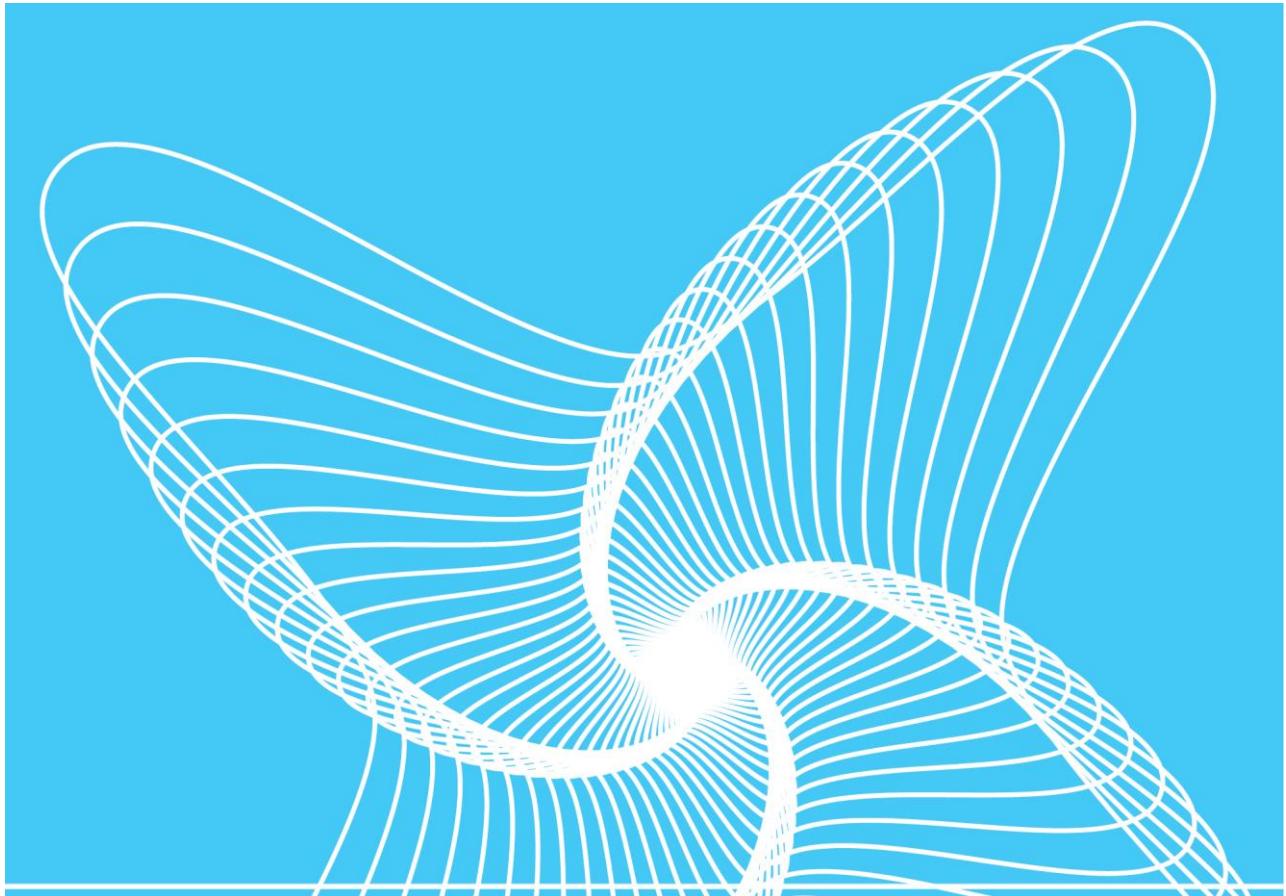


SESAM USER MANUAL

# GeniE

Vol 1 - Concept design and analysis of offshore structures

Valid from program version V7.3



Sesam User Manual

GeniE

Date: 26 Feb 2016

Valid from GeniE version V7.3

Prepared by DNV GL - Software

E-mail support: [software.support@dnvgl.com](mailto:software.support@dnvgl.com)

E-mail sales: [software@dnvgl.com](mailto:software@dnvgl.com)

© DNV GL 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 GL AS.

# GeniE User Manual

## Table of Contents

<b>1. INTRODUCTION.....</b>	<b>6</b>
1.1 GENIE - AN INTEGRATED DESIGN SOLUTION .....	6
1.2 GENIE IN THE SESAM SYSTEM.....	8
1.3 STATUS LISTS .....	8
1.4 ON-LINE HELP SYSTEM AND TUTORIALS .....	8
1.5 HOW TO READ THIS MANUAL.....	10
1.6 ACRONYMS FREQUENTLY USED IN THE MANUAL .....	10
<b>2. FEATURES OF GENIE .....</b>	<b>11</b>
2.1 THE FUNDAMENTAL PRINCIPLES BEHIND GENIE.....	11
2.2 CONCEPT MODELLING .....	12
2.3 A TYPICAL WORKFLOW WHEN USING GENIE .....	12
2.4 SETTING THE DESIGN PREMISE.....	13
2.5 BASIC STRUCTURE MODELLING .....	16
2.6 ADVANCED STRUCTURE AND PANEL MODELLING .....	16
2.7 EQUIPMENT AND LOAD MODELLING.....	18
2.7.1 <i>Equipment modelling</i> .....	18
2.7.2 <i>Weight lists</i> .....	19
2.7.3 <i>Acceleration loads</i> .....	20
2.7.4 <i>Explicit loads</i> .....	20
2.7.5 <i>Loadcases and load combinations</i> .....	21
2.8 PREPARE FOR STRUCTURAL ANALYSIS .....	21
2.9 RUN ANALYSIS AND LOOK AT RESULTS .....	23
2.10 RUN EIGENVALUE ANALYSIS .....	24
2.11 RUN HYDRODYNAMIC ANALYSIS .....	25
2.12 RUN PILE AND SOIL ANALYSIS .....	26
2.13 TENSION/COMPRESSION ANALYSIS .....	27
2.14 CODE CHECKING AND ADVANCED RESULTS PROCESSING .....	30
2.15 RE-ANALYSIS .....	31
2.16 HULL GIRDER LOAD ADJUSTER .....	31
2.16.1 <i>How to run HGLA</i> .....	31
2.16.2 <i>Running the analysis</i> .....	32
2.16.3 <i>What happens when the analysis is run</i> .....	33
2.16.4 <i>Checking the output from HGLA</i> .....	33
2.17 DOCUMENTING THE MODEL.....	34
2.18 JOURNALLING AND READING COMMANDS, WIZARDS.....	35
2.19 UNIT HANDLING AND SECTION LIBRARIES .....	36
2.20 TRANSFER OF A MODEL THROUGH THE INPUT AND RESULTS INTERFACE FILE FORMAT .....	37
2.21 IMPORT AND EXPORT FROM OTHER SYSTEMS.....	37
2.22 PANEL MODELLING .....	37
<b>3. USER'S GUIDE TO GENIE .....</b>	<b>39</b>
3.1 GETTING STARTED .....	39
3.1.1 <i>The Graphical User Interface (GUI)</i> .....	39
3.1.2 <i>Starting up</i> .....	39
3.1.3 <i>Setting up your preferences</i> .....	40
3.1.4 <i>View manipulations (LMB &amp; RMB)</i> .....	45
3.1.5 <i>Selecting objects</i> .....	46
3.1.6 <i>Graphic modelling and editing</i> .....	50
3.1.7 <i>Browser modelling and editing</i> .....	52
3.1.8 <i>The command line interface and journalling</i> .....	53
3.1.9 <i>Defaults and naming</i> .....	54
3.1.10 <i>Using the clipboards</i> .....	54
3.1.11 <i>Making a named set</i> .....	55
3.1.12 <i>Making a dynamic set</i> .....	56

3.1.13	<i>Making a report</i> .....	63
3.1.14	<i>Saving a plot</i> .....	81
3.1.15	<i>Documenting mass and centre of gravity</i> .....	82
3.1.16	<i>Labelling</i> .....	83
3.1.17	<i>Show length and angle</i> .....	84
3.1.18	<i>Working with units</i> .....	84
3.1.19	<i>Switch between wireframe, outline, solid view</i> .....	85
3.1.20	<i>Customise the tool buttons</i> .....	86
3.1.21	<i>Work with visible model</i> .....	86
3.1.22	<i>Cut planes</i> .....	89
3.2	THE DESIGN PREMISE .....	95
3.2.1	<i>Rules for beam creation</i> .....	95
3.2.2	<i>Rules for transformation</i> .....	96
3.2.3	<i>Rules for joint creation</i> .....	96
3.2.4	<i>Rules for joint design</i> .....	97
3.2.5	<i>Rules for tolerances</i> .....	98
3.2.6	<i>Rules for connected move</i> .....	98
3.2.7	<i>Rules for units</i> .....	99
3.2.8	<i>Rules for meshing</i> .....	99
3.2.9	<i>Rules for scripting of sets</i> .....	102
3.2.10	<i>Property libraries</i> .....	103
3.2.11	<i>Compatibility</i> .....	104
3.3	GUIDING GEOMETRY .....	105
3.3.1	<i>Guide points</i> .....	106
3.3.2	<i>Guide lines</i> .....	108
3.3.3	<i>Guiding poly-lines</i> .....	112
3.3.4	<i>Guiding conic sections</i> .....	114
3.3.5	<i>Guiding free-form curves (splines)</i> .....	116
3.3.6	<i>Split/Join Curves</i> .....	125
3.3.7	<i>Guiding curves on surfaces</i> .....	127
3.3.8	<i>Guiding planes</i> .....	128
3.3.9	<i>Transformations</i> .....	133
3.3.10	<i>Advanced</i> .....	135
3.3.11	<i>Find, select and display guiding geometry</i> .....	150
3.3.12	<i>Move and Copy guiding geometry entities</i> .....	152
3.3.13	<i>Offsetting a guiding curve</i> .....	159
3.3.14	<i>Converting a guiding curve from its old-format to NURBS</i> .....	161
3.4	STRUCTURE .....	162
3.4.1	<i>Beams and Piles</i> .....	163
3.4.2	<i>Flat Plates</i> .....	167
3.4.3	<i>Shells with circular sections</i> .....	168
3.4.4	<i>Free Form Shells</i> .....	170
3.4.5	<i>Shells from Point-Nets</i> .....	172
3.4.6	<i>Hull Design Tools</i> .....	175
3.4.7	<i>Holes</i> .....	191
3.4.8	<i>Reference documents on Computer-Aided Geometric Design:</i> .....	195
3.5	LOADS, NEW MENU .....	196
3.6	BASIC STRUCTURE MODELLING OF BEAMS .....	198
3.6.1	<i>Regular Straight Beam</i> .....	198
3.6.2	<i>Overlapping beams</i> .....	200
3.6.3	<i>Modelling perpendicular beams</i> .....	202
3.6.4	<i>Snap planes</i> .....	202
3.6.5	<i>Split, join and divide beams</i> .....	203
3.6.6	<i>Insert split points</i> .....	205
3.6.7	<i>Beam local co-ordinate system</i> .....	205
3.6.8	<i>Delete, move and copy beams</i> .....	206
3.6.9	<i>Change beam properties</i> .....	208
3.6.10	<i>Beam eccentricities</i> .....	209

3.6.11	<i>Connecting beams to an eccentric beam end</i> .....	211
3.6.12	<i>More about the Edit Beam dialog</i> .....	213
3.7	BASIC PLATE MODELLING.....	216
3.7.1	<i>Regular plate in a plane</i> .....	216
3.7.2	<i>Overlapping plates</i> .....	218
3.7.3	<i>Split and join plates</i> .....	218
3.7.4	<i>Plate local co-ordinate system</i> .....	219
3.7.5	<i>Delete, move and copy plates</i> .....	220
3.7.6	<i>Change plate properties</i> .....	220
3.7.7	<i>Stiffened panels</i> .....	221
3.8	ADVANCED STRUCTURE MODELLING .....	222
3.8.1	<i>Segmented beam modelling</i> .....	222
3.8.2	<i>Tubular joint modelling</i> .....	225
3.8.3	<i>Hinges</i> .....	234
3.8.4	<i>Local joint flexibility</i> .....	235
3.8.5	<i>Non-structural beams, shims, and truss elements</i> .....	237
3.8.6	<i>Reference point modelling of beam structures</i> .....	242
	<i>Curved structure</i> .....	243
3.8.7	<i>Release X-joints</i> .....	251
3.8.8	<i>Point-point connection</i> .....	252
3.8.9	<i>Modelling a pile inside a leg - ungrouted case</i> .....	253
3.8.10	<i>Modelling a pile inside a leg – grouted case</i> .....	258
3.8.11	<i>Importing and exporting a model</i> .....	259
3.9	LOADCASES AND LOAD COMBINATIONS .....	261
3.9.1	<i>Some principles for mass and load calculations</i> .....	261
3.9.2	<i>Making basic loadcases and account for selfweight</i> .....	263
3.9.3	<i>Making load combinations</i> .....	265
3.9.4	<i>Making a load interface</i> .....	266
3.9.5	<i>Controlling the loadcase numbering</i> .....	267
3.9.6	<i>Copy a loadcase</i> .....	269
3.9.7	<i>Result cases</i> .....	269
3.9.8	<i>Converting point and line load to point masses</i> .....	270
3.10	EQUIPMENT MODELLING .....	272
3.10.1	<i>Creating equipments</i> .....	272
3.10.2	<i>Editing the cog and footprint</i> .....	272
3.10.3	<i>Placing the equipment</i> .....	274
3.10.4	<i>Creating forces from placed equipments</i> .....	277
3.10.5	<i>Verifying the applied loads</i> .....	278
3.11	WEIGHT LISTS.....	279
3.11.1	<i>Format of weight list data</i> .....	280
3.11.2	<i>Import weight list data</i> .....	281
3.11.3	<i>Creating and placing equipments from weight lists</i> .....	281
3.11.4	<i>Calculating the loads</i> .....	283
3.11.5	<i>Verifying the calculated forces</i> .....	284
3.12	EXPLICIT LOADS .....	285
3.12.1	<i>Point loads</i> .....	285
3.12.2	<i>Line loads</i> .....	287
3.12.3	<i>Pressure loads</i> .....	288
3.12.4	<i>Temperature load</i> .....	290
3.12.5	<i>Prescribed displacements</i> .....	290
3.12.6	<i>Acceleration and rotational fields</i> .....	292
3.13	LOAD ANNOTATIONS .....	297
3.13.1	<i>Displaying Load Annotations</i> .....	297
3.13.2	<i>Annotation Layout and the Refresh Graphics button</i> .....	299
3.13.3	<i>Annotation Settings</i> .....	300
3.13.4	<i>Performance options</i> .....	302
3.13.5	<i>Annotation Slices</i> .....	304
3.14	MAKING A FINITE ELEMENT MODEL .....	306
3.14.1	<i>Boundary conditions</i> .....	306
3.14.2	<i>Create and control a mesh</i> .....	310

3.14.3	<i>Element types</i> .....	321
3.14.4	<i>Controlling the FEM Loadcase numbers</i> .....	322
3.14.5	<i>Mesh only parts of model</i> .....	323
3.15	RUN STRUCTURAL ANALYSIS.....	327
3.15.1	<i>The Meshing option of the Activity Monitor</i> .....	329
3.15.2	<i>About the different meshing steps</i> .....	330
3.15.3	<i>Modify the analysis parameters</i> .....	331
3.15.4	<i>Smart load combinations</i> .....	332
3.15.5	<i>Continue on element shape error</i> .....	333
3.16	RESULTS .....	334
3.16.1	<i>View preferences</i> .....	334
3.16.2	<i>Displacements</i> .....	334
3.16.3	<i>Element General Stresses</i> .....	336
3.16.4	<i>Beam forces</i> .....	337
3.16.5	<i>Principal stresses</i> .....	339
3.16.6	<i>Surface load</i> .....	339
3.16.7	<i>Other results presentation</i> .....	340
3.17	RUN CODE CHECKING USING FRAMEWORK .....	341
3.18	CREATE MASS MODELS .....	348
3.18.1	<i>Mass model for hydrodynamics</i> .....	349
3.18.2	<i>Mass model for structural dynamics</i> .....	350
3.18.3	<i>Mass model when neglecting eccentricities</i> .....	351
3.18.4	<i>Point mass versus node mass</i> .....	352
3.18.5	<i>Scale structural mass</i> .....	353
3.19	RUN EIGENVALUE ANALYSIS .....	355
3.20	RUN WAVE LOAD ANALYSIS .....	357
3.21	CHANGE AND RUN ANALYSIS AGAIN .....	358
3.22	PANEL MODELLING .....	358
3.23	WIND LOADS .....	364
3.23.1	<i>Wind loads on equipment</i> .....	364
3.23.2	<i>Wind loads on plates</i> .....	368
	<i>Wind loads – general</i> .....	370
4.	EXECUTION OF GENIE .....	371
4.1	PROGRAM ENVIRONMENT .....	371
4.2	FILES USED BY GENIE .....	374
4.3	PROGRAM LIMITATIONS .....	375
4.4	PROGRAM EXTENSIONS .....	375
4.4.1	<i>The GeniE Lite licence</i> .....	375
4.4.2	<i>The GeniE RCLite licence</i> .....	376
4.5	MESSAGES PRODUCED BY GENIE .....	377
4.6	SOFTWARE AND HARDWARE RECOMMENDATIONS .....	377
4.7	DATABASE VERSION CHECKING .....	378
4.8	BACKUPS.....	379
4.8.1	<i>Default settings for Backup (only for new workspaces)</i> .....	379
4.8.2	<i>Manage Backups (for the current workspace)</i> .....	380
4.8.3	<i>How to recover a backed up workspace</i> .....	381

<b>5. SHORT DESCRIPTION OF COMMANDS .....</b>	<b>383</b>
5.1    MENUS.....	383
5.1.1 <i>Pull-down menus</i> .....	383
5.1.2 <i>Tool buttons</i> .....	388
5.1.3 <i>Context sensitive menus</i> .....	393
5.2    SHORT COMMANDS AND WINDOWS COMPLIANCE .....	398
5.3    THE COMMAND LINE INTERFACE SYSTEM .....	399
5.3.1 <i>The GeniE JSript Command Reference</i> .....	401
5.3.2 <i>Useful script commands</i> .....	404
<b>6. APPENDIX A: REFERENCES .....</b>	<b>407</b>
<b>7. APPENDIX B: CONSISTENT UNITS .....</b>	<b>408</b>
7.1    CONSISTENT SI UNITS .....	408
7.2    CONSISTENT IMPERIAL UNITS .....	408

# 1. INTRODUCTION

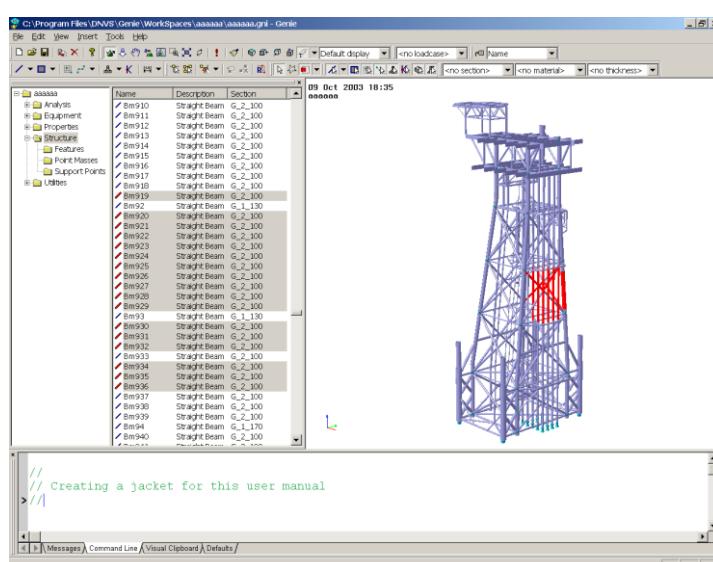
This User Manual aims at guiding new users of GeniE. Emphasis has also been put on advanced features for the more experienced users by separating commands from the textual & graphic description of features.

The User Manual is valid from GeniE version 7.2 – both for the full version of GeniE and the GeniE-lite version. The User Manual is split in six volumes. The program limitations that apply are listed in Section 4.3 Program Limitations.

## 1.1 GeniE - An integrated design solution

GeniE is the new design analysis tool in SESAM and represents the latest generation design and analysis software supporting designers and engineers. The development has been motivated and driven by end-user needs for new solutions offering significantly faster modelling tightly integrated with advanced strength assessment. GeniE supports work phases from initial concept studies to mature design and re-analysis:

- Intuitive user interface and strong features for 3D visualisation of model and results.
- Interactive modelling capabilities relevant for design of topside structures, jackets and similar types of structures. Regular curved structures can also be modelled provided they are plated structures.
- Combined plate and beam modelling (curved as well as planar).
- Intelligent tubular joint design based on user defined rules.
- Easy to interrogate the model by using browser techniques.
- Flexible load application by modelling equipments, their footprints and load transfer rules.
- Flexible handling and conversion of units.
- Openness to CAD systems and other analysis systems, with import of section libraries and existing weight list, as well as support for MS Office applications.
- A powerful journalling system based on the Jscript language.
- Integrated analysis (hydrodynamic, pile-soil and structural) and results processing.



*A Windows based user interface*

GeniE builds on DNV Software's long time experience as a solution provider in the offshore market, as well as recent advances in IT technology. This includes

- ACIS geometry/topology modeller from Spatial Corporation, USA
- AISC Shapes Database v3.0 from AISC Inc., USA
- HOOPS from TechSoft3D licensed from Spatial Corporation, USA
- MFC for the graphical user interface from Microsoft Corp., USA
- Objective Toolkit for grids from Rogue Wave Software, USA
- ObjectStore PSE Pro for data storage in object-oriented databases from eXcelon Corporation, USA

These industry standard technologies are combined with DNV Software's own proven and unique technologies, including

- Finite element mesh generation
- Finite element analysis
- Finite element results visualisation
- Environmental loads calculation
- Code checking and rule based design
- Openness towards leading CAD vendors

GeniE may be used as a stand-alone tool using a direct analysis approach (all modelled in one and same finite element model) where the user can

- Model structure, equipments, environment and other loads
- Calculate hydrodynamic loads and run static structural analyses including non-linear pile soil analysis
- Visualise and postprocess results
- Perform code checking based on recognised standards

GeniE is also ideal for creating superelements (parts) in a superelement analysis. Typically, topsides and modules are created in GeniE and assembled with other parts of the structure like the hull of a FPSO, Semi-submersible, TLP, Spar or similar structure.

