elements will no longer belong to the modified portion.
- **Assign only the selected elements to portion:** The modified portion will contain only the selected elements.

Renaming the Portion can be done by clicking the name of the portion and click on ***RENAME*** icon (#5).

Portion can be deleted by clicking by clicking the name of the portion and click on ***DELETE*** icon (#4).

3.5 OBJECT NAMES AND RENUMBERING

Names and labels are used for the better organization of the different objects in the model and for the better transparency of the documentation. The names and numbers of the objects are generated automatically while modeling.

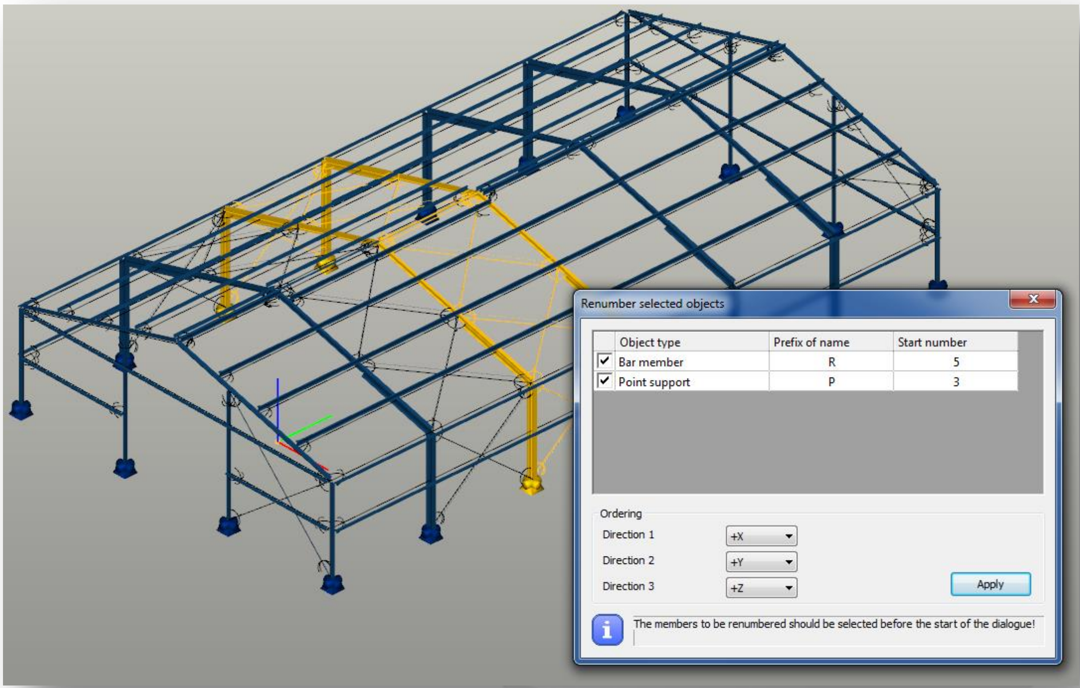
The following name and renumbering options are available:

- **numbering of finite element points** – generated fully automatically, the user can't influence the process.
- **names of structural element** – these are generated automatically while the user creates, copies structural objects (beam, column) consisting of a prefix and a number (for instance: "B1" for a bar element). These names appear on the object tree in the right side table. The names can be modified by the ***RENUMBER SELECTED OBJECTS*** tool on the ***STRUCTURAL MEMBERS*** tab. Using this function a specific prefix and a start number can be defined, and a direction can be set in which the renumbering will be processed. On the finite element view these names denote the numbering of the finite elements of the members.

- **names of structural supports, releases and links** – these names show the type of the placed support, release or link objects (for instance: “Fixed” or “yy,zz,w”)

Renumbering the objects

To renumbering the objects first select the relevant objects, then click on the ***RENUMBER SELECTED OBJECTS*** icon on the ***STRUCTURAL MEMBERS*** tab.



The ***RENUMBER SELECTED OBJECTS*** window will appear, with the automatically recognized object types contained by the set of selection. With the checkboxes could be selected the different types objects for renumbering.

The following functions can be used for renumbering:

- Prefix of the name can be defined
- Start number can be defined
- Priority of the renumbering can be set in the 3 main directions (X, Y Z)

The new names of the objects will built up from the given prefix and the new serial number.

Label's visibility

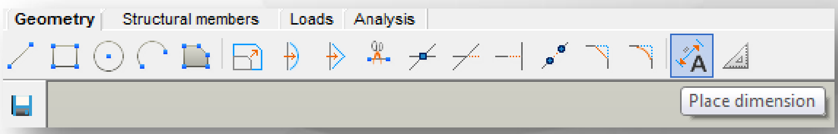
The visibility of the labels can be set on the bottom status bar as it was discussed above (***1.3.6 The status bar***). The following label options are available:

- **material** – the used material name of structural members (for instance: "S235")
- **shape** – the name of the used cross-sections of structural bar members (for instance: "HEA 200")
- **load intensity**
- **units** – the units of load and thickness if set (for instance: "kN" or "mm")
- **initial curvature** – the value of the initial curvature on structural bar members
- **coordinate system** – the names of the local coordinate axes (X,Y,Z)

Additionally arbitrary text can be assigned to structural elements manually on the ***Dimensioning*** dialogue using the last tool.

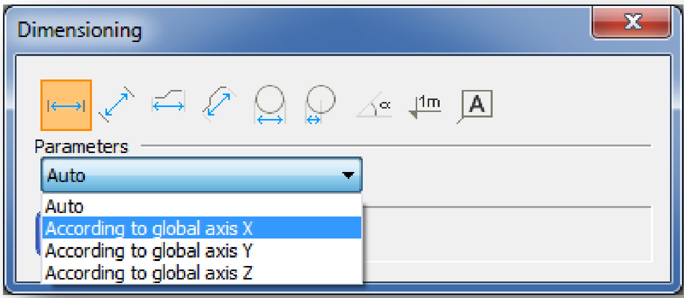
3.6 Dimensions

For the documentation of the calculated structural model it is very important to be aware of the exact dimensions of the model. The dimensioning tool can be found on the ***Geometry*** tab.



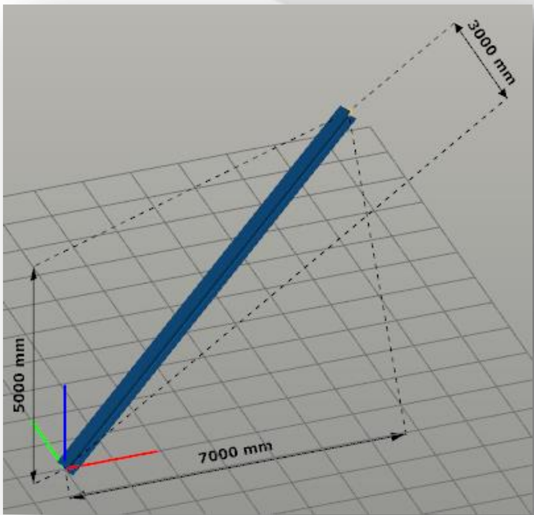
This provides a great number of different dimensioning options:

- **Projected dimension of a linear object** : The projected length of the selected linear object onto the global X or Y or Z axis will be dimensioned.

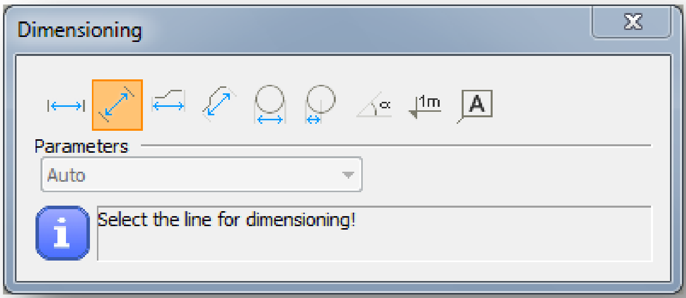


Parameters:

- Auto: moving the mouse, the system detects the desired reference axis.
- According to global axis X: the X axis projection of the length of the linear object.
- According to global axis Y: the Y axis projection of the length of the linear object.
- According to global axis Z: the Z axis projection of the length of the linear object.

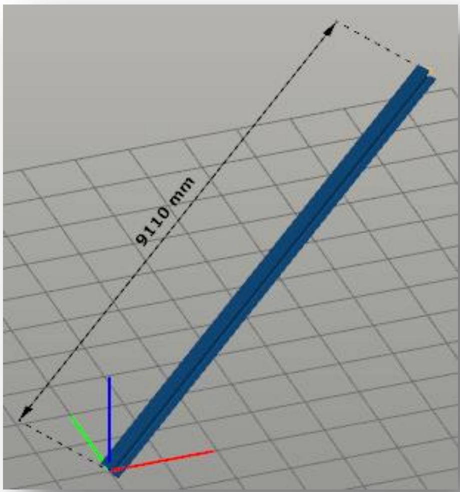


- **Lengthwise dimension of a linear object** : The length of the selected linear object could be placed in one of the plane which is defined by centerline of the object and the global coordinate system X,Y plane, and a plane perpendicular to this plane and containing the centerline of the object.

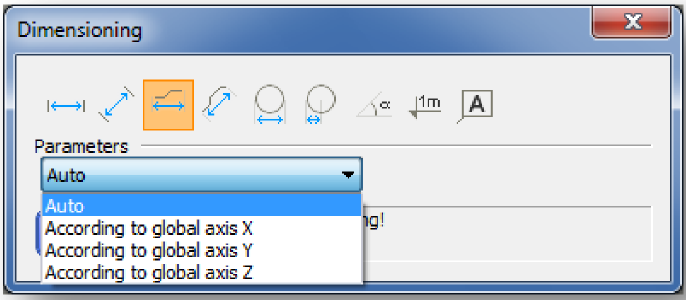


Parameters:

- Auto: moving the mouse, the system is detects the desired plane.



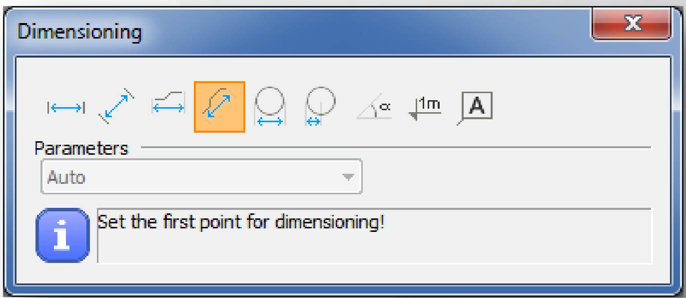
- **Projected dimension of the distance between two selected points** : The length of the distance between two selected points will be projected onto the global X or Y or Z axis.



Parameters:

- Auto: moving the mouse, the system is detects the desired reference axis.
- According to global axis X: the X axis projection of the distance between two selected points.
- According to global axis Y: the Y axis projection of the distance between two selected points.
- According to global axis Z: the Z axis projection of the distance between two selected points.

- **Lengthwise dimension between two points** : The distance between the selected two points could be placed in one of the following plane.
  - Plane defined by centerline of the object and the X,Y plane global coordinate system
  - The plane perpendicular to the above plane and containing the centerline of the object.

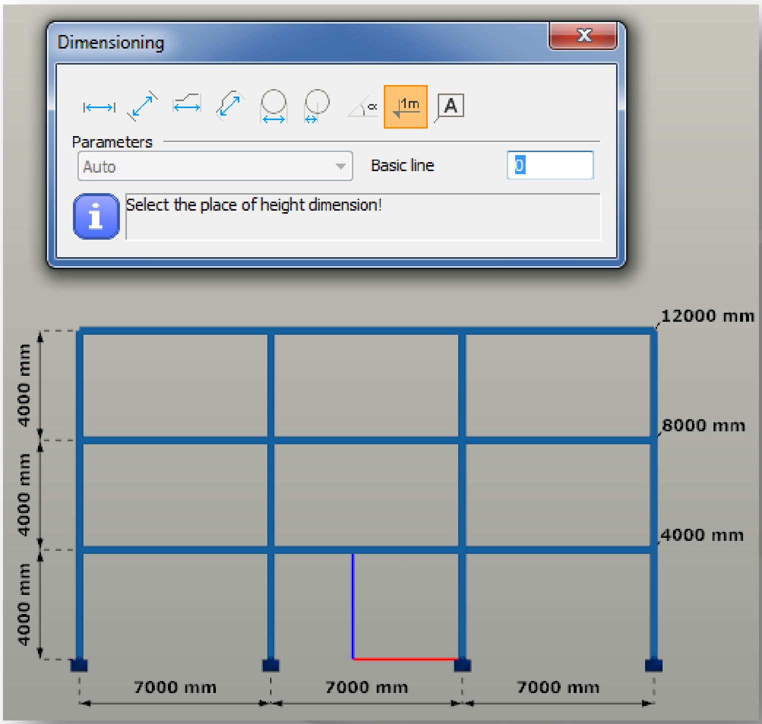


Parameters:

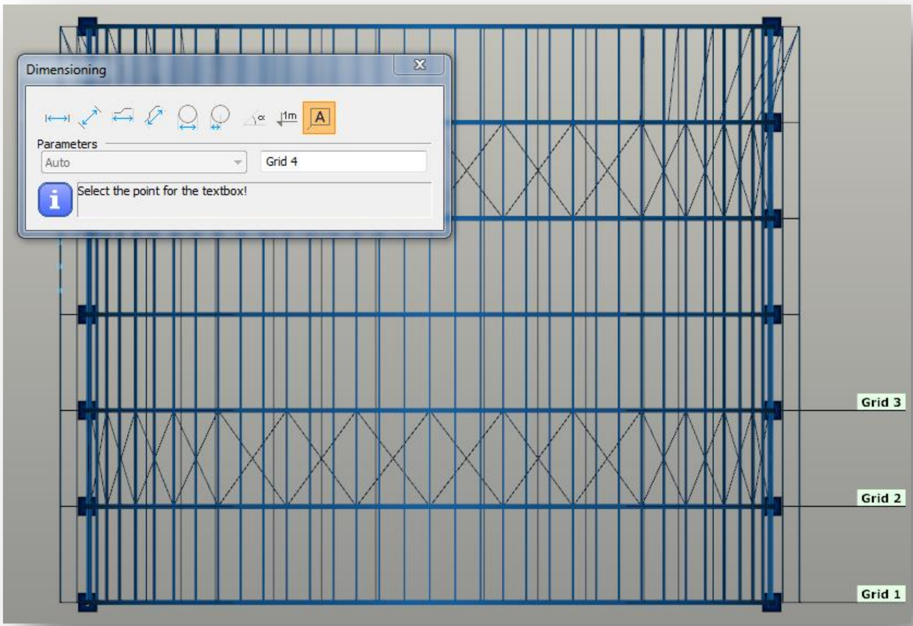
- Auto: moving the mouse, the system is detects the desired reference axis.

- **Diameter of a circle**
- **Radius of a circle**
- **Angle of two lines** : After selecting two lines any of the four angle could be dimensioned.
- **Height dimension** : This function helps for quick dimensioning the high positions of the selected points of a construction relative to global Z direction related to the given value in Basic line filed.

  The content of the **Basic line** field means the height of the 0 level in the global Z direction. All the picked points will be measured from to this value as a 0.

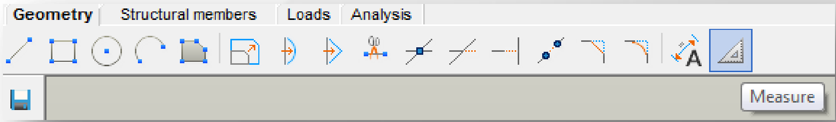


- **Textbox to a given line** : The given text (Grid 4) in the script field can be placed into the model.

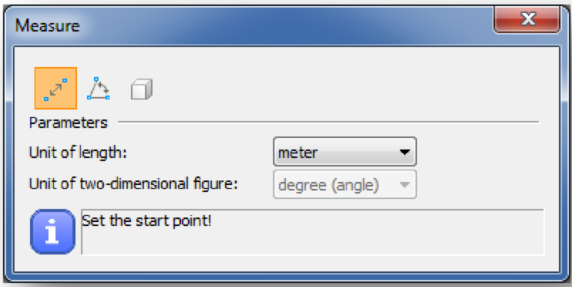


3.7 Measure

During modeling, sometime necessary to measure distances, and angles. The ***Measure*** tool can be found at the end of the ***Geometry*** tab.



The following window offers the measuring functions:



- **Measure distance** : Shows on the screen the measured distance between two picked points in meter or in millimeter, selected from the “unit of length” list box.
- **Measure angle** : Function measure the angle between two lines was defined by clicking three snap points in the 3D space in the plane of the two lines in degree or radian in.
- **Data of structural member** : Function shows on the screen the length, the surface and the weight of the selected structural member.

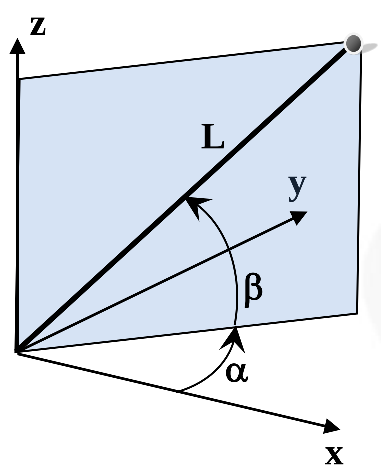
4 DRAWING GEOMETRY

4.1 BASICS

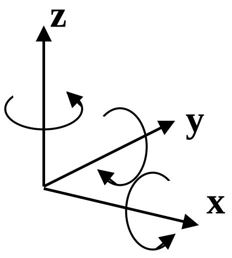
The first step of the modeling phase is the geometry drawing of the structure. In StabLab all the geometrical objects can be easily created and modified in a 3D space. Among the basic CAD drawing, modifying and moving functionality advanced snap options make the modeling efficient. All the CAD functionality is placed on the ***GEOMETRY*** tab, the further view, select and snap options are on the side and status bar. Additionally all the relevant modeling functions are placed on the dialogues of structural members, supports and loads.

4.2 COORDINATE SYSTEMS

For the appropriate modeling and interpretation of the analysis results it is of high importance to be aware of the applied coordinate systems. This section summarizes in detail all the coordinate systems used in the *StabLab*.



Two different type of coordinate systems can be used: the rectangular Descartes system (XYZ) and the polar system (αβL). In the polar system "α" denotes the angle between the axis "X" and the vertical plane defined by the axis "Z" and the point, "β" denotes the angle between the section line of that plane and the coordinate plane "XY" and the line from the origin to the point, and "L" denotes the distance from the origin. The polar coordinate system is less frequently used so the rest of this section describes the Descartes system.



As a general convention all the Descartes coordinate systems (axes directions and rotation signs) follow the most commonly applied right hand rule. This rule is valid for the definition of geometry and loads as well. The positive moments and rotations are defined as counter clockwise about the axis if it is viewed in front (towards the origin) in both the global and local system.

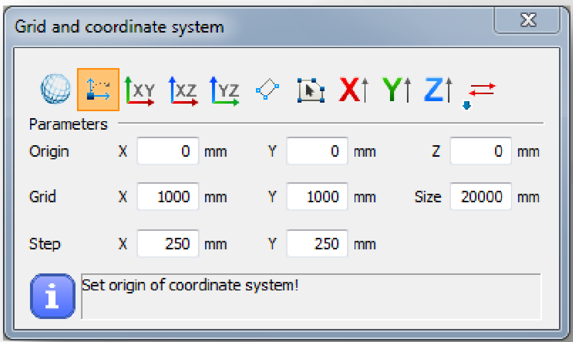
In the further text the following denotations are used:

- X, Y,Z: global coordinate system
- x, y, z: local coordinate system

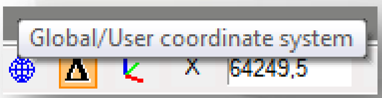
4.2.1 User coordinate system

The *User coordinate system* (*UCS*) is used for making the modeling easier and faster. The *UCS* is a specially positioned Descartes system (XYZ).

