SESAM TUTORIAL

GeniE
Learning the Basics and Getting Started

Valid from program version 8.2

Sesam Tutorial

GeniE – Learning the Basics and Getting Started

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

➢ This tutorial explains how to create and analyse a simple structural model with two basic loads.

➢ The following modelling topics are covered:

  o Beam modelling

  o Load application

  o Support point modelling

  o Static analysis

  o Results presentation

  o View options

➢ The tutorial does not require any previous knowledge in the use of GeniE.

➢ A GeniE input file for creating the model is provided.

➢ The appearance of the GUI and dialogs in later versions of GeniE may change.
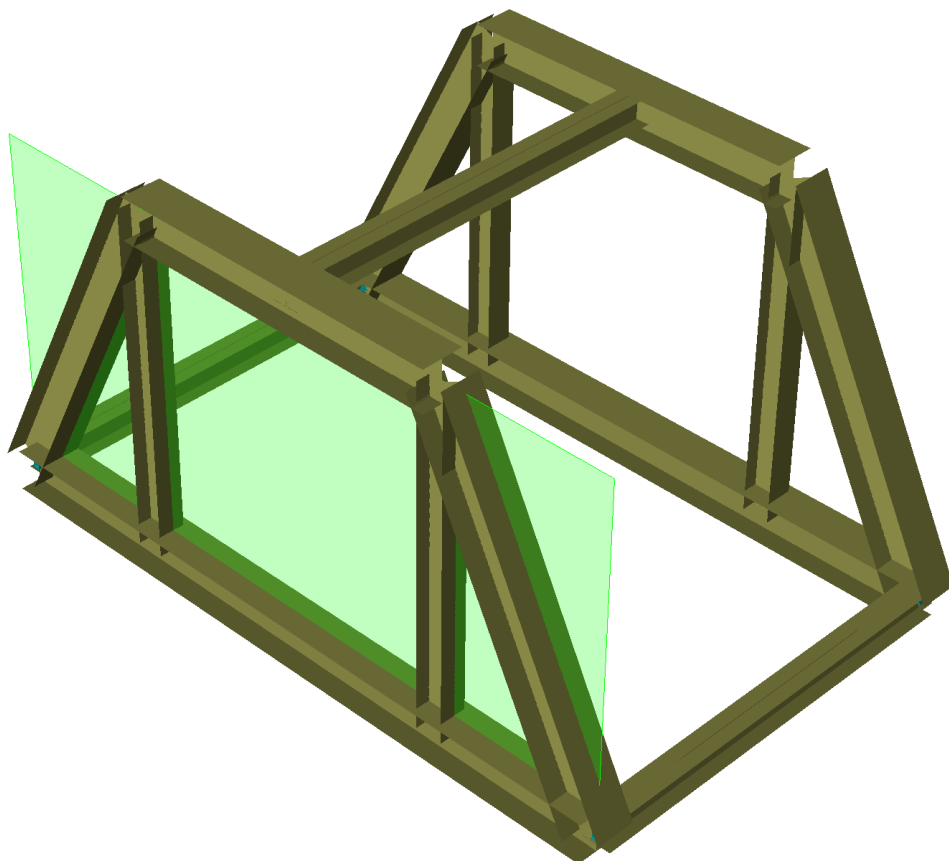
## 2   OPEN WORKSPACE, DEFINE MATERIAL AND BEAM CROSS SECTIONS

➢ Start GeniE and open a new workspace.

- o Give a *Workspace name*.

   - ▪ Notice that the workspace name also appears as the workspace folder in the *Location* field.

- o Accept default *Output Units* m and N and click *OK*.

   - ▪ Unless otherwise specified, all values in this tutorial are in these units.

- o Check the *Jacket mode* button to limit menus to those relevant for jacket (spaceframe) modelling.

➢ Define steel material.
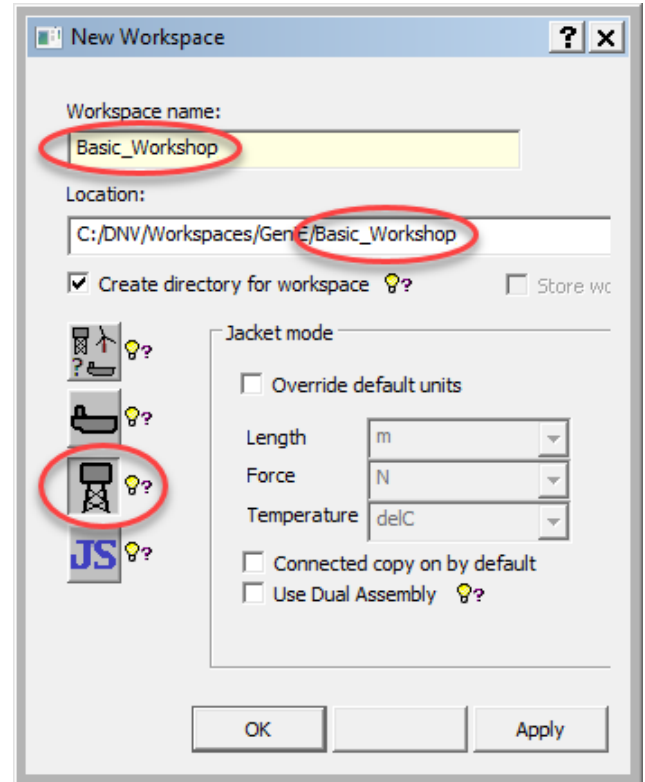
- o Use *Edit | Properties* to open the *Properties* dialog.

- o In the *Material* tab click *Create/Edit Material*.

- o In the *Material* dialog give a material name and a *Yield* value. Accept default values and click *OK*.
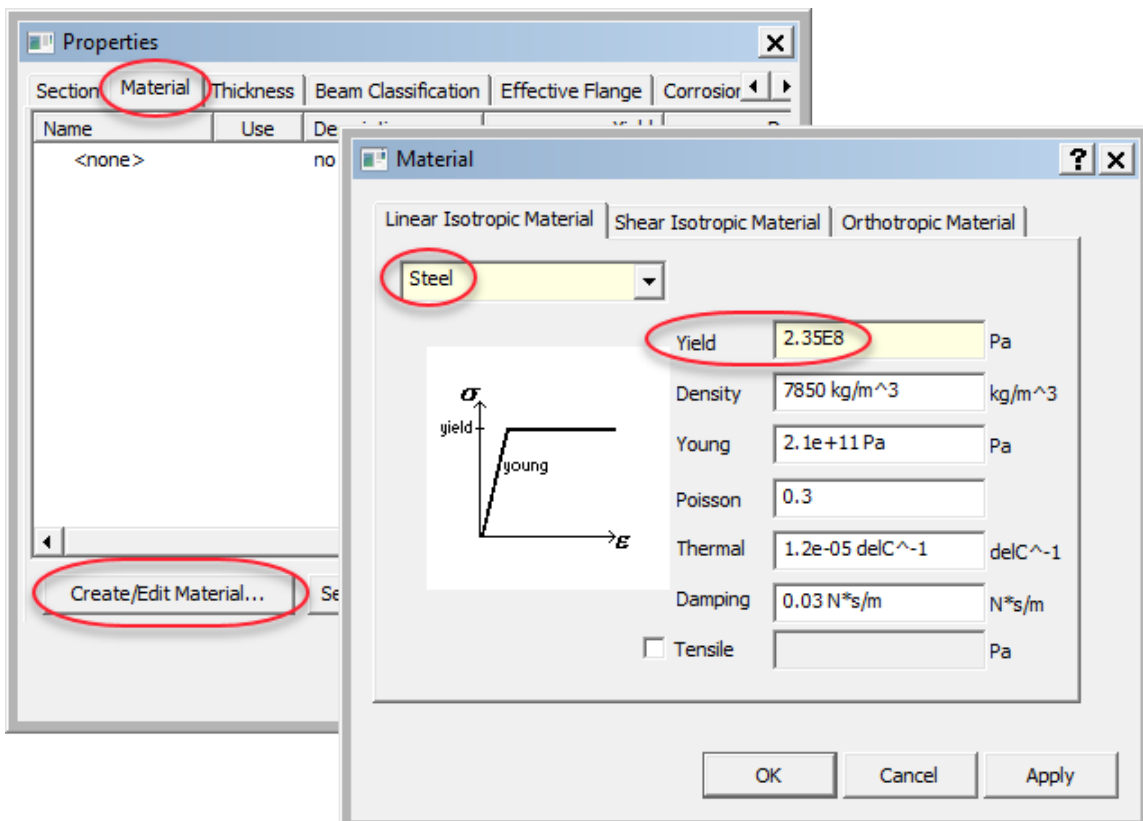
➢ Define beam cross section properties by clicking *Create/Edit Section* in the *Section* tab of the *Properties* dialog. In the *Sections* dialog go to the *I Section* tab and create sections named I600 and I400 with data as shown below.
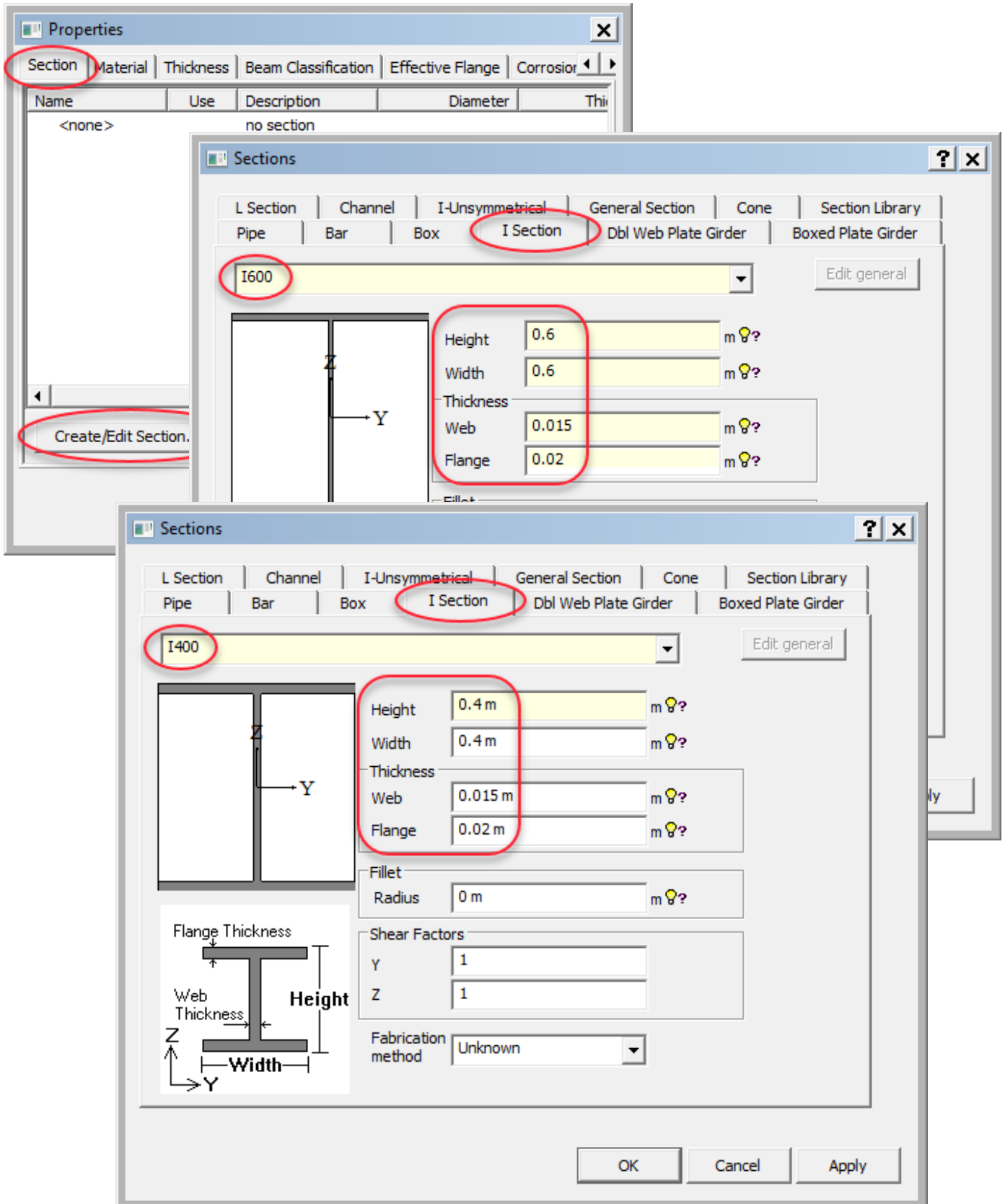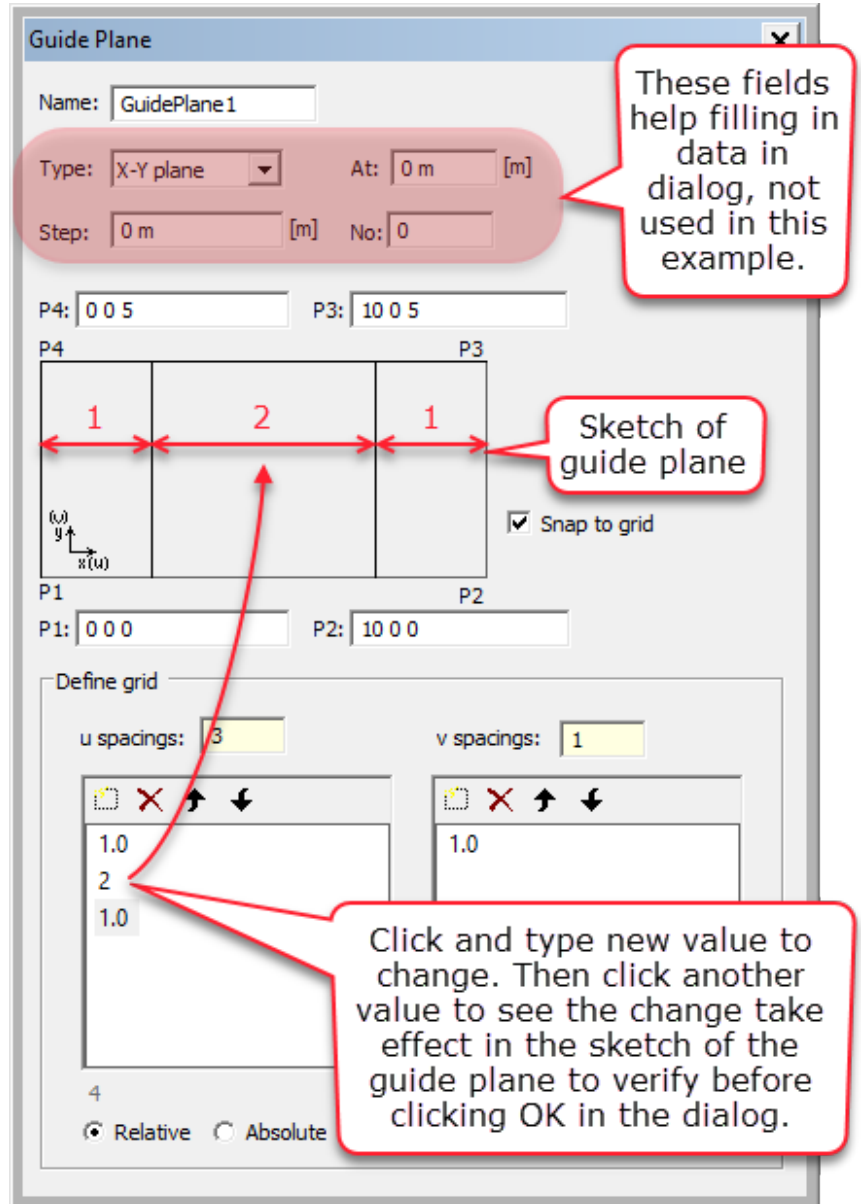
3 GUIDING GEOMETRY

