SESAM TUTORIAL

GeniE

Modelling and Code Checking a Module Frame

Valid from program version 8.2

Sesam Tutorial

GeniE – Modelling and Code Checking a Module Frame

Date: June 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

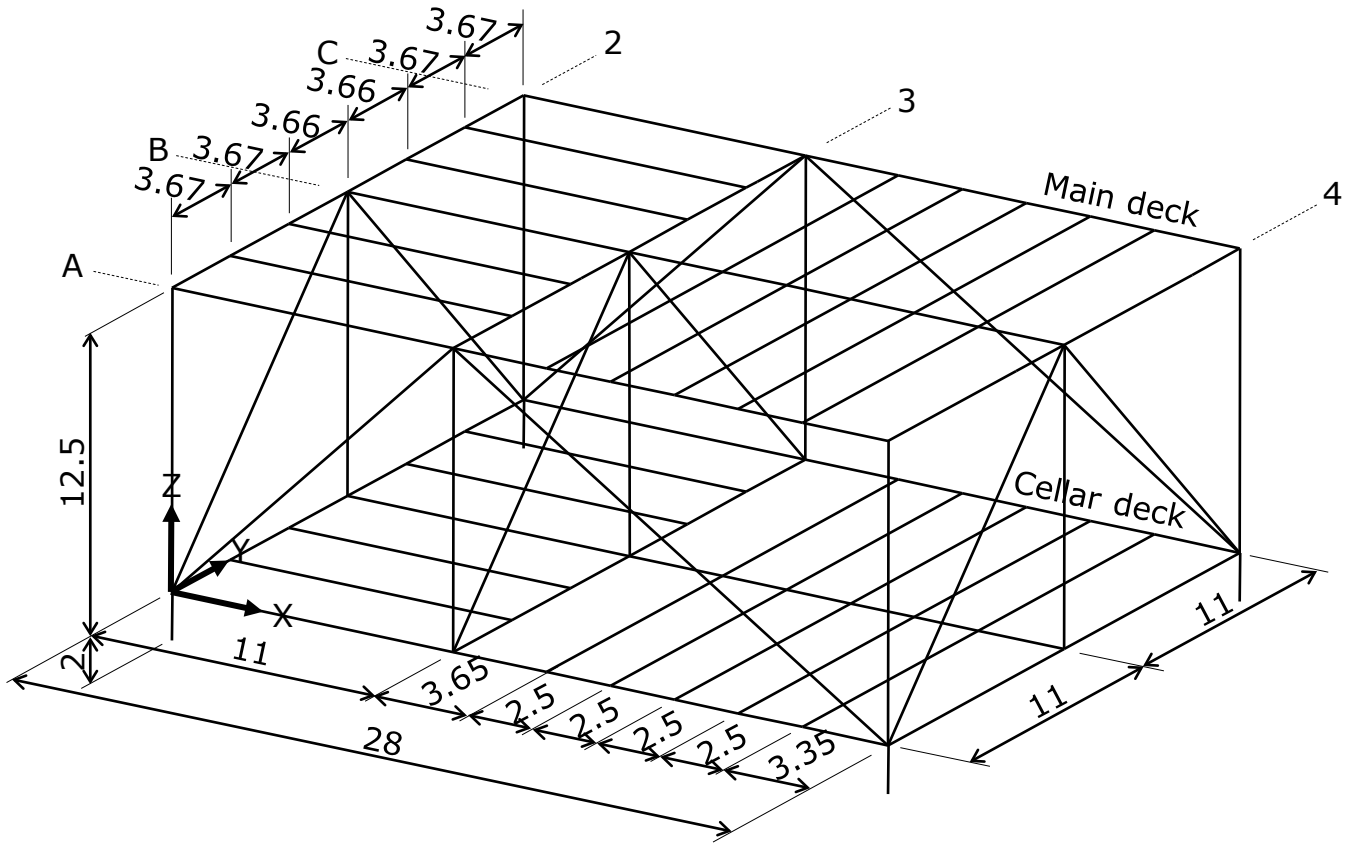
E-mail support: software.support@dnv.com

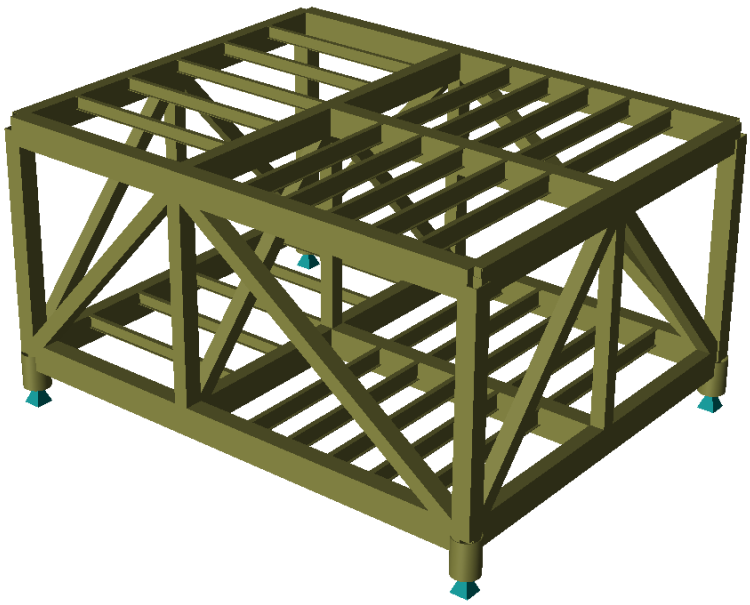
E-mail sales: digital@dnv.com

TABLE OF CONTENTS

1 INTRODUCTION

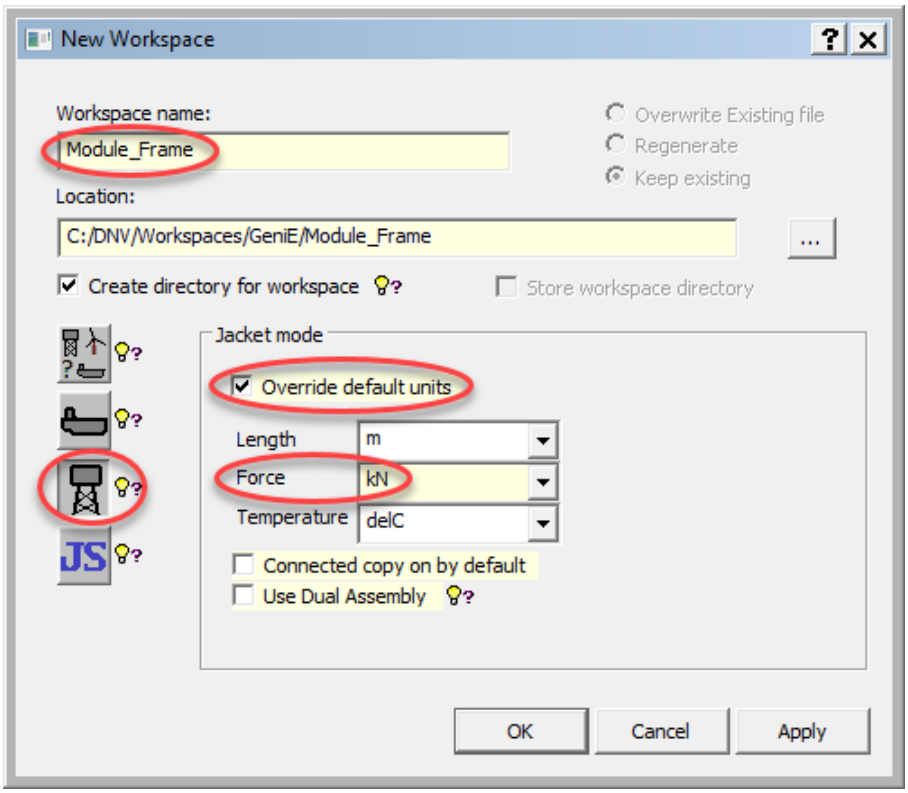
- In this tutorial the module frame below is modelled, a static analysis is run and code checking is performed.
- The main dimensions are given below.
- A GeniE input file for creating the complete model is provided.
- The appearance of the GUI and dialogs in later versions of GeniE may change.

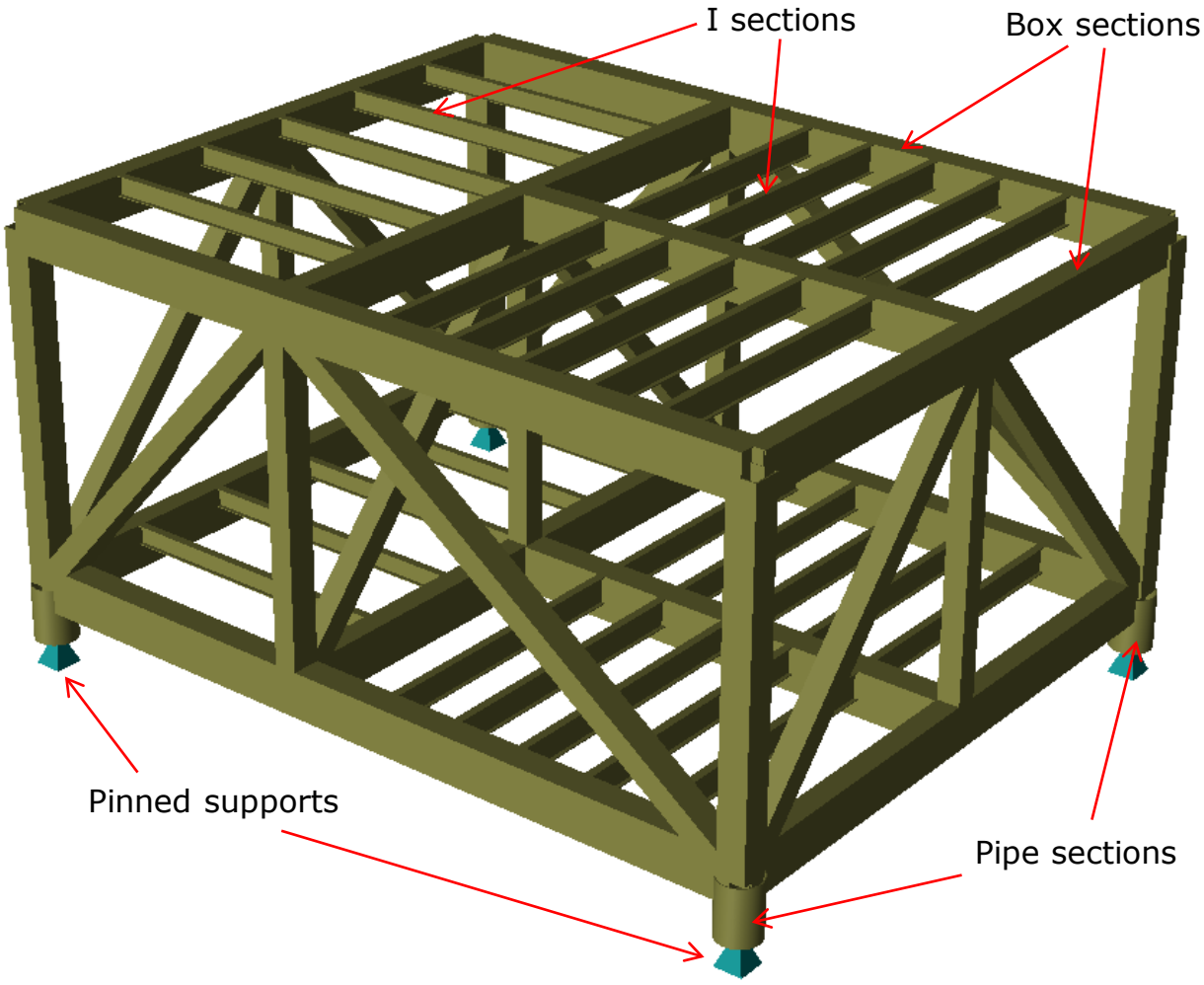




2 OPEN WORKSPACE

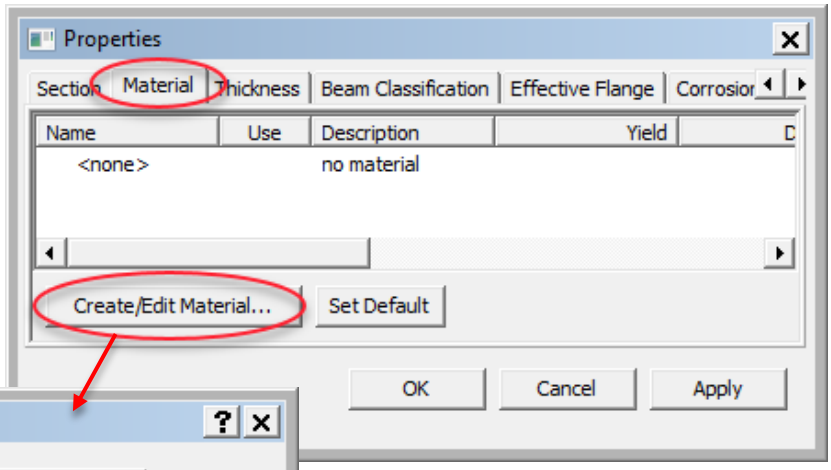
- Start GeniE and open a new workspace.
  - Give a *Workspace name*.
  - Check *Override default units* and change *Force* unit to kN.
  - Press the *Jacket mode* button to limit menus to those relevant for jacket (spaceframe) modelling.
  - Click *OK*.