The tools for creating and modifying the coordinate systems can be reached by the ***Set grid and coordinate system*** button on the side bar.



The User coordinates system can be switched on (or switch back to the global system) at the bottom status bar

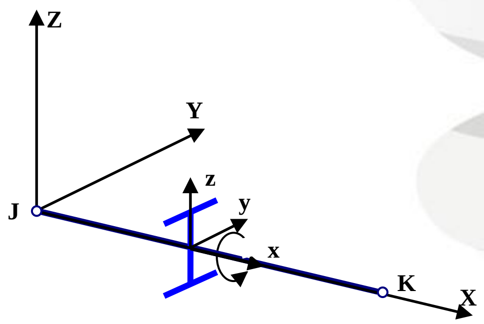


The following creation options are available (from left to right):

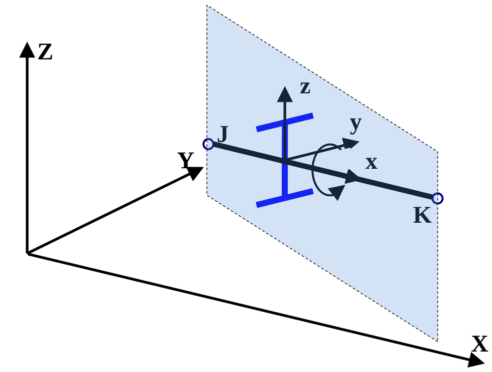
- Switch to global system
- Set new origin for the UCS
- The "XY" plane of the global system will be the "XY" plane of the UCS
- The "XZ" plane of the global system will be the "XY" plane of the UCS
- The "YZ" plane of the global system will be the "XY" plane of the UCS
- Set the UCS by 3 points: the origin, the direction of "X" and "Y" axis
- Set the standing of the UCS identical to the local coordinate system of the selected plane with origin placed into the first node of this plane
- Set the direction of the axis "X" by two points
- Set the direction of the axis "Y" by two points
- Set the direction of the axis "Z" by two points
- Reverse the direction of the selected axis

4.2.2 Local coordinate system of bar elements

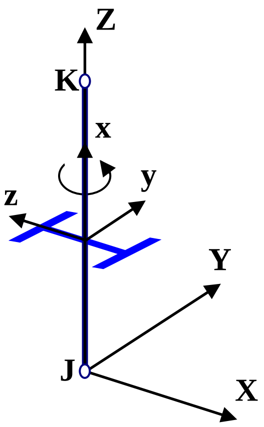
The reference line of bar members defines the axis "x" of the local coordinate system of the bars. The direction of the axis "x" is given by the start (**J**) and end (**K**) node of the reference line.

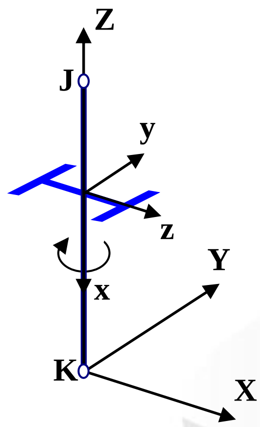


In the basic case the local coordinate system of a bar can be determined considering its reference line ("x") identical to the global axis "X". In this case the direction of local axes "y" and "z" are identical to the global axes "Y" and "Z". The position of the cross section on the bar is the following: the reference line goes through its centre of gravity, and the section “YZ” system coincides with the local “yz” system of the bar.



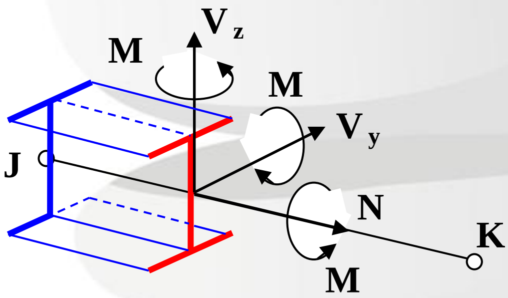
In case of a generally positioned beam member (not column with vertical reference line) the plane defined by the local “xz” system takes always vertical position and the axis “z” points towards the positive “Z” direction. For beams with (single) curvature the axis “x” is always the tangent.





In case of vertical reference line (column members) the local axis “y” has the same direction as the global axis “Y”, the direction of the local axis “z” depends on the placement of the column (position of the start and end points).

The interpretation and signs of the inner forces on a member is defined by its local coordinate system and the mentioned general sign convention as described in the following figure:



4.3 Defining action points

Action points (snap points) can be set by moving the cursor on the icon on the right side of the bottom status bar. You can find the description about the snap point settings above in **Status bar** section (*1.3.6 The status bar*).

4.4 Drawing

The functions for drawing described below are collected on the *Geometry* tab.

Draw line

Simple line drawing. First select the start point. Then select the end point.

Polyline drawing. First select the start point. Then select the next points. Press *Esc* to interrupt line drawing.

Draw rectangle with lines

Draw rectangle. Select the corner point. Select the opposite corner.

Draw leaning rectangle in the space. Select the corner point. Select end point of the side of the rectangle. Select third point of the rectangle.

Draw Circle

Draw circle by radius. Select the centre-point of the circle. Select the radius of the circle. The circle will be created in the plane containing the centre-point and parallel to the actual *UCS*.

Draw circle by diameter. Select the start point. Select the end point. The circle can be created out of the actual *UCS* system.

Draw circle by 3 points. Select first point. Select second point. Select third point. The circle will be created in the plane defined by the three definition points.

Draw Arc

Arcs could be created out of the actual *UCS*.

Draw arc by center-point, start point and angle. Select the center of the arc. Select the start point of the arc. Select the angel of the arc or type it to the dialog window

Draw arc by 3 points. Select the start point. Select the end point. Select a point in the arc.

Draw arc by start point, end point and tangent. Select the start point. Select the end point. Select the other point of the tangent.

Draw two-dimensional plane elements

Draw two-dimensional plane element. Define the geometry of the plane element.

Draw hole. Select the plane element. Define the geometry of the hole (Rectangle, closed polyline or circle).

Change the direction of the local “x” axes of a plane element by defining two points. Select the surface. Select the start point. Set the direction with the second point.

The construction modes of the plane elements:

The construction mode of a rectangle type plane element can be selected: rectangle or leaning rectangle.

The construction mode of a circular type of the plane element can be selected: draw circle by radius, by diameter, or circle by 3 points.

Draw polygon (a closed polyline). Set the first point. Set the next points. To close the polygon press right click on the mouse.

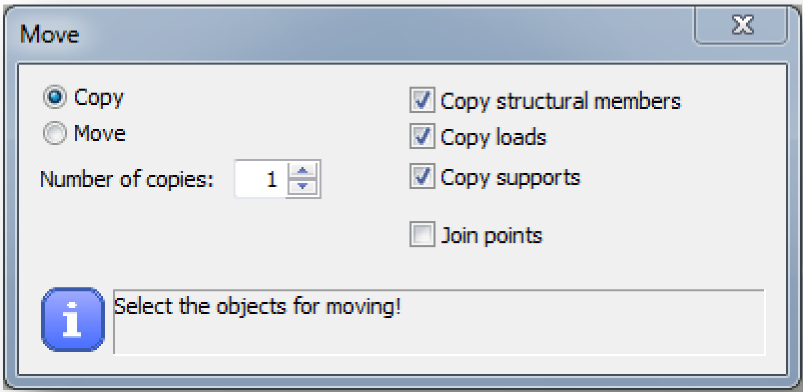
4.5 MODIFYING

The four mostly used modifying functions are placed on the left sidebar:

Move point and edge. Select the point and (or) edge to be moved. Set the new position of the point (edge). This function is used for change the position of any of the end point of a linear or circular element, or to change the position of an edge or a corner point of a polygon, or of hole in a polygon.

Move selected objects. The selected objects can be moved or copied.

Select the objects, and click the ***MOVE SELECTED OBJECTS*** icon:



Select the transformation (Move or Copy). The copy function can be activated by radio button near the Copy function. The number of copies can be selected.

Set start point of the vector for moving/copying. Set the end point of vector for moving/copying.

The selected members, loads and supports will be moved or copied to the position which is defined by the given vector accordingly to the selected options. The Move/Copy function is independent from the actual *UCS*.

Effect of the options:

Copy structural members:

Unchecked: Use when only the loads and/or the supports of the structural member need to be coped or moved.

Checked: Default. Structural members will be copied or moved.

Copy loads:

Unchecked: The loads will not be copied with the selected structural members.

Checked: The loads will be copied with the selected structural members.

Copy supports:

Unchecked: The supports of the selected structural members will not be copied.

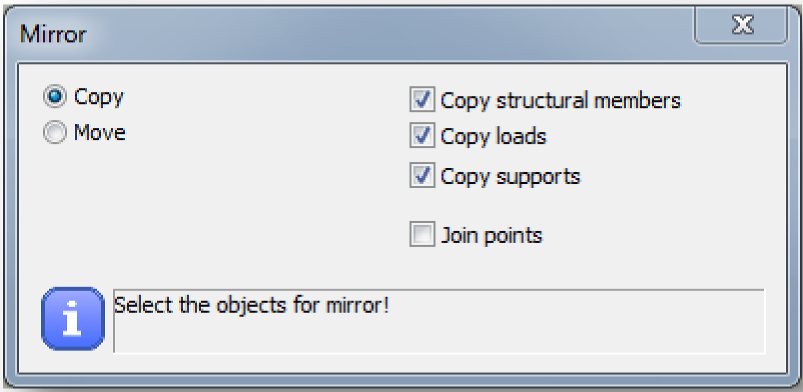
Checked: The supports of the selected structural members will be copied. (In case of the selection set does not contain at least one selected member, than the selected supports will not be copied.)

Join point:

If the **Join points** box is checked then the end points of the copied objects will be connected with lines.

 **Mirror selected objects.** The two given points represent mirroring axis will be projected into the actual *UCS* system and the mirroring will be performed in the actual *UCS* symmetrically to the axis of mirroring.

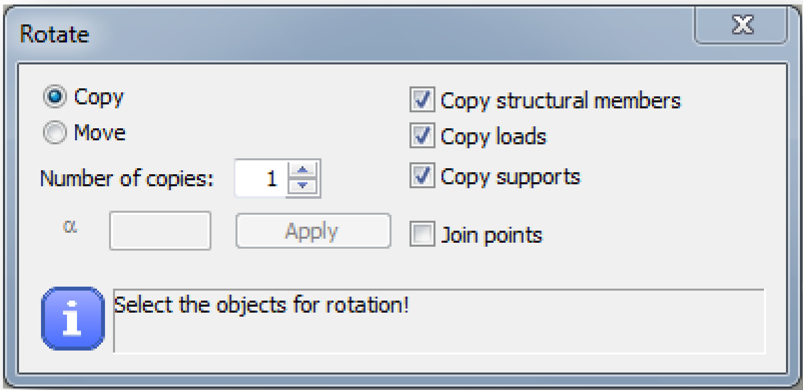
As first step select the *UCS* for mirroring (if it is needed), select objects to be mirrored and click on the ***MIRROR SELECTED OBJECTS*** icon.



Selecting the mirror line by picking two points for the mirror axis, the selected objects will be moved or copy to the mirrored position.

The effects of the options are the same as it was described above.

 **Rotate selected objects.** Select the objects for rotation.



Select the Move or Copy option for the transformation. The number of copies can be selected. Set the center of rotation. Set reference point of rotation. Set the direction point or type in the rotation angle and click on the ***Apply*** button.

The effects of the options are the same as it was described above.

The further modifying functions are on the *GEOMETRY* tab:

Sizing of the selected objects. Select objects to sizing. Set the sizing center, the sizing reference point and the sizing multiplier point (the distances of the sizing reference point and the sizing multiplier point to the sizing origin point determines the sizing factor). The sizing will be performed. The length (and depending on the sizing origin maybe the position) of the objects will be changed.

The length of the line-loads will be changed proportionally.

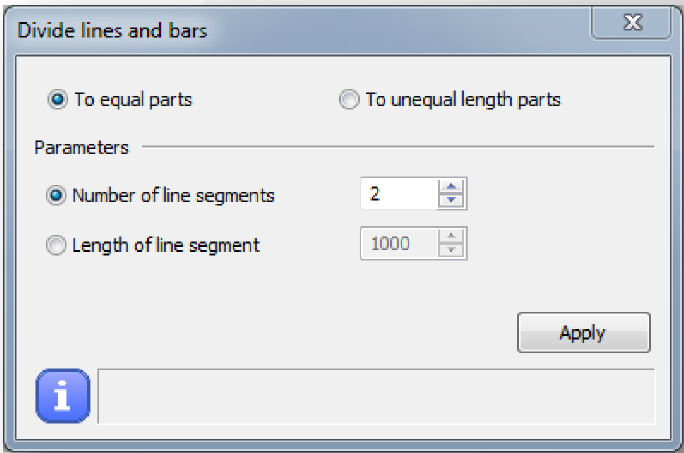
Curve selected line element. Select the edge for bend. Set the insert point.

Refract the selected line element. Select the edge to refract and the insert point along the axis of the selected object. Set the new position of the selected point. The original object will be broken into two parts.

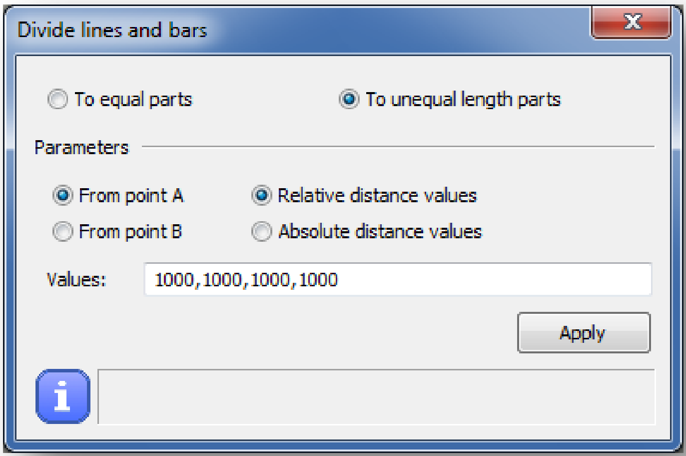
The coordinates of the new position of the selected point also can be given manually accordingly to the selected coordinate system.

Divide selected objects. Lines and bars can be divided into equal or unequal parts.

By selecting the **To equal parts** option the number of the segments or the length of the segments can be set.



By selecting the **two unequal parts** option a series of relative or absolute lengths can be defined measured from A or B endpoint of the object.



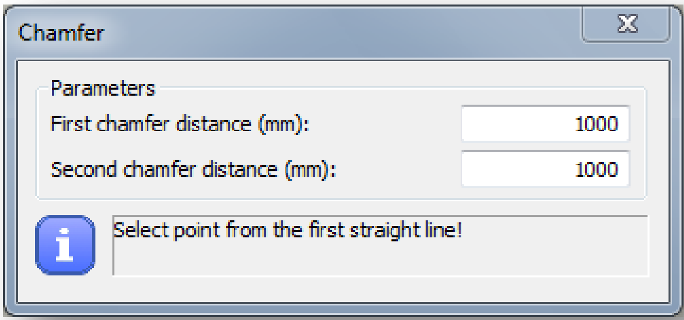
Break two selected objects. Select the two intersected objects to break. Both object will be broken by the intersection point. It works with linear and curved objects.

Trim selected element by cutting edge. Select cutting edge. Select the part to be trimmed. It works both with linear and curved objects.

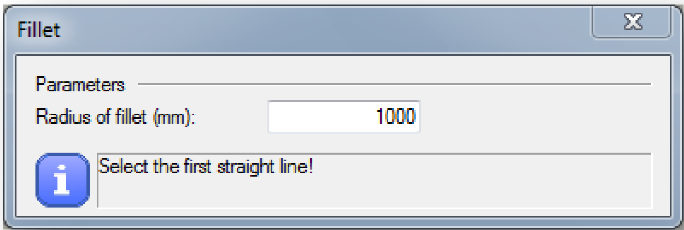
Extend selected line elements to limit line. Selecting the limit line and click on the linear object to extended to the limit line.

Divide by trim. Cut out a part of the selected element. Select object for cut out. Select start point and the end point of the segment of the cut out.

Chamfer of two selected linear element. Select the first and the second linear element. The chamfer will be applied according to the order of selection.

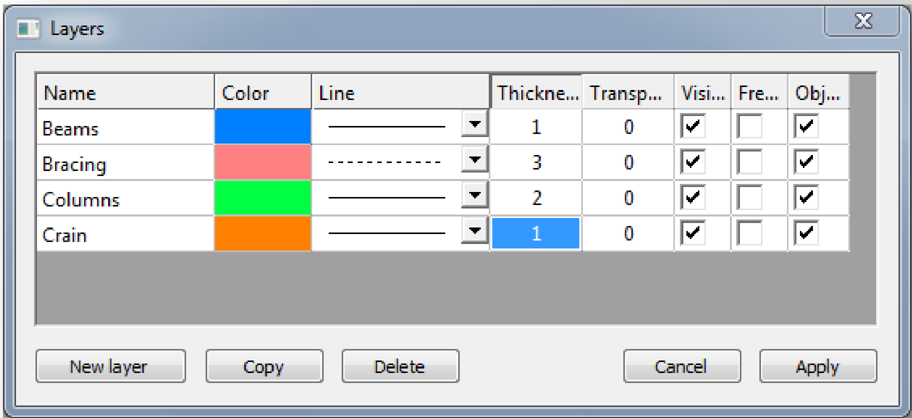


Filleting of the edge of the section of two line element. Select the first and the second linear element. Chose a fillet radius.



4.6 Layers

The layers dialog window can be used for:



- Create new layers
- Copy existing layer-definition to a new layer
- Delete layers
- Edit properties of existing layers (colour, line style, thickness and transparency)
- Turn the visibility of layers on and off
- Freeze layers from activity whilst keeping them visible
- Turn on and off own style. If Objects of own style checkbox is turned OFF then all the objects on the layer use the selected colour, line style, thickness etc. If it is turned ON (default setting) then all the objects use the global style.

All CAD systems have a layer concept of some sort. Objects are placed on appropriate layers as a practical way of managing the objects within the model.

5 STRUCTURAL MODELING

5.1 BASICS

In *StabLab* great emphasis was taken on the user friendly structural modeling in which the engineer can build the structural model using real structural elements. Accordingly the user model, built by the engineer using whole structural members (beams with haunches, tapered columns, tension braces etc.), is completely separated from the calculation model, which is an automatically generated sophisticated finite element model used by the structural analysis. It follows that the engineer does not need to deal with the calculation model; he/she can concentrate solely on the direct productive labour building the real structural model.

IT SHOULD BE KEPT IN MIND THAT THE ENGINEER SHOULD BE AWARE OF THE FEATURES, POSSIBILITIES AND LIMITATIONS OF THE APPLIED ANALYSIS MODEL ALREADY AT THE MODELLING STAGE BECAUSE THE MISUNDERSTANDING OF THE MODELLING OPPORTUNITIES CAN LEAD TO UNEXPECTED ANALYSIS AND DESIGN RESULTS.

All the functionality connecting with the structural modeling are placed on the ***STRUCTURAL MEMBERS*** tab.

5.2 LINE MEMBERS

5.2.1 CREATING LINE MEMBERS

In *StabLab* the line members with steel thin-walled, reinforced concrete, and composite cross-section can be modeled. The members – depending on the placement – can be columns or beams. The column is a special position line member, which is always vertical, so the placement is simpler, because only the height should be defined.

