

SESAM TUTORIAL

GeniE

Modelling a Topside with Equipment

Valid from program version 8.2



Sesam Tutorial

GeniE – Modelling a Topside with Equipment

Date: June 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

E-mail support: software.support@dnv.com

E-mail sales: digital@dnv.com

© DNV AS. All rights reserved

This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

TABLE OF CONTENTS

1. Introduction	Page 4
2. The GeniE Graphical User Interface	Page 5
3. Create the Model	Page 22
1. Create Main Support Frame	Page 23
2. Add Plates	Page 26
3. Flush Stiffener Beams with Deck Plate	Page 27
4. Extend the Deck	Page 28
5. Create Sets	Page 31
6. Create the Upper Deck	Page 32
4. Add Boundary Conditions	Page 34
5. Create Loads	Page 35
1. Create and Place Equipments	Page 36
2. Import and Place Weight List	Page 40
3. Create Explicit Loads	Page 43
4. Create Load Combinations	Page 46
6. Create and Run an Analysis	Page 47
7. Present Results	Page 48

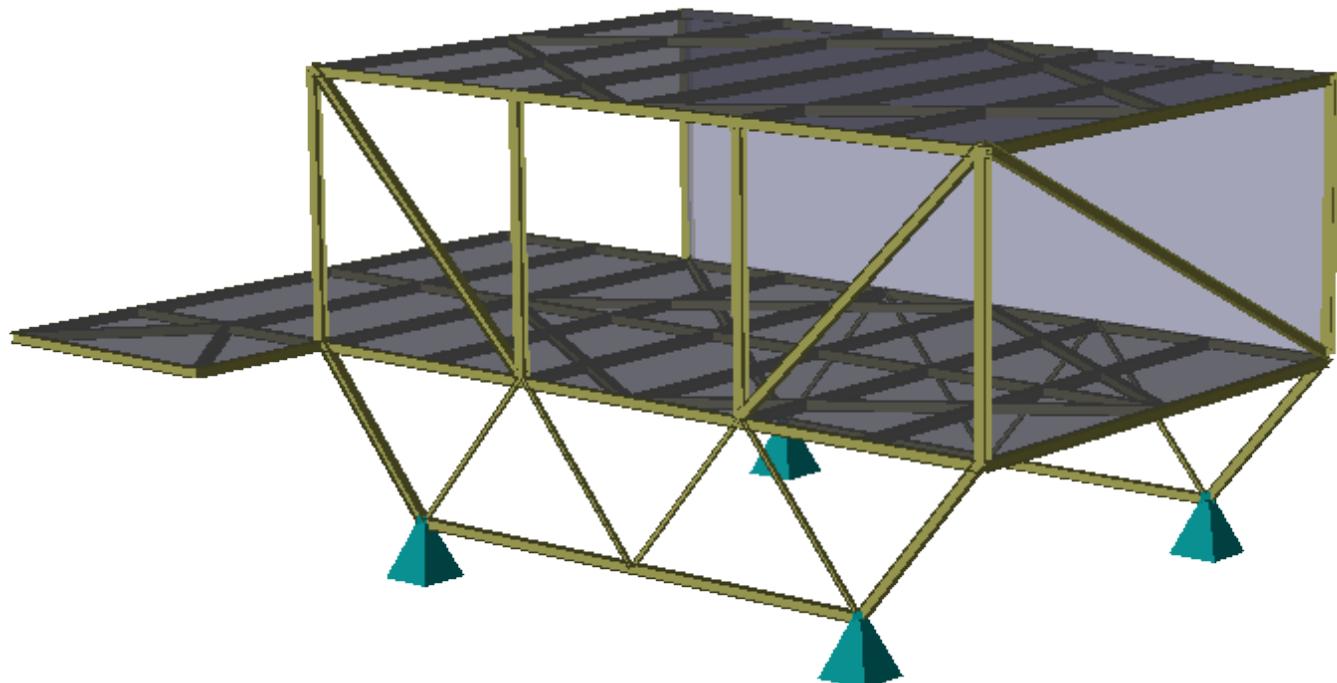
1 INTRODUCTION

➤ This tutorial consists of two parts:

- Chapter 2 The GeniE Graphical User Interface
explains the GeniE graphical user interface (GUI):
 - Graphics screen manipulations like rotating, zooming, etc.
 - The tabs of the *View Options* dialog determining what and how to display
 - Selecting objects and filtering display of objects
 - Labelling and colour coding objects in the display
 - Controlling naming of objects
 - Help pages
 - Logging commands and undo/redo
- Chapters 3 - 7
Model and analyse a topside structure with equipment including:
 - Creating guiding geometry
 - Modelling structure with boundary conditions, explicit loads and equipments
 - Running analysis and presenting results

➤ A GeniE input file for creating the complete model is provided.

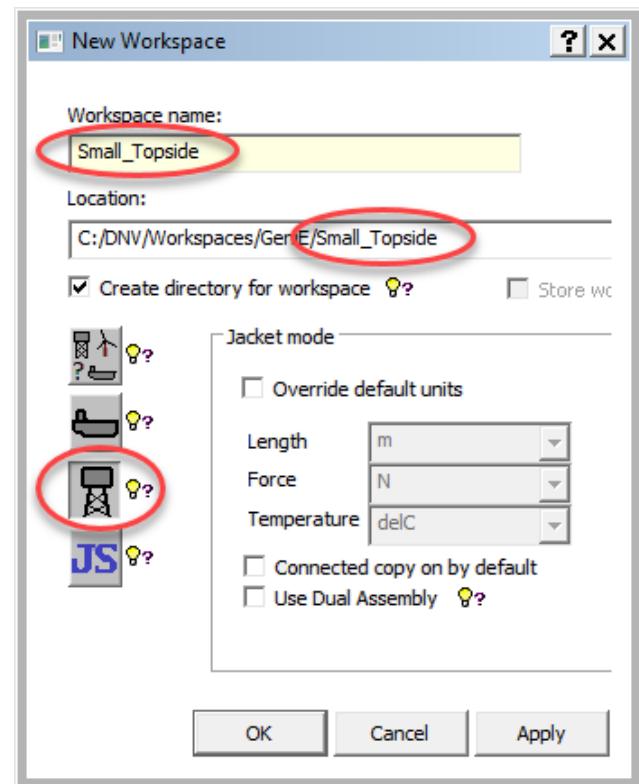
➤ The appearance of the GUI and dialogs in later versions of GeniE may change.



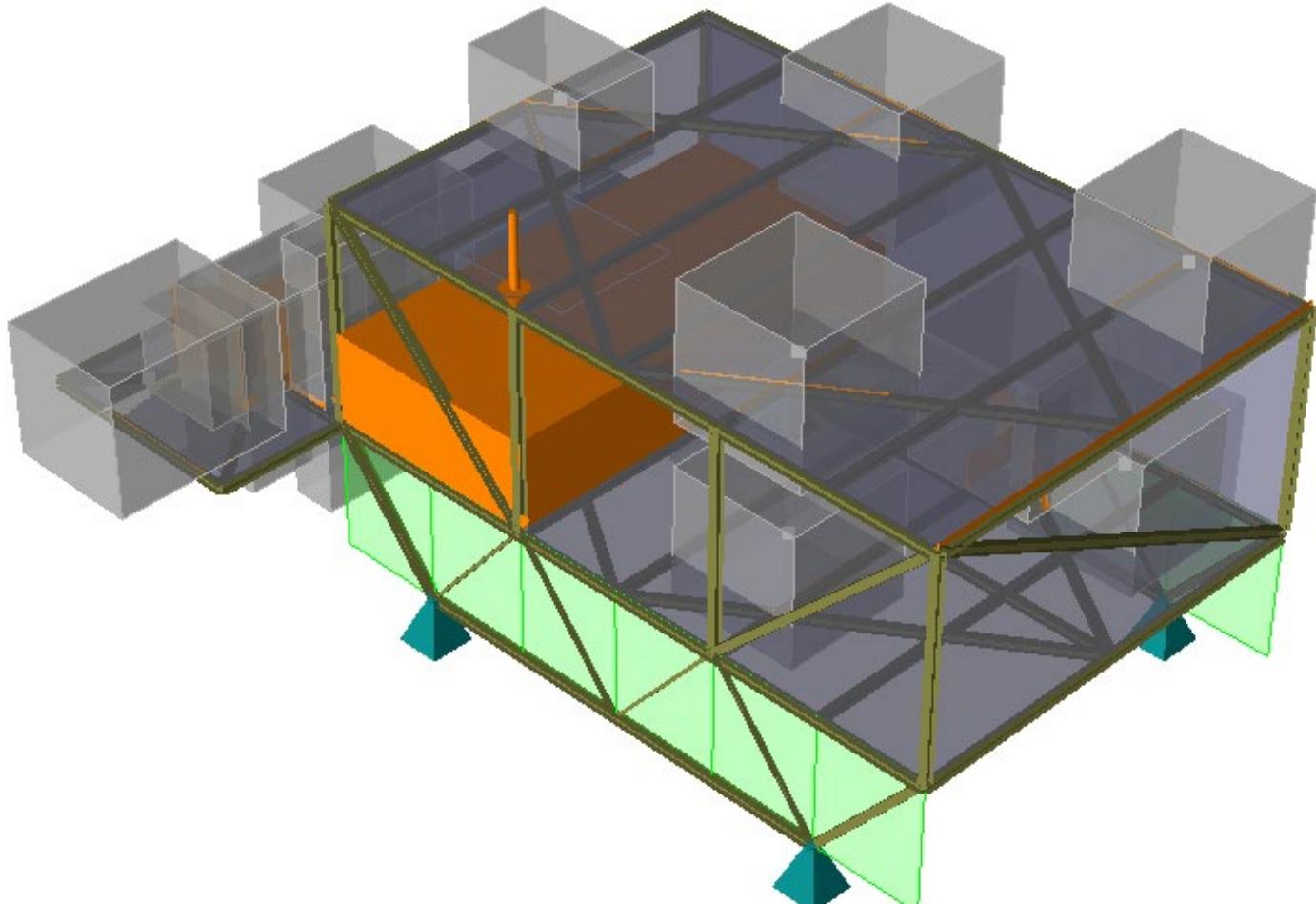
2 THE GENIE GRAPHICAL USER INTERFACE

- Start GeniE and open a new workspace.
 - Give a *Workspace name*.
 - Accept default *Output Units* m and N and click *OK*.
 - Press the *Jacket mode* button to limit menus to those relevant for jacket (spaceframe) modelling.

- Use *File | Read Command File* to read the file *Small_Topside_input.js* found in the installation folder typically named
 <path>\GeniE VX.Y-ZZ\Help\Tutorials\TutorialsBasicAndCodechecking\B2_GeniE_Small_Topside\JS

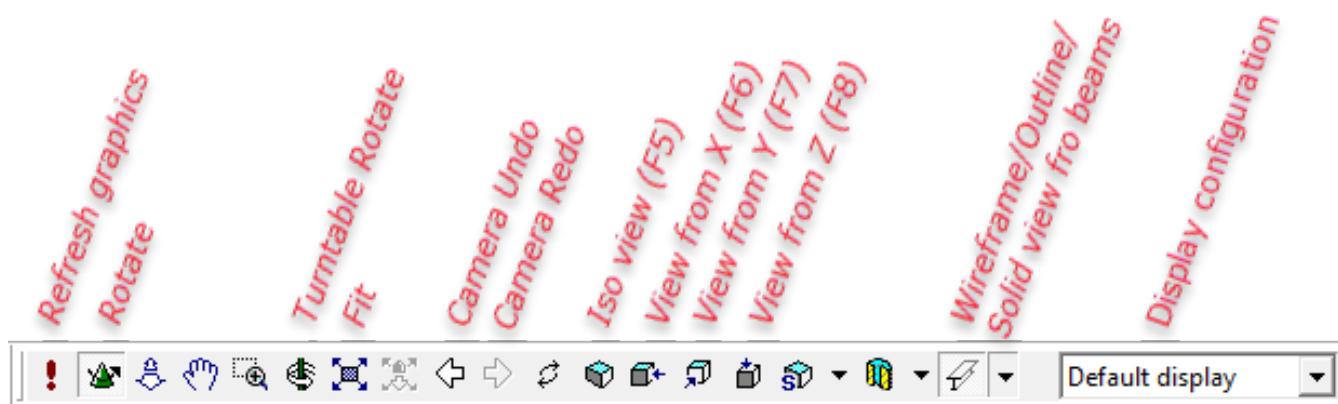


- The complete topside model shown below should appear in the display area.



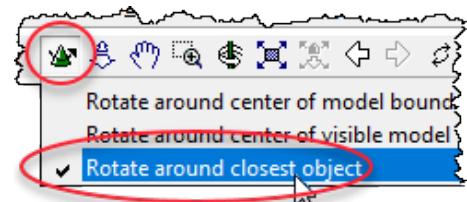
Graphics Screen Manipulations

- The *Graphics Screen Manipulations* buttons are shown below.



- Use these as follows (most important buttons mentioned):

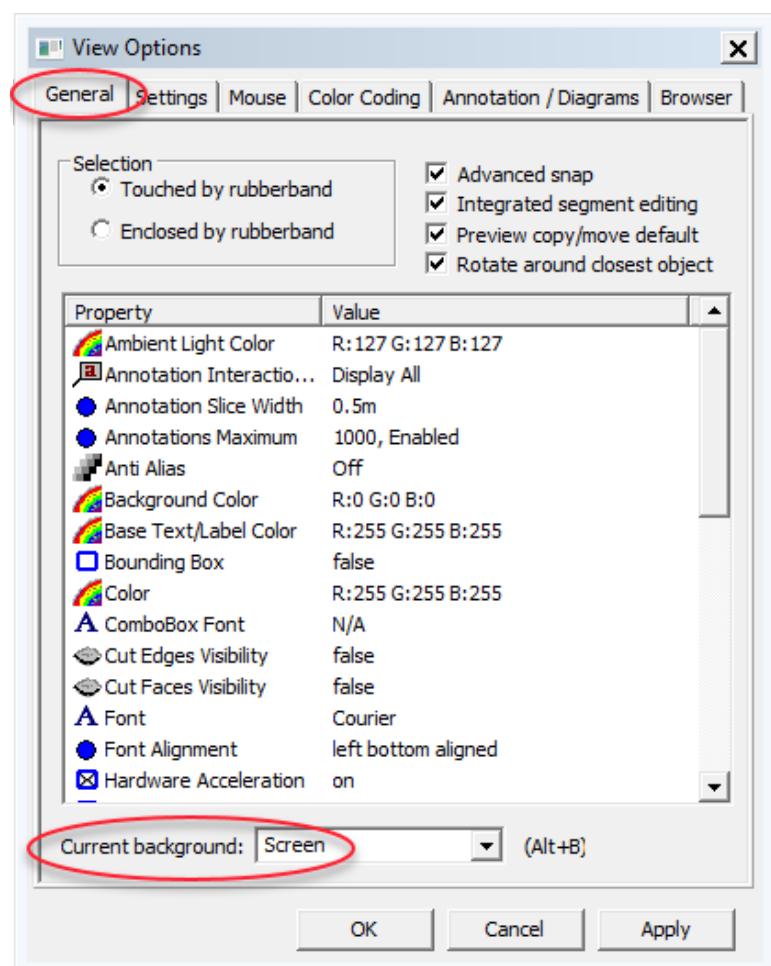
- *Refresh graphics* – Remove dimensions and angles set off.
- *Rotate* – Normally keep this button pressed down. This allows rotating with the right mouse button (RMB), zooming with Shift+RMB, panning with Ctrl+RMB and rubberband zooming with Shift+Ctrl+RMB.
 - Right-click the *Rotate* button to select rotation centre. Normally *Rotate around closest object* is the most convenient option.



- *Turntable Rotate* – Use this as an alternative to the *Rotate* option to keep vertical the currently vertical axis projection (on the screen). Normally this is the Z-axis.
- *Fit* – Fit the model, or the currently selected part of the model, to the screen.
- *Camera Undo* – Go back to the previous view, i.e. before the last rotation, zooming or panning.
- *Camera Redo* – Relevant after a *Camera Undo*.
- *Iso view* – View the model from +X,-Y+Z
- *View from X/Y/Z (F6)/(F7)/(F8)* – Toggle between viewing the model from the positive and negative X/Y/Z axis.
- *Wireframe/Outline/Solid* – Choose between three display modes for beams. *Solid* shows the thicknesses of the beam sections while *Outline* does not.
- Select display configuration: *Default display*, *Mesh - Transparent*, *Modelling - Transparent*, *Results - with Mesh*, etc.

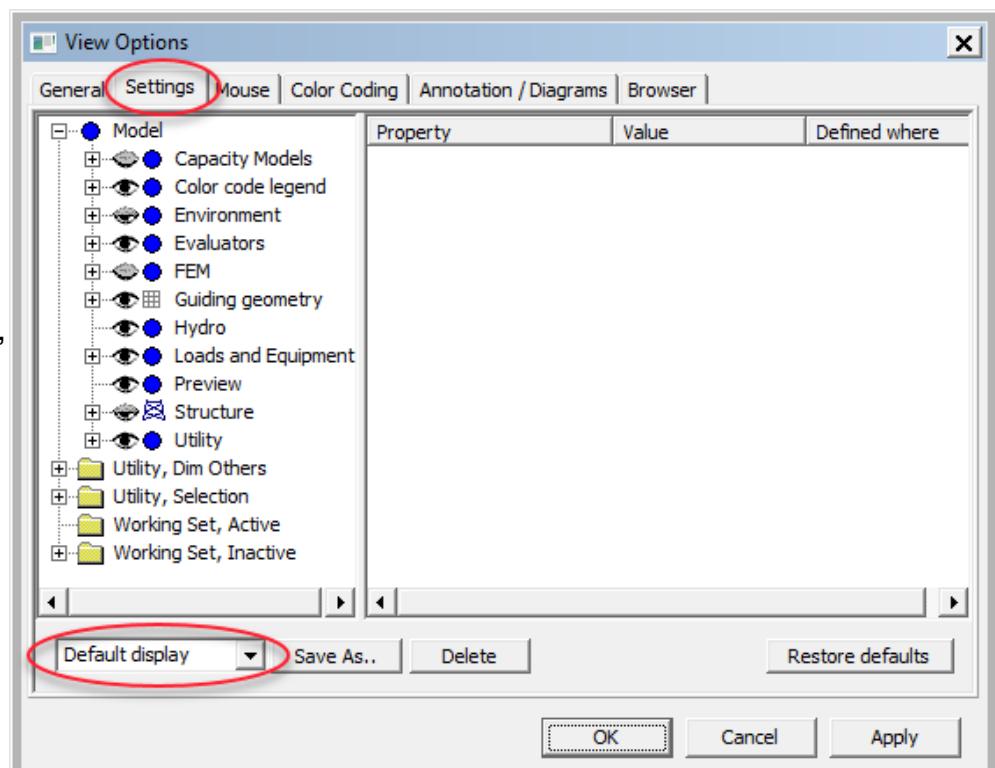
View Options | General Tab

- View | Options | General opens the dialog shown to the right.
- Double-click properties in the General tab to change settings:
 - Background colour
 - Fonts
 - Perspective/orthographic display
 - Sizes
 - Etc.
 - Set *Current background* to *Paper* for white background.
Alt+B toggles between the *Screen* and *Paper* backgrounds.

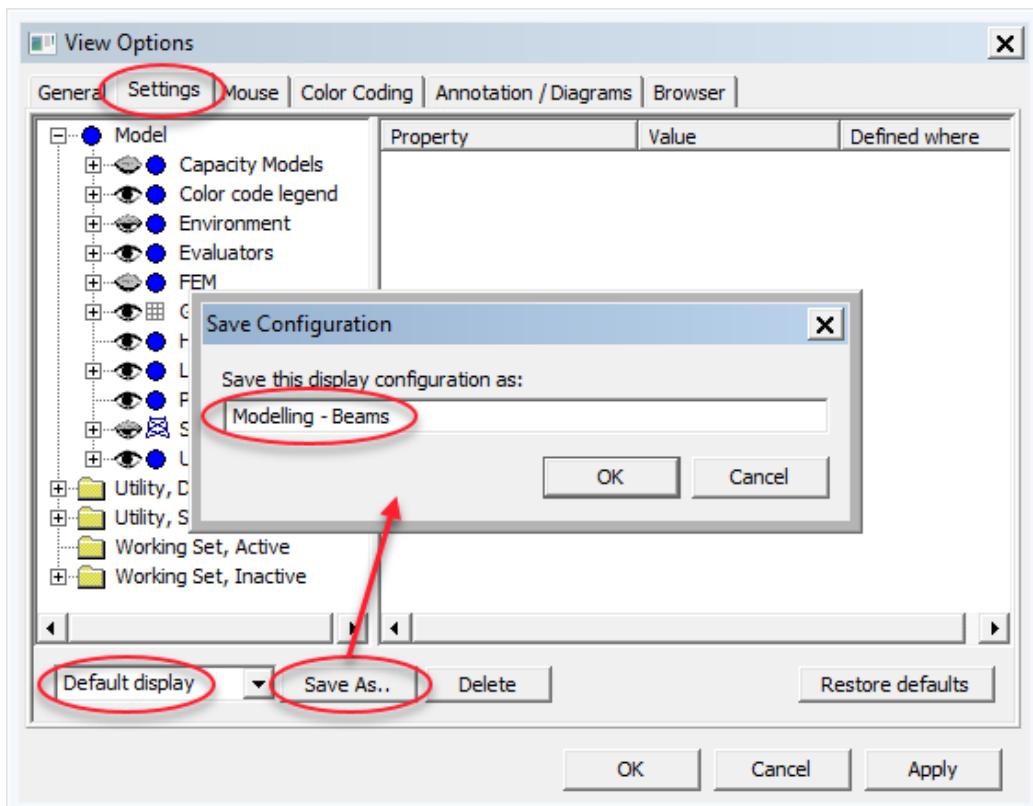


View Options | Settings Tab

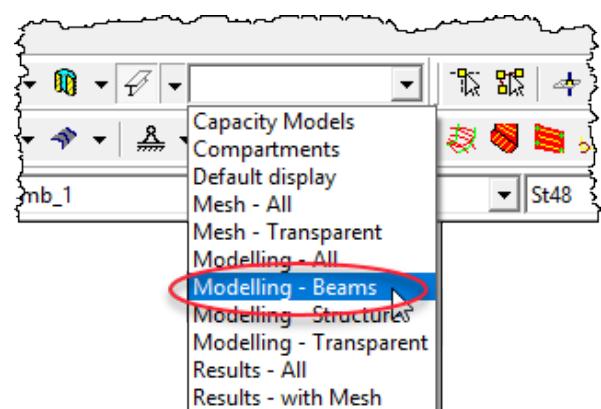
- Change how the model is displayed by View | Options | Settings (or Alt+O). Such changes are made for the currently selected display configuration, *Default display* in the figure below, and stored in the computer registry.



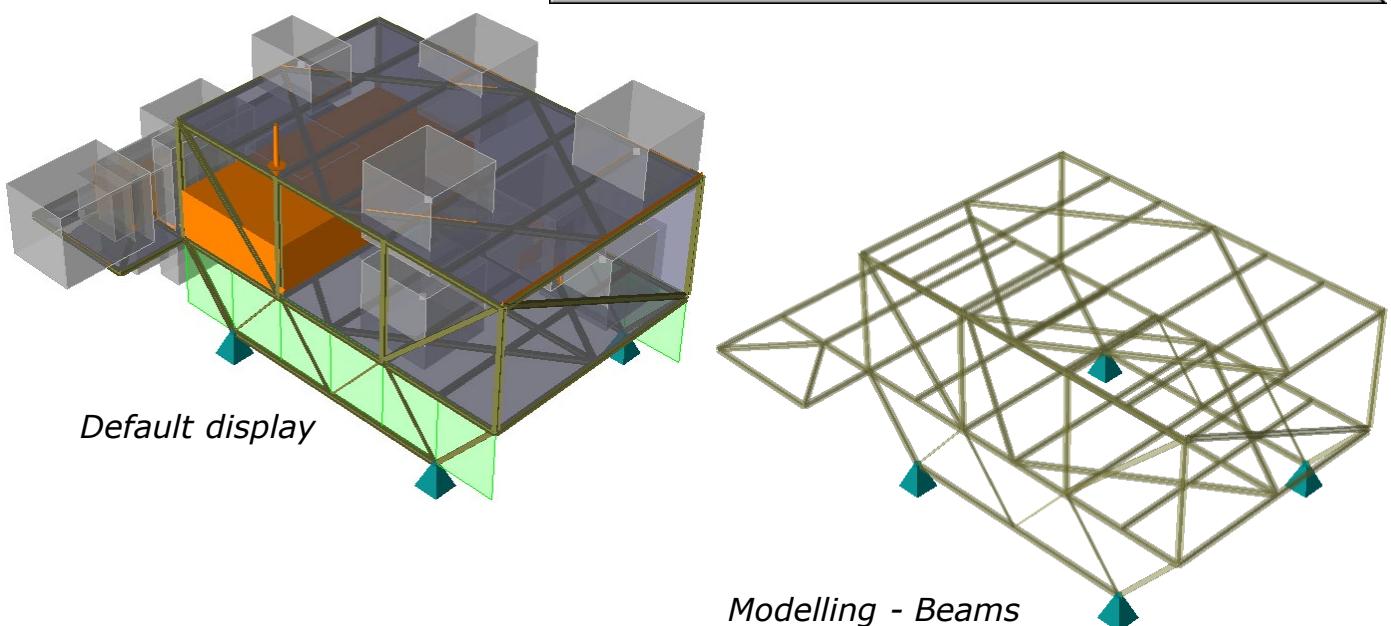
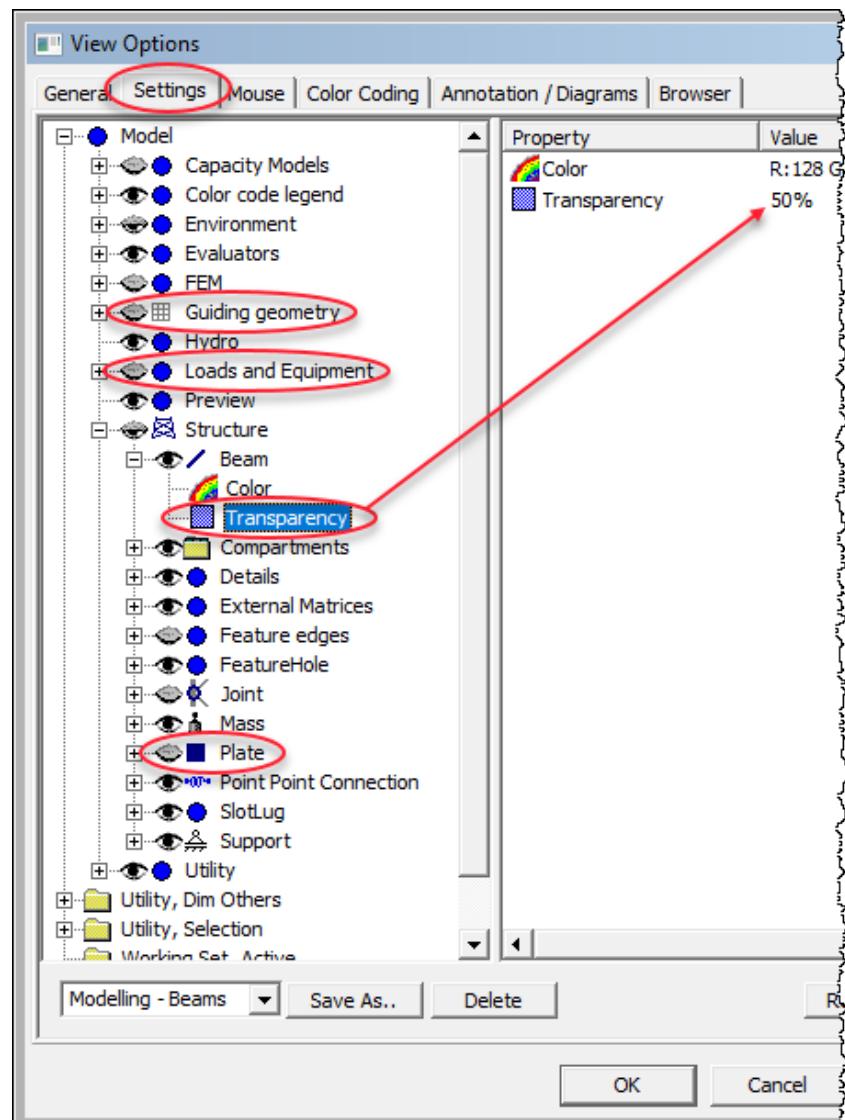
- Create a new display configuration named *Modelling - Beams* as follows.
- Base it on the *Default display* configuration.
 - Click **Save As..**.
 - Give the name *Modelling - Beams* and click OK.



