

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: <a href="mailto:software.support@dnv.com">software.support@dnv.com</a>

E-mail sales: <u>digital@dnv.com</u>

© 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.



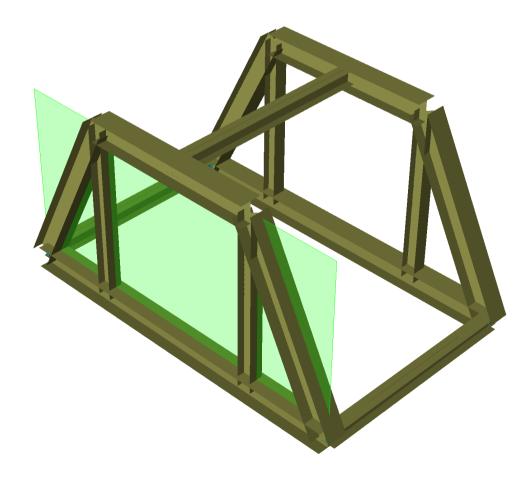
# TABLE OF CONTENTS

1.	Introduction	Page 4
2.	Open Workspace, Define Material and Beam Cross Sections	Page 5
3.	Guiding Geometry	Page 7
4.	Create Beams	Page 8
5.	Create Supports	Page 14
6.	Create Loads	Page 15
7.	Create and Run an Analysis	Page 17
8.	Present Results	Page 18
9.	View Options	Page 20



## **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
  - Load application
  - Support point modelling
  - Static analysis
  - Results presentation
  - 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

New Workspace

- > 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.
  - Accept default Output Units m and N and click OK.
    - Unless otherwise specified, all values in this tutorial are in these units.
  - Check the *Jacket mode* button to limit menus to those relevant for jacket (spaceframe) modelling.
- > Define steel material.
  - Use *Edit* | *Properties* to open the *Properties* dialog.
  - o In the Material tab click Create/Edit Material.
  - In the *Material* dialog give a material name and a *Yield* value. Accept default values and click OK.

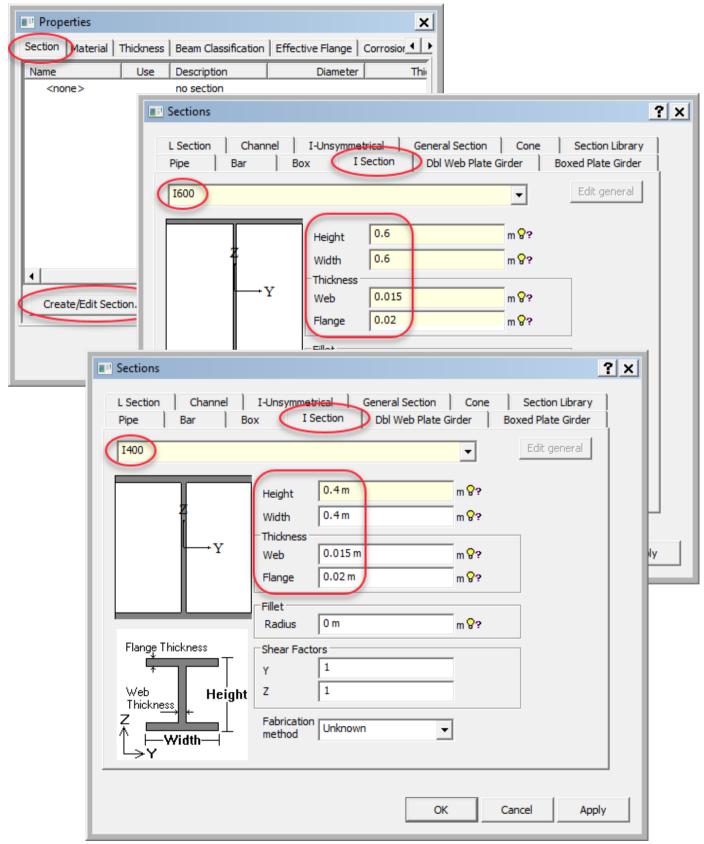
Properties Section Material Thickness Bea	m Classification   Effective Flange   Corrosior	
Name Use De		
<none> no</none>	Material	? ×
	Linear Isotropic Material Shear Isotropic Material Orthotropic Ma	terial
	Yield 2.35E8	Pa
		kg/m^3
	yield Young 2.1e+11 Pa	Pa
	Poisson 0.3	
•	<i>I</i> → <i>ε</i> Thermal 1.2e-05 delC^-1	delC^-1
Create/Edit Material	Damping 0.03 N*s/m	N*s/m
	Tensile	Pa
	OK Cancel	Apply