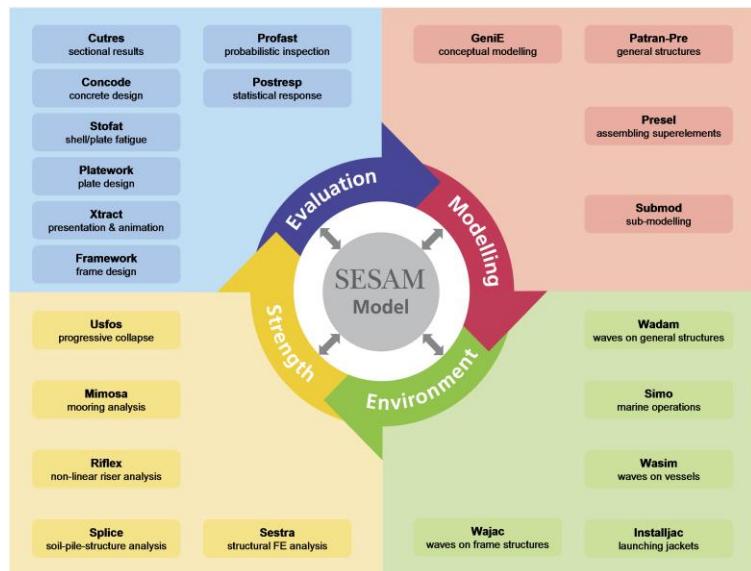
GeniE may also be used to create panel models of fixed or floating structures for subsequent stability or hydrodynamic analysis in HydroD.

GeniE is fully owned, maintained and supported by DNV GL - Software, an independent business unit of DNV GL AS, Norway.

## 1.2 GeniE in the SESAM system

GeniE is related to the rest of the SESAM system through the SESAM Interface File. It may either be used as a stand-alone tool or in a superelement analysis. On its own, it will do all modelling, analysis, and results presentation within the same user interface.

GeniE allows you to do basic results presentation of beam axial, force and moment diagrams. The program also transfers the concept model and results to Framework where the code checking is performed.



GeniE may also be executed and controlled from SESAM Manager or Workflow Manager. Typical examples are when more complex models are to be generated (e.g. a complete jacket or a tension leg platform). The topside may very well be created in GeniE, while the rest is modelled in e.g. Patran-Pre and assembled in Presel. In other words, the FEM file (T-file) is read into Presel and the superelement produced by GeniE becomes part of a superelement hierarchy.

## 1.3 Status lists

In addition to this User Manual you may find additional information in the Status List (as for any other SESAM program). Such information may be:

- Reasons for update (new version)
- New features
- Errors found and corrected

The status list is accessed either from

- Inside GeniE using pulldown menu **Help/Status List**
  - From the help page there is also a section *Release Notes* that explains the new features
- At the following internet address <http://www.dnvsoftware.com> ->Support tab
- From SESAM Manager

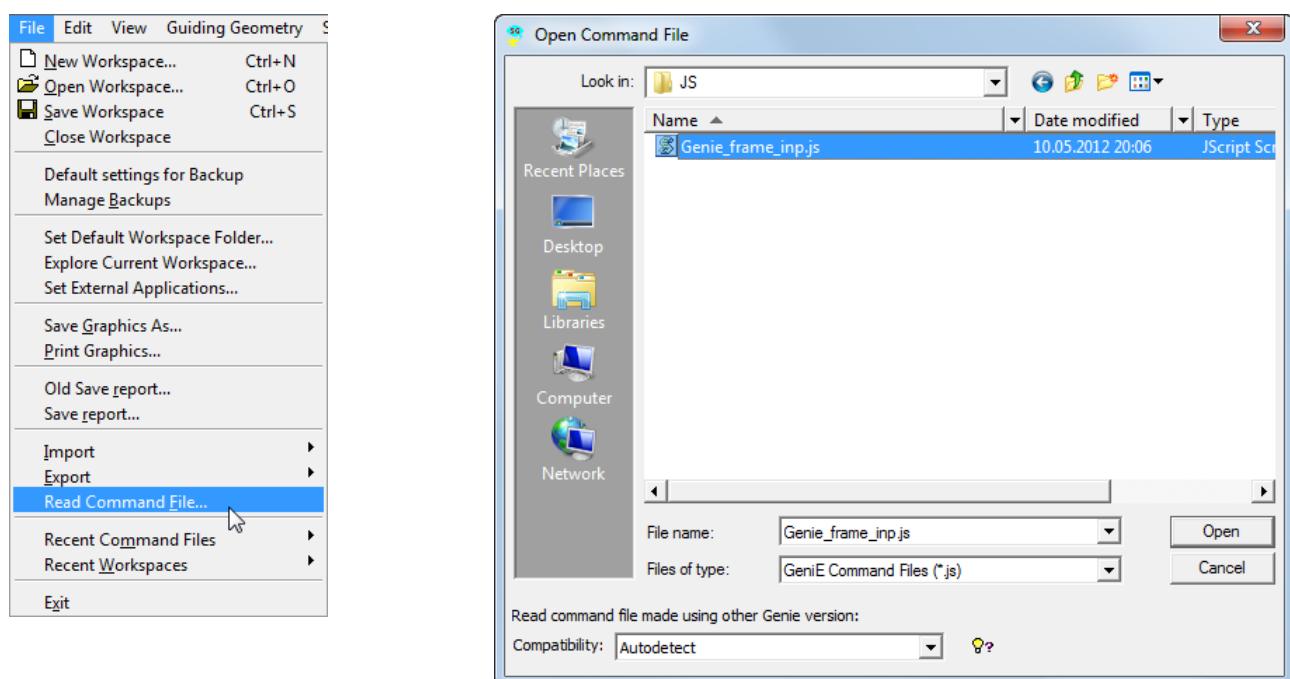
## 1.4 On-line help system and tutorials

GeniE comes with an on-line help system (**Help/Help Topics** or **F1**). Its purpose is to provide easy access to release notes, limitations, tutorials, wizards and this user manual. In addition it contains a detailed documentation of all available commands in the journalling system (based on J-script). There are also videos showing how to do certain operations, these are best viewed using resolution 1280x1024.

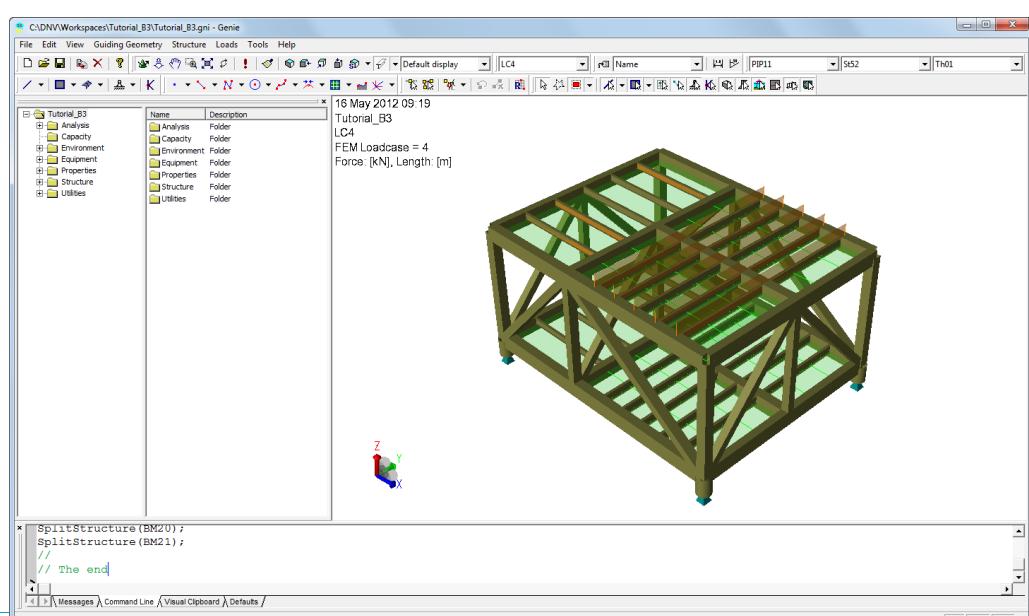
The most efficient way to work with the tutorials is to make a print-out of the tutorials, start GeniE, create a new workspace (command **File/New workspace**) and follow the steps in the tutorial. Each tutorial comes with a pre-defined journal file (command file) – you find these in the folder C:\Program Files\DNVS\GeniE Vxx-xx\Examples (xx-xx is the version number of your installation, for example 6.3-00). If you want to use these files the steps are as follows:

1. Create a new workspace **File/New Workspace/<name>**  
(Keep the default settings for tolerant modelling and database units)
2. Read in the journal file from **File/Read Command File|<browse until you find the desired input file>**
3. Save your model by **File/Save**
4. You have now created the same model as in the tutorial.

In the example below the <name> has been set to *Tutorial\_B3* and the imported file is for Tutorial no. B3 “GeniE Frame Module”.



The sequence above creates the following view in GeniE:



## 1.5 How to read this manual

Read Chapter 2 *Features of GeniE* to get a quick overview of the features of GeniE (what you can do).

Read Section 3.1 *Getting started* for an introduction to the graphical user interface (GUI) and what needs to be done prior to modelling and analysis tasks.

Read the rest of Section 3 to learn how to set up a complete analysis including all modelling, executing it and post-processing the results. This section explains the most common operations you would use when modelling a topside, a module frame, a jacket or relevant curved structures using concept modelling techniques. It also describes how to do hydrodynamic and structural analysis followed by result assessments. A more comprehensive guidance in how to create such models (including journal files) may be found on the on-line help system.

Read Chapter 4 *Execution of GeniE* for information on files associated with GeniE and about alternative ways of running the program. This Chapter also documents high level limitations.

Read Chapter 5 *Command Description* for a detailed documentation of all commands (pull down menus, tool buttons, and context sensitive menus). Observe that the available commands in the journalling system are documented in the on-line help system available from GeniE. They are thus not documented herein.

Read Volume 2 of the User Manual to learn about hydrodynamic analysis of framed structures and non-linear analysis of the pile-soil interaction.

A command from the menu list (also referred to as the pulldown menu) is written like this:

**Insert/Beam/Dialog**. The name of a tool button is written like this: **Basic plate**. A function buttons is referred to like this: **F1**.

GeniE comes with a context sensitive menu. You invoke this menu by pushing your right mouse button when the mouse is located above a selected object. In this manual this operation is termed **RMB**. The commands on the context sensitive menu are written like this: *Join Beams*.

Viewing this manual assumes the usage of Adobe Acrobat Reader version 4.0 or higher. You may use older versions, but then you don't have access to important features like e.g. free text search and bookmarks (table of content + hyperlinks).

It is particularly noted that this User Manual documents all capabilities of GeniE. If you are running the GeniE.lite version or GeniE without the extension Curved geometry, there are several items in this manual you do not have access to in your program. These features are blanked out in your program version.

## 1.6 Acronyms frequently used in the Manual

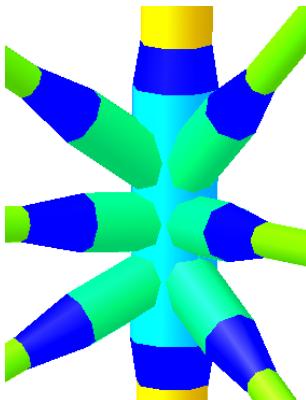
Acronym	Explanation
<b>RMB</b>	Right Mouse Button
<b>LMB</b>	Left Mouse Button
<b>GUI</b>	Graphical User Interface
<b>DOF</b>	Degree Of Freedom
<b>CLI</b>	Command Line Interface
<b>LJF</b>	Local Joint Flexibility
<b>FEM file</b>	SESAM Input Interface File

## 2. FEATURES OF GENIE

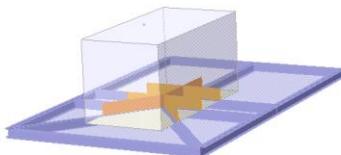
This Chapter gives an overview of the capabilities of GeniE. A more complete description is given in the following Chapters 3, 4 and 5. The tutorials and videos found under **Help/Help Topics** (or F1) also give a thorough explanation of some of the features.

### 2.1 The fundamental principles behind GeniE

The visions of the program GeniE were established in the Joint Industry Project *SESAM 2000* (1994 -2000). The program itself has been developed by DNV Software following this project.



*Built-in knowledge for joint design*



*Built-in knowledge when deriving equipment loads*

One of the most important goals was to make the design analysis process – for topsides and jackets initially - more efficient and based on the engineering terms rather than finite element terms. These goals have been achieved by introducing concept modelling and building engineering knowledge into the program.

Concept modelling allows the user to very rapidly build a model for design purpose while the built-in knowledge ensures quality of the model and the results. It is important that the user can understand and control every step in a design process; hence all logics built into the program can be checked and modified by the user if necessary.

Focus has also been on continued usage of existing data or information. Typical examples are how data from a CAD system or other FE systems can be imported, automatically translated to a concept model and enriched with necessary data to do a design analysis. Equally important is the need for data transfer from GeniE to other SESAM programs – examples are models created in GeniE for a superelement analysis and code checking of plated structures in Platework.

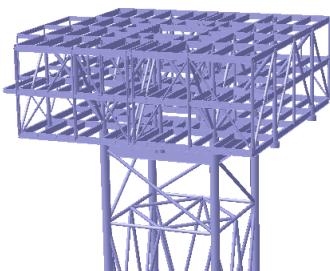
GeniE will let you:

- Model a structure using libraries for sections, plates, material and boundary conditions. The modelling itself may be done directly, using guiding geometry or by importing an existing model.
- Make loadcases where you assemble equipments, weight lists or explicit point/line/surface/temperature loads.
- Generate a finite element model or panel model – the mesh is either automatically generated or controlled by you.
- Run analysis – linear static, hydrodynamic, pile/soil analysis - and do results processing or code checking.
- Efficiently change your model (e.g. mesh density, add loadcases and add structural components), do a re-analysis and check the new results.
- Document every step of your analysis

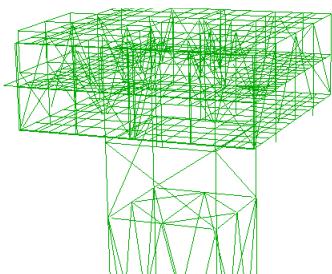
## 2.2 Concept modelling



The structure



The concept model



The FE model

A design vision in GeniE has been the realisation of *concept modelling* techniques. In previous generation design and analysis software the end-goal was to perform structural assessment based on the Finite Element Method (FEM). These solutions solved many problems and were extremely helpful in assessing the status of a given design. However, these software solutions had architectural limitations prohibiting efficient integration with CAD software and rule based capacity check software.

The main problem was a missing vehicle for communication between CAD software, structure analysis software and rule based capacity check software. These different domains had too little in common for efficient information exchange. For example, the CAD system had a detailed geometry/topology model, the analysis system had a FE model while the capacity check software had specialised capacity models. It was very hard or impossible to communicate model changes across these domain boundaries. The consequence was excessive and costly re-modelling within each domain.

Concept modelling provides a means of overcoming many of these issues because the user's design intent is better captured. Instead of representing e.g. deck structures as element, nodes, faces or edges, the user may model the plates and beams explicitly. These new concepts capture the design intent much better because it is now possible to model items such as whole deck plates and segmented beams as single design concepts. It also relieves the user from tedious and unnecessary work. Finally, the model becomes richer as plates with holes, supports, equipments etc. can be modelled explicitly.

The concepts hold information about attributes (for example section profiles and hydrodynamic properties) as well as connectivities to other structural members. When moving a structural part the connectivity (topology) is automatically updated.

The picture to the left shows the concept model and the automatically created analysis model – when a linear static analysis has been performed the concepts hold information about results as well.

## 2.3 A typical workflow when using GeniE

The most common steps when performing a design analysis in GeniE are listed below. Details for each step may be found in the next sections.

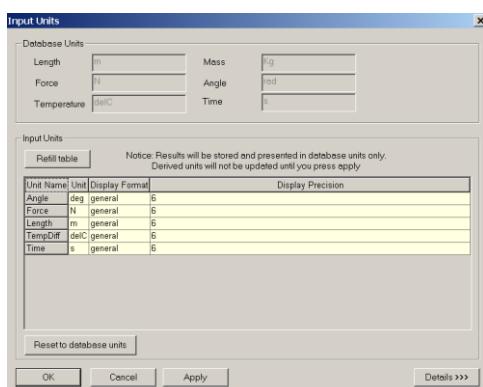
1. Decide the design premise – typically units, section libraries, mesh rules, gap calculations, tolerances and name settings.
2. Model the structural part of model – either from importing from other systems or other SESAM programs, using wizards or modelling from scratch. This part is normally the most demanding task depending on the complexity of the structure to be modelled. The program is intended for many and frequent changes in the structure layout – it is easy to modify the model.
3. Create loadcases – basic loadcases and load combinations. The loadcases are built using loads from explicit point/line/surface/temperature loads, loads from placed equipments or imported weight lists, or acceleration loads (gravity or other). For hydrodynamic analysis using Wajac relevant attributes may be added to the model also.
4. Perform the analysis – a finite element model is automatically created using either default mesh settings or those specified by you. When of many changes has been made to the model tools are

available to ensure a clean topology in which unnecessary points are removed (hence fewer finite element nodes). The user may also perform a hydrodynamic analysis and include the wave loads in the structural analysis.

5. Investigate the result – either from looking at results directly inside GeniE or do code checking in Framework. The concept model is brought over to Framework so that you can use the same names.
6. Re-run the analysis – it is very easy to change parts of the model, re-run and look at the new results.
7. For all steps documenting the model and results – emphasis has been put on documenting the model graphically and/or tabulated using standard office tools like for example the MS Office package.

## 2.4 Setting the design premise

Prior to modelling you should decide the design premise, that is change some of the built-in knowledge of GeniE if this is needed. You may change the following default settings:



*Example of changing units*

- Settings for lengths of can, stub and cone and how to perform the gap calculation
- Rules for tolerant modelling
- Rules when moving members – connected members to follow or not
- Units – typically SI or Empirical units
- Meshing rules – you may instruct GeniE how to create a finite element mesh given your settings for mesh parameters like e.g. mesh angles and Jacobi determinants

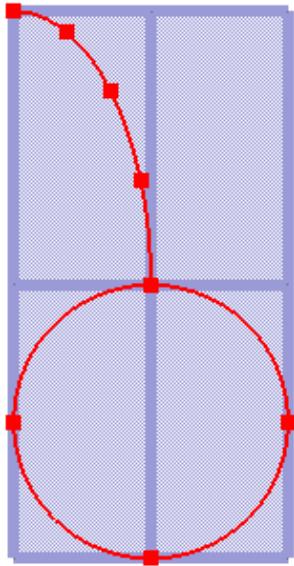
## 2.5 Use of guiding geometry

Guiding geometry is the first level for modeling structures in Genie. It contains a set of points, grids and curves, which are not part of the model, but they are created in order to help the user during the modeling process. These objects are purely geometrical in the sense that they do not have any physical property (such as material, permeability etc) but only dimensions. Thus, guiding geometry objects play no role in the meshing of the final model.

Guiding geometry may help the users in the following ways:

- It provides us with *snapping-points*, which can be used directly in concepts creation (e.g., plate from polygon) or in other guiding –geometry entities (e.g., spline interpolation).
- The guiding points and curves can be used directly in concepts creation (e.g. creation of curved beams from guiding-curves and shells by applying skinning or curve-net interpolation schemes to sets of guiding curves).

The guiding geometry entities can be categorized into the following groups:

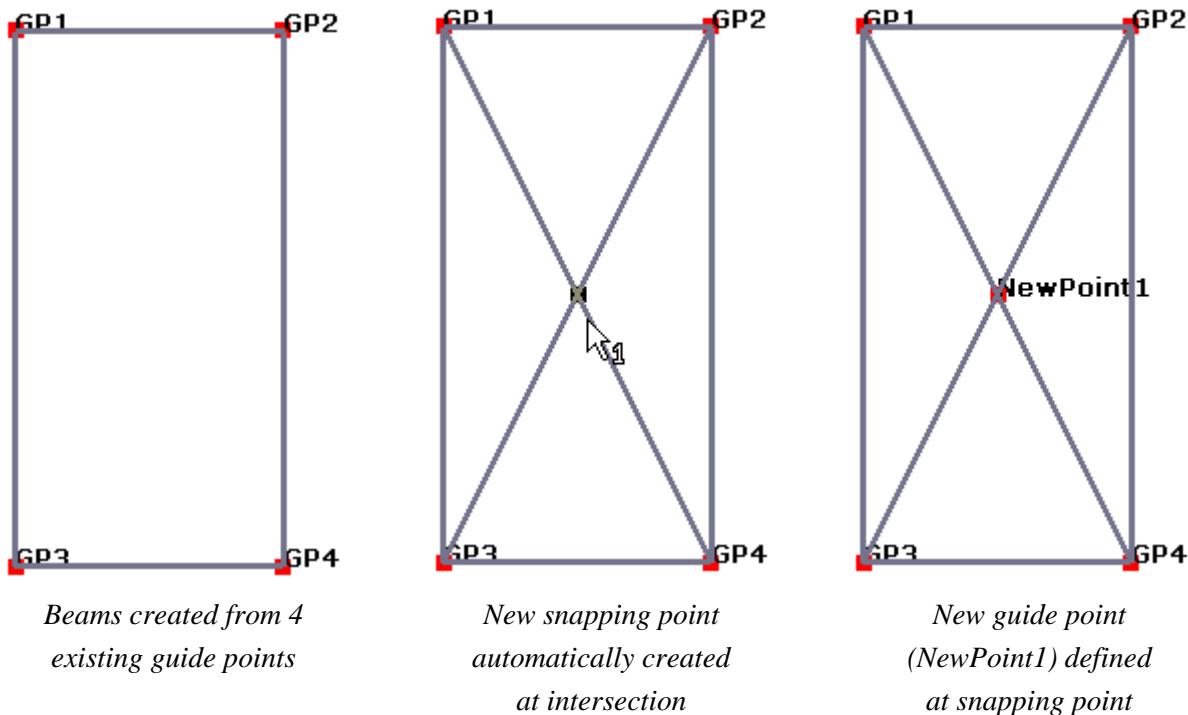


- Points, single or point-sets and point-nets.
- Curves, which can be lines, polylines, conic sections, splines and curves on the surface of the structure. All curves are defined as NURBS, which gives the possibility to read them from Rhino (v4.0).
- Planes, which are rectangular, triangular, trapezoidal grids of snapping points.