The dialogue allows the following modeling options for creating members:

Draw the reference line of the member

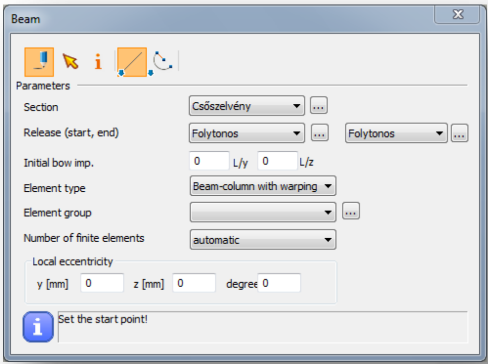
draw as a line segment or continuous line

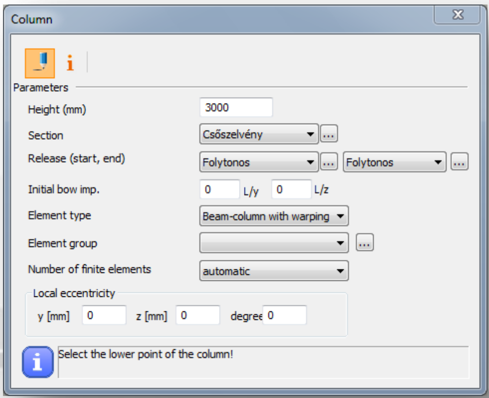
draw as an arc by center point, start point and angle, 3 points or start point, endpoint and tangent

Select an existing line for the section as a reference line of a member

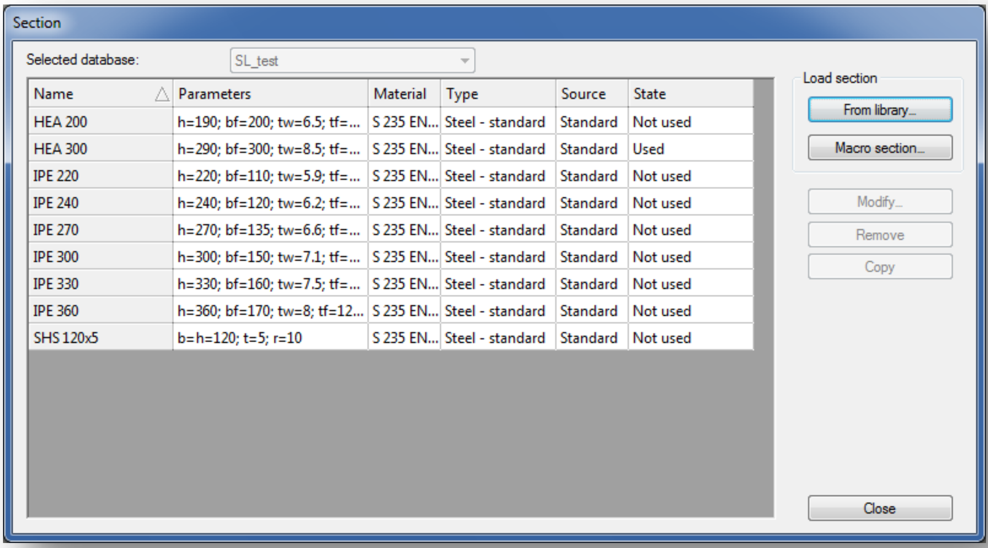
Read out the parameter of an existing member by clicking on it and assigning those parameters to the one to be created

Member parameters can be defined in the middle part of the dialog:



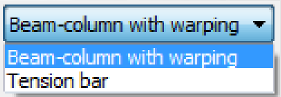


- *Section*: before creating a beam or column member cross-sections should be loaded, and the appropriate section can be selected from the combo including the loaded sections. If no section has been loaded in advance the **SECTION** dialogue can be called by pressing the ... button next to the combo.



- *Release (start, end)*: predefined releases can be assigned to the ends of the members. If new release type is needed the **RELEASE** dialogue can be called by pressing the [...] button next to the combo.
- *Initial bow imp.*: half-sine wave shaped initial bow imperfection can be defined in the two local direction perpendicular to the member reference axis ("y,z") with the given amplitude at the mid-length.
- *Element type:*

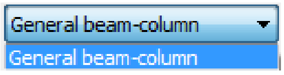
<u>*Section made from steel:*</u>



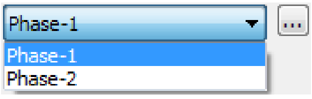
Two choices are possible *Beam-column with warping* and *Tension bar*, these types influence the finite element type used in the analysis. The *beam-column with warping* is a special element with 14 degree of freedom, including the warping of the thin-walled cross section. It is an important effect in case of structures with usual steel profiles.

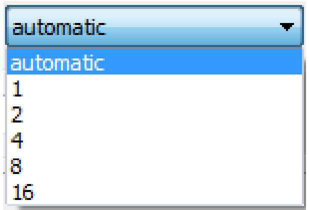
The *tension bar* can only resist tensional axial force (no bending or torsional moments and shear) if it got compression the analysis neglects its effect.

Reinforce concrete or Composite sections:

There is only one choice, 

the *General beam-column* type this is the commonly used 12 degree of freedom finite element.

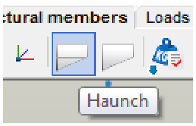
- *Element group:* the members can be sorted out into element groups for various purposes (selection, dominant results, sensitivity analysis etc.). 

if new element group is needed the **Groups of structural members** dialogue can be called by pressing the button next to the combo.
- *Number of finite elements:* 

the required number of finite elements used on the member in the analysis can be set. The default automatic option gives a sufficient result in the most cases.
- *Local eccentricity:* member eccentricity can be defined in the local coordinate system ("y,z") of the member, and the section can be rotated about the local coordinate "x".
- At the bottom of this dialog there is an instruction and command area for guiding the user.

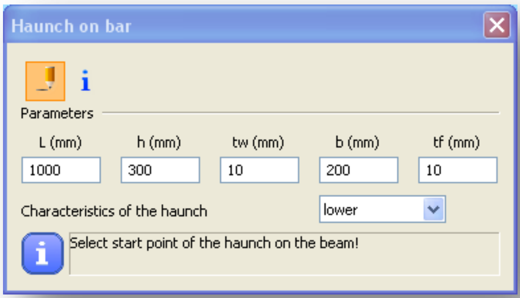
IT SHOULD BE NOTED THAT NOT ONLY THE ECCENTRIC AXIAL FORCE PRODUCES INFLUENCE IN THE ANALYSIS (ADDITIONAL BENDING MOMENTS) BUT THE ECCENTRIC BENDING AND TORSIONAL MOMENTS (ADDITIONAL BENDING AND TORSIONAL MOMENTS AND **BIMOMENT**)!

5.2.2 Haunched members

The created members can be strengthened, if necessary, by using *Haunch* 

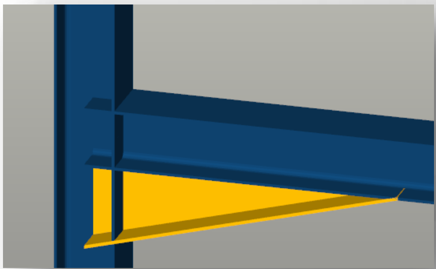
function.

The haunch can only be used for members with I type (IPE, HEA, welded I) sections, and the shape of the haunch is considered as half of a welded I section (with one flange) with decreasing web height. On the dialogue panel the length (L), height (h), web thickness (tw), flange width (b), flange thickness (tf) and the characteristics of the haunch should be set.

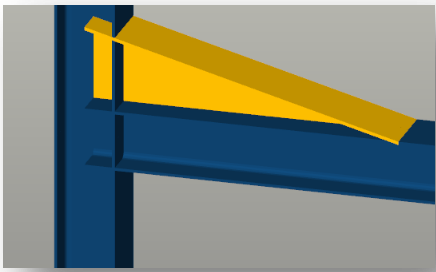


The position of the haunch is set by simply clicking on the member the start point (where the height of the haunch is full) and the direction point of the haunch. It should be noted that if the start point is a common end point of more members (this is the usual case, for instance at a beam-to-column connection point) then this point should be approached and clicked on the member to be haunched. The characteristics of the haunch denote the appropriate side of the haunch on the member according to the direction of its local "z" axis. In the case of usual beam position it results the followings:

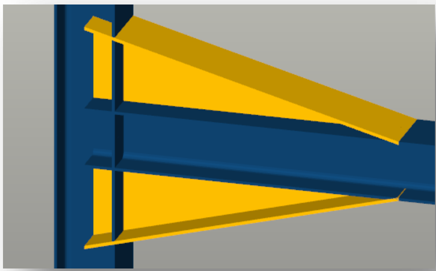
- lower



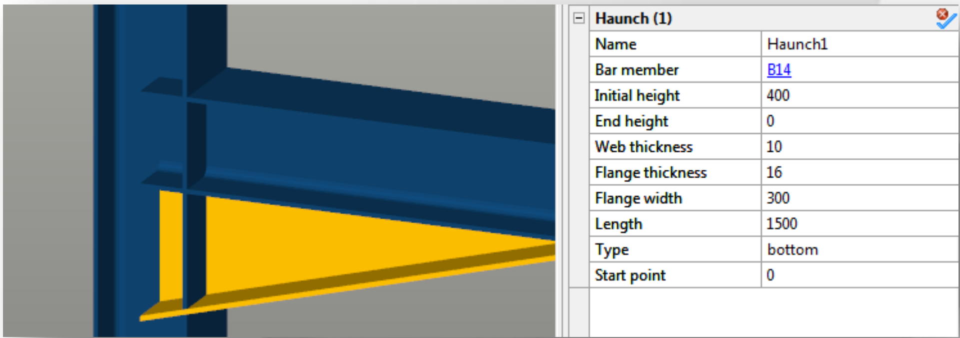
- upper



- symmetric



The haunches can be selected and modified in the parameter table as separate objects.



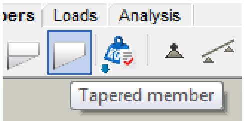
Important to know that for the haunched part of the member new sections are created during the automatic finite element generation which consist of the original section and the haunch with appropriate web height. These new sections are placed eccentrically on the reference line of the member (except the symmetrical haunch type).

This eccentricity causes additional effects in the analysis results due to the eccentric position of the sectional forces (for instance at the beam-to-column connection point of a frame with haunched beams and/or columns the equilibrium of the in-plane bending moments exists only if the additional moments from the eccentric axial forces are taken into account)

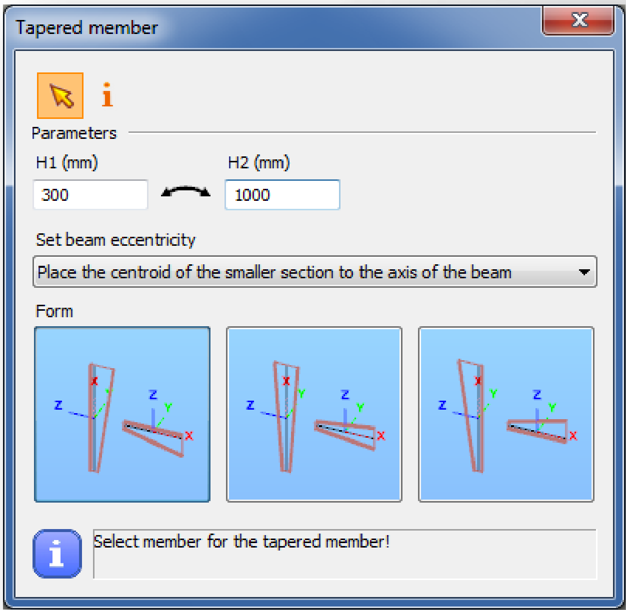
5.2.3 Tapered members

Tapered members are frequently used in the economic design of steel framed structures, so the fast and simple modeling of tapered members is of high importance. For the definition of tapered member first a line member with welded I or H, box, or cold formed C section should be created in the model.

Hot rolled and other shape of Macro section can't be tapered.

With the *Tapered member* 

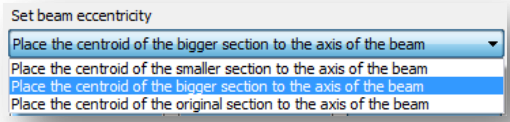
function the section height of these sections can be set to linearly varying along the member length.



First the user has to specify the parameters and beam eccentricity options for the member to be tapered on the ***TAPERED MEMBER*** dialogue.

Specify the start (H1) and end section height (H2) independently of the original section height of the cross-section. The start value for section height applied at the start point of the member, the end value applied at the other end. To change the H1 and H2 values click the icon. The values will be changed, and selecting again the member for tapering, the sizes of the tapering will change adequately.

The rules for the beam eccentricity of the tapered member relates to the axis of the originally positioned member to be tapered.

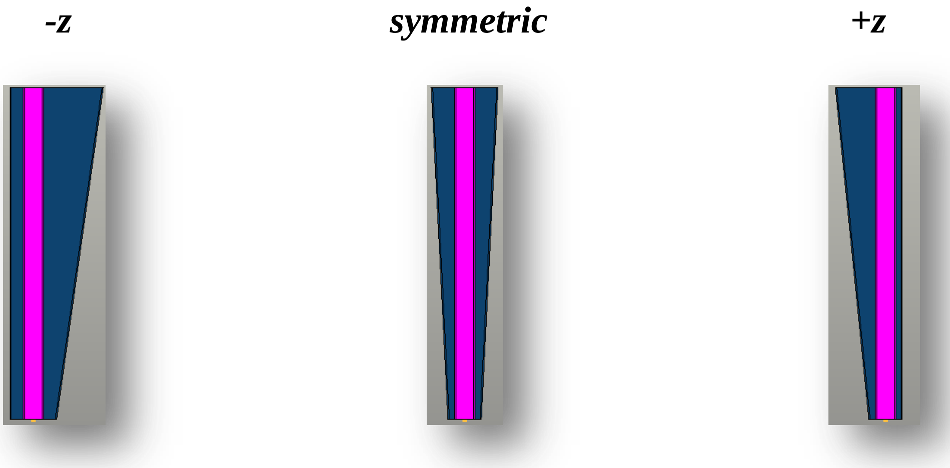


1. *Place the centroid of the smaller section to the axis of the beam:* the center of smaller H value of the tapered member will be positioned to the axis of the original beam
2. *Place the centroid of the bigger section to the axis of the beam:* the center of bigger H value of the tapered member will be positioned to the axis of the original beam
3. *Place the centroid of the original section to the axis of the beam:* the edge of the tapered member is coincident with the original member end and the tapering will start from this position.

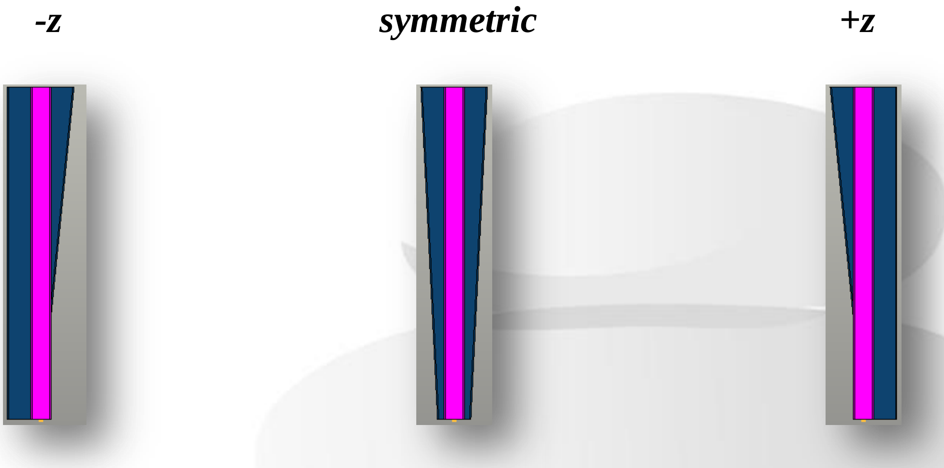
The relative position of the tapering can be –z (the left side of the tapered member will be parallel to the axis of the originally placed member), symmetric or +z (the right side of the tapered member will be parallel to the axis of the originally placed member). These definitions regulate the directions of the offset of the given height values along the local “z” axis of the tapered beam.

Below shown the effect of the different relative positioning by the chosen eccentricity:

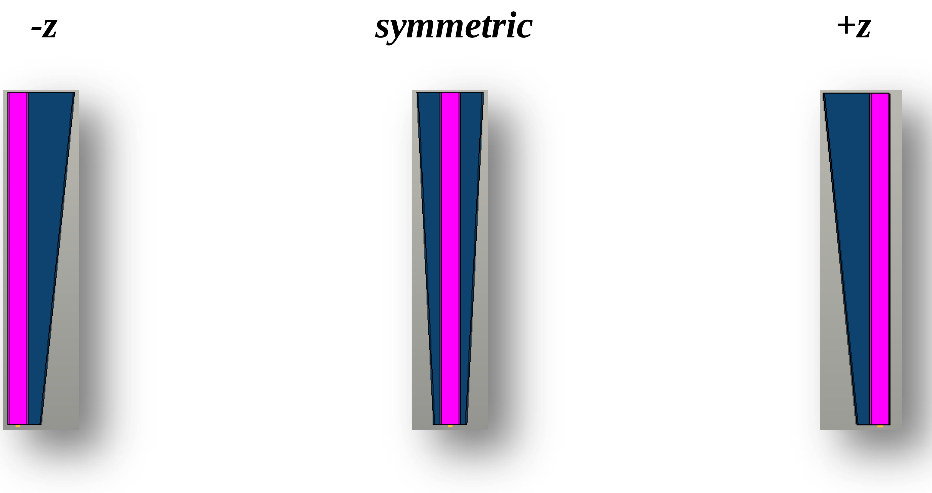
1. *Place the centroid of the smaller section to the axis of the beam:* the center of smaller H value of the tapered member will be positioned to the axis of the original beam



2. *Place the centroid of the bigger section to the axis of the beam:* the center of bigger H value of the tapered member will be positioned to the axis of the original beam



3. ***Place the centroid of the original section to the axis of the beam:*** The edge of the tapered member is coincident with the original member end the tapering starts from this position



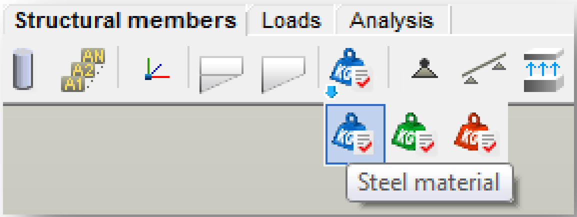
Activating the icon, by clicking an earlier created tapered member in the model, the values, the eccentricity and the relative positioning will be read out of that clicked, and appear in the dialog box. By clicking another member for taper, these parameters will be applied.

In case of selecting a tapered member not only the member but the tapering will be selected automatically and can be modified in the parameter table as separate object.