	Workspace name: Basic_Workshop						
Location:							
C:/DNV/Works	paces/GenE/Basic_Workshop						
Create direc	tory for workspace 💡?	🗖 Store wc					
■	Jacket mode Override default units Length Force N Temperature delC Connected copy on by Use Dual Assembly	✓ ✓ ✓ ✓					
	ОК	Apply					

2 1



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 1600 and 1400 with data as shown below.





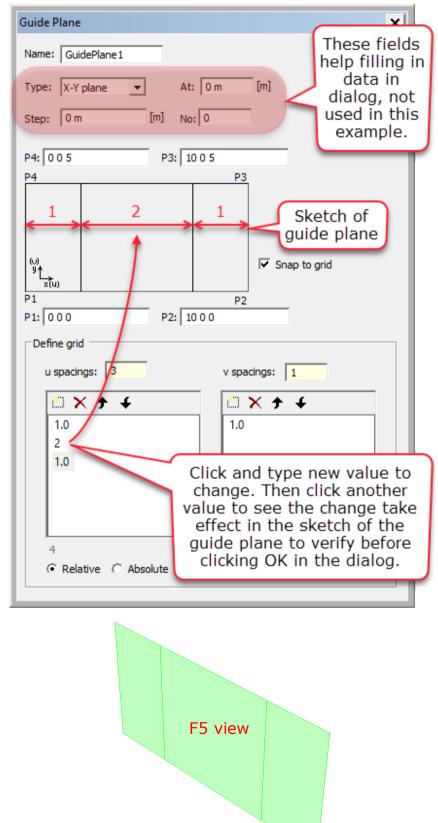
## **3 GUIDING GEOMETRY**

- 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.





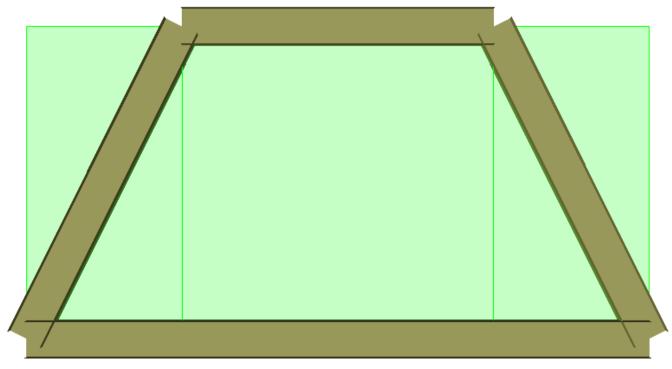


#### 4 CREATE BEAMS

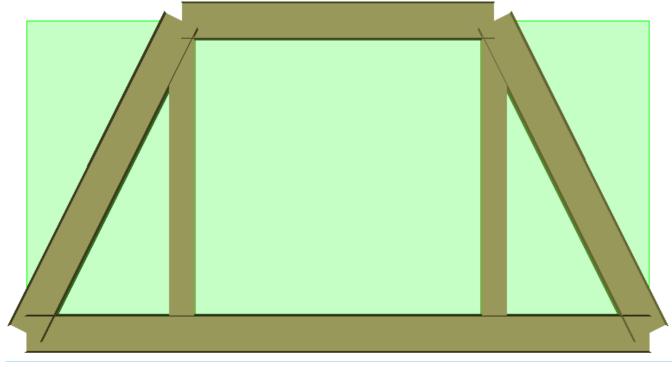
Prior to creating beams set proper default beam cross section (I600) and material (Steel) as shown to the right.