When modelling the structure point-snapping option may be used. It helps the user to focus on the modelling process, instead of calculating the exact coordinates of specific points, when, e.g., modelling a beam or a plate. Implicitly, it also helps towards model integrity. This technique may be compared with the MS PowerPoint snap-to-grid.

A snapping point can be a single point, the end point of a guiding curve or a 1D concept, a predefined point on a guiding curve, or it can be automatically calculated when guiding geometry entities or structural objects, are intersected. A snapping point may be used for modelling new objects. When the user moves the mouse cursor relatively close to a snapping-point, then it snaps on that point automatically.

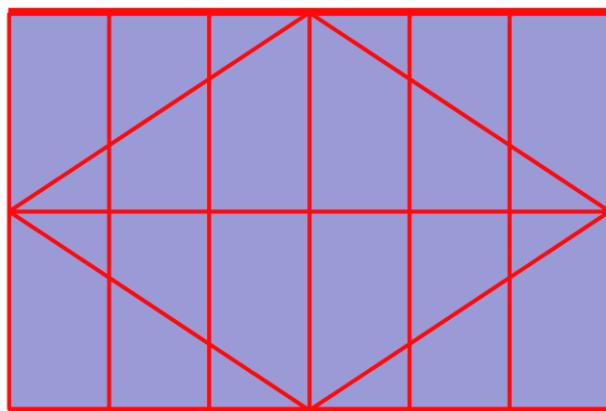
The figures below show how snapping points are created during modelling. Let us assume we are given four guiding points (which are always snapping points). Then, we can graphically construct the four beams, shown in the first figure. After this, we construct another two beams, from GP1 to GP4 and from GP2 to GP3, which intersect each other. The intersection point becomes a snapping point and behaves as the guide points, ready to be used when modelling a new object.



Thus, the introduction of such significant points as snapping points lets us start a modelling task with a minimum of guiding geometry and continue building using the new snapping points, which have been automatically established.

## 2.5 Basic structure modelling

Basic structure modelling is characterised by modelling straight beams or plates in a plane(s). A regular structure is easily generated by inserting beams and plates using guiding geometry (graphic modelling snap-to-grid technique). Beams and plates may also be created using the more traditional method by specifying end co-ordinates manually (line based or graphic modelling where input data is explicitly given). As explained earlier in this chapter, all modelling is done on a concept level and not on a geometry or finite element level. In other words, GeniE will let you model the structure as you see it. A beam can cross many plates and other beams, GeniE will keep track of all intersections and topology for you. Similarly, it does not matter whether you insert plates or beams first. Calculating the intersections is done on the fly by GeniE and when making a finite element model a consistent mesh will be generated.



The figure to the left illustrates a regular structure which has been modelled using beams and plates inserted in random order. Also observe that a beam is considered as a beam in its full length, i.e. start and end points are at the outer ends of the beam and not at each intersection with another beam.

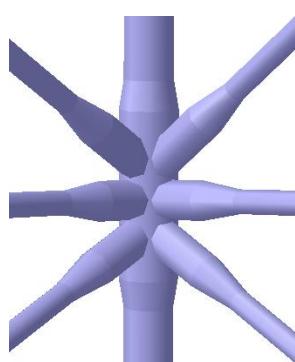
You may of course decide to split the beam into smaller beams. The selected beams to the left (highlighted in read colour) are examples of continuous beams intersected by several beams.

GeniE has several features that will allow you to change and modify the model after it has been created. Typical examples may be copy, move, change or split for single or a group of members in one go.

## 2.6 Advanced structure and panel modelling

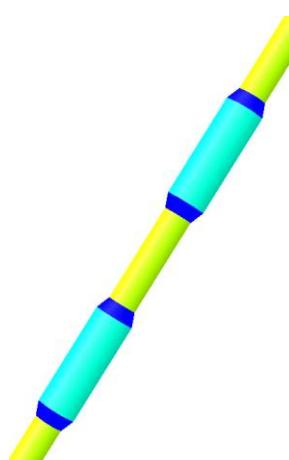
GeniE comes with several features for advanced structure and panel modelling, for example automatic tubular joint design, segmented beam modelling, punching, automatic creation of shim elements, flushing stiffeners (offset of beams) or curved beams and plates.

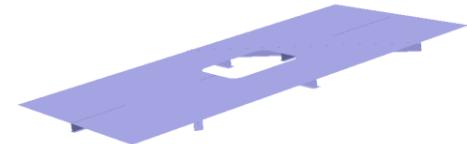
The figures below illustrate a number of examples where GeniE's features for advanced modelling have been used. GeniE is also capable of handling double-curved plates and connected beams.



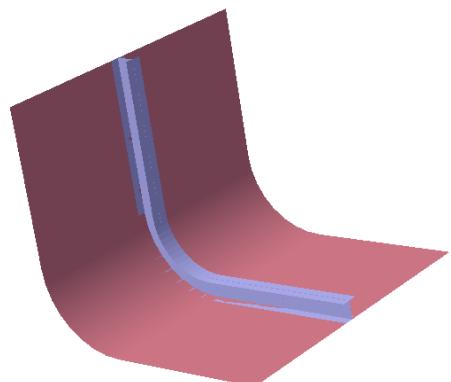
Tubular joint including cans, stubs, cones and planewise gaps

A segmented beam consisting of pipes and cones

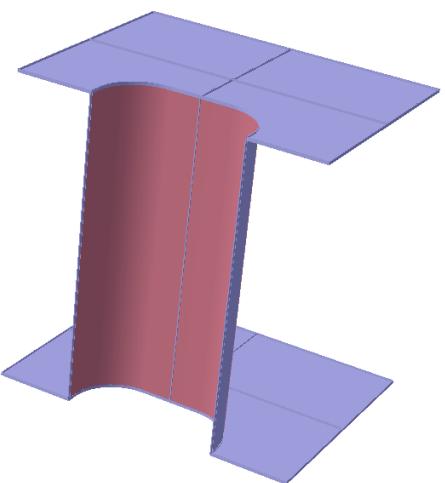




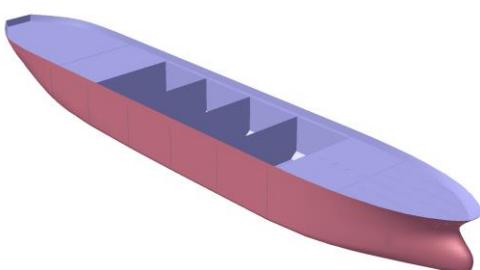
A stiffened panel punched by a 2D-profile



A single curved plat containing one flushed beam.



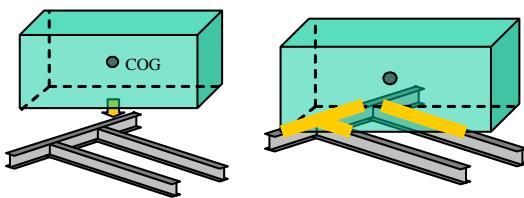
Two plates connected by a half cylinder.



A ship model intended for hydrodynamic and stability analysis (a panel model).

## 2.7 Equipment and load modelling

Gravitational and inertia loads relevant for a topside structure are normally caused by equipments placed on the structure. Thus, the designer needs to calculate these loads prior to applying them to the analysis model. GeniE offers the possibility to model the equipments and to place these on the concept structural model for given load conditions. The program will compute the necessary line loads or masses automatically.



*Placing an equipment and automatically calculate line loads on supporting beams*

Structural assessment cannot be limited to dealing with structural concepts only. Often, the design of the structure will be influenced by other concept information such as heavy equipment. Equipments may be seen as non-deformable objects that generate loads but do not contribute to structural stiffness. If this assumption does not hold the equipment should be modelled as a structural entity.

Prior to generating an analysis model you need to specify whether the model is intended for a linear structural analysis (loads from equipments must be used) or for a dynamic analysis model (where mass from equipments is used).

### 2.7.1 Equipment modelling

The equipment items are independent concepts that may be modelled explicitly or imported via weight lists. The equipments have mass and dimensions and via gravity or other acceleration this mass induces loads. The loads are distributed over the load carrying interface between the equipment footprint and the structure and can be represented as line loads applied to all the beams part of the load carrying interface.

The footprints may be modelled exactly to ensure correct load transfer to the structure. The user may also specify a load pattern rule, that is specify which parts of the structure that shall receive loads.

Note, however, that data imported from weight lists do not contain information like footprints and relative centre of gravity. User defined equipments are thus termed *Prism Equipment* while an equipment imported from a weight list is called *Bounding Box Equipment*.

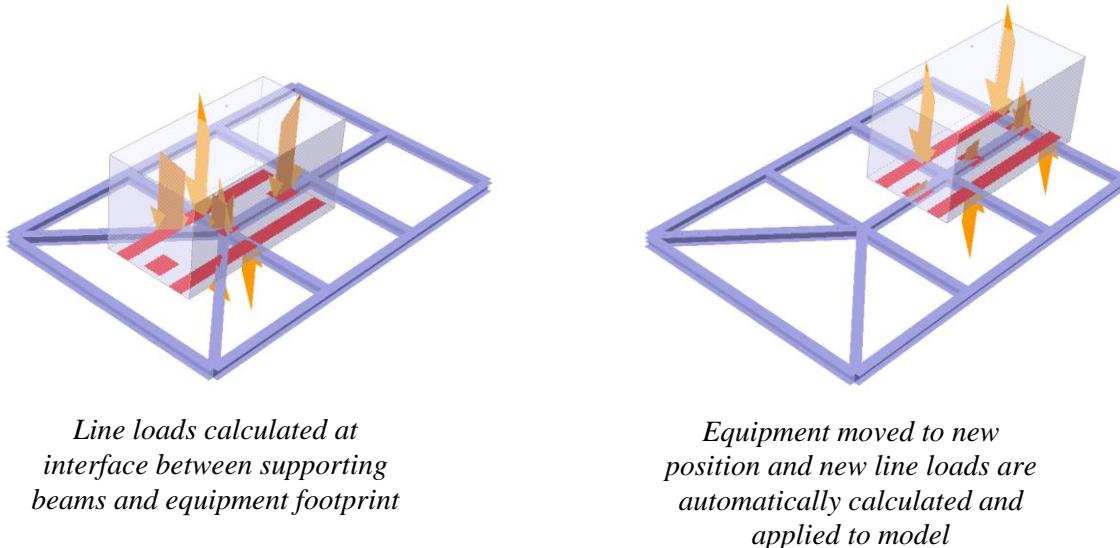
GeniE will always calculate a correct equipment mass and load accounting for

- Relative centre of gravity of the equipment itself
- The load transfer pattern – typically skid beams or support points
- Load transfer rules specifying which beams will receive loads – typically beams in one direction

Once the load carrying interface is established the load generated by the equipment is automatically calculated. First, the total equipment force is calculated as  $F_{equipment} = Mass_{equipment} * acceleration field$ . This force is then distributed over the load carrying interface while ensuring force and moment equilibrium.

When the acceleration contains a horizontal part shear forces as well as a pair of forces is automatically calculated – the vertical location of the centre of gravity plays an important role.

When moving equipments the load calculation is done on the fly. In other words, you do not need to calculate loads when you have moved equipments or inserted a new supporting beam. There are also several ways of verifying and documenting that the loads have been correctly applied to the model. The figures below show loads calculated on the same equipment, but with different locations.



The illustrations above show an equipment with many footprints (support surfaces). Where the footprints intersect with beams linearly varying loads are calculated to ensure moment equilibrium. An equipment may be used for several loadcases and in various positions (on top, below or along supporting beams). You may also create new equipments based on existing equipments, for example from a copy and change operation.

The resulting COG of mass from Equipment when using Footprint-Mass will usually be in the centre of the projected surface, even if the equipment COG is not in the middle of the Equipment box. The mass of the equipment is in this case split into equal parts and put into the supporting nodes. The COG will be near the middle of the supporting plane, unless different number of nodes at different beam edges under the equipment.

### 2.7.2 Weight lists

Many weight control systems can produce a neutral file containing information about the equipment objects, their mass, their size and where they are located. GeniE can use this information to automatically calculate loads to a model. Weight lists may be specified in .csv or .xml format.

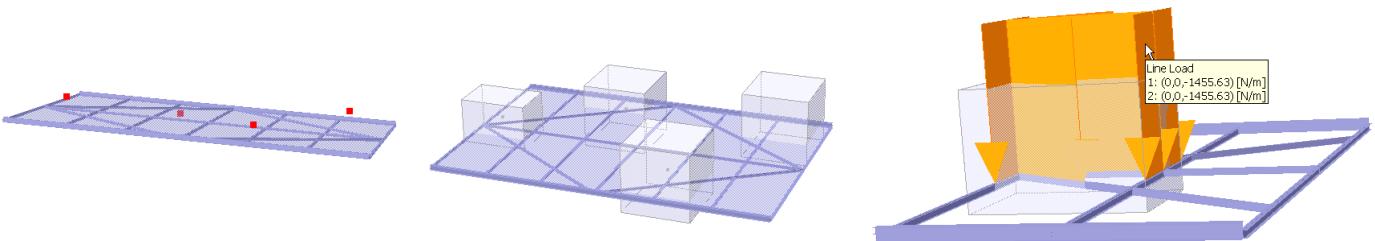
Equipments in a weight list may not always have information about its size, you can therefore decide a bounding box so that GeniE will know which structural members will receive load (where the box intersects with structure).

When calculating loads from weight list objects the total force (mass times acceleration) is distributed as constant line loads on the load interface. For weight list objects the load interface is the intersection between a beam and the bounding box. This means that using the weight list approach is not as accurate as modelling equipments since these hold more information (relative centre of gravity, footprints, calculating linearly varying loads).

The loads or masses produced from a weight list may be used in many loadcases in the same way as other equipment. They can, however, not be used in different positions since their locations are specified in the weight list.

You can read in many weight lists and use these in different loadcases or load combinations. You can also choose to not load all items from an imported weight list into a loadcase.

The figures below show how an imported weight list read and converted to line loads. The first step is to import the data, the next step (automatic) is loading a loadcase and the final step is automatic calculation of constant line loads.



*Weight list objects read by program – the red dots indicate locations*

*Weight list objects (bounding box equipments) placed in a loadcase*

*Constant line loads calculated where beams intersect the bounding box equipments*

### 2.7.3 Acceleration loads

You may specify two types of acceleration loads in GeniE. The first one – which is the most used – is applying a constant acceleration field (x,y,z directions) to the model. The selfweight of the structure is represented by applying a constant acceleration field in vertical direction (e.g.  $-9.81 \text{ m/s}^2$  using SI units). Note that the loads calculated will include contributions from both structure mass and any point masses defined.

The other option is to define an angular acceleration field. Normally you use this option when acceleration results from a hydrodynamic analysis are known and you want to model and analyse e.g. a topside structure without any connection to the hull.

### 2.7.4 Explicit loads

There are a number of explicit loads that you can apply to the model. Typical loads are point loads, line loads, surface loads (pressure loads on plates), temperature and prescribed displacements. Observe that none of these loads can contribute to the mass model required for dynamic analysis.

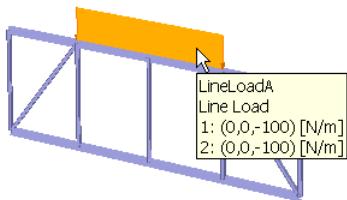
GeniE is flexible in how the loads are applied. For example, a point load does not have to be located at a previously defined point or at beam ends. Similarly, a line load may be applied to a part of the length of a beam. For surface loads the pressure may be applied to parts of a plate if desired. It is required, however, that all explicit loads intersect structural elements to contribute to the analysis. Hence, moving a beam will not move loads defined to act on the beam.

A point load including moments, i.e. not only force components, cannot be applied somewhere along a beam. It must be applied at the beam end or at a defined point along the beam. A point load including force components only may, however, be applied anywhere along a beam.

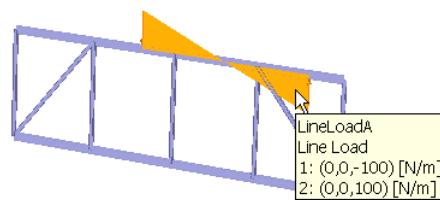
Point and line loads apply to beams while surface loads apply to plates. In other words, if you apply a line load to a plate no loads will be calculated for use in the analysis.

Once an explicit load has been created it is very easy to modify and move the load. Contrary to equipment modelling, an explicit load can only be used in one loadcase.

In the example below a constant line load has been applied between two structural joints. Then the load is moved and the load intensities changed so that it becomes a linearly varying load.



*Line load applied to part of continuous beam*



*Same line load moved and with new intensities*

### 2.7.5 Loadcases and load combinations

A loadcase is set up by either placing equipments on the structure, including items from a weight list, defining explicit point/line/surface loads, including effects from temperature loading or accounting for acceleration (mass and constant or angular acceleration).

Load combinations are easily created by including loadcases where you also may add scaling factors. A load combination can be used in other load combinations (also called nesting), with or without scaling factors.

A load combination may be viewed graphically with the scaling factors included in the graphic presentation.

When using the feature for several analyses, different loadcases may be referred to.

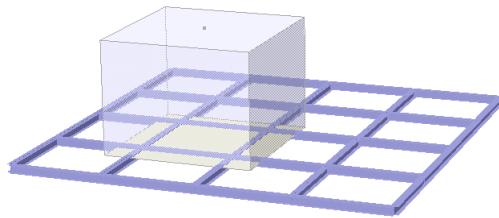
## 2.8 Prepare for structural analysis

Prior to performing an analysis it is necessary to generate a finite element model. GeniE will create a finite element mesh either automatically using default settings or those specified by you. In case it is not possible to create a finite element mesh you will be informed which parts are causing the problems and also guided in how to refine the model so as to be able to successfully create a finite element mesh.

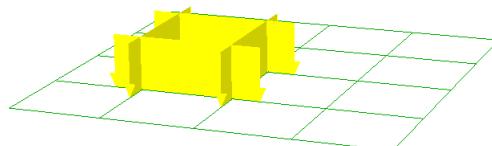
A model created in GeniE can be used for several purposes (see next page for illustrations):

- Linear static analysis controlled from the GeniE user interface. In this case GeniE will automatically create a finite element mesh, perform the analysis using Sestra and produce a result file that can be accessed by GeniE. For such analysis it is important that the equipment mass is applied to the model as line loads (this is the default option).
- Eigenvalue analysis controlled from the GeniE user interface. GeniE will again create the finite element mesh and do the eigenvalue analysis using Sestra. The eigenvalue results are ready for visualisation in GeniE. In this case it is important that the equipment mass is applied to the model as point masses connected to the model (you need to change default settings).
- A model intended for use outside the GeniE domain. Examples may be a superelement created by GeniE for subsequent import into Presel, you may want to refine the Sestra input and run the analysis separately or you may want to add a GeniE created mass model to a hydrodynamic analysis. In such case you need to decide how to represent the equipments (loads or masses), apply the relevant boundary conditions (super-nodes in case of a superelement analysis) and create the finite element mesh prior to exporting the finite element model for its intended use.

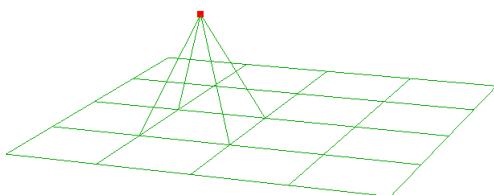
A common approach when making an analytical representation of a real structure is to neglect structural parts and coating that are not relevant for the stiffness of the structure. However, these have importance regarding the total mass of the structure. To overcome this limitation, it is possible to scale masses to a target mass value.



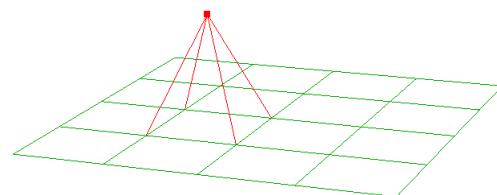
*Equipment placed on top of frame.*



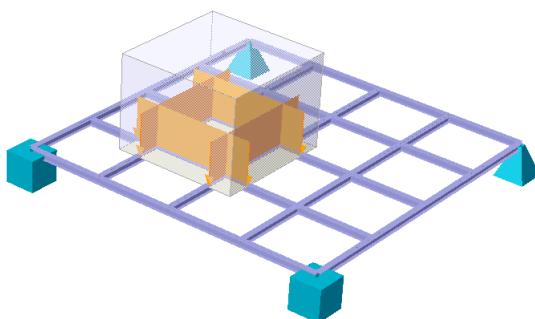
*Linear static analysis: Equipment mass represented as line load in finite element model.*



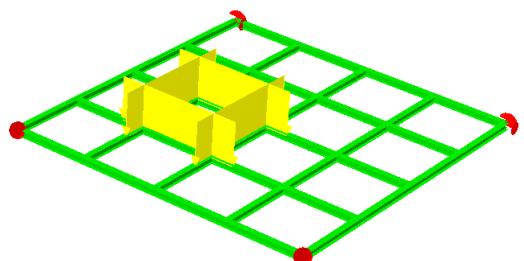
*Mass model: Equipment mass represented as point mass plus beam connecting the structural part. Hinges are introduced between connecting beams and structure to avoid moment transfer when point mass is subjected to accelerations.  
Eccentricities may also be neglected.*



*Mass model: Equipment mass represented as eccentric mass elements connected to the structural parts, note moment transfer when subjected to accelerations. This option should be used with care, and preferably only when global results are the target of an analysis.*



*Model containing boundary conditions (fixations and supernodes).*



*The finite element model is ready for inclusion in a superelement assembly. A supernode is visualised as a red ball.*

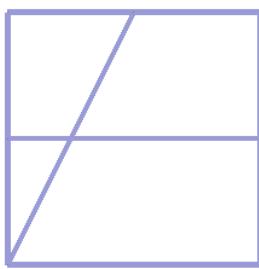
## 2.9 Run analysis and look at results

GeniE can run a linear static analysis using pre-defined parameters for running Sestra in the background. When the analysis is completed you can verify the analysis details by accessing the output files from Sestra.

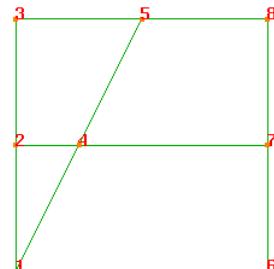
When you start the analysis GeniE will automatically recalculate all the loads, generate a new finite element model, run the analysis and create a result file that you can postprocess from the GeniE user interface.

GeniE has built-in knowledge in how to assess the finite element mesh quality – you may alter these settings if desired. However, it will always be your responsibility to ensure that the results found in the analysis are good enough. Hence, it is important that you check attention to the mesh quality. If you are not satisfied it is very easy to change mesh parameters and run a new analysis.