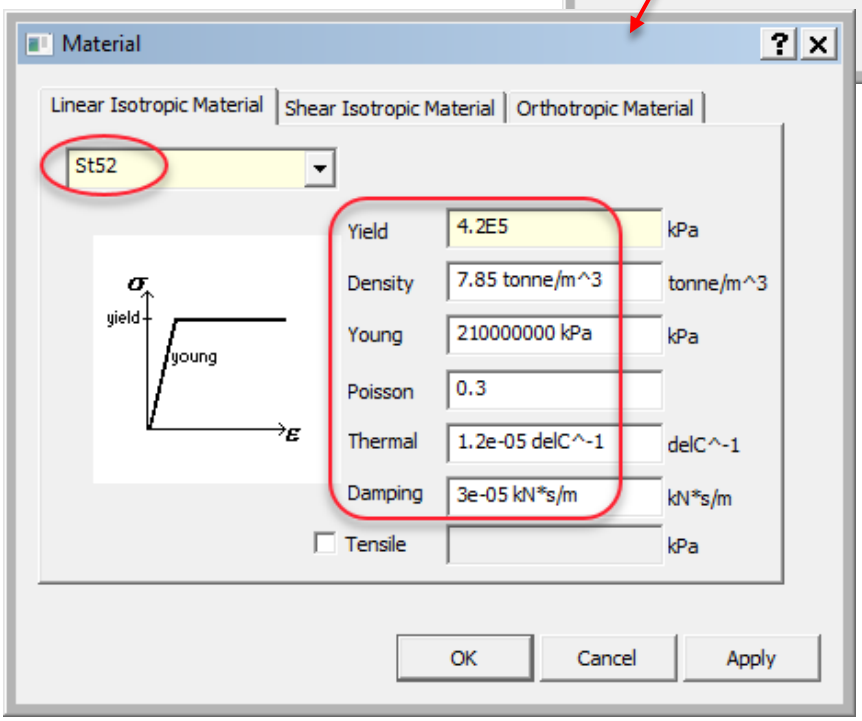




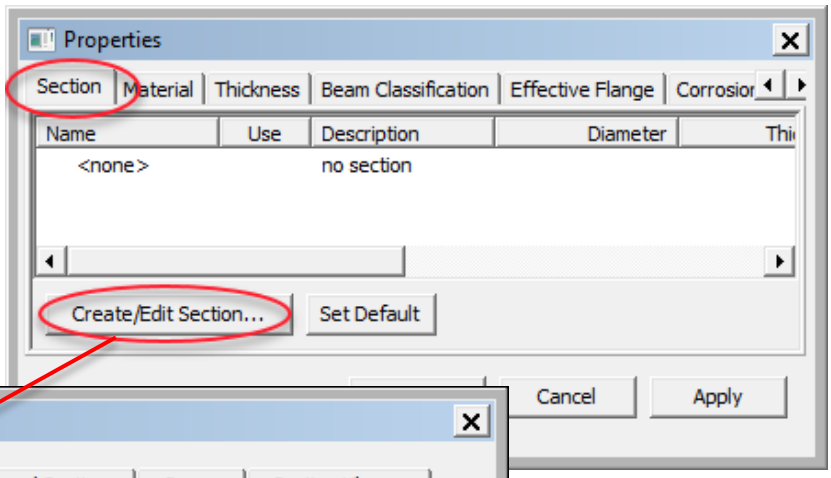
3 MATERIAL AND BEAM CROSS SECTIONS

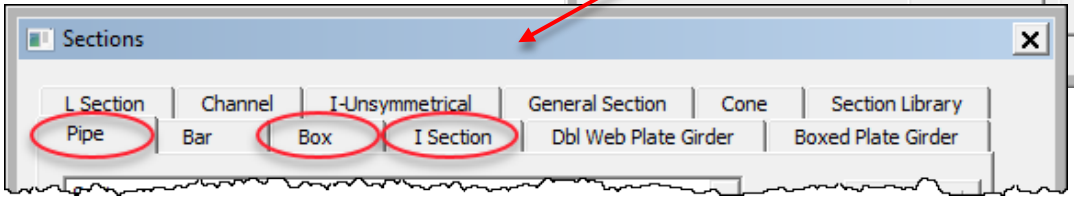
- Use *Edit* | *Properties* to open the *Properties* dialog. Go to the *Material* tab, click *Create/Edit Material* and give data as shown below.





- Create beam cross section by *Edit* | *Properties* and go to the *Section* tab in the *Properties* dialog.
  - Use the *Pipe*, *Box* and *I Section* tabs to manually create the sections to be used in this tutorial.





- The three tables below show the beam cross sections to define.
  - *Box* sections:

Name	Height	Width	Web thickness	Flange thickness
Box1	1.6	1.0	0.025	0.040
Box2	1.6	0.8	0.025	0.040
Box3	1.0	1.0	0.050	0.050
Box4	1.0	0.8	0.035	0.035
Box5	1.0	1.0	0.035	0.035
Box6	0.8	0.8	0.025	0.025
Box7	1.5	1.0	0.025	0.040
Box8	1.5	0.8	0.025	0.040

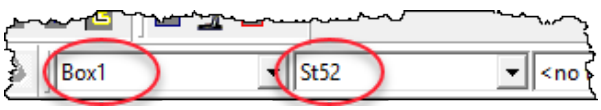
  - *I Sections* (I or H):

Name	Height	Width	Web thickness	Flange thickness
IH1	1	0.3	0.019	0.036
IH2	0.7	0.3	0.017	0.032

  - *Pipe* section:

Name	Diameter	Thickness
Pip1	1.5	0.04

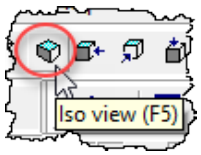
- When defining the cross sections accept default values for *Shear Factors*, *Fabrication method* and *Fillet*.
- Set Box1 as default beam cross section and St52 as default material. I.e. these will then apply to beams being created.



- When creating beams set the proper section as default prior to their creation. Alternatively, assign the proper section afterwards.

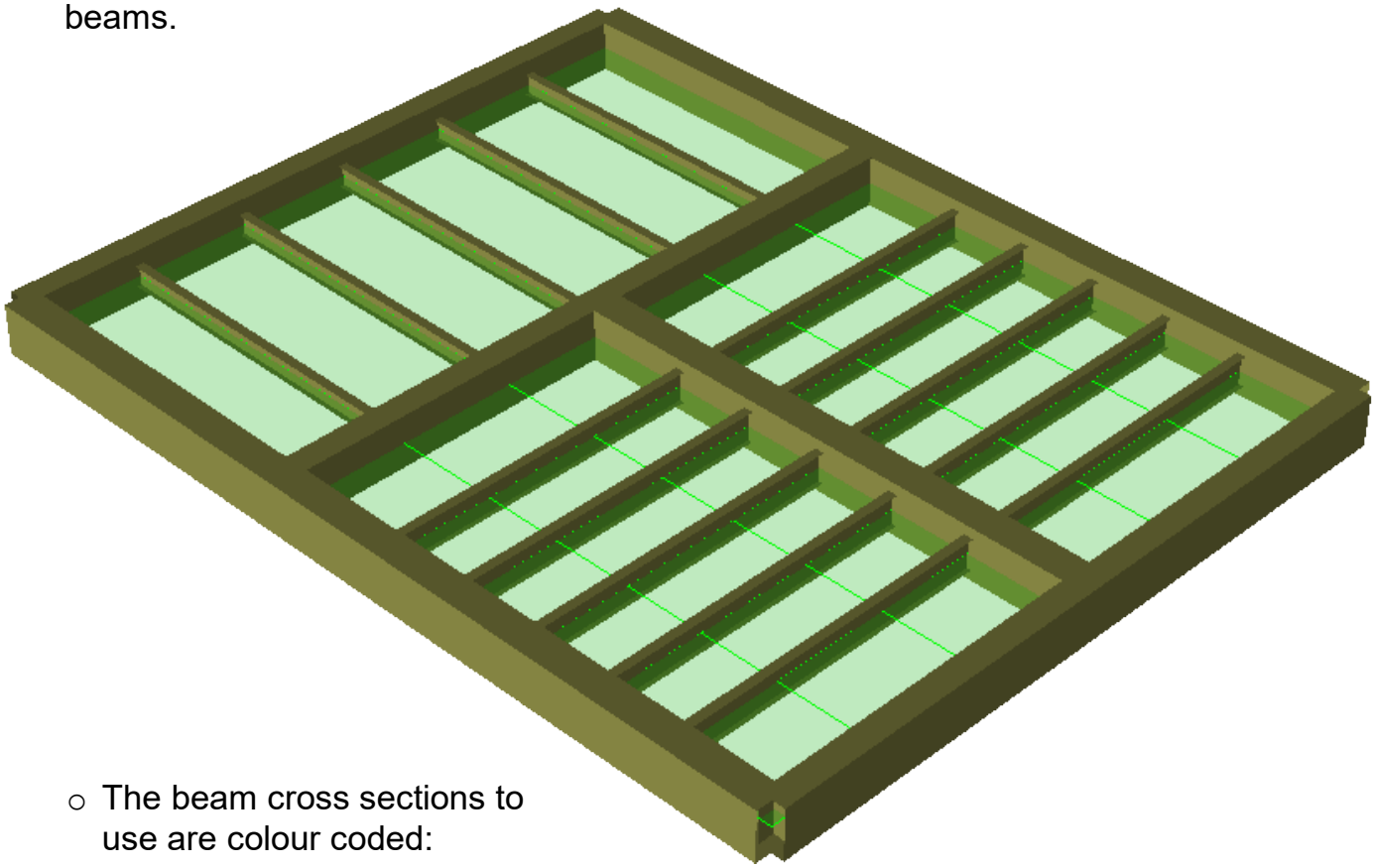
4 GUIDE PLANE

- Use *Guiding Geometry* | *Planes* | *Guide Plane Dialog* to open the dialog shown to the right. Fill in data as shown.
  - An easy way of filling in points *P1 – P4* is to fill in *P4* as shown and then use Tab key to jump to the other three points.
  - Having filled in the *u* and *v spacings* make sure the sketch of the guide plane in the dialog is correct before clicking *OK*.
- Press F5 to see the guide plane as shown below.

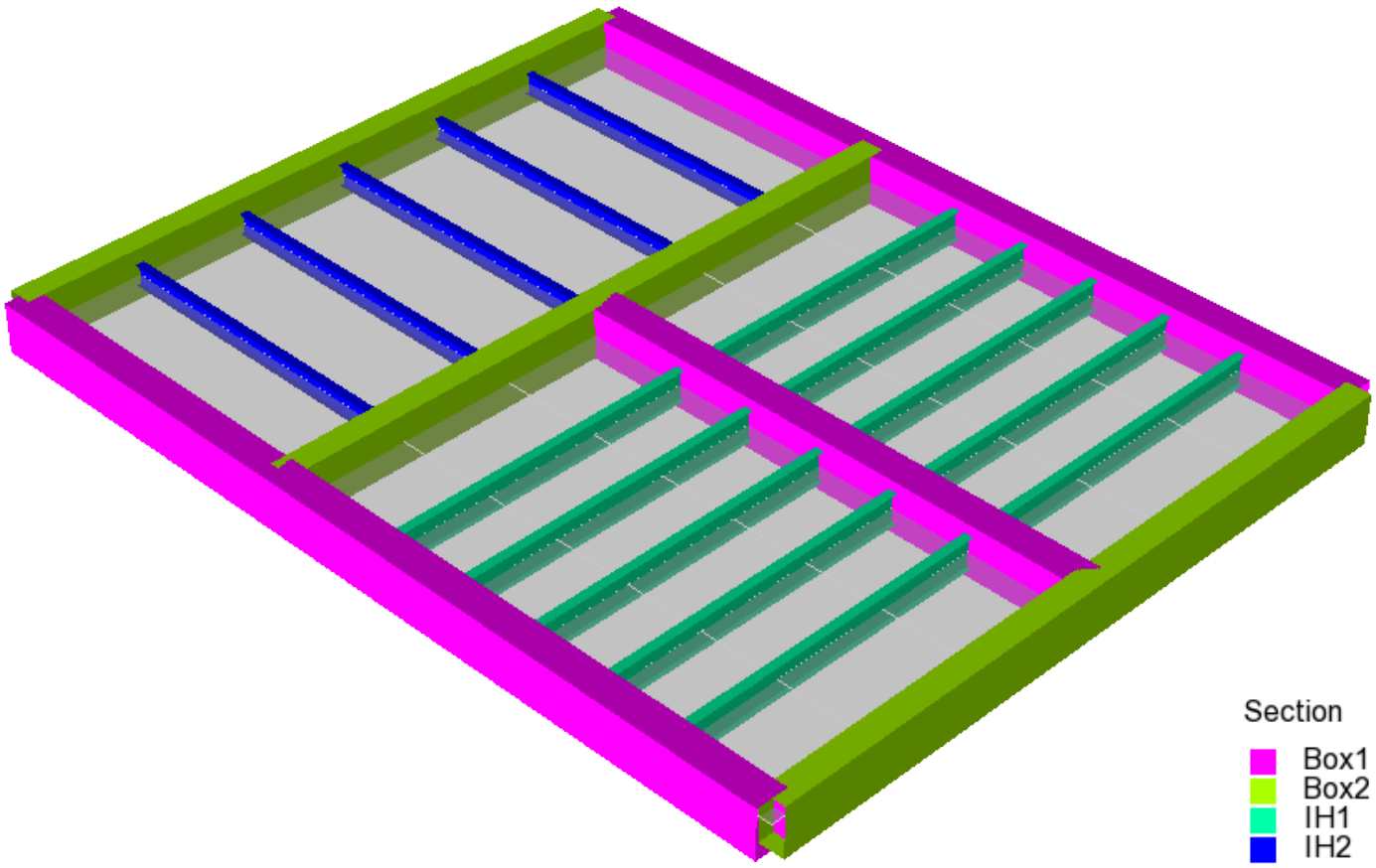


5 CELLAR DECK

➢ Use *Structure* | *Beams and Piles* | *Straight Beam* (or) to create the cellar deck beams.