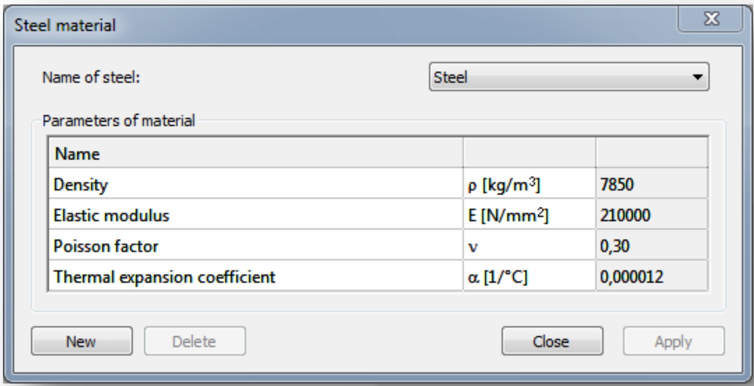
Important to know that for the tapered members new sections are created during the automatic finite element generation with appropriate section heights. These new sections are placed eccentrically on the reference line of the member (except the symmetrical tapering). This eccentricity causes additional effects in the analysis results due to the eccentric position of the sectional forces (for instance at the beam-to-column connection point of a frame with tapered beams and/or columns the equilibrium of the in-plane bending moments exists only if the additional moments from the eccentric axial forces are taken into account)

5.3 Materials

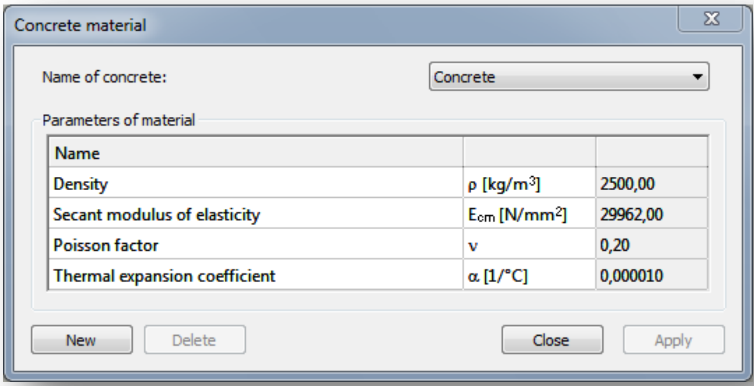
New material grades can be defined as one of the three different types of material: steel, concrete and concrete reinforcement.



The latter is only used for the rebar reinforcement of concrete or composite cross sections. New materials can also be created with arbitrary parameters.



In *StabLab* only elastic material is considered in the analysis, so the calculation results are only affected by the elastic modulus, Poisson factor, density (if the self-weight of the structure is considered) and temperature expansion factor (if temperature load is applied).



IMPORTANT TO NOTE THAT IN CASE OF BAR MEMBERS (BEAMS, COLUMNS) THE MATERIAL IS THE PARAMETER OF THE CROSS SECTION OF THE MEMBER, SO THE CURRENT MODIFICATION SHOULD BE APPLIED FOR THE APPROPRIATE CROSS SECTION. THIS FEATURE ALLOWS THE DEFINITION OF CROSS SECTIONS WITH MULTIPLE MATERIALS (E.G. COMPOSITE SECTION). ACCORDINGLY IF IDENTICAL CROSS SECTIONS WITH DIFFERENT MATERIAL ARE LIKED TO BE USED IN ONE MODEL THEN MULTIPLE CROSS SECTION DEFINITION IS NEEDED!

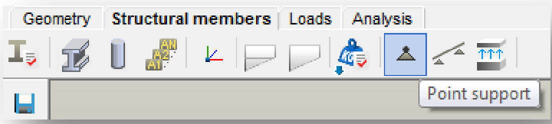
5.4 SUPPORTS

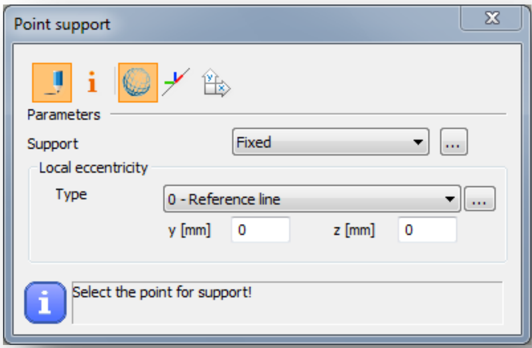
There are two types of supports in *StabLab*: point support and line support. Point supports can be placed on any part of a line member (predefined points are not needed; end points or snap points can be used). Line supports can be assigned to existing lines.

Supports can be placed according to the global ( ), local ( ) or user ( ) coordinate system.

Placing supports according to the member local coordinate system is very useful feature when working with sloping members. The visibility of the local coordinate system can be turned on with the *Visibility of the coordinate systems* option and also the name of the axes can be shown on the screen using the appropriate options of the *Visibility of labels* functions.

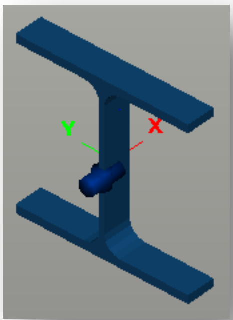
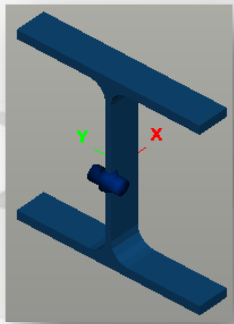
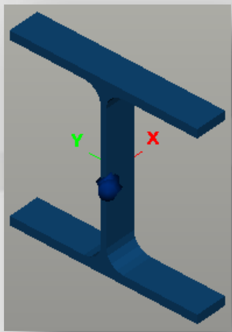
5.4.1 POINT SUPPORT

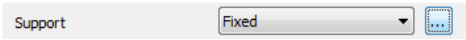


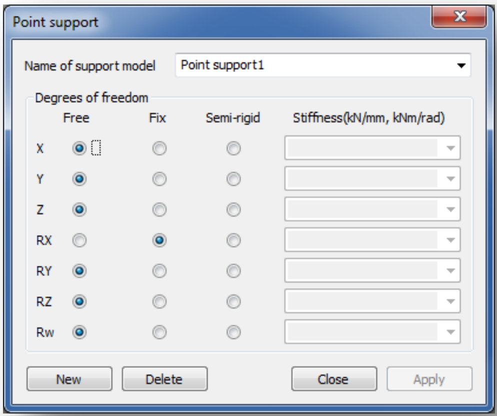


After selecting the coordinate system, the type of the support has to be selected from the list-box. There are several point supports types predefined.

By positioning, the used coordinate system determines the orientation of constrains of the support. Using the Global, Local or the *User coordinate system* the orientations of the constrains represented by the support will be different:

Global coordinate system	***Global coordinate system***	***User coordinate system***
		

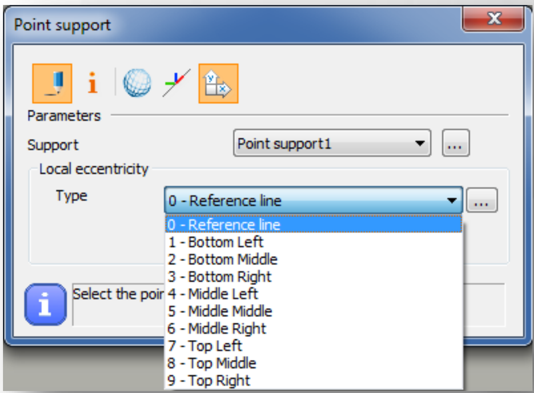
If any special supports are needed during the modeling process different from the predefined support types then click on the support definition icon (). Here you can define new supports. 7 DOFs can be set to free, fix or semi-rigid. For semi-rigid DOF the stiffness must also be set in kN/mm, or kNmm/rad.



The support type names can be clearly understood. For instance "x,y,z, xx" means any movement is fixed in *x*, *y* and *z* direction and the rotation around *x* axe is also fixed. All the rest DOFs (Degrees of Freedom) are free.

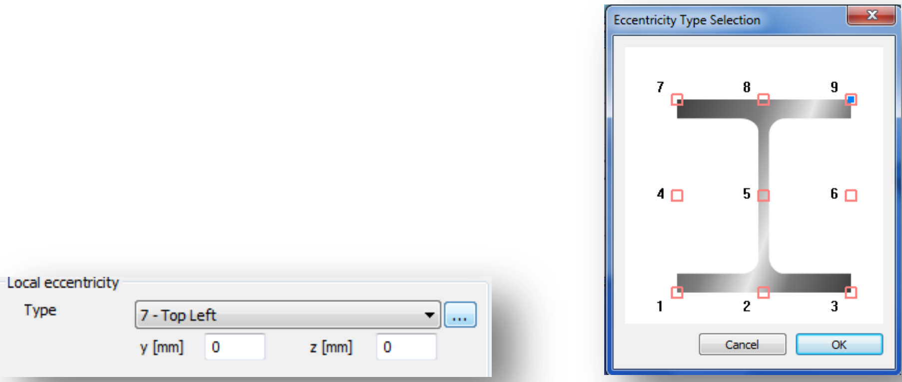
Local eccentricity can also be defined to the supports:

(This feature can be used for example for modeling the support effect of the bracing which is not connected to the reference line of the member but supporting the flange of the beam.)

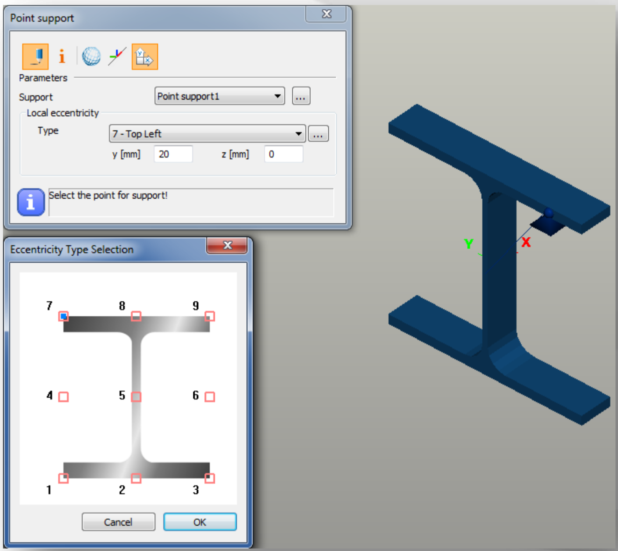


The eccentricity of the support can be defined **relative to the reference line** of a section, or **relative to the section geometry**. The *0 - Reference line* local eccentricity type can be selected only from this list-box.

You can specify the support eccentricity **relative to the section geometry** in two ways. The first way is: select one of the typical point of the section geometry (1-9) from the list, the second way is select one of the typical point (1-9) of the section relative position of the support by clicking the position on the graphical imitation of a section (click the small icon left to the list-box):



Giving value for the *y* or *z* parameters for local eccentricity these values will be added to the above selected position.



Changing the eccentricity of the supported object the new position of the support will be calculated accordingly.

Depending on the type of the selected local eccentricity used by placing the support the transformation rules are the following:

By selecting the *0 – Reference line eccentricity* type:

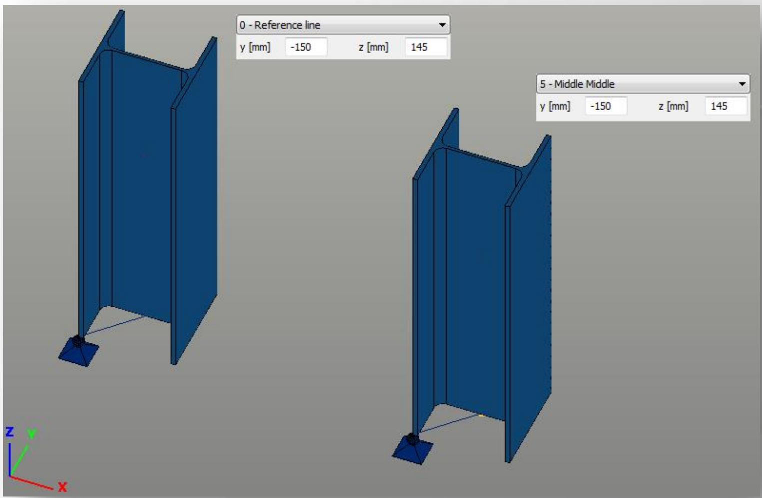
> Placing a support with this eccentricity type, changing the eccentricity of the supported object, the support will keep its position relative to the reference line, not to the section. By changing the "Rotation angle" attribute of the supported object, the reference line also rotates, and the position of the support will rotate too.

By selecting the 1 -9 eccentricity type:

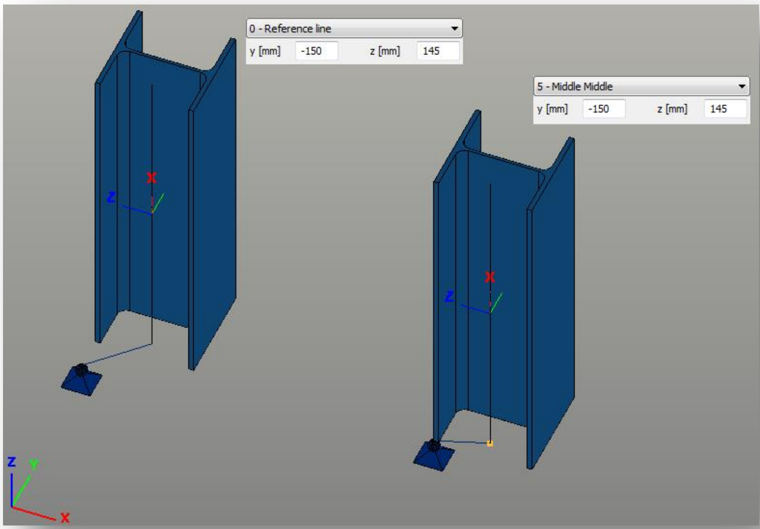
> Placing a support with any of these eccentricity types, changing the eccentricity of the supported object, the support will keep its position relative to the section. By changing the "Rotation angle" attribute of the supported object, the position of the support will rotate too.

Example for the different types of eccentricity:

Example 1: positioning



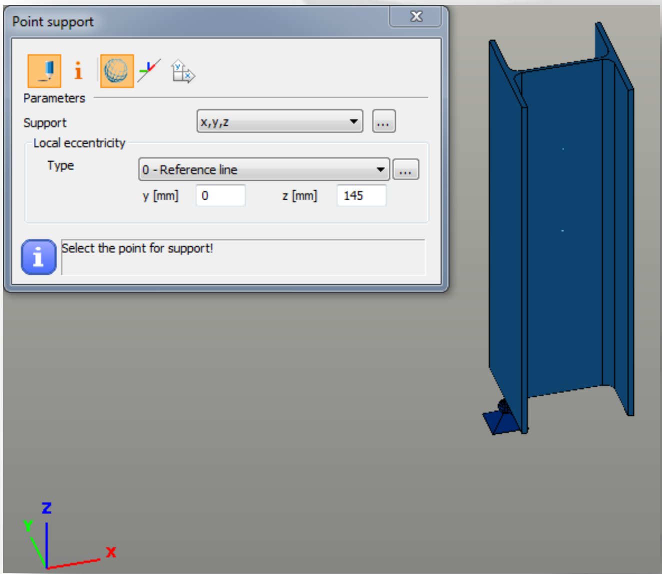
After changing the eccentricity of the beams y= 100 mm, the new positions of the supports are as follows:



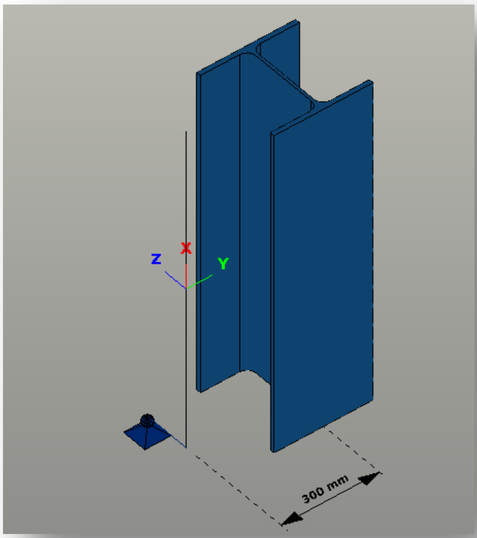
In case of the *0 – Reference line eccentricity* type, the support kept its position relative to the reference line. In case of the *5 – Middle Middle* eccentricity, the support moved with the point of the section.

Example 2

Positioning a support by selecting the global coordinate system, the eccentricity is 145mm from z direction of the local system of the supported column.

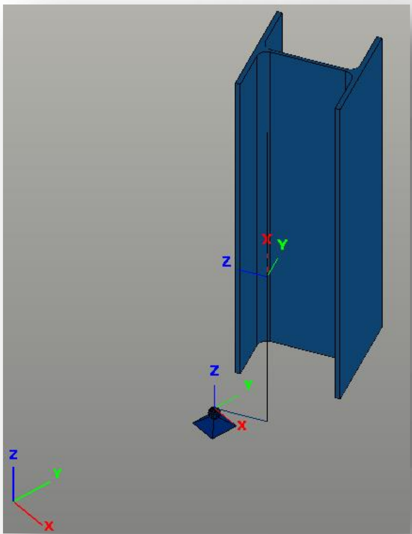


Changing the eccentricity of the column to y=300:



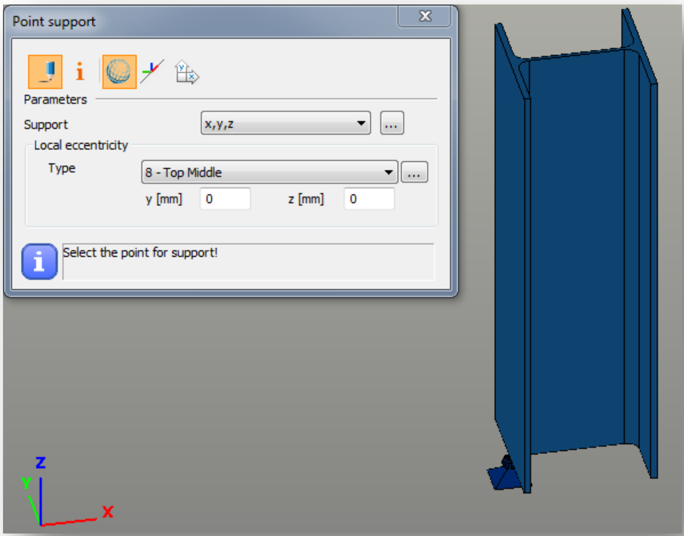
The column eccentricity from the centerline is 300 mm in Y direction. Position of the support was not changed. The support kept its original relative position to the centerline of the column.

Rotating the column with 30 degree:

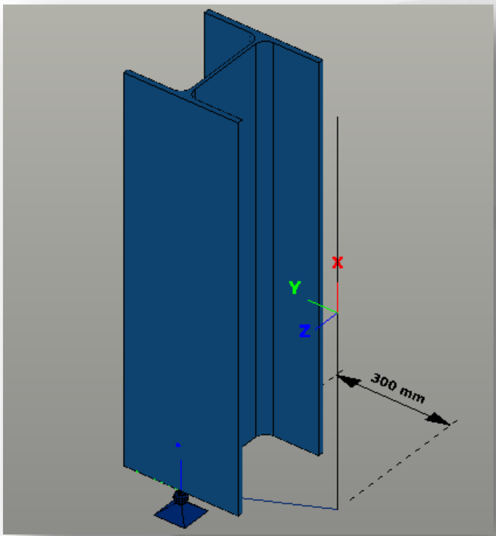


The support was rotated accordingly, but it kept the directions of its local axis parallel to the axis of the global coordinate system, which was selected by for positioning the support.