A topology point in the model will introduce a finite element node in the analysis model. In case you have done changes to your model you may want to remove unnecessary topology points to simplify the finite element model. GeniE comes with a special tool for this purpose – the simplify topology feature. Below is illustrated a simple frame where the inclined beam is moved and how this effects the finite element model.

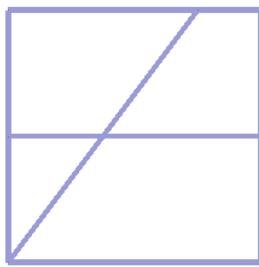


*Small frame model.  
Before moving member*

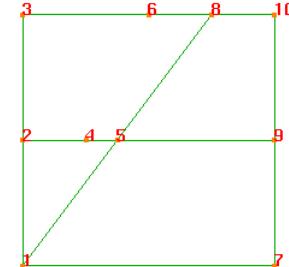


*Small frame FE model.  
Before moving member.  
8 FE nodes created.*

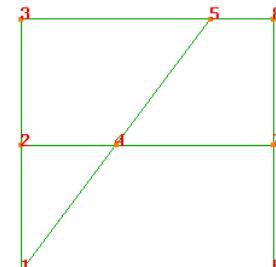
*Simplify topology is accessible  
from the toolbar:*



*Model after moving upper end  
of inclined member.*



*FE Model after moving member.  
10 FE nodes created.  
2 topology points not necessary.*



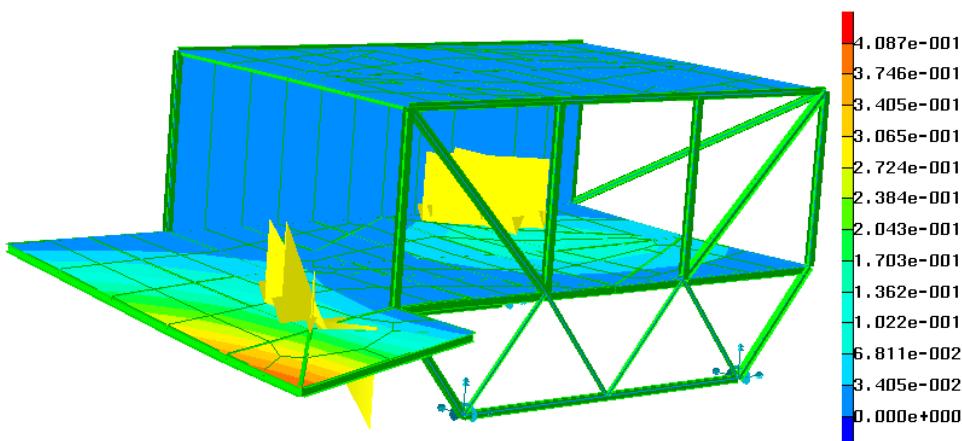
*FE Model after simplify  
topology.  
8 FE nodes created.*

The above example shows a pure beam but the same principles apply to a plate or mixed beam/plate model also on how finite element nodes are created where topology points are generated.

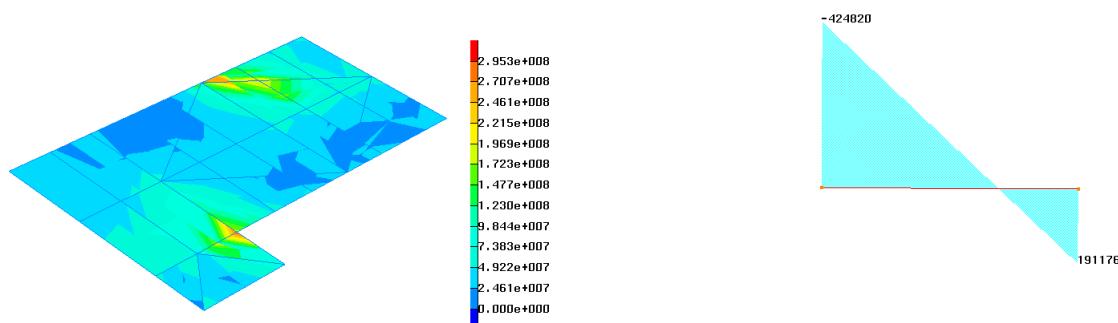
GeniE will let you look at certain results attributes like displacements, general plate stresses, principal stresses, and beam forces and diagrams. You may look at results for the whole model or parts of the model. Another advantage of the concept technology is that a concept holds information about the finite element details and results. This means that you can select one (or several) concepts and show results for given loadcases.

Furthermore, you may specify how much information to be presented per view. The examples below show a complete model (the finite element mesh, loads and displacements), a pre-defined set (VonMises stresses only) and one single beam (moment diagram).

Observe that results are showed in the database units specified when you start a new modelling session in GeniE.



*Displacements for global model.  
Figure shows FE mesh, load, and results.  
A scaling factor of 2.0 has been applied*



*VonMises stresses for bottom deck.  
Figure shows results on a selected set.*

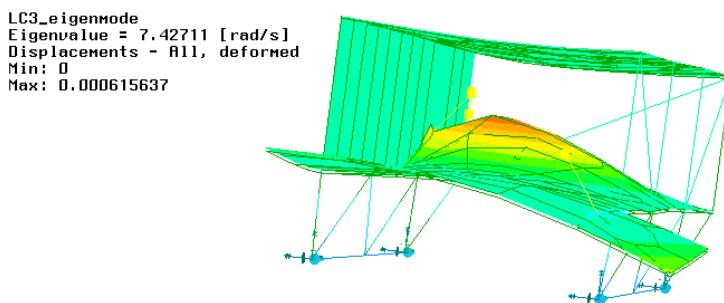
*Moment diagram for one selected beam.*

In case you want to re-run your analysis because the model has been changed (e.g. new beams, moved equipments) you only need to start the analysis process – GeniE will create a new finite element model and re-calculate all loads prior to running the analysis. This ensures full consistency between the concept model and the finite element model.

## 2.10 Run eigenvalue analysis

GeniE comes with a pre-defined input for running an eigenvalue analysis in Sestra. The input involves calculation of the 10 first eigenmodes using a number of methods (e.g. Lanczos'm Householder). If you want to calculate other eigenmodes or use another calculation method you can set up the necessary input data and run it from inside GeniE. Alternatively, you can export the finite element model and do the analysis from SESAM Manager (executing Sestra).

Prior to analysis you should remember to represent the equipment as masses.

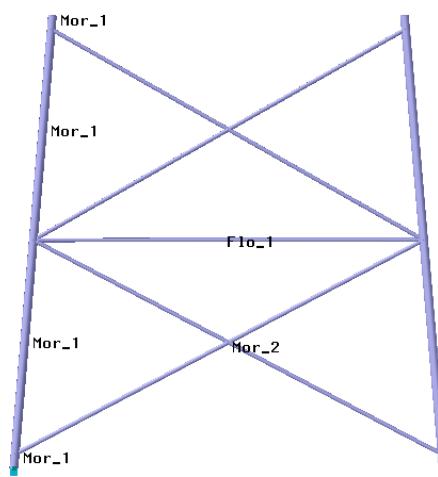


The example shows an eigenvalue analysis performed from the GeniE user interface. All equipment masses contribute to the mass matrix (structure mass plus equipment masses).

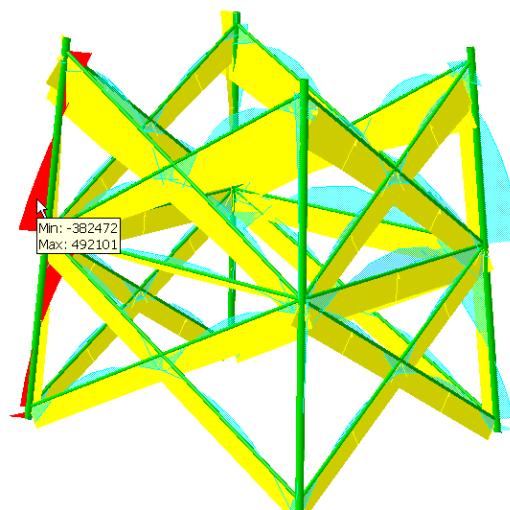
## 2.11 Run hydrodynamic analysis

Before running a hydrodynamic analysis based on Morison's theory you need to add necessary attributes to your model. Several of these belong to the concept model while other data belong to the environment. The data applied to the concept model are typically Morison coefficients, non-structural members and flooding. Remaining data (such as water depth, wave theories, current) constitutes the environment – Vol. 2 of this user manual describes how to model such data. GeniE creates the necessary input data for hydrodynamic analysis using Wajac.

Following the hydrodynamic analysis you may run a structural analysis including the loadcases from the hydrodynamic analysis.



*Two different sets of Morison coefficients and one flooding parameter applied to model*



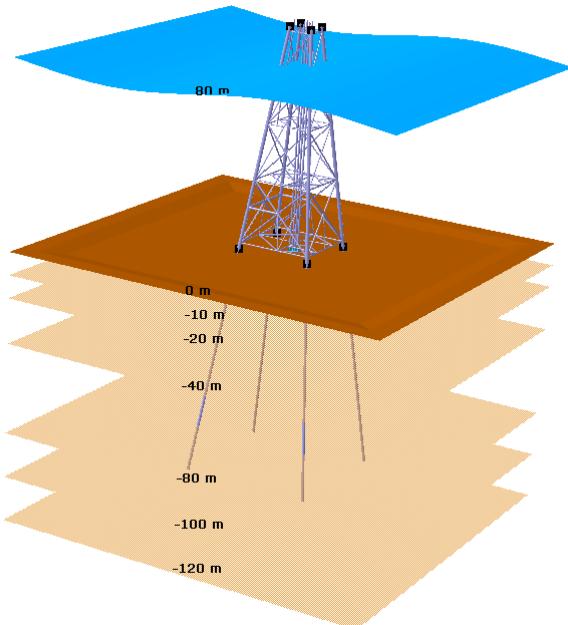
*Beam moment diagrams (blue colour)  
on structure exposed to wave loads  
(yellow colour)*

The examples above show a small jacket where Morison coefficients and flooding parameters are applied to the model and the results (wave loads and moment diagrams) from a structural analysis based on hydrodynamic loads.

## 2.12 Run pile and soil analysis

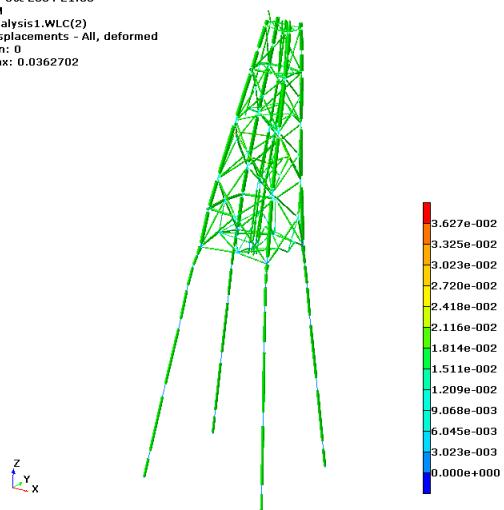
You may also model all data necessary to do an integrated analysis consisting of structure, wave (or wind), piles and soil as well as to do the analysis itself. Vol 2 of this User Manual explains the modelling features and also how to run such analysis, either as separate activities or as one integrated analysis.

The picture below shows the various parts of a complete model and an example on how results may be investigated for a complete model.



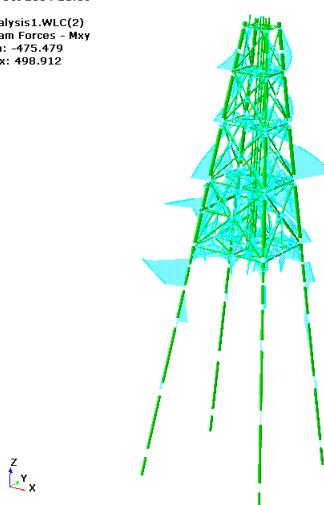
A complete model consisting of both environment and structure.

24 Oct 2004 21:38  
UM  
Analysis1.WLC(2)  
Displacements - All, deformed  
Min: 0  
Max: 0.0362702



Deformed shape

24 Oct 2004 21:39  
UM  
Analysis1.WLC(2)  
Beam Forces - Mxy  
Min: -475.479  
Max: 498.912



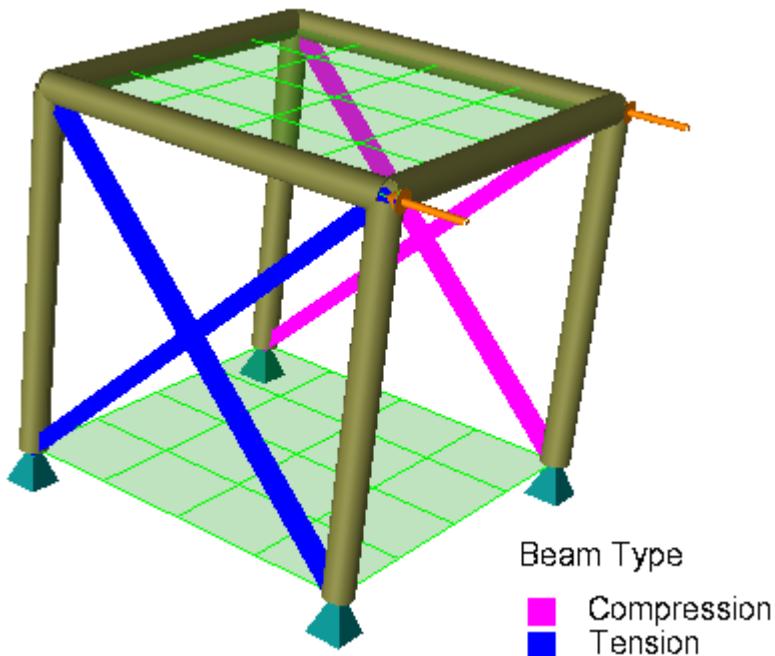
Bending moments

## 2.13 Tension/Compression analysis

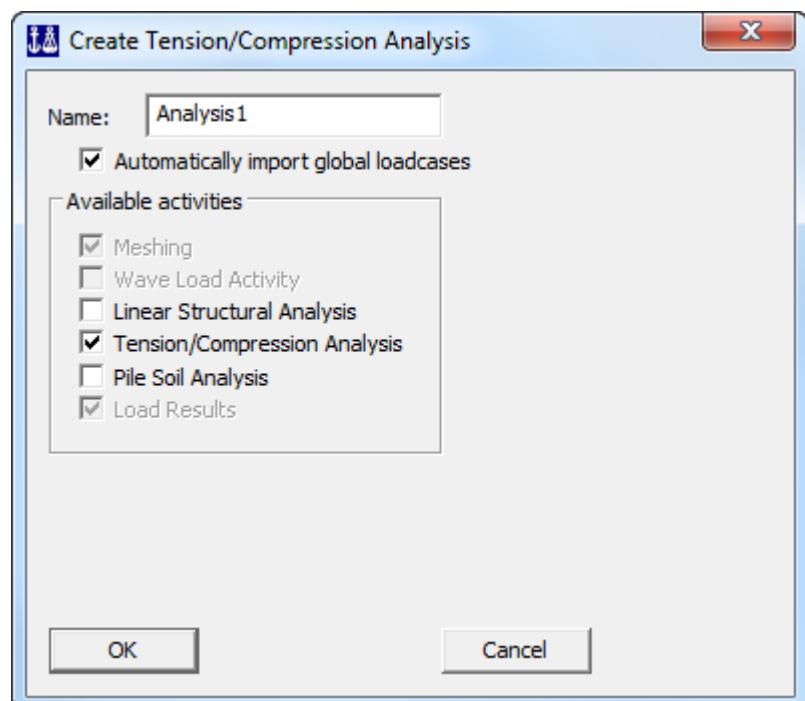
GeniE gives you the possibility to run a tension/compression analysis. A brief explanation on how to do this in GeniE follows below. For a more detailed explanation and examples, see the tension/compression manual in your Sestra installation.

To run a tension/compression analysis you need to set the beam type of the truss elements you want to analyse to Truss Tension Only or Truss Compression Only.

Note that the beam type Truss Tension and Compression cannot be used in this context.



To create a tension/compression analysis you need to check the Tension/Compression Analysis checkbox as shown in the illustration.



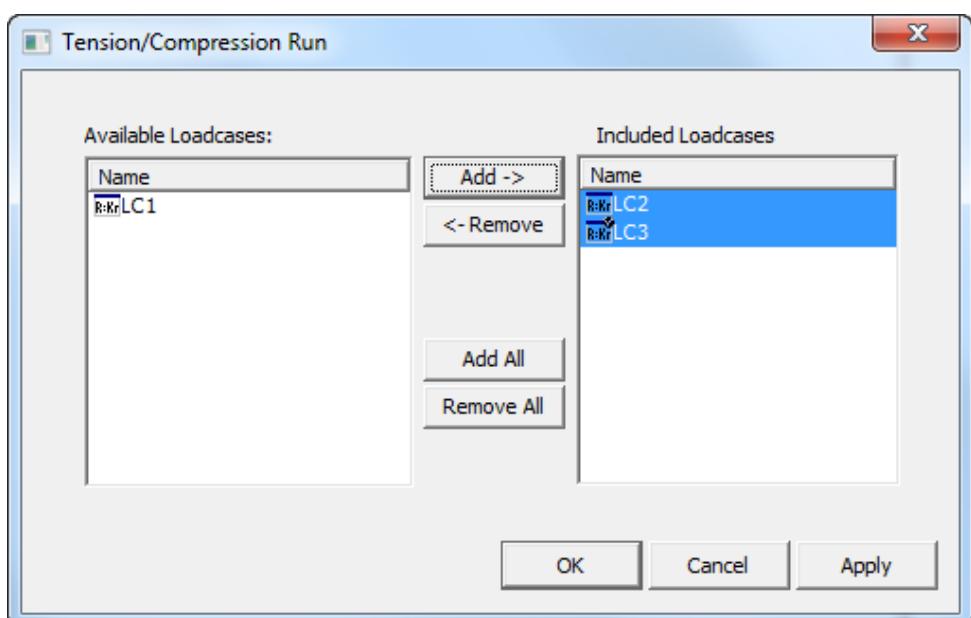
Name	Description	FEM Loadcase	FEM LC Rule
Analysis1.step(1)	Meshing (Always Regenerate)		
Analysis1.step(2)	Tension Compression Analysis		
Analysis1.step(3)	Load		
R:Kr LC1	Reference to LoadCase	1	Manual
R:Kr LC2	Reference to LoadCase	2	Manual
R:Kr LC3	LoadCase	3	Automatic

Before running the analysis you need to specify which loadcases that should be run as tension/compression. The remaining loadcases will be run as a linear structure analysis.

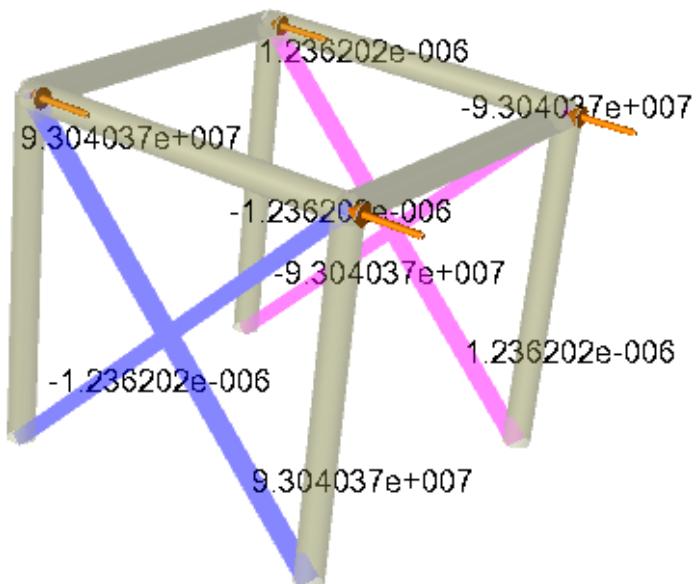
Note that the loadcases you include for tension/compression need to be the last in the FEM loadcase number sequence of the analysis.

In this example you have the following alternatives for loadcases you can include:

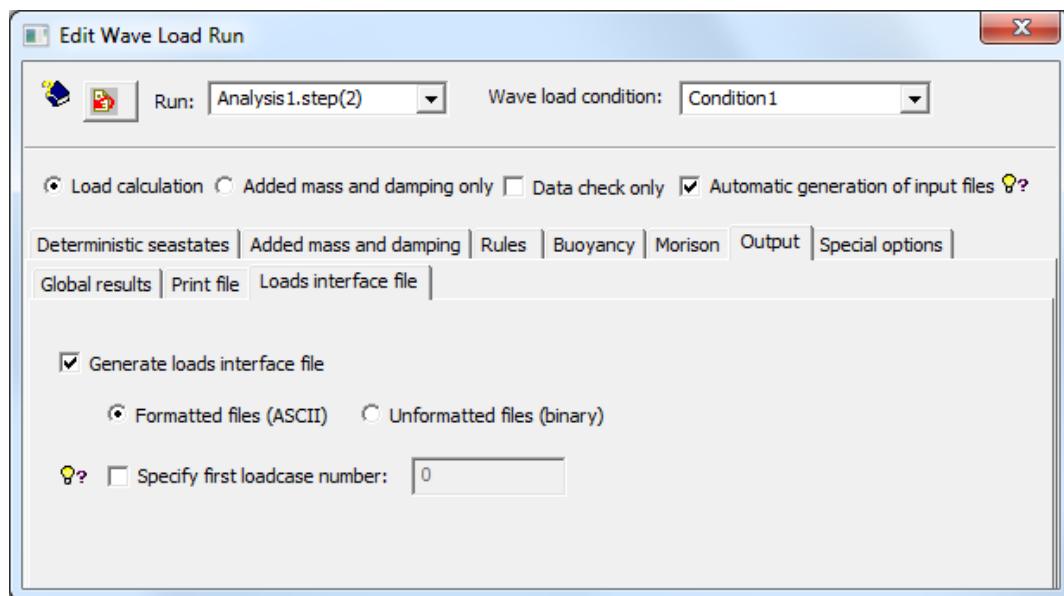
- LC1, LC2 and LC3
- LC2 and LC3
- LC3



```
24 Jan 2013 09:59
tension_compression
Analysis1
LC2
FEM Loadcase = 2
Force: [N], Length: [m]
G-stress - sigxx, Top, deformed
Min: -9.30404e+007
Max: 9.30404e+007
```



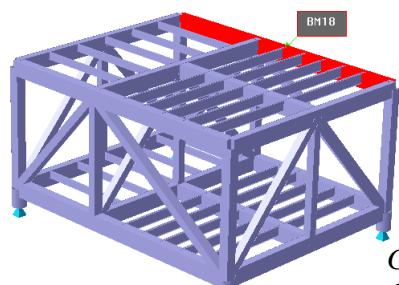
**Note:** It is possible to run an integrated wave load and tension/compression analysis using Wajac. You then need to specify Wajac to write loads to a Formatted loads interface file to make this work. If Unformatted files are specified, the analysis will fail with an error message in the TensionCompressionDriver.log file.