1 martine	<u>~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~</u>	. <b>3</b> .21	Mar 1	┚┖╍╼ॽ
1600	)	Steel	$\mathbf{>}$	<ul> <li><n<sup>3</n<sup></li> </ul>
hand				~~~~~

- Start inserting beams by Structure | Beams and Piles | Straight Beam, or click the Beam button ( / ) in the toolbar.
  - $\circ\,$  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 90° about their axes.
- ➤ To select the beams the Selection button 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.
  - 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*.

Edit Beams	
Local System       Beam offset       Beam end sniping       Hinges       Split Points       Move I       Image: Split Points       Image: Split Points       Image: Split Points       Move I       Image: Split Points       Imag	Edit Beam Convert to Plate Elements Centre of Gravity Copy Move
OK Cancel Apply	

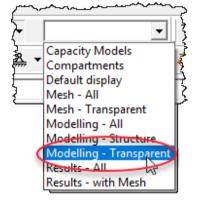


See that the beams are listed in the Structure folder in the browser to the left in the GeniE window.

					¢
⊡ Basic_Workshop	Name	Description	Section	Needs remesh	
🗄 💼 Analysis	/ Bm1	Straight Beam	1600	Yes	
Capacity	/ Bm2	Straight Beam	1600	Yes	
🕂 💼 Environment	/ Bm3	Straight Beam	1600	Yes	
🗄 💼 Equipment	/ Bm4	Straight Beam	1600	Yes	
+ Properties	/ Bm5	Straight Beam	1400	Yes	
Structure	Bm6	Straight Beam	I400	Yes	
Connections	Connections	Folder			
L	$\downarrow_{\overline{a_{a}}}$	~~~~~	~	~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	Д

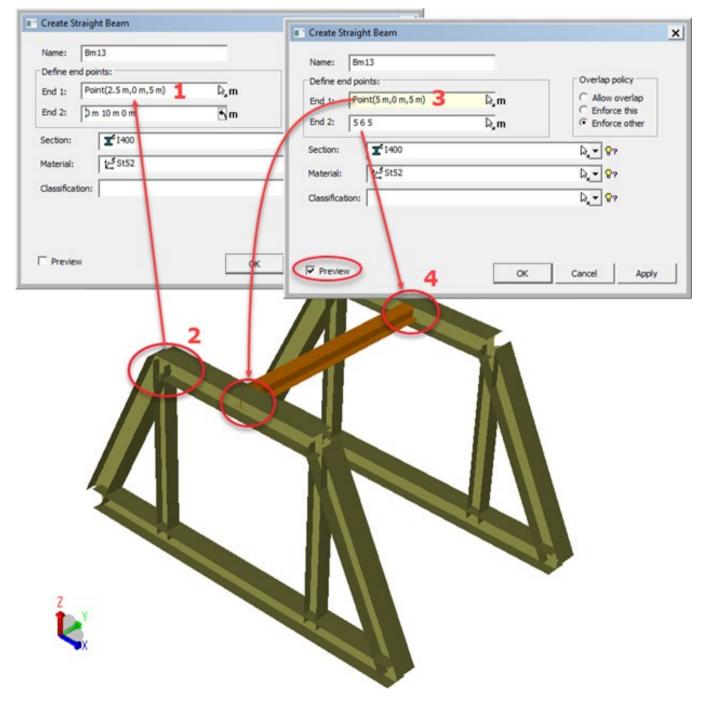
- Change display configuration to *Modelling Structure* or *Modelling - Transparent* (i.e. don't show guide plane).
- Select all beams by dragging a rubberband. Then right-click to open the *Copy* dialog.
  - Make a copy of the frame using the *Translation Vector* (0,6,0) as shown below (copy in Y-direction).
  - Check *Preview* to see a preview of the copy.
  - Click Apply to perform the copying. Then click Cancel as no more copying is desired.