➢ Use *Guiding Geometry | Planes | Guide Plane Dialog* to create a guide plane with values as shown.

- ○ Take care in entering correct data for P1 to P4 as well as the grid.

- ○ Note that a coordinate may be specified using the full syntax:
      Point(0 m,0 m, 5m)
  or simplified separating values by space:
      0 m 0 m 5 m
 or, if m is default unit, even simpler:
      0 0 5

- ○ An easy way of entering the data is to start in P4, fill in '0 0 5', jump with Tab key to P3, fill in '10 0 5' and so on.

- ○ Check the sketch of the guide plane before clicking OK.

- ○ Click OK and see that the guide plane appears in the display area.

- ○ Provided the *Default display* configuration is chosen the guide plane will be displayed. Use the *Iso view* (F5) and *View from Y* (F7) buttons, see below, in the toolbar to view the guide plane from different view points.
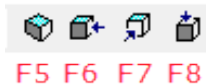
F5 F6 F7 F8

Guide Plane dialog

These fields help filling in data in dialog, not used in this example.

Sketch of guide plane

Click and type new value to change. Then click another value to see the change take effect in the sketch of the guide plane to verify before clicking OK in the dialog.
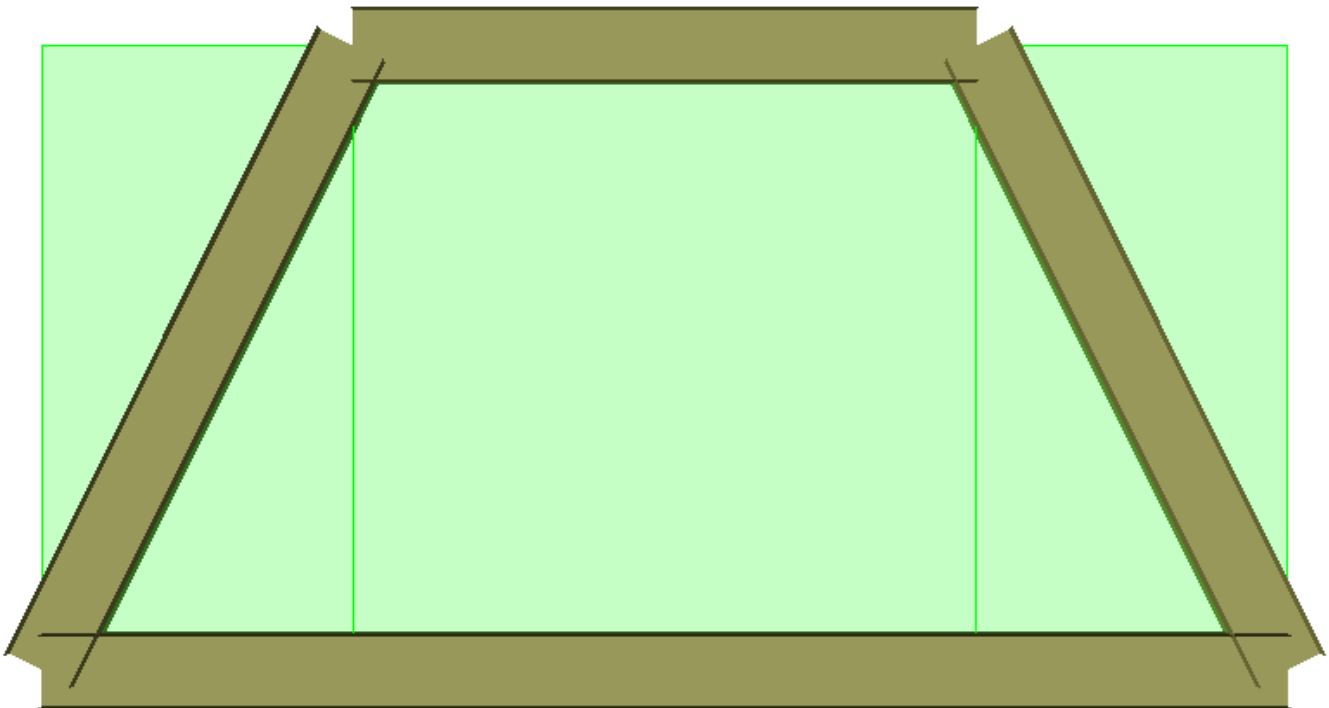

F5 view
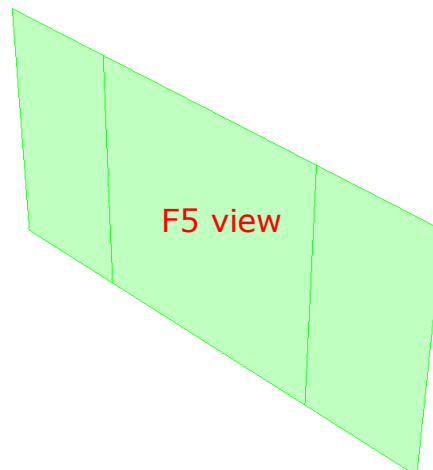
4 CREATE BEAMS

➢ Prior to creating beams set proper default beam cross section (I600) and material (Steel) as shown to the right.


➢ Start inserting beams by *Structure | Beams and Piles | Straight Beam*, or click the *Beam* button (╱) in the toolbar.

 ○ Click in the guide plane to insert the four beams shown below.

➢ Change default section to I400 and insert two more beams as shown below.
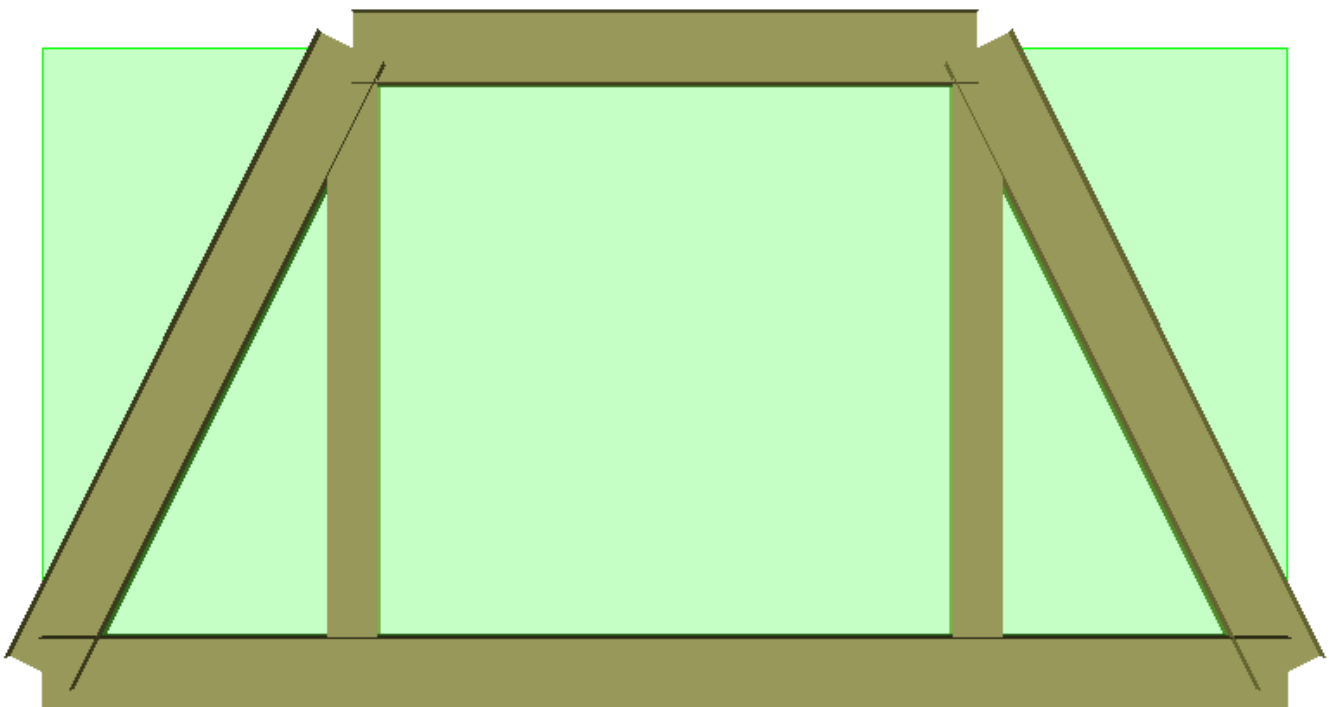
➢ Select *Iso view* (F5). The vertical beams (columns) should be rotated 90º about their axes.

➢ To select the beams the *Selection* button ![cursor] must be depressed. Hint: The Esc key will in most cases depress the *Selection* button. Click one beam and Shift+click the other beam. Right-click to open the *Edit Beams* dialog as shown below.

➢ Select *Rotate around local X-axis* 90° and click *OK*.