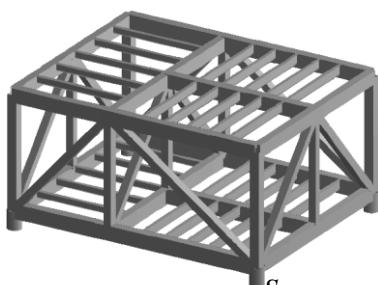
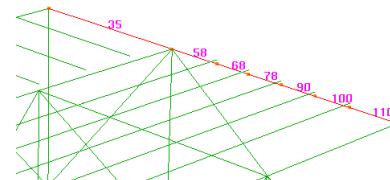
## 2.14 Code checking and advanced results processing

The code checking of frame structures is done in Framework while code checking of plate structures is done in Platework. More advanced post-processing can be done in Xtract. Platework and Xtract are based on finite element technology meaning that the concept information cannot be referred to in these programs. The programs will read the sets you have created in GeniE.

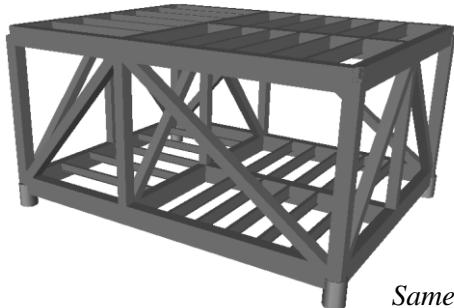
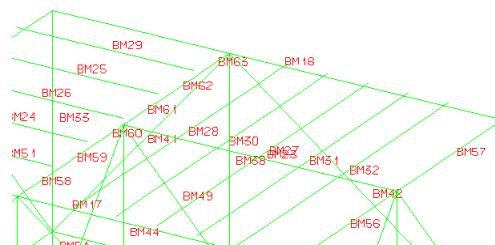
The concept information built into the model created in GeniE is understood and used by Framework meaning that all the naming conventions (e.g. beams, loads, materials, sets) are the same in the two programs. Note, however, that you should limit all names to 8 characters. The examples below show a small model viewed in GeniE, Framework and Xtract.



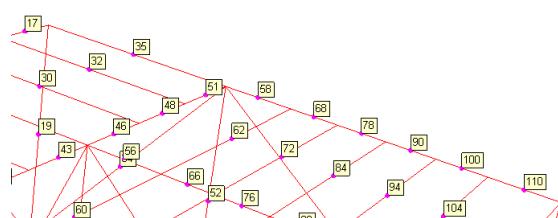
*GeniE created model  
Concept level: BM18  
Finite element numbers: 35, 58, 68, 78, 90, 100, 110*



*Same model viewed in Framework.  
Program reads concept information.  
The selected beam is referred to as BM18.*



*Same model viewed in Xtract.  
Program does not read concept information.  
The selected beam is referred to as finite elements  
35, 58, 68, 78, 90, 100, 110.*



## 2.15 Re-analysis

A re-analysis may easily be performed after changing some of the properties. Often new structural members have been inserted, section types edited and changed, mesh density refined and so on.

Only one command is necessary to generate a new mesh, running the analysis again and present the same type of results again.

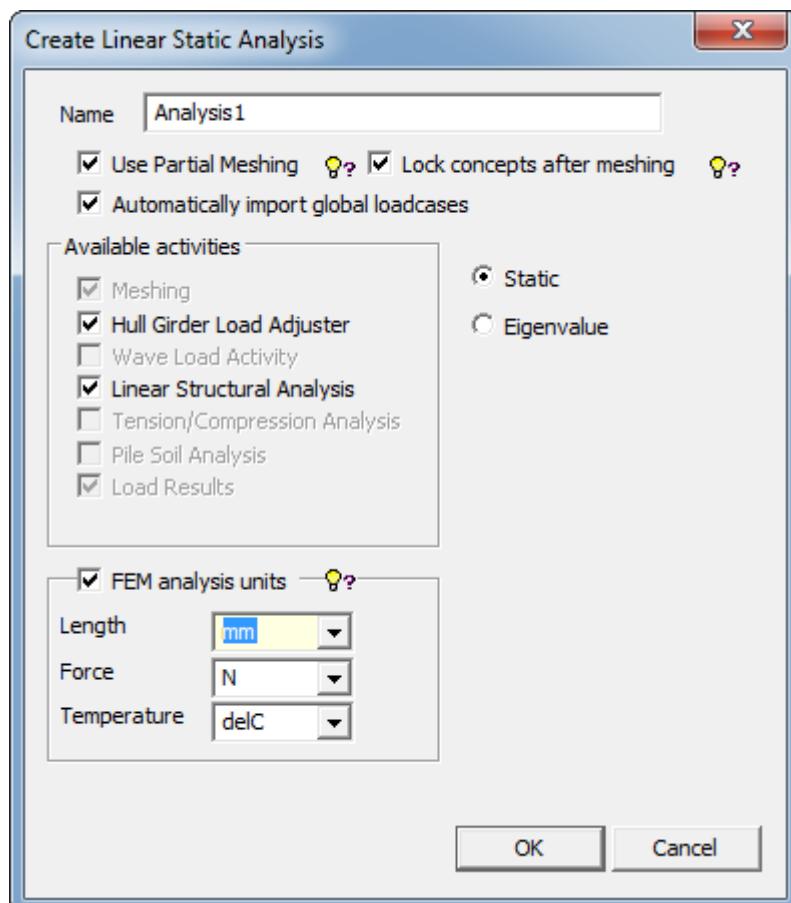
You may execute as many re-analyses as you want. Observe also, that a new finite element model will be created every time you do an analysis. GeniE takes care of consistency between finite element model, results and the model itself.

## 2.16 Hull Girder Load Adjuster

The Hull Girder Load Adjuster (HGLA) is an application that computes and applies hull girder load adjustments for 3 cargo hold models according to CSR BC&OT or DNV GL rules. It requires both a ShipData.xml and a ruleloads xml from Nauticus Hull. Note that the load adjustments are applied to the FE model only, e.g. the load concepts are not updated. If the model or loads have been changed, HGLA has to be run again.

### 2.16.1 How to run HGLA

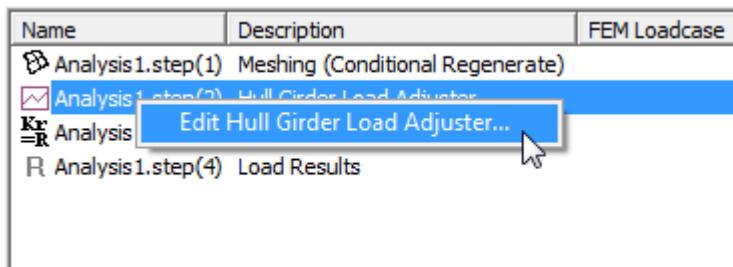
Create an analysis in the normal way. In the “New Analysis” dialog, check the checkbox “Hull Girder Load Adjuster”.



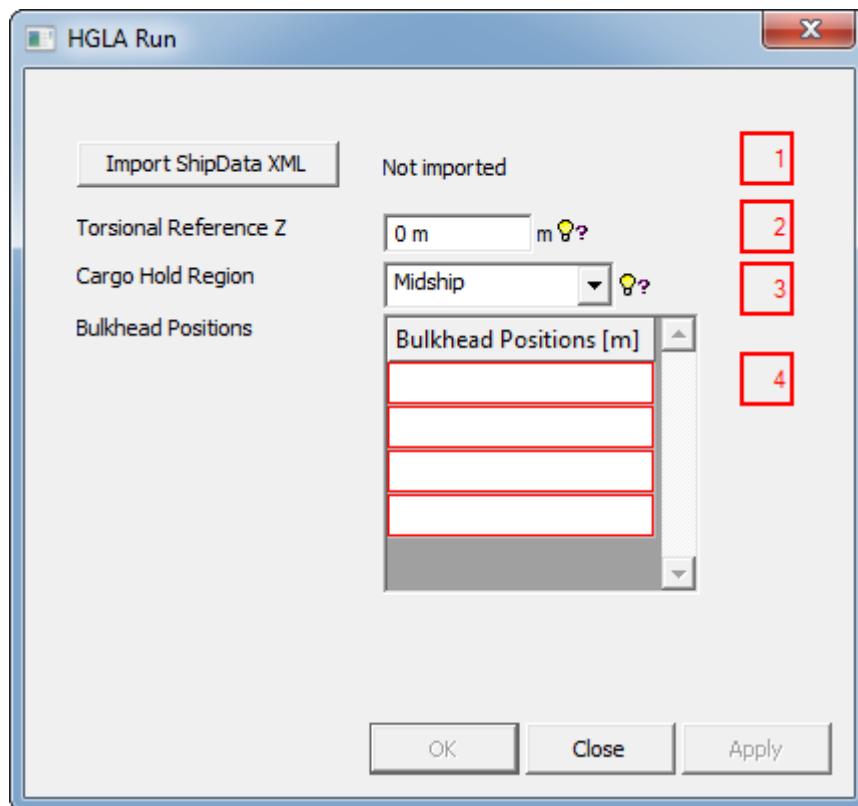
The FEM analysis units must be the same as are used for xml file units for the ruleloads xml export (compartments). An imported FE model may have different FEM analysis units than database units.

In the resulting analysis, the HGLA analysis activity will be positioned after the meshing activity and before the linear static analysis. None of the other analysis types will be available when running HGLA.

Before you can run the analysis, the HGLA activity needs to be set up. In the browser, RMB on the HGLA activity, and select “Edit Hull Girder Load Adjuster”



This opens the HGLA setup dialog.



The following input is required:

Import the ShipData.xml file from Nauticus Hull. Clicking on the button will open up a File dialog.

Torsional Reference Z value. This can be obtained from Nauticus Hull. For Bulk Carriers this should be 0.0.

Which cargo hold region your model represents; Aft, Midship or Forward.

The four bulkhead positions for the 3 cargo hold model.

### 2.16.2 Running the analysis

Once the HGLA activity has been set up, and the rule loads XML file has been imported from Nauticus Hull, the analysis is ready to be executed. The analysis is executed the normal way, by using the Activity Monitor in GeniE.

### 2.16.3 What happens when the analysis is run

When you execute the HGLA analysis activity, the following happens:

A sub-directory is created to store the input and output files to HGLA. This subdirectory is called HGLA and resides within the Analysis directory

The following files are copied in to the HGLA directory:

Ship data XML file

Rule loads XML file

FEM file

An input file to HGLA is created.

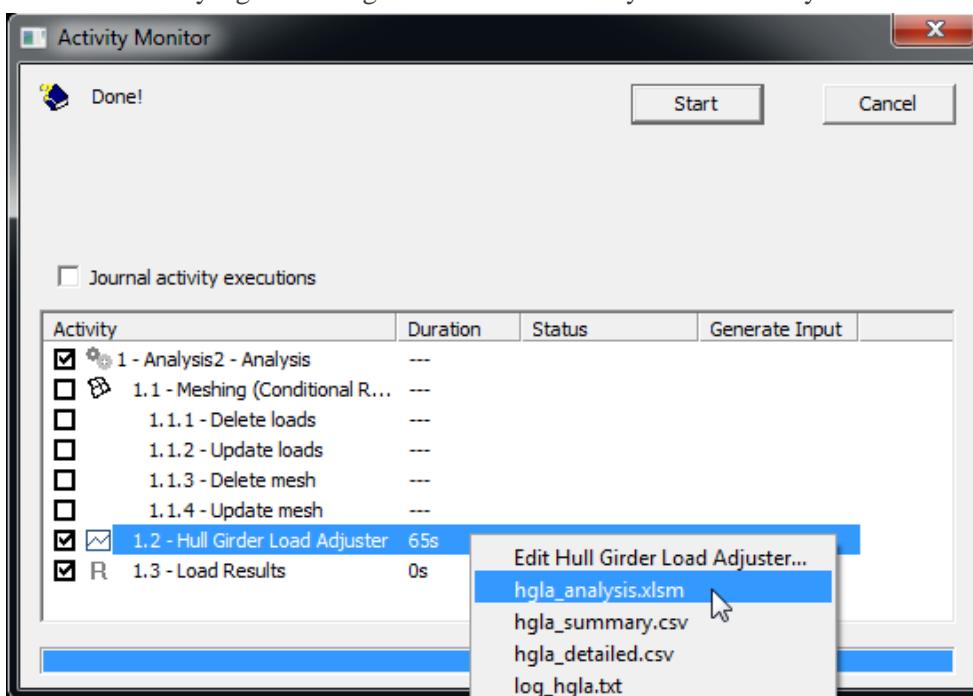
HGLA is executed

After HGLA has updated the FEM file a utility program called HGLAMerger will merge some results into an Excel spreadsheet file named hgl\_a\_analysis.xlsx.

The output FEM file from HGLA is copied back to the Analysis folder. The original FEM file is backed up.

### 2.16.4 Checking the output from HGLA

After a successful run of HGLA, some result files are produced in addition to the new FEM file. These files can be viewed by right-clicking on the HGLA activity in the Activity Monitor.



The files available for viewing are:

hgl\_a\_analysis.xlsx: This Excel spreadsheet is used to view graphs for vertical and horizontal shear forces and bending moments for each loadcase.

hgl\_a\_summary.csv: This file contains summary data that will also be present in the spreadsheet above.

hgl\_a\_detailed.csv: This file contains detailed data that will also be present in the spreadsheet above.

log\_hgl\_a.txt: This file contains logging information produced by the HGLA program. This file may be useful in case HGLA fails.



## 2.17 Documenting the model

There are a number of ways to document and verify the model covering the range from on-line verification using the graphic capabilities and pop-up property sheets to documenting the model using either print-out or graphic file options.

(15 m,0 m,5 m)

(15 m,0 m,0 m)

It is easy to select whole or parts of the model to investigate and to which you want to add additional information. Typical examples may be labelling of section and material names. The figure to the left shows an imported graphic file where name and the coordinate values have been added to a beam.

Hard copies of the graphic window may be directed to your default printer or saved to file. A graphic file may easily be imported into an office system. You may also document your model by saving the details to files that can be accessed by e.g. Notepad, any Internet Explorer or MS Excel. The example below shows how you can verify and document the model data using MS Excel.

	A	B	C	D	E	F	G	H
1	Name	Mass [Kg]	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	Length [m]	FlexLen [m]	Type
2	BM1	3.4289E+01	14.000	0.000	0.000	28.000	28.000	Straight Beam
3	BM10	2.5915E+00	5.500	14.660	0.450	11.000	11.000	Straight Beam
4	BM11	2.5915E+00	5.500	18.330	0.450	11.000	11.000	Straight Beam
5	BM12	6.7754E+00	14.650	11.000	0.300	22.000	22.000	Straight Beam
6	BM13	6.7754E+00	17.150	11.000	0.300	22.000	22.000	Straight Beam
7	BM14	6.7754E+00	19.650	11.000	0.300	22.000	22.000	Straight Beam
8	BM15	6.7754E+00	22.150	11.000	0.300	22.000	22.000	Straight Beam
9	BM16	6.7754E+00	24.650	11.000	0.300	22.000	22.000	Straight Beam
10	BM17	3.3190E+01	14.000	0.000	12.500	28.000	28.000	Straight Beam
11	BM18	3.3190E+01	14.000	22.000	12.500	28.000	28.000	Straight Beam
12	BM19	2.3315E+01	0.000	11.000	12.500	22.000	22.000	Straight Beam
13	BM2	3.4289E+01	14.000	22.000	0.000	28.000	28.000	Straight Beam
14	BM22	2.5915E+00	5.500	3.670	12.900	11.000	11.000	Straight Beam
15	BM23	2.0151E+01	19.500	11.000	12.500	17.000	17.000	Straight Beam
16	BM24	2.5915E+00	5.500	7.340	12.900	11.000	11.000	Straight Beam
17	BM25	2.5915E+00	5.500	14.660	12.900	11.000	11.000	Straight Beam
18	BM26	2.5915E+00	5.500	11.000	12.900	11.000	11.000	Straight Beam
19	BM27	6.7754E+00	10.650	11.000	12.750	22.000	22.000	Straight Beam

*The model data documented using MS Excel.*

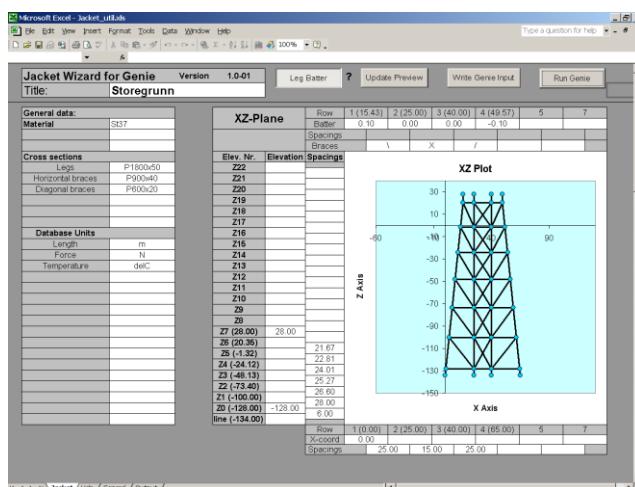
*In this case the mass of BM17 is in focus*

## 2.18 Journalling and reading commands, wizards

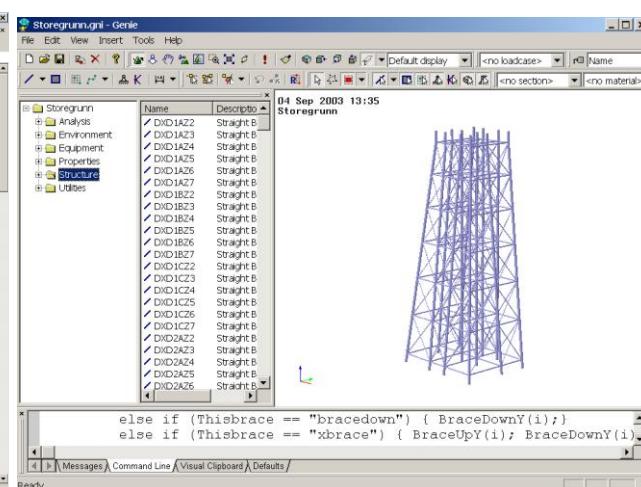
GeniE comes with full journal support of all modelling and analysis commands. This means that every change of the model is journalled to a file unique for each modelling session. The journal file (file extension \*.js) can be read in to re-create the model and it can be modified in a standard text editor to suit the user's needs.

The syntax used on the journal file is based on the JScript™ scripting language. This means that the user can extend the modelling features of GeniE by utilising the powerful features of the JScript programming language. Examples of such are macros and wizards, or journal files that you want to re-use from time to time (e.g. your own section properties).

GeniE comes with two wizards (jacket and deck wizards) that use the programming feature in the journalling system. The figures below show MS Excel running the jacket wizard and GeniE with the same model (automatically created using the journal file).



*The wizard guides the user in how to specify a jacket (or topside) structure. The wizard will create a journal file and start GeniE that automatically creates the model. The wizard also contains information on how it is build up so that you can create your own wizards.*



*The jacket model sent to GeniE. The wizard allows you to specify section and materials – this means that the beams created are more or less complete. To run an analysis of this model you only need to specify boundary conditions and loading conditions.*

The wizards are accessed from the help system and they include guidance on how to use the wizards. Consult the Readme.txt and also the "Help"-tab in the Excel spreadsheets.

Sesam GeniE Help - Microsoft Internet Explorer provided by Det Norske Veritas

- File Edit View Favorites Tools Help
- Favorites
- Vol 1 - Concept engineering
- Vol 2 - Waves, pile and soil
- Vol 3 - Plate/Shell Structures
- Vol 4 - Beam code checking
- Vol 5 - Plate code checking
- Vol 6 - Menu description
- Reference Documents
- Training Documents
- Command Reference
- JScript commands
- Tutorials
- Example Index
- Wizards
- Wizard templates

Readme.txt - Notepad

Readme.txt for the wizard for Genie scripting

- The wizard itself contains information on how to define inputs, how to store a journal file and how to open the Genie program using a saved journal file.
- The wizard includes a total of 5 different structure types (Tubular K-joint, a Box-joint, a Tanker (for panel modelling purposes), a typical corrugated Bulkhead and a semi-sub (for panel modelling purposes)).
- You can decide if you want to run with the latest official version installed (this is the default option) or an alternative program version (a development version or another official version).
- Read more under the Help in the wizard

Readme.txt for Genie Jacket and Topside wizards

These wizards are implemented as MS Excel files. The provided files are to be understood as templates for you to copy and modify.

How to use the wizards:

- Create a new Genie workspace.