- See that the new display configuration appears in the pulldown menu among the predefined display configurations.
- But yet it is identical with the *Default display* configuration.
- Expand branches in the *Settings* tab and adjust visibility (eye symbol) and transparency as explained next page.

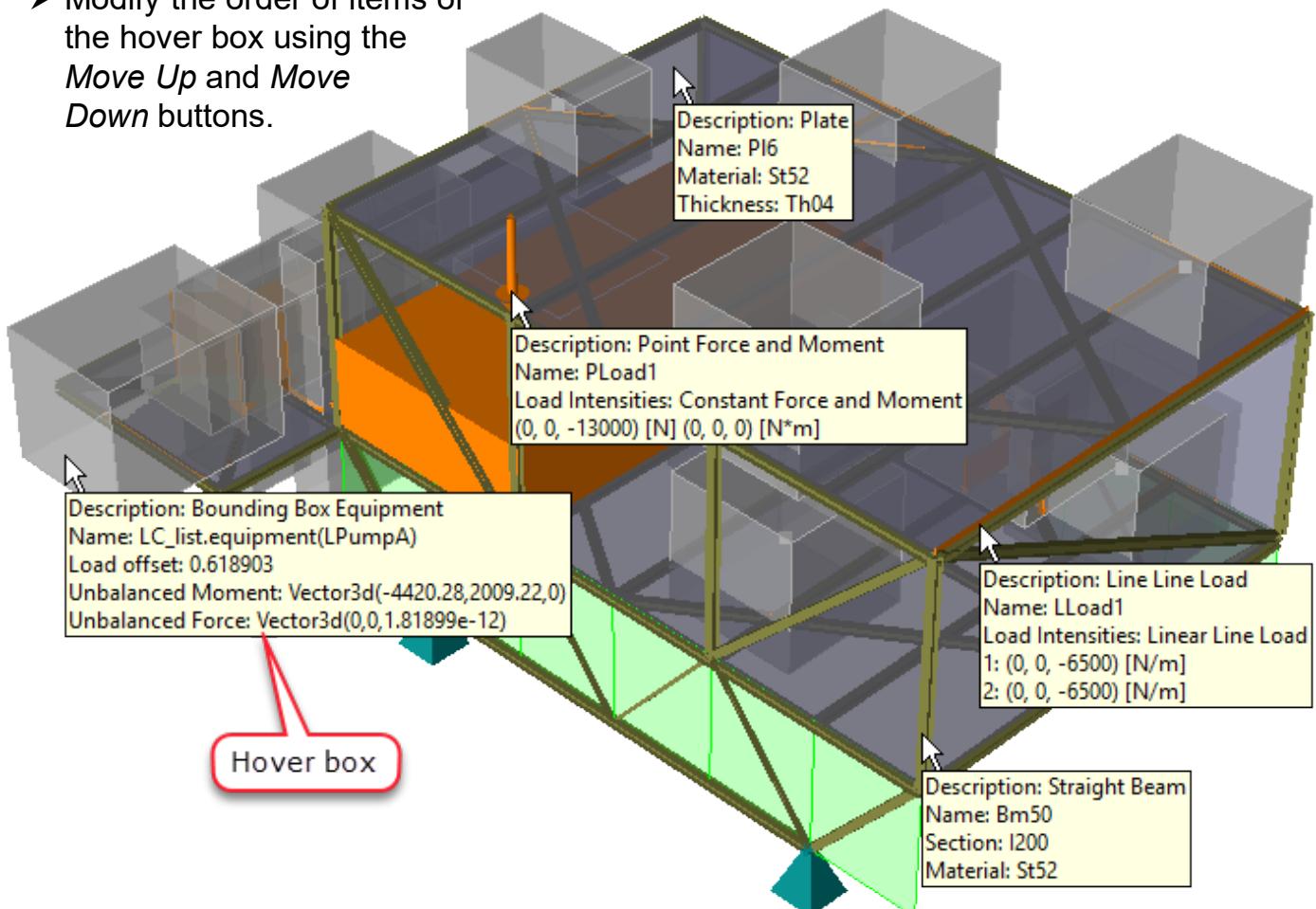
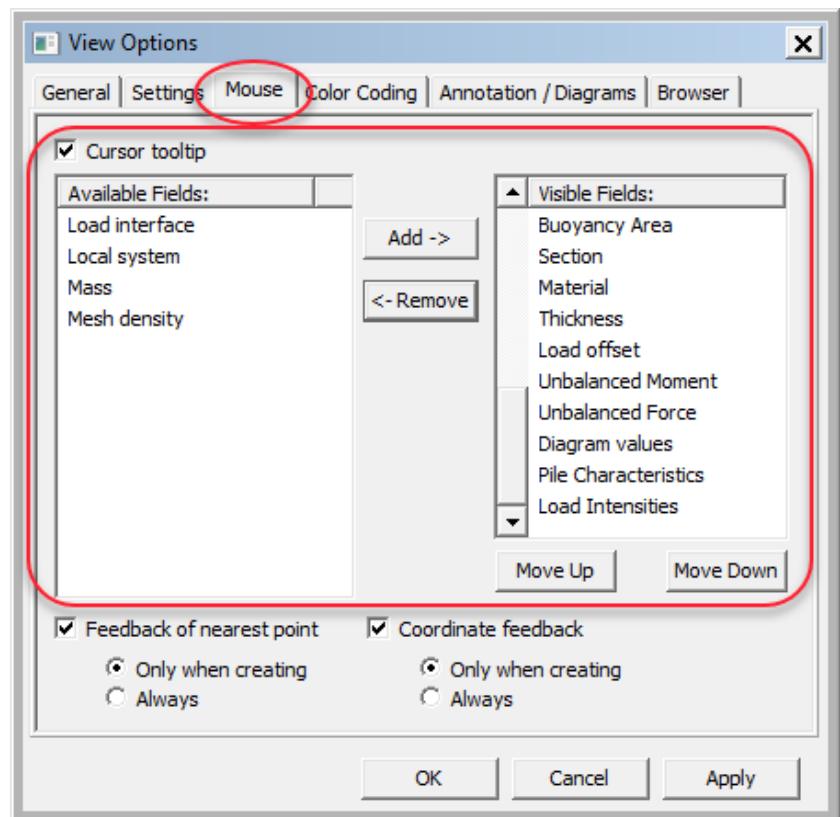


- Click the eye symbol to close it for:
 - *Guiding geometry*
 - *Loads and Equipment*
 - *Plate*
- Double-click *Transparency* for *Beam* to change it from 100% to 50%.
- The figure to the left below shows the model using the *Default display* configuration.
- The figure to the right below shows the model using the *Modelling - Beams* display configuration.
- Play with other settings, e.g. colours.



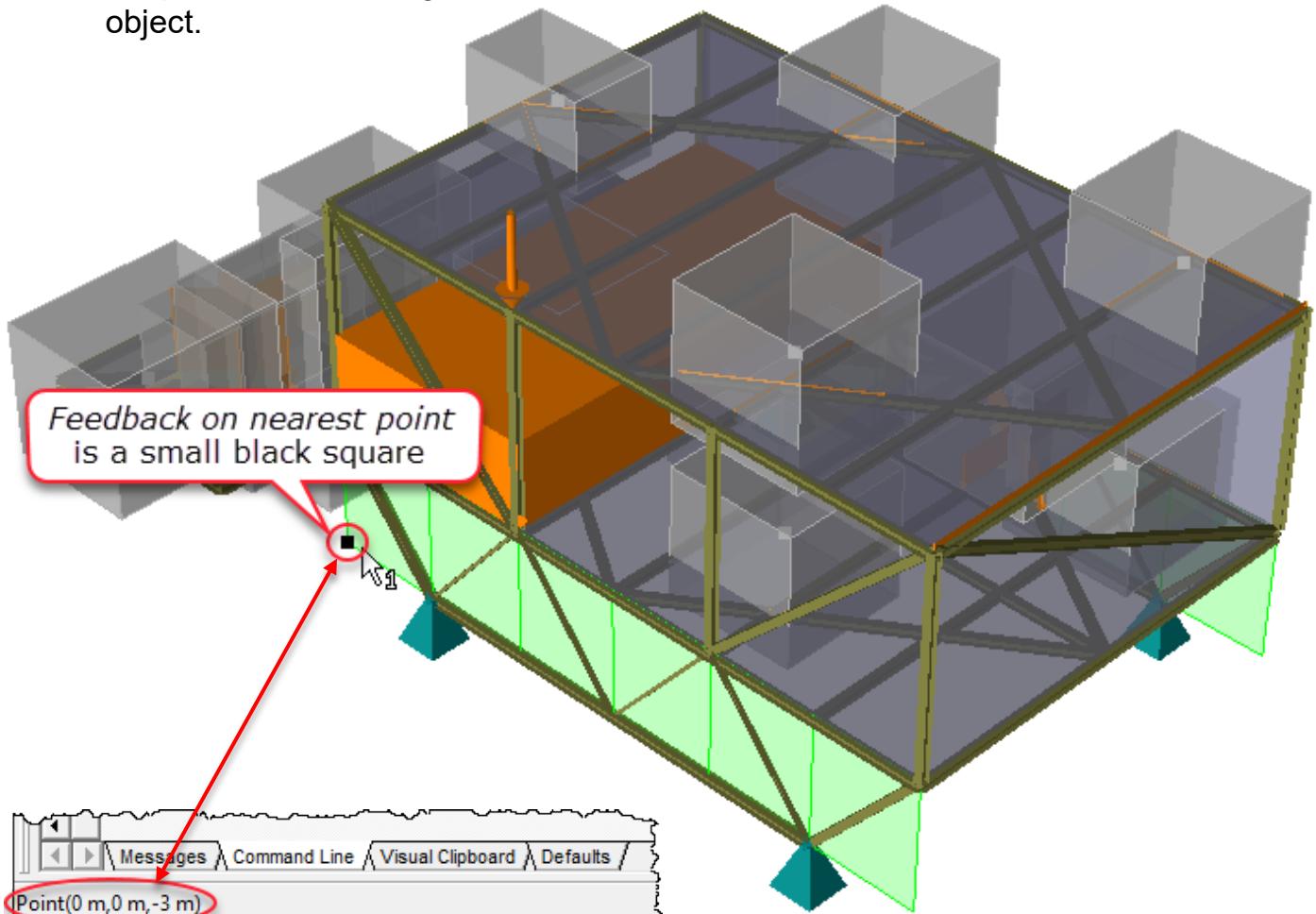
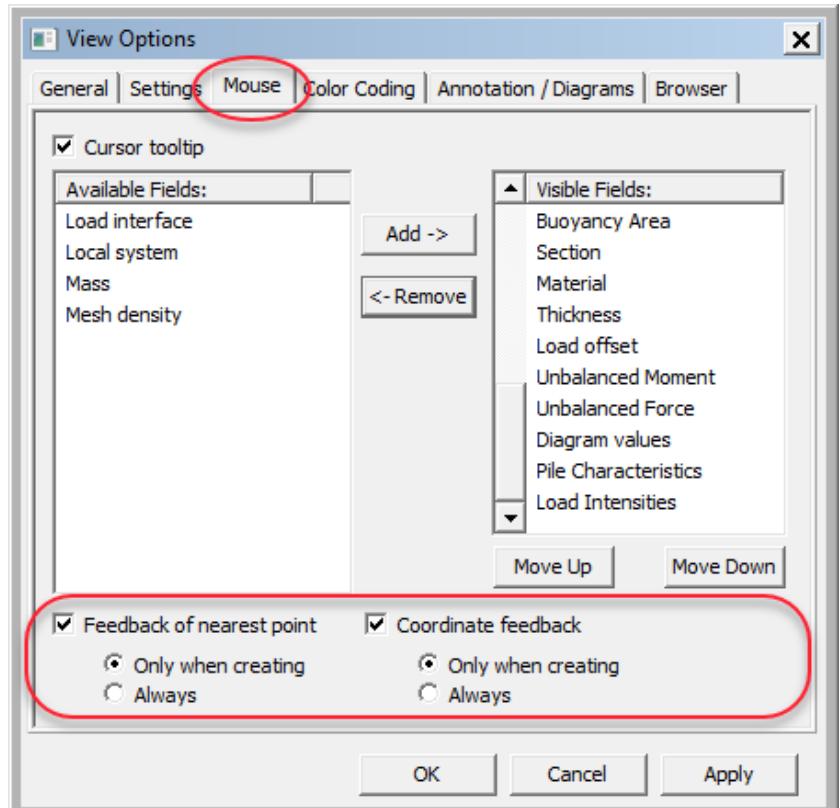
View Options | Mouse Tab

- **View | Options | Mouse**
- The **Mouse** tab of the **View Options** dialog controls the contents of the hover box (hovering mouse over object).
 - In the case of beams this is *Description*, *Name*, *Section* and *Material* as seen in the figure below.
 - The figure also shows the hover box for other objects.
- Modify the hover box content by moving items between the *Available Fields* and *Visible Fields*.
- Modify the order of items of the hover box using the *Move Up* and *Move Down* buttons.



➤ The lower area of the *Mouse* tab is used to control the small black square and the coordinates in the lower left corner of the GeniE window when hovering over the model.

- The figure below shows an example of these two feedbacks.
- By default, both *Feedback on nearest point* and *Coordinate feedback* are switch on when in the process of creating an object, e.g. a beam.
- Select *Always* to get this feedback even when not in the process of creating an object.



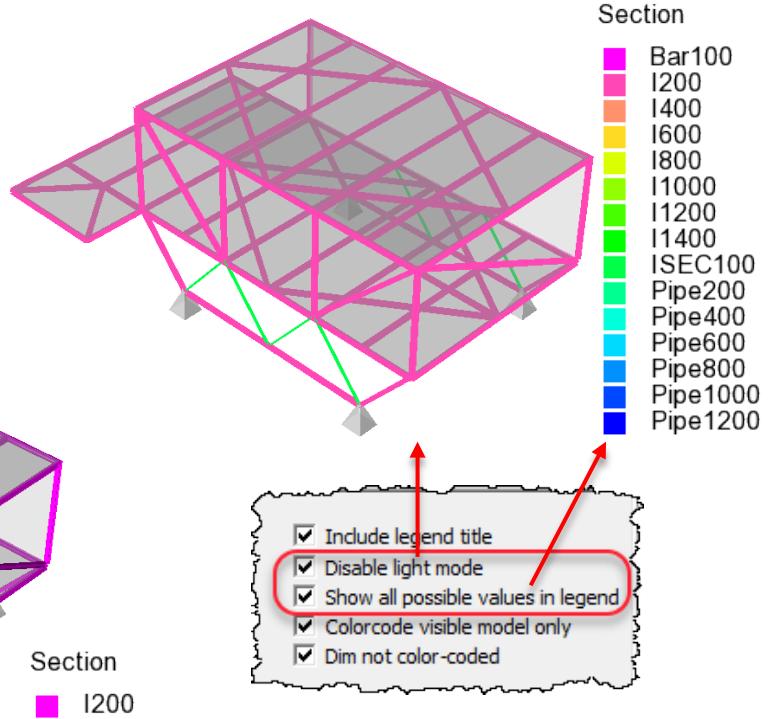
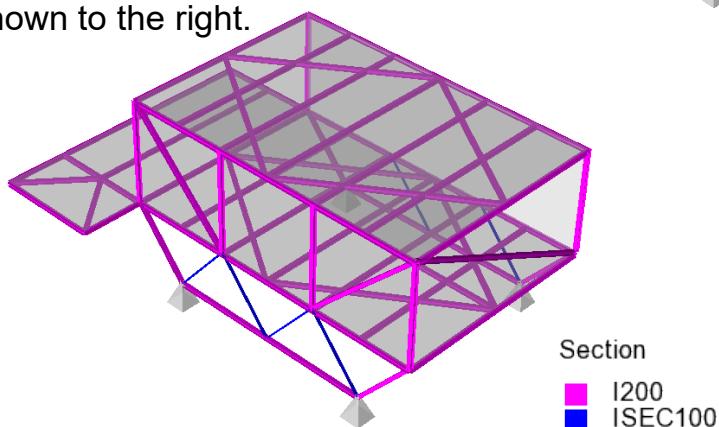
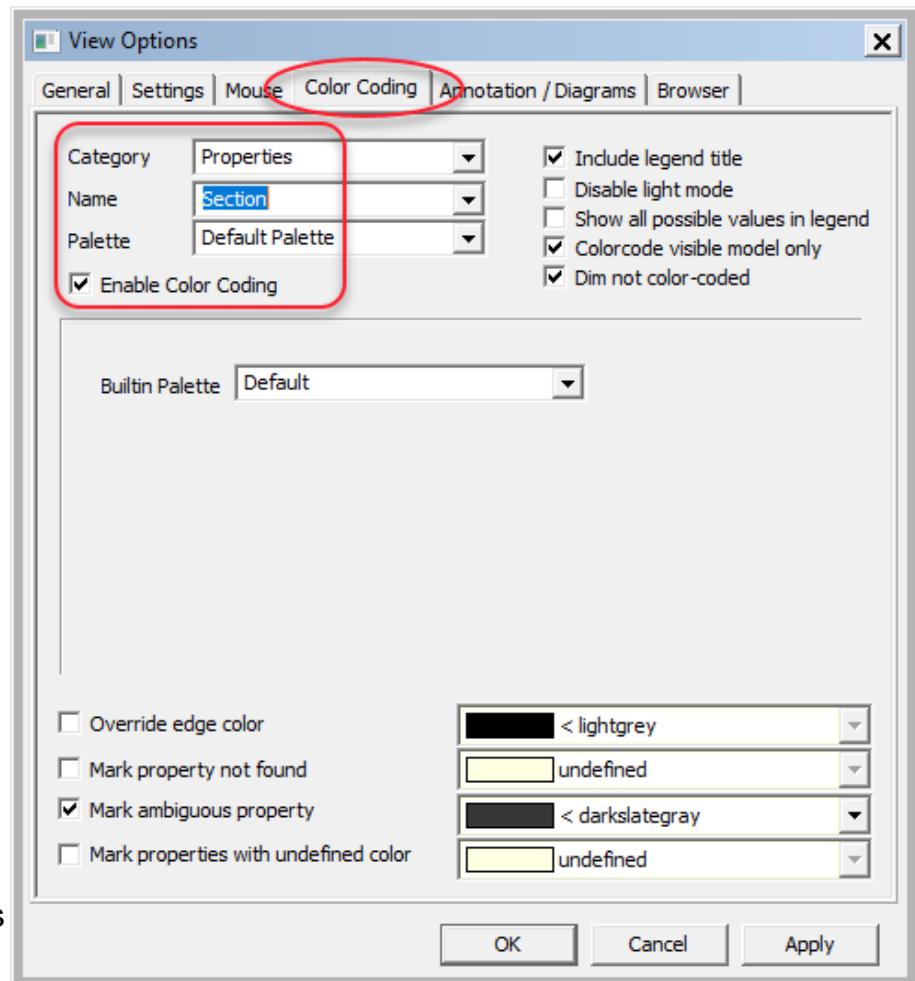
View Options | Color Coding Tab

- **View | Options | Color Coding**
- The **Color Coding** tab of the **View Options** dialog controls what to colour code and colours to use.

- An easy way of colour coding, e.g. sections, is to right-click the **Sections** folder in the browser and selecting **Color code all visible properties**. This is described later in this tutorial.
- The **Color Coding** tab, however, offers more control of the colour coding.

- Colour coding beam cross sections using the default palette is shown below.

- The **Modelling - Transparent** display configuration is chosen and no load case.
- The effect of **Disable light mode** (brighter colours) and **Show all possible values in legend** is shown to the right.

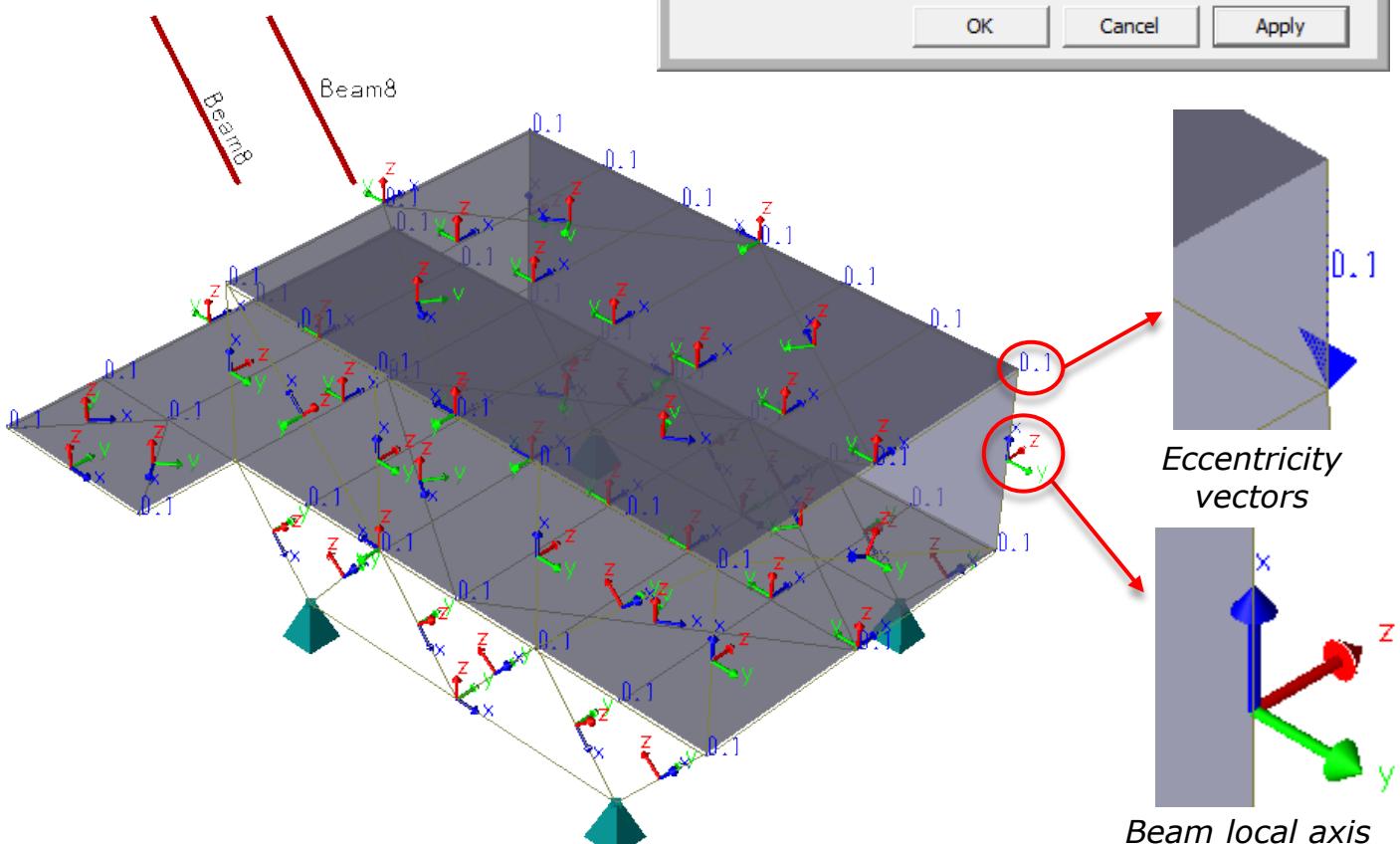
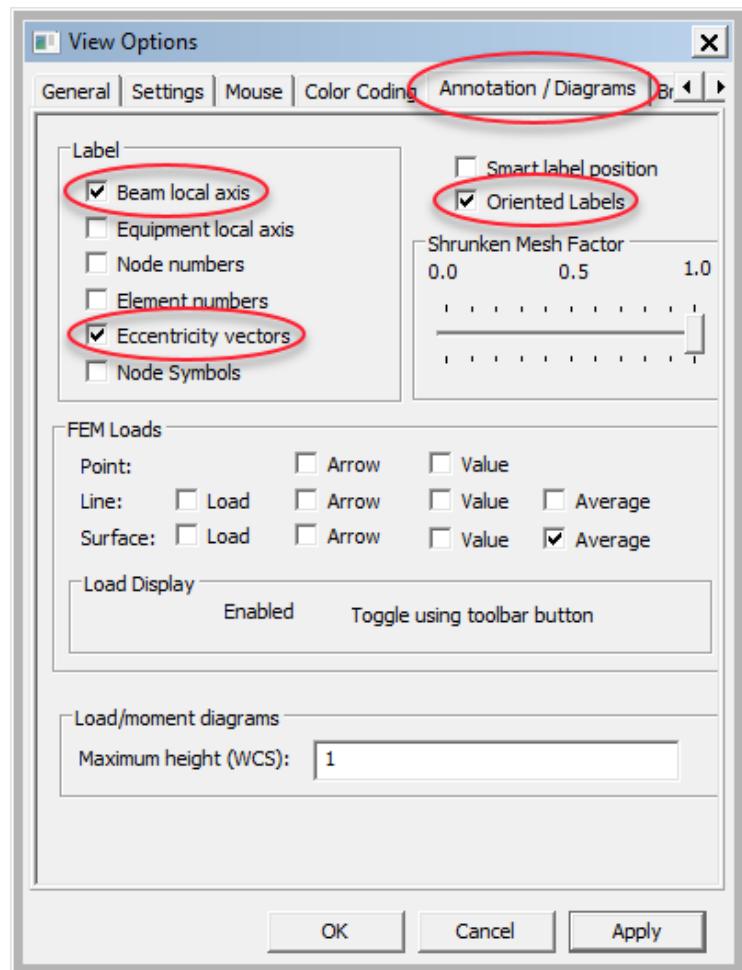


View Options | Annotation / Diagrams Tab

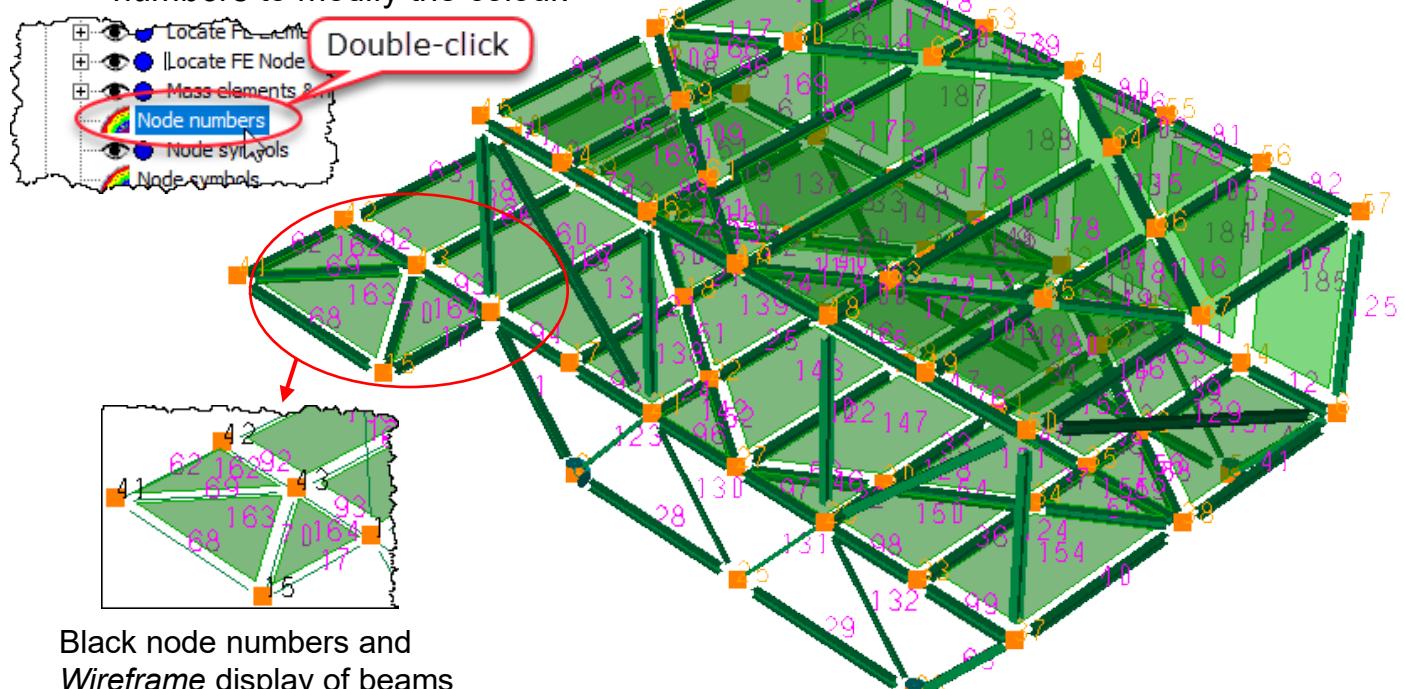
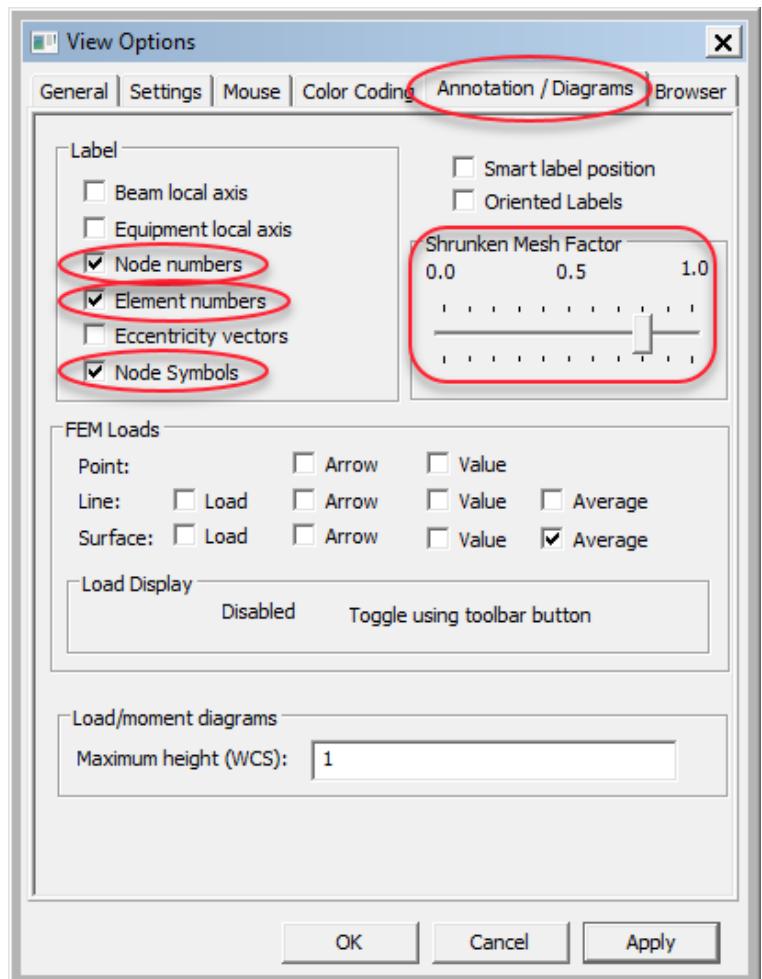
- *View | Options | Annotation / Diagrams*
- The *Annotation / Diagrams* tab of the *View Options* dialog controls global labelling, i.e. labelling for the whole model.
 - The *Beam local axis* and *Eccentricity vectors* options are exemplified below with beams displayed in *Wireframe* mode.