   o Note that clicking *Apply* rotates the beams and keeps the dialog open. If you then click *OK* the operation is repeated and the beams are rotated 180°. In such case click *Cancel* rather than *OK*.
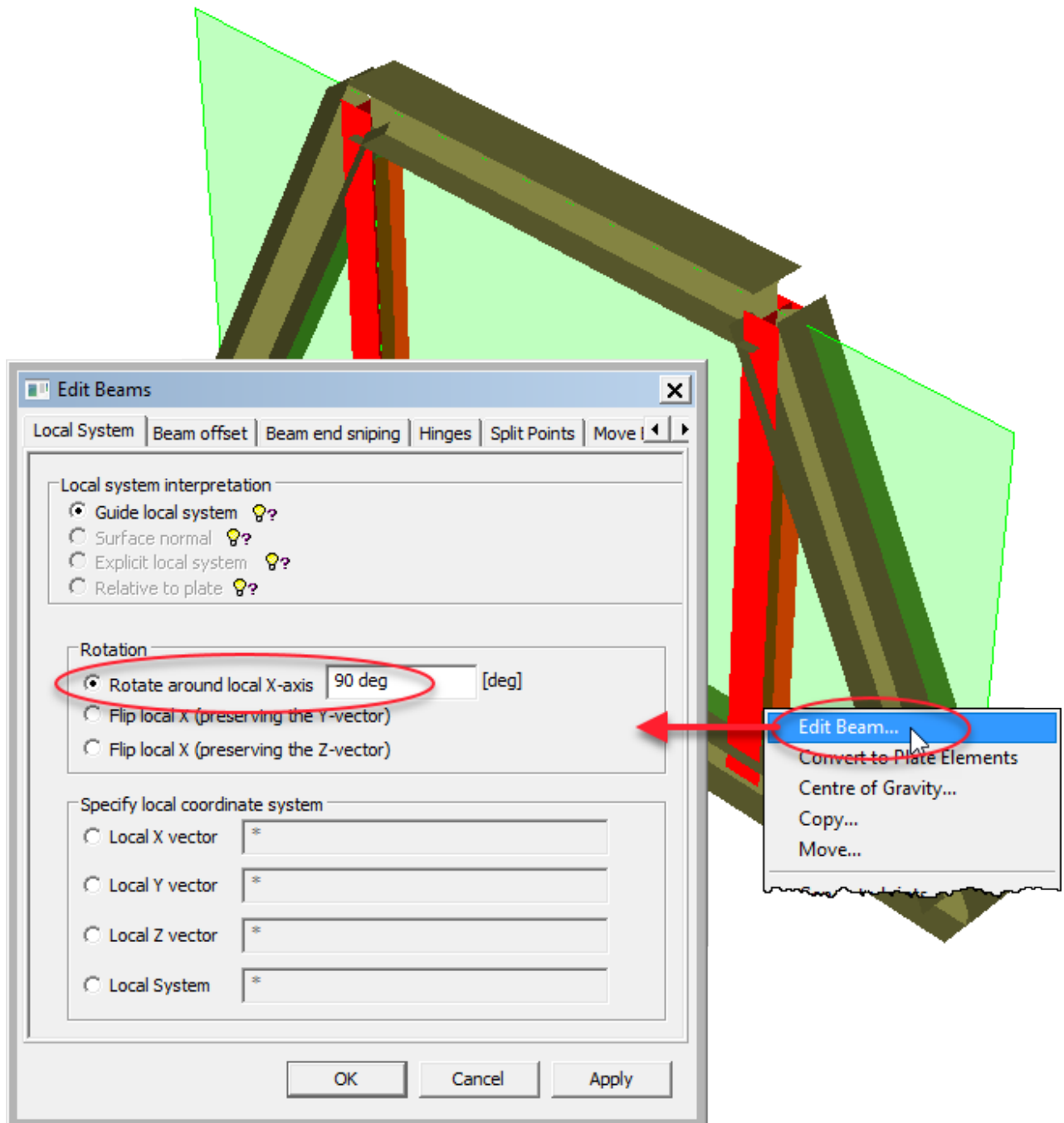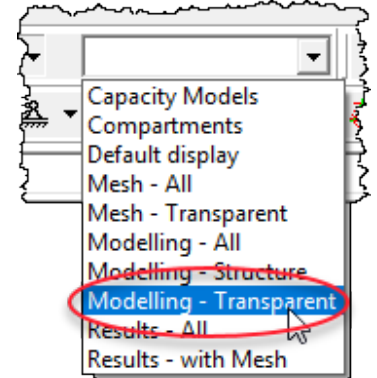
> See that the beams are listed in the *Structure* folder in the browser to the left in the GeniE window.
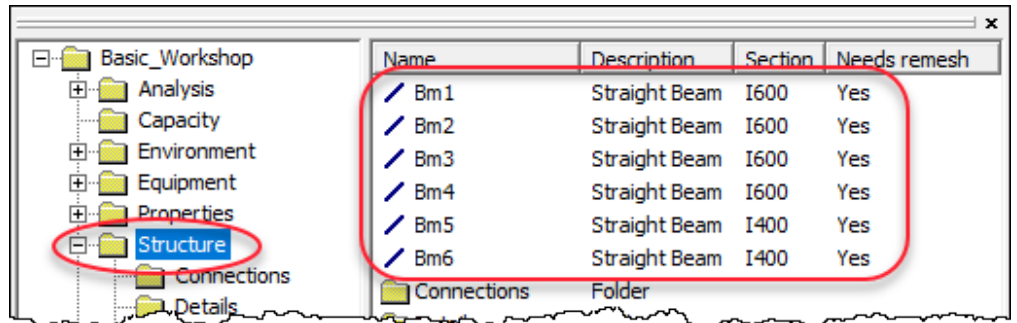
> Change display configuration to *Modelling - Structure* or *Modelling - Transparent* (i.e. don't show guide plane).
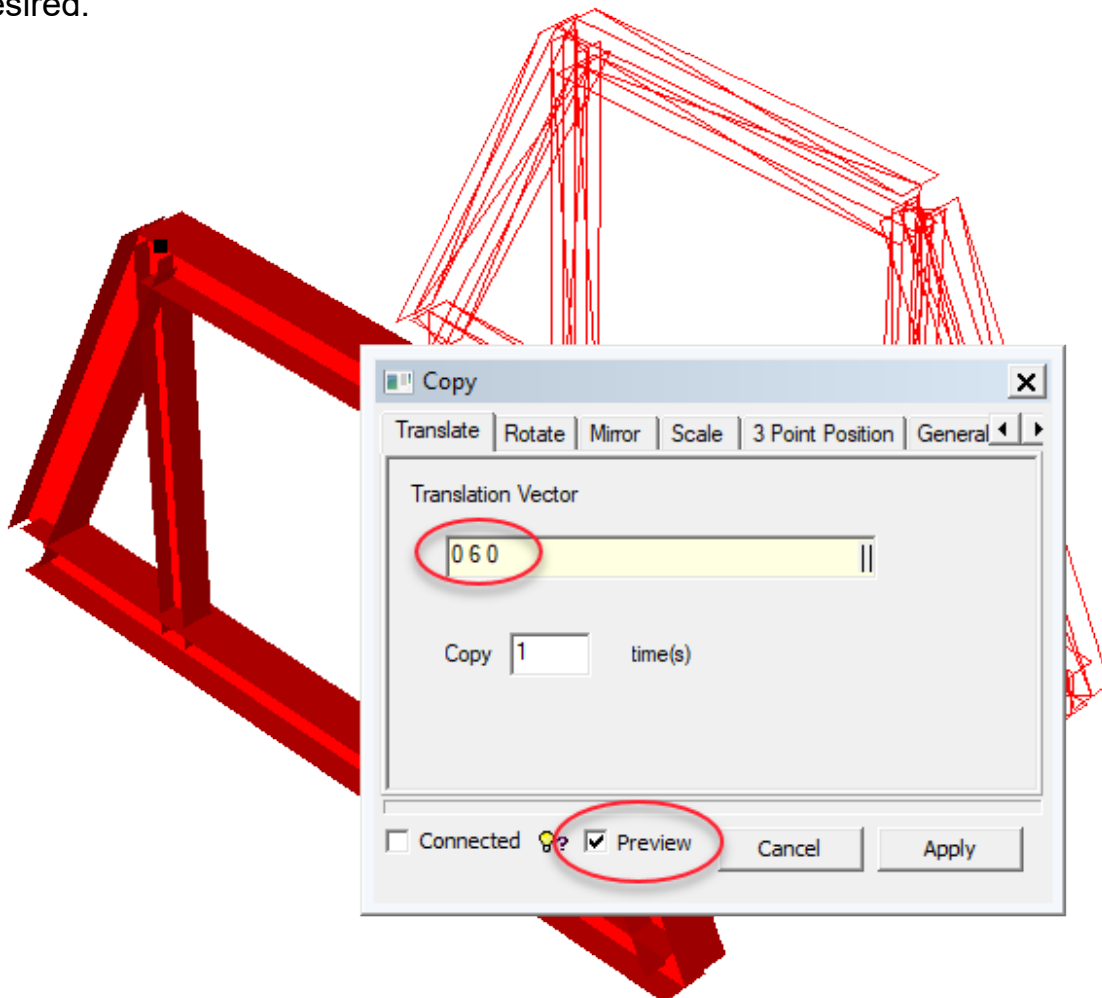
> Select all beams by dragging a rubberband.
> Then right-click to open the *Copy* dialog.

   o Make a copy of the frame using the *Translation Vector* (0,6,0) as shown below (copy in Y-direction).

   o Check *Preview* to see a preview of the copy.

   o Click *Apply* to perform the copying. Then click *Cancel* as no more copying is desired.

➢ Insert a beam using a dialog: *Structure > Beams and Piles > Straight Beam Dialog*. Use the existing geometry to specify appropriate end coordinates:

1. Click in the *End 1* field (you don't have to delete the data there).

2. Click the end of one of the two upper horizontal beams and see that the coordinate of this point (2.5,0,5) pops into the *End 1* field.

3. Change the X coordinate from 2.5 to 5.

4. Do a similar process for *End 2* to get coordinate (5,6,5), or simply enter 5 6 5.
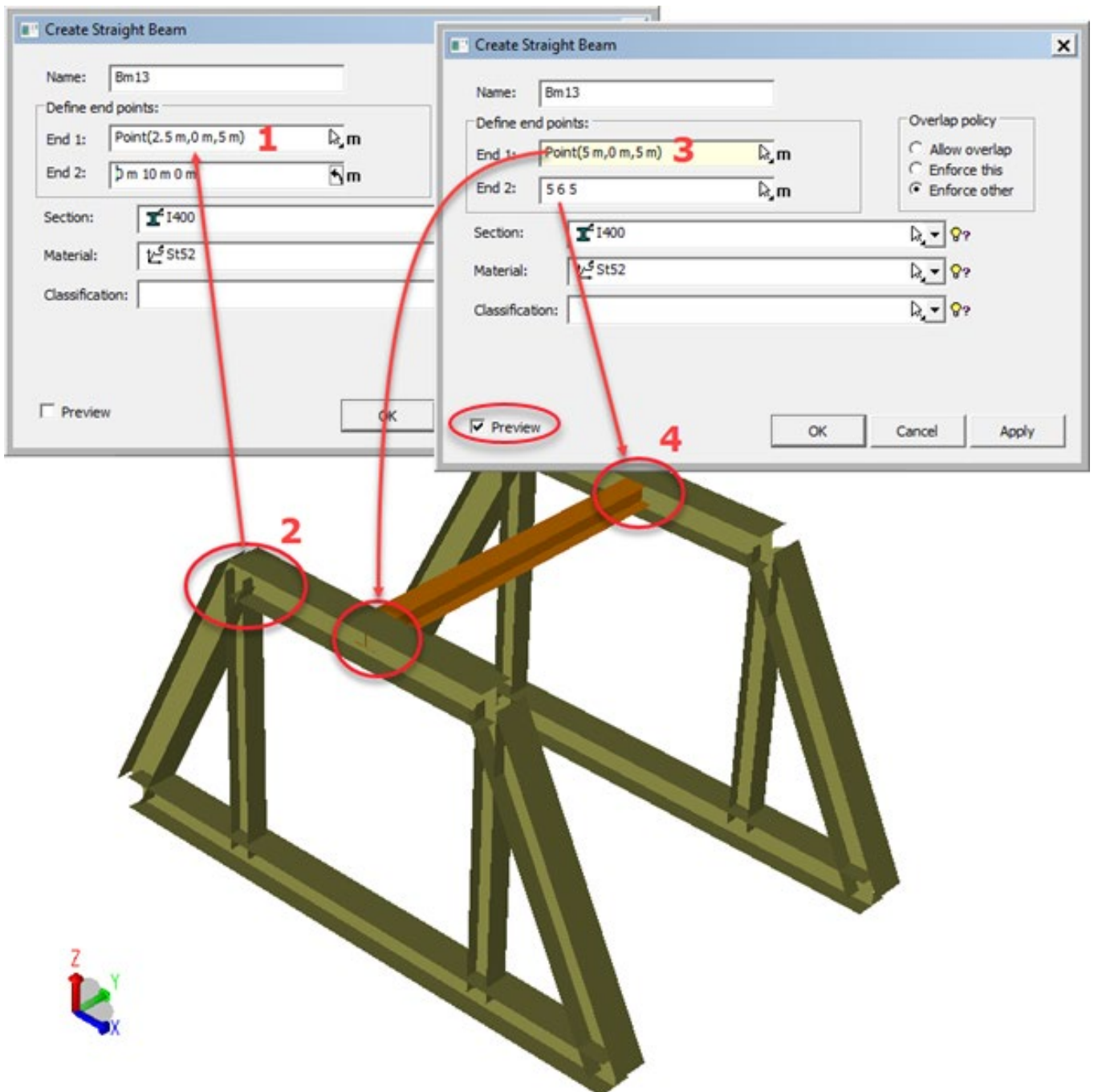
➢ Select the new beam, right-click and select *Copy*. Specify the *Translation Vector* of the copy process as follows:

1. Click in the *Translation Vector* field.

2. Click an end of the beam to copy.

3. Click the point to where the new beam should be copied and see that the appropriate vector appears in the field.

4. Click *Apply*.
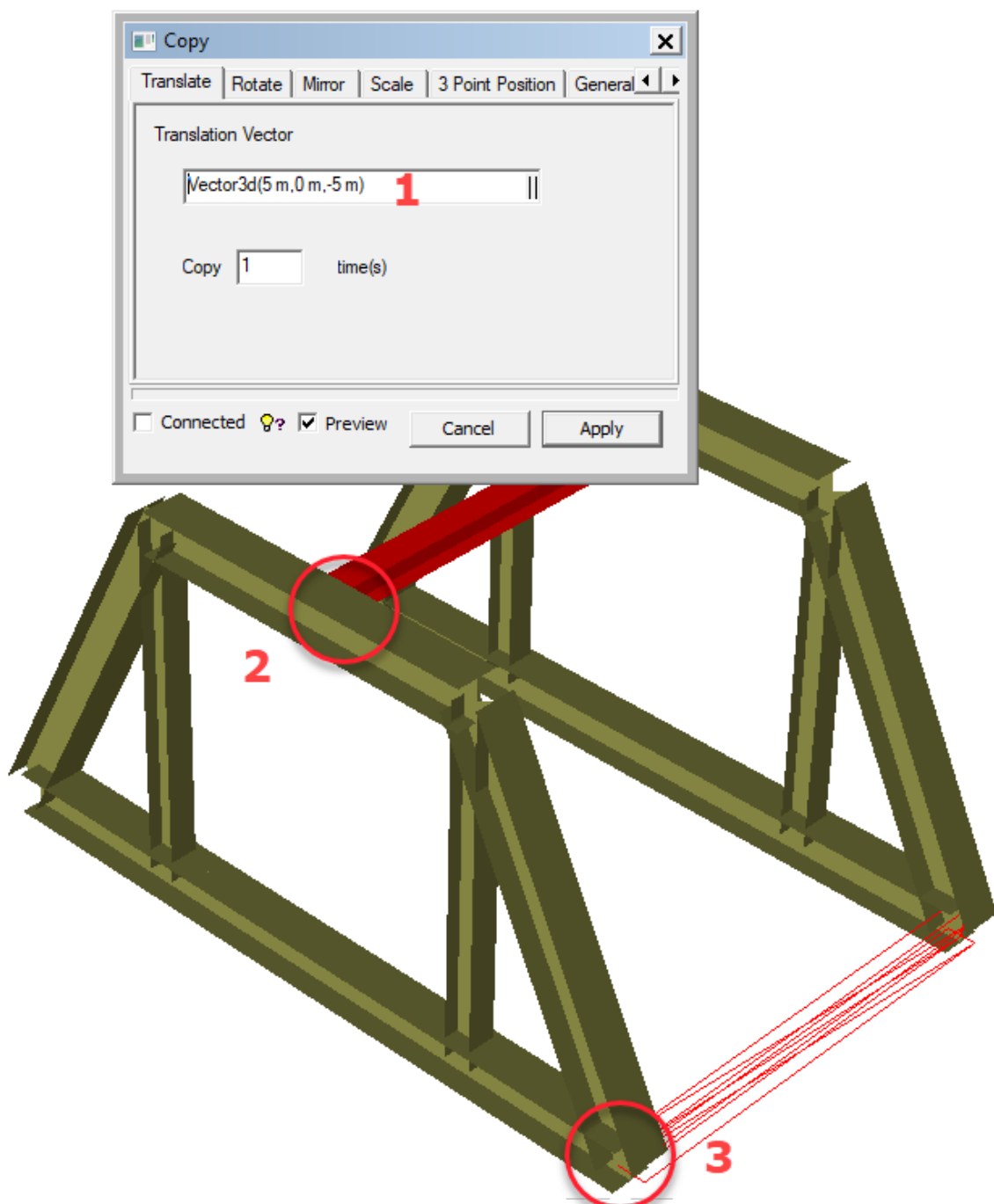
➢ Make another copy of the beam to connect the other ends of the two lower horizontal beams. The complete model is shown below.
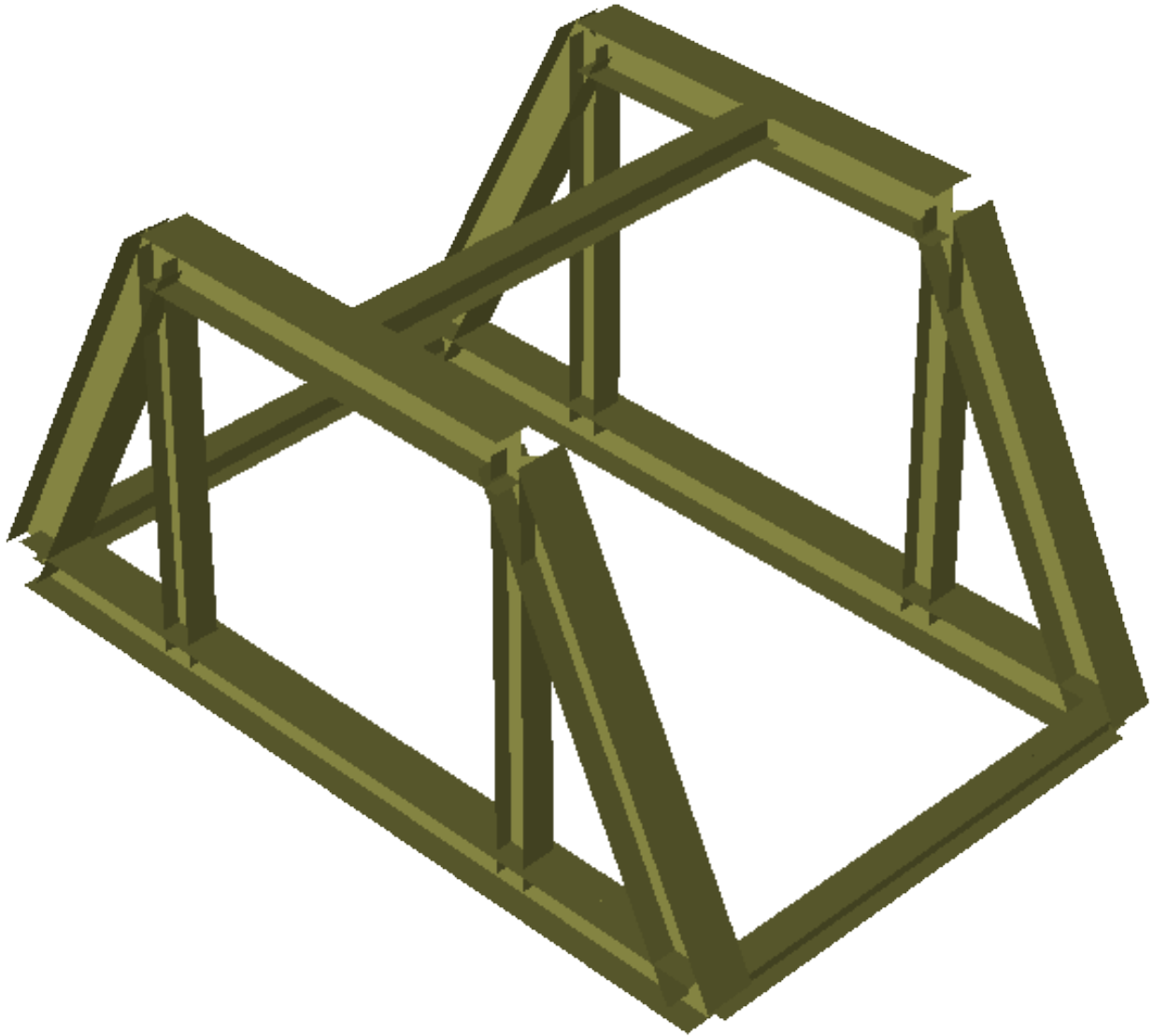
5 CREATE SUPPORTS

➢ Add supports by *Structure > Support > Support Point* (or click ⚓) in the lower four corners as shown. By default, all six degrees of freedom (dofs) will be fixed.

➢ Select the supports one-by-one, right-click and select *Properties*. In the *Support* tab adjust the boundary conditions so that all rotations are free and the three translations are fixed or free as indicated by the figure below.
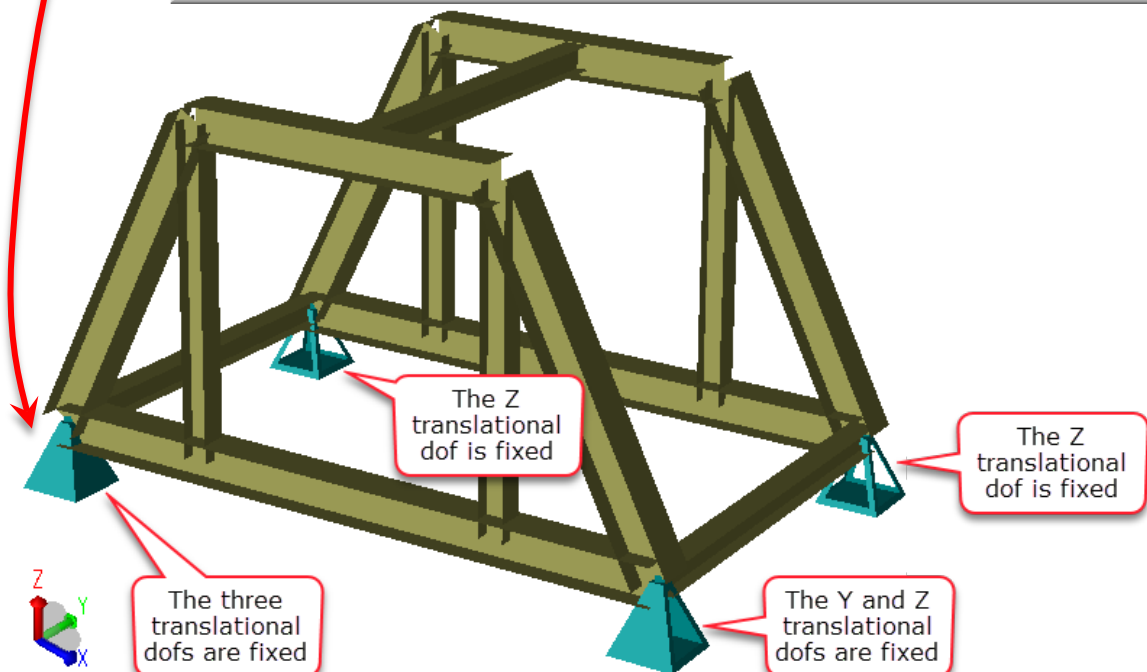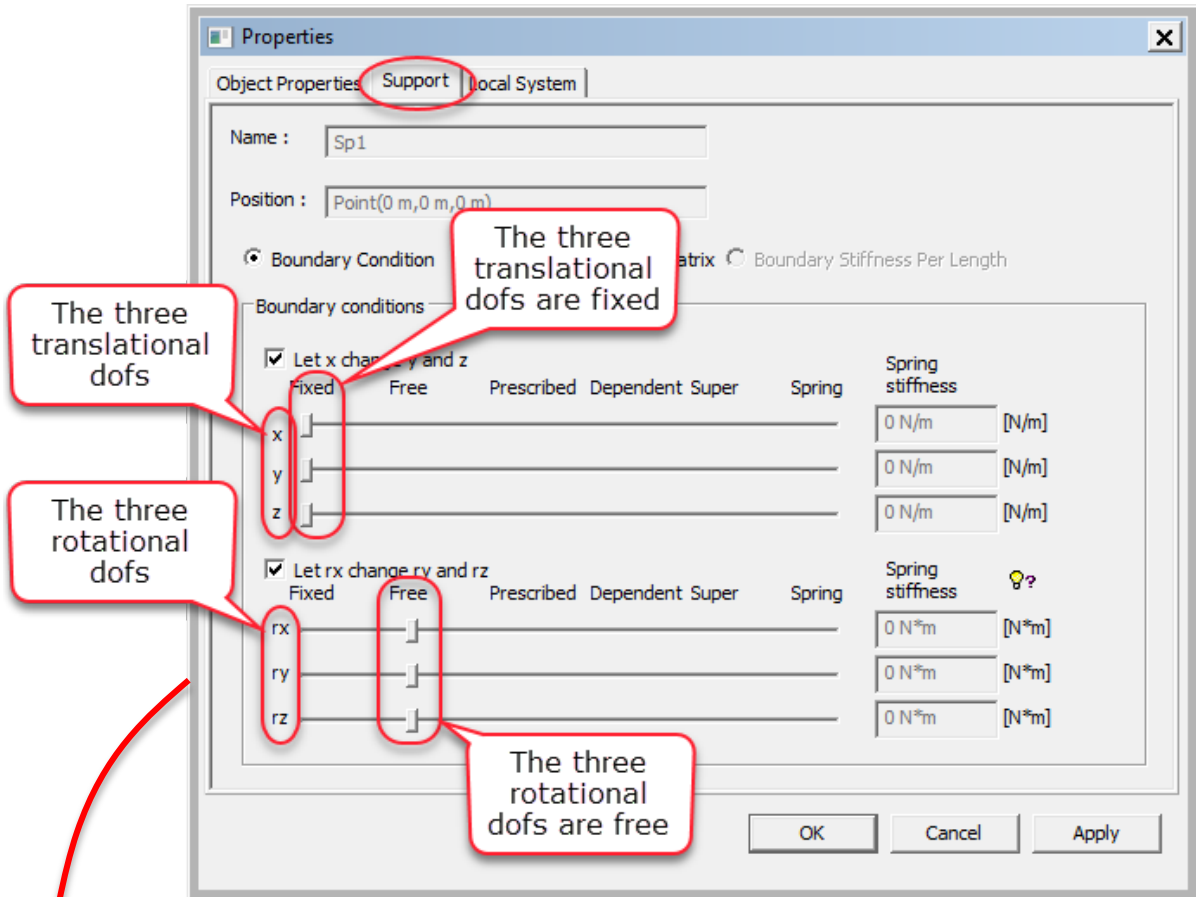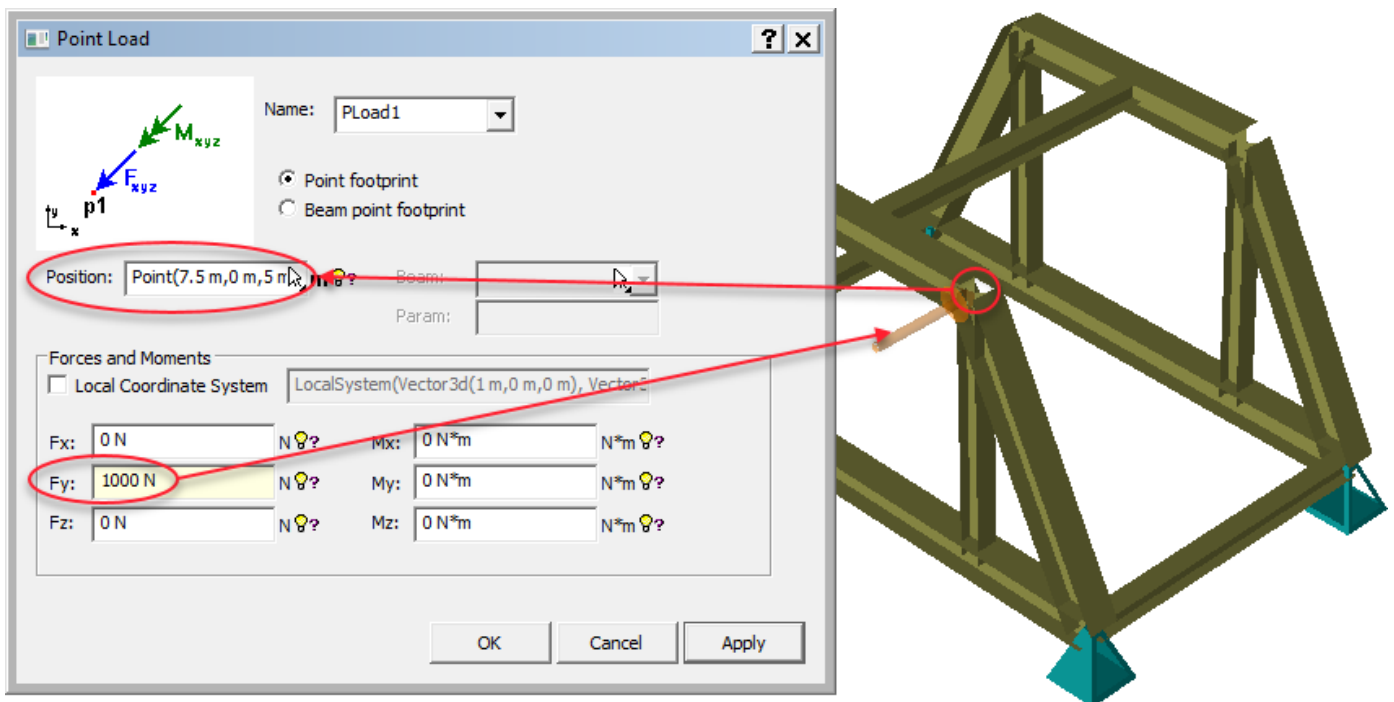
6 CREATE LOADS