For back-up purposes you are advised to create a copy of the journal file including your own comments (either created interactively during modelling session or manually added afterwards). This file is of great importance when recreating the model in newer versions of GeniE and if you need support assistance.

```
//Tolerances Rules
GenieRules.Tolerances.angleTolerance = 0 deg;
GenieRules.Tolerances.pointTolerance = 0.01 m;
GenieRules.Tolerances.useTolerantModelling = false;

***** STRUCTURE *****/
//Beams
St52.setDefault();
I200.setDefault();
Bm1 = Beam(Point(0 m,0 m,0 m), Point(10 m,0 m,0 m));
Bm2 = Beam(Point(0 m,0 m,0 m), Point(10 m,10 m,0 m));
Bm3 = Beam(Point(0 m,10 m,0 m), Point(10 m,10 m,0 m));
Bm4 = Beam(Point(0 m,0 m,0 m), Point(0 m,10 m,0 m));
Bm5 = Beam(Point(0 m,0 m,0 m), Point(10 m,10 m,0 m));
Bm6 = Beam(Point(0 m,0 m,0 m), Point(0 m,10 m,0 m));

//Plates
//Supports
Sp1 = SupportPoint(Point(0 m,0 m,0 m));
Sp2 = SupportPoint(Point(10 m,0 m,0 m));
Sp3 = SupportPoint(Point(10 m,10 m,0 m));
Sp4 = SupportPoint(Point(0 m,10 m,0 m));

***** GUIDING GEOMETRY *****/
//Guiding Geometry
GP_UserMANUAL = GuidePlane(Point(0 m,0 m,0 m), Point(10 m,0 m,0 m), Point(10 m,10 m,0 m));

***** EQUIPMENTS AND LOAD MODELLING *****/
//Definition of LoadCases
LC1 = LoadCase();
LC1.setMemLoadcase(1);
```

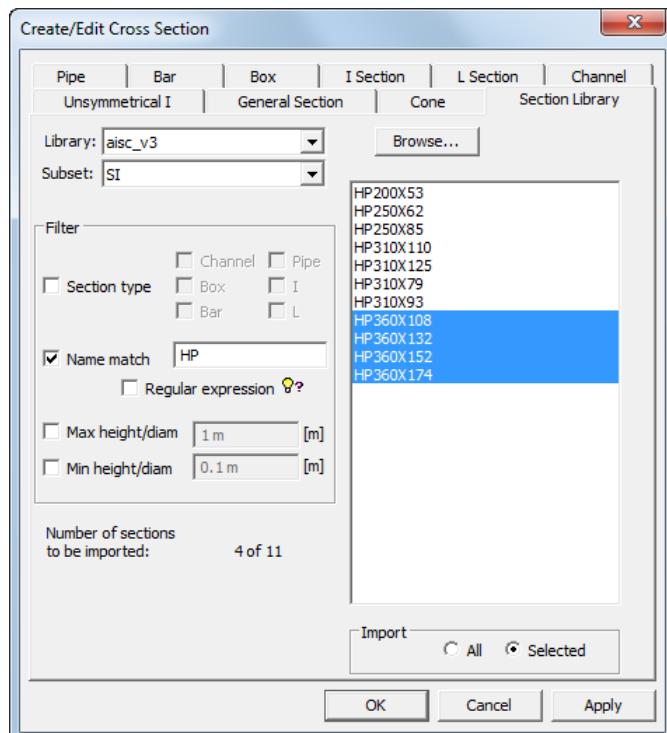
You may also create a clean journal file to easily reproduce your model (except for curved structure, imported weight lists and cut-outs in the structure). This file does not include the history on how the model was created. The clean journal file includes comments and it is easy to understand how it is built up. The picture to the left shows an example of a clean journal file.

The journalling system is documented on the help pages (**Help/Help Topics** -> Jscript commands). Further information may be found on <http://msdn.microsoft.com/scripting/default.htm?scripting/JScript/> and <http://msdn.microsoft.com/scripting/jscript/download/jsdoc.exe>.

## 2.19 Unit handling and section libraries

GeniE comes with a powerful unit handling system. The user may choose a preferred consistent set of units for the model work space – the database units. This must be done when the work space is created and this cannot be changed.

During modelling the user may choose to use another input set – the input units. GeniE will automatically convert the input quantity to data base units. When documenting a model, either graphically or textually, the input units are used. The results from a structural analysis are always shown in database units. The unit system is fully supported by the JScript implementation.



GeniE also comes with the AISC and Euronorm&Norwegian Standard section libraries. You may choose to import all or parts of these libraries. When importing section properties these are logged on the journal file meaning that you don't have to import every time you recreate a model using the journal file as input. The profiles are logged on the journal file according to the input units you have specified.

The example to the left shows that only a few of I-profiles with a given name HP\* shall be imported for further usage in GeniE. The profiles are found in the AISC section library.

## **2.20 Transfer of a model through the Input and Results Interface File format**

GeniE creates a finite element model, the \*.FEM file, when you make a finite element mesh and when you run an analysis. This file can be exported to other SESAM programs, either to Sestra for more advanced type of analysis, or read into Presel for inclusion in a superelement analysis (supernodes in GeniE are then required). The \*.FEM file contains the concept information that some SESAM programs (e.g. Framework) can understand. Other programs are not capable of using the concept information since they are pure finite element based programs, examples of such are Presel, Patran-Pre, Usfos and Xtract. In such programs, the finite element model forms the basis.

Following a linear analysis performed in GeniE (Sestra running in background) a result file \*.sin will be created. This file can be read into Framework for e.g. beam code checking, Platework for plate code checking and Xtract for more advanced postprocessing.

## **2.21 Import and export from other systems**

GeniE can import/export data and build a concept model from SESAM or other systems:

- Preframe, Prefem, Patran-Pre – structure & loads - reading SESAM's Input Interface File \*.FEM
  - From Ansys (neutral file \*.cdb) when Prepost has been used to create a \*.FEM file
- SESAM – structure & loads – write SESAM's Input Interface File \*.FEM
- Sacs – structure & loads – reading neutral file \*.inp
- StruCad3D – structure & loads – reading neutral file \*.s3d
- Spatial Technology SAT file – structure only – read a neutral file \*.sat
- CadCentre PDMS – structure only - read and write neutral file \*.sdn (also referred to as sdnf file)
- Intergraph PDS – structure only – read and write neutral file \*.sdn
- The AISC Section library, version 3 – read a database
- The Euronorm and Norwegian Standard Section library – read a database
- Equipment data from weight list systems – read a \*.csv or \*.xml file created by a weight list system
- Documenting the model and loads using \*.txt, \*.xml or \*.html that can be accessed using e.g. MS Notepad, MS Excel, MS Word or any Internet Explorer.
- Graphic files for inclusion into other office systems - \*.gif, \*.eps, \*.bmp, \*.tga, \*.tif, \*.ps, \*.vrml, \*.dfile

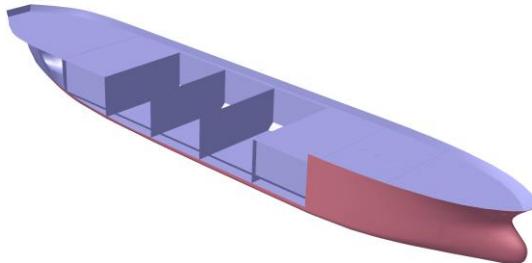
## **2.22 Panel modelling**

GeniE may also be used to create panel models to be used in stability or hydrodynamic analysis by HydroD. Furthermore, the panel models may be used in coupled analysis in DeepC. Common for all such analysis is that they require a panel model in addition to what is referred to as wetted surfaces.

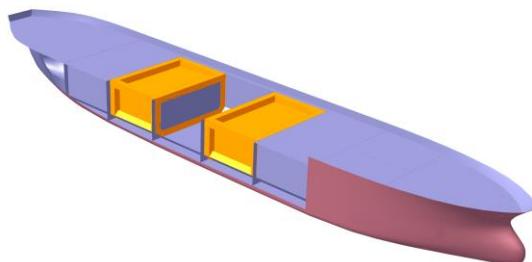
The hull form of a floating vessel is often built up from double curved surfaces. GeniE can handle such curves either from a direct import using existing structure as found on a SAT file or by a manual definition of all cross sections followed by a plate skinning operation.

It is also possible to define tanks whereby you can specify tank fillings during stability analysis in HydroD.

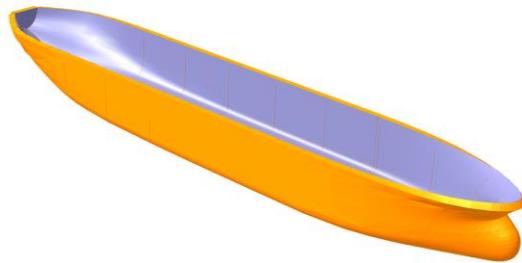
The pictures below show an example where the hull has been imported from a SAT format and tanks have been defined.



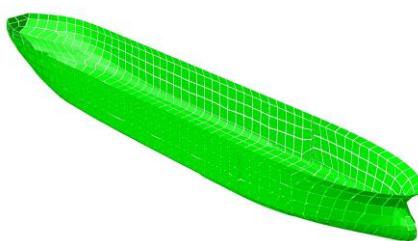
A ship hull form imported from SAT format. Parts of the structure removed for visualisation.



Two internal tanks have been modelled. These tanks may be referred to in HydroD and tank filling may be specified there (filling fraction and fluid density).



The outer surface has been assigned wetted surface such that hydrodynamic loads and accelerations may be computed in HydroD (and Wadam). These loads may be transferred back to the finite element model (see picture below) for subsequent analysis. Such analysis cannot be done from GeniE's interface, but should be done using SESAM Manager.



The finite element model automatically created by GeniE using default settings.



## 3. USER'S GUIDE TO GENIE

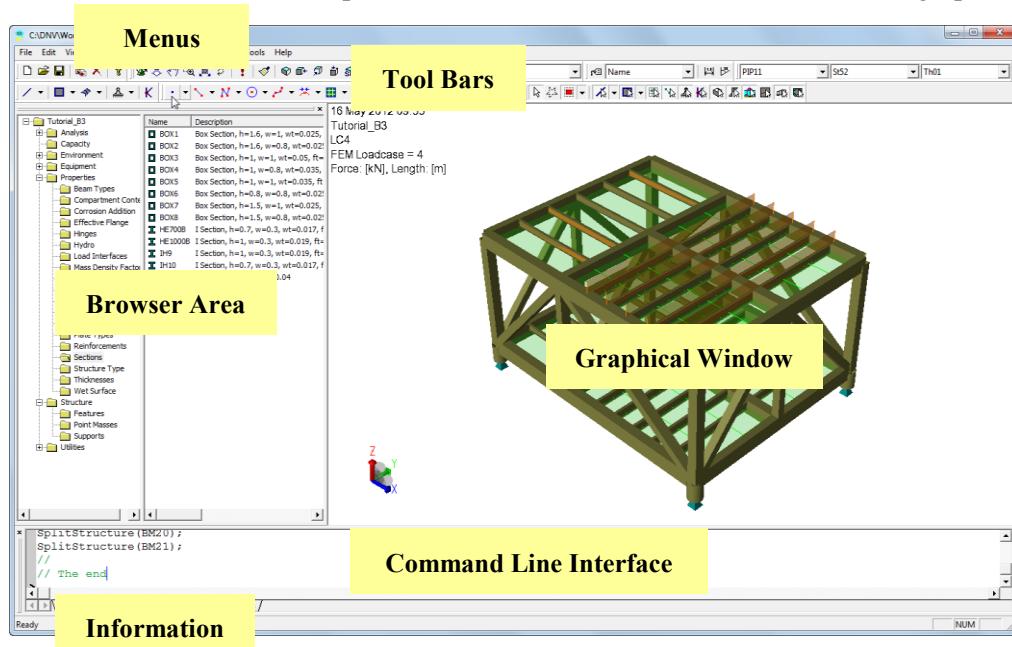
This Chapter focuses on how to use the program. The Chapter had been built up according to the sequence of steps you would normally follow if you were to do a full design analysis. The first Section describes the basics on the user interface and how you can interact with the program when doing modelling, verification, and documentation of your model.

### 3.1 Getting started

#### 3.1.1 The Graphical User Interface (GUI)

The GUI consists of 5 main parts - the menus, the tool bars, the browser, the graphical window, and the

command line  
interface (CLI). In  
addition there is also  
an information field:



Tool bar area.

Graphic modelling and interpretation of the model is performed in the *Graphical Window*.

A tabulated view of the model is available from the *Browser area* – here you may also edit and change the model.

The journaling system is running in the Command Line Interface – here you type in explicit commands if you want to do modelling using a line based input approach.

#### 3.1.2 Starting up

You create a new project or model (termed workspace in GeniE) by

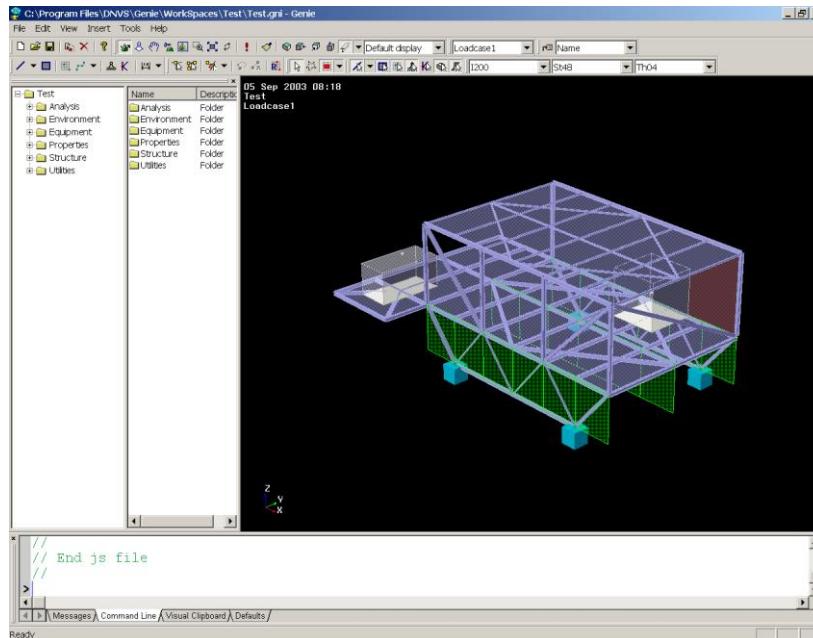
**File/New Workspace <model name>**

This will create a directory on your disk c:\ DNVGL\Workspaces\GeniE\<model name> when using default installation settings. All files and information for this model is stored under this directory.

You can save, close, and open up the same model.

Note that when you make new workspace you need to decide the database units and whether you want tolerant modelling to be default, see Chapter 3.2 for further details. It is not possible to change the database units during modelling.

For the remaining parts of Section 3.2 you may want to have access to a model while reading the content. For this purpose you may a) start the program (see Chapter 4 on how to start GeniE), b) create a new workspace and call it e.g. Test, c) save a predefined model that you find on the on-line help system (F1, then find a file under Tutorial 1 called Small\_topside.js and save to your PC), d) read in the file by using command **File|Read Command File <path>Small\_topside.js**, e) save the model by **File|Save**. You now have a model (with model name Test) where you can test out many of the features listed in the following. The model should look as follows:

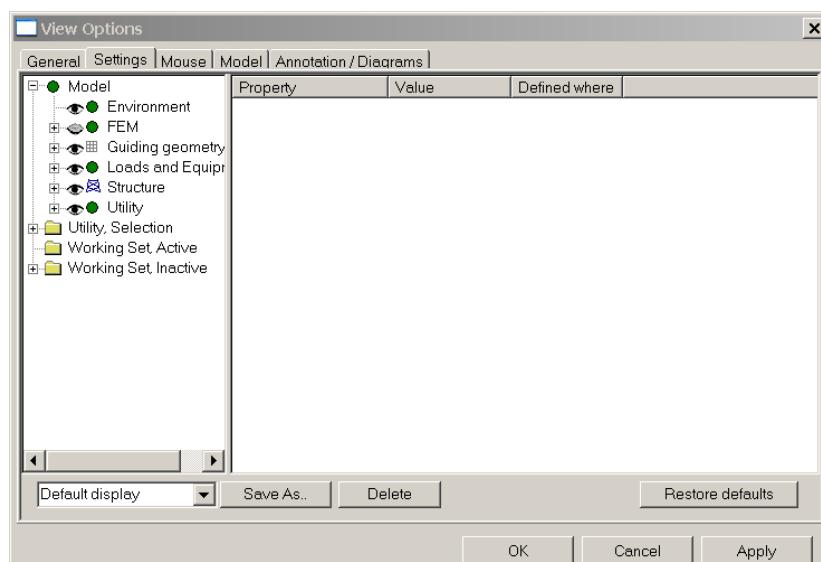


The picture shows GeniE with default setting enabled for graphic viewing. In the following several options on how to change these are mentioned.

If you want to test these out and you for some reason want to return to the default settings, you may do this by **View|Options** and select **Restore Defaults**.

### 3.1.3 Setting up your preferences

You define your own user profile via the menu **View/Options**. Here you can change system settings for the *display*, the *categories*, and the *cursor feedback*. The changes you make here are permanent, meaning that the new system settings also apply for new modelling sessions you do in GeniE.



General – colours and appearance of the user interface

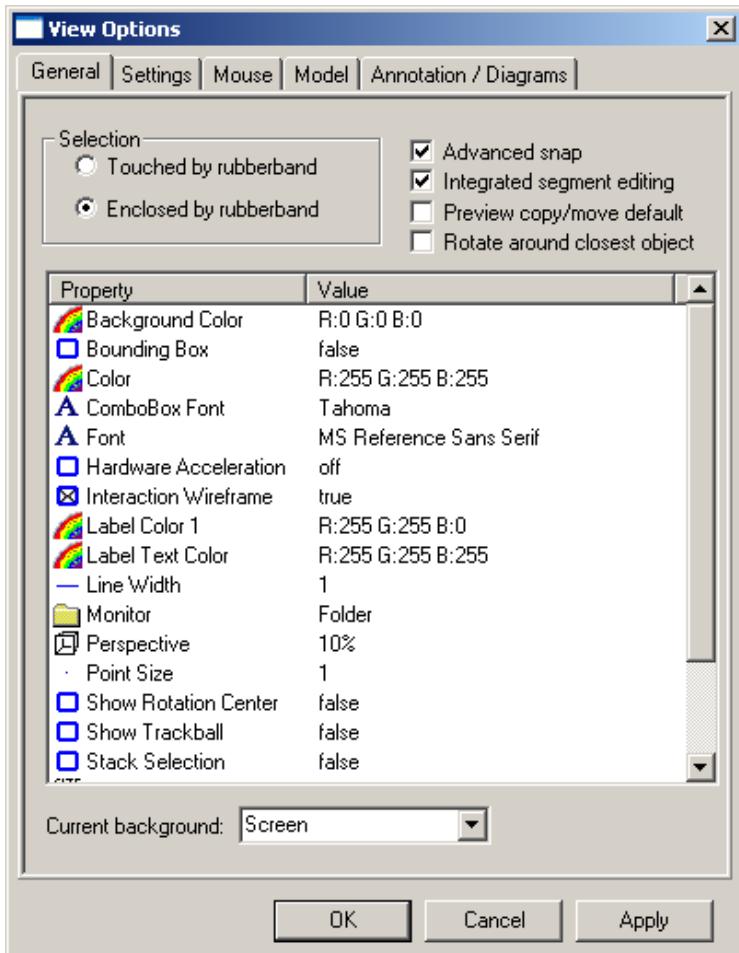
Settings – predefined model views

Mouse – cursor tooltips and feedback

Model – how to view the model and use of property colour coding features

Annotation – labelling of global model

### 3.1.3.1 Change the appearance of the user interface



You may double-click any of these attributes to change settings.

If you have a slow/medium PC, you should at least activate the tick-off option for Interaction Wireframe – this will show your model in wireframe view when moving, rotating, zooming etc.

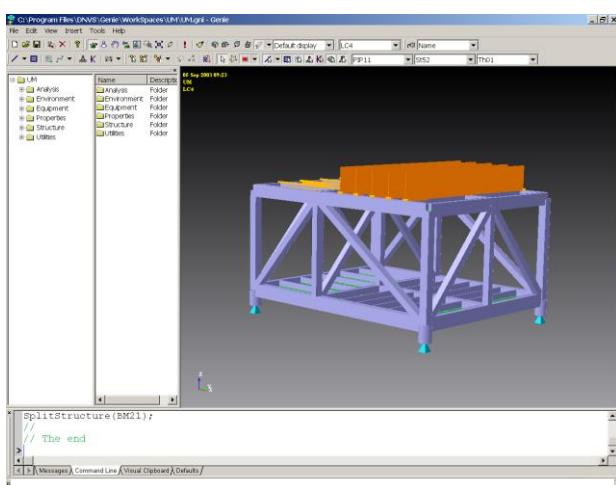
Also observe that if you want a preview of copy or move operations, you may set this to your default option by ticking the right box.

You may also want to change the background colour, or appearance of text.

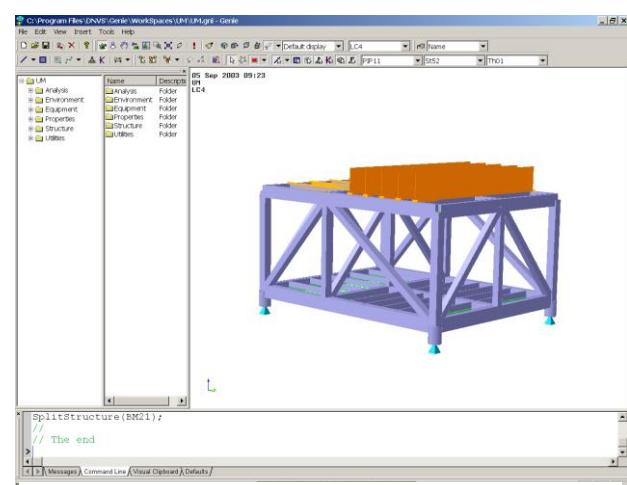
When producing a graphic file, it may be beneficial for you to change Current background from Screen to paper – this will in many cases make a better view when the picture has been imported to an office system like e.g. MS Word.

Note also the tick-off options for default preview when copy/move and for setting the rotation centre.

Below is showed an example where background colour has been changed, the text font is changed to Times New Roman 14, bold, and yellow. Also is showed how it appears when switching background to Paper.



View model when background is set to Screen



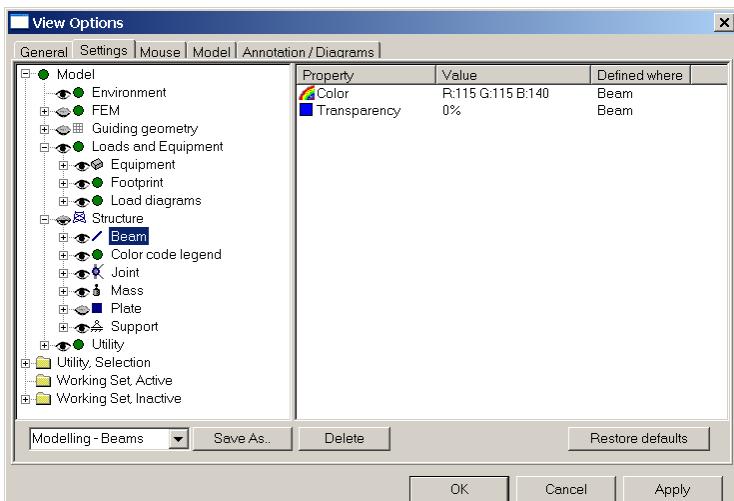
View model when background is set to Paper

### 3.1.3.2 Change and make new predefined model views

GeniE comes with a set of predefined model views – these focuses on different attributes and results:

- Default display, shows all except finite element and results
- Mesh – All, shows all mesh information except results
- Mesh – Transparent shows mesh, but no finite element loads. Shell elements are transparent
- Modelling – All, same as default display, many users remove Guiding geometry from this view
- Modelling – Structure, shows structure but no guiding geometry or loads
- Modelling – Transparent, shows structure where plates are transparent plus equipments and their footprints, no loads
- Results – All, shows all results but no structure or finite element mesh
- Results – with mesh, shows all results including the finite element mesh

You may change or delete any of these. Using Restore defaults will override any of your changes.