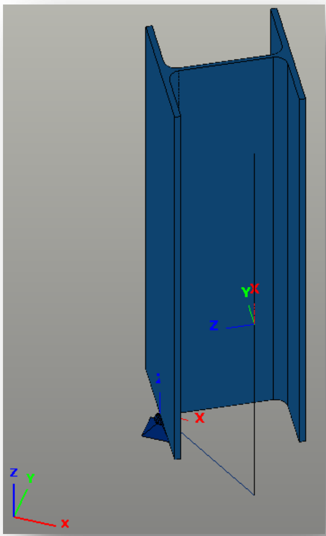
By selecting any of the 9 other type of eccentricity:



Changing the eccentricity of the column to y=300:



Rotating the column with 30 degree:

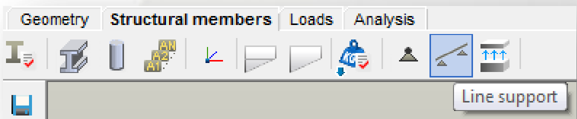


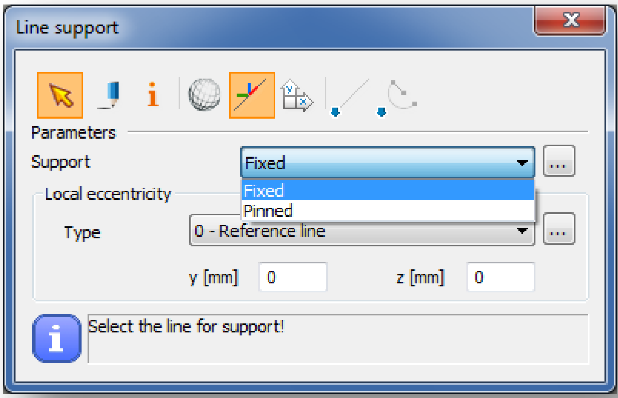
Selecting any of the above eccentricity types for positioning a support, changing the eccentricity of the supported object, the support will keep the originally given relative position from the new position of the supported object, and will keep the orientation defined by the applied coordinate system for placing this support.

WARNING!

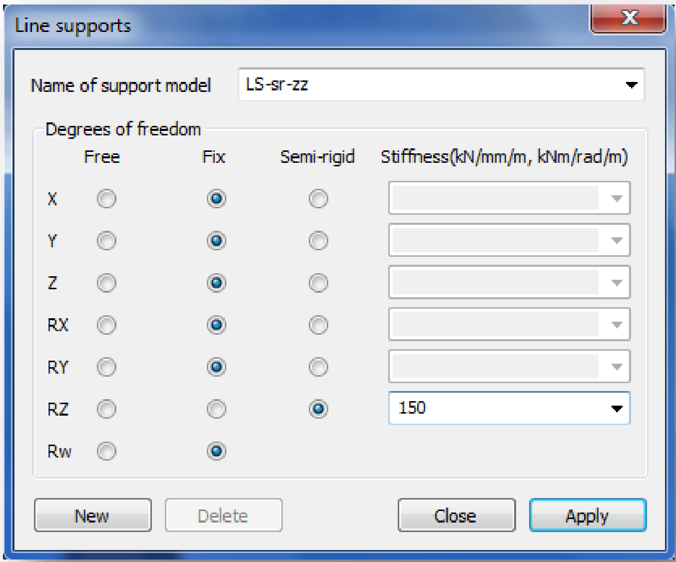
BE AWARE THAT SINCE THE FINITE ELEMENT FOR THE STEEL BEAMS AND COLUMNS HAS 7 DOFS THE POINT SUPPORT ALSO HAS 7 DOFS, THE 7^{TH} DOF REPRESENTS THE WARPING OF THE CROSS SECTION. ACCORDINGLY IF A CROSS SECTION ON A CERTAIN PLACE IS CONSIDERED TO FIXED FOR WARPING (FOR INSTANCE IN HEAVILY STIFFENED JOINTS) THE 7^{TH} DOF SHOULD BE FIXED. IN JOINTS CONSISTING SEVERAL MEMBERS (ESPECIALLY WHEN MEMBER ECCENTRICITIES ARE PRESENT) IT IS RECOMMENDED TO APPLY WARPING SUPPORT!

5.4.2 LINE SUPPORT





There are two default types for line supports the *Fixed* and the *Pinned*. It is also possible to define a new line support by giving the attributes of constrains as it is needed.



By positioning a line support the effect of the applied coordinate system and local eccentricity type are the same as for the point support.

The select function :

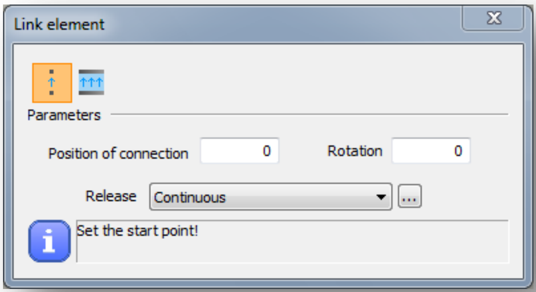
The line support will be placed to the selected edge of a surface (an edge of boundary or an edge of a hole), or to an axis of a beam.

The draw function :

The line support will be placed between the first and the second point defined by the draw function on an axis of a beam.

5.5 LINK ELEMENTS

Link element ( ) can be used to connect elements which are not directly connected to each other.



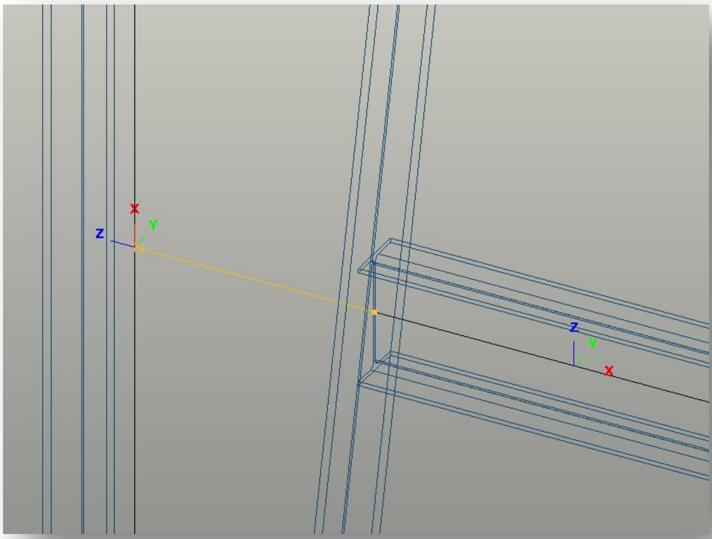
There are two different types of link element:

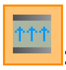
Connect two structural points :

For example if there is a cantilever on a relatively high tapered column (see image below) it is advisable to connect it not directly to the reference line of the column but with the link element. If it is connected the way like that the analysis and design results are more accurate and the modeling is more exact.

The position of connection parameter is a value between 0 and 1, and defines the place where the continuity acts in the percentage of the length of the link element.

The rotation value defines the angle of the link element in degrees.



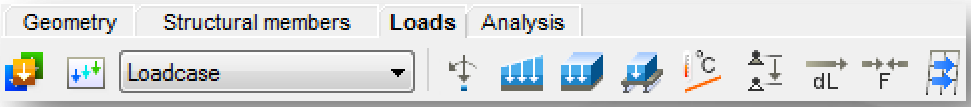
Connect two structural edges that are equal length and parallel **:**

This type of link element can be used to connect two equal length and parallel beams.

6 Structural loads

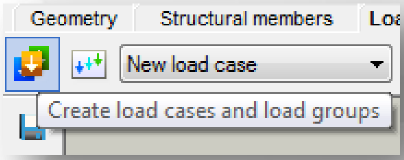
6.1 Basics

The definition of loading on a structural model is one of the most important modeling phases. Contrary to the modeling of structural members the load modeling is minutely controlled and supported by the structural codes and standards since the appropriate definition of loads ensures the major part in the reliability of structural performance. In *StabLab* several types of loading options help the engineer in this work. In accordance with the modeling of structural member the engineer can work with the load types and the applied loads are automatically converted into finite element loads for the calculation model. All the loading functionality connecting with the load modeling is placed on the ***Loads*** tab.



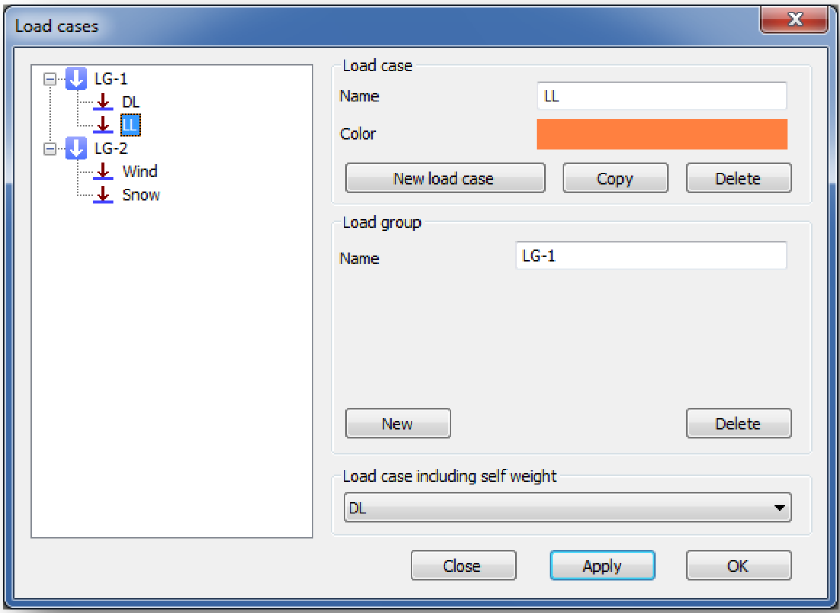
6.2 Load cases

Before place any loads in the model load cases must be defined. As a default, one load case has been defined in *StabLab* for quick calculations.



In a Load group several load cases can be defined. By clicking on the ***New*** button a new load group will appear.

By clicking on the ***New load case*** button, a new load case can be created. Name and the color of the load case can be set.



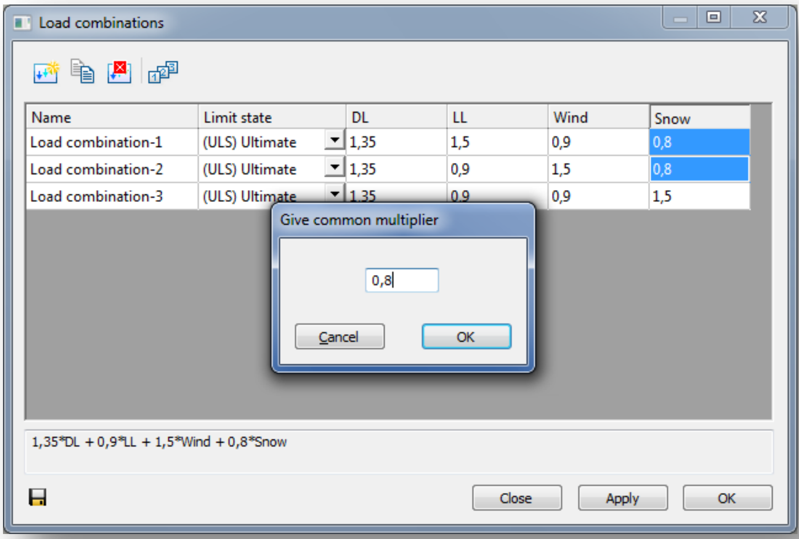
With drag-and-drop gesture load cases can be moved between the load groups.

StabLab can add the structure's dead load to one of the load cases if necessary. In order to do so please select the appropriate load case at the bottom right corner of the dialogue.

6.3 Load combinations

Load combinations can be set from load cases. Load combinations can be created manually by giving the safety and combination factors manually for each load cases. To do click on the ***Create new load combination*** button ().

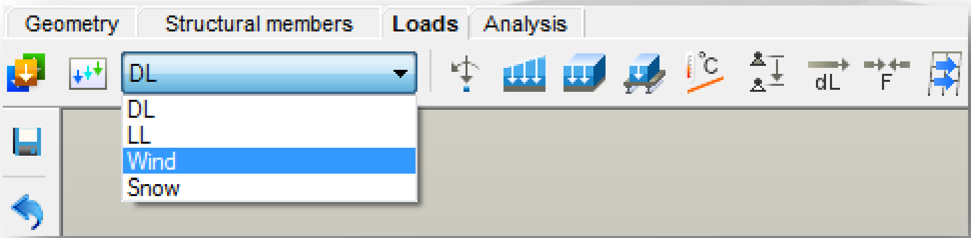
It is possible to edit multiple safety factors at a time: select safety factors and right click over it.



The selected load combination (multiple LC-s can be selected with using ***Ctrl*** or ***Shift*** buttons) can be copied ( ), deleted ( ) and renumbering ( ).

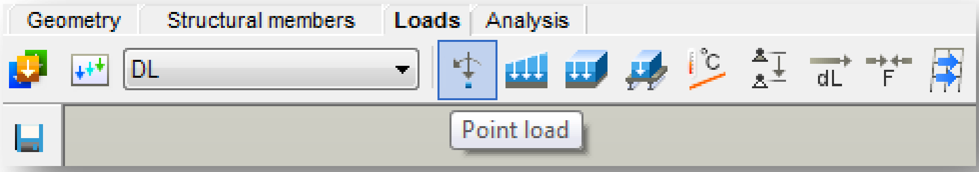
6.4 Load types

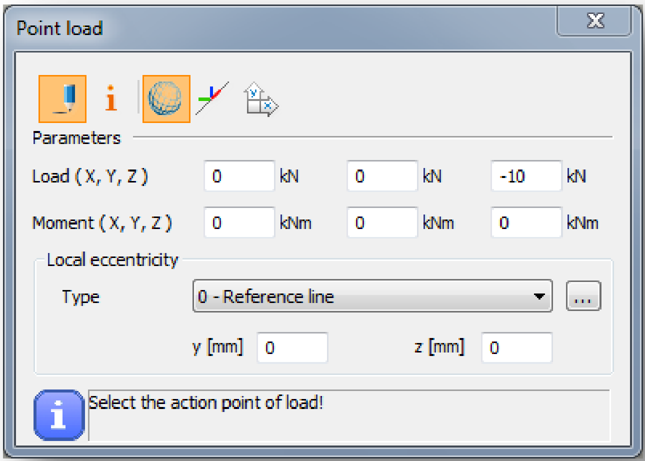
Any placed load will belong to the load case selected from the list:



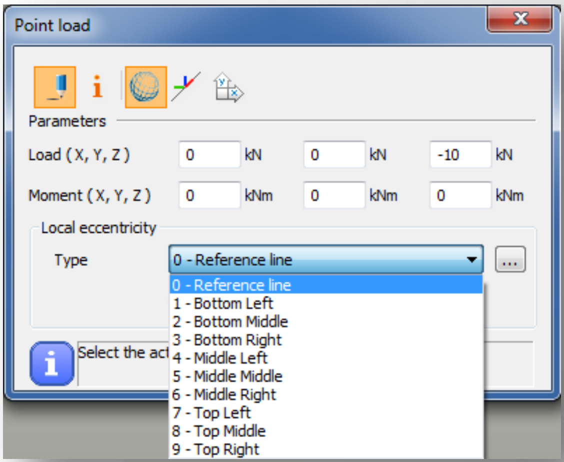
6.4.1 Point load

Point loads can be placed according to the global, local or the user defined coordinate system.



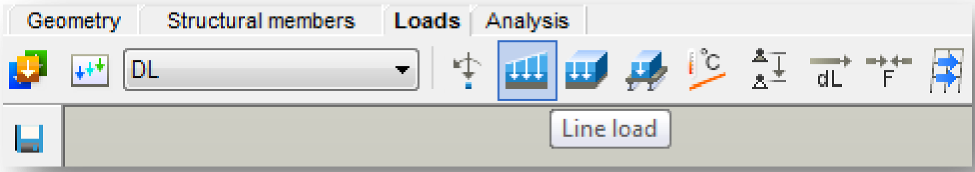


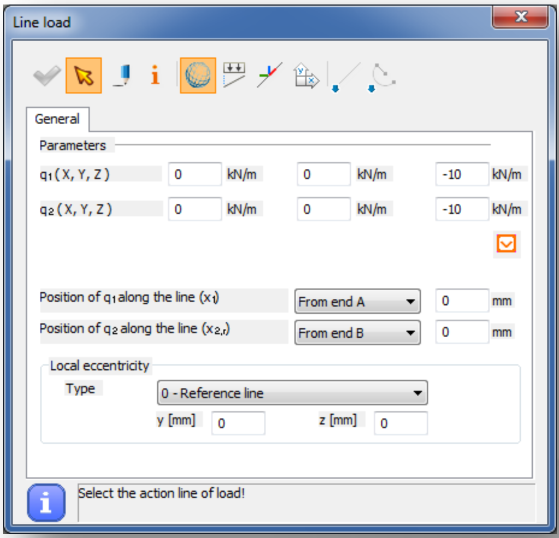
Eccentricity can be applied on the point load also. The effect of the selected coordinate system, the applied local eccentricity type and values are the same as point support (*5.4.1 Point support*).



6.4.2 Line load

Line loads can be placed along a member.



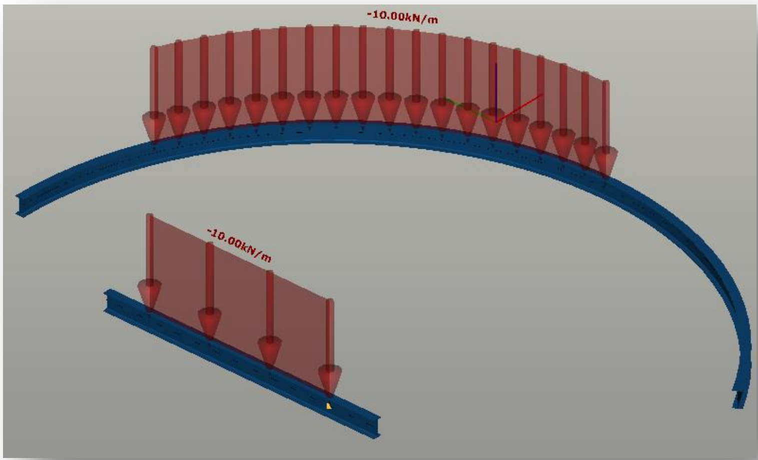


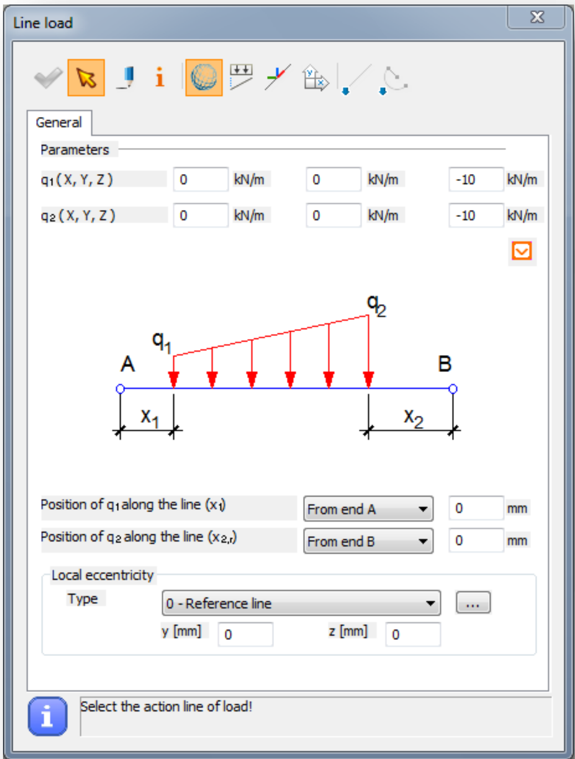
In order to place a line load click on the member using the *Select* function (). Line load can be applied to multiple members at the same time by using *Place loads* icon (). This case the members must be selected before clicking on the ***Line load*** icon on ***Loads*** tab.

The 6th icon from the left () is the ***Global projection system*** which is commonly used when modeling the snow load.