➢ Create a load case named LC1 by *Loads > Load Case*. Accept the default load case name LC1. See that this appears in the *Load Cases* folder in the browser as well as the current load case.
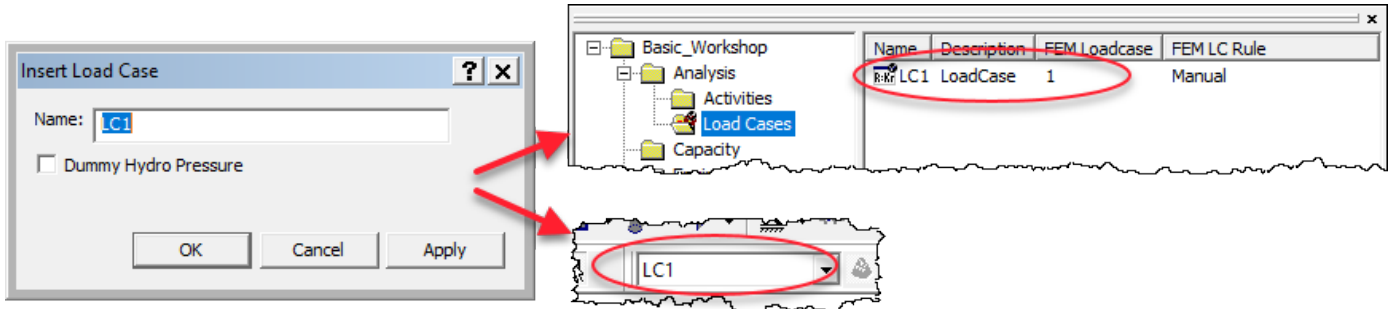
➢ Note that when a load case is set as current, new loads will be added to this load case.

➢ Fill LC1 with a point load by *Loads > Explicit Load > Point Load* and enter data as shown below.

  o The position of the load is specified by clicking in the *Position* field followed by clicking in the model.

  o Give a force of 1000 N in Y-direction.

  o To see the load in the display select *Modelling - Transparent* display configuration. *Modelling - Structure* display configuration will by default not display loads.

- ➢ Create a new load case named LC2. See that this appears as the new current load case.

- ➢ Fill LC2 with a line load by *Loads > Explicit Load > Line Load*. Enter data as shown below.

  - ○ Select points *p1* and *p2* by clicking the model (the insertion point jumps from *p1* to *p2*).

 - ○ Give a line load of 500 N/m in Y-direction.
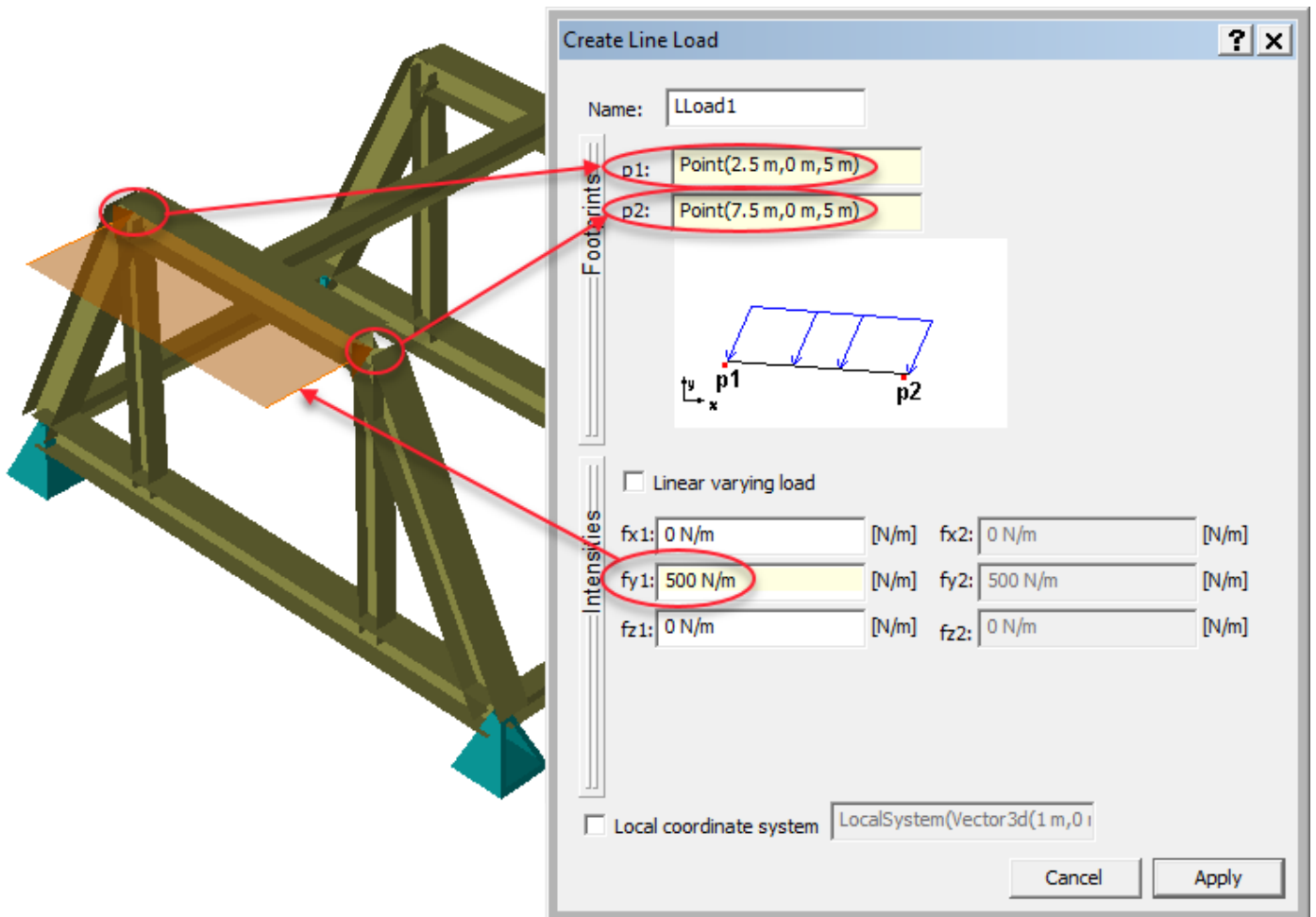
7 CREATE AND RUN AN ANALYSIS

➢ Create an analysis activity by *Mesh & Analysis | Activity Monitor* (or Alt+D).

- o By default a *Static Linear Structural Analysis* will be done.

- o Accept this and click *OK*.

➢ The *Activity Monitor* opens. See that the analysis consists of:

- o *1.1 - Meshing*

- o *1.2 - Linear Structural Analysis*

- o *1.3 - Load Results* (loading analysis results into GeniE)
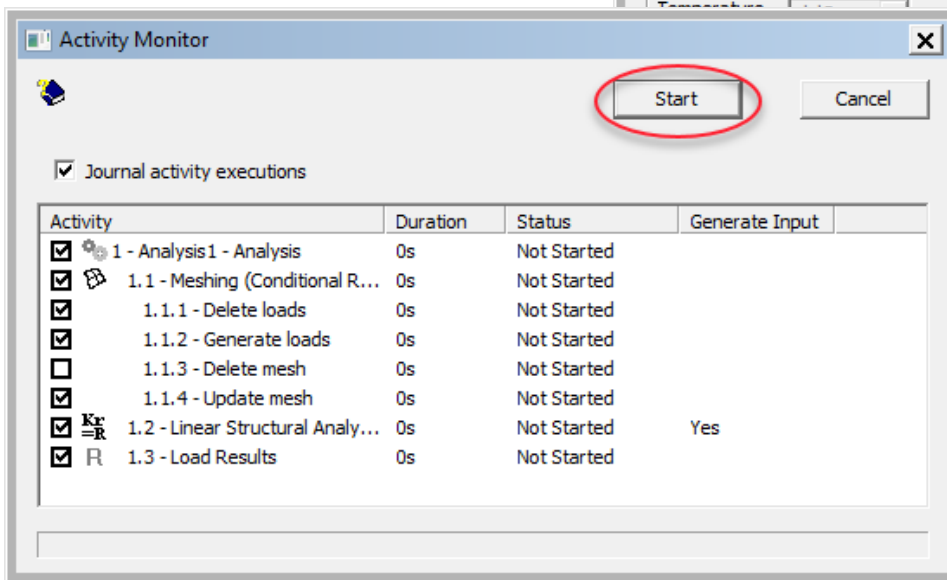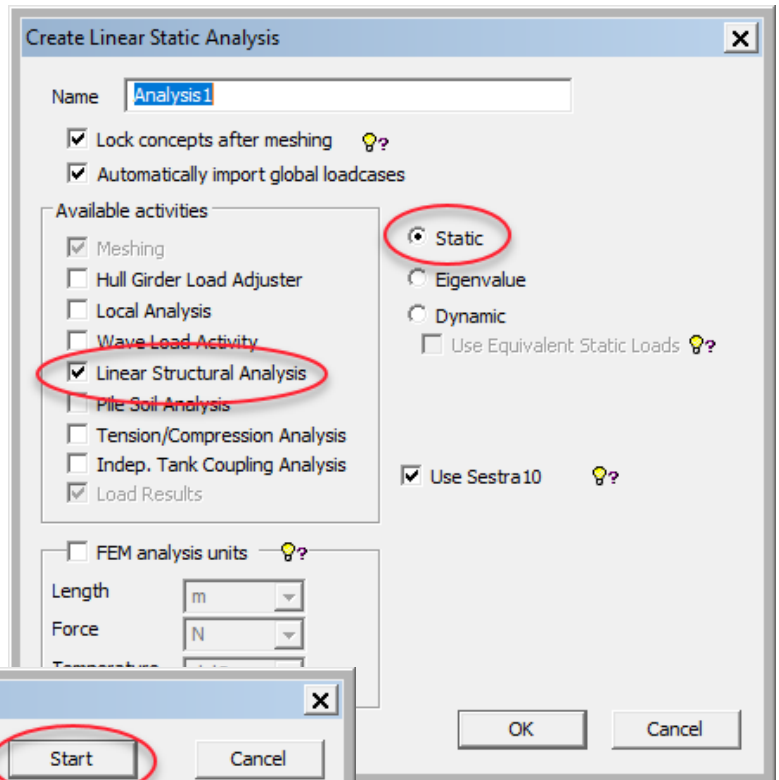
➢ Click *Start* in the *Activity Monitor* to run the analysis.