- The *Oriented Labels* option checked (left) and unchecked (right) is illustrated below.



- The *Node numbers*, *Element numbers* and *Node symbols* options of the *Annotation / Diagrams* tab are relevant only when displaying the FE mesh.
- Use Alt+M to open the *Activity Monitor* and click *Start* to create a FE mesh (a very coarse mesh).
- Switch to *Mesh - Transparent* display configuration.
- The three said options plus 0.8 as *Shrunken Mesh Factor* are illustrated below.
- *Node numbers* and *Node symbols* are orange.
- *Element numbers* are magenta.
- These colours may be changed in the *Settings* tab. With white background (*General* tab, *Paper* background) black node numbers may be preferred. Expand *FEM* | *Mesh* and double-click *Node numbers* to modify the colour.

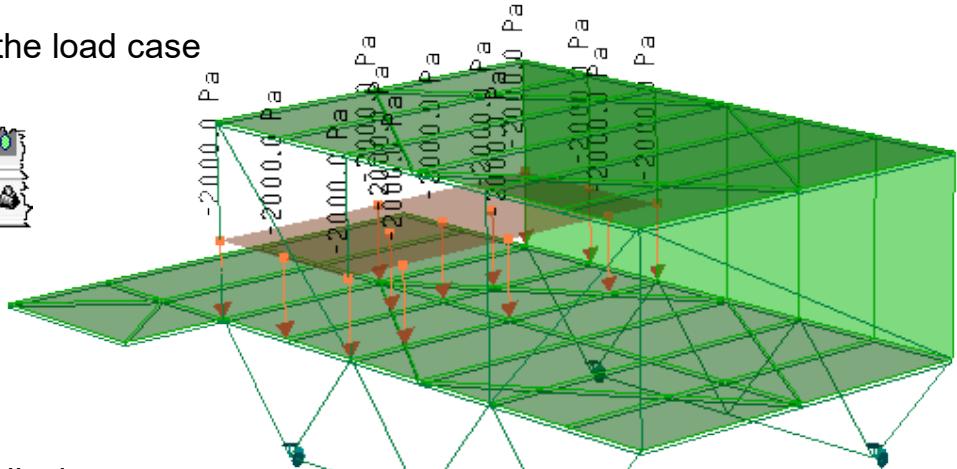
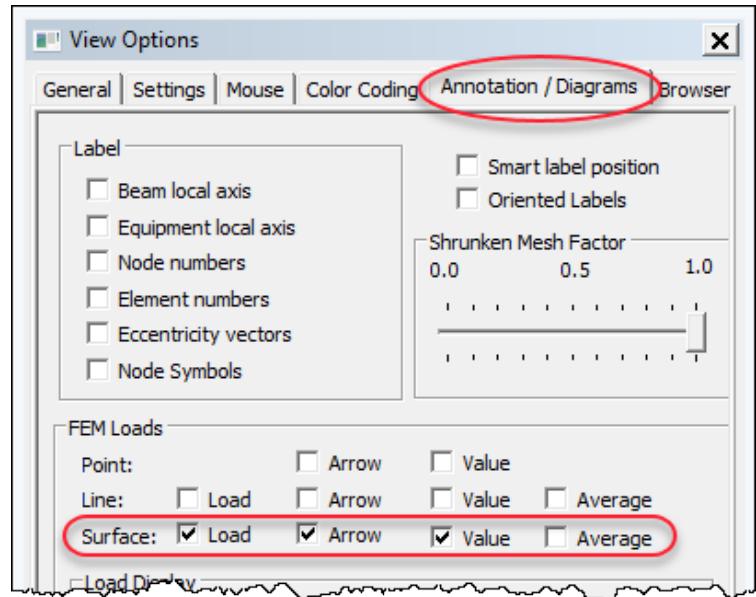
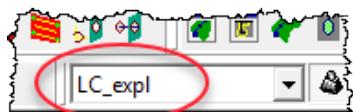


- The *FEM Loads* area in *Annotation / Diagrams* tab are for adding display of loads to the mesh display.

- *FEM Loads* are loads created by the meshing process. These may be a subset of loads added to the geometry (concept) model. If a part of a load fails to match any geometry it will not be represented by the *FEM Loads*.

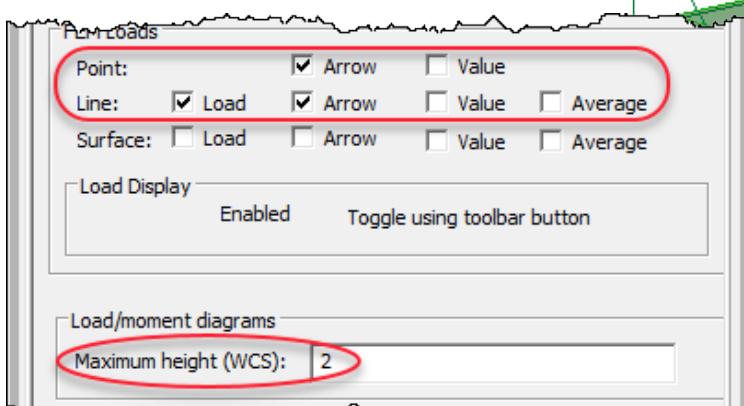
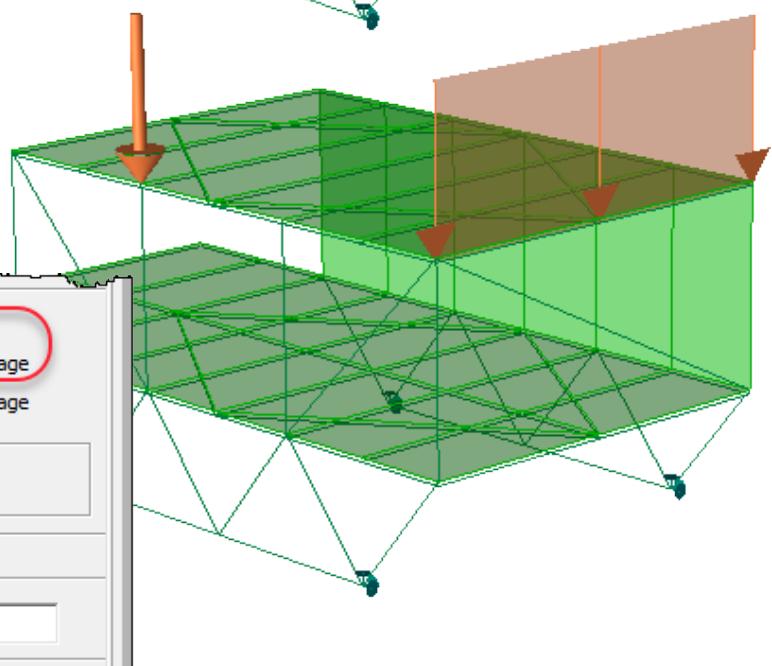
- An example of adding *Surface* load for the load case LC_expl is shown to the right below.

- Select load case in the load case selector:



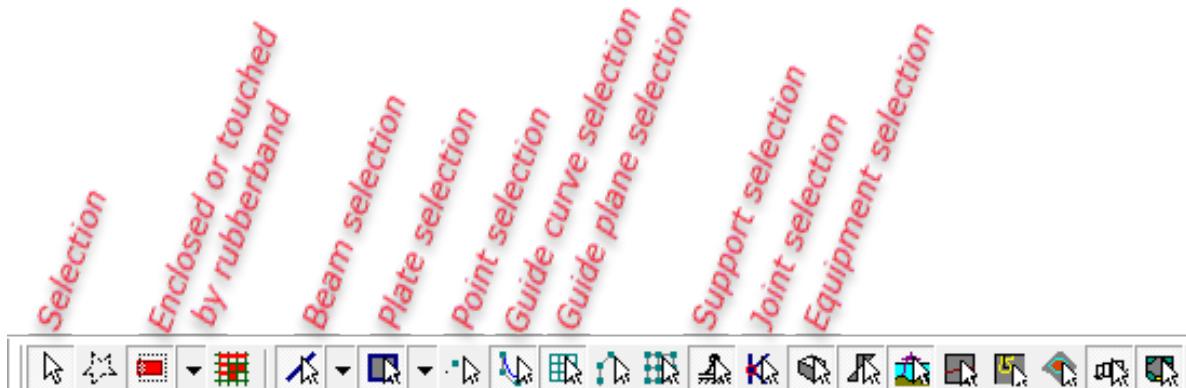
- The figure to the right displays *Point* and *Line* loads for LC_expl.

- Arrow and Value may be added.
- Use *Maximum height* to scale up the displayed load.



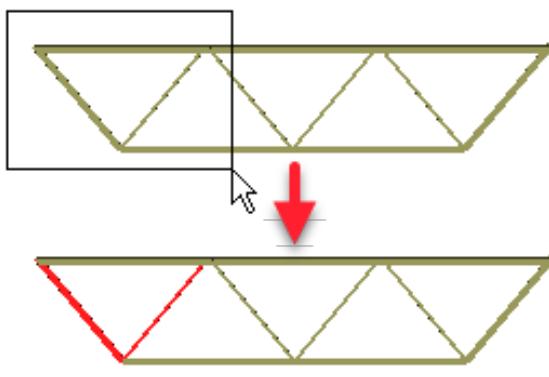
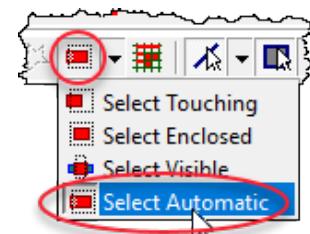
Selection and Filters

- Beams, plates and other objects may be selected in the browser and graphically.
- The *Selection and Filter* buttons shown below control graphic selection.

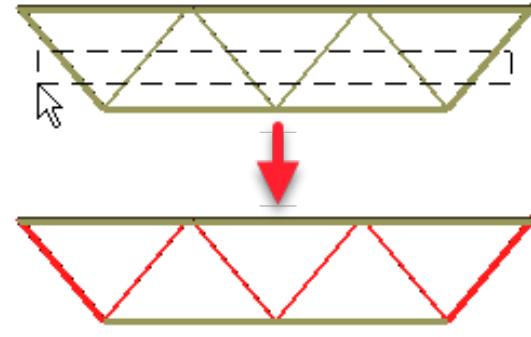


- Use these as follows:

- *Selection* – To select objects in the display this button must be pressed down.
- *Enclosed or touched by rubberband* – Of the four alternatives as shown to the right, the *Select Automatic* is normally the best choice. This allows enclosed selection when dragging the rubberband from left to right and touched-by selection when dragging from right to left as illustrated below.

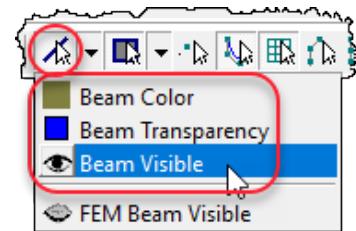


Enclosed selection



Touched by selection

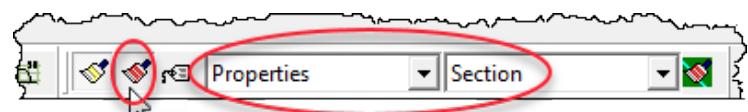
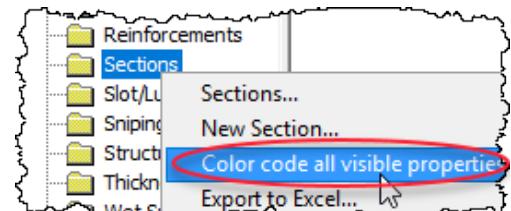
- The selection and filter buttons *Beam selection*, *Plate selection*, etc. have three functionalities:
 - Button depressed enables selection in the display area, button lifted disables selection. This is convenient for selecting beams but not plates and vice versa.
 - Right-click to change colour and transparency.
 - Right-click to close/open eye symbol to hide/display.



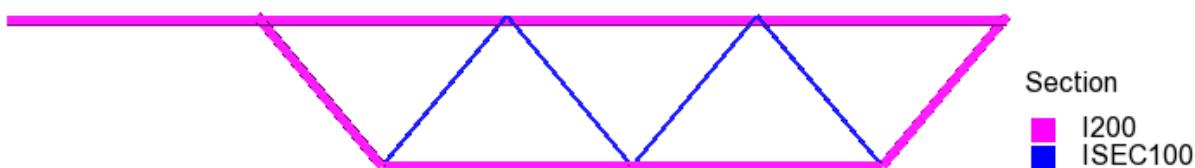
Label and Colour Code Selected Objects

- Additionally to colour coding using the *View | Options | Color Coding* tab as explained earlier, the following methods are available:

- Right-clicking a folder in the browser and selecting *Color code all visible properties* as shown to the right.
- The *Color Code and Label* as shown to the right are normally in the upper right corner of the GeniE window. Select category, e.g. *Properties*, and sub-category, e.g. *Section*, and click the button as indicated.

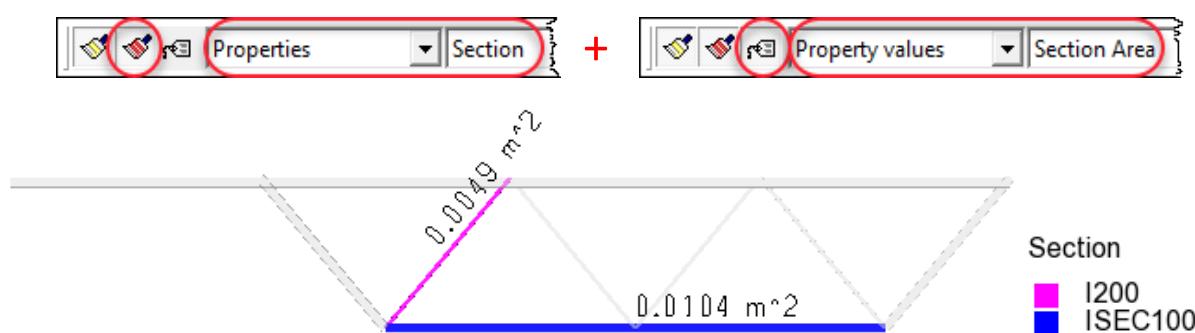


- Both methods above give the same display:

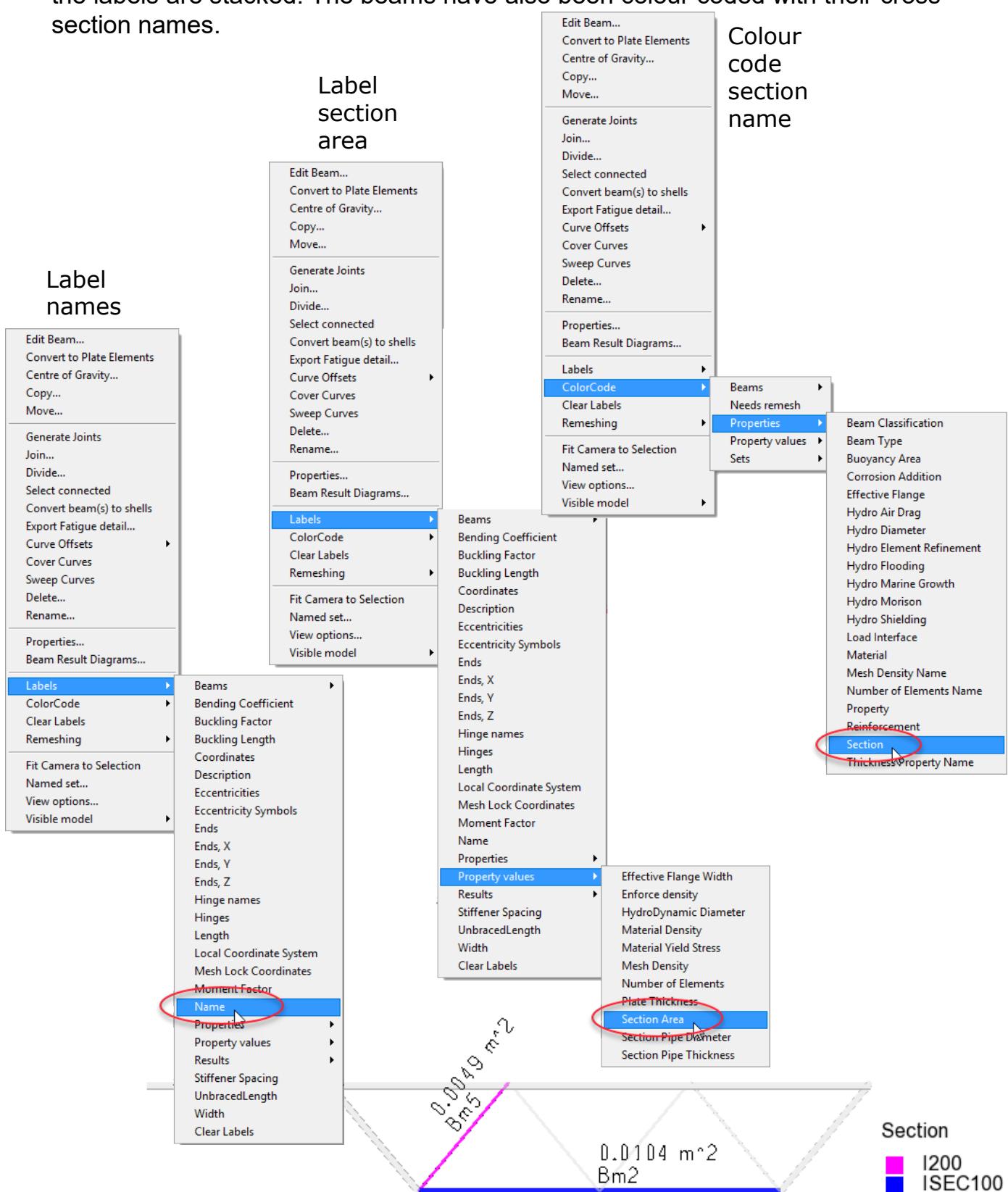


- The above methods colour code the entire displayed model. The *Color Code and Label* options also allow colour coding plus labelling a selection only.

- In the example below a couple of beams have been selected and then colour coded with *Properties Section* and labelled with *Property values Section Area*.

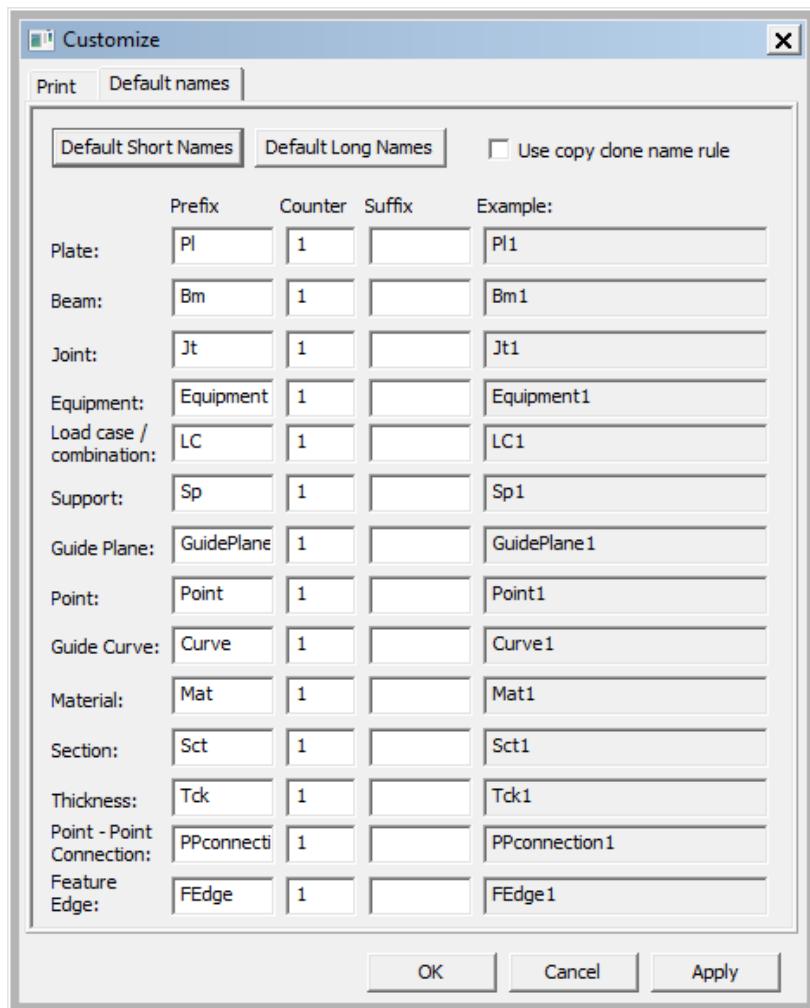


- A final method for colour coding and labelling is to use the context sensitive menu. Below is an example where a couple of beams have been selected and, using the context menus shown, labelled with their names and cross section areas. As seen, the labels are stacked. The beams have also been colour coded with their cross section names.



Default Names

- View | Options | Default Names is used to open the *Customize* dialog shown below. This dialog is used to set default names for the objects to create.



- The settings of the dialog involves that:
- Plates will be named Pl1, Pl2, Pl3, etc.
 - Beams will be named Bm1, Bm2, Bm3, etc.
- An example of use of the dialog is to change the *Prefix* and reset the *Counter* to 1 before creating parts of the model. For example:
- *Prefix* = LoBm, *Counter* = 1, and then create all beams in the lower deck.
 - *Prefix* = UpBm, *Counter* = 1, and then create all beams in the upper deck.
 - *Prefix* = Col, *Counter* = 1, and then create all columns.

Help

- Click *Help | Help Topics* (or press F1) to open the GeniE help pages in a web browser.
- In the left column click the topic of interest.

- These are:

- License – Information about the GeniE license system
- Release Notes – Information on new features and bug fixes for the last few versions
- Support Request – How to get help from our support staff
- User Documentation (User Manual) – How to use GeniE
- Guiding Documents – Additional user documentation on various modelling topics
- Tutorials – Basic and advanced tutorials for self-education in using GeniE
- Reference Documents – Additional user documentation on implementation of code checks and import of data
- JScript Commands – Overview and documentation of GeniE JScript commands
- Wizard Templates – Downloadable Excel files for creating jacket and deck models
- GeniE SnackPack – Script files for highly efficient modelling of various objects
- Sesam Examples – Link to a web page with several examples of use of Sesam, i.e. not only GeniE
- Sesam Portal – Portal to launching web applications, downloading applications and documents, help and training, support, etc.



DNV

Sesam GeniE D8.2-01

05-Jun-2021

[License](#)

[Release Notes](#)

[Support Request](#)

[User Documentation](#)

[Guiding Documents](#)

[Tutorials](#)

[Reference Documents](#)

[JScript Commands](#)

[Wizard Templates](#)

[GeniE SnackPack](#)

[Sesam Examples](#)

[Sesam Portal](#)

Final Notes Before Modelling

➤ Logging of commands:

- All commands creating, or in any way modifying, a model are logged as JavaScript commands in a file with the name of the workspace with extension js.
- The log file does not distinguish between commands read using *File | Read Command File*, clicking in the menu and filling in dialogs, and commands entered in the *Command Line* area.
- Commands for colour coding and labelling are not logged.
- A typical log file:

```
// GenIE V7.13-11 64bit started 04-Dec-2019 09:04:47
GenieRules.Tolerances.useTolerantModelling = true;
GenieRules.Tolerances.angleTolerance = 2 deg;
GenieRules.Compatibility.version = "V7.13-11";
GenieRules.Meshing.autoSimplifyTopology = true;
GenieRules.Meshing.eliminateInternalEdges = true;
GenieRules.BeamCreation.DefaultCurveOffset = ReparameterizedBeamCurveOffset();
GenieRules.Transformation.DefaultConnectedCopy = false;
// GenIE V7.13-11 64bit ended 04-Dec-2019 09:04:47
// GenIE V7.13-11 64bit started 04-Dec-2019 09:04:47
//
// Cross sections
Bar100 = BarSection(0.1,0.03);
I200 = ISection(0.2,0.2,0.015,0.02);
I400 = ISection(0.4,0.4,0.015,0.02);
//
// Material library
St42 = Material(240E6, 7.85E3, 2.1E11, 0.3, 1.2E-5, 0.03);
//
I200.setdefault();
St42.setdefault();
//
// Define guide plane
GuidePlane1 = GuidePlane(Point(0,0,-3),Point(15,0,-3),Point(15,0,0),Point(0,0,0),6,1,1,1,1,1,1,1,1);
//
// Create beams
//
Bm1 = Beam(Point(2.499999762,0.,-3.),Point(12.5,0.,-3.));
Bm2 = Beam(Point(12.5,0.,-3.),Point(15.,0.,0.));
Bm3 = Beam(Point(15.,0.,0.),Point(0.,0.,0.));
```

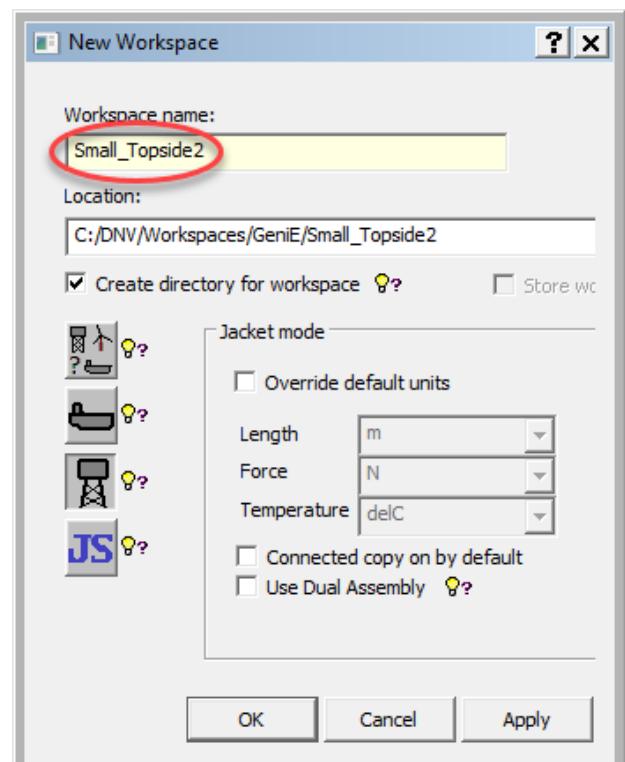
➤ Undo and redo:

- An unlimited number of undo and redo actions may be taken by Ctrl+Z and Ctrl+Y, respectively.
- An undo can only be done back to the last time the workspace was saved.