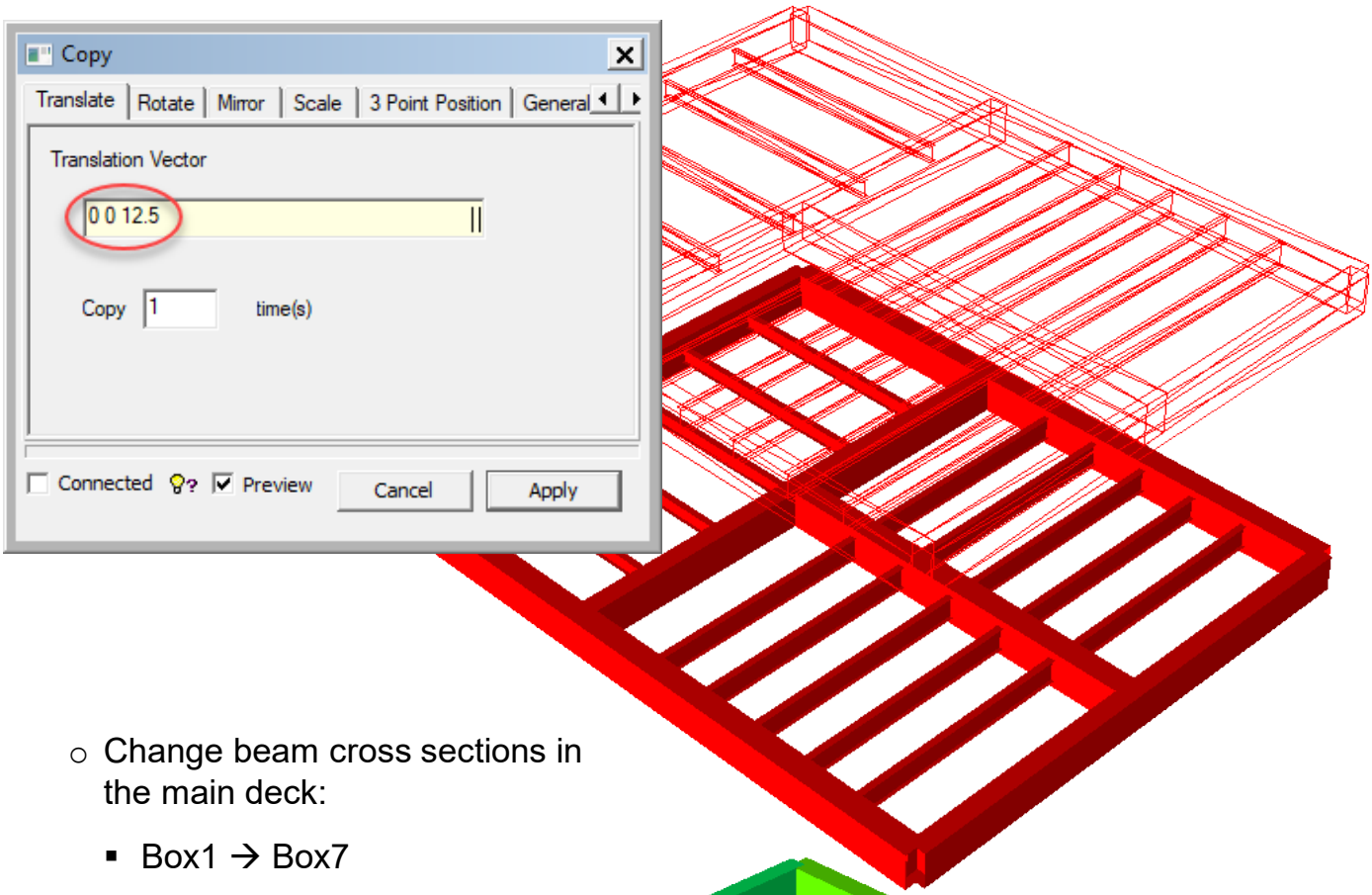


- The beam cross sections to use are colour coded:

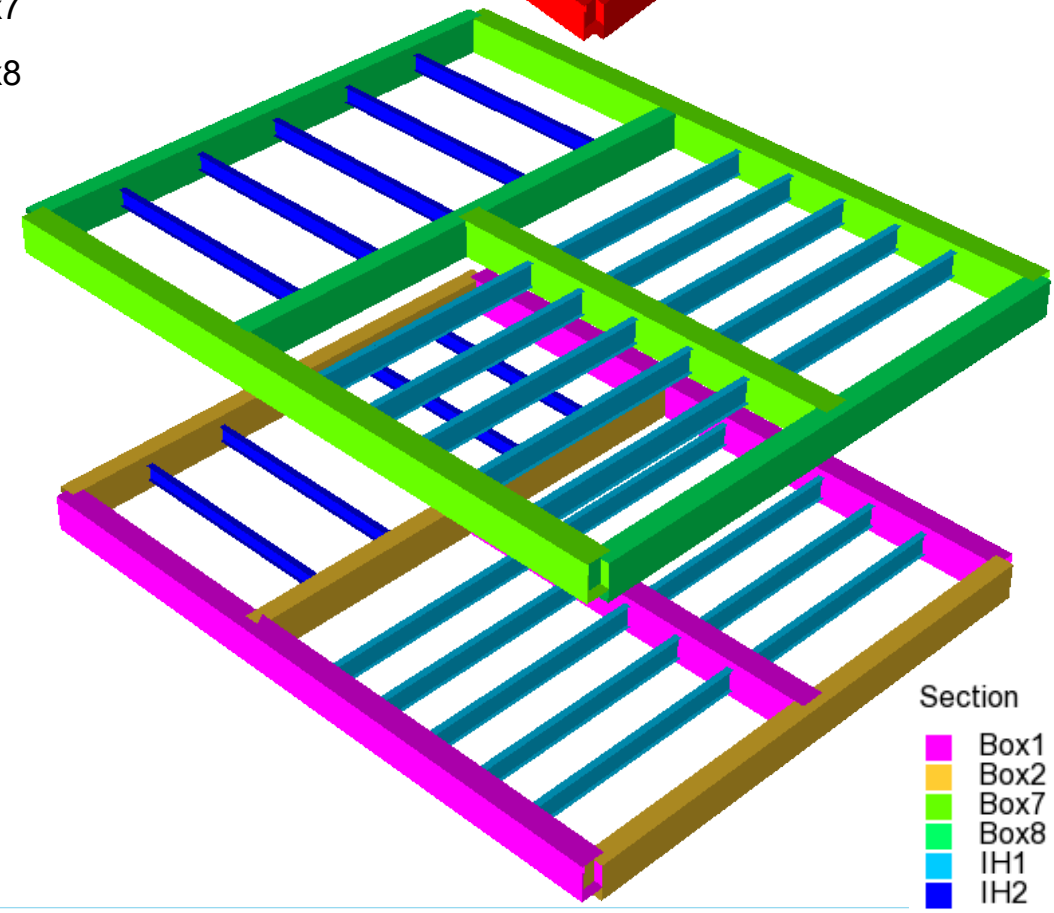


6 MAIN DECK

➢ Create the main deck by copying the cellar deck 12.5 m upwards:

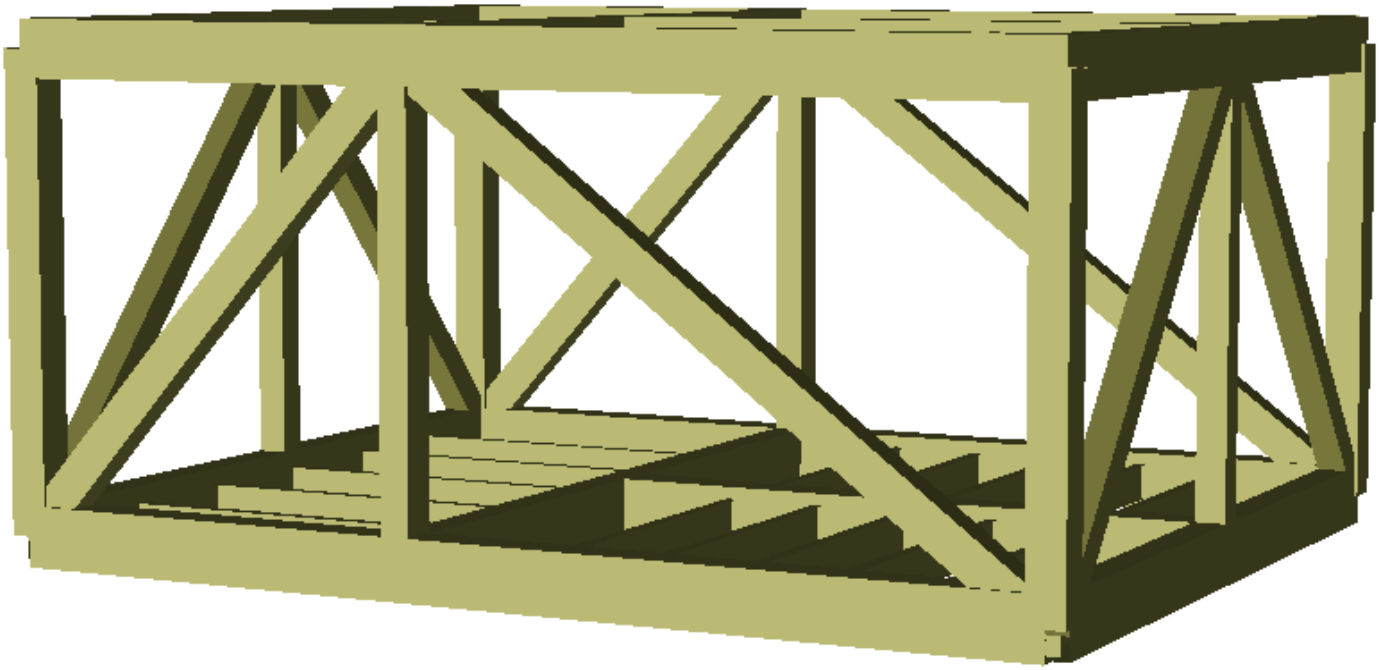


- Change beam cross sections in the main deck:
  - Box1 → Box7
  - Box2 → Box8



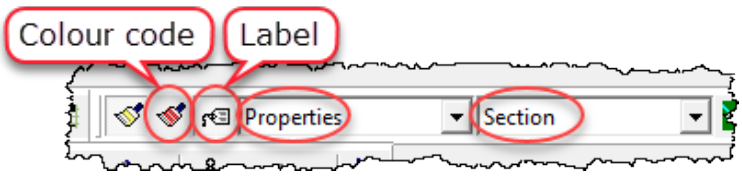
7 COLUMNS AND BRACINGS

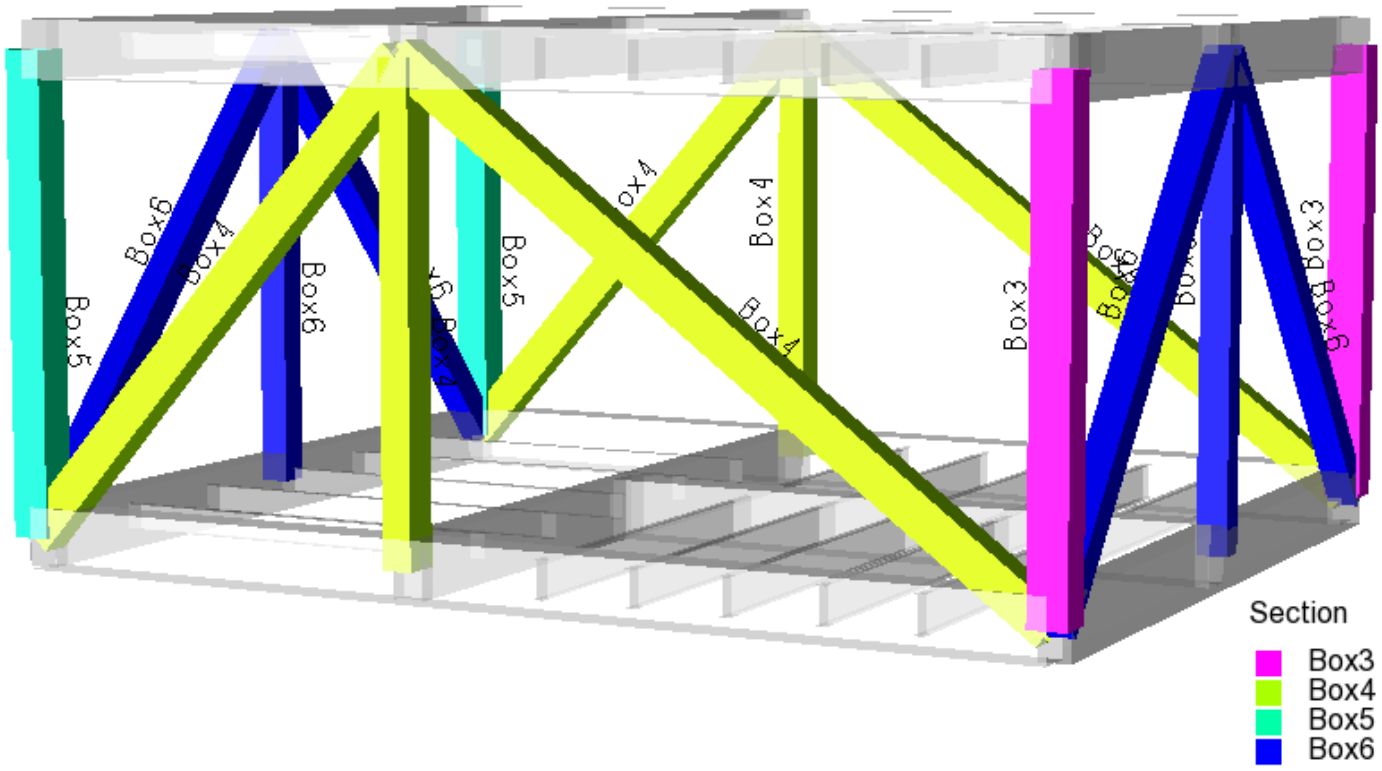
➢ Create columns and bracings between the cellar and main deck as shown.



➢ The cross sections to use are shown below both in colours and with labels.