➢ Make sure the status of all sub-activities are *Success*.

- o If not, right-click the *Linear Structural Analysis* to open the file Sestra.mlg and look for messages.
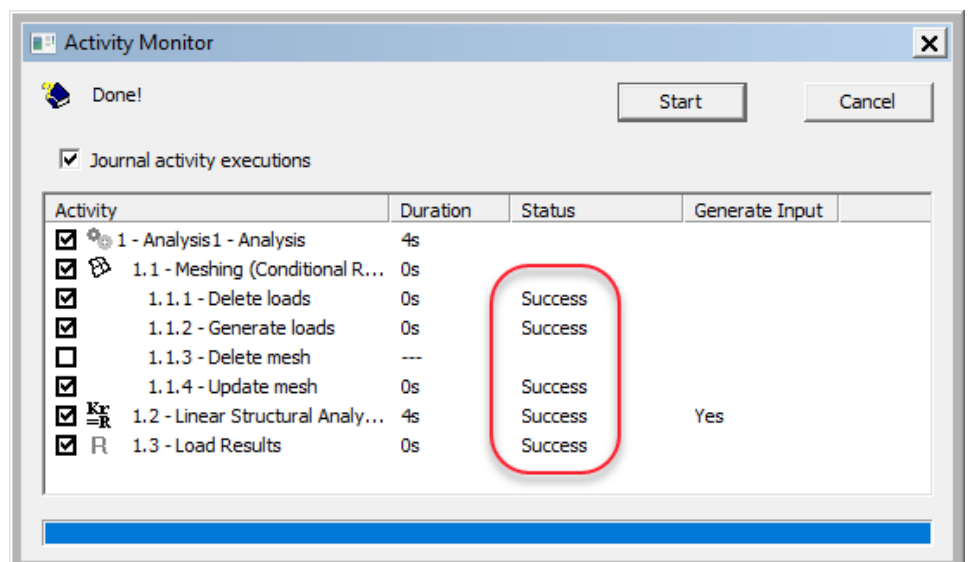
8 PRESENT RESULTS

➢ To present results do as follows:

  o Switch to *Results - with Mesh* display configuration.

  o Switch to *Wireframe* display of beams.
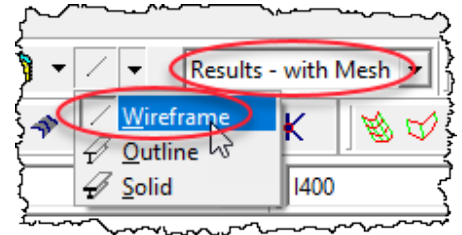
  o Open the *Result presentation* dialog by *Results > Presentation* (or Alt+P).

  o Select a load case and an *Attribute* plus a *Component* to display. Only *Displacements*, *Beam Forces* and *Reaction Forces* are available for this pure beam model.

➢ Deformed shape is shown below. Add display of the undeformed by right-clicking the *Beam selection* button and opening the eye symbol.
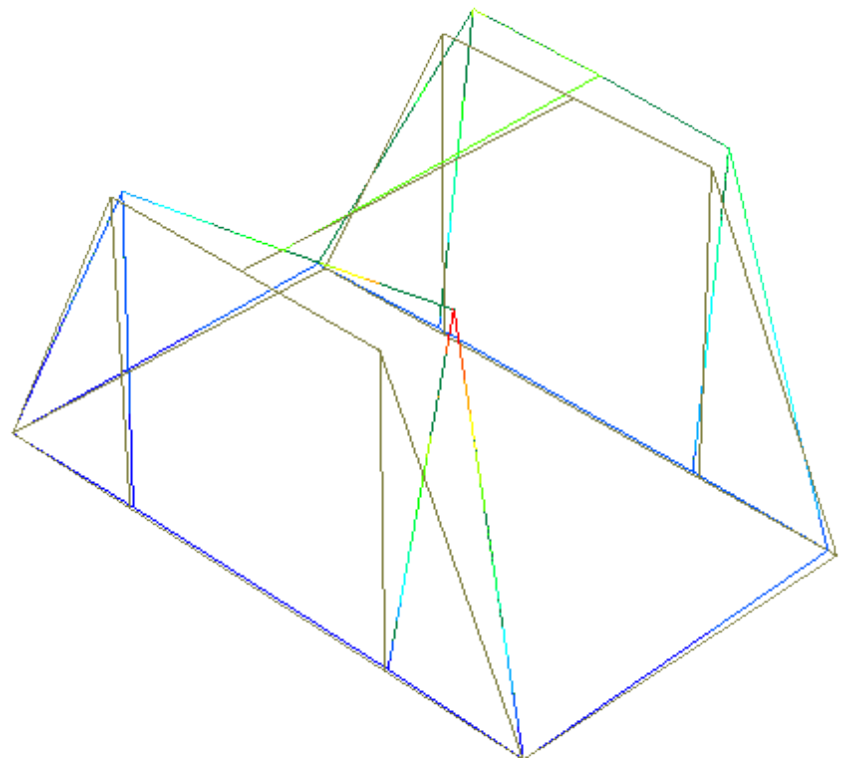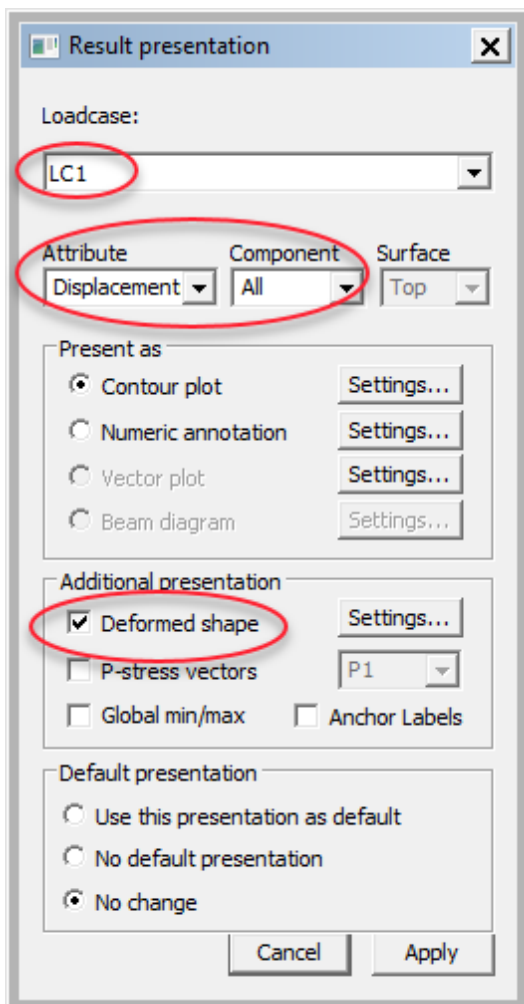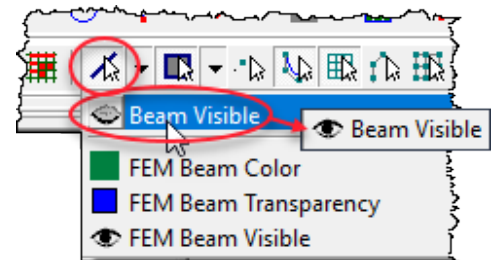
➢ A moment diagram is shown below.

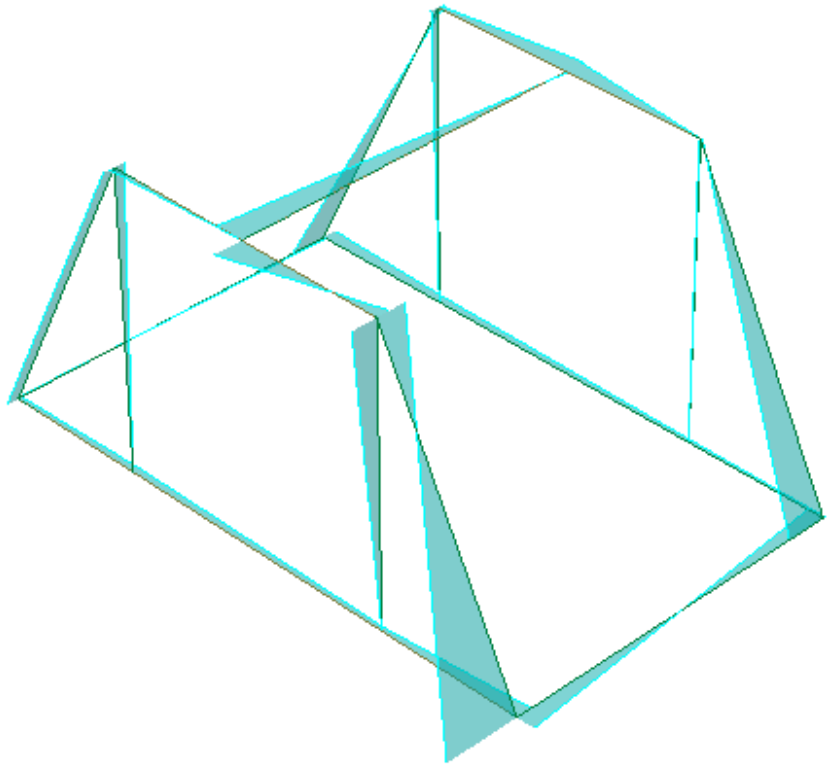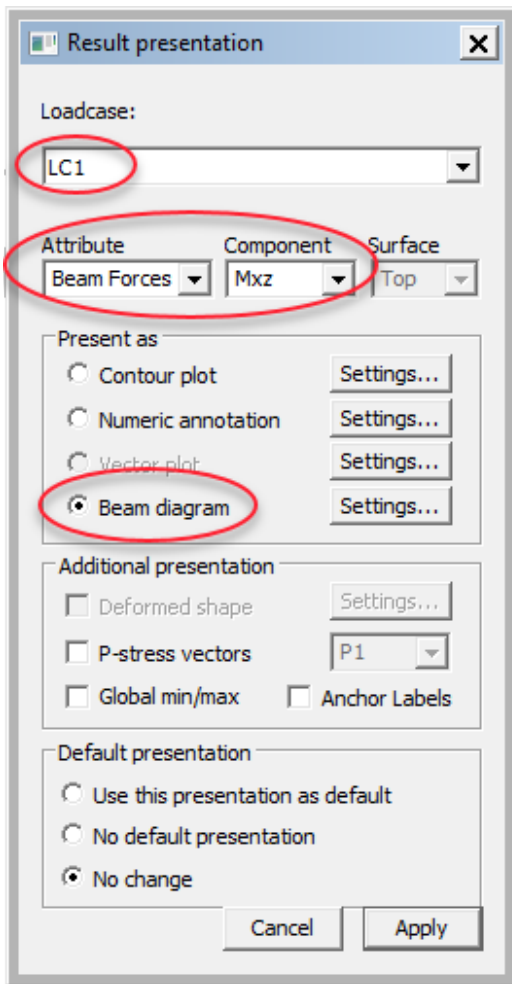➢ To interpret the beam forces the local axis systems of the beams must be known as well as the definitions of beam forces and moments.

o Add beam local axis systems by switching to *Modelling - Transparent* display configuration, selecting all beams, right-clicking and clicking *Labels | Local Coordinate System*.

o Positive moment M_{xy} gives tension on negative local z-axis.

o Positive moment M_{xz} gives tension on negative local y-axis.
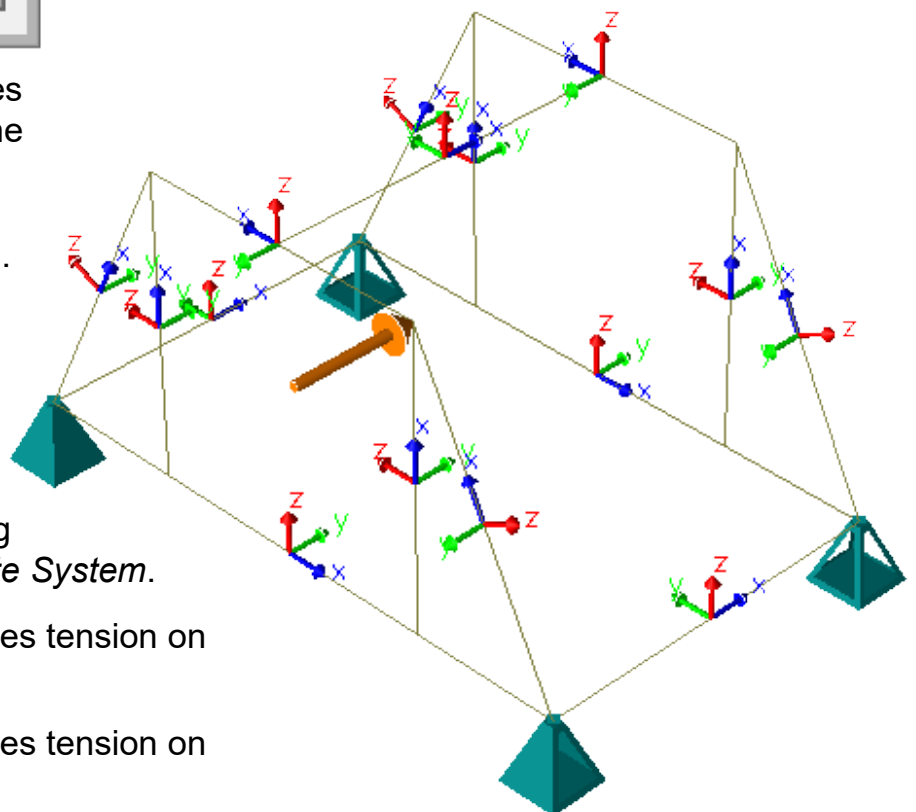
9 VIEW OPTIONS

➢ This chapter explains different ways of working with your model in an efficient way by making sets, labelling and colour coding objects as well as using different view options to filter and select certain parts of your model.
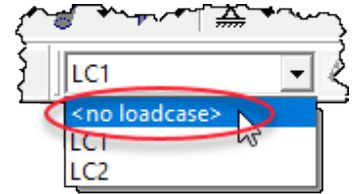
- o Remove labels by selecting labelled objects, right-clicking and selecting *Labels | Clear Labels*.