Translate Rotate Mirror Scale 3 Translation Vector	S Point Position   General	
Copy 1 time(s)		
Connected Street Preview	Cancel Apply	



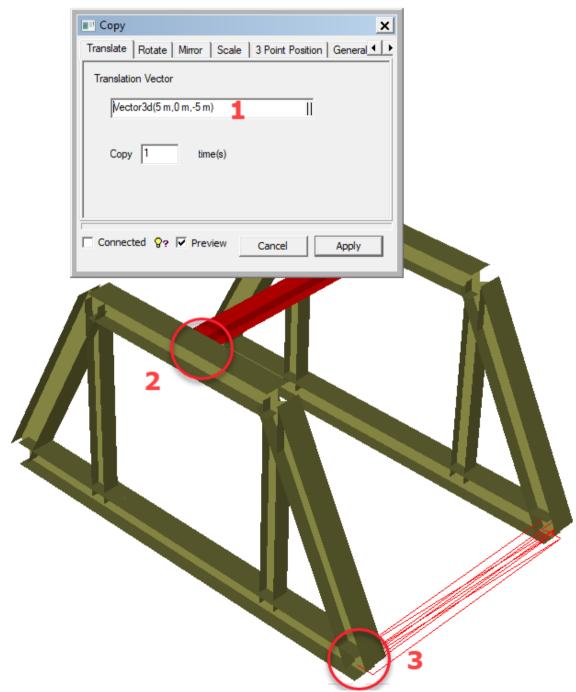


- Insert a beam using a dialog: Structure > Beams and Piles > Straight Beam Dialog. Use the existing geometry to specify appropriate end coordinates:
  - 1. Click in the *End* 1 field (you don't have to delete the data there).
  - 2. Click the end of one of the two upper horizontal beams and see that 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.



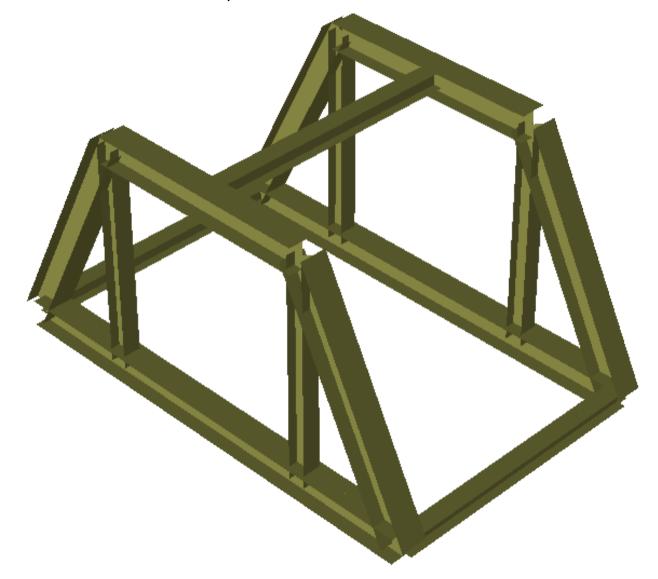


- Select the new beam, right-click and select Copy. Specify the Translation Vector of the copy process as follows:
  - 1. Click in the *Translation Vector* field.
  - 2. Click an end of the beam to copy.
  - 3. Click the point to where the new beam should be copied and see that the appropriate vector 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**

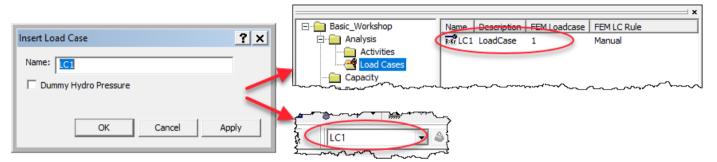
- ➤ Add supports by Structure > Support > Support Point (or click A) in the lower four corners as shown. By default, all six degrees of freedom (dofs) will be fixed.
- Select the supports one-by-one, right-click and select *Properties*. In the *Support* tab adjust the boundary conditions so that all rotations are free and the three translations are fixed or free as indicated by the figure below.