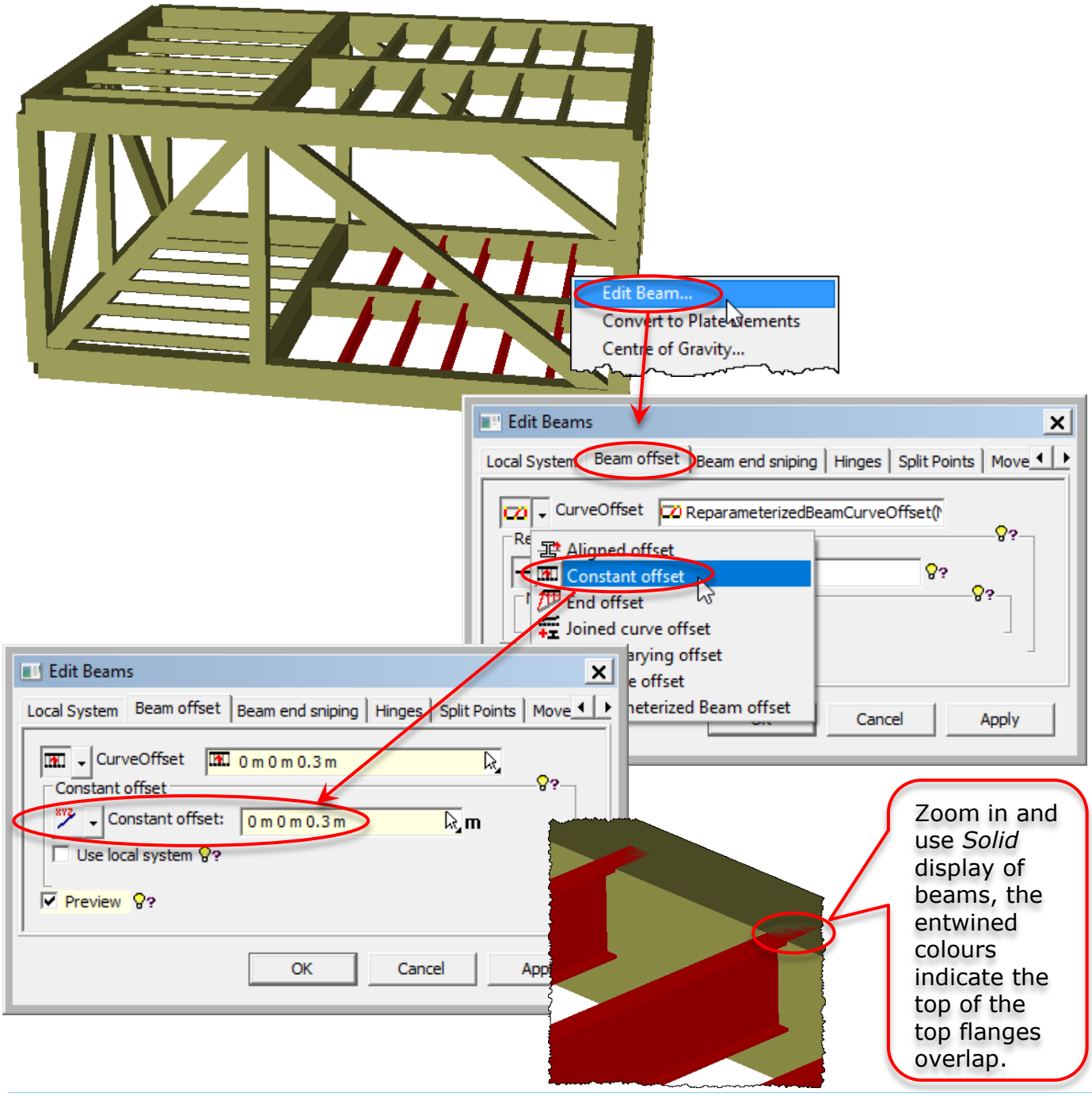
- Colour code and label a selection by selecting objects and then either using *the* buttons shown to the right, or right-clicking to access the *Labels* and *ColorCode* menus.





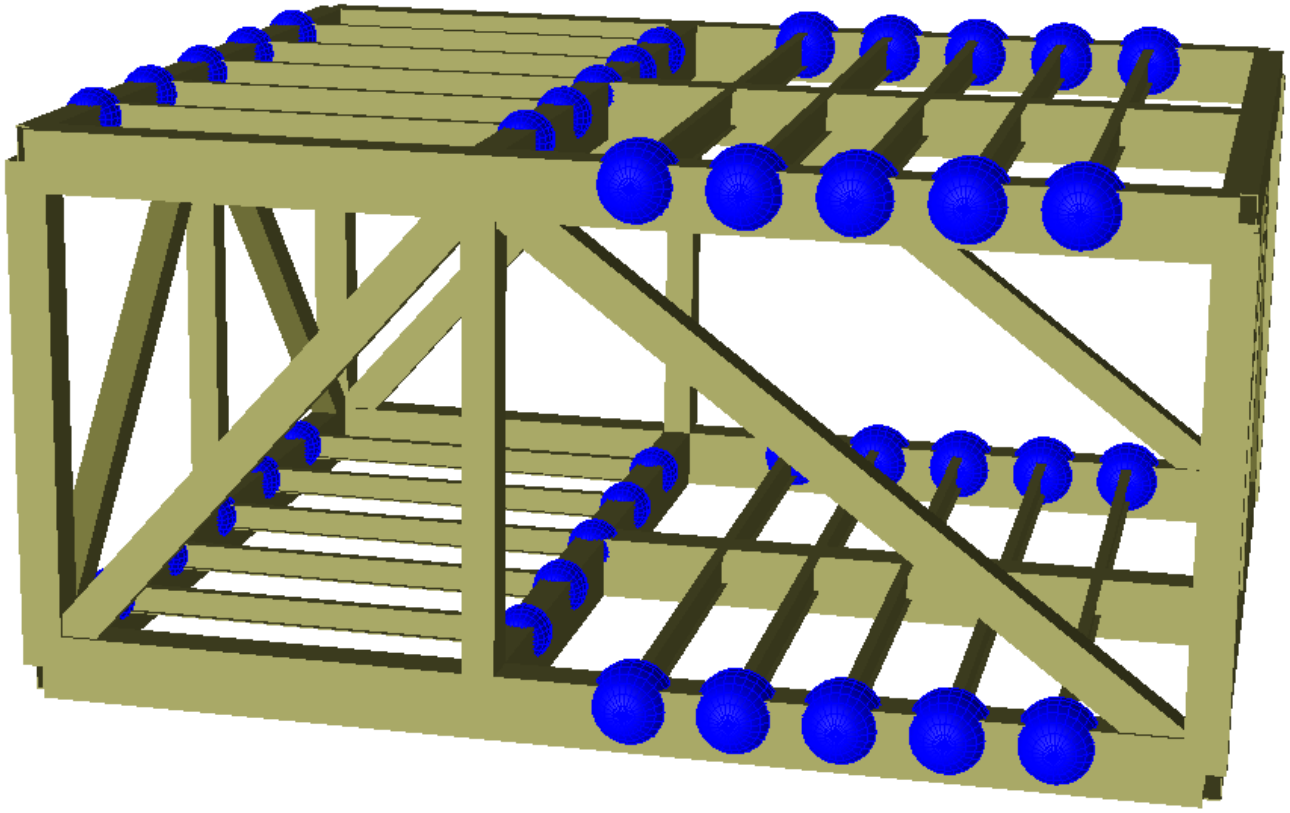
8 OFFSETS (ECCENTRICITIES) FOR HORIZONTAL BEAMS

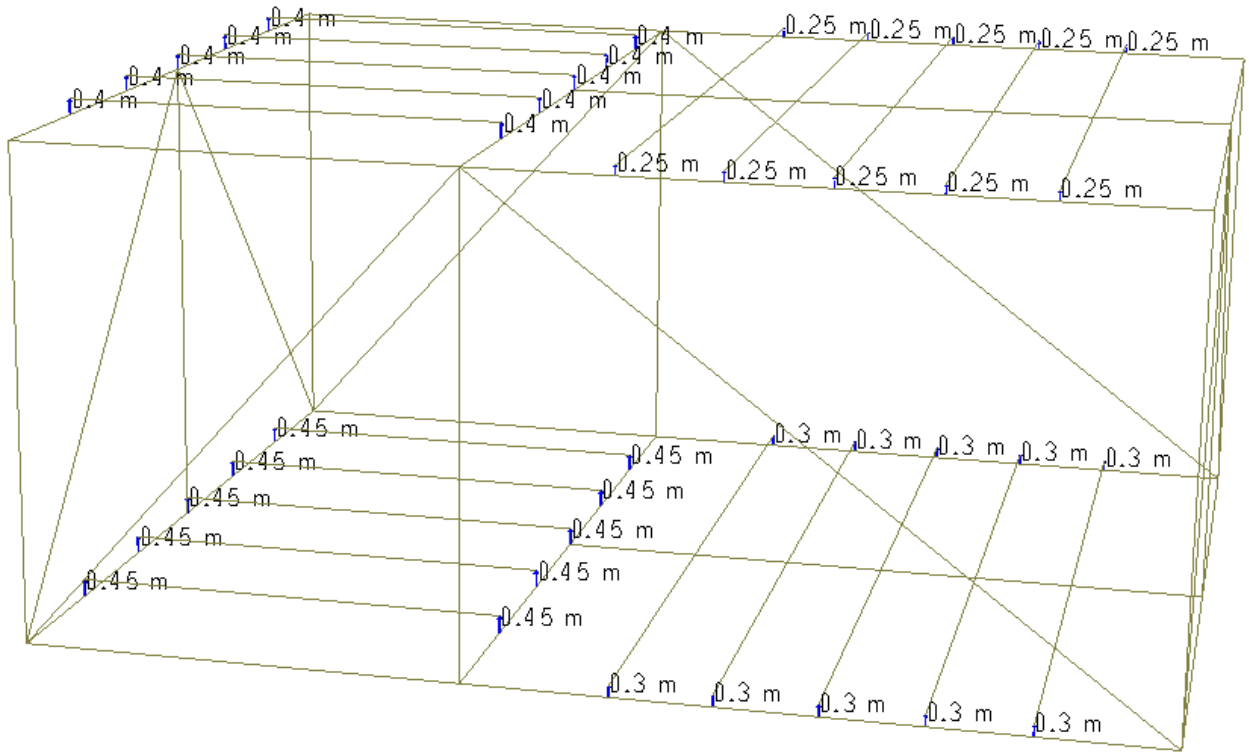
- The top of all beams in the cellar deck and likewise in the main deck shall be flush (at equal elevation). Beams must be given offsets (eccentricities) to achieve this.
  - The two box sections in the cellar deck have the same height, 1.6 m. So only the I sections (IH1 and IH2) need to be given offsets equal to half the difference in height:
    - IH1: (1.6-1)/2 = 0.3
    - IH2: (1.6-0.7)/2 = 0.45
  - Creating offsets is shown below for the IH1 beams in the cellar deck:



➢ Create appropriate offsets for all beams with I section in the cellar and main decks.

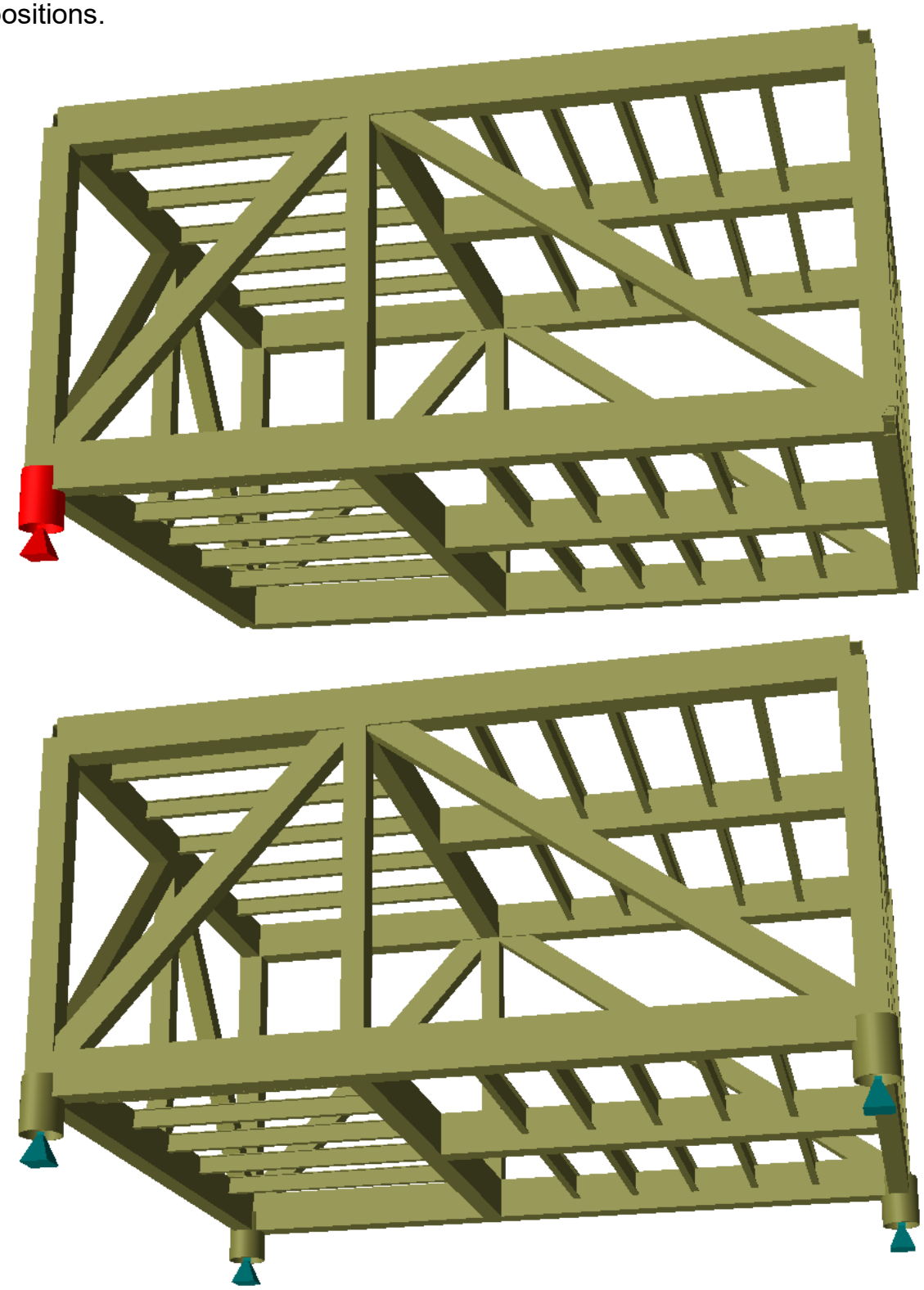
- The below figures show all beams labelled with *Eccentricity Symbols* (top) and *Eccentricities* (bottom).