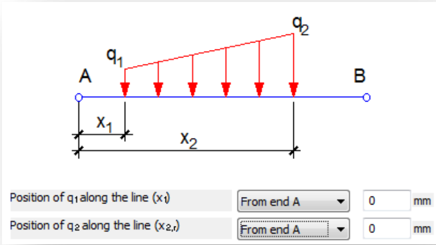
If just a partial line load need to be applied along the member then there are two ways to apply:

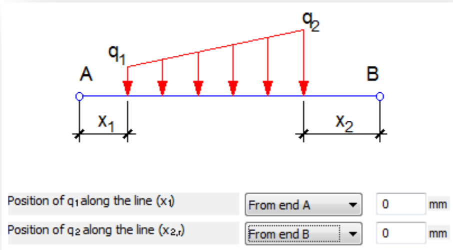
The first way is to use the draw function by selecting the icon and set the start point and the end point of the line load. This function can be used both for linear and curved members in accordance with the selected drawing function .

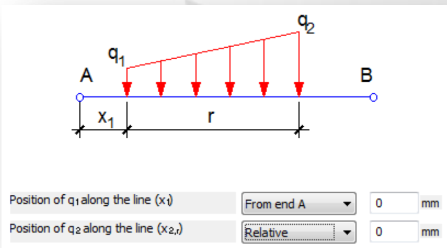




The second way is to give the exact distances of the start and end points of the line load from the member's end points. Using the setting of Position of q1, q2, there are 2x3 = 6 possible option to define a partial line load. The "Position of q1" set to "From end A" has 3 options, and setting it to "From end B" has also 3 options:





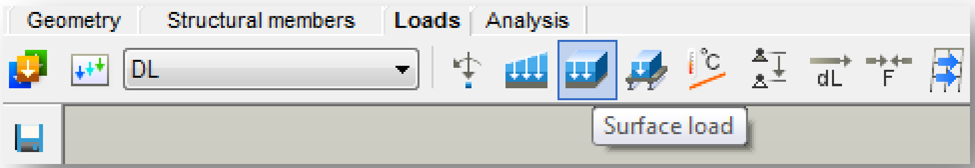


This is a flexible way for numerical input the line load positions.

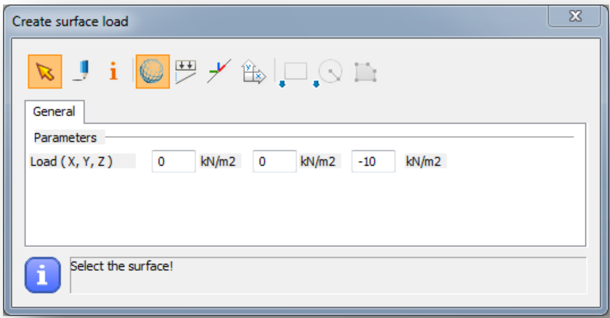
Eccentricity can be applied on the line load also. The effect of the selected coordinate system, the applied local eccentricity type and values are the same as point support (*5.4.1 POINT SUPPORT*).

By clicking the icon, the attributes of the selected line load fill out the parameters of the line load window, even if it was set by the draw function.

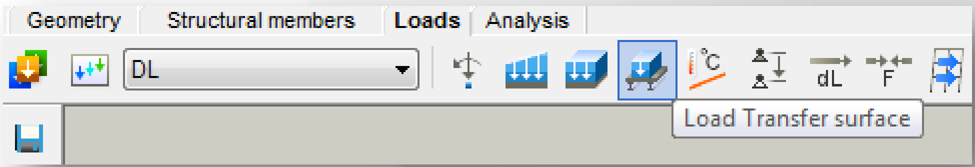
6.4.3 Surface load



Surface loads can be defined using the same method like in point or in line loads. No surface moment loads can be placed.

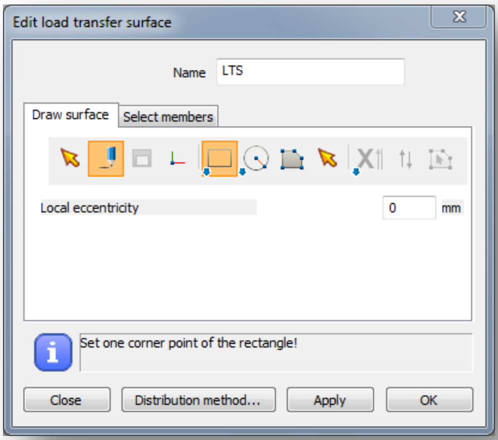


6.4.4 Load transfer surface

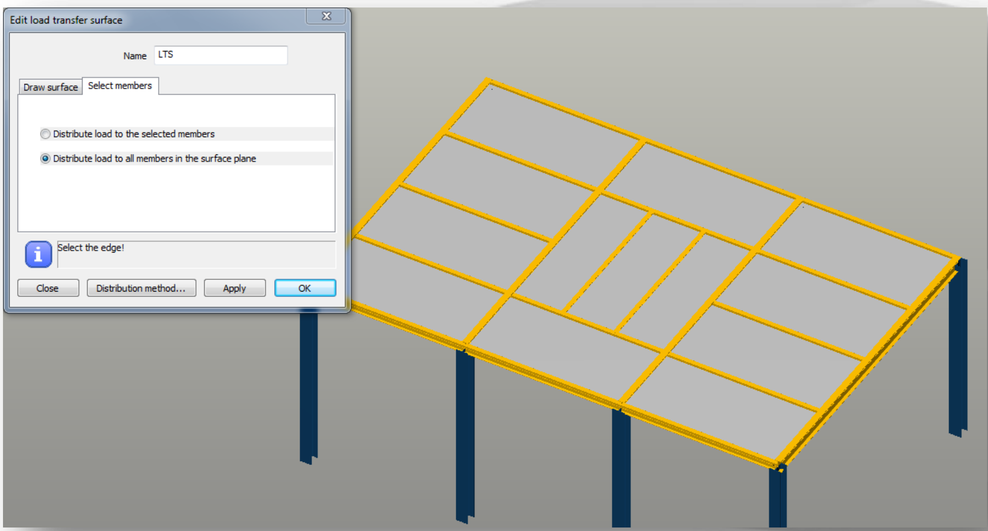


Load transfer surface is a special surface which distributes surface loads to members as line loads. This is very useful in those cases when surface load need to be distributed to members, like floor loads, snow and wind load, etc.

After clicking on the  icon on the *LOADS* tab a dialog window appears.



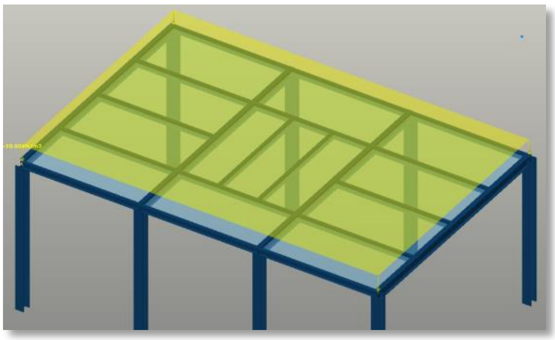
Just like in the two-dimensional figure drawing there are different possibilities to draw the surface: draw a rectangle, draw leaning rectangle, draw circle, draw polygon. After drawing the surface it is possible to select the members to which the surface distributes the load. There are two options: distribute load to the selected members or distribute load to all members covered by the surface in the surface plane. If second option is clicked then the appropriate members will be highlighted. If none of the options is clicked, then the second one will be applied as a default.

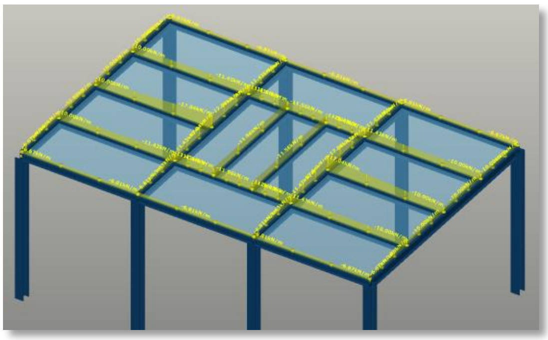


It is also possible to choose the first option and select members for carrying the surface load or remove members from the selection using the ***SHIFT*** + left click. After the corresponding members have been selected and ***OK*** is clicked then the surface is created.

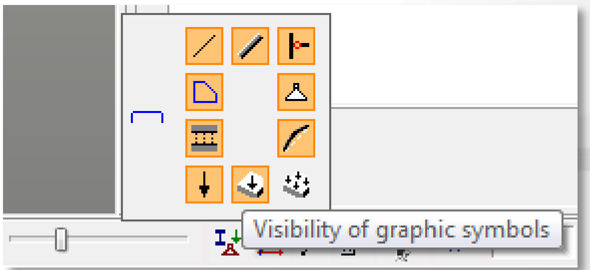
Surface load can be placed by using the method described in ***6.4.3 SURFACE LOAD***

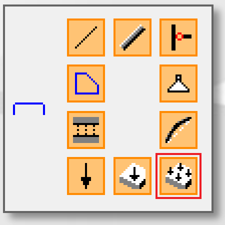
There are two visibility options: view the surface load or view the distributed load.





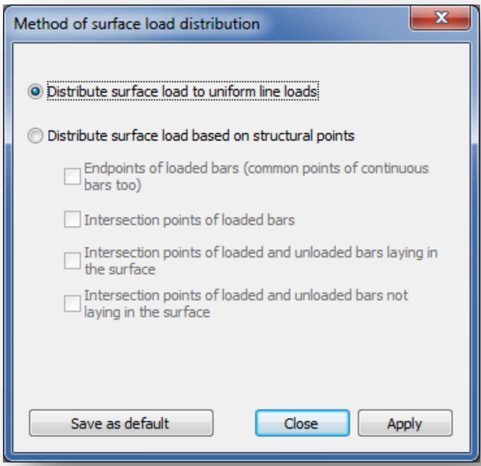
Changing the views is possible by clicking on the dedicated icon which can be found among the visibility of graphics symbols setting.



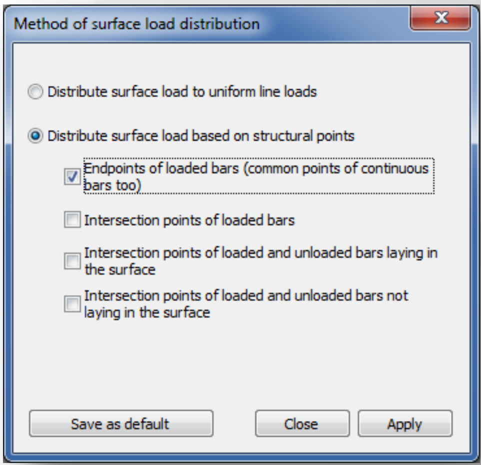


There are two possible methods to convert surface load to the selected members:

1. Converting surface loads to uniform line loads is performed by a meeting the following requirements:
    - the resultant force of all the line loads is the same as for the surface load
    - he line loads are constant on all the selected members



2. Converting the surface load to line loads based on structural points has the following background:
   (1) the surface load is first converted to concentrated point loads acting on the selected structural points using the Delaunay triangulation technique
   (2) the concentrated loads are then converted to line loads on the selected members meeting with the following requirements:
      - the resultant force of all the line loads is the same as for the surface load
      - the line loads are linear on all the selected members
      - the end value of the line loads on the selected structural points for all the selected members are equal

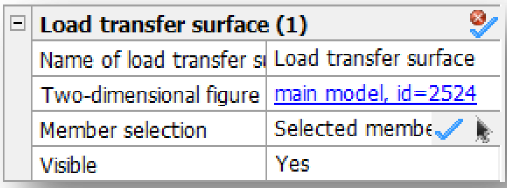


The basic working method is the following:

First create the load transfer surface. Select the members which it distributes load to, or accept the default setting which is distribute loads to all planar members. Apply surface loads to the surface at every load case where it is necessary.

6.4.4.1 Modifying load transfer surface

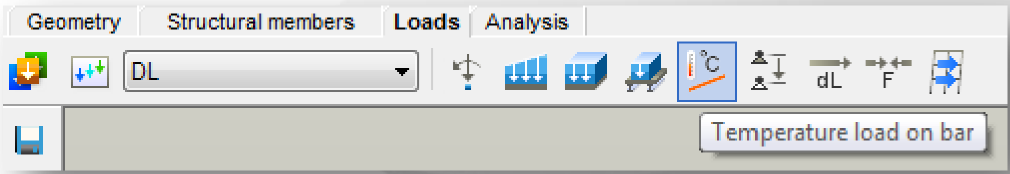
Load transfer surface can be modified by selecting it and changing the properties in the property bar. Member selection can be easily modified from the planar members to selected members. It is possible to highlight the previously selected members by clicking on the blue thick ( ).



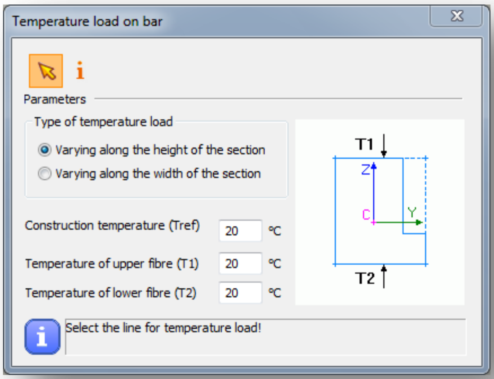
If selected members option is chosen and then the black arrow ( ) is clicked then previously selected members can be removed from the selection or new members can be added.

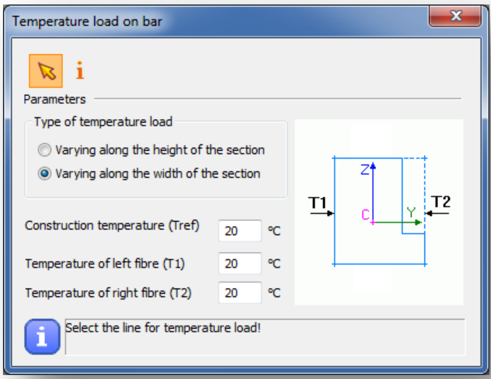
6.4.5 Temperature load

Temperature loads can be defined to members.



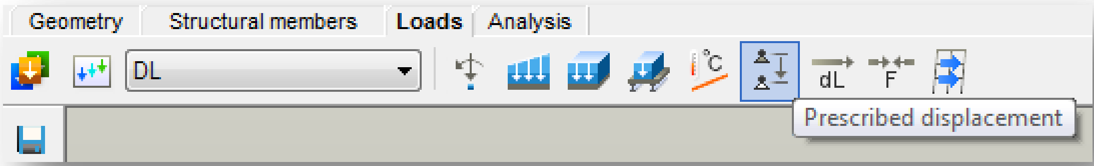
First choose the type of the temperature load: the temperature is changing along the cross section’s height or the width. Then add the construction temperature (reference temperature) and the upper and lower temperature of the cross section. The last step is to select the member where you would like to apply the load.



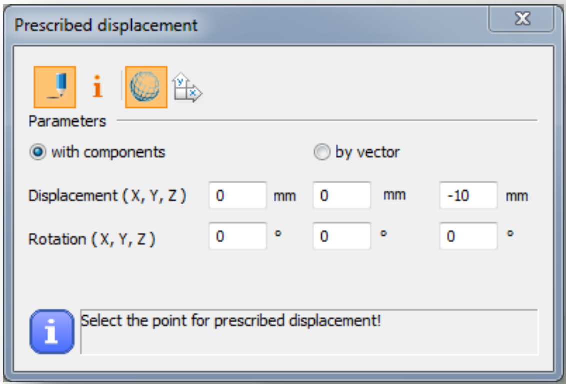


6.4.6 Prescribed displacement

Prescribed displacement can be applied for point supports.

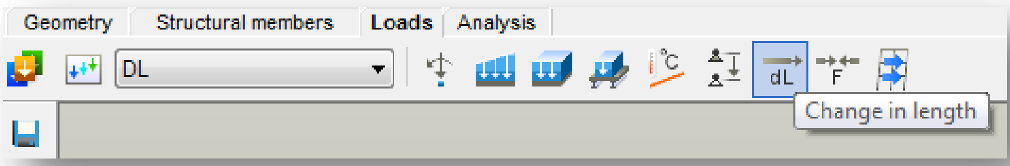


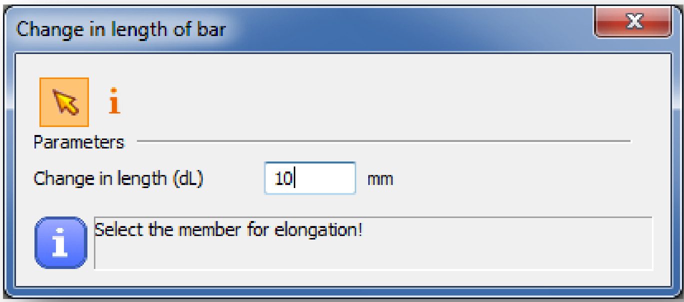
Prescribed displacement can be set with components or by vector.



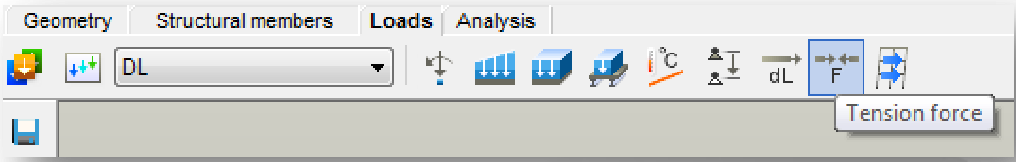
6.4.7 Prestres, prestrain

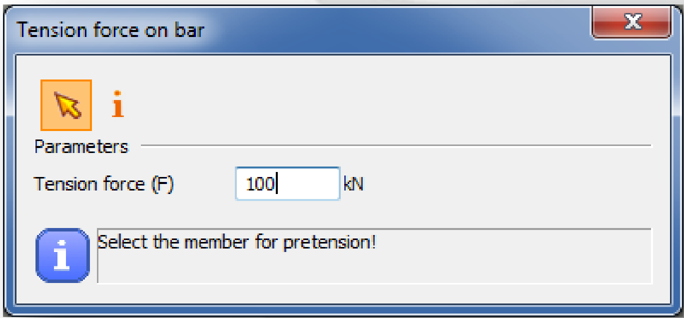
The change of the original member system length can be set and applied as a load.





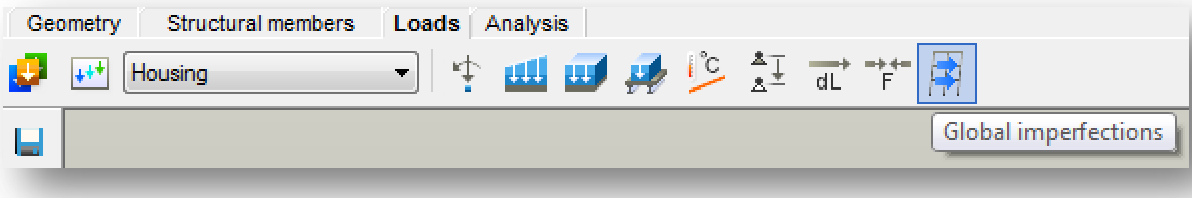
Tension force can be applied on bar members as a load.





6.5 Global imperfections

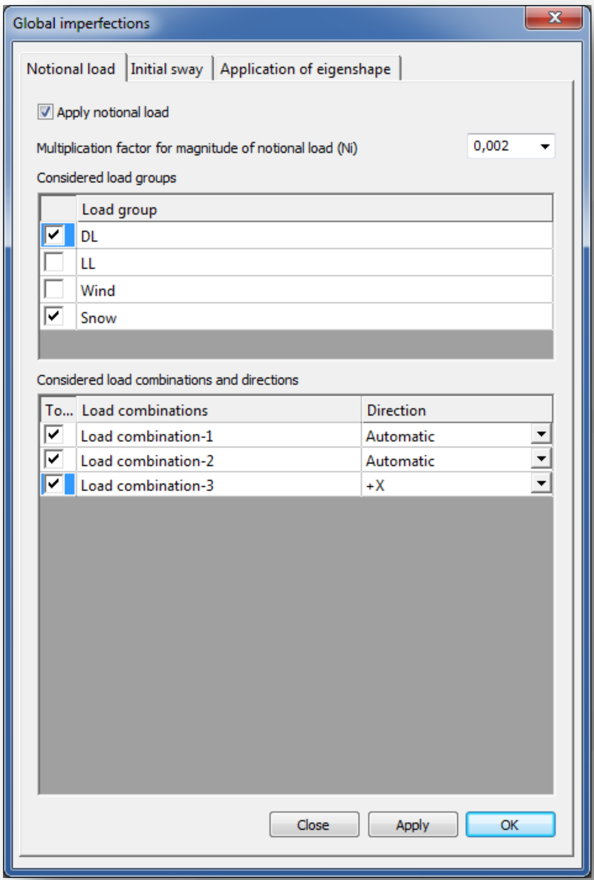
Three types of global imperfection can be applied on the whole model.