	Properties X
	Object Properties Support Jocal System
	Name: Sp1
	Position : Point(0 m,0 m,0 m)
	The three
	Boundary Condition translational atrix C Boundary Stiffness Per Length     dofs are fixed
The three translationa	
dofs	I let x char y and z Fixed Free Prescribed Dependent Super Spring stiffness
	0 N/m [N/m]
	y 0 N/m [N/m]
The three rotational	0 N/m [N/m]
dofs	Let rx change ry and rz Fixed Free Prescribed Dependent Super Spring stiffness
	ry [N*m]
	[72] [0 N*m [N*m]
	The three rotational
	dofs are free OK Cancel Apply
	The Z translational dof is fixed The Z translational dof is fixed The Y and Z
	rslational translational dofs are fixed



#### 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.
  - The position of the load is specified by clicking in the *Position* field followed by clicking in the model.
  - Give a force of 1000 N in Y-direction.
  - To see the load in the display select *Modelling Transparent* display configuration. *Modelling - Structure* display configuration will by default not display loads.

Point Load	? ×
Name: PLoad1	
Point footprint p1 © Beam point footprint	
Position: Point(7.5 m,0 m,5 m) Beam:	
Forces and Moments  Local Coordinate System LocalSystem(Vector3d(1 m,0 m,0 m), Vector3	
Fx: 0 N N 8? Mx: 0 N*m N*m 8?	
Fy: 1000 N N 8? My: 0 N*m N*m 8?	
Fz: 0 N N ?? Mz: 0 N*m N*m ??	
OK Cancel A	vlqc



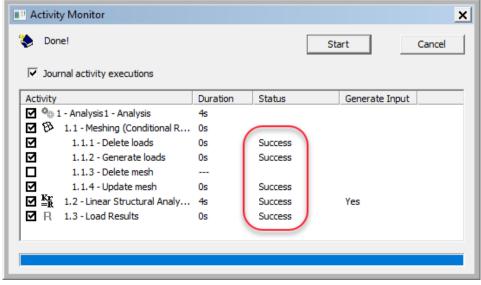
- Create a new load case named LC2. See that this appears as the new current load case.
- Fill LC2 with a line load by Loads > Explicit Load > Line Load. Enter data as shown below.
  - Select points *p1* and *p2* by clicking the model (the insertion point jumps from *p1* to *p2*).
  - $_{\odot}\,$  Give a line load of 500 N/m in Y-direction.

	Create Line Load	? ×
	Name: LLoad1 p1: Point(2.5 m,0 m,5 m) p2: Point(7.5 m,0 m,5 m)	
	<pre> p1 p2 Linear varying load fx1: 0 N/m fy1: 500 N/m [N/m] fy2: 500 N/m fz1: 0 N/m [N/m] fz2: 0 N/m</pre>	[N/m] [N/m] [N/m]
*	Local coordinate system LocalSystem(Vector3d(1 m,0 r	Apply



# 7 CREATE AND RUN AN ANALYSIS

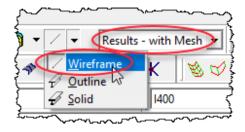
- Create an analysis activity by Create Linear Static Analysis x Mesh & Analysis | Activity Monitor Name Analysis 1 (or Alt+D). Lock concepts after meshing 02 Automatically import global loadcases • By default a Static Linear Available activities Structural Analysis will be done. Static Meshina Hull Girder Load Adjuster Eigenvalue Accept this and click OK. Local Analysis C Dynamic Wave Load Activity 🔲 Use Equivalent Static Loads 💡 > The Activity Monitor opens. Linear Structural Analysis See that the analysis consists of: Plie Soil Analysis Tension/Compression Analysis ○ 1.1 - Meshing Indep. Tank Coupling Analysis ▼ Use Sestra 10 02 Load Results o 1.2 - Linear Structural Analysis FEM analysis units 8? Length 1.3 - Load Results (loading m Force N analysis results into GeniE) Activity Monitor × OK Cancel ۲ Start Cancel ✓ Journal activity executions Activity Duration Status Generate Input 🗹 🌯 1 - Analysis 1 - Analysis Not Started 0s P 1.1 - Meshing (Conditional R... 0s Not Started  $\checkmark$ 1.1.1 - Delete loads 0s Not Started  $\checkmark$ 1.1.2 - Generate loads Not Started 0s 1.1.3 - Delete mesh Not Started 0s 1.1.4 - Update mesh  $\checkmark$ 0s Not Started ☑ <sup>K</sup><sub>**r**</sub> 1.2 - Linear Structural Analy... 0s Not Started Yes R 1.3 - Load Results Not Started 0s
- Click Start in the Activity Monitor to run the analysis.
- Make sure the status of all sub-activities are Success.
  - If not, right-click the Linear Structural Analysis to open the file Sestra.mlg and look for messages.