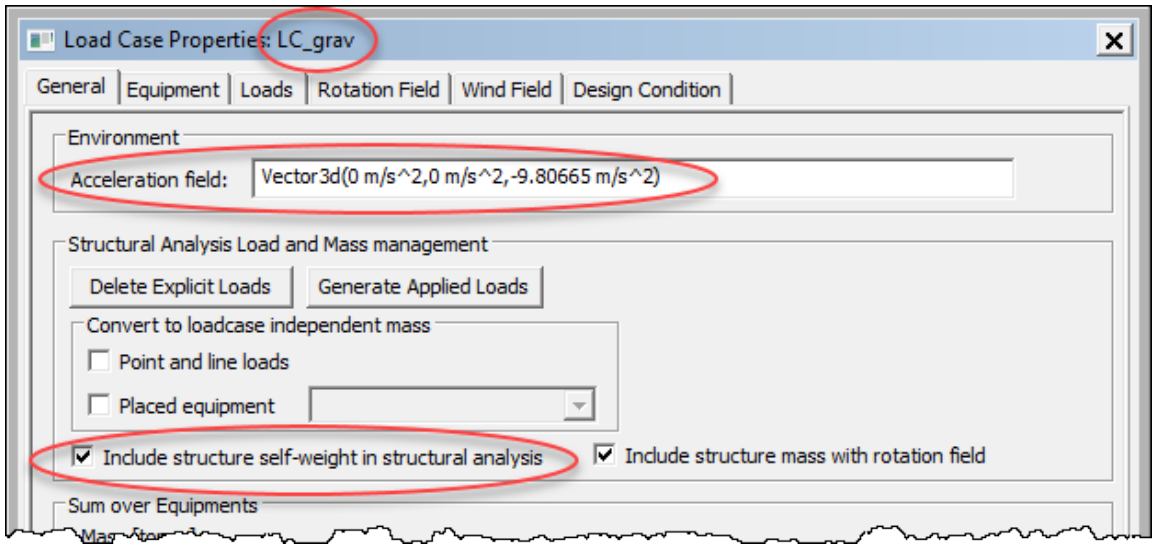
9 SUPPORTS

- The module frame is supported by four 2 m high pipe stools (beam cross section Pip1) for which the lower ends are fixed for the three translations and free for the three rotations (i.e. pin support).
  - Use the *Create Straight Beam* dialog to create one of the stools, add boundary conditions and copy the stool plus boundary condition to the three other positions.

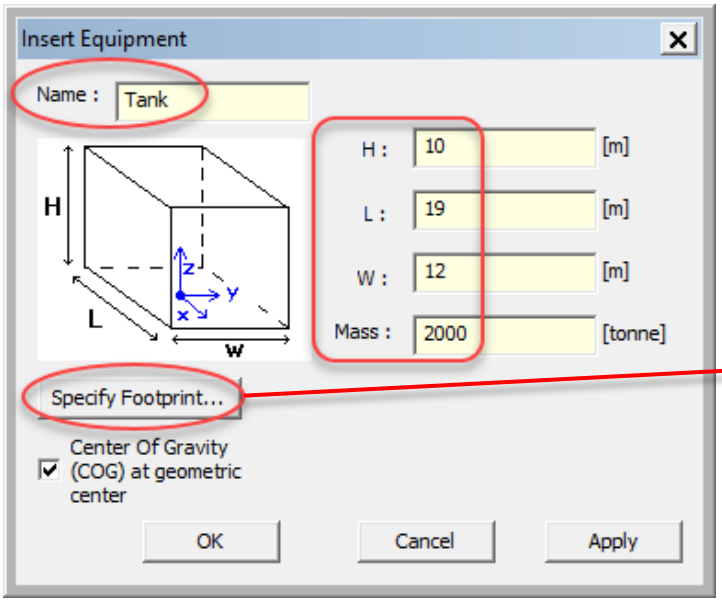


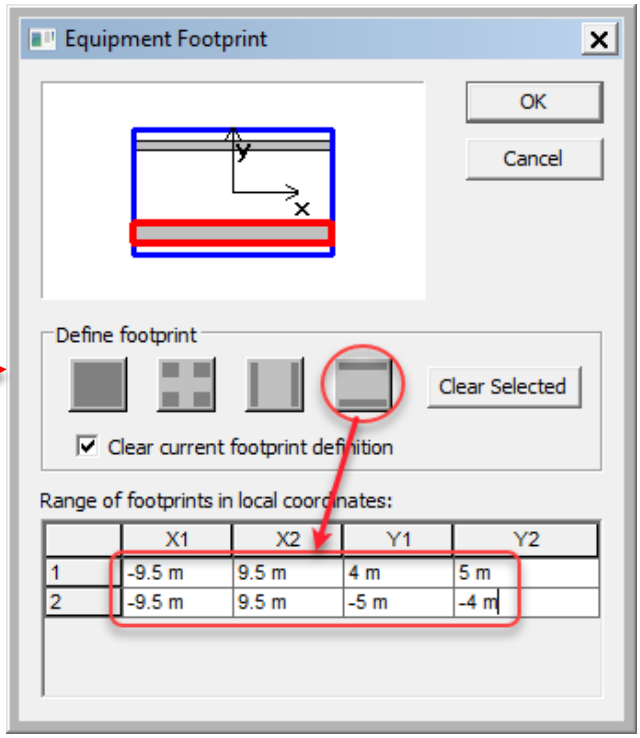
10 LOADS

- LC_grav is a gravity (self weight) load case. Create it, select it in the *Load Cases folder* and open the *Load Case Properties* dialog:



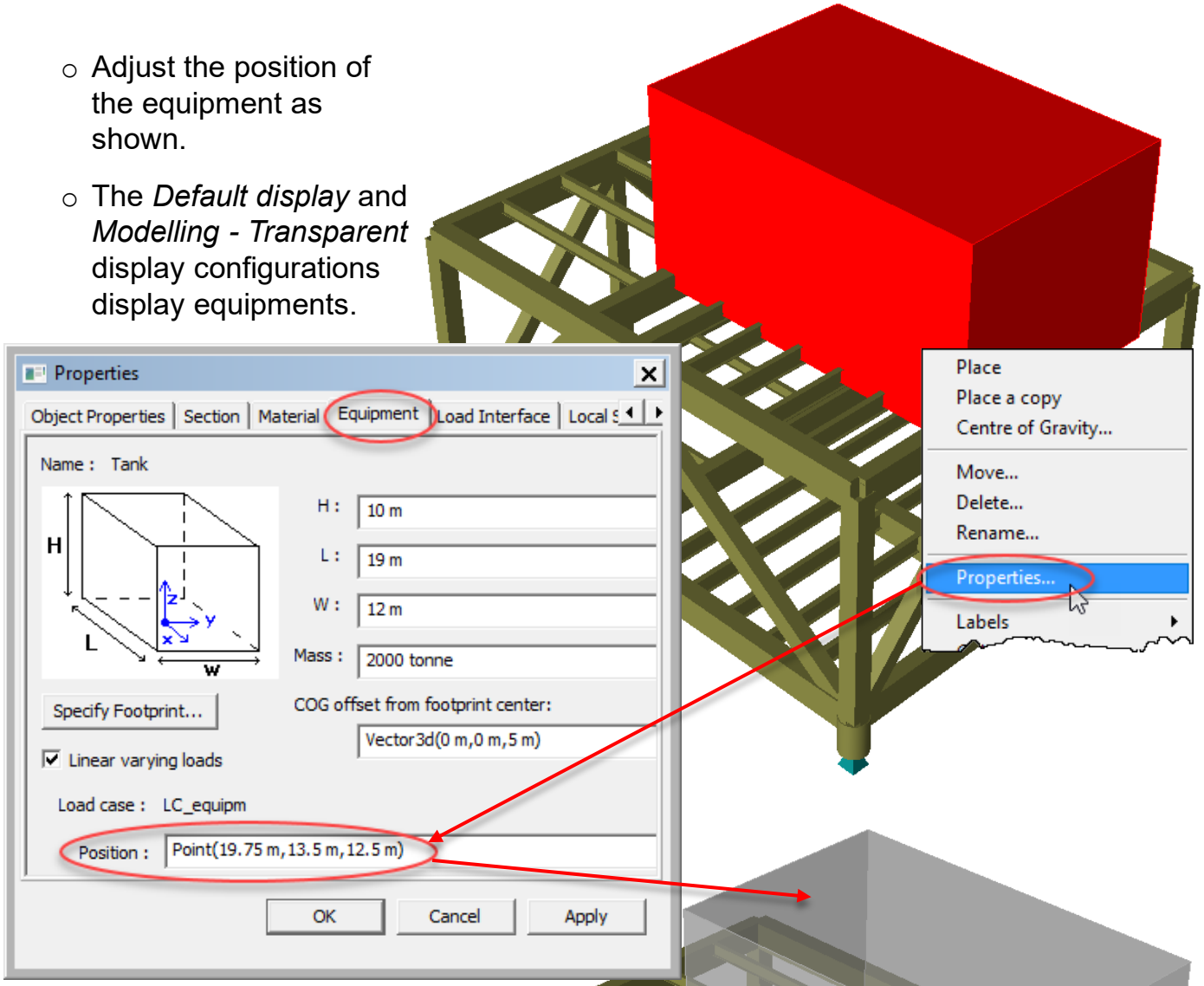
- LC_equipm is a load case containing an equipment.
  - Create the load case which then becomes the currently selected one.
  - Create the equipment by *Loads* | *Prismatic Equipment* with height, length, width, mass and footprint as shown.



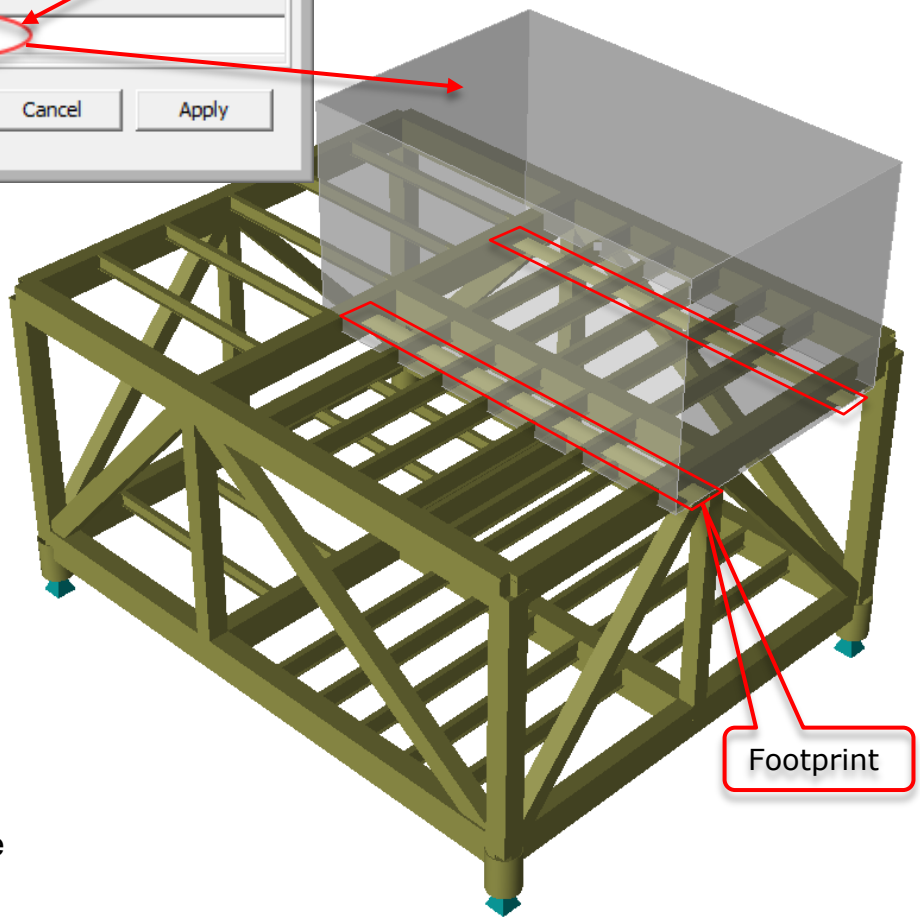


  - Position the equipment at (19.75, 13.5, 12.5) by right-clicking it in the *Equipment* folder, selecting *Place in Loadcase* (LC_equipm) and clicking in the model at a clickable point approximately at the centre of the main deck.