➤ Save the workspace from time to time. This is especially important for large and complex models.

3 CREATE THE MODEL

- If a GeniE session is already running, use *File | New Workspace* (or *Ctrl+N*) to create a new workspace. Otherwise, start GeniE and open a new workspace.
 - Give a *Workspace name* different from the previous name.
 - Accept default *Output Units* m and N and click *OK*.
 - Unless otherwise specified, all values in this tutorial are in these units.
- Rather than importing a section library (*File | Import | Section library*) or creating the sections manually, read the file (*File | Read Command File*) *SectAndMat.js* found in the installation folder typically named <path>\GeniE VX.Y-ZZ\Help\Tutorials\ TutorialsBasicAndCodechecking\B2_GeniE_Small_Topside\JS
 - This file also contains material and plate thickness definitions.
 - Verify that sections, materials and plate thicknesses are found in the relevant *Properties* folders in the browser. The *Sections* folder shown below.



	Name	Description	Type
Bar100	Bar Section, h=0.1 m, w=0.03 m		Bar Section
I200	I Section, h=0.2 m, w=0.2 m, wt=0.015 m, ft=0.02 m		I Section
I400	I Section, h=0.4 m, w=0.4 m, wt=0.015 m, ft=0.02 m		I Section
I600	I Section, h=0.6 m, w=0.6 m, wt=0.015 m, ft=0.02 m		I Section
I800	I Section, h=0.8 m, w=0.8 m, wt=0.015 m, ft=0.02 m		I Section
I1000	I Section, h=1 m, w=1 m, wt=0.015 m, ft=0.02 m		I Section
I1200	I Section, h=1.2 m, w=1.2 m, wt=0.015 m, ft=0.02 m		I Section
I1400	I Section, h=1.4 m, w=1.4 m, wt=0.015 m, ft=0.02 m		I Section
ISEC100	I Section, h=0.1 m, w=0.1 m, wt=0.015 m, ft=0.02 m		I Section
Pipe200	Pipe Section, d=0.2 m, t=0.02 m		Pipe Section
Pipe400	Pipe Section, d=0.4 m, t=0.02 m		Pipe Section
Pipe600	Pipe Section, d=0.6 m, t=0.02 m		Pipe Section
Pipe800	Pipe Section, d=0.8 m, t=0.02 m		Pipe Section
Pipe1000	Pipe Section, d=1 m, t=0.02 m		Pipe Section
Pipe1200	Pipe Section, d=1.2 m, t=0.02 m		Pipe Section

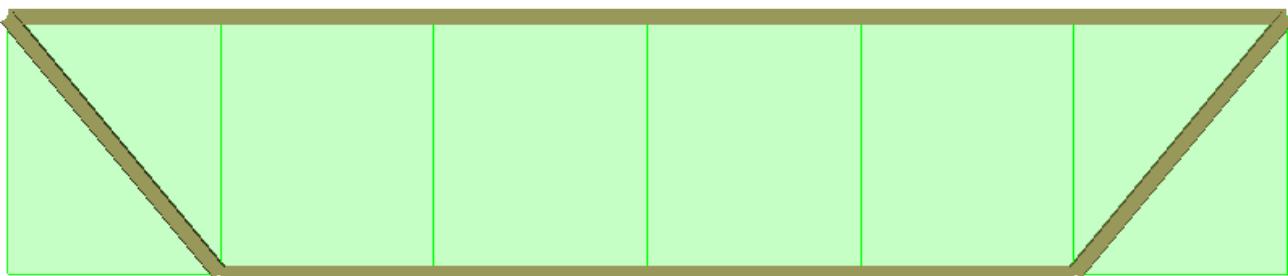
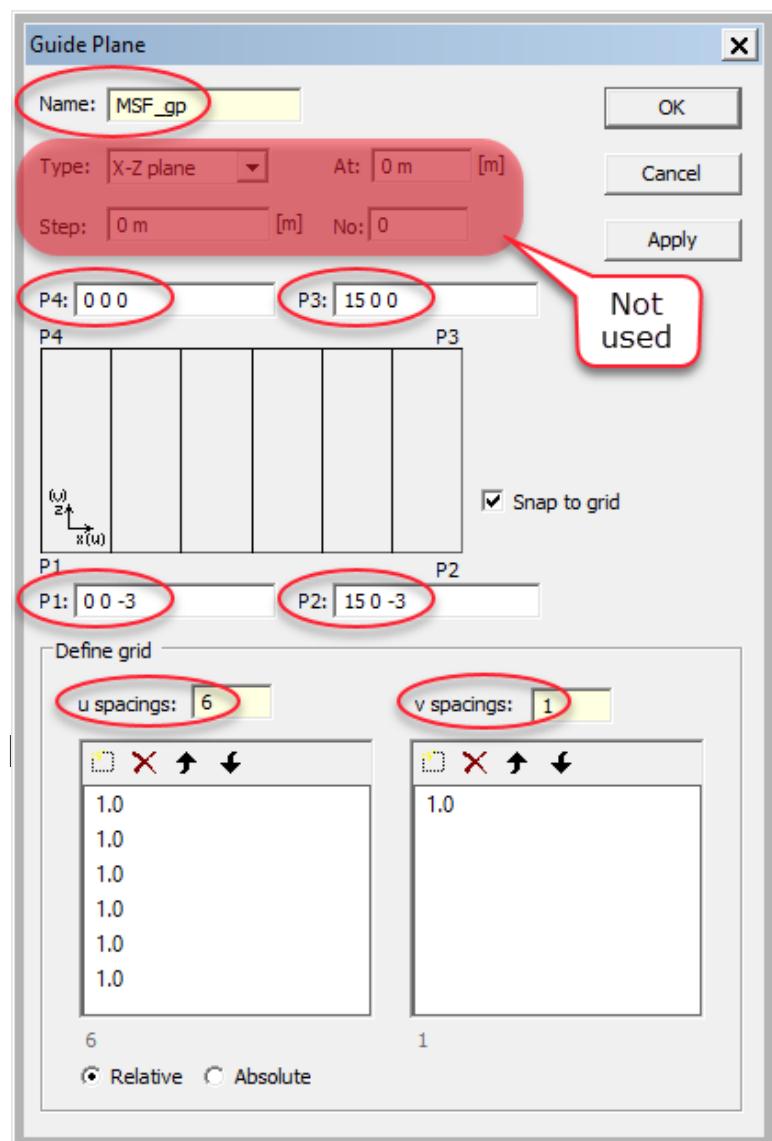
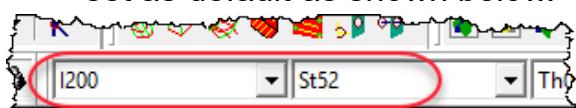
3.1 CREATE MAIN SUPPORT FRAME

- Use *Guiding Geometry | Planes | Guide Plane Dialog* to open the dialog and enter data as shown.

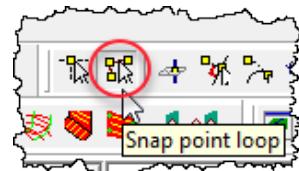
 - Easy entering of coordinates of points $P1 – P4$ is to enter data as shown for $P4$ and step through the fields using Tab.
 - Make sure the spacings are correct and click **OK**.

- The guide plane named **MSF_gp** (main support frame, guide plane) shown below should appear.
- Click in the guide plane to create the four beams shown below.

 - Use *Structure | Beams and Piles | Straight Beam* (or press the button ) to create beams.
 - The beam cross section should be I200 and the material St52 so ensure these are the ones set as default as shown below.



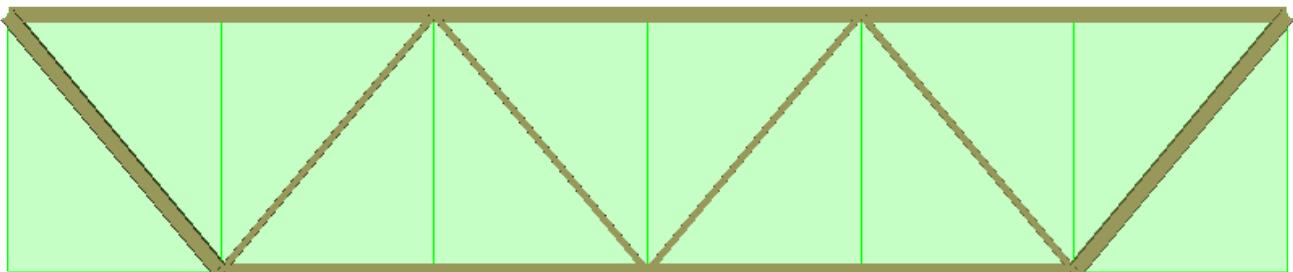
- Note that to create linked beams (chain of beams) as shown above the *Snap point loop* button may be pressed as shown to the right. This involves that the next beam to create starts where the previous ended, i.e. only click the beam end points. Use this method for the next beams.



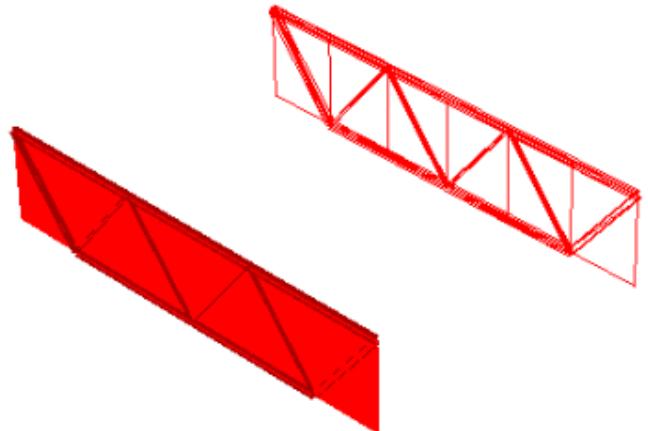
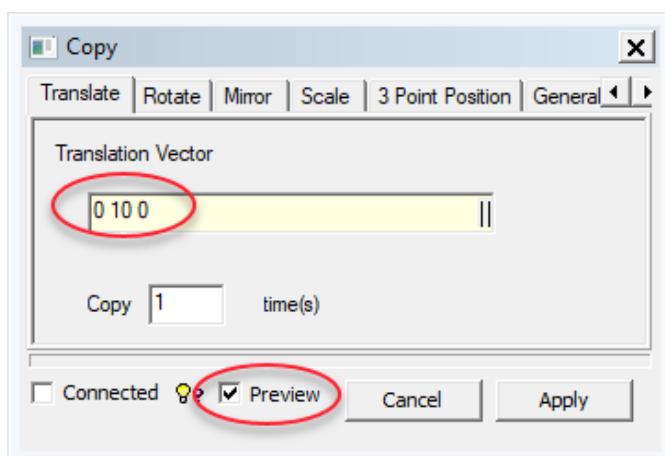
- Also be aware of the *Undo snap point* and *Clear snap points* buttons shown to the right. Use these to undo clicking last point or all points, respectively, when creating beams and plates.



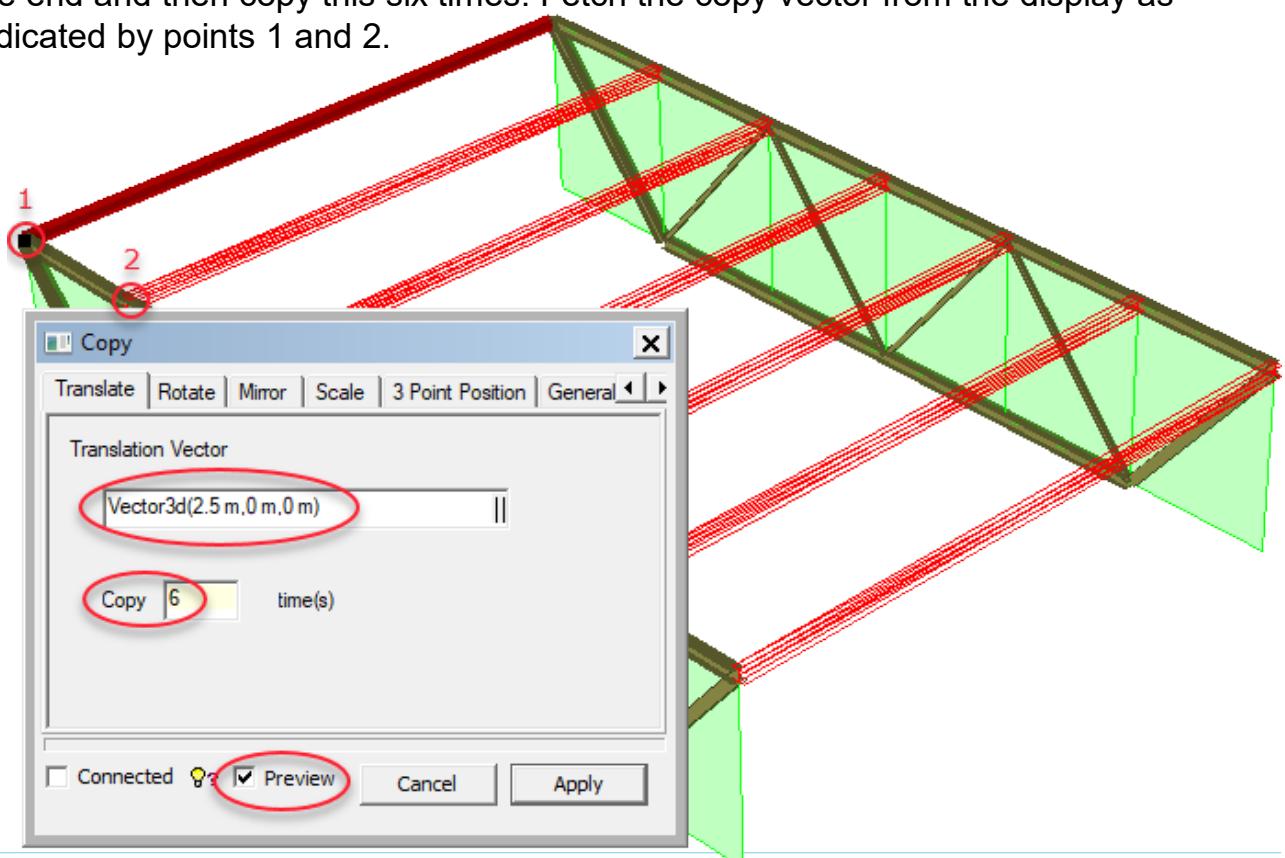
- Set default beam cross section to ISEC100:



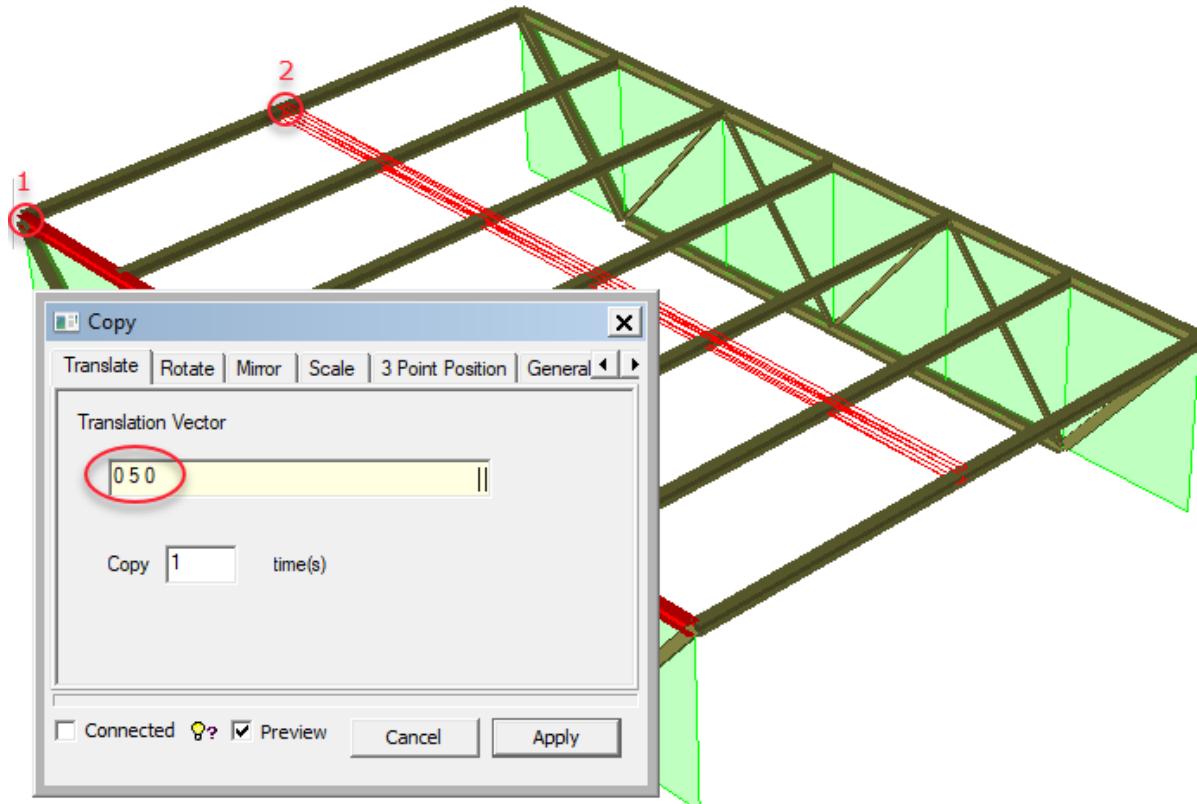
- Copy the guide plane and beams a distance of 10 m in Y-direction:



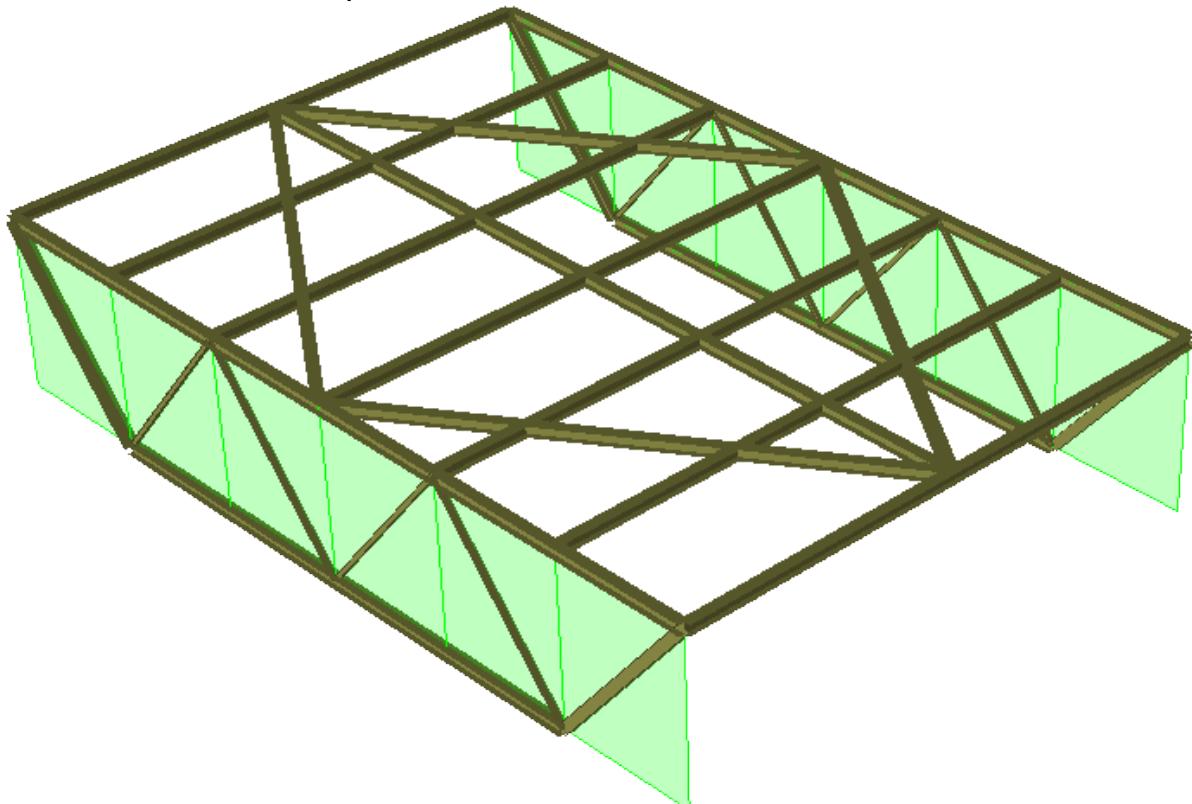
- Connect the two frames by horizontal beams with cross section I200. Create one at the end and then copy this six times. Fetch the copy vector from the display as indicated by points 1 and 2.



- Copy the top horizontal beam of the support frame as shown below. Again, fetch the copy vector from the display as indicated by points 1 and 2.

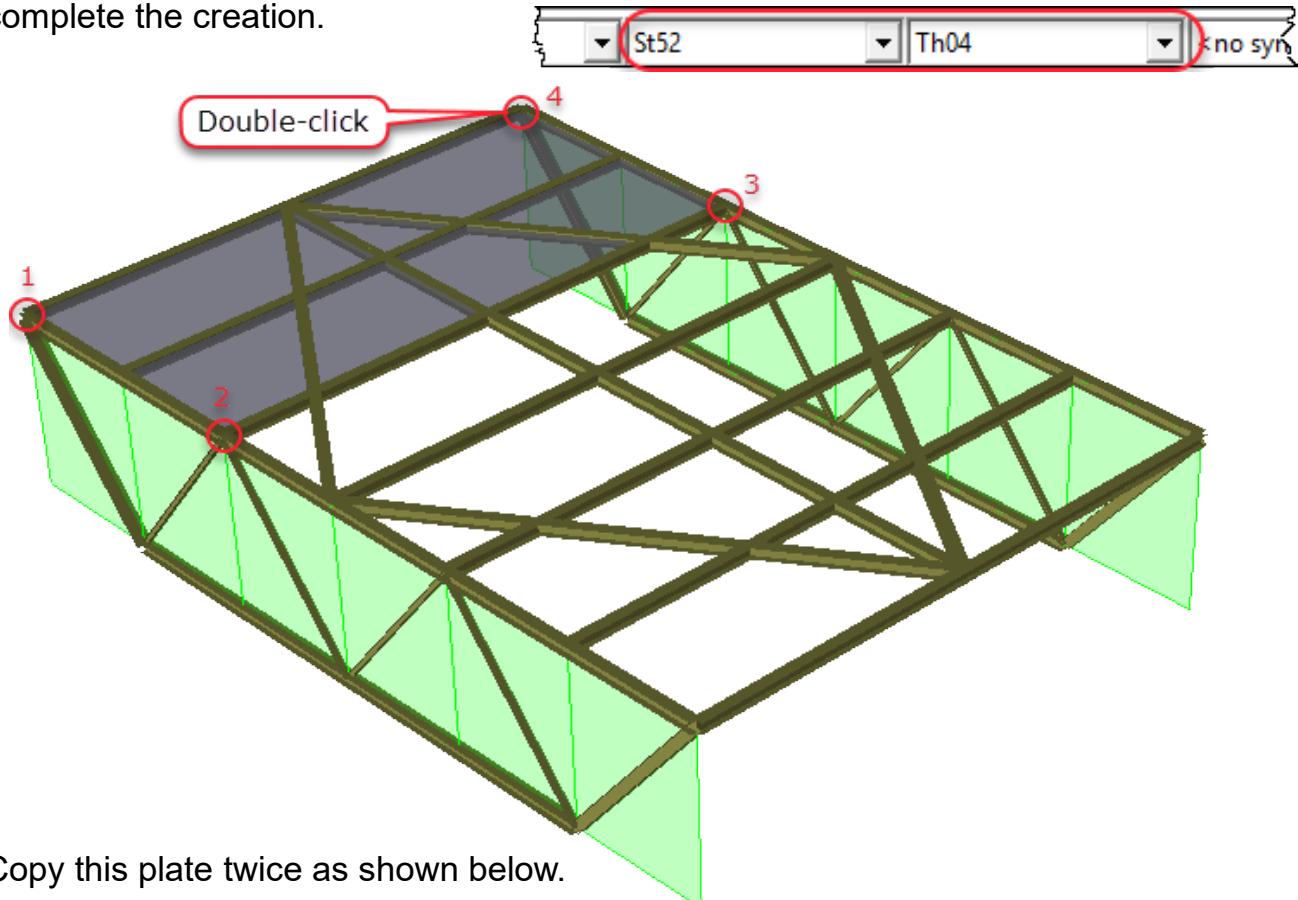


- Create the diamond shaped beams with cross section I200 as shown below.

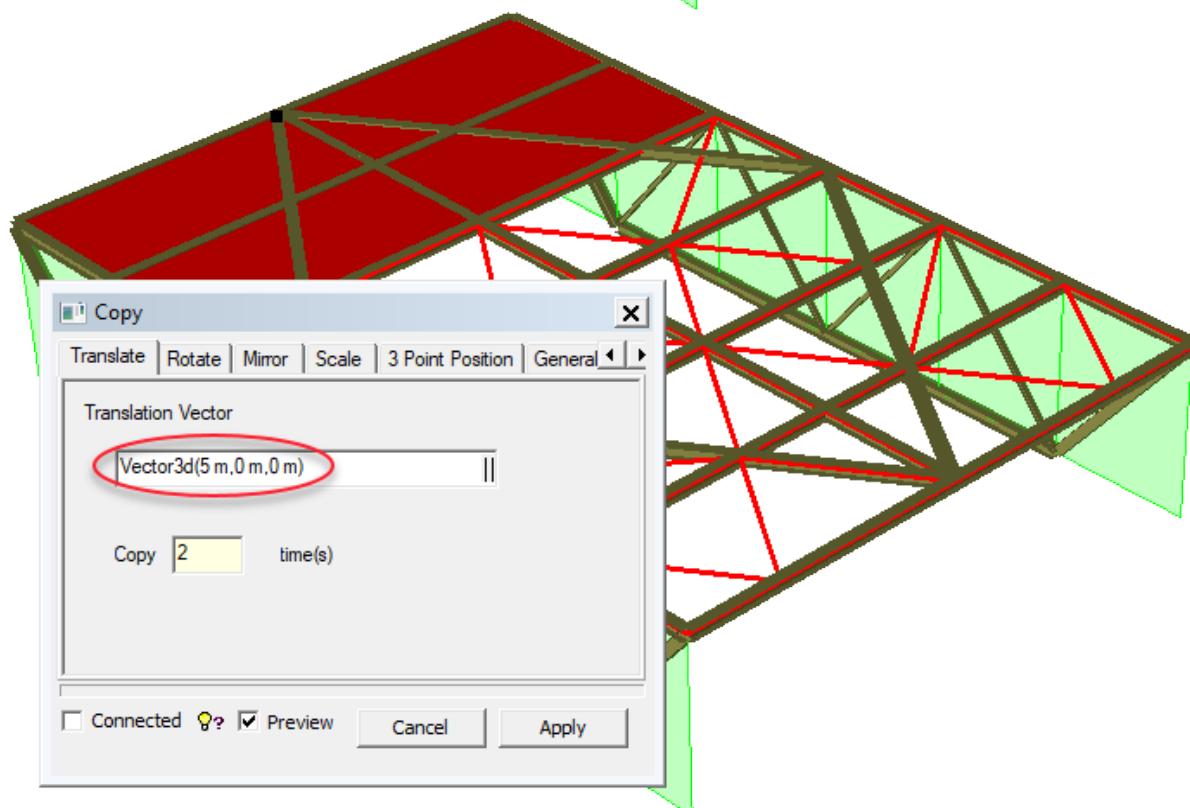


3.2 ADD PLATES

- Use *Structure | Flat Plates | Flat Plate* (or press the button ) to create a plate with thickness Th04 and material St52 as shown below. Double-click the fourth corner to complete the creation.