- o Do not display any load by selecting *<no loadcase>* in the *Select loadcase* field.

➢ Explore options for selecting and displaying parts of the model and creating sets.

- o Switch to *Modelling - Transparent* display configuration, select some members or objects, right-click and select *Visible model | Show selection only* (or Alt+S).

- o Right-click the subset and select *Visible model | Show complement* (or Alt+Q).

- o Displaying a subset, right-click and select *Visible model | Show All* (or Alt+A).

- o Select, right-click and select *Visible model | Remove selection* (or Alt+minus).

- o Learning the keyboard shortcuts Alt+S, Alt+Plus/Minus, Alt+A and Alt+Q makes working in GeniE much more efficient.

- o Try selecting a few members and create a named set by right-clicking and selecting *Named set*. In the *Regular Set* dialog give a name of the set.

- o Sets is a way to group related parts in the model for later selection within GeniE and in subsequent programs.

- o All sets are listed in the browser in the folder *Utilities | Sets | Regular Sets*.
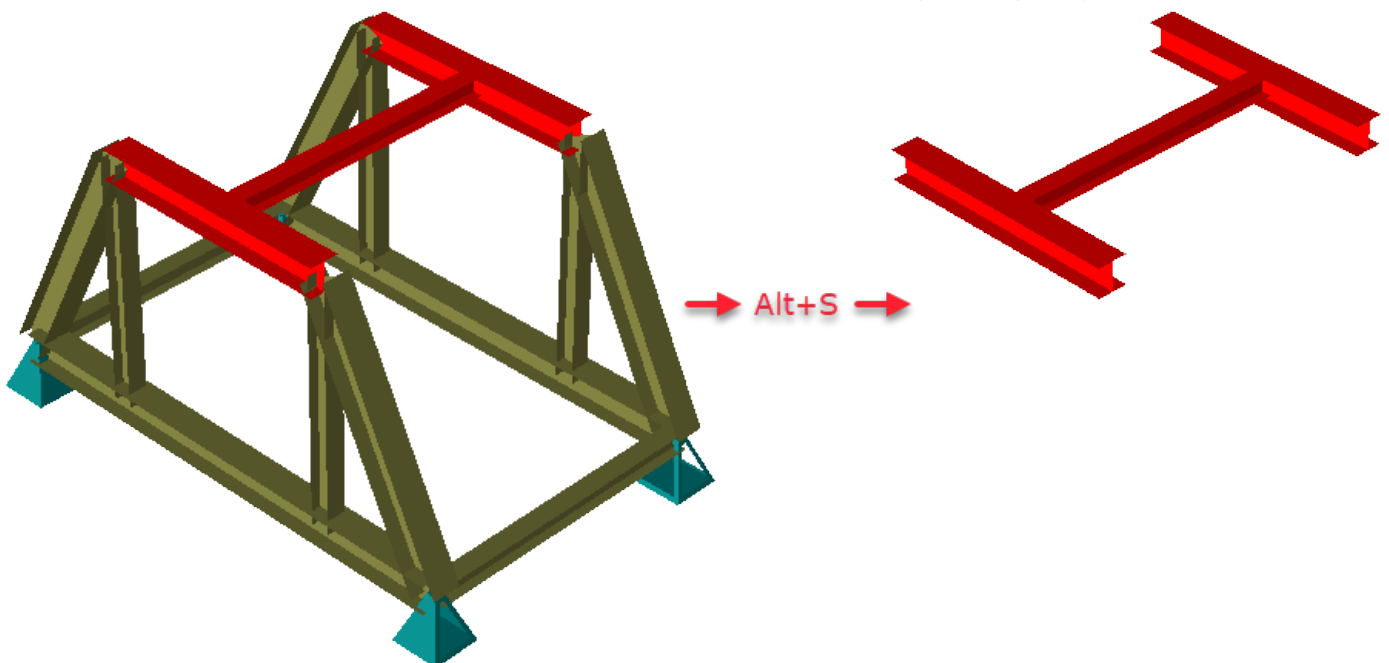
➢ Customize the view and selection options using the selection toolbar shown below. Changes are stored in the registry for the active display configuration. I.e. the changes are persistent after closing and restarting GeniE.
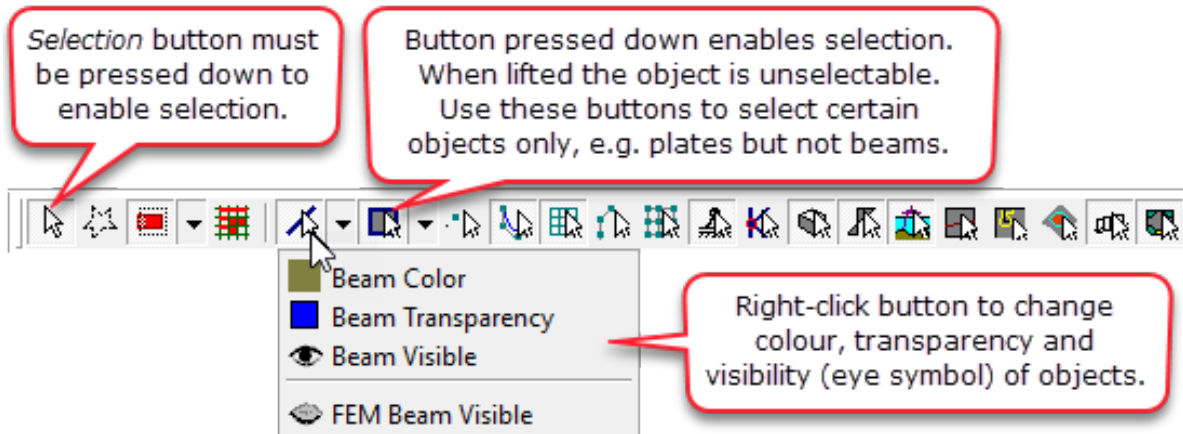

➢ Restore default colours, visibility, etc. for all objects and all display configurations by *View | Options | Settings* (or Alt+O) and clicking *Restore defaults*.
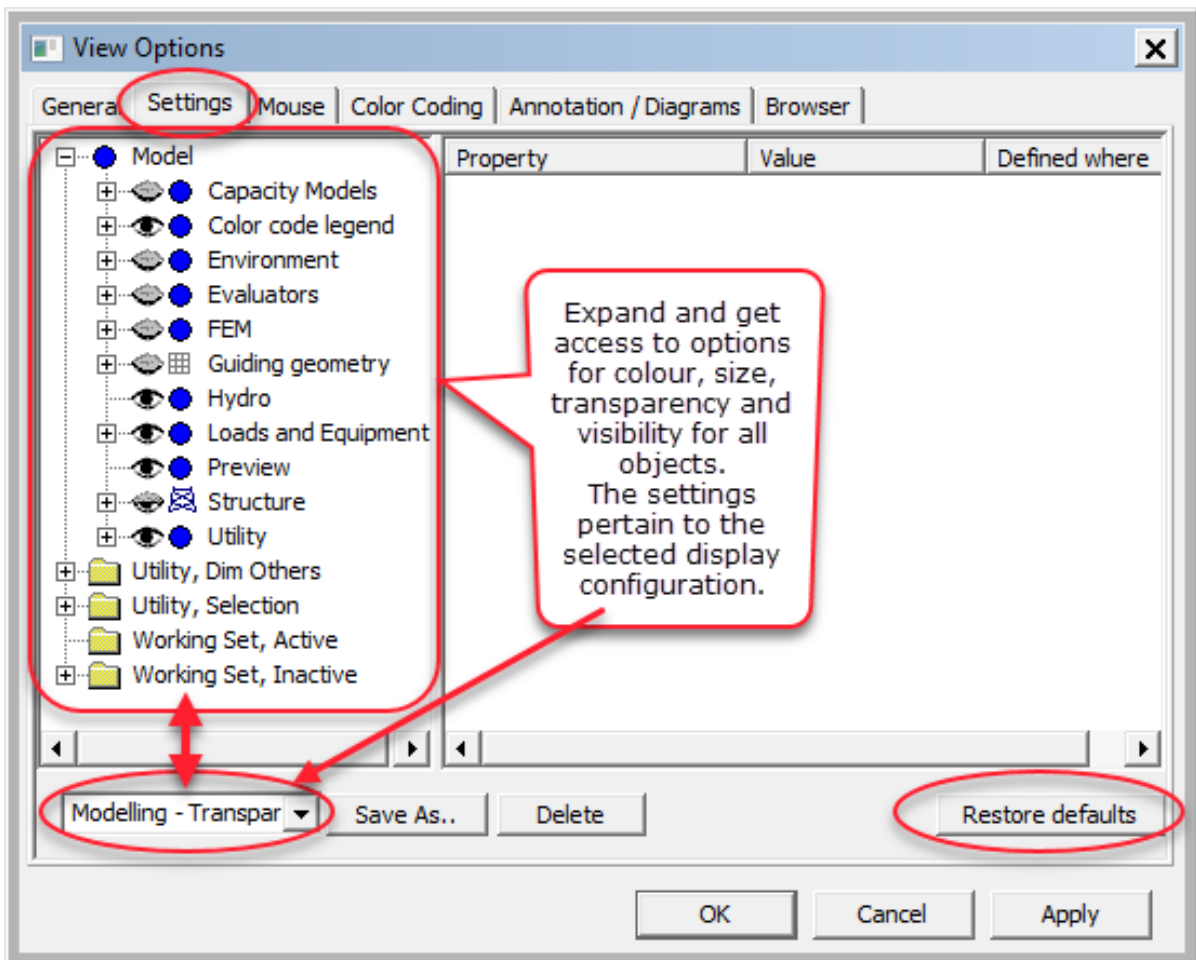
- ➢ Use labels and colour coding for beam sections and names, plate thicknesses, local coordinate (axis) systems, etc. Do so for the whole model or a selected part.

    - o Find options for colour coding properties in the upper right area of the GeniE window.

    - o Select some beams prior to clicking 🖌 to colour code parts as shown to the left below. Do not select anything to colour code the whole mode as shown to the right below.