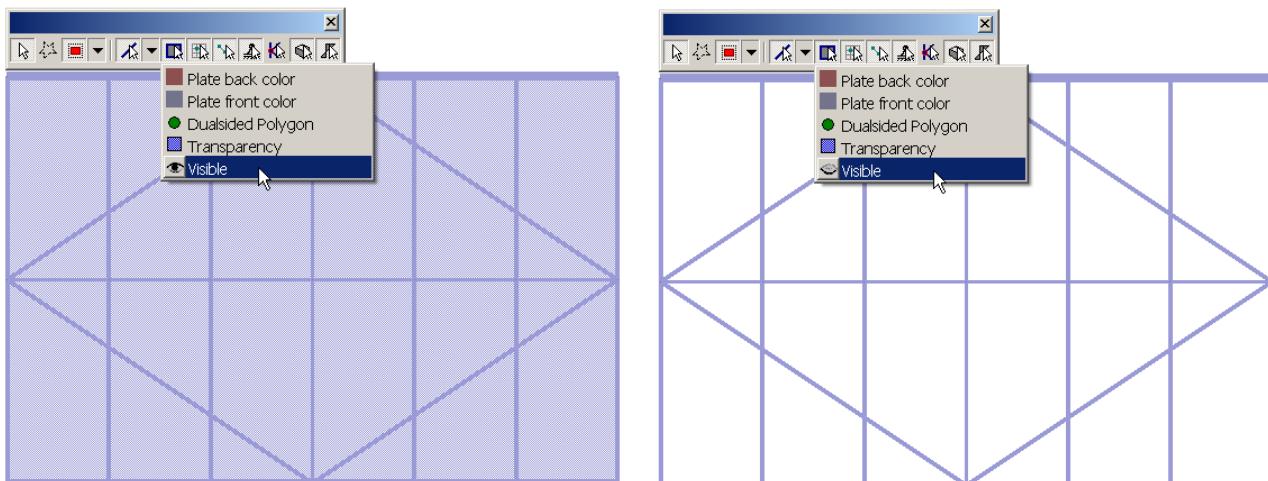
In the example to the left a new model view Modelling – Beams has been created from saving the Modelling – Transparent and removed visibility of plates. As can be seen, the eye symbol is closed.

Furthermore, the colours of the beams are changed to Red:155, Green:115, Blue:140.

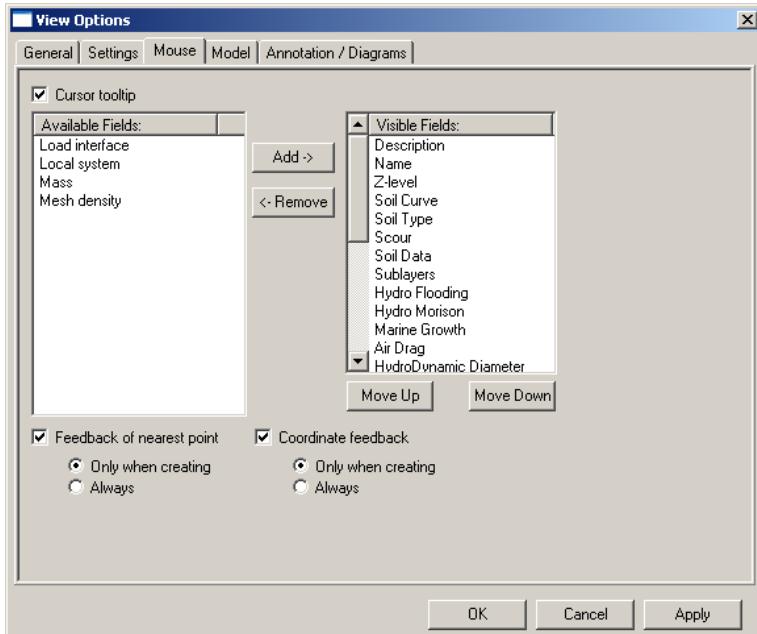
You should be careful in using red colour since any selected object is highlighted in red.

The new model view is permanent so that you can use it in other GeniE sessions as well.

You may also change the model views directly from the toolbars. You click with the RMB on a filter tool button to get access to the quick menu to the available graphic settings for the object. When doing so, the current view settings as found under ***View/Options*** are automatically adjusted accordingly. Below is an example where visibility of plates has been deselected.



## Change actions associated with the cursor



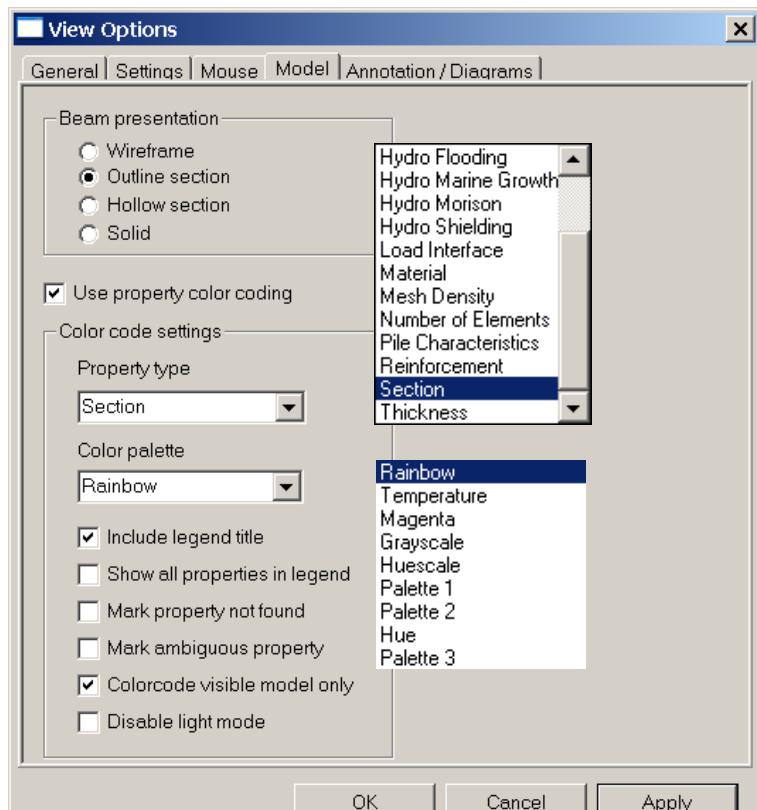
You may decide to see relevant information when moving the cursor over an object. For example, using the settings to the left will result in the following graphical feedback of a beam:

- Type of object
- The name
- Section
- Material

The mouse feedback is context sensitive, i.e. the feedback depends on which type of object is in focus.

The coordinate feedback is showed in the lower left corner of the GeniE user interface (the *Information area*).

### 3.1.3.3 Change graphical views and property colour coding



There are a number of ways to view your model:

Wireframe will draw a beam as a line disregarding any section property. Plates are not showed as such except for the boundaries.

Outline section will draw the section as is, but with no thickness information. The plates are also drawn without thickness.

Hollow section will do the same as outline, but in addition the boundaries of thickness are included.

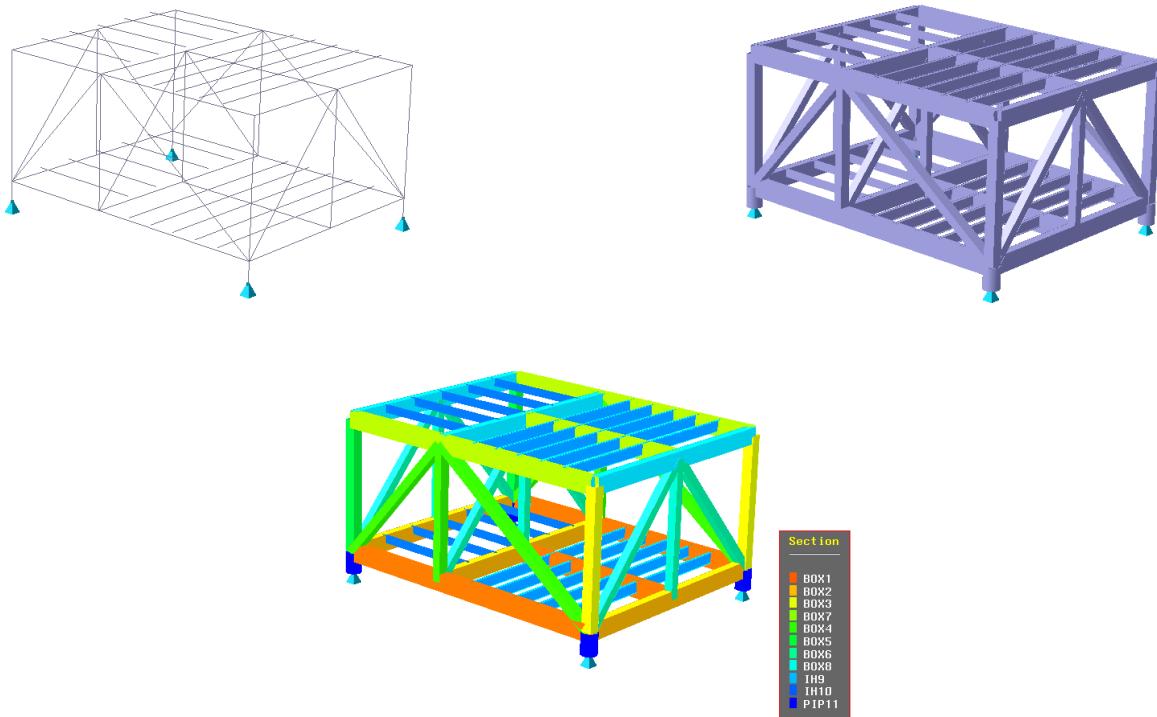
Solid will draw a section in full including thickness view. Plates are showed without thickness.

Observe there is a Toolbar pulldown button that gives you a quicker access to switching beam views.

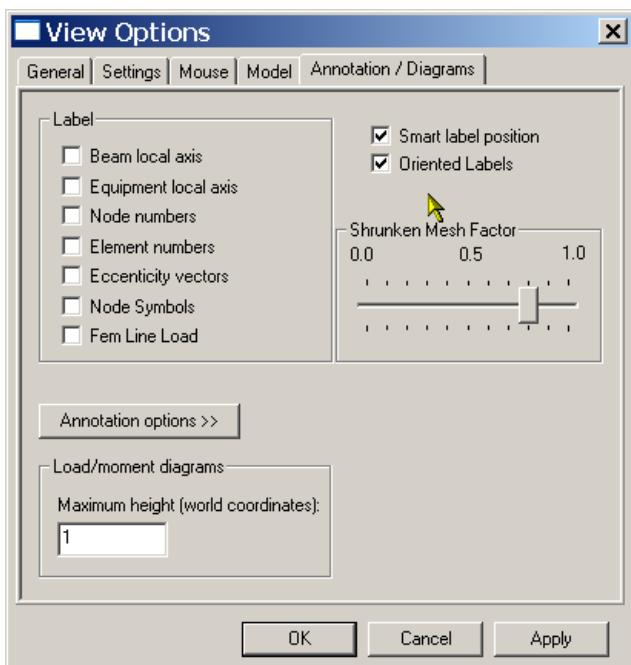


The property colour coding applies for the attributes shown above using a set of predefined colour palettes.

The examples below show a model viewed in wireframe, solid, and outline including colour coding of section types. All pictures have been produced using option *Paper background*.



### 3.1.3.4 Global labelling



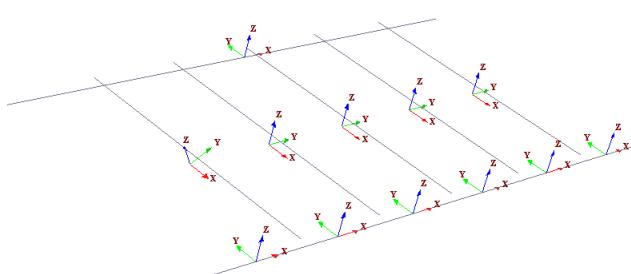
You may label the global model (or the selection as seen in the graphic window) and label certain attributes to it. You can add the following information to your model

- Local axis of beams
- Local axis of equipments
- Node numbers
- Element numbers
- Eccentricity vectors
- Node symbols
- Shrunken mesh factor when displaying the FE mesh
- Ensure that labels are not stacked on top of each other (smart label positions)
- Make labels oriented along beam axis (oriented labels), names only

Maximum height (world coordinates) controls the appearance of load symbols.

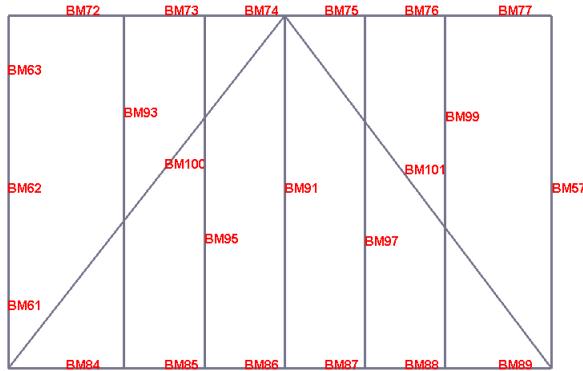
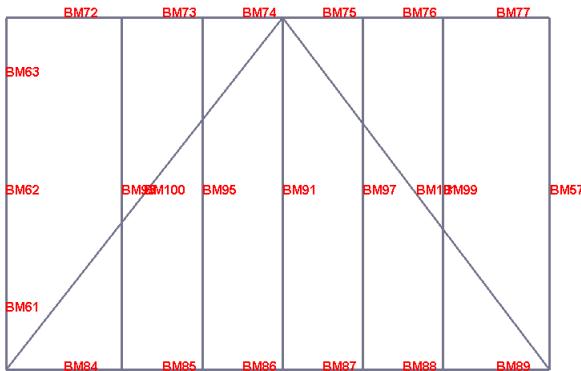
You may also do labelling per individual object(s); see later for a description on how to do it.

In the examples below are showed beam local axis (one of the beams have been rotated around local x-axis) and node and element numbers for a set of beams (note that the default colours for node and element numbers have been changed to red and blue in this case).



15	35	30	58	36	68	42	78	48	90	56	101	62	110	66
17		51												
13	32	28												
14		48												
11	30	26												
12		46												
6	28	24	56	34	66	40	76	46	88	54	98	50	100	85
9			43											
8	26		22											
7		41												
5	24		20											
4		38												
0	22	18	54	32	64	38	74	44	86	52	96	58	106	83

The example below shows the differences from using *Smart Label Positioning* or not. The picture to the left shows labels stacked on top of each other, while in the picture to the right (using Smart Label Positioning) all labels are separated from each other. For both pictures, label text colour has been set to red, and font changed to *Arial* with size 20.



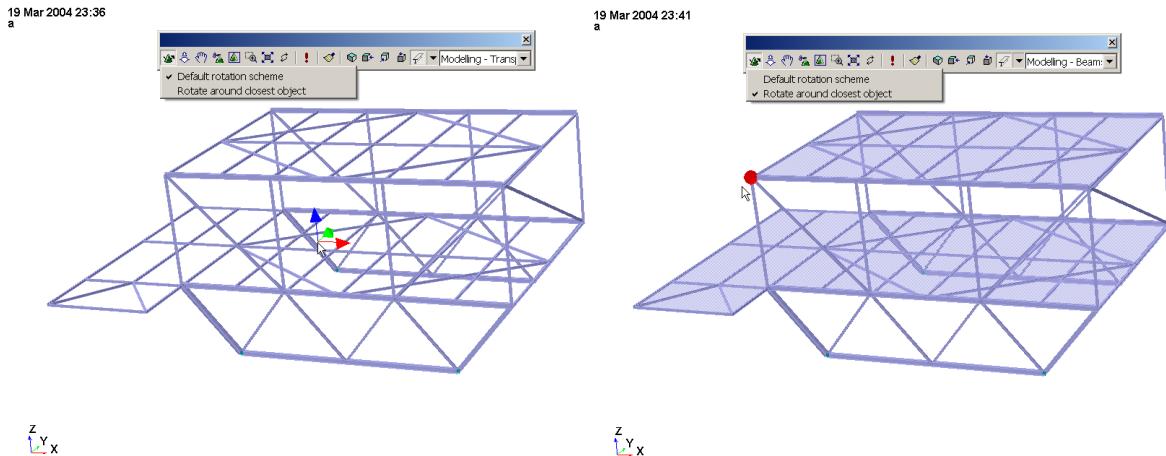
### 3.1.4 View manipulations (LMB & RMB)

The left mouse button (LMB) is reserved for selections (including rubberband selection), input fields, and graphic modelling. The right mouse button (RMB) is used for view manipulations like rotate, zoom, and pan.

When a cursor is placed above a selected item, the use of RMB will force the context sensitive menu to appear. The use of TAB+RMB will always force the context sensitive menu to appear.

It is possible to decide on which position to rotate around. The examples show rotation around

- the system default rotation centre (governed by global coordinate system and the size and location of the model), and
- a user defined position by selecting an object to rotate around.



To access the quick menu for selecting rotation scheme, click RMB on the tool button for rotation. You may also change these settings from *View/Option/General*.

### 3.1.5 Selecting objects

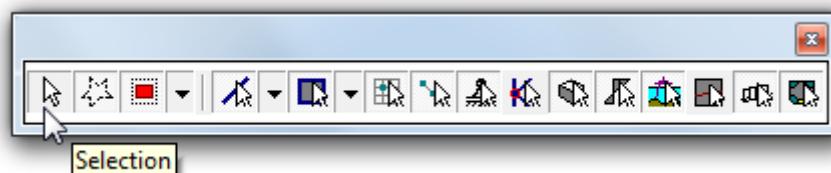
When selecting objects that are overlapping, the following prioritization order will be used, top priority first:

1. guide point
2. all other concepts
3. beam
4. guide curve
5. plate
6. guide plane

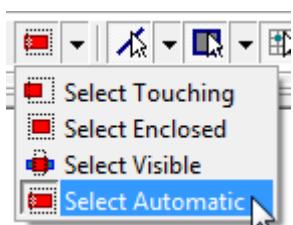
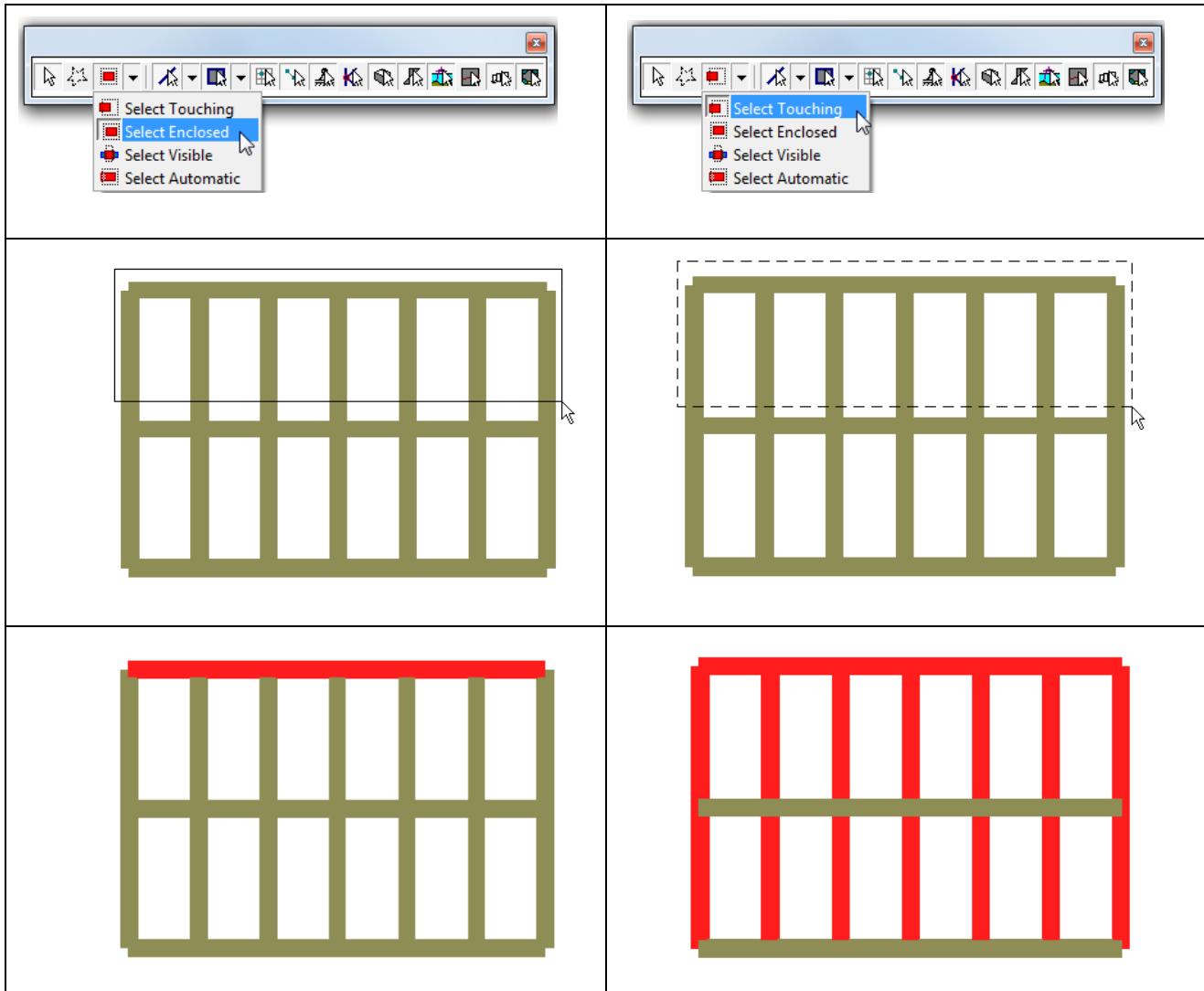
There are two ways of selecting objects (beam, plate, equipment), graphically or via the browser. A combination of these two is also possible. You may also select the finite element mesh while in graphic mode.

#### 3.1.5.1 Graphic selection

Single objects (e.g. beam, plate, equipment) are selected graphically by clicking the object. If you want to select more than one object you can click (LMB) the first one, then shift+click on the additional ones. Note that you need to set GeniE to selection mode by activating the selection button showed below.