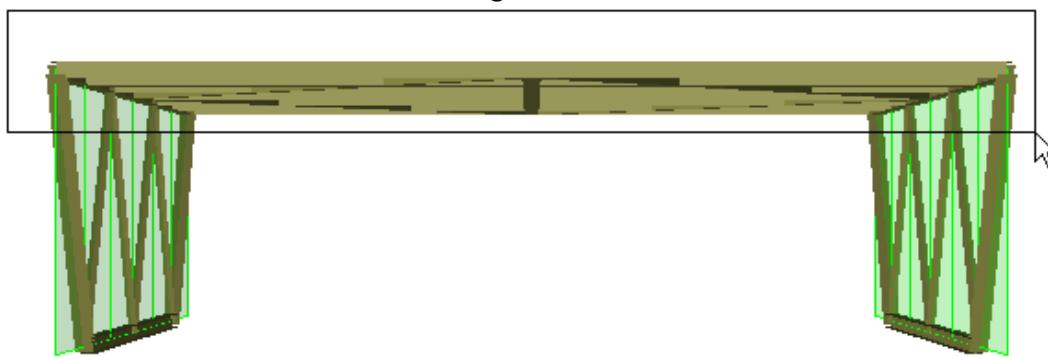
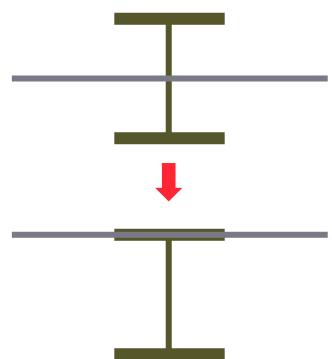
- Copy this plate twice as shown below.



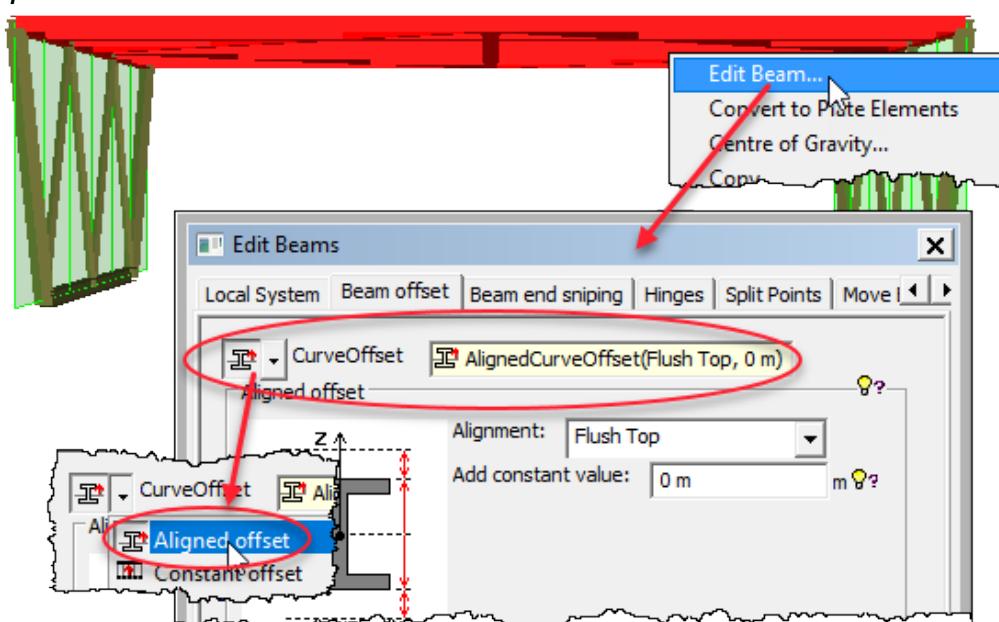
3.3 FLUSH STIFFENER BEAMS WITH DECK PLATE

- By default, the neutral axes of the horizontal beams in the deck coincide with the deck plate. This must be corrected so that the top flanges are flush with the deck.

- Select all beams of the deck, right-click and select *Edit Beams*.
 - Selecting all beams of the deck is easy when using F6 or F7 to view the model along the X- or Y-axis.



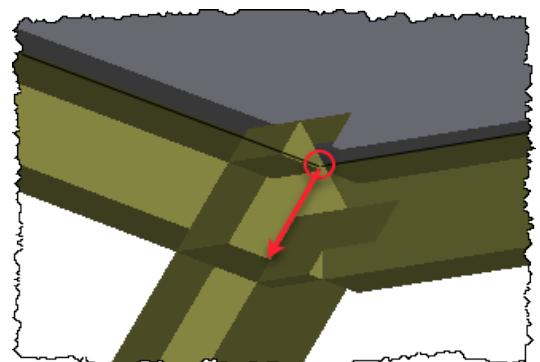
- In the *Edit Beams* dialog select *Aligned offset*, ensure that the *Alignment* is *Flush Top* and click *OK*.



- See that the top flange is flush with the plate.

- See also that the inclined beams in the vertical plane connect to the plate plane (where the FE node will be) – the point encircled in red.

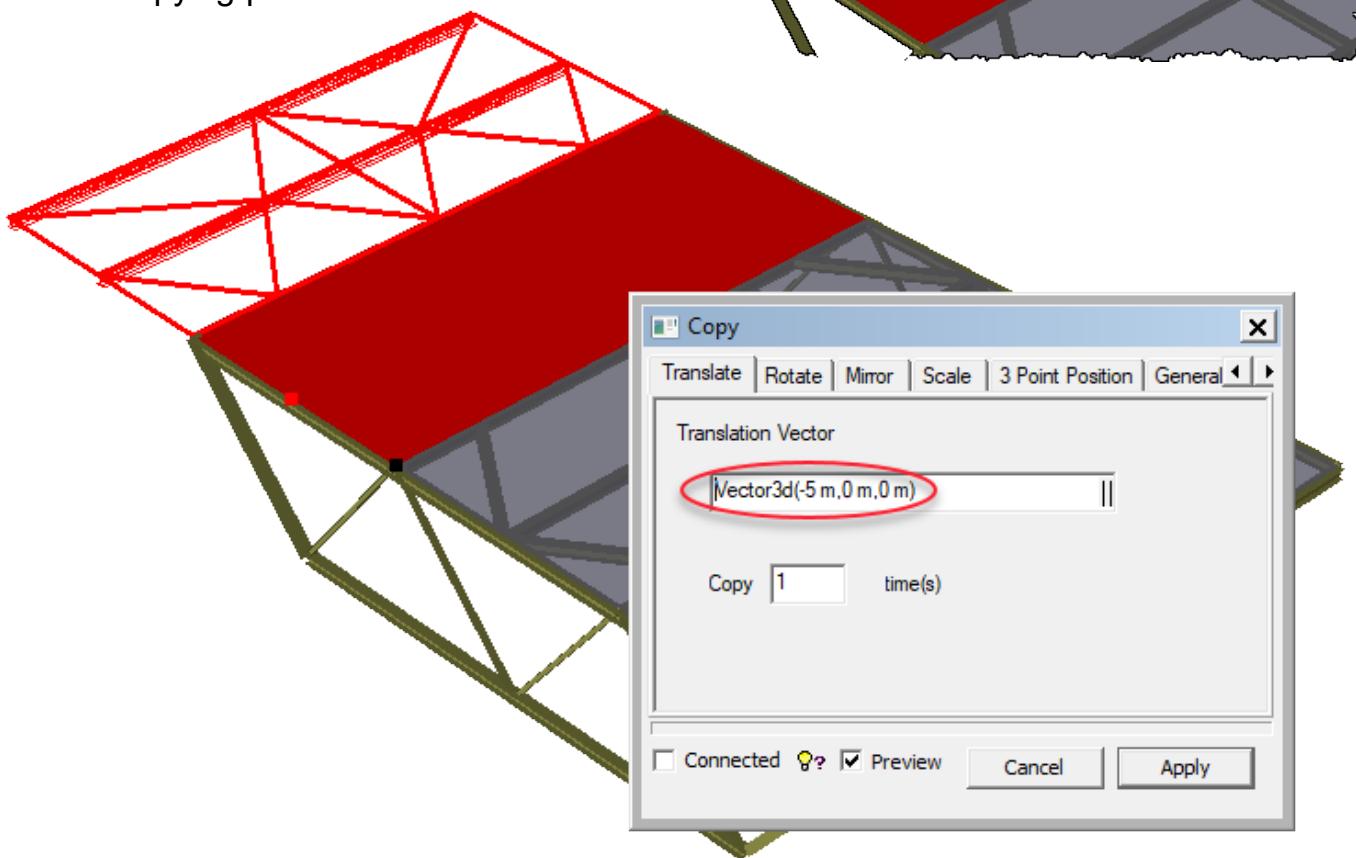
- This may be dealt with by giving these beams an end offset (eccentricity) as indicated in red. This is, however, neglected in this tutorial.



3.4 EXTEND THE DECK

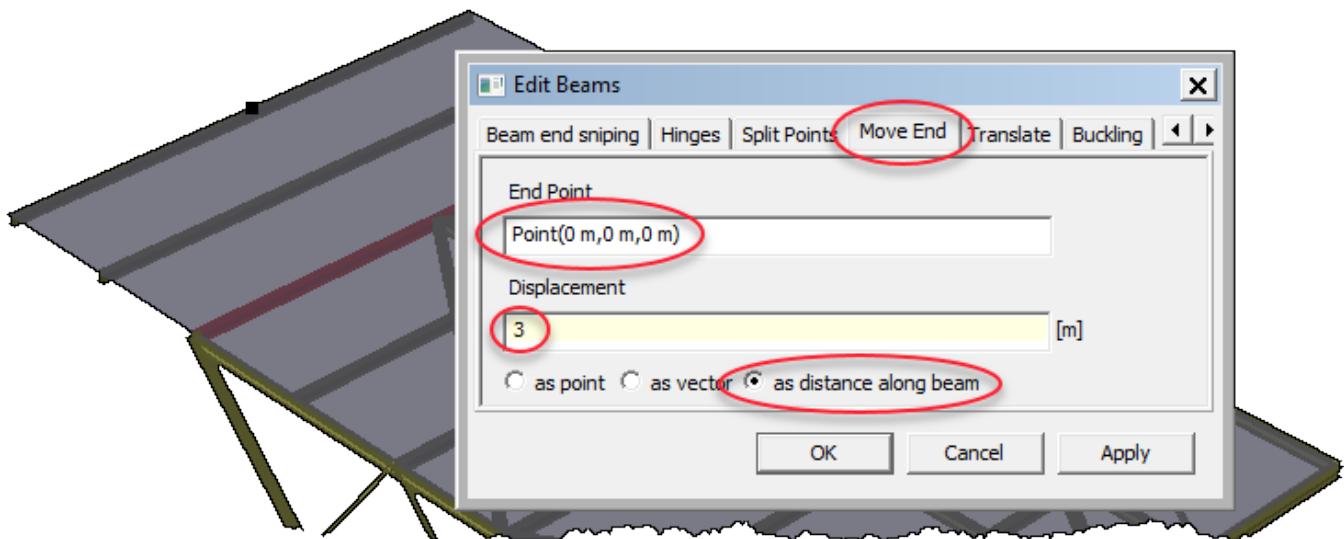
- Extend the deck by copying two stiffener beams and a deck plate -5 m in the X-direction.
- To enable selection of the stiffener beams that are hidden beneath the deck plate (unless you rotate the model) lift the *Plate selection* button as shown to the right.
- Then press the *Plate selection* button and Shift+click to select also the plate.

- The copying process is shown below.

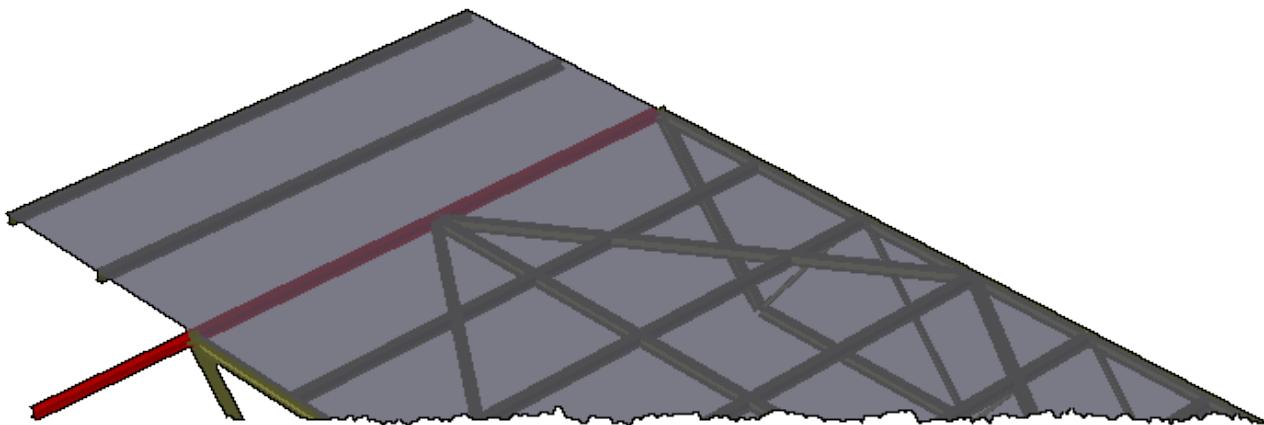


- The deck extension should be further extended in the negative Y-direction a distance of 3 m.
- First extend two of the beams.
- Select and right-click a beam, click *Edit Beam* and in the *Edit Beams* dialog go to the *Move End* tab. As *End Point* click the beam end to extend, select as *distance along beam* and enter 3 m.

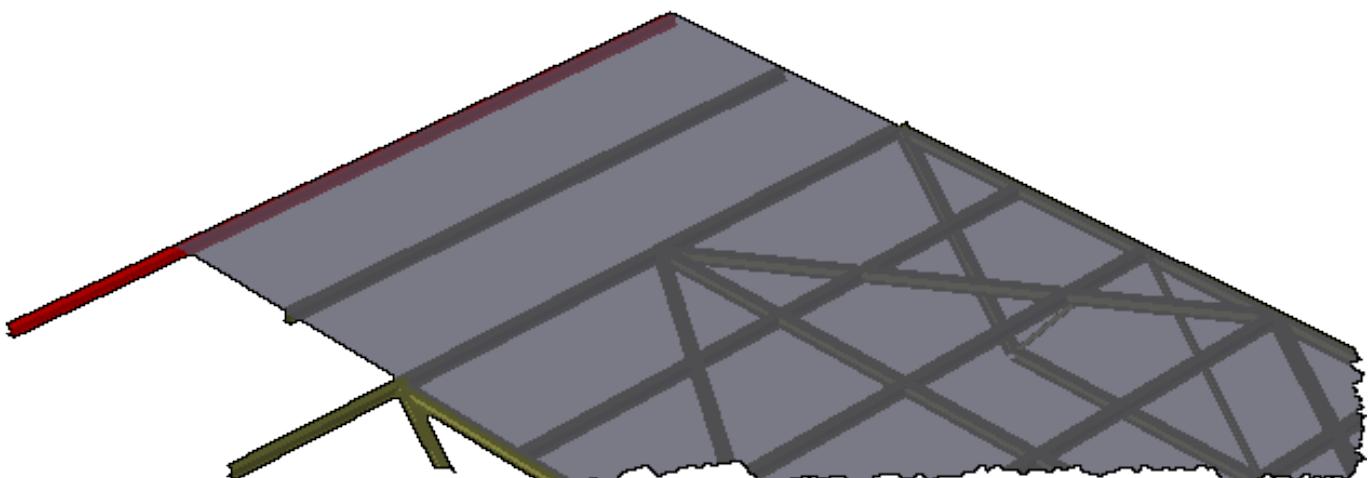
- The beam extension process:



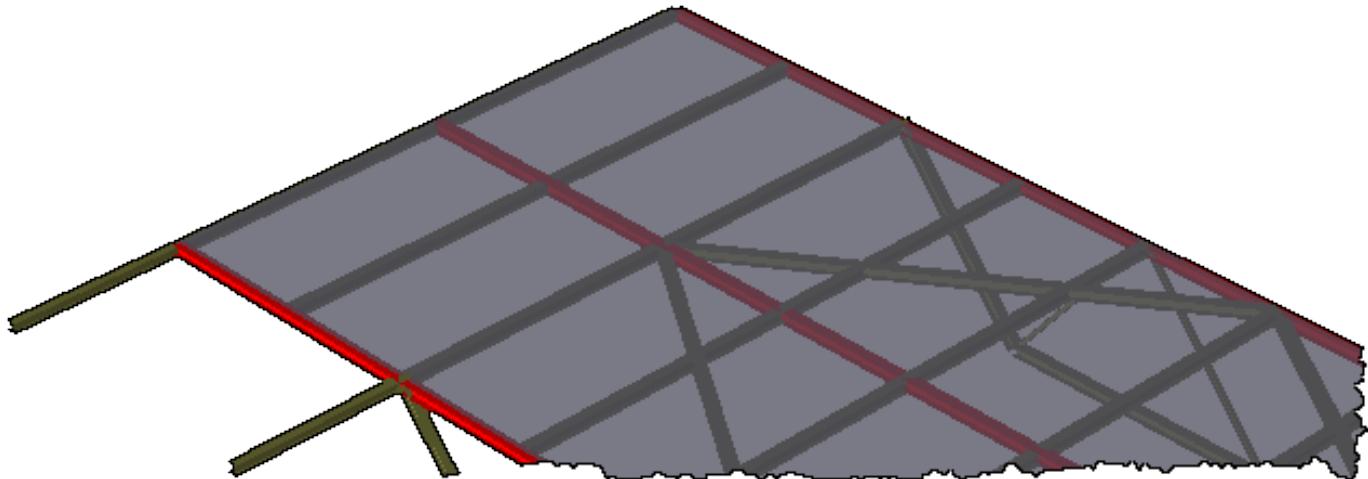
- The result:



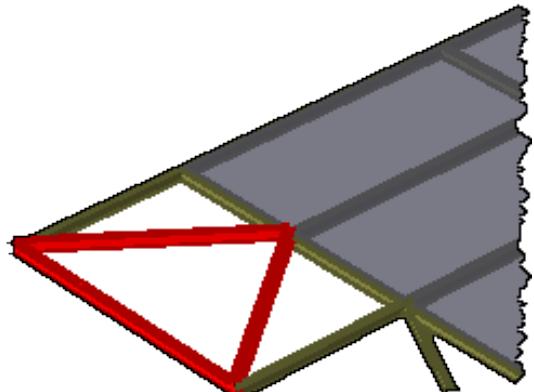
➤ Do the same for the beam at the edge of the deck:



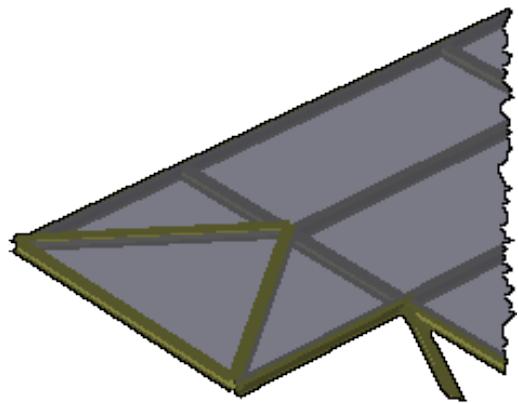
- Extend also the three highlighted beams shown below in X-direction so that the model becomes as shown. The extension for these beams is 5 m.



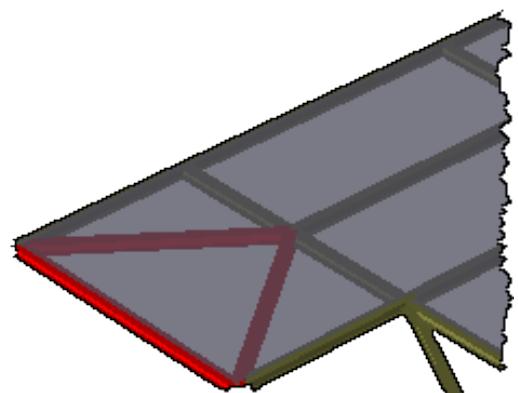
- Add the three beams shown to the right.
All with section I200.



- Add a plate with thickness Th04 as shown to the right.



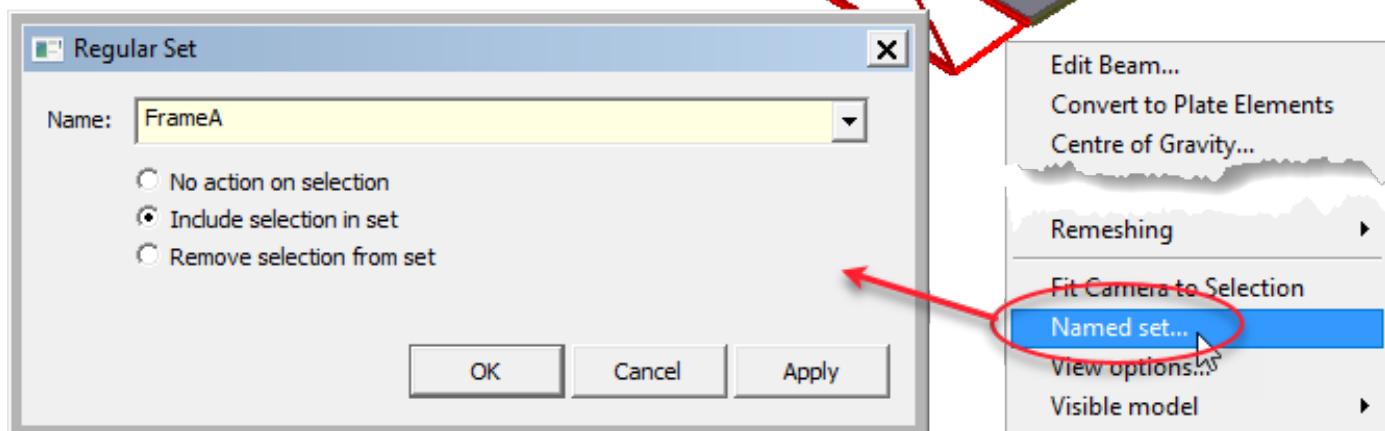
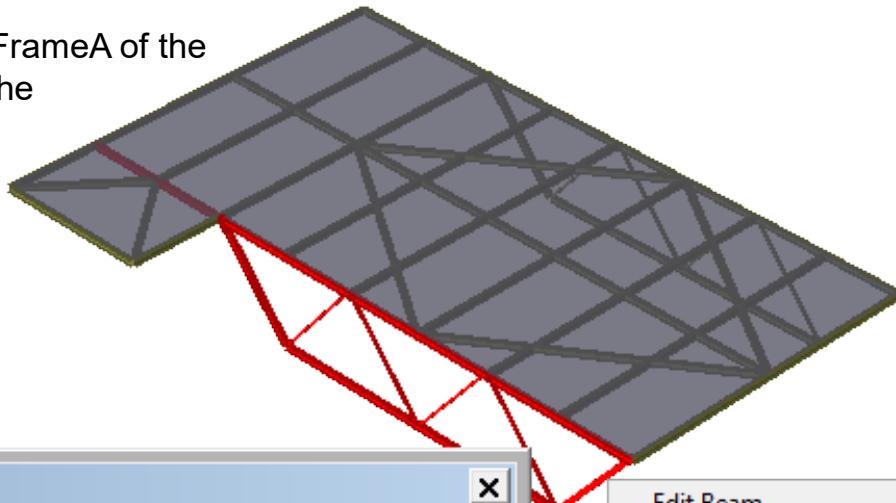
- The three beams shown to the right must be given offsets so that their top flanges are flush with the plate.



3.5 CREATE SETS

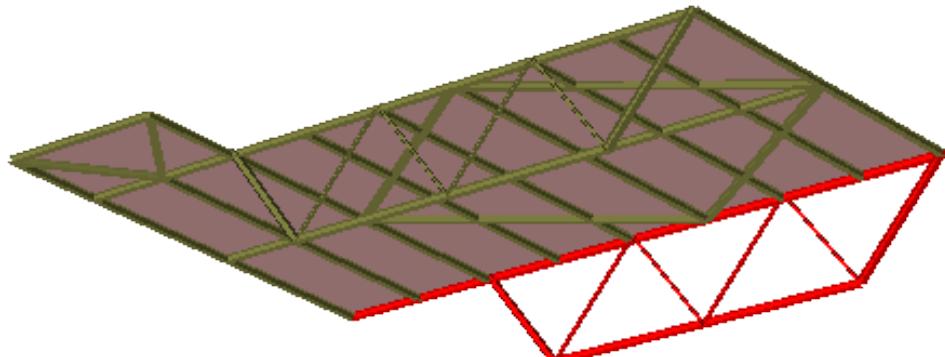
- Create a set named FrameA of the beams highlighted. The process is shown.

- The set appears in the browser in the folder *Utilities | Sets | Regular Sets*. Confirm this.

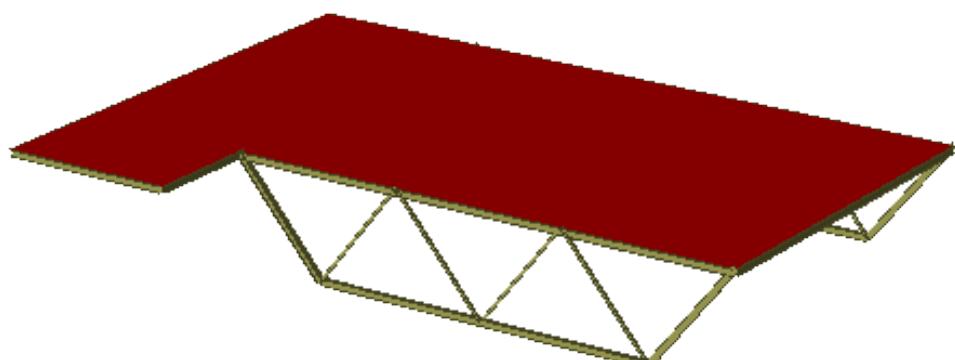


- Create more sets:

- FrameB shown to the right.