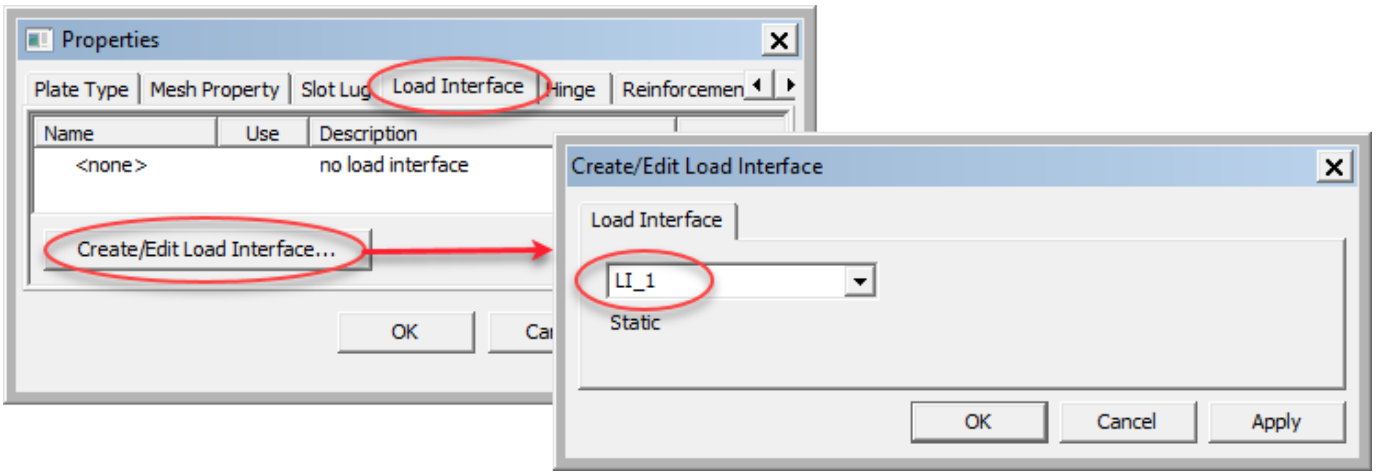
- Adjust the position of the equipment as shown.
- The *Default display* and *Modelling - Transparent* display configurations display equipments.



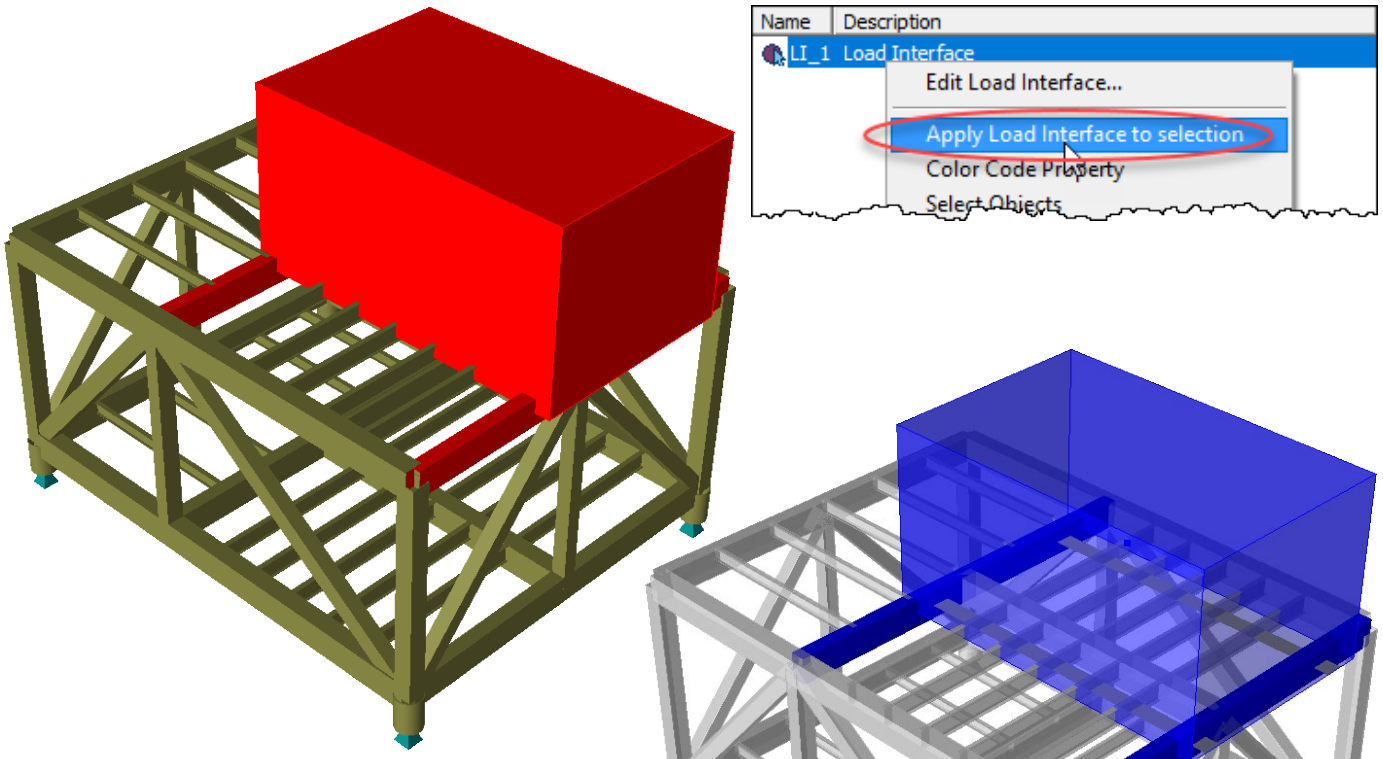
- Notice that the footprint touches the two main beams in the Y-direction with box ections plus the secondary beams with I section. This means that all these beams will carry the load from the equipment.
- However, only the two main beams shall carry the load. This is achieve by a *Load Interface*.



- Create a load interface by *Edit* | *Properties, go to the Load Interface* tab and enter a name as shown. There is no data associated with a load interface.

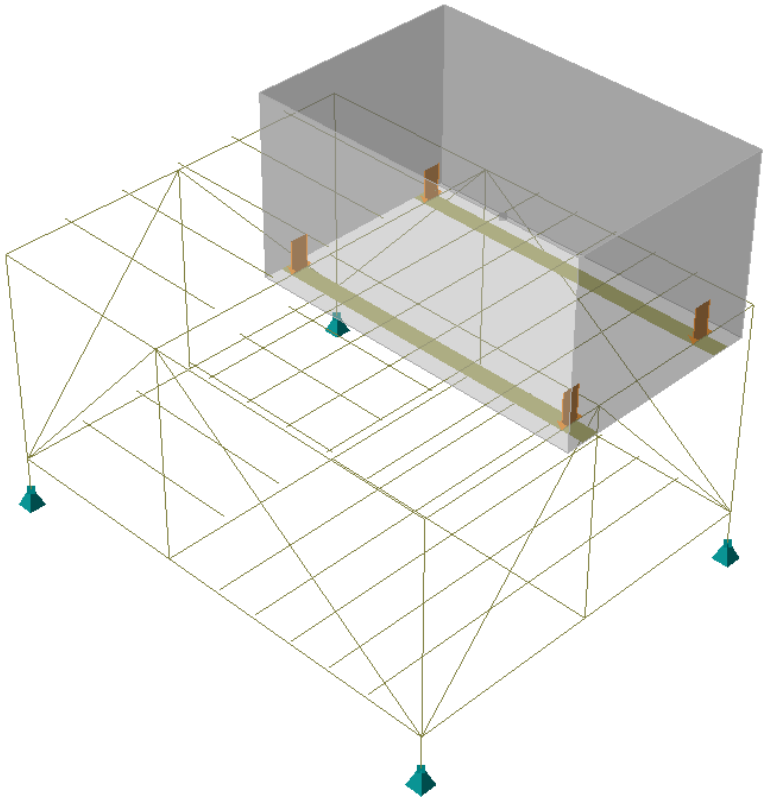


- Select the two main beams and the placed equipment and apply the load interface to the selection:

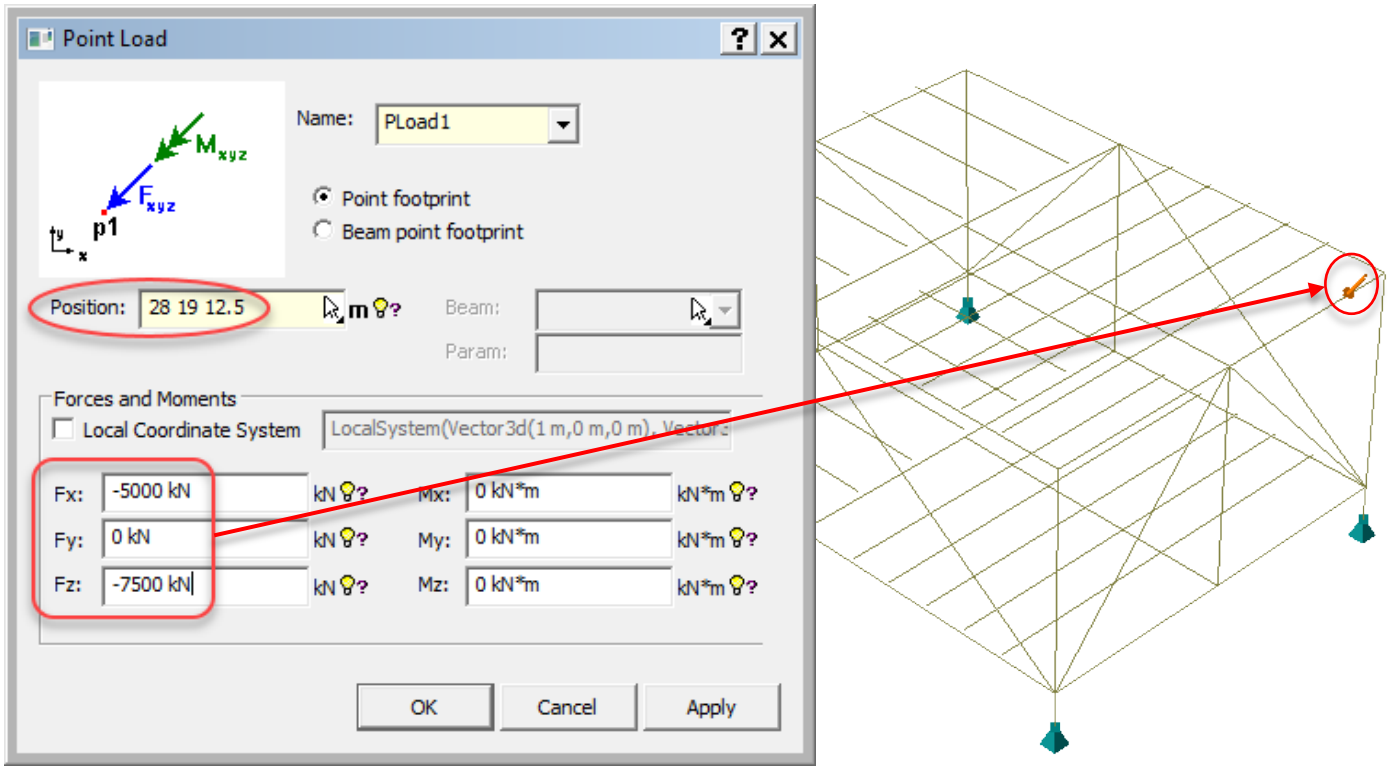


- The application of the load interface is verified by colour coding it.

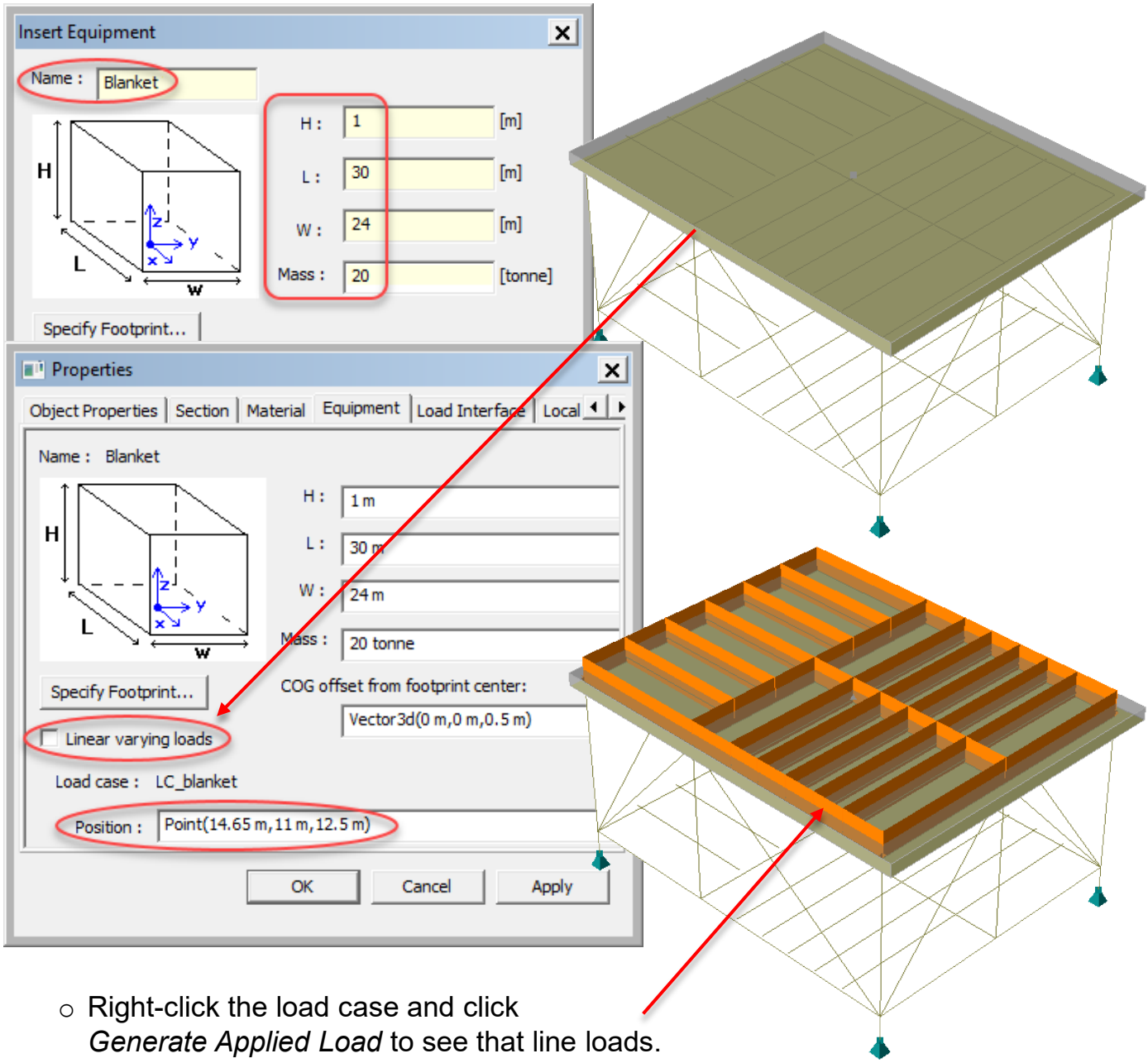
- Right-click the load case LC_equipm and click *Generate Applied Load* to see that line loads are created only where the footprints touches the two main beams.