Section

▮ I400
▮ I600
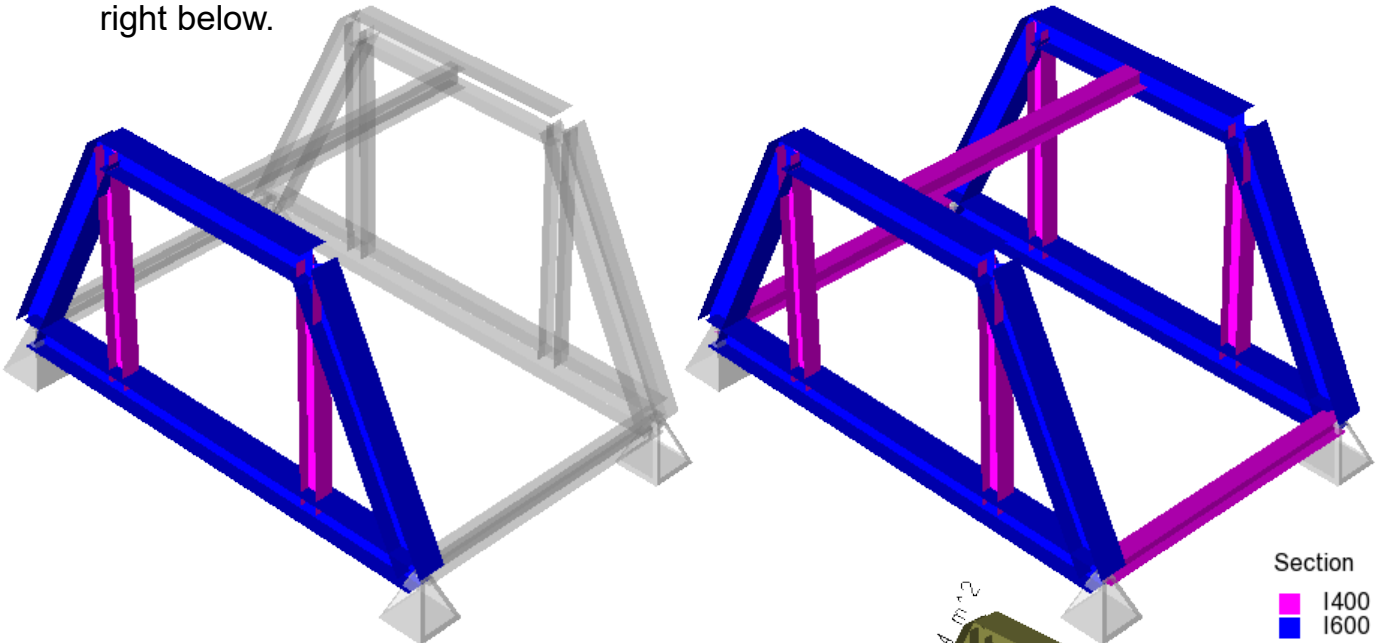
- ➢ In a similar way as above, label the whole or part of the model with property values. Here shown for beam cross section area.
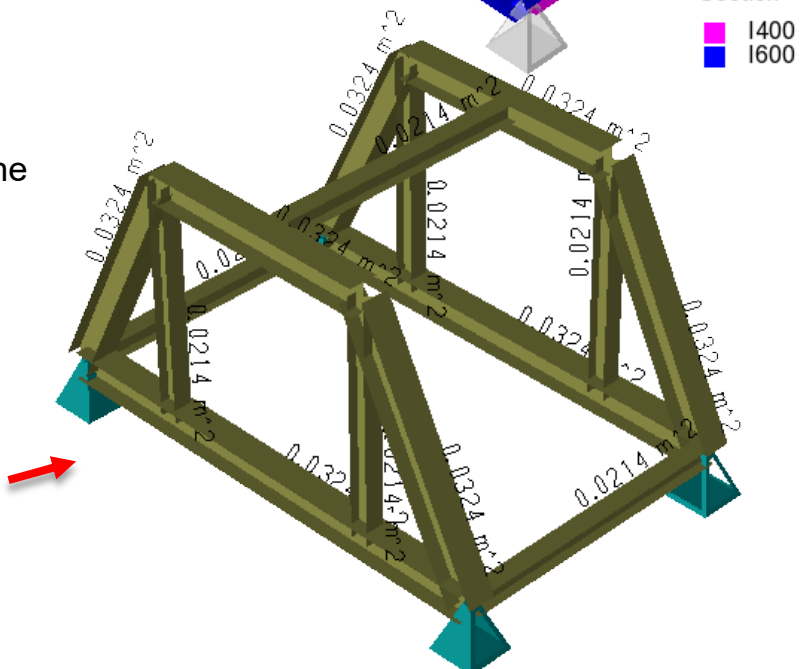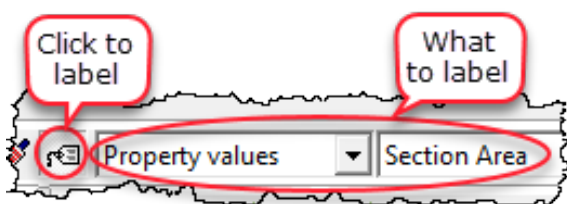
- Colour coding the whole model may also be done by *View | Options | Color Coding* that opens the *Color Coding* tab.

  - This dialog allows more control of the colouring.
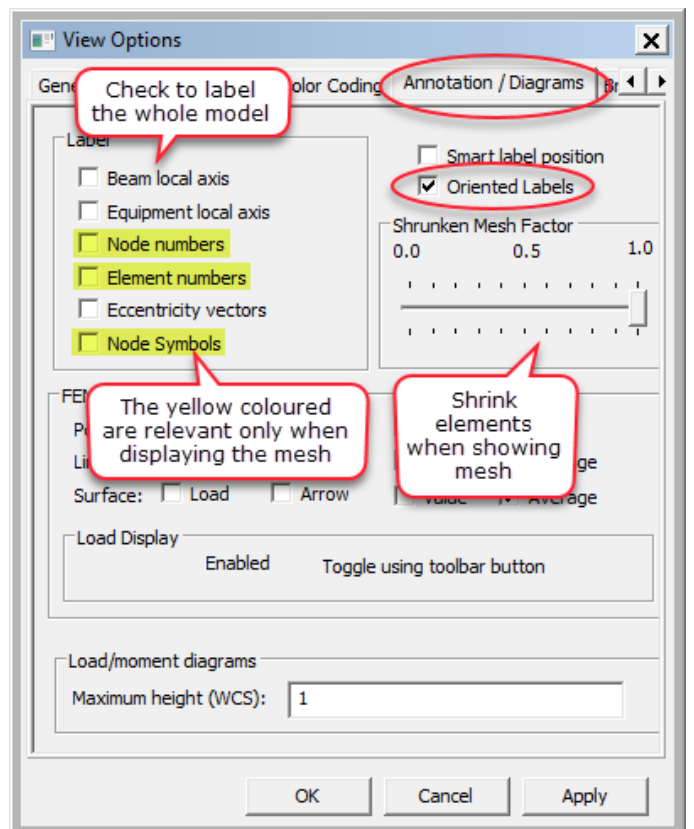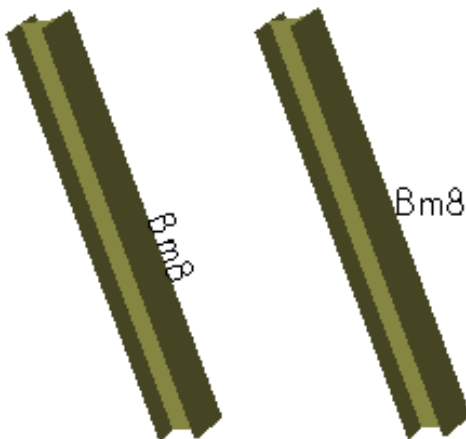
- Moreover, colour coding may be done by right-clicking a folder in the browser and clicking *Color code all visible properties* as shown below for sections.
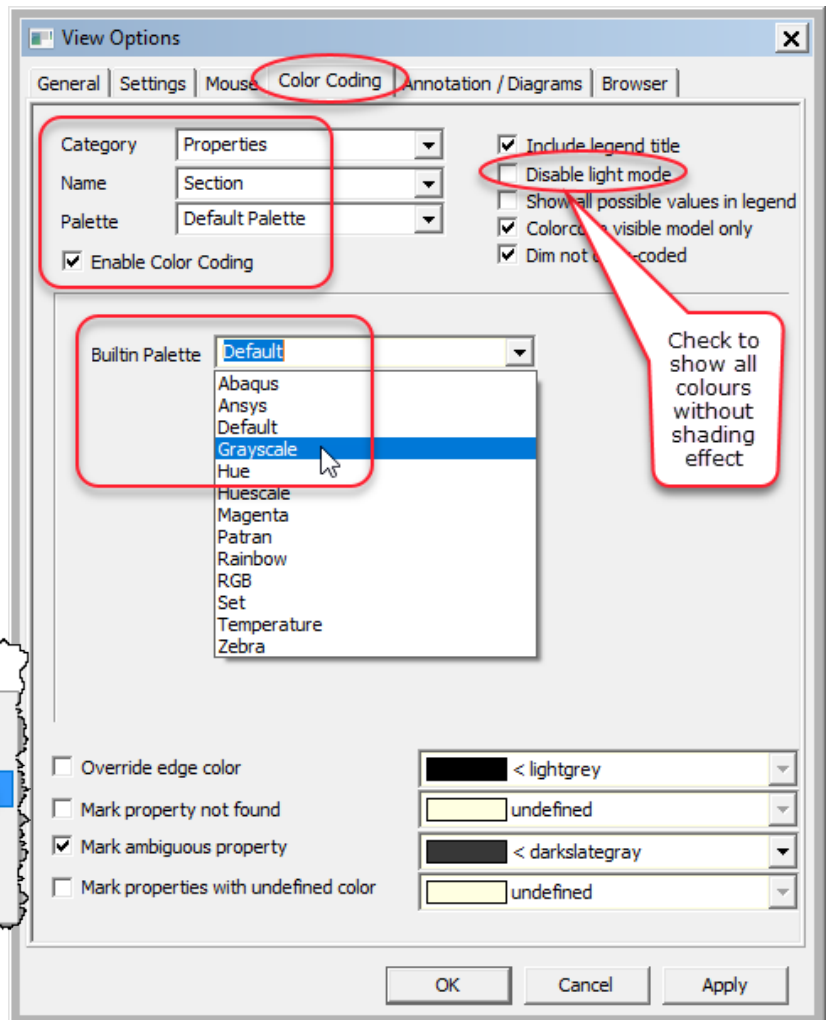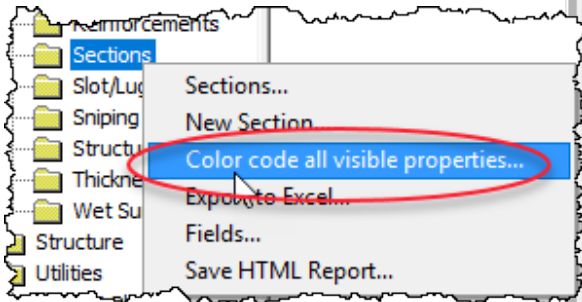
- Labelling the whole model may also be done by *View | Options | Annotation / Diagrams*.

  - Shrinking elements is often useful to see individual elements.

  - The effect of the *Oriented Labels* option is shown below.

➢ Colour coding and labelling parts of the model may also be done by selecting objects, right-clicking and selecting *Labels …* or *ColorCode ….*

    o Deselect the objects (click anywhere) to see the colour coding.

    o Remove labels by selecting labelled objects, right-clicking and selecting *Clear Labels*.

    o Toggle the current colour coding by clicking the paint brush button.

➢ Here is shown an example where a part of the model has been labelled with *Name* (beam names) and colour coded with *Section Area*.

Section Area
   ■ 0.0214 m^2
   ■ 0.0324 m^2
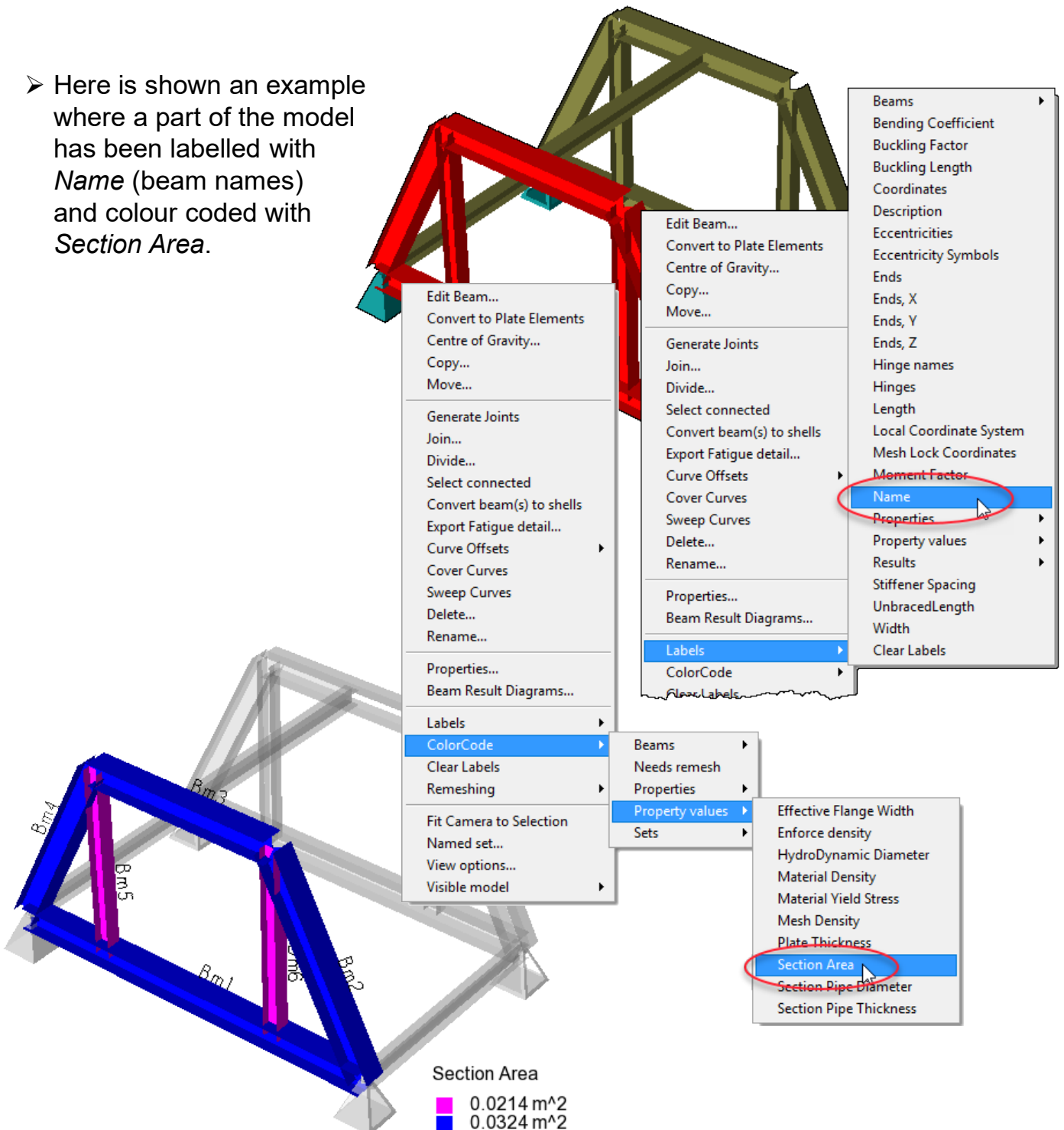
About DNV

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