6.5.1 Notional load

Initial imperfections can be taken into account by the application of notional loads.

On the basis of the selected load groups, the notional loads will be generated automatically in the selected load combinations. Direction of the notional loads can be defined automatically or can be set for every load combination independently.



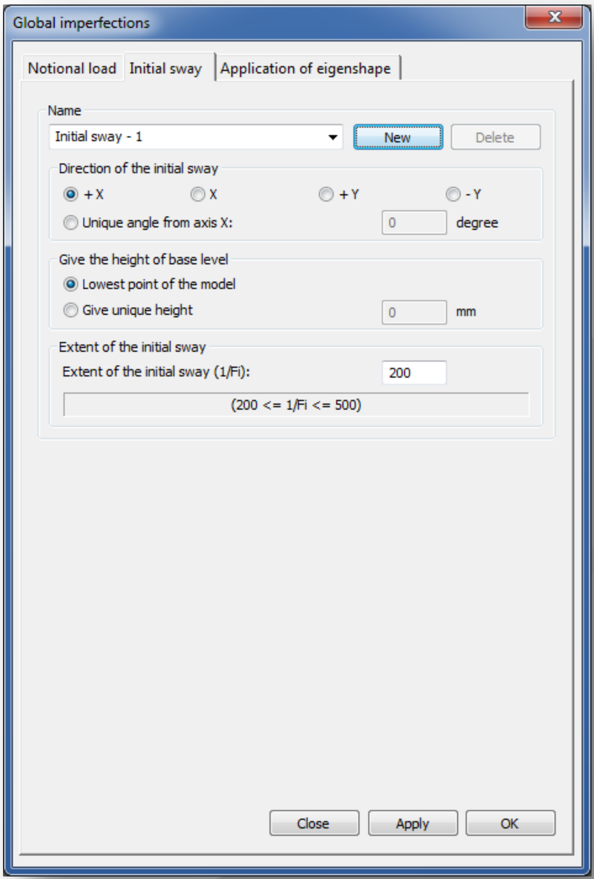
6.5.2 Initial sway

The finite element model is automatically modified according to the applied initial sway imperfection.

Direction of the initial sway can be set according to the horizontal global coordinate axis or with a unique angle from global axis X.

Initial sway can be applied from the lowest point of the model or from a given height.

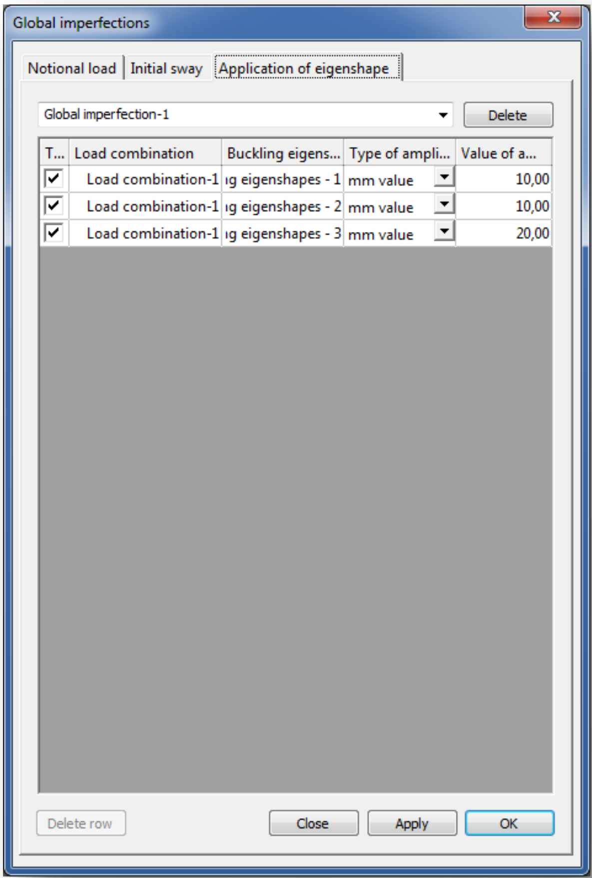
Extent of the initial sway (1/ϕ) must be between 200 and 500. Φ is in radian.



6.5.3 Aplication of eigenshape

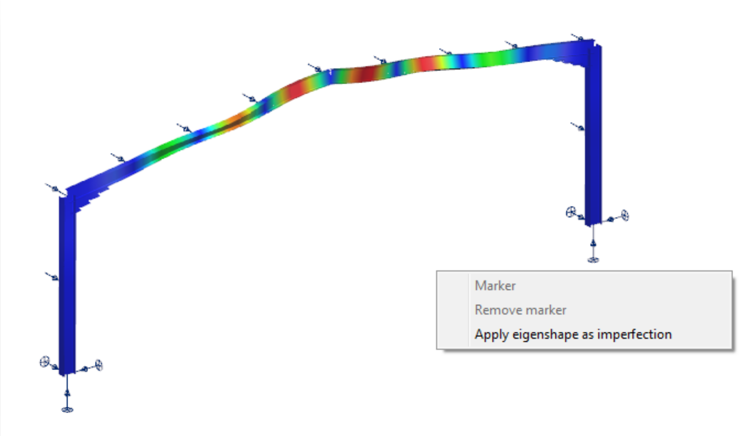
Selected eigenshape(s) can be applied as a global imperfection. The finite element model is automatically modified according to the applied eigenshape(s).

On the ***Global imperfections*** dialog the selected eigenshape(s) can be turn on/off with the checkbox(s) and the value of amplitude can be also modified.

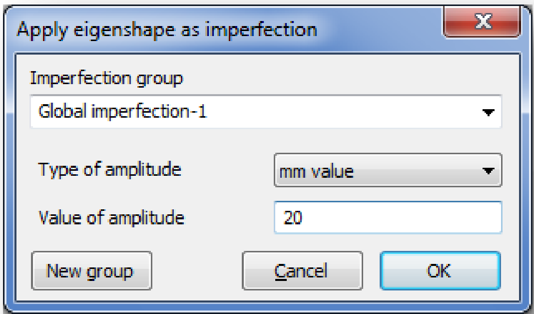


6.5.3.1 Selection of eigenshape(s)

Eigenshape(s) can be selected for global imperfection from Buckling results on the ***Analysis*** tab. To select a buckling shape press right click anywhere on the graphical window and click on the Apply eigenshape as imperfection function of the dropdown menu.



Every selected eigenshapes are arranged to an imperfection group. Groups can contain one or more eigenshapes. New group can be created with the ***NEW GROUP*** button.



Value of the amplitude must be given. It can be given in two ways:

- Maximum value of amplitude can be given in millimeter. Other values are interpolated
- With the multiplication factor of the calculated normalized eigenshape values (these values can be seen on the graphical window or in the result tables)

7 Structural analysis

7.1 Basics

The analysis of the structural model can be the most “black box” type phase of the design process for the engineer, and additionally the modern structural standards usually define the appropriate and required analysis type for the used design formulas. The StabLab applies the finite element method for all the calculations using two beam-column element types (traditional 12 DOF Timoshenko and 14 DOF thin-walled including warping of the section). The great variety of calculation possibilities includes first and complete second order (not only P-δ effect) analysis; flexural-, torsional-, and lateral-torsional buckling solutions; and static eigenvalue analysis. The exceptionally fast and robust equation solvers yield optimal calculation time even for unusually huge models. The functionalities connected with the structural analysis are placed on the ***Analysis*** tab. In the first step of the analysis the finite element mesh generation is automatically performed.

7.2 Finite elements

7.2.1 Basics

The mechanical performance of the calculated model is always highly influenced and limited by the applied finite element type. The considered displacements, forces, shape functions, second order effects within the finite element model basically determine the quality of the analysis, the expectable results and accordingly the applicable standard verification methods. On the other hand the engineer should be aware of the important characteristics of the used finite elements already at the model building phase in order to avoid the non-expected structural behavior and calculate the mechanical performance of the imagined structure as accurate as possible. In StabLab all the finite elements model always have the real 3D behavior; there are no options for special reduced degree of freedom calculations (plane frames etc.) if it is required appropriate support and/or load system should be applied. All the defined loads and supports are converted to nodal forces by load cases and nodal supports, no forces or supports are considered along the finite elements, this feature is taken into account during the automatic FE mesh generation.

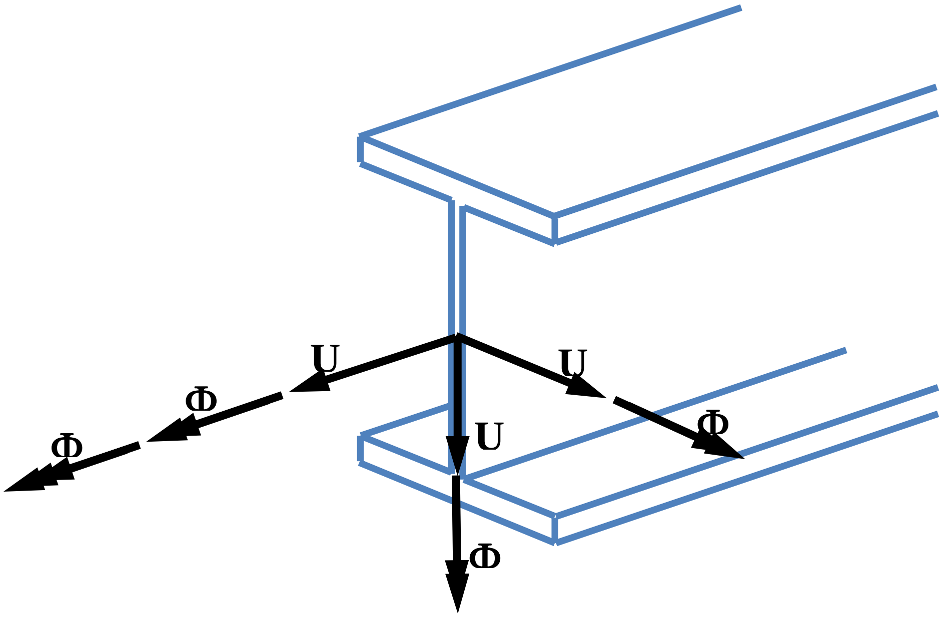
The subsequent sections do not intend to introduce the deep theory behind the applied finite elements – it can be found in the literature – only the most significant features are presented and explained which are important to know for the appropriate interpretation of the results.

7.2.2 Line elements

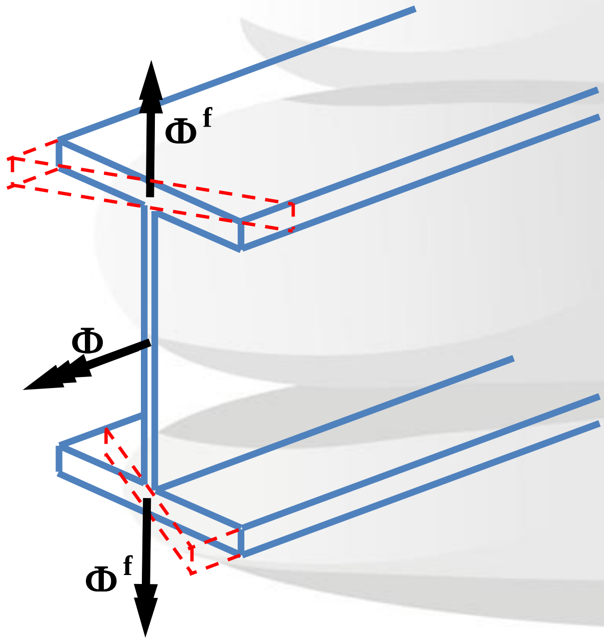
There are three options for line elements:

- a 6 degrees-of-freedom (6DOF) general beam-column element for the bars subjected to axial force, shear force, bending moment, torsion or any interaction of these
- a 7 degrees-of-freedom (7DOF) beam-column with warping element for the bars subjected same as 6DOF element + warping effect
- a tension-only (TO) element for bars subjected by tension force

The 7DOF element is specially developed for thin-walled members where the warping of the cross-section is of high importance in the behavior, this effect is considered by the 7th DOF. In the following figure the considered nodal displacements are illustrated:



The first 6 DOF are the conventional displacements (U_x, U_y, U_z) and rotations (Φ_x, Φ_y, Φ_z) according to the local coordinate system of the member (see section ***4.2 Coordinate systems***). What needs more explanation is the 7th DOF which is mathematically the first derivative of the twist about the longitudinal axis (Φ'_x); mechanically it represents the warping of the section which is straight consequence of torsion on thin-walled members. The next figure illustrates the warping effect of I shaped cross-section when the flanges step out of the original plane of the section.



In this case the warping DOF can be considered as a dual and opposite rotation of the flanges about the axis perpendicular to their width (in this case the local axis “z”).

Since steel members are usually relatively slender various modes of global stability failure can occur: flexural buckling, torsional buckling, lateral-torsional buckling and any interactions of these, all modes can be calculated by the 7DOF element. It is a quite important and advantageous feature in the stability design of these members but since the accurate calculation of all the torsional modes is highly dependent on the 7th warping DOF it is essential to consider this effect already in the modeling phase (see chapter ***5 Structural modeling*** for the modeling warnings).

The TO elements have only 1 DOF which is the longitudinal displacement (U_x). These elements are considered in the calculation only if they are subjected to tension accordingly the calculations are iterative in nature. At first an analysis is performed considering all the TO elements modeled by one finite element. Next a force check is executed on the TO elements and the compressed ones are neglected from the model and a new calculation is initiated. It is continued until all the TO elements has tension. This model configuration is taken into account in the eigenvalue calculations.

Since the eigenvalue calculations can not be performed iteratively it may happen that in the eigenshape the TO element gets compression i.e. its length shortens. In this case if this effect is significant and should be avoided a new eigenvalue analysis should be run without the TO element.

7.3 Model check (diagnostics)

In *StabLab* there is a possibility to perform a model check previous to executing any calculations. This function automatically runs before starting the finite element mesh generation or analysis but can be initiated any time (switching the **Diagnostics** on the ***View*** menu and run) examining the recent state of the model. There are two kinds of diagnostics messages:

- **ERROR:** the errors make the calculations impossible or meaningless to execute so the detected errors stop further calculations
- **WARNING:** the warnings allow the calculations but notice the possible errors

Part of the model checks is performed on the user model (basic check), these are basic requirements for the normal performance of a model; the other part is performed on the generated finite element model (pre-calculation check).

The following basic checks are performed:

- existence of load on the structure
- existence of support on the structure
- length of bars, line loads and line supports
- overlap, length and compatibility of haunches
- multiple supports on the same place
- compatibility of tension bars

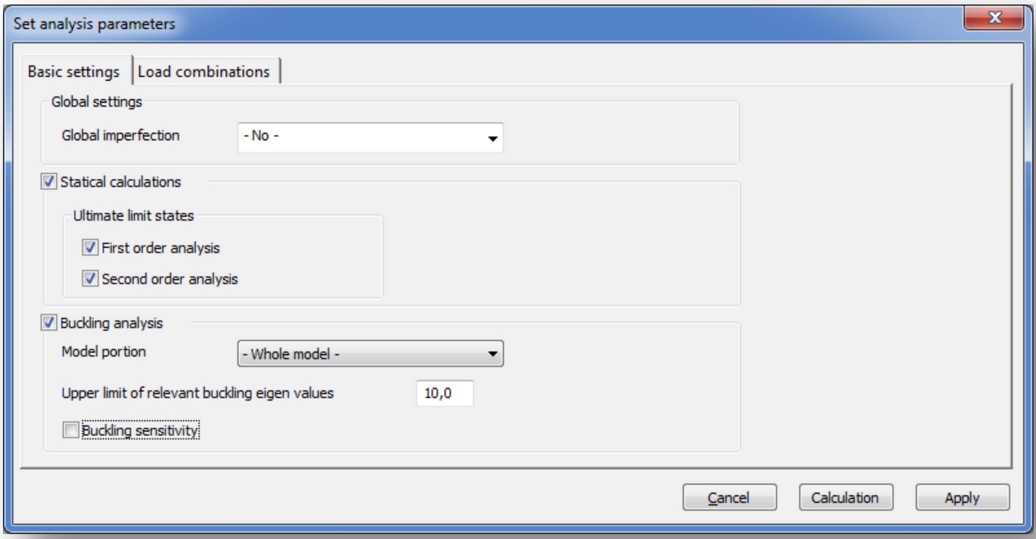
The following pre-calculation checks are performed:

- overhang of line loads and line supports
- point loads and point supports are not on the model
- overlap of bar members
- very small distance (< 5 mm) between points or lines of bars, loads or supports (the limit distance can be set in the **OPTIONS** menu)

The object which the errors or warnings are detected on can be selected and deleted from the diagnostics results table (in the middle of the right tables).

7.4 ANALYSIS TYPES

The required analysis types can be set on the **ANALYSIS PARAMETERS** panel. All types can be run for all the finite elements. Basically the analysis types can be defined for the existing load combinations.



By using the first tab it is possible to set analysis parameters for all load combinations at the same time.

Global settings and analysis types are separated in three groups.

In the first group global imperfection can be set for the whole model (see section **6.5 GLOBAL IMPERFECTIONS**).

In the second group the types of the statical calculation can be set.

In the last group buckling analysis and parameters can be set. If buckling analysis is clicked by default 10 eigenvalues are calculated. The upper limit of relevant buckling eigen values can be given. The buckling analysis can be performed for the whole model or for a model portion.

For each load combination unique settings can be set on the second tab. Load combinations can be turned off or type of analysis can be set.