- LC_point_load is a load case containing the point load (-5000,0,-7500) in position (28,19,12.5):



- LC_blanket is a load case containing evenly distributed line loads on all beams of the main deck. Use the equipment feature.
  - Create an equipment named Blanket with data as shown. The length and width should be somewhat larger than the extent of the main deck in X- and Y-directions.
  - Place the equipment at the midpoint of the main deck.
  - Select and right-click the equipment and select *Properties* to uncheck the option *Linear varying loads*. This involves that all line loads will be constant. (The moment balance of the equipment weight will not be preserved.)
  - Adjust *Position* if necessary so that the equipment extends outside the deck.

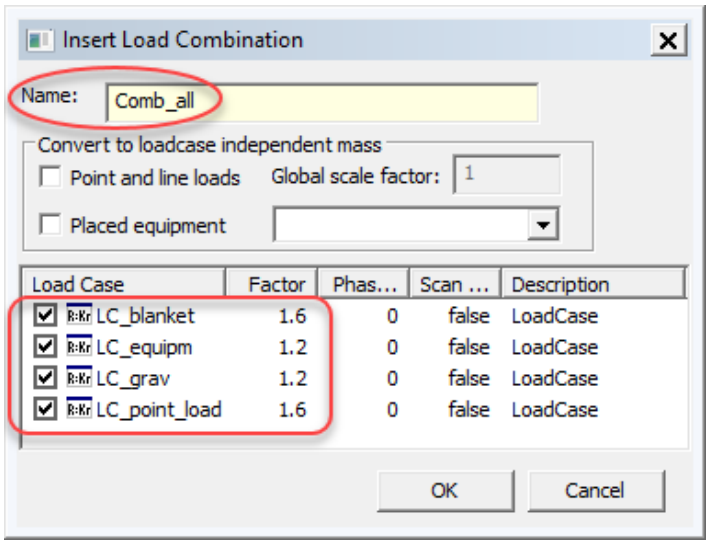


  - Right-click the load case and click *Generate Applied Load* to see that line loads.

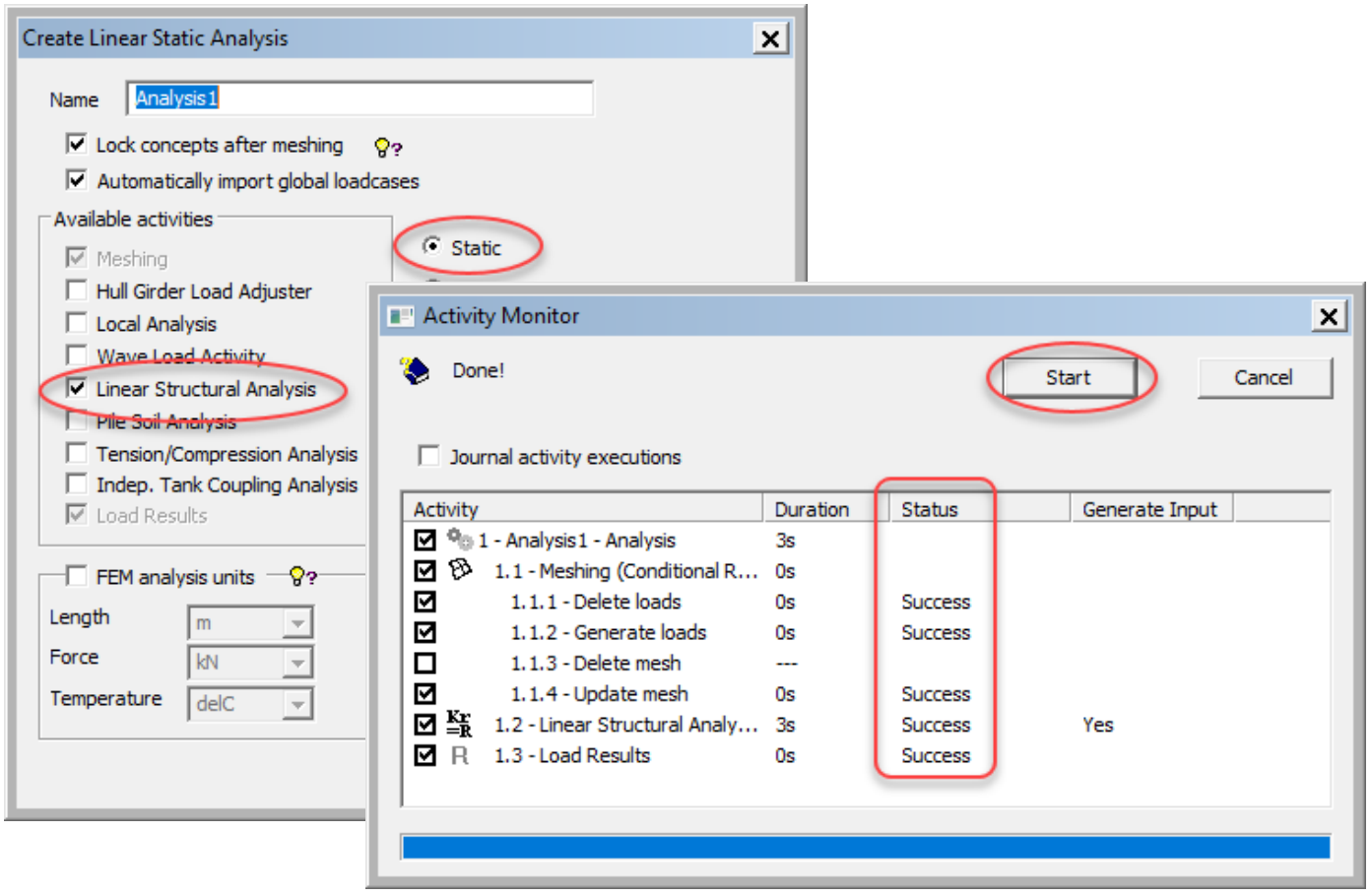
11 STATIC ANALYSIS

- Prior to running an analysis create a combination of all load cases with factors 1.2 and 1.6 as shown.
  - The *Load Cases* folder now contains:

Name	Description	FEM Loadcase	FEM LC Rul
LC_grav	LoadCase	1	Manual
LC_equipm	LoadCase	2	Manual
LC_point_load	LoadCase	3	Manual
LC_blanket	LoadCase	4	Manual
Comb_all	LoadCombination	5	Automatic



- Use *Mesh & Analysis* | *Activity Monitor* (or Alt+D) to create an analysis activity and click *Start* in the *Activity Monitor* to run the analysis. Make sure the analysis succeeds.

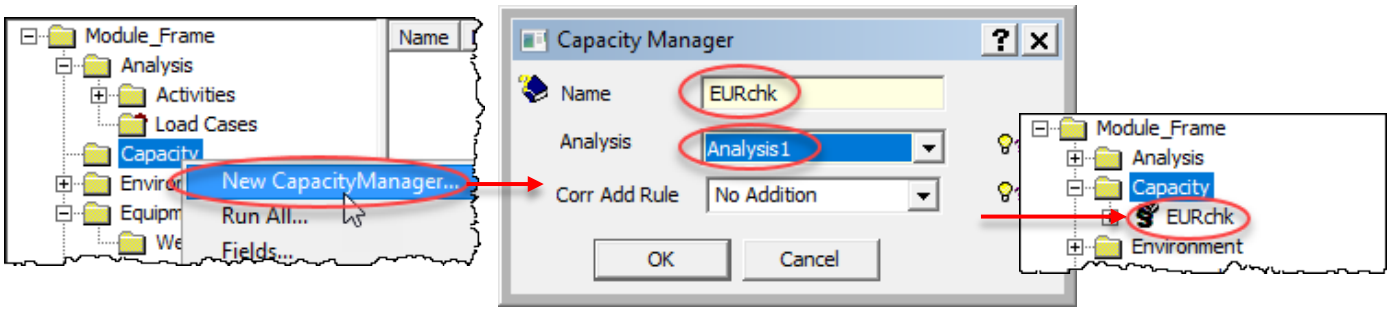


- Right-click the *Linear Structural Analysis* activity to open the Sestra.lis file and see load sums for the four load cases (internal number = *FEM Loadcase*):

```
Load sum for all result cases (internal number):
   result case;           tx;           ty;           tz;           rx;           ry;           rz
             1;  2.842171e-14;  0.000000e+00; -6.263183e+03; -6.889502e+04;  9.039593e+04; -3.126388e-13
             2;  0.000000e+00;  0.000000e+00; -1.961330e+04; -2.647795e+05;  3.873627e+05;  0.000000e+00
             3; -4.999995e+03;  0.000000e+00; -7.499993e+03; -1.424999e+05;  1.474999e+05;  9.499991e+04
             4;  0.000000e+00;  0.000000e+00; -1.961330e+02; -2.157463e+03;  2.862961e+03;  0.000000e+00
```

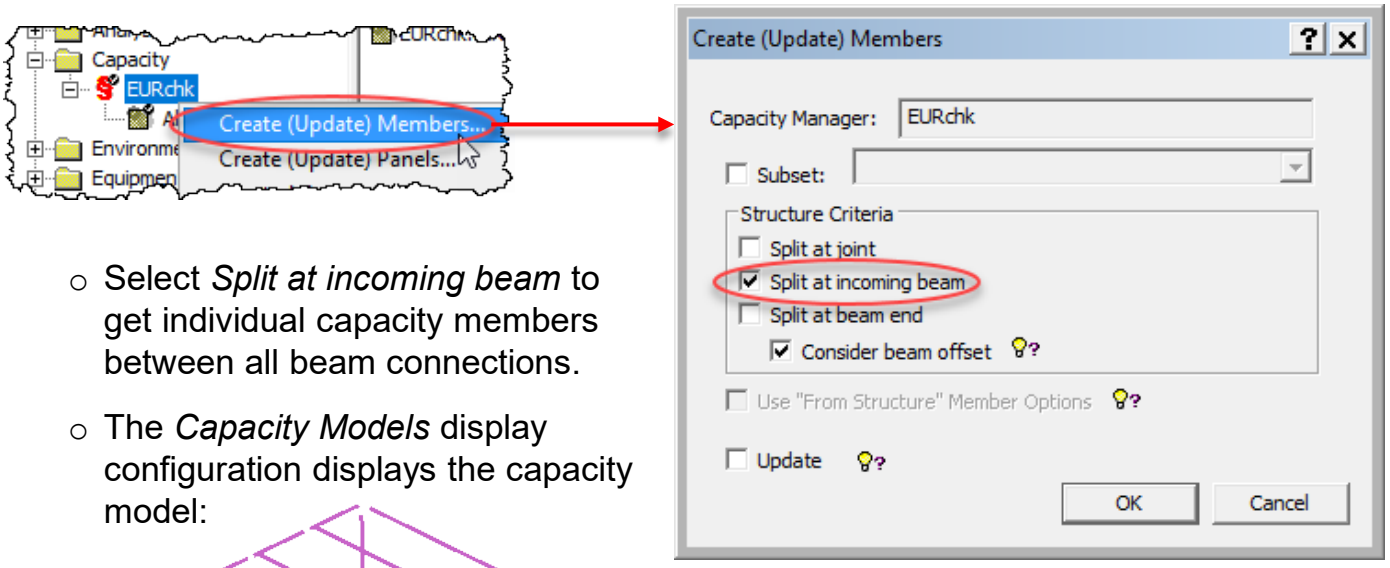
12 CODE CHECK

➢ Create a capacity manager by right-clicking the *Capacity* folder:

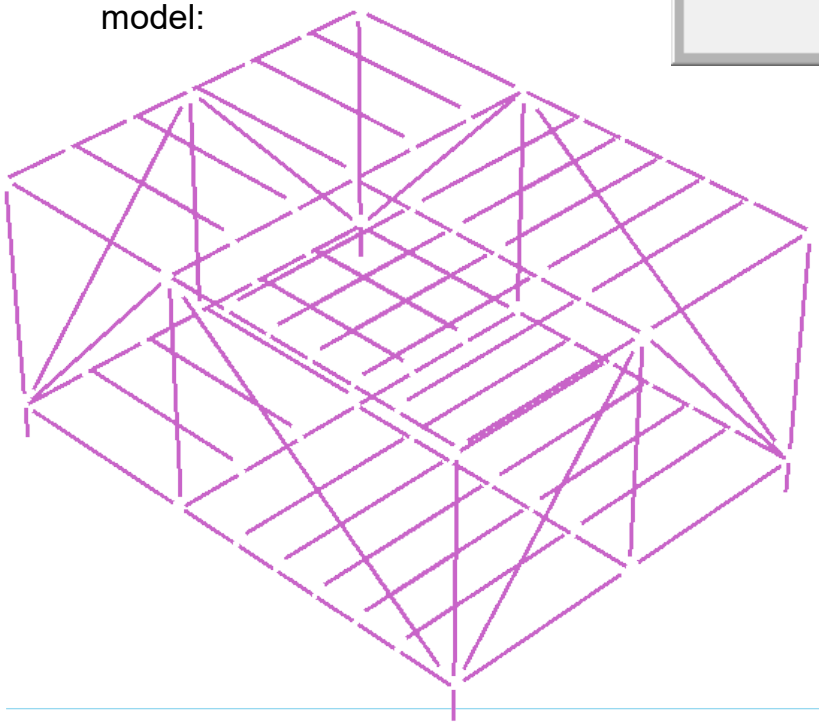


- The capacity manager contains:
  - Analysis results (Analysis1)
  - Capacity members to be code checked.