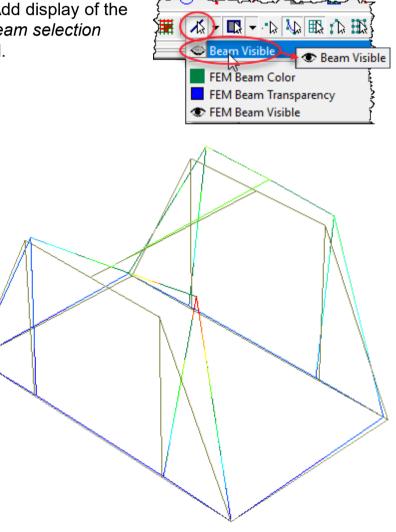
#### 8 PRESENT RESULTS

- > To present results do as follows:
  - Switch to *Results with Mesh* display configuration.
  - Switch to Wireframe display of beams.



- Open the *Result presentation* dialog by *Results > Presentation* (or Alt+P).
- 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.

	Result presentation	×				
	Loadcase:					
0	LC1	•				
	Attribute Componen Displacement  All	Surface				
	Present as					
	Contour plot	Settings				
	C Numeric annotation	Settings				
	C Vector plot	Settings				
	C Beam diagram	Settings				
Li	Additional presentation					
0	Deformed shape	Settings				
1	P-stress vectors	P1 -				
	🔲 Global min/max 📃 🖉	Anchor Labels				
	Default presentation					
	$\ensuremath{\mathbb{C}}$ Use this presentation as	default				
	$\ensuremath{\mathbb{C}}$ No default presentation					
	• No change					
Ľ	Cancel	Apply				





> A moment diagram is shown below.

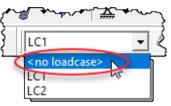
Result presentation	×	
Loadcase:		
LC1	•	
Attribute Componen Beam Forces 🖵 Mxz	t Surface	
Present as		
C Contour plot	Settings	
O Numeric annotation	Settings	
O Vector plot	Settings	
• Beam diagram	Settings	
Additional presentation		
Deformed shape	Settings	
P-stress vectors	P1 🔻	
🗌 Global min/max 🔲 /	Anchor Labels	
Default presentation		
C Use this presentation as	default	
C No default presentation		
• No change		
Cancel	Apply	

- > 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.
  - Add beam local axis systems by switching to Modelling - Transparent display configuration, selecting all beams, right-clicking and clicking Labels | Local Coordinate System.
  - $_{\odot}\,$  Positive moment  $M_{xy}$  gives tension on negative local z-axis.
  - $_{\odot}\,$  Positive moment  $M_{xz}$  gives tension on negative local y-axis.

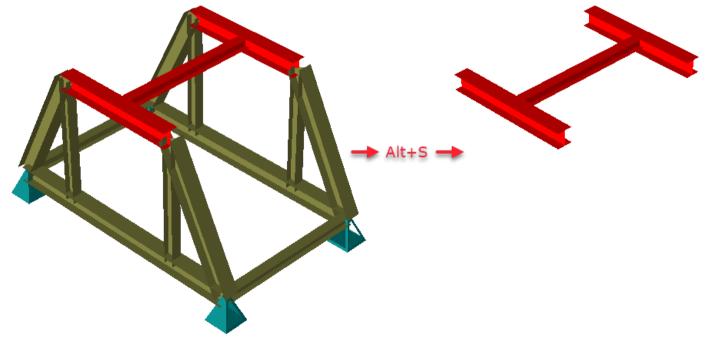


#### **9 VIEW OPTIONS**

- 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.
  - Remove labels by selecting labelled objects, right-clicking and selecting *Labels* | *Clear Labels*.
  - Do not display any load by selecting <*no loadcase*> in the Select loadcase field.