7.4.1 First order

The first order analysis calculates the structural response considering the initial stiffness of the model. It is advisable to run a first order analysis in order to check the model performance before executing more costly calculations (for instance eigenvalue analysis). The steps of the first order analysis are the following:

1. Calculation of first order (initial) stiffness matrices ($\mathbf{K}^{el}{}_{s}$) of the finite elements in their local coordinate system
2. Compiling the global stiffness matrix ($\mathbf{K}_{s}$) and nodal force vector ($\mathbf{P}$) of the whole model by transforming the element stiffness matrices into the global coordinate system
3. Modifying the global stiffness matrix and nodal force vector considering the special boundary conditions (supports, continuity releases between the elements, prescribed displacements, temperature loads etc.)
4. Solution of the basic linear system of equations which writes the relationship between the applied nodal forces – known variables generated from the loads – and the nodal displacements – unknown variables ($\mathbf{U}$) – in the global coordinate system for the global model (see section *7.2 Finite elements* for the interpretation of nodal displacements and forces):

$$\mathbf{K}_s\mathbf{U} = \mathbf{P} \quad \rightarrow \quad \mathbf{U} = \mathbf{K}_s^{-1}\mathbf{P} \tag{1}$$

5. Calculation of the internal forces (and stresses) of the elements ($\mathbf{f}^{el}$) in their local coordinate system by transforming the global nodal displacements of the element into local system ($\mathbf{u}^{el}$) using their local stiffness matrices:

$$\mathbf{f}^{el} = \mathbf{K}_s^{el}\mathbf{u}^{el} \tag{2}$$

7.4.2 Second order

The second order calculations take into account that the loaded and deformed structure can behave differently than the initial configuration. This effect can be considered as if the initial stiffness was changing during the loading history. The steps of the second order analysis are the following:

- Performing the whole first order analysis as described in the previous section
- Calculation of geometric stiffness matrices ($\mathbf{K}^{el}{}_{g}$) of the finite elements in their local coordinate system by the internal forces of the elements ($\mathbf{f}^{el}$)

- Compiling the second order global stiffness matrix ($\mathbf{K_s}$ + $\mathbf{K_g}$) and nodal force vector (**P**) of the whole model by transforming the element stiffness matrices into the global coordinate system
- Modifying the second order global stiffness matrix and nodal force vector considering the special boundary conditions (supports, continuity releases between the elements, prescribed displacements, temperature loads etc.)
- Solution of the basic linear system of equations which writes the relationship between the applied nodal forces – known variables generated from the loads – and the nodal displacements – unknown variables (**U**) – in the global coordinate system for the loaded and deformed global model (see section *7.2 Finite elements* for the interpretation of nodal displacements and forces):

$$\left(\mathbf{K}_s + \mathbf{K}_g\right)\mathbf{U} = \mathbf{P} \quad \rightarrow \quad \mathbf{U} = \left(\mathbf{K}_s + \mathbf{K}_g\right)^{-1}\mathbf{P} \qquad (3)$$

- Calculation of the internal forces (and stresses) of the elements (**fel**) in their local coordinate system by transforming the global nodal displacements of the element into local system (**uel**) using their local stiffness matrices:

$$\mathbf{f}^{el} = \mathbf{K}_s^{el}\mathbf{u}^{el} \qquad (4)$$

- If the difference between the new nodal displacements and the ones obtained earlier exceeds a certain limit repeat the calculations from step 2

7.4.3 Statical eigenvalue – buckling analysis

In a mechanical interpretation the eigenvalue analysis approximates the elastic critical load levels where the structure is subjected to some modes of loss of stability. Mathematically it means that the second order equation of (3) has no unique solution because the second order stiffness matrix is singular. In StabLab linear eigenvalue analysis is performed considering one parameter, conservative loading and that the geometric stiffness matrix depends linearly on the load factor (λ):

$$\mathbf{K}_g(\lambda \mathbf{f}) = \lambda \mathbf{K}_g(\mathbf{f}) \qquad (5)$$

In this case the eigenvalue analysis can be written in the following form:

$$(\mathbf{K}_s + \lambda \mathbf{K}_g)\mathbf{U} = 0 \qquad (6)$$

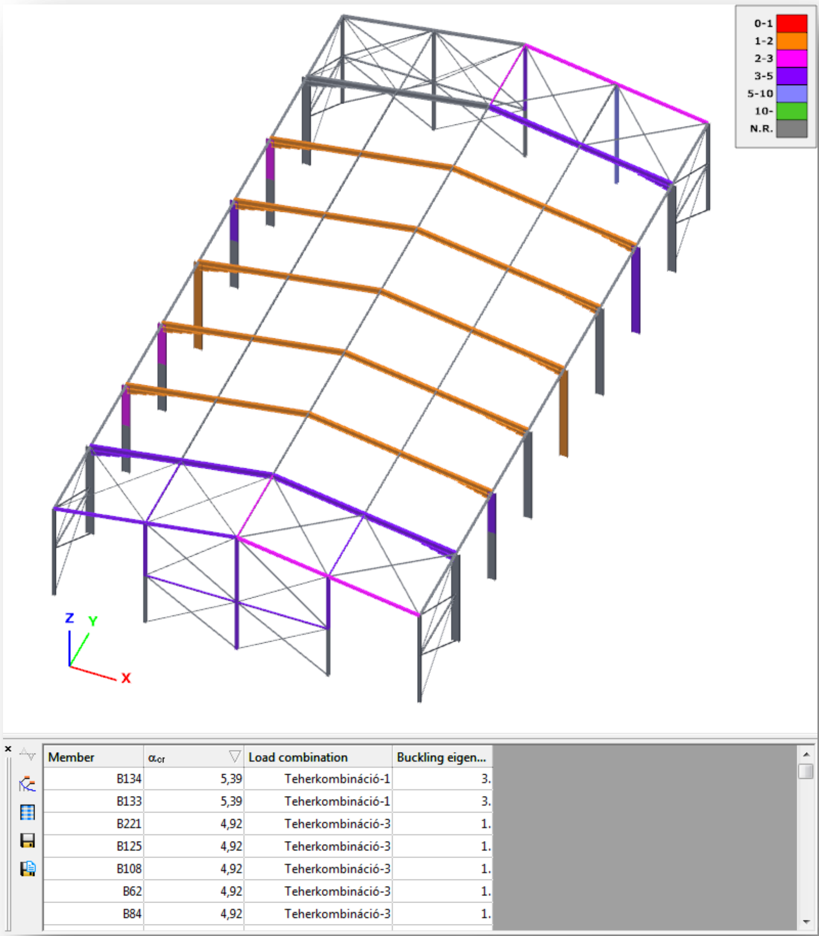
The solutions are certain critical load factors (buckling loads factor λ^{i}_{cr}) which make the second order stiffness matrix singular and corresponding displacements (buckling shapes $\mathbf{U}^{\mathbf{i}}$).

The possible buckling shapes which can be calculated by this eigenvalue analysis are basically influenced by the considered second order effects which are determined by the applied finite element. In case of beam-column structures the 7 DOF finite elements have the capability to consider all modes of global buckling shapes: flexural buckling, torsional buckling, lateral-torsional buckling and any interactions of these.

7.4.4 Buckling sensitivity

Buckling sensitivity analysis is a very useful function to assist the global stability design. Buckling sensitivity gives a review about the eigenshapes of the structure, and gives for every member the relevant eigenshape which is the best for the buckling analysis.

The summary view shows the crucial members for buckling considering all the available load combination and buckling modes by analyzing the bucking deformation energy accumulated in the certain members



Member	α_{cr}	Load combination	Buckling eigen...
B134	5,39	Teherkombináció-1	3.
B133	5,39	Teherkombináció-1	3.
B221	4,92	Teherkombináció-3	1.
B125	4,92	Teherkombináció-3	1.
B108	4,92	Teherkombináció-3	1.
B62	4,92	Teherkombináció-3	1.
B84	4,92	Teherkombináció-3	1.

7.5 Analysis results

7.5.1 Result types

For the line elements the following analysis results are available:

- First and second order calculations:
  - Deformations – on the finite element nodes in the global coordinate system
    - XYZ – all components
    - X – only displacements in the "X" direction
    - Y – only displacements in the "Y" direction
    - Z – only displacements in the "Z" direction
  - Equilibrium – signed summation of the internal nodal force components and the appropriate external force

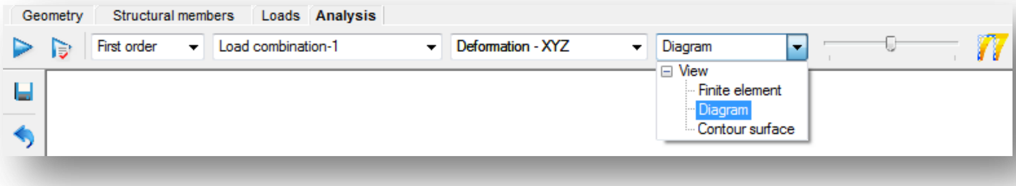
and reaction components, all the values should be zero which means the model is in equilibrium in the calculated deformed state

- Internal forces – on the end nodes of each finite element in the local coordinate system
  - N – axial force
  - Vy – shear force in the local “y” direction
  - Vz – shear force in the local “z” direction
  - Mx – torsional moment
  - My – bending moment about the local “y” axis
  - Mz – bending moment about the local “z” axis
  - B – bimoment
- Reactions – on the support nodes in the global coordinate system
  - R – all the reaction forces and moments
  - RR – all the reaction forces
  - RRR – all the reaction moments
  - Rx – reaction force in the in the global “X” direction
  - Ry – reaction force in the in the global “Y” direction
  - Rz – reaction force in the in the global “Z” direction
  - Rxx – reaction moment about the global “X” direction
  - Ryy – reaction moment about the global “Y” direction
  - Rzz – reaction moment about the global “Z” direction

- Static eigenvalues and corresponding eigenshapes – on the finite element nodes in the global coordinate system

7.5.2 Visualization options

The results of the performed analysis are usually a huge amount of data so the efficient handling of it has great importance. There should be opportunities for global overview and for obtaining accurate, detailed information about a certain part or problem. There are two main possibilities for the demonstration of the analysis results: the *graphical visualization* and the *result tables* (for the general functionality of tables used in *StabLab* see section ***1.5 General functions for tables***). The two visualization options are obviously in strong connection and can be manipulated by the functions placed on the upper part of the ***Analysis*** tab. The four combos contain the following selection options (from left to right):

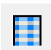


- selection of analysis type (discussed in section ***7.4 Analysis types***)
- selection of load combination or load case
- selection of result type (discussed in section ***7.5.1 Result types***)
- selection of type of view

All the model view options discussed in section ***3.2 Model views*** are applicable on the result graphics. Moving the slide beside the combos the scaling of the results can be adjusted on the graphics. In case of partial or submodel view the graphics and the tables show only the results of the actual model part. Three types of result tables can be viewed:

 Extreme values by members: select the maximum and minimum values for each structural members

 User defined values: show the values belonging to the markers defined by the user (discussed more deeply in section ***7.5.3 Result markers***)

 All values.

The result tables arrange the values according to the current result type:

- displacement type results (first or second order deformations, static eigenshapes): finite element node number, displacement components

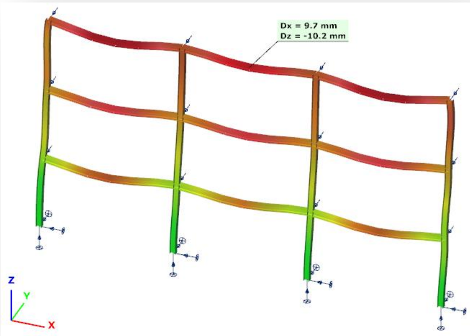
- force type results (internal forces): finite element node number, finite element number, force components
- reaction type results (reactions, equilibrium): finite element node number, reaction components.

7.5.3 Result markers

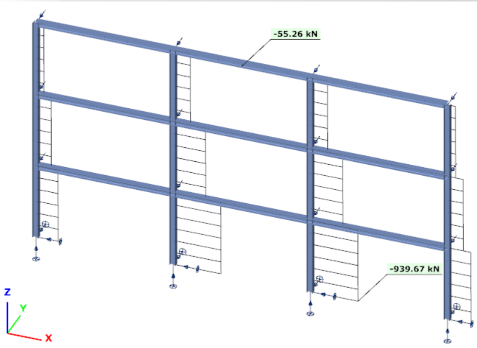
On the graphical interpretation of the results markers can be placed in order to show certain significant values. When moving the mouse along the structural elements (or more correctly the finite elements) the result markers continuously appear showing the actual values. These markers can be fixed by clicking on the right mouse button and choosing the *Marker* option.

The fixed markers will appear on every result views where it has interpretable value:

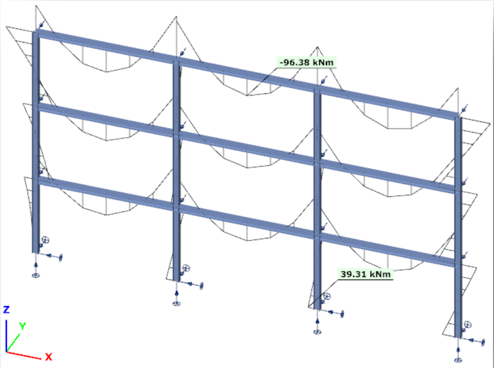
- Deformation



- Normal force



- Bending moment



When there are fixed markers on the model the *User defined values* table contains the appropriate (view dependent) values of the marked points. In this table the markers can be switched to disabled by uncheck the proper row.

☐	5	37	-60,130	0,000	4,900	0,000	-56,820	0,000	0,000
☑	4	56	-71,400	0,000	52,980	0,000	186,740	0,000	0,000
☑	3	9	-75,880	0,000	-78,680	0,000	221,200	0,000	0,000

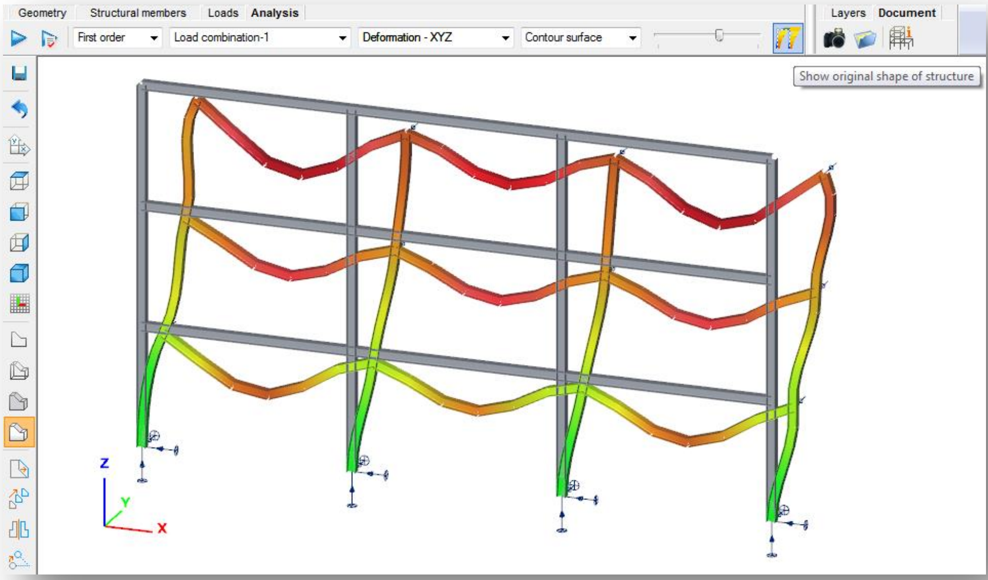
The markers can be deleted by:

- click on the proper row in the table by the right mouse button
- click on the marked point on the graphics by the right mouse button and chose *Remove marker* option

Extreme values can automatically marked by the check boxes on the *Extreme values* table rows.

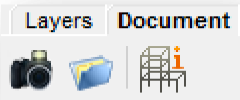
7.5.4 Show original shape of structure

Show original shape of structure function  will show the original shape of structure and the deformed structure at the same time:


Geometry
Structural members
Loads
Analysis
Layers
Document
First order
Load combination-1
Deformation - XYZ
Contour surface
Show original shape of structure
Z
Y
X

8 DOCUMENTATION

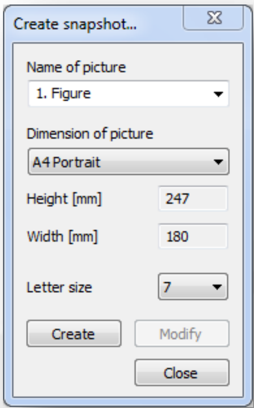
8.1 THE DOCUMENT TAB



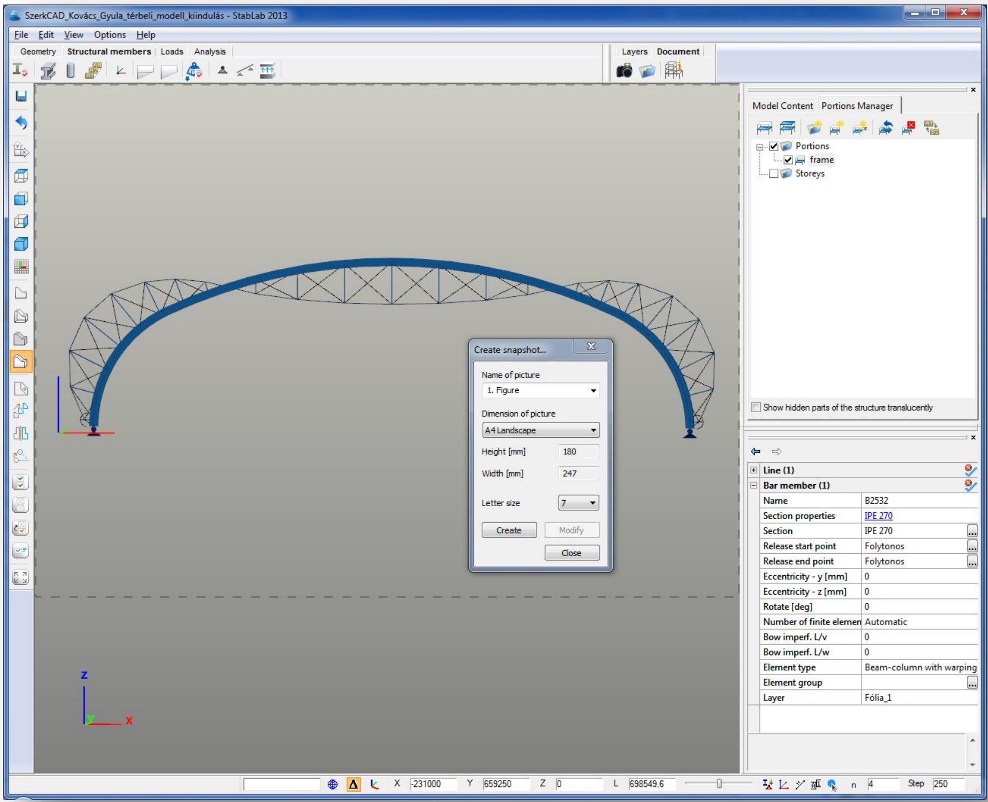
All icons which related to the documentation are placed in the ***DOCUMENT*** tab.

8.1.1 CREATING SNAPSHOTS

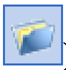
The first icon () on the ***DOCUMENT*** tab can be used to take a snapshot of the model.

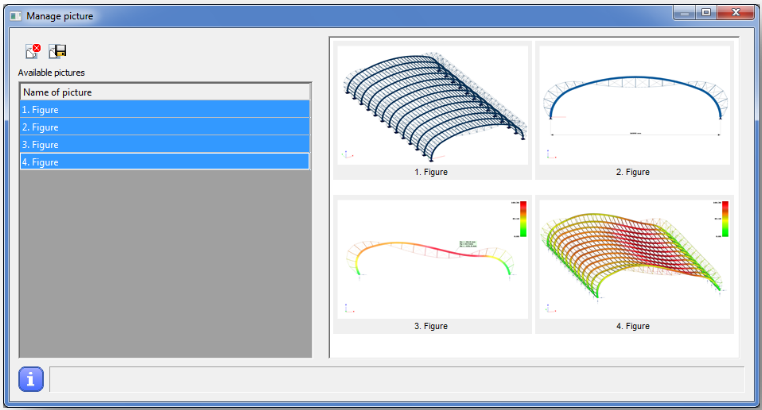


The size of the snapshot can be set and a dashed rectangle shows which part of the model would appear on the picture. The model can be moved or rotated to fit into the dashed rectangle.



8.1.2 Snapshot manager

The second icon on the documentation tab is the Snapshot manager () where the taken snapshots can be exported to a file or can be deleted from the model in order to reduce model file size. Multiple selected pictures can be saved and deleted at the same time. The name of the images can also be edited by double clicking the name of the image.



8.1.3 Model information

With using the third icon on the ***Document*** tab the model information can be viewed or can be exported to a file. The model information window shows the most important features of the model like Number of bars, Number of supports, Total weight of the structure, etc. The mass, surface and the length can be viewed for each section in the model. The model information can be shown based on the full model or on the current view if the model view shows only a part of the full model using the sub model view.