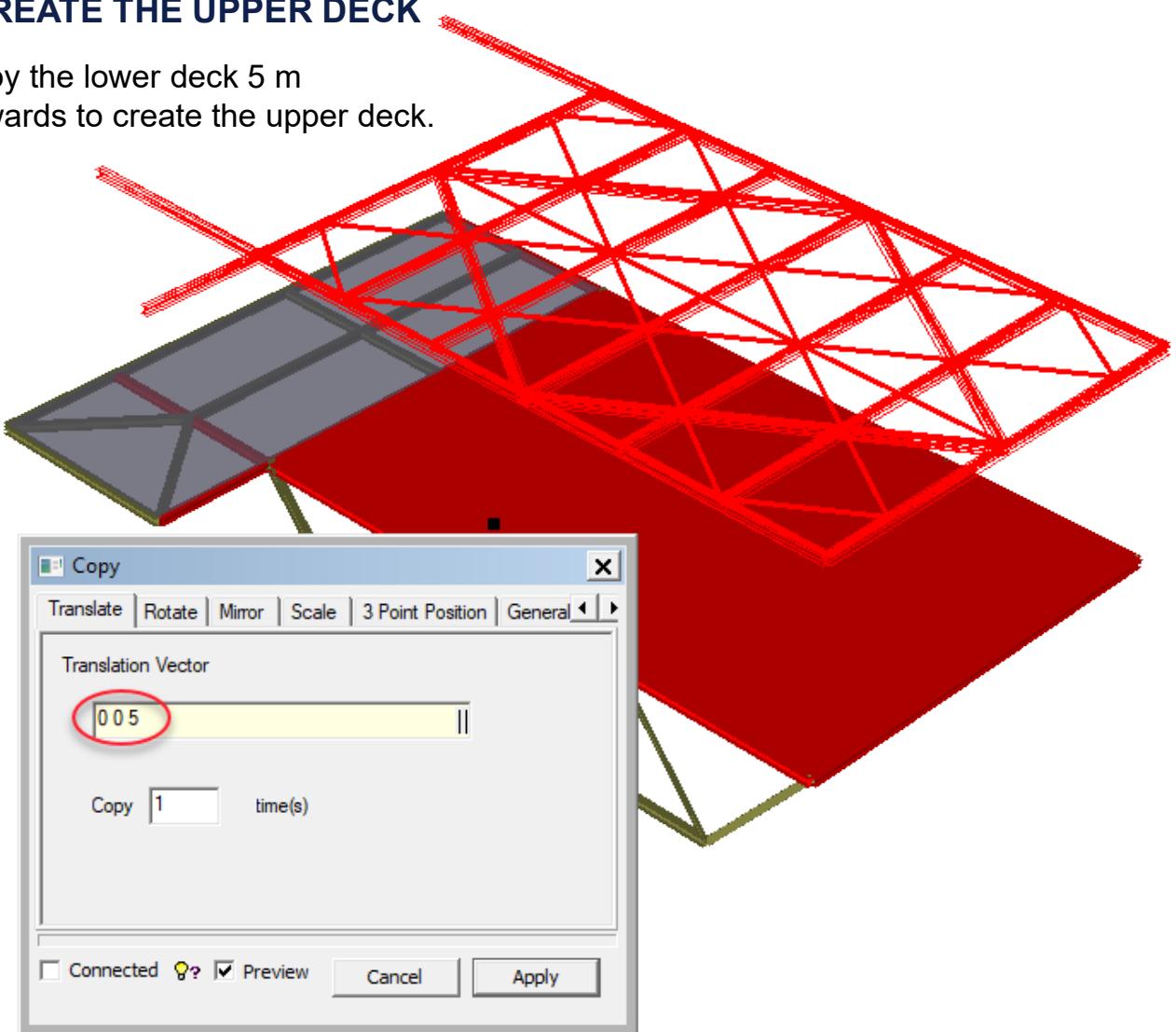


- L_Deck shown to the right. Include only the deck plate and no beams.

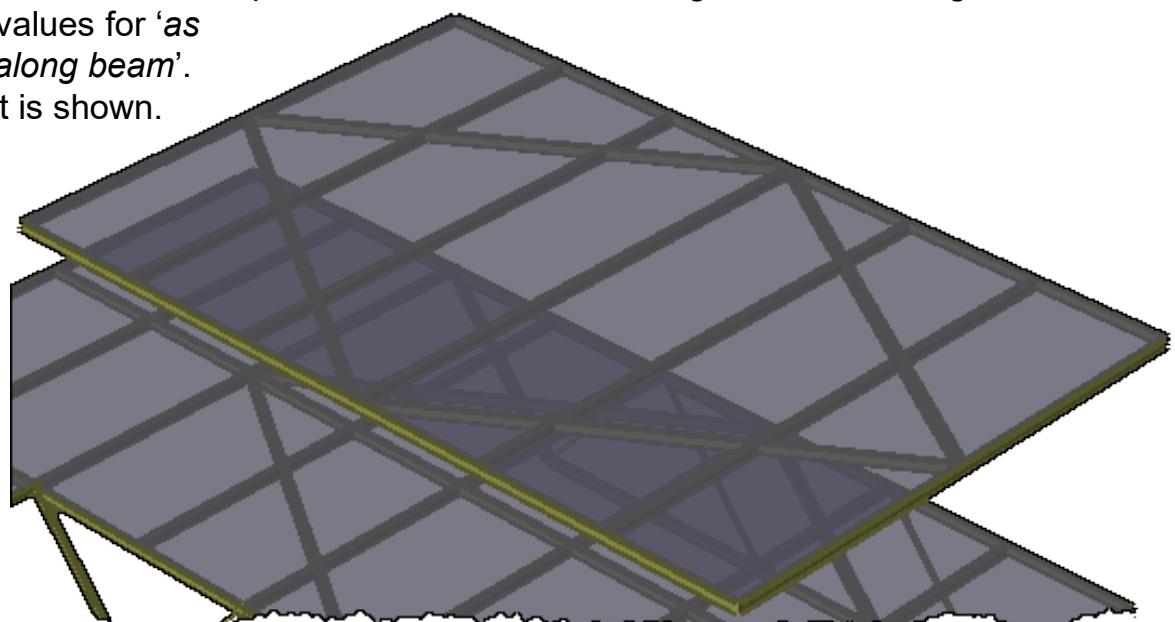


3.6 CREATE THE UPPER DECK

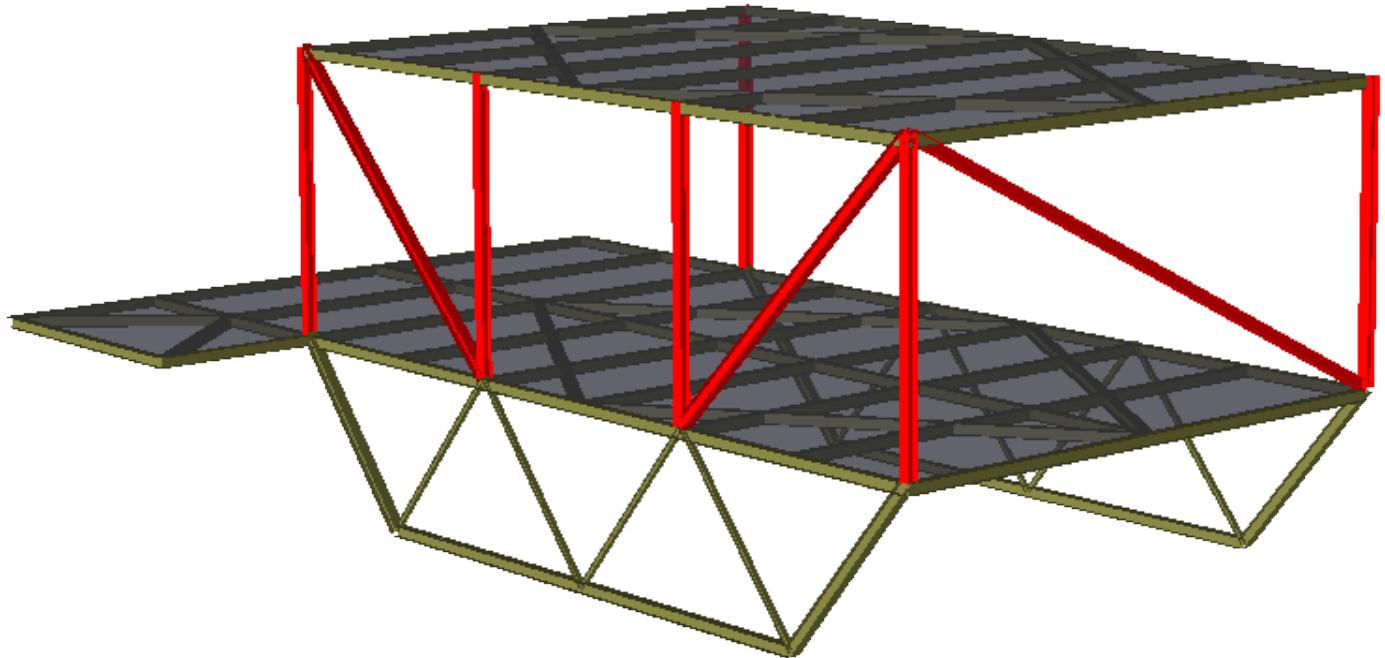
- Copy the lower deck 5 m upwards to create the upper deck.



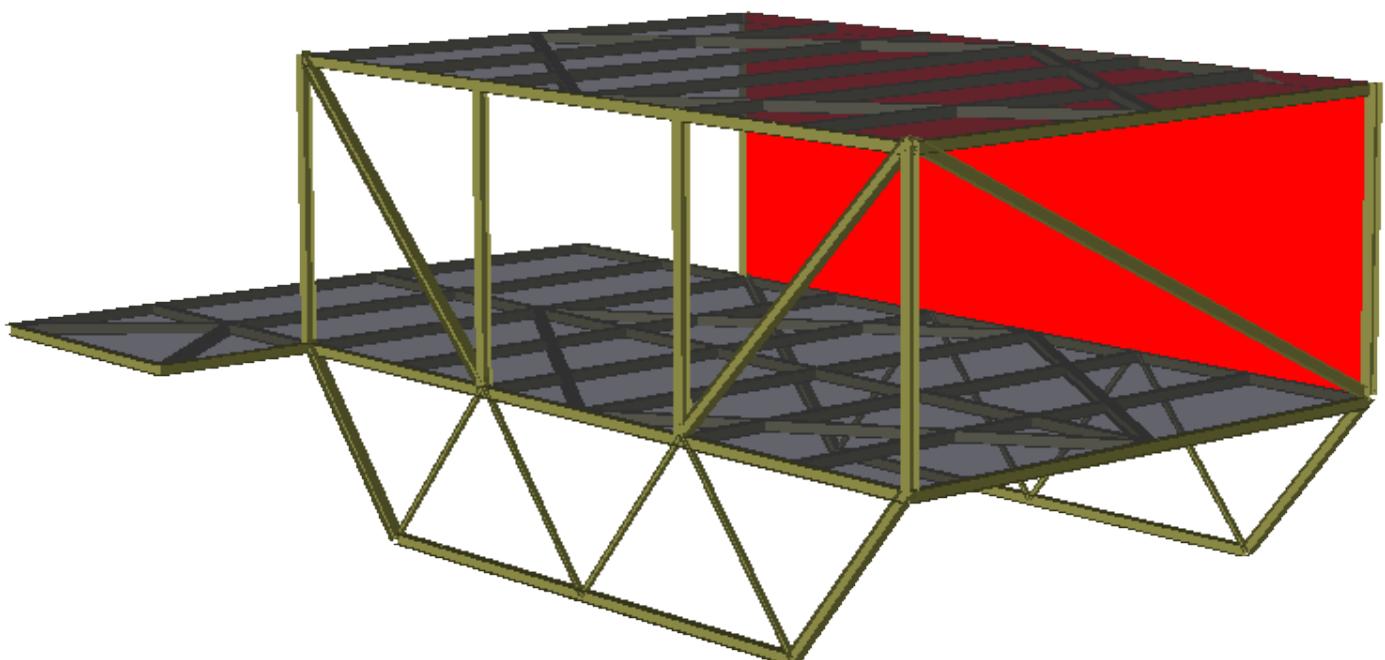
- The upper deck shall have no extension, so the three protruding beams must be shortened. Use the same procedure as when extending the beams but give negative values for 'as distance along beam'.
The result is shown.



- Create a set named U_Deck of the plates in the upper deck.
- Create columns and diagonal bracings as shown between the lower and upper decks. All with beam cross section I200.

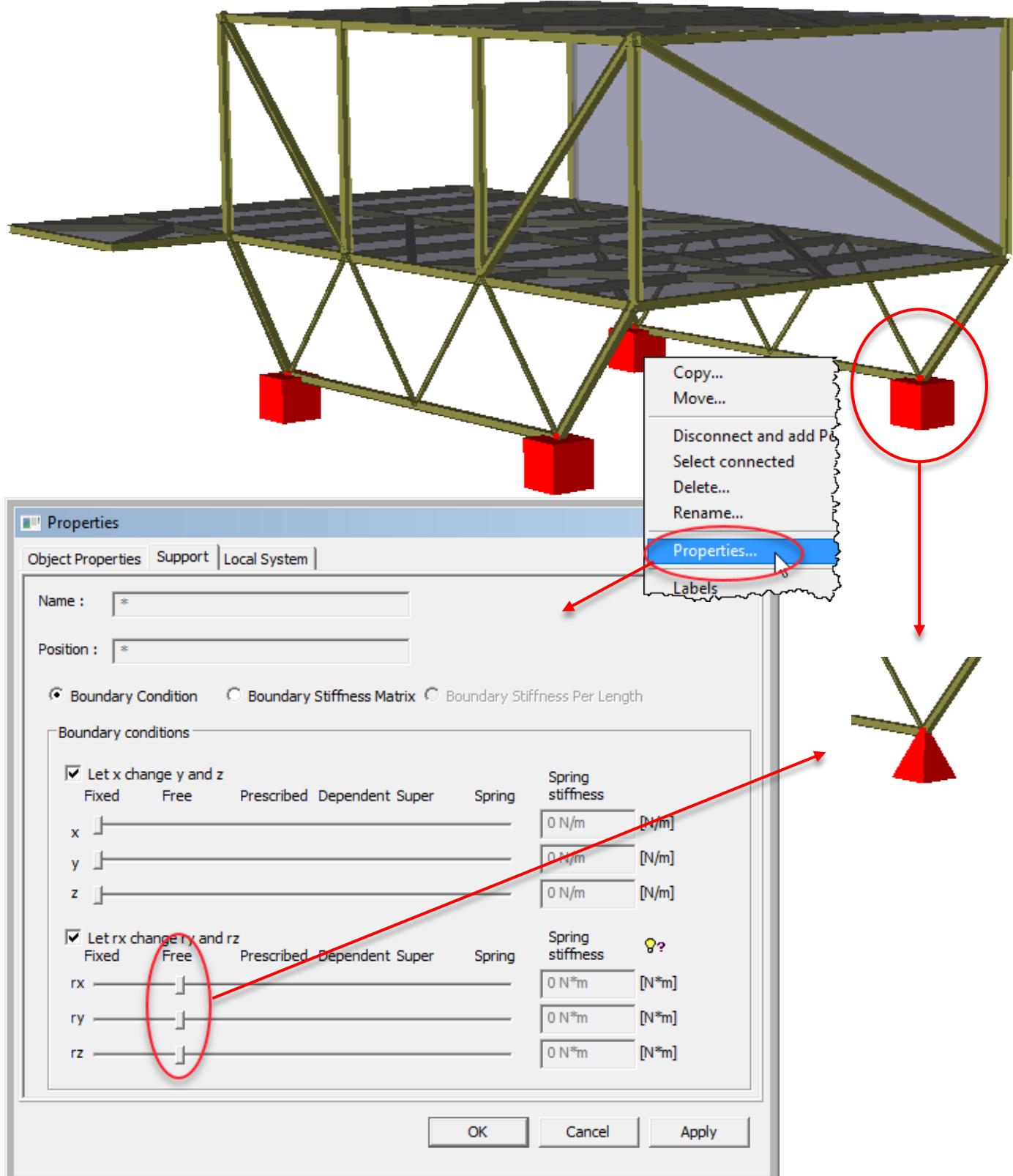


- Create a vertical plate with thickness Th03 between the two decks as shown.



4 ADD BOUNDARY CONDITIONS

- The topside rest on four points. All with free rotations. Use *Structure | Support | Support Point* (or press ) to add the supports followed by selecting them, right-clicking and adjusting the individual boundary conditions. All supports should appear as pyramids.

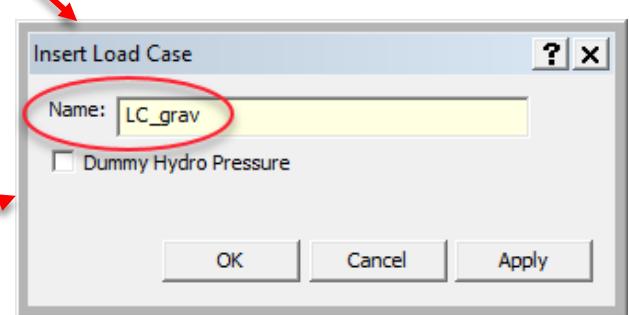
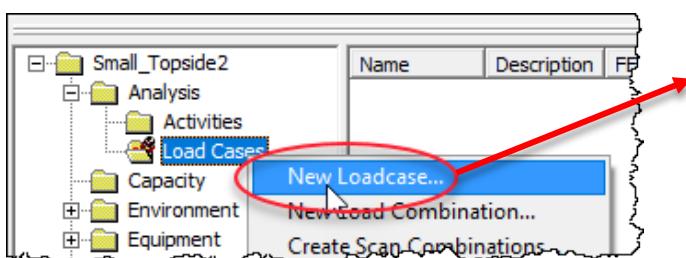


5 CREATE LOADS

- Modelling equipments and loads typically include:
 - Create load cases. These are empty containers until filled with contents.
 - Define a load case as gravity load case.
 - Create equipments and place in appropriate load case.
 - Import weight list and place in appropriate load cases.
 - Include explicit loads (point, line and surface loads) in appropriate load case.
 - Create load combinations.
 - Check load sums.

➤ Create load case LC_grav by *Loads | Load Case*.

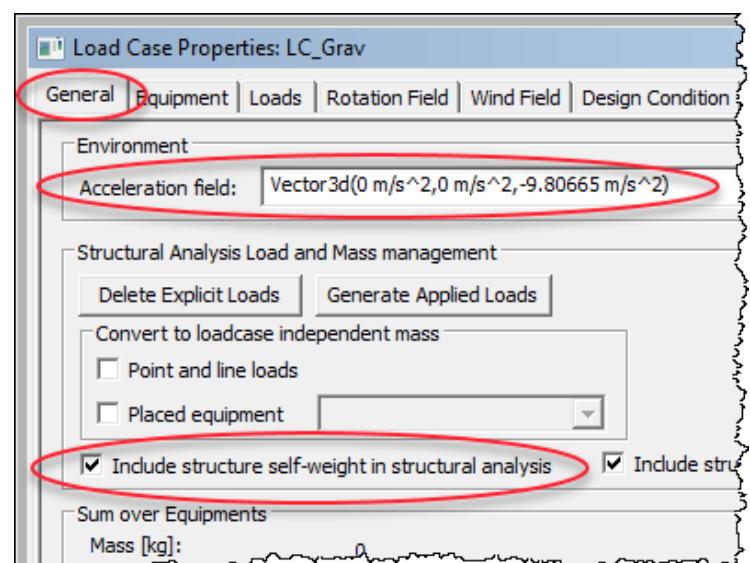
- An alternative way of opening the *Insert Load Case* dialog is to right-click the *Analysis | Load Cases* folder:



➤ Create three more load cases:

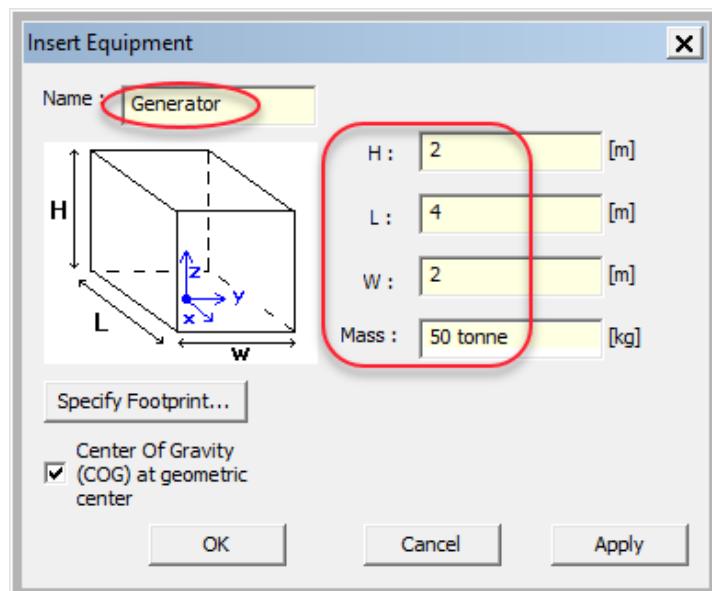
- LC_eqpm
- LC_list
- LC_expl

➤ Make the load case LC_grav the gravity (self weight) load case by right-clicking it and in the *General* tab of the *Load Case Properties* dialog check *Include structure self-weight in structural analysis* as shown to the right. The *Acceleration field* is by default filled in with the acceleration field of gravity.

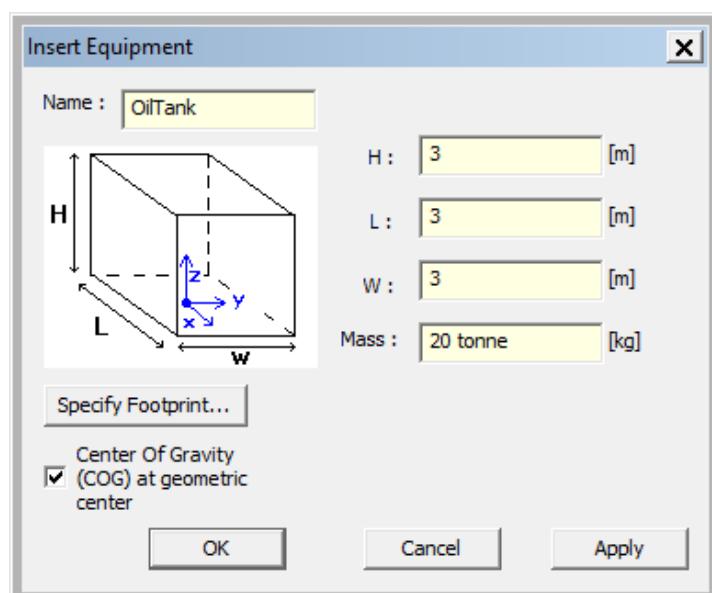


5.1 CREATE AND PLACE EQUIPMENTS

- Use *Loads | Prismatic Equipment* to create an equipment named Generator with data as shown below.



- The height, length and width determine the default centre of gravity and default footprint (that both may be changed) but have otherwise merely display purposes. The footprint determines the load distribution to the supporting beams.
- Create another equipment named OilTank as shown below.



- Find the equipments in the *Equipment* folder.

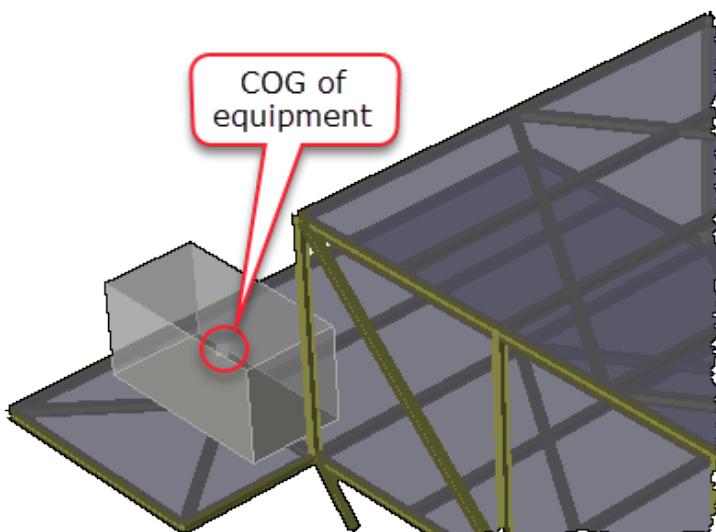
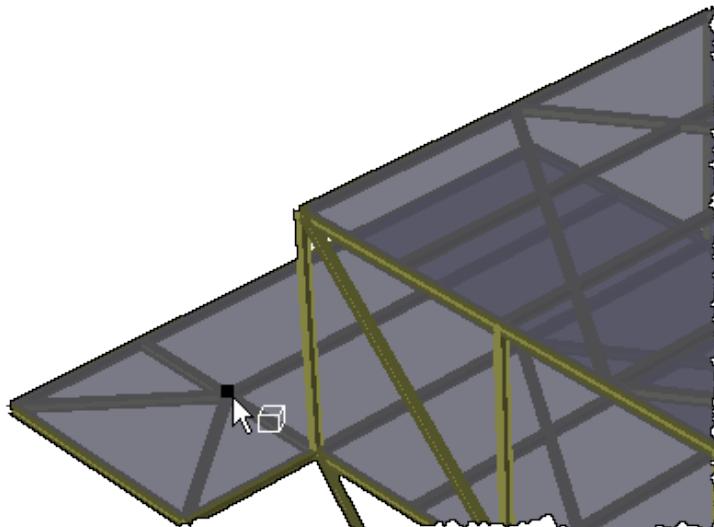
Name	Description	X [m]	Y [m]	Z [m]	Equi
Generator	Prism Equipment	-2.5	0	0	500
OilTank	Prism Equipment	10	8.33333	0	200

➤ Place both equipments in the load case LC_eqpm.

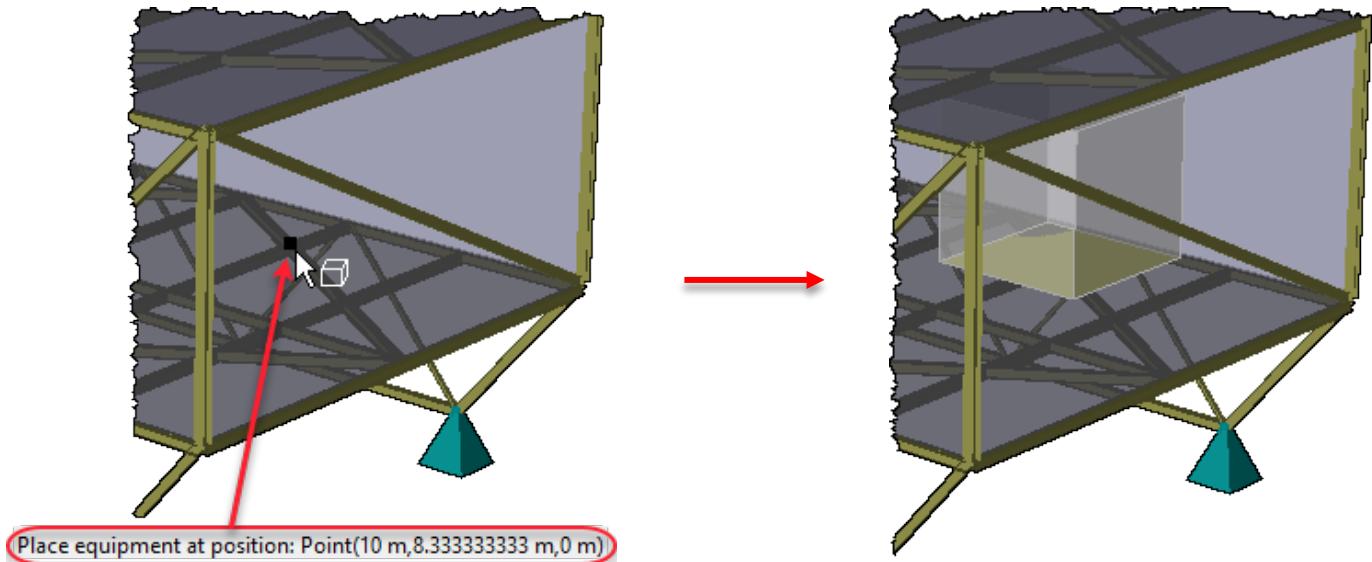
- First make this load case the currently selected one as shown to the right.
- Then place the equipment Generator in the selected load case and in the model by the operation:
 - Right-click the equipment in the *Equipment* folder in the browser and select *Place in Loadcase* as shown to the right.
 - Click in the model where the midpoint of the bottom surface of the equipment should be positioned. Notice the cube symbol when hovering over the model.
 - The centre of gravity (COG) of the equipment is shown as a small light grey square.