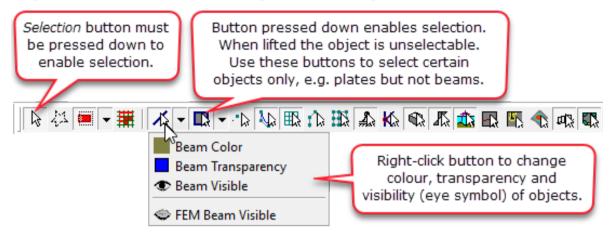


- > Explore options for selecting and displaying parts of the model and creating sets.
  - Switch to *Modelling Transparent* display configuration, select some members or objects, right-click and select *Visible model* | *Show selection only* (or Alt+S).
  - Right-click the subset and select *Visible model* | *Show complement* (or Alt+Q).
  - Displaying a subset, right-click and select *Visible model* | *Show All* (or Alt+A).
  - Select, right-click and select *Visible model* | *Remove selection* (or Alt+minus).
  - Learning the keyboard shortcuts Alt+S, Alt+Plus/Minus, Alt+A and Alt+Q makes working in GeniE much more efficient.
  - 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.
  - Sets is a way to group related parts in the model for later selection within GeniE and in subsequent programs.
  - 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.

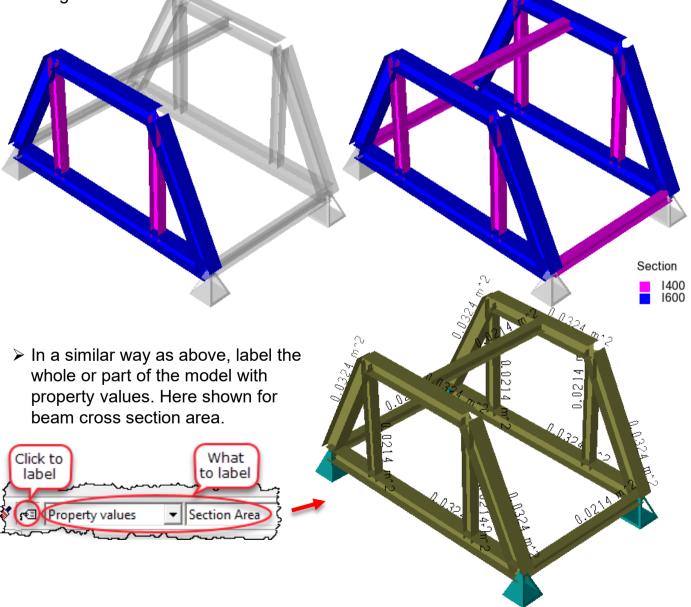
View Options			×
General Settings Mouse Color Co	oding Annotation / Diagrams	Browser	
<ul> <li>Model</li> <li>Capacity Models</li> <li>Color code legend</li> <li>Environment</li> <li>Evaluators</li> <li>FEM</li> <li>Guiding geometry</li> <li>Hydro</li> <li>Loads and Equipment</li> <li>Preview</li> <li>Vility, Dim Others</li> <li>Utility, Selection</li> <li>Working Set, Active</li> <li>Working Set, Inactive</li> <li>Modelling - Transpar Save As</li> </ul>	objects. The settings pertain to the selected displa configuration	ons e, ind II s e ay b.	Defined where estore defaults
	ОК	Cancel	Apply



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



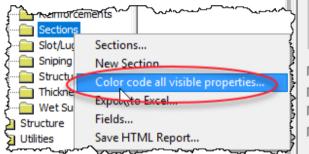
 Select some beams prior to clicking stocolour code parts as shown to the left below. Do not select anything to colour code the whole mode as shown to the right below.