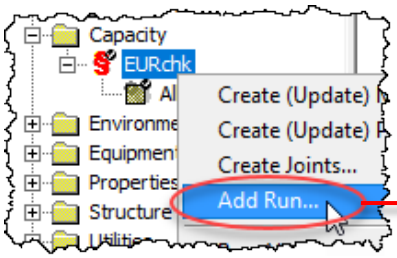
➢ Create capacity members by right-clicking the newly created capacity manager:

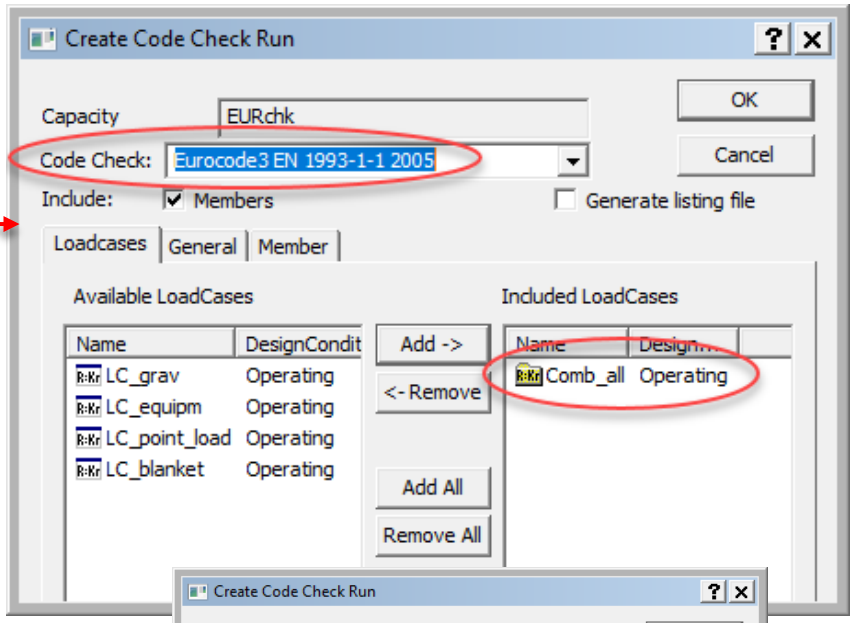


- Select *Split at incoming beam* to get individual capacity members between all beam connections.
- The *Capacity Models* display configuration displays the capacity model:

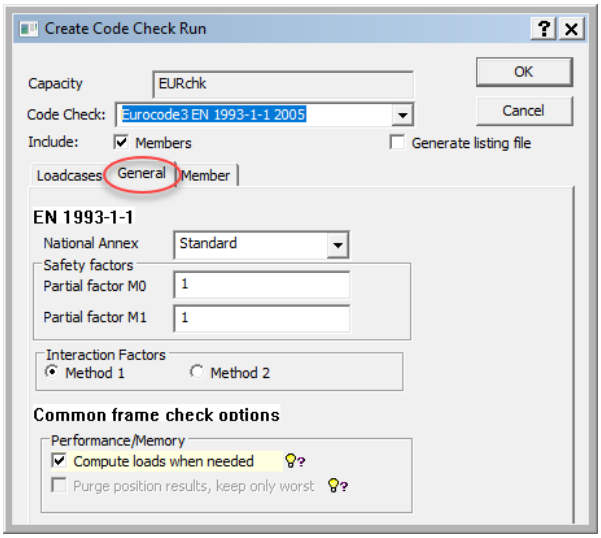


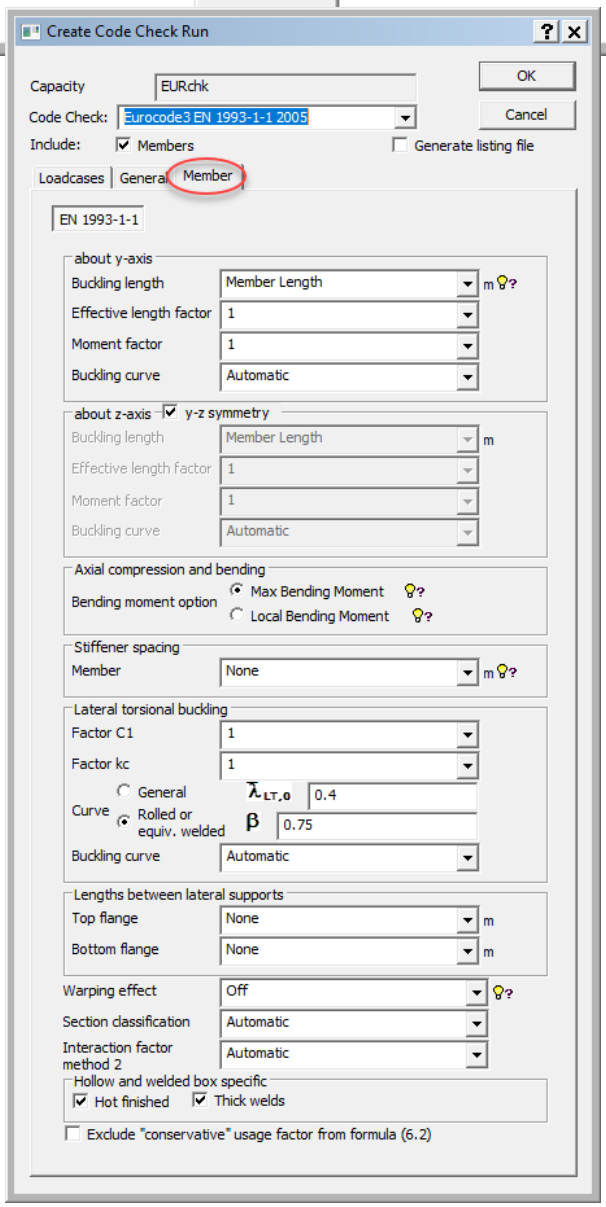
➢ Add a run to the capacity manager by right-clicking the created capacity manager:



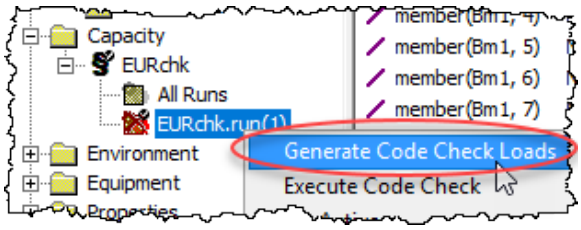


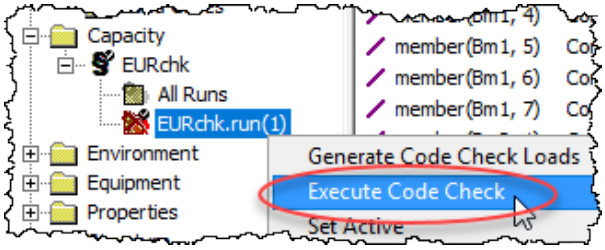
- In the *Create Code Check Run* dialog select *Eurocode* and move the load combination over to the *Included LoadCases* field.
- The tabs *General* and *Member* of the dialog are shown below. Accept all defaults in these tabs.



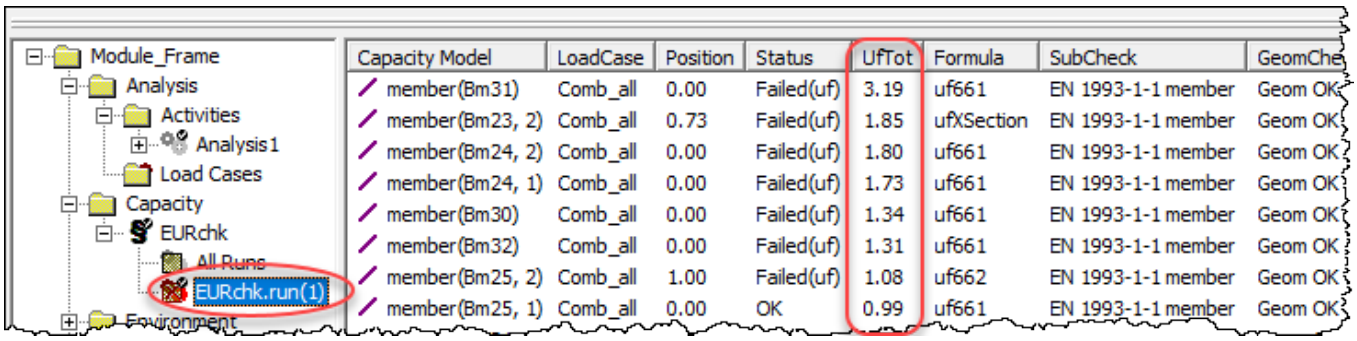


- In sequence, right-click the capacity manager or capacity manager run and click *Generate Code Check Loads* and *Execute Code Check*:





- View the code check results in the browser, *UfTot* is the worst usage factor:

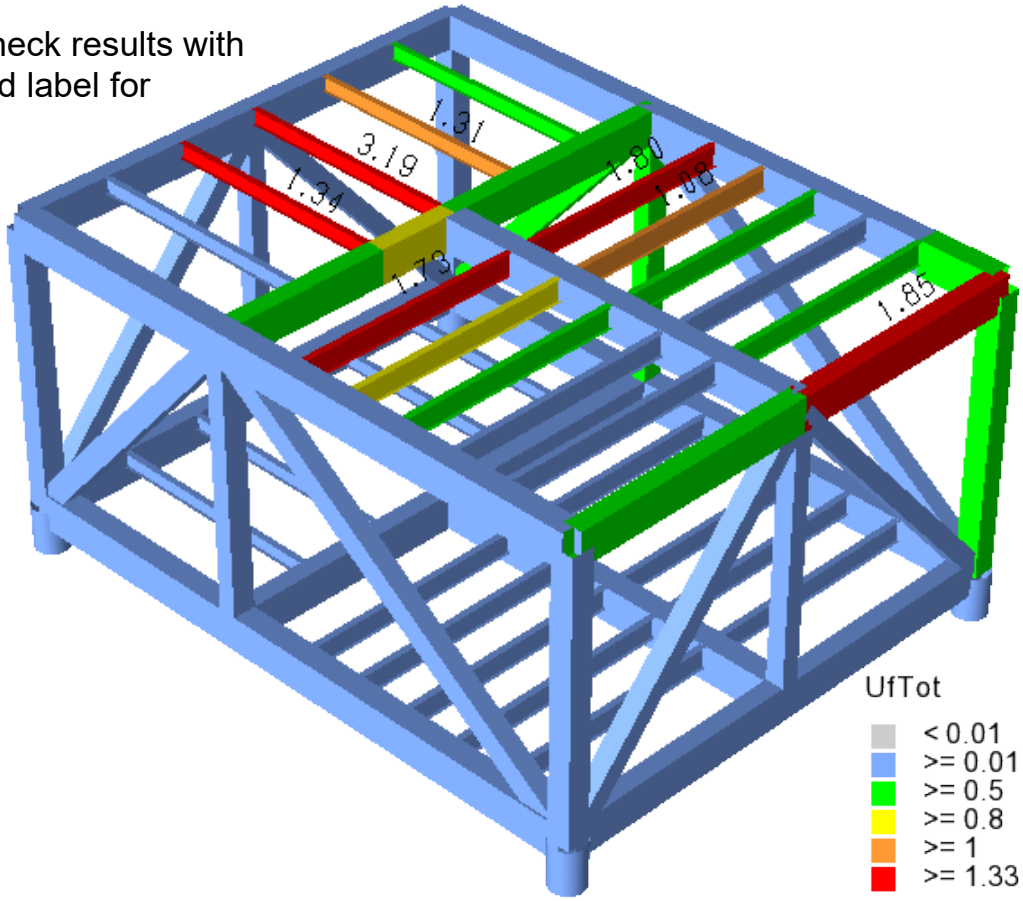


Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomChe
member(Bm31)	Comb_all	0.00	Failed(uf)	3.19	uf661	EN 1993-1-1 member	Geom OK
member(Bm23, 2)	Comb_all	0.73	Failed(uf)	1.85	ufXSection	EN 1993-1-1 member	Geom OK
member(Bm24, 2)	Comb_all	0.00	Failed(uf)	1.80	uf661	EN 1993-1-1 member	Geom OK
member(Bm24, 1)	Comb_all	0.00	Failed(uf)	1.73	uf661	EN 1993-1-1 member	Geom OK
member(Bm30)	Comb_all	0.00	Failed(uf)	1.34	uf661	EN 1993-1-1 member	Geom OK
member(Bm32)	Comb_all	0.00	Failed(uf)	1.31	uf661	EN 1993-1-1 member	Geom OK
member(Bm25, 2)	Comb_all	1.00	Failed(uf)	1.08	uf662	EN 1993-1-1 member	Geom OK
member(Bm25, 1)	Comb_all	0.00	OK	0.99	uf661	EN 1993-1-1 member	Geom OK

- The *Formula* column above contains references to equation numbers in Eurocode. *Uf661* is reference to equation (6.61).

$$\frac{N_{Ed}}{\frac{\chi_y N_{Rk}}{\gamma_{M1}}} + k_{yy}\frac{M_{y,Ed}+\Delta M_{y,Ed}}{\chi_{LT}\frac{M_{y,Rk}}{\gamma_{M1}}} + k_{yz}\frac{M_{z,Ed}+\Delta M_{z,Ed}}{\frac{M_{z,Rk}}{\gamma_{M1}}} \le 1 \qquad (6.61)$$

- Display the code check results with colour coding of and label for *UfTot* for the load combination.



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.

DNV | digital@dnv.com | dnv.com/digital