Name	Description	FEM Loadcase	FEM LC R
R-Kr LC_Grav	LoadCase	1	Manual
LC_eqpm	LoadCase	2	Manual
R-Kr LC_list	LoadCase	3	Set Current
R-Kr LC_expl	LoadCase	4	Move to Load

Name	Description	X [m]	Y [m]	Z [m]
Generator	Prism Equipment			
OilTank	Prism Eq			
Weight Lists	Folder			

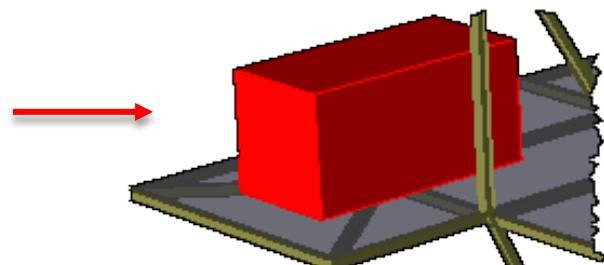
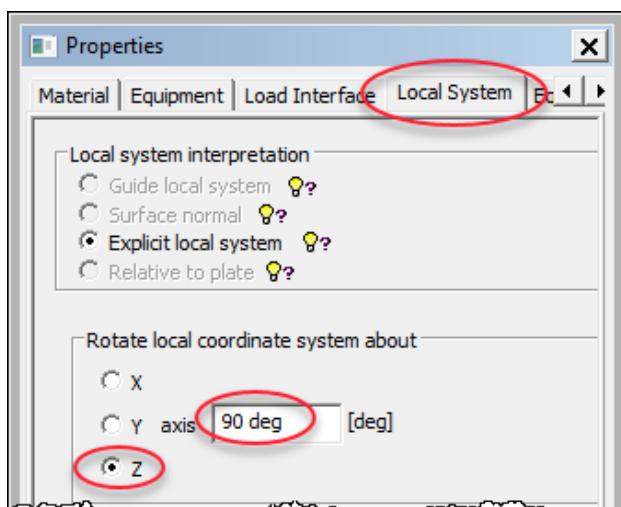
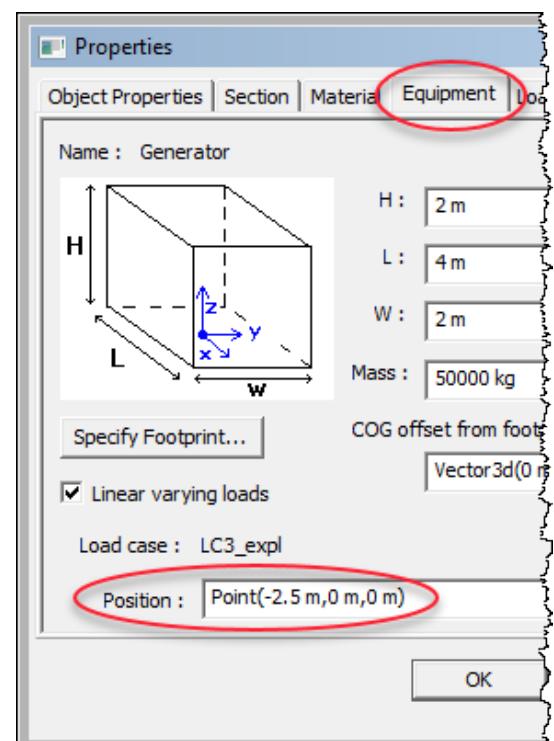


- Place the equipment OilTank at the point (10,8.3333,0). When hovering the mouse the coordinate appears in the lower left corner of the GeniE window.

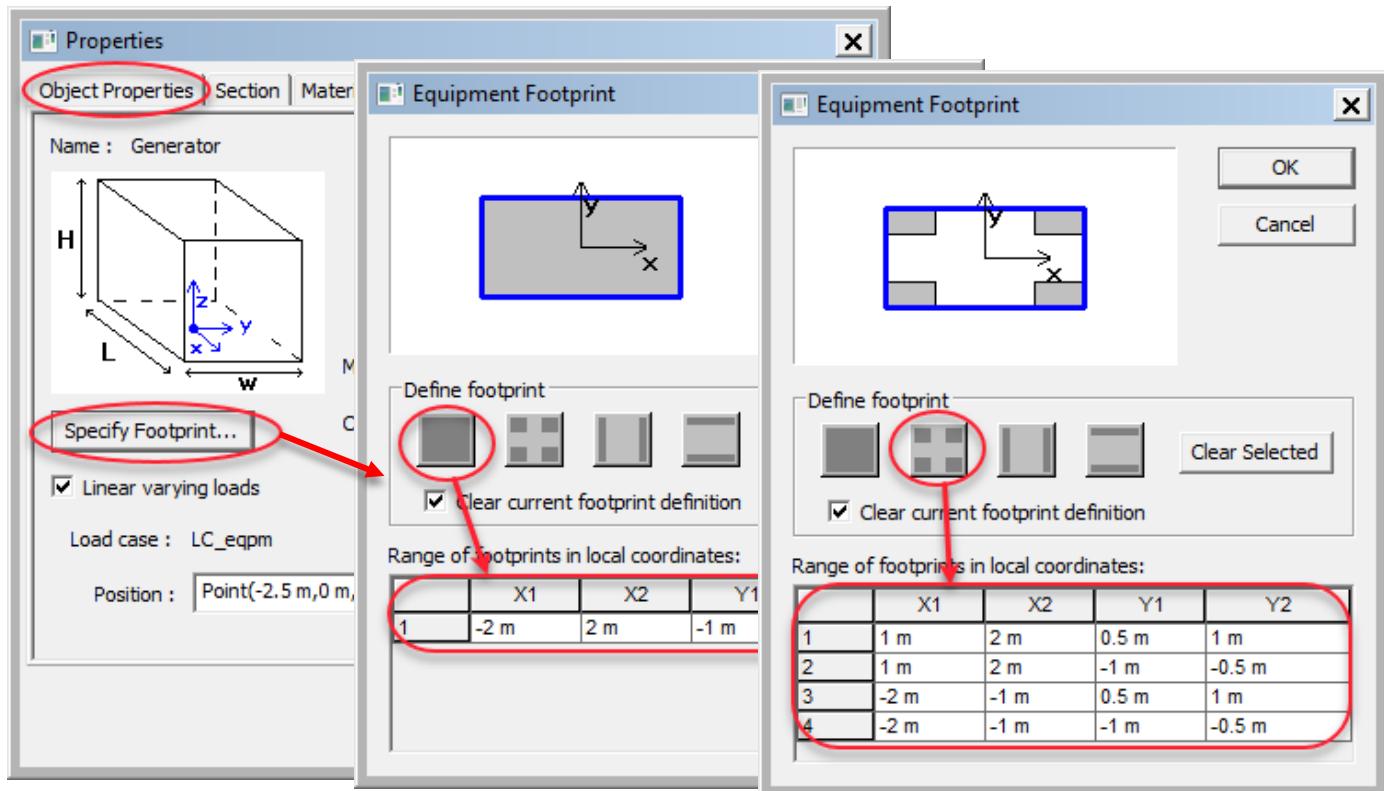


Both equipments have been placed by clicking in the model. Being restricted to clickable points, these may not be the exact positions. Moreover, equipments may need to be rotated.

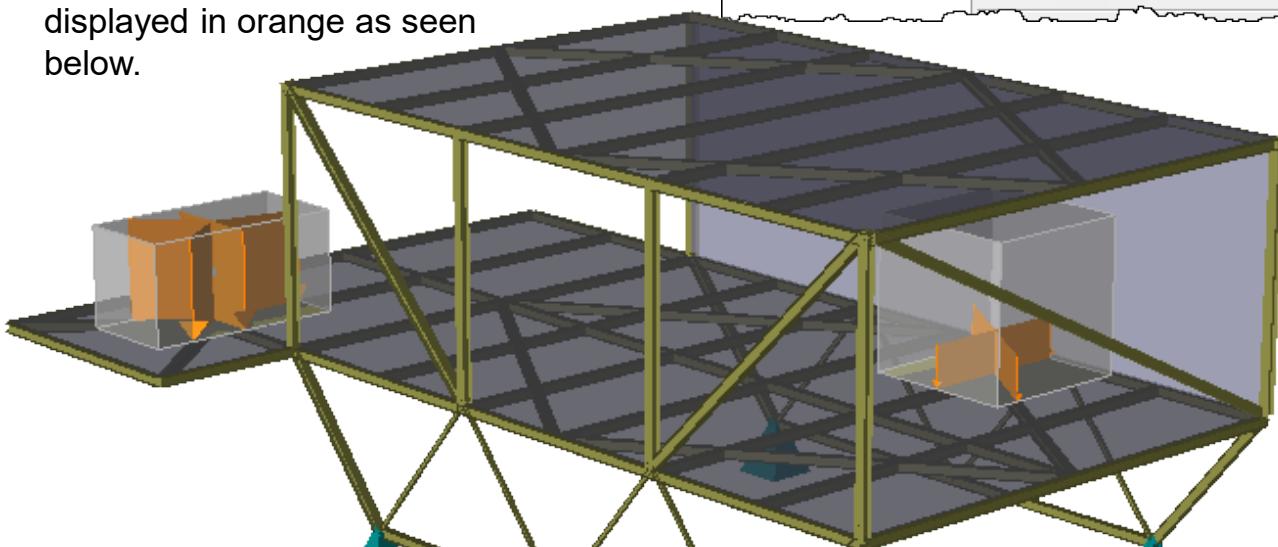
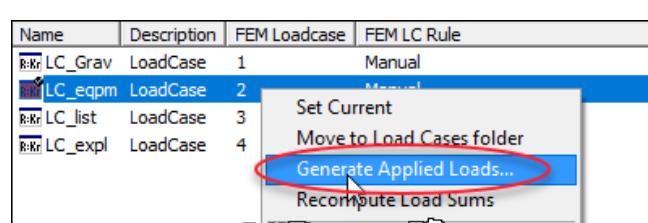
- Position the Generator correctly by right-clicking it and selecting *Properties*. In the *Equipment* tab of the *Properties* dialog shown to the right, notice that not only the *Position* but also the height, length, width and mass may be adjusted.
- Accept the position but rotate the equipment 90° about the vertical axis using the *Local System* tab as shown below.



- The footprint of the equipment determines the load distribution to the supporting beams. By default, the bottom surface of the equipment is the footprint. I.e. all beams touched by the bottom surface are loaded.
- The footprint may be adjusted by right-clicking the equipment and clicking the *Specify Footprint* button as shown below. Keep default footprint for equipments.



- The line loads on supporting beams may be displayed by right-clicking the load case containing the equipments and selecting *Generate Applied Loads* as shown to the right. The loads are displayed in orange as seen below.



5.2 IMPORT AND PLACE WEIGHT LIST

- Make LC_list the currently selected load case.
- The process for a weight list is as follows:
 - Import the weight list.
 - Create *Bounding Box Equipment* objects.
 - Place the *Bounding Box Equipment* objects in a load case.
- Use *Loads | Import Weight List* to import the weight list named *Weight_list_with_size.xml* found in the installation folder typically named <path>\GeniE VX.Y-ZZ\Help\Tutorials\TutorialsBasicAndCodechecking\B2_GeniE_Small_Topside\JS
 - A weight list is useful as a simplified alternative to creating several equipments when less control of the load distribution to supporting beams is required.
 - The weight list is typically created by the user as an XML file:

```

<weight_report name='dimension'>
  Name of weight list

  <weight_item name='LPumpA' description="Light pump">
    Name of weight list item
    <position x="-2" y="-3" z="1" />
    <dimension dx="3" dy="4" dz="3"/>
  </weight_item >
  <weight_item name="LPumpB" description="Light pump">
    <position x="-2" y="1" z="0.5"/>
    <weight dry="400"/>
    <dimension dx="2" dy="5" dz="4"/>
  </weight_item >
  <weight_item name="LPumpC" description="Light pump">
    <position x="-1" y="2" z="0.5"/>
    <weight dry="600"/>
    <dimension dx="1" dy="4" dz="6"/>
  </weight_item >
  <weight_item name="LPumpD" description="Light pump">
    <position x="-3" y="3" z="1.5"/>
    <weight dry="500"/>
    <dimension dx="2" dy="3" dz="3"/>
  </weight_item >

  <weight_item name="HPumpA" description="Heavy pump">
    <position x="1" y="5" z="5.5"/>
    <weight dry="1500"/>
    <dimension dx="3" dy="2" dz="2"/>
  </weight_item >

```

- Having imported the weight list, it appears in the *Equipment | Weight Lists | dimension* folder of the browser as shown to the right. (*dimension* is the name of the weight list.)

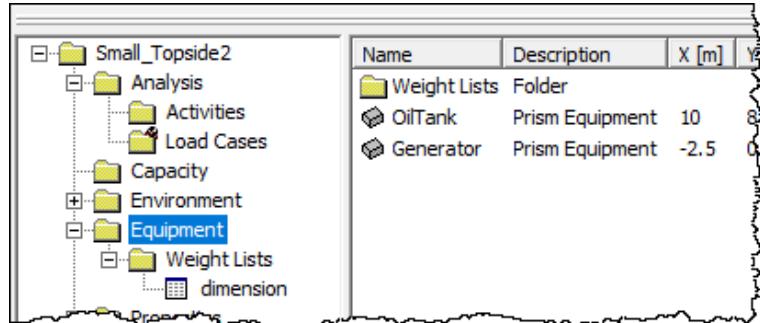
The screenshot shows the GeniE software interface. On the left, the browser tree displays a project structure with 'Small_Topside2' at the root, followed by 'Analysis', 'Activities', 'Load Cases', 'Capacity', 'Environment', 'Equipment', 'Weight Lists', and 'Properties'. The 'dimension' folder under 'Weight Lists' is highlighted with a red circle. On the right, a table lists equipment items with their names, descriptions, groups, total weights, and dry weights. The table has columns for Name, Description, Group, Total Weight [kg], and Dry Weight [kg].

Name	Description	Group	Total Weight [kg]	Dry Weight [kg]
HPumpA	Heavy pump		1500	1500
HPumpB	Heavy pump		1500	1500
HPumpC	Heavy pump		1500	1500
HPumpD	Heavy pump		1500	1500
LPumpA	Light pump		800	800
LPumpB	Light pump		400	400
LPumpC	Light pump		600	600
LPumpD	Light pump		500	500

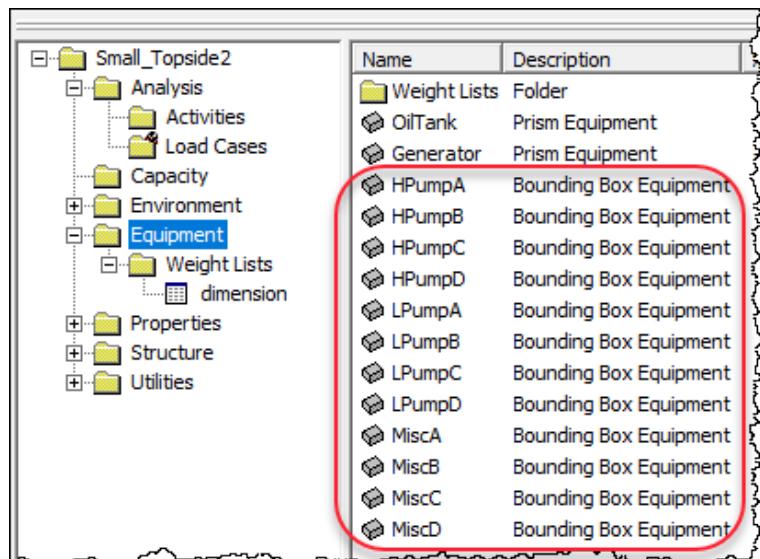
- Create so-called *Bounding Box Equipment* objects for all items contained in the weight list by right-clicking the weight list and clicking *Create Equipment* as shown to the right.

- The weight list items are by this operation moved from the *dimension* folder into the *Equipment* folder where they are listed as *Bounding Box Equipment* objects.

- Before creating *Bounding Box Equipment* objects:

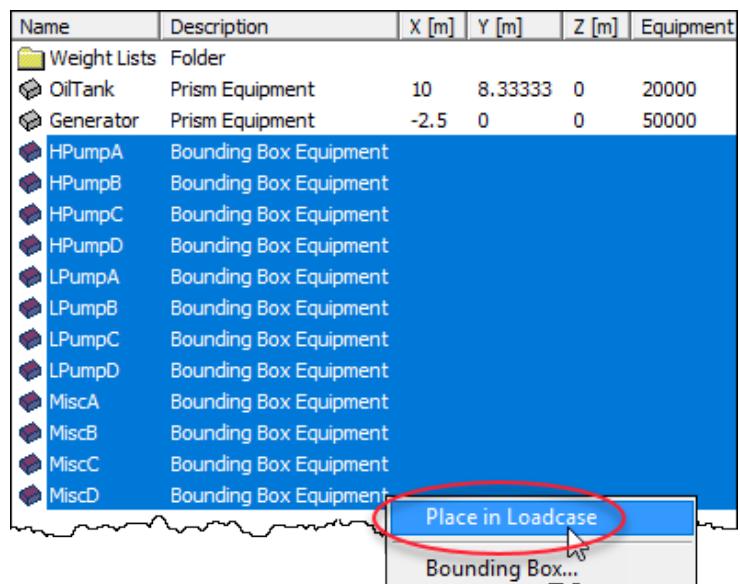


- After creating *Bounding Box Equipment* objects:

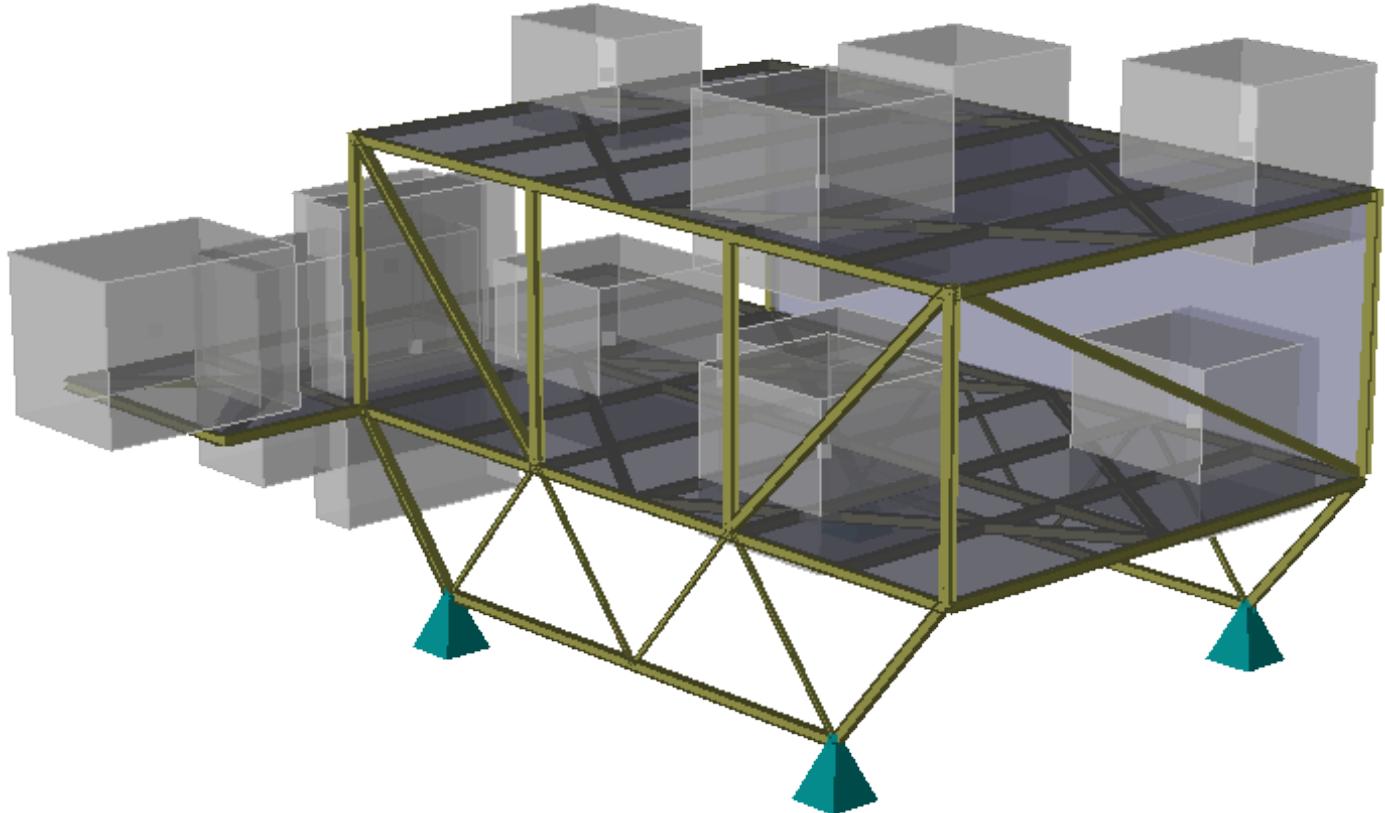


- Place the *Bounding Box Equipments* in the selected load case and in the model by right-clicking all or a selection, right-clicking and selecting *Place in Loadcase* as shown to the right.

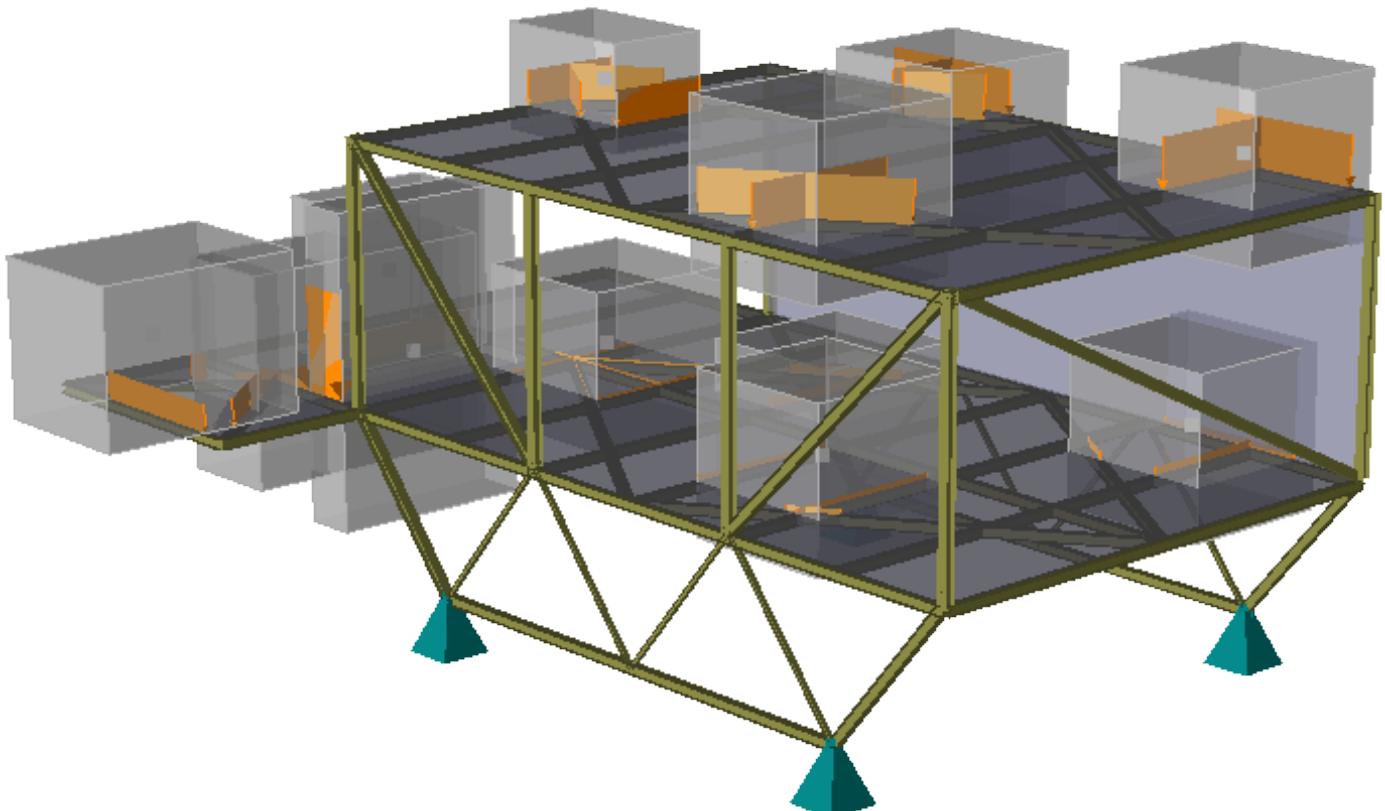
- As compared with prismatic equipments (Generator and OilTank) the weight list objects have positions as part of their definitions and therefore need not be positioned in the model.