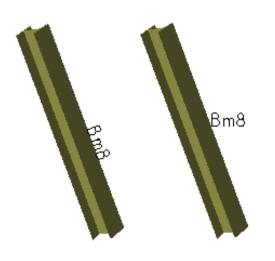


Colour coding the whole model may also be done by View | Options | Color Coding that opens the Color Coding tab. Ge

- 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.



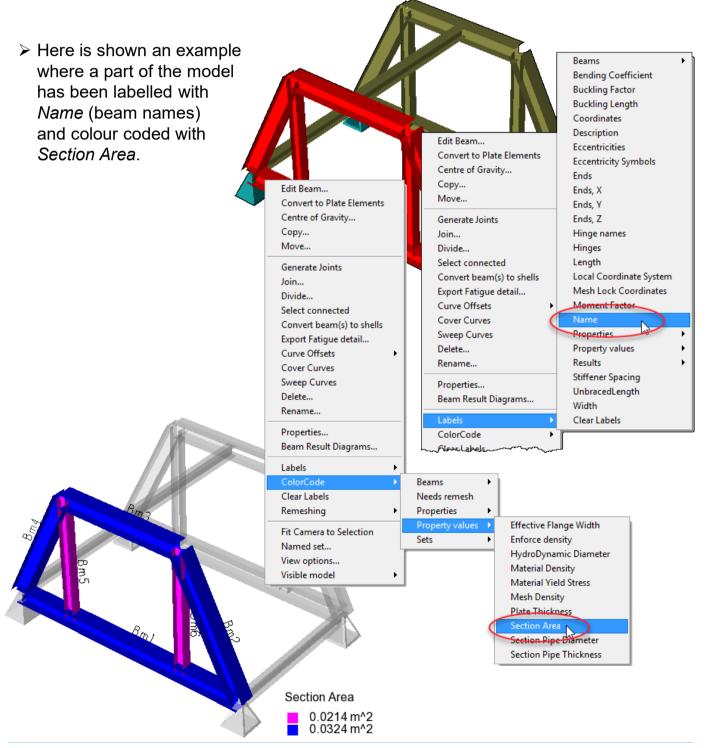
- Labelling the whole model may also be done by View | Options | Annotation / Diagrams
  - Shrinking elements is often useful to see individual elements.
  - The effect of the *Oriented Labels* option is shown below.



View Option	IS			×
eneral Setting	gs Mouse Color C	Coding Annotat	ation / Diagrams   Browser	
Category Name Palette IV Enable Co	Properties Section Default Palette lor Coding	<b>•</b>	<ul> <li>✓ Include legend title</li> <li>Disable light mode</li> <li>Show all possible values in le</li> <li>✓ Colorce visible model only</li> <li>✓ Dim not coded</li> </ul>	gend
		_		_
Builtin Pak	ette Default Abaqus Ansys Default Grayscale Hue Huescale Magenta Patran Rainbow RGB Set Temperature Zebra		Check to show all colours without shading effect	
Override ed	dae color		< lightgrey	Ţ
Mark prope				뒥
✓ Mark ambig	juous property		< darkslategray	Ī
Mark prope	rties with undefined	color	undefined	-
		ОК	Cancel Apply	
ms.	View Options			×
	Uiew Options		Coding Annotation / Diagrams	
ms.	Gen Check to the whole Laver Beam local Equipment Node numb Element nu Eccentricity Node Symb	axis local axis pers mbers y vectors ools low coloured ant only whe ing the mesh oad Arr Enabled	Coding Annotation / Diagrams Smart label position Shrunken Mesh Factor 0.0 0.5 Shrunken Mesh Factor Shrunken Mesh Shrunken Mesh Factor Shrunken Mesh Shrunken Mesh Shrunke	



- 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.
  - Remove labels by selecting labelled objects, right-clicking and selecting *Clear Labels*.
  - $\circ$  Toggle the current colour coding by clicking the paint brush button  $\boxed{\checkmark}$ .





#### **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