- The weight list items should now appear in the model:

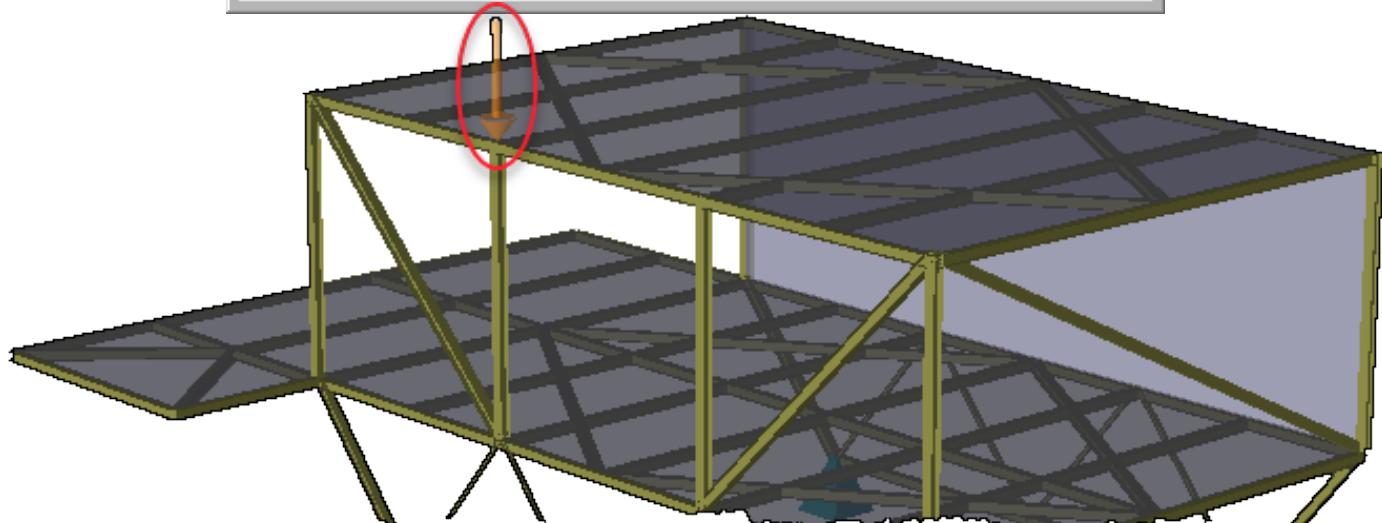
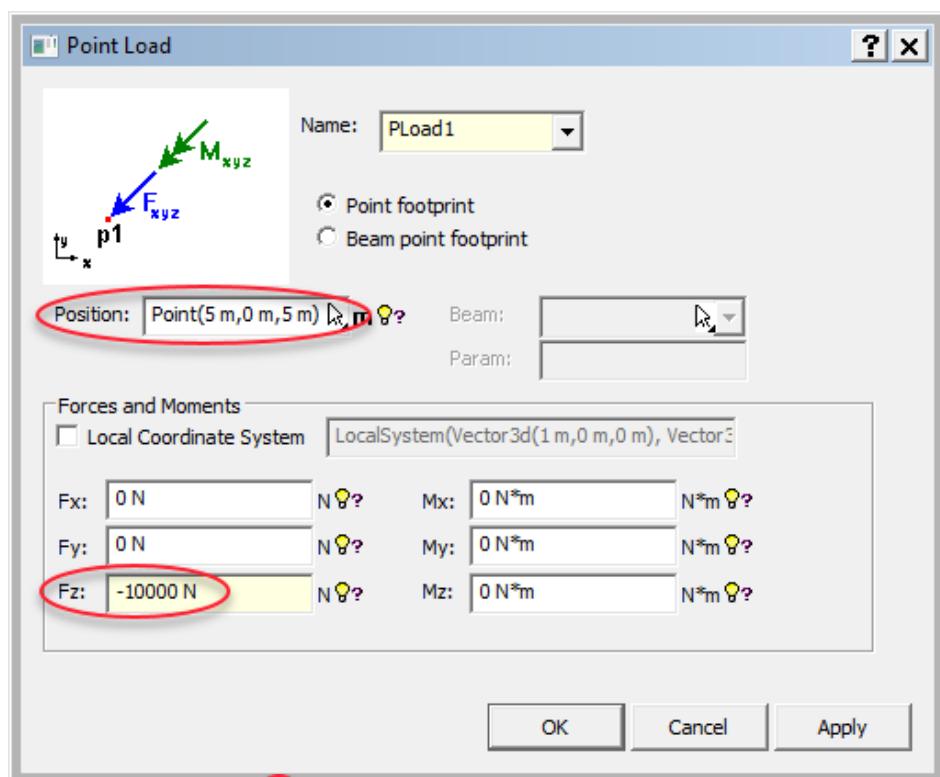


- Right-click the load case containing the weight list and select *Generate Applied Loads* to display the line loads:

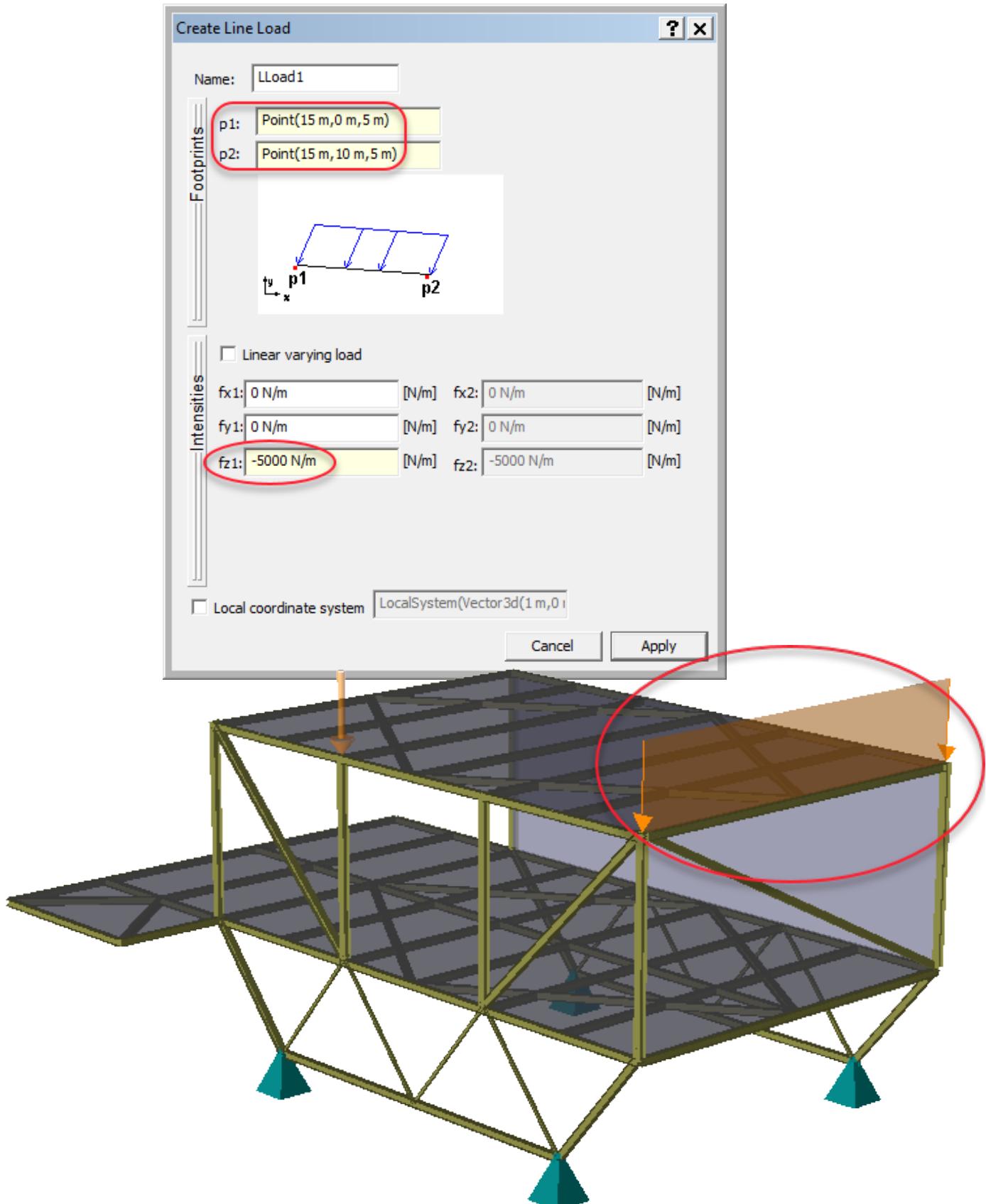


5.3 CREATE EXPLICIT LOADS

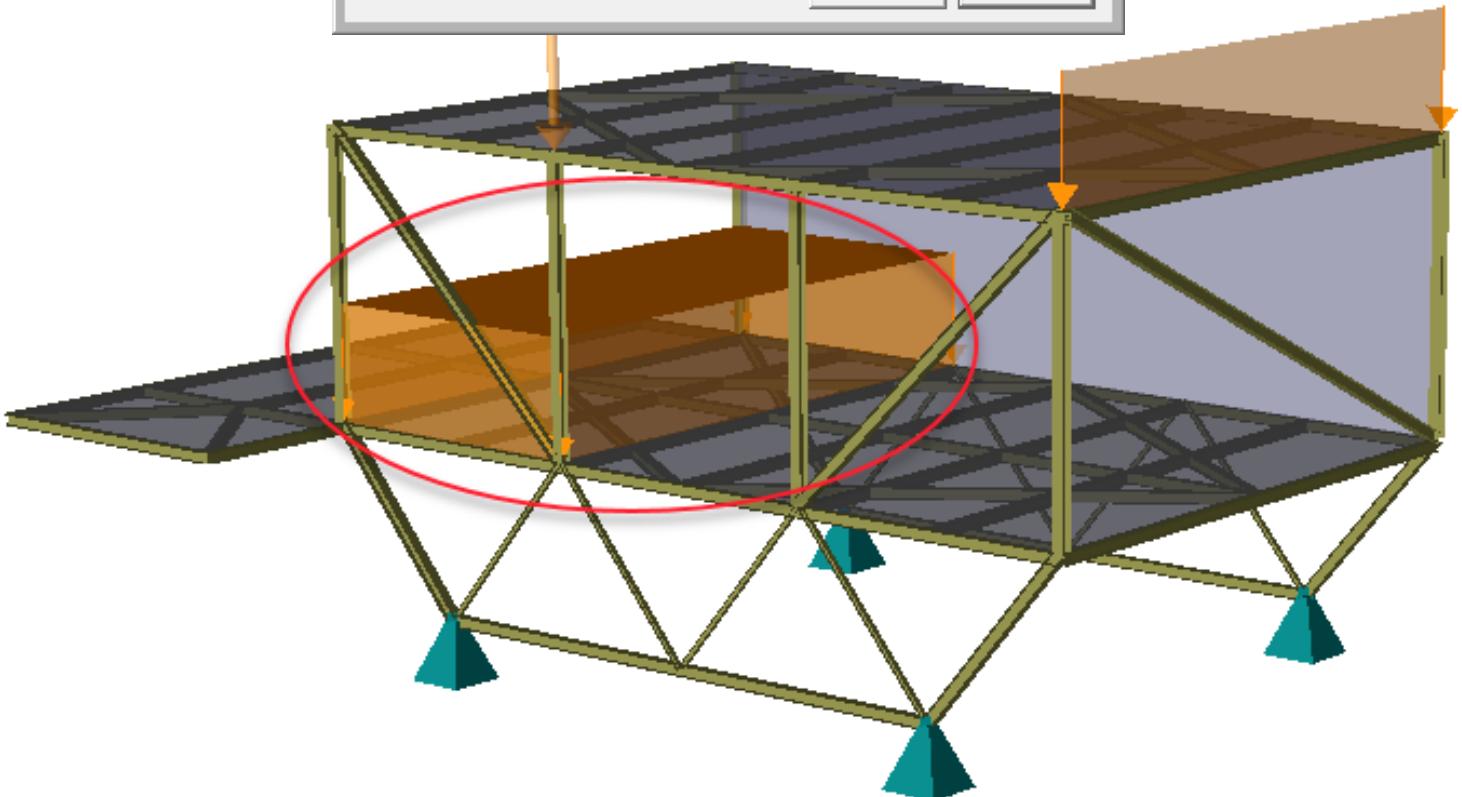
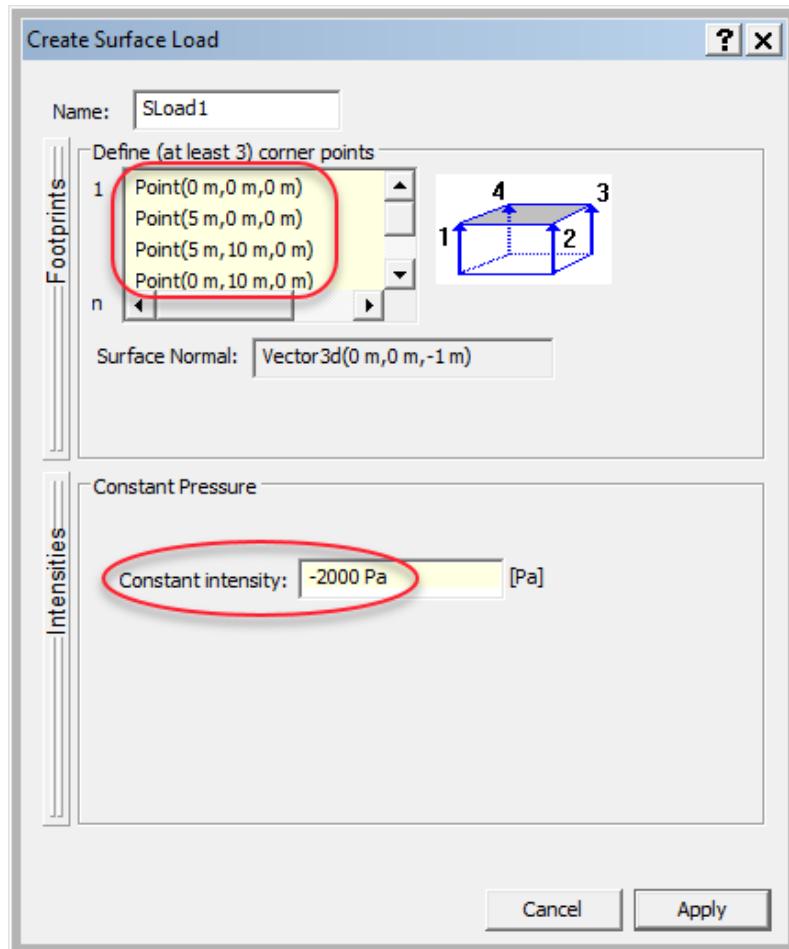
- There are three different types of explicit loads:
 - Point load
 - Line load
 - Surface load
- First set LC_expl as the currently selected load case so that the explicit loads are put into this load case.
- Use *Loads | Explicit Load | Point Load* to insert a vertical force as shown. Click in the model to position the load.



- Use *Loads | Explicit Load | Line Load* to insert a constant line load as shown. Click twice in the model to position the load.

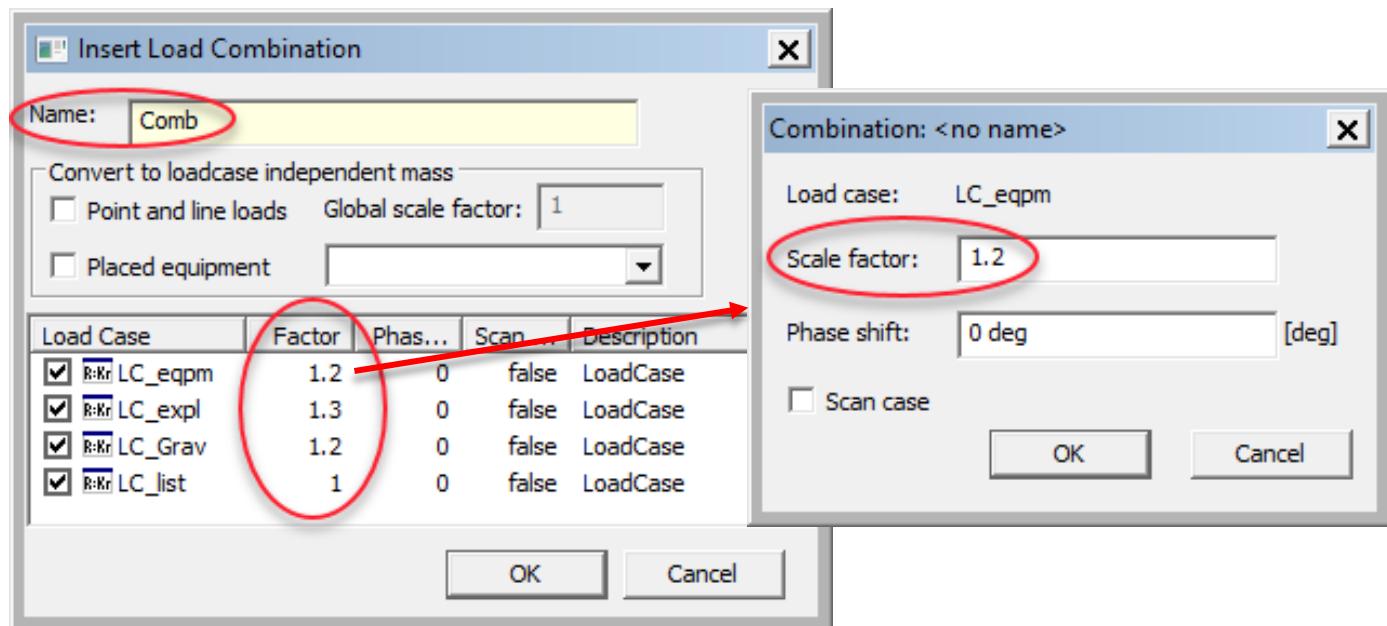


- Use Loads | Explicit Load | Surface Load to insert a constant surface pressure as shown. Click four times in the model to position the load.

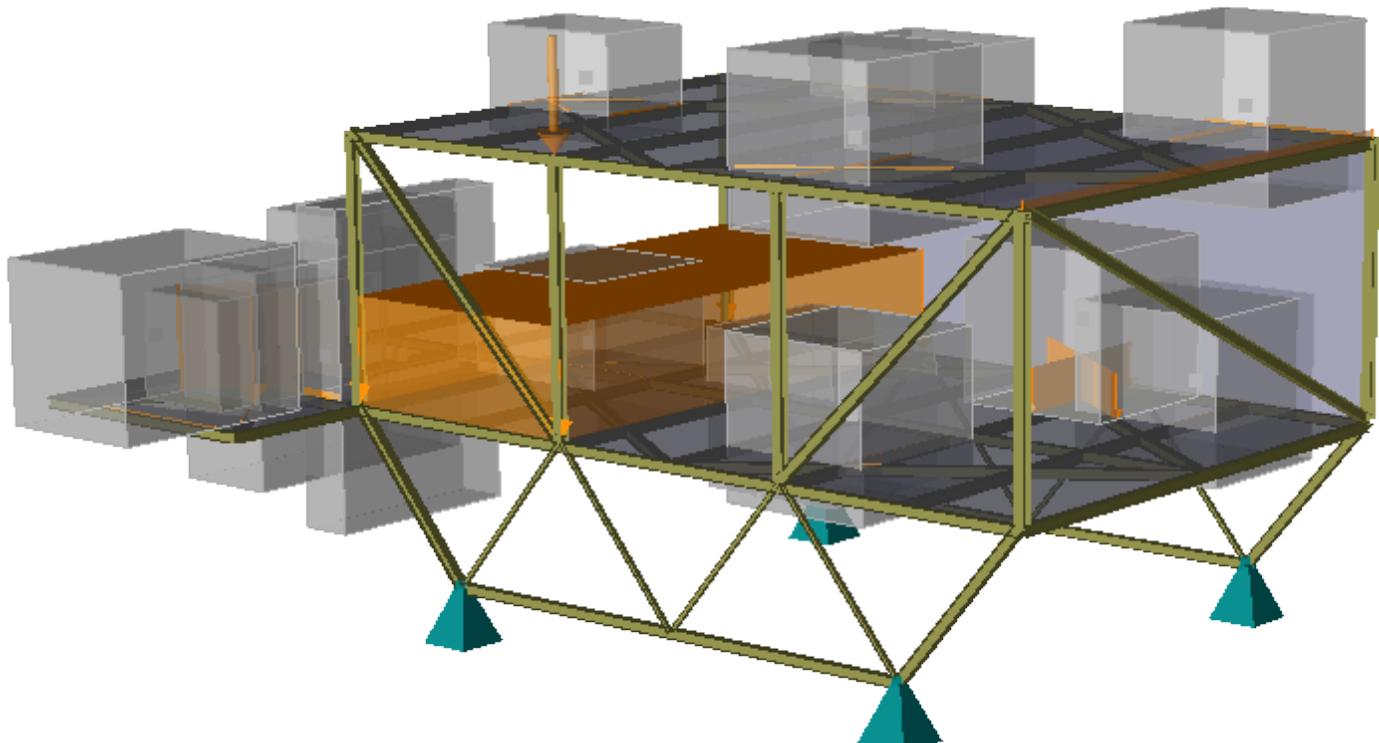


5.4 CREATE LOAD COMBINATIONS

- Use *Loads | Load Combination* to open the *Insert Load Combination* dialog. Or select and right-click the relevant load cases in the *Analysis | Load Cases* folder and select *New Load Combination* to open the same dialog.
- Create a combination named Comb being a combination of all four load cases.
- Adjust the *Factor* for the individual load cases by double-clicking the default value 1.

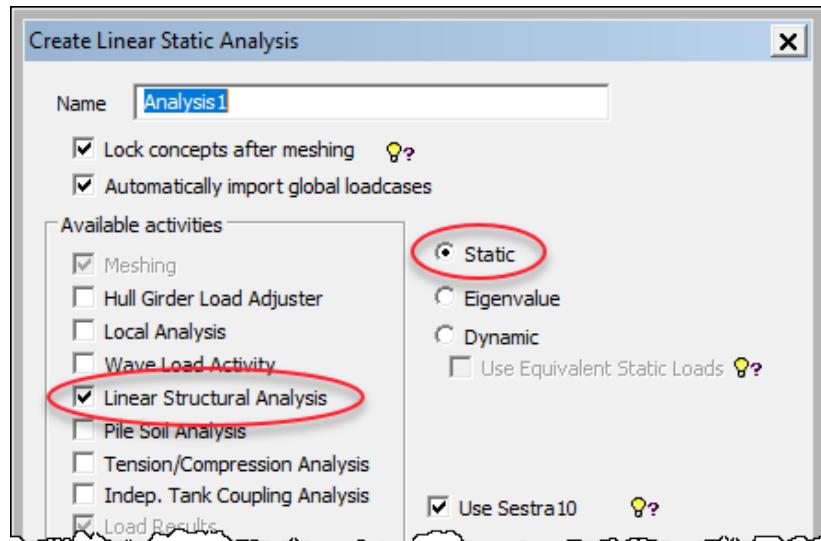


- The loads of the load combination are displayed:

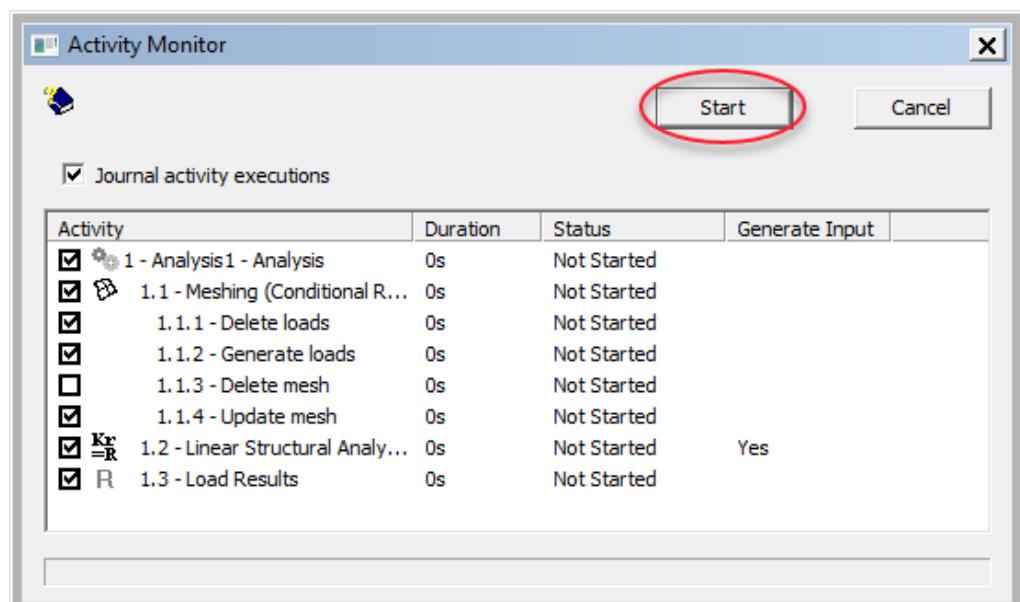


6 CREATE AND RUN AN ANALYSIS

- Use *Mesh & Analysis | Activity Monitor* (or Alt+D) to create an analysis activity. By default a *Static Linear Structural Analysis* is performed. Accept this and click *OK* to close the dialog.



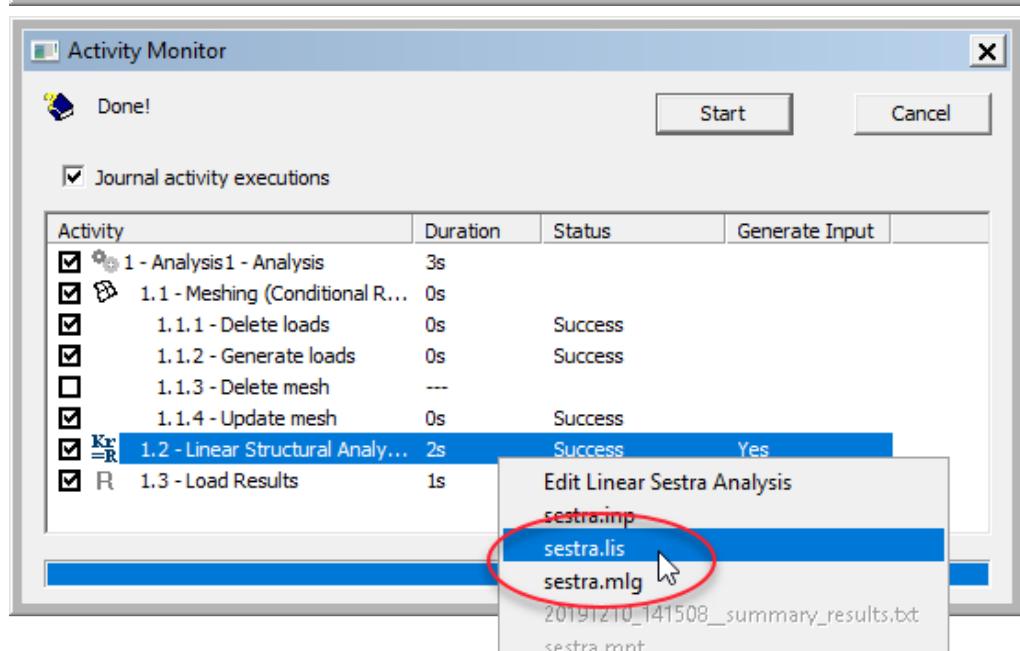
- This opens the *Activity Monitor* shown to the right. Click *Start* to run the analysis.



- When the analysis is done make sure the *Status* of all sub-activities is *Success*.

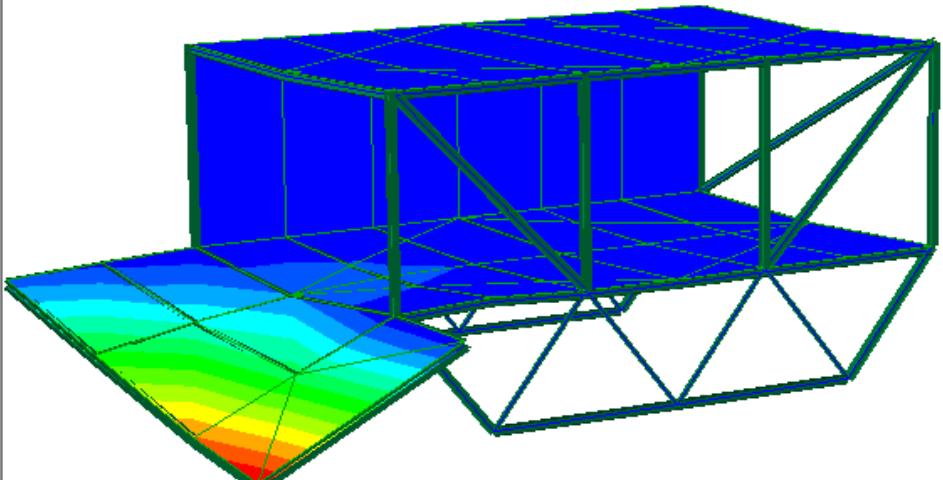
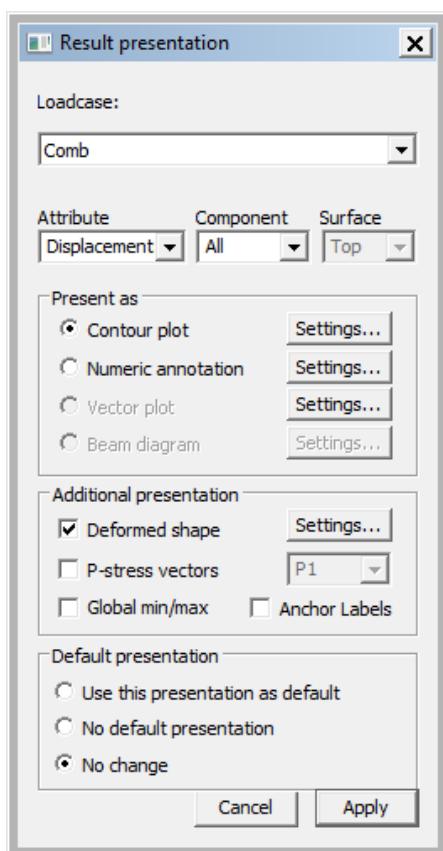
- Right-click the *Linear Structural Analysis* activity to open Sestra files:

- Sestra.lis: Sum of loads and sum of reactions.
- Sestra.mlg: Messages



7 PRESENT RESULTS

- Switch to *Results - with Mesh* display configuration use *Results | Presentation* (or Alt+P) to open the *Result Presentation* dialog. In the dialog select:
- Load case or load combination
 - *Attribute* (*Displacements, Beam Forces, G-stress, etc.*)
 - *Component* (dependant on *Attribute*)
 - How to present the selected results
 - An example is shown below.



- Try out different ways of presenting results.

About DNV

We are the independent expert in risk management and quality assurance. Driven by our purpose, to safeguard life, property and the environment, we empower our customers and their stakeholders with facts and reliable insights so that critical decisions can be made with confidence. As a trusted voice for many of the world's most successful organizations, we use our knowledge to advance safety and performance, set industry benchmarks, and inspire and invent solutions to tackle global transformations.

Digital Solutions

DNV is a world-leading provider of digital solutions and software applications with focus on the energy, maritime and healthcare markets. Our solutions are used worldwide to manage risk and performance for wind turbines, electric grids, pipelines, processing plants, offshore structures, ships, and more. Supported by our domain knowledge and Veracity assurance platform, we enable companies to digitize and manage business critical activities in a sustainable, cost-efficient, safe and secure way.