You may use rubberband selection to select either all objects touched by or enclosed by the rubberband. You can switch between these two options using the rubberband button on the selection toolbar. The examples below show the difference between enclosed, touched by, select visible and select automatic. Note that all selected items are highlighted in red colour.



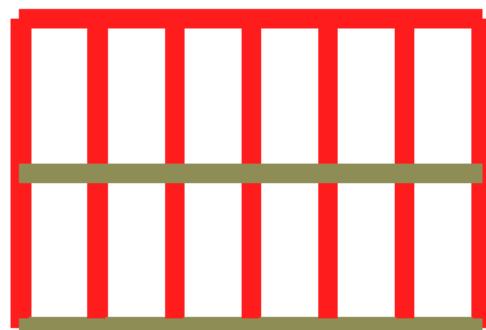
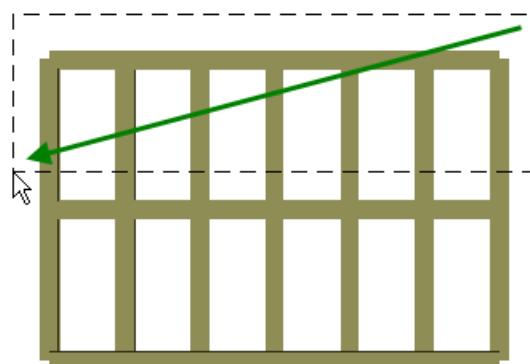
Select Automatic gives you the option of choosing between Select Touching and Select Enclosed based on how you move the mouse when making the selection.

## How to use Select Automatic:

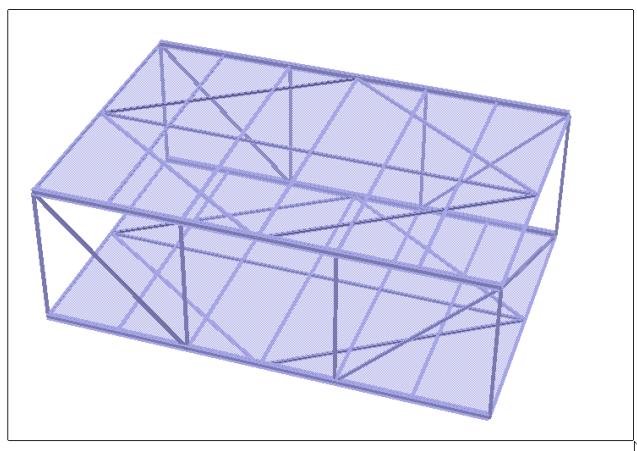
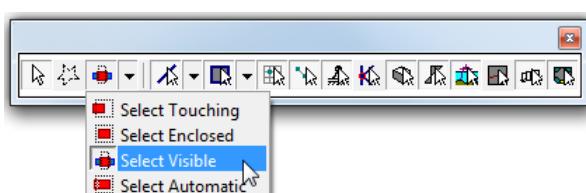
Dragging the mouse pointer from left to right selects enclosed:



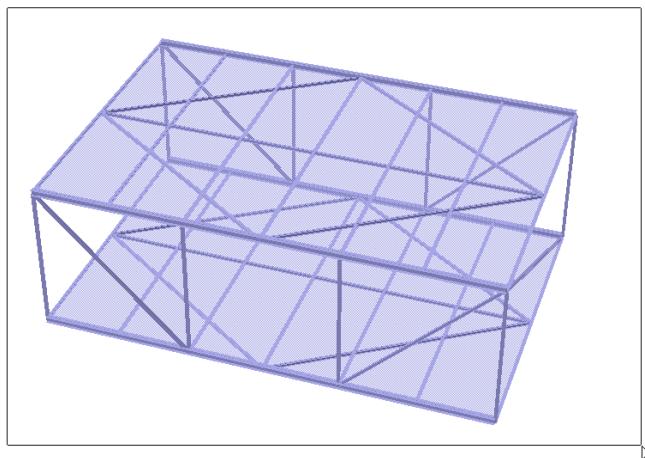
Dragging the mouse pointer from right to left selects touching:



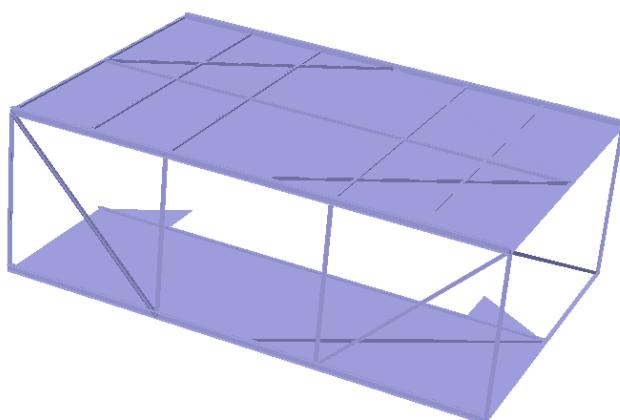
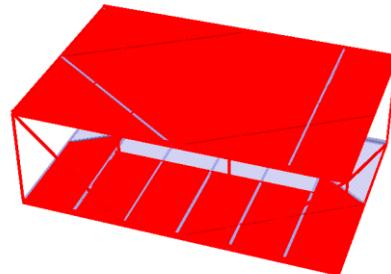
Observe that these selection methods will select all objects (visible or not) that are inside the rubberband. If you want to select only what you have selected, you should use the option *Select Visible*.



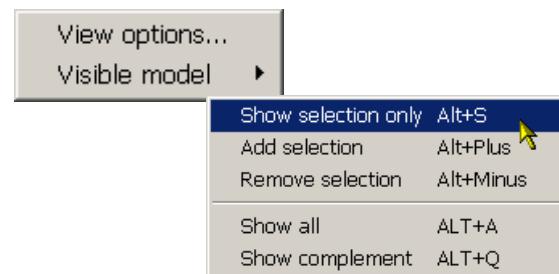
The model consists of plates and beams. When viewing from this eye-position, some of the beams and plates are not visible. In this example they are visible because transparency has been used for illustration effects.



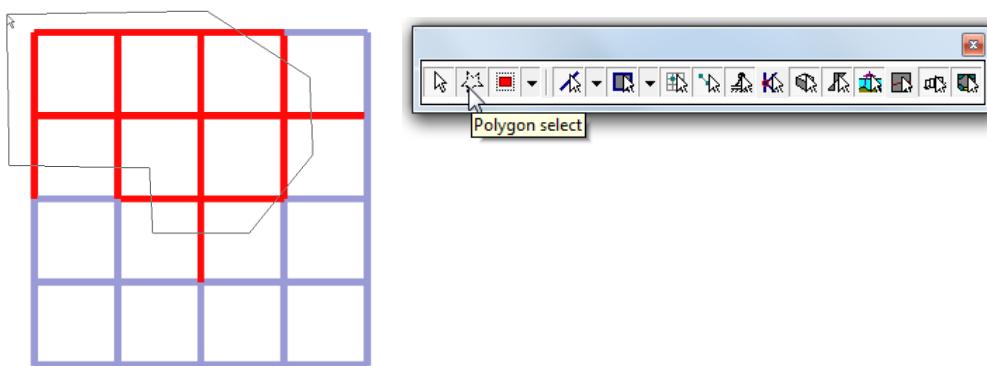
When making a rubberband select using the *Select Visible* option, the non-visible structural parts on the bottom level are not selected.



By using ALT+S or use the commands shown below, you visualise those selected. As can be seen, parts of the lower level are not selected.



The example above assumed using rubberband selection. You may also use the polygon selection. In this example this method is used together with rubberband *Select Touching*.

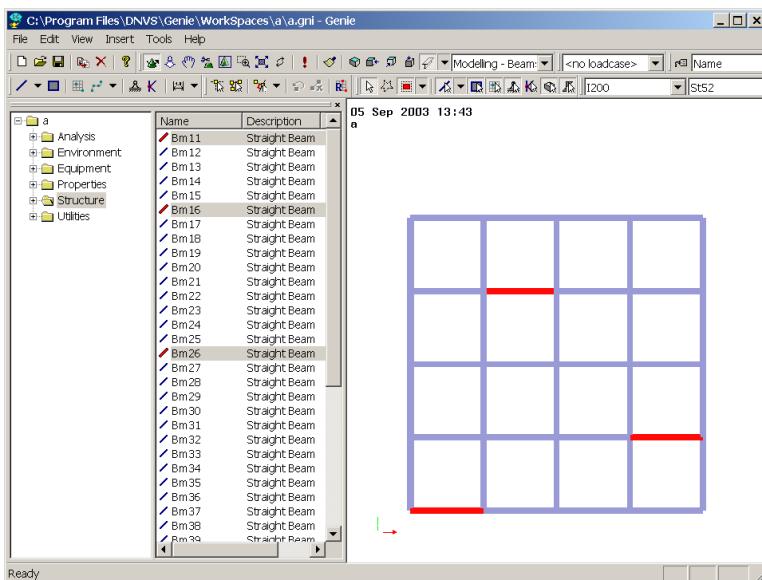


Note that you can also use Polygon Select together with *Select Automatic*. In this case if you make a selection by clicking points in a clockwise pattern you will select objects enclosed by the rubberband. If you make a selection by clicking points in a counter clockwise pattern you will select objects touched by the rubberband.

### 3.1.5.2 Browser selection

You may select objects from the browser also – the selection is Windows compliant. You may compare it with how you select in Windows Explorer:

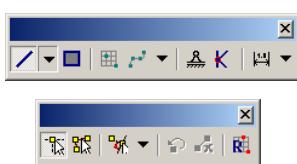
- Click (LMB) on individual object
- Click (LMB) and shift+click (LMB) to select all objects in-between a list.
- Click (LMB) and shift+arrow (up or down) to select all objects in-between a list.
- Click (LMB) and ctrl+click (LMB) to select individual objects



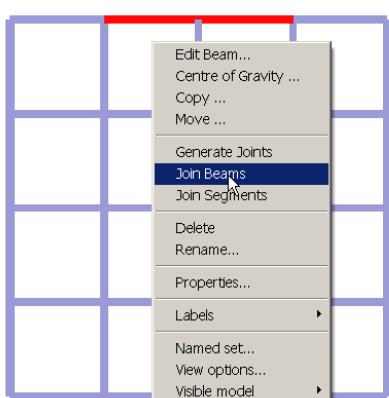
This example show that Bm11, Bm16, and Bm26 have been selected using the Ctrl+click option. Note that selected items are automatically highlighted in the graphic window (and vice versa if graphic selection is performed).

### 3.1.6 Graphic modelling and editing

Graphic modelling is done by deciding the task to be performed and then the necessary click operations. For example inserting a beam graphically requires that activate insert straight beam and then click between two points.

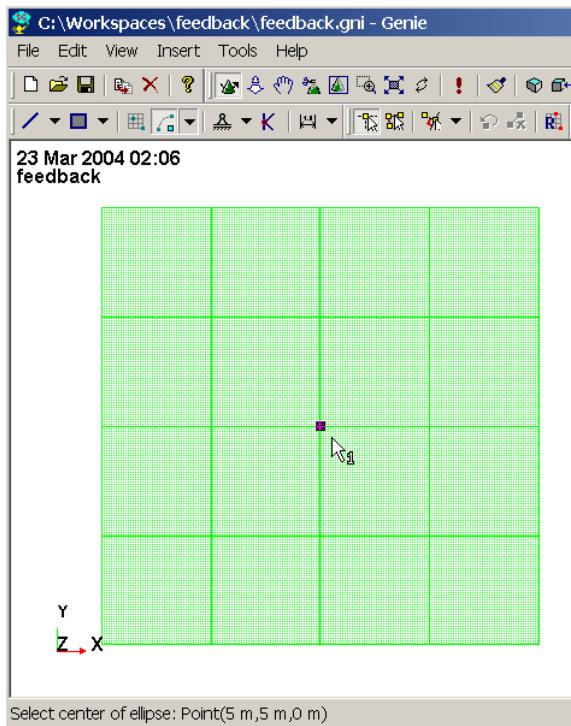


From the Toolbar you click Insert Straight Beam and make sure that GeniE is in modelling modus. Normally, when clicking *Insert Beam* (or other) this will automatically switch GeniE to modelling modus. You may also do the same from the pull-down menu by using command **Insert|Beam|Straight Beam Dialogue**.



Graphic editing requires that you have selected an object(s) and to enforce the context sensitive menu to appear. The example to the left shows that two beams have been selected, the context sensitive menu has been activated, and command Join Beams is chosen.

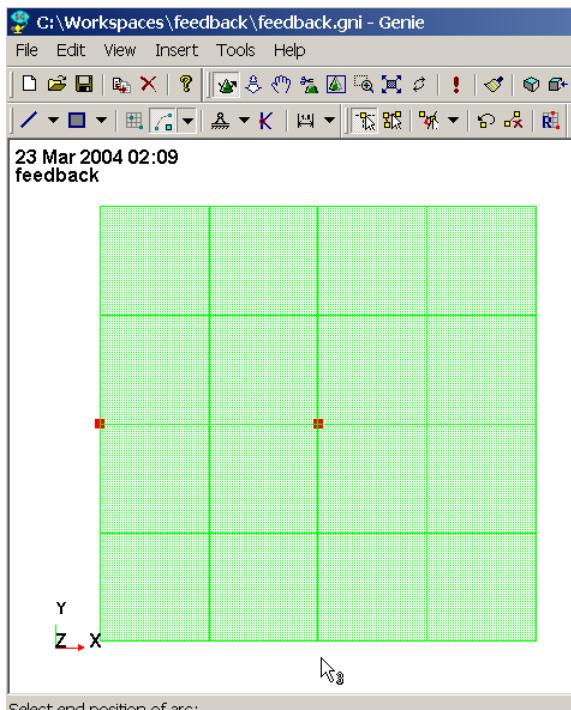
GeniE will query for expected input in the status bar (at the bottom left part of the GUI). The sequence of pictures below depicts how the program will ask for expected input. The example is based on inserting a guide spline requiring 3 input parameters – origin, start and end positions.



#### *Step 1: Insert the origin*

GeniE will query at the status bar: “Select center of ellipse”.

Since the mouse is “touching” a snap point, the status bar will also report the actual coordinate values.



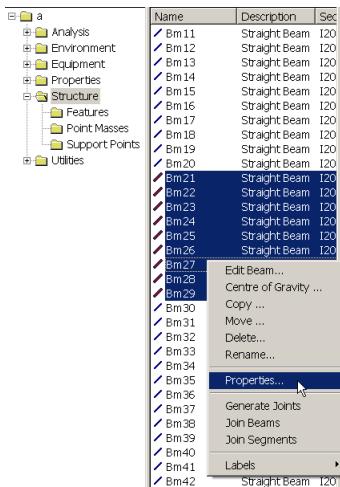
#### *Step 3: Insert the end position*

GeniE will query at the status bar: “Select end position of arc”.

Notice that in this case the mouse is not “touching” a snap-point; hence no coordinate values are listed on the status bar.

### 3.1.7 Browser modelling and editing

The browser works the same way as Windows Explorer and the browser comes with predefined folders for Analysis, Equipment, Properties, Structure, and Utilities. A double click on a folder will expand it.

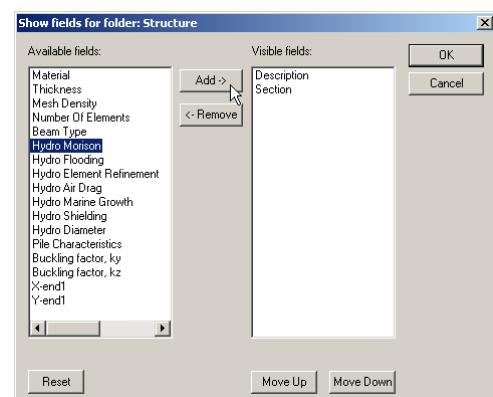
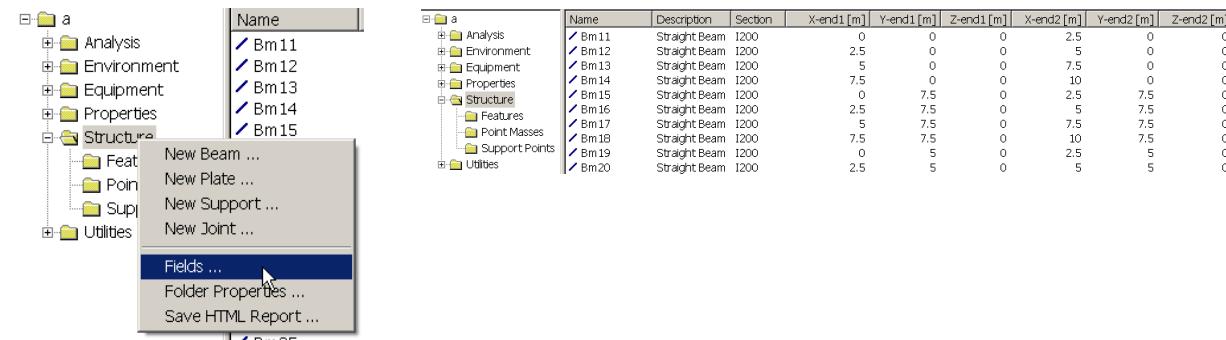


When you right click (RMB) in the browser area you activate the available commands. For example, when right clicking while in the Structure folder, you get access to commands for defining a new beam, plate, support or joint.

Selecting objects in the browser and the use of RMB will force the context sensitive menu. You can then make changes or operations to the objects selected. The example to the left shows that some beams have been selected, RMB to force the context sensitive menu, and Properties have been chosen

The browser is also powerful with regards to sorting. You click the headers to sort the actual column. This may help you in selecting particular objects in a very efficient manner. One example is to sort all beams with a given section, select these and change section for all in one command.

You may also customise the browser. GeniE comes with a predefined browser setting; you may add or take away information as needed. Please note that such changes are not permanent, i.e. the changes only apply to the current GeniE session. The example below shows how you can add a new column, for example coordinate values for beam ends.



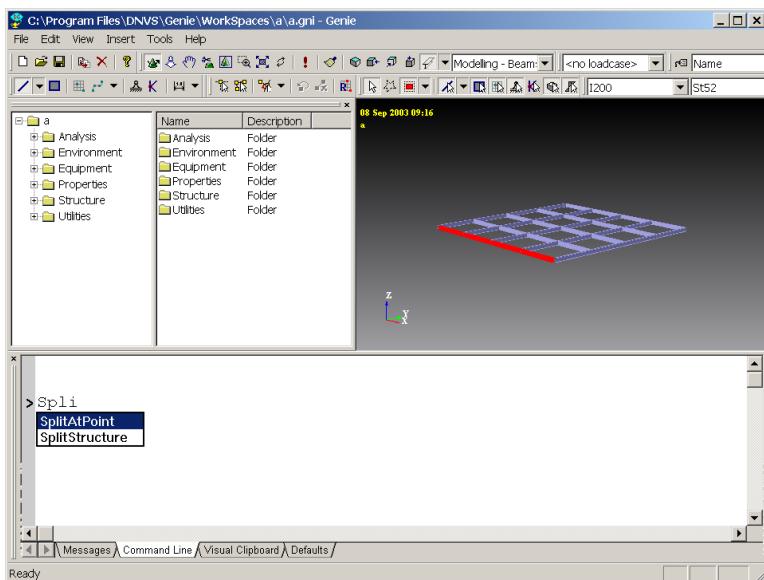
The picture to the left shows the fields you may add to the right tree of the browser.

In this example the field *Hydro Morison* shall be added.

From the left tree of the browser you can also efficiently change properties. One example is changing the section for selected members. You select the objects, open up the catalogue *Properties* and *Sections*, then select the desired new section and right click on it, before you select **Apply section to selection**.

### 3.1.8 The command line interface and journalling

All commands, either given graphically, from the browser, or from the pull-down menu are logged on a journal file. The language of this file is based on the J-script language and enhanced with the commands necessary for operating GeniE. All commands are easy to understand, they are documented under the On-line Help system and you can use this to edit a complete input file. Many use it to specify their own property and equipment library and re-use this from project to project.



The journal file is created and read by the command line interface system (CLI) running GeniE, you see this window at the bottom of the GUI. You can type the commands directly from this window, if you don't remember the complete command you can type parts of it and push TAB to get further guidance. See the picture below where a beam has been graphically selected and a split operation shall be done.

A command is activated when pressing ENTER.

You may also give comments in the CLI while running an interactive session. You type // <your text>.

When you model you should build up a clean journal file, i.e. a copy of the most current active journal file where you remove unnecessary actions and insert comments for historical reasons. You may edit this file by using a standard text editor. This file is important for back-up purposes and also when you want to recreate your model in newer versions of GeniE – the compatibility over program versions is on journal file level.

The screenshot shows a Microsoft Internet Explorer window displaying the Sesam Genie Help documentation. The address bar shows 'C:\Program Files\DNVS\Genie\Help\Modeler\_Help.html'. The left sidebar has links for User's Guide, Tutorials, Command Reference, and HowTo-Videos. The main content area is titled 'Function Detail' for the 'Beam' command. It includes the command signature 'BasicBeam Beam(Point p1, Point p2, [....])', a description 'Construct a new straight or curved beam from the given points', parameters 'p1 - 1st point on Beam', 'p2 - 2nd point on Beam', and '... - Specify more points if you want to create a curved beam', and an example 'Bm1=Beam(Point(0,0,0),Point(0,10,0)); //Create straight'.

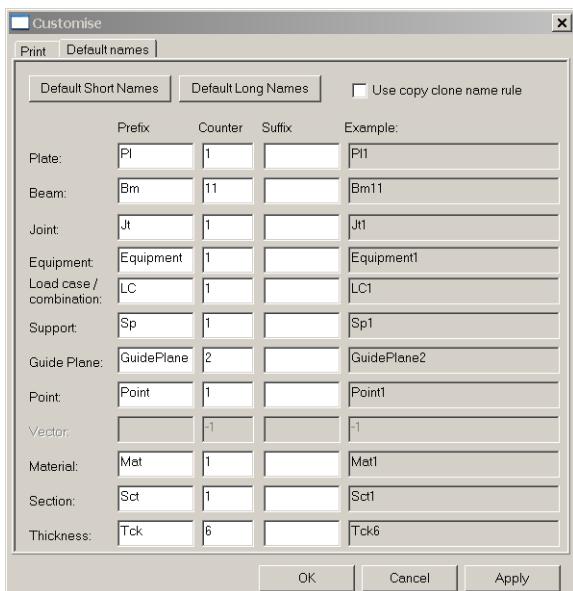
A journal file is read into GeniE using the following command **File/Read Command File**. You may read in several journal files to the same model, but they will be logged as one current journal file.

The picture to the left shows how the command Beam is documented on the On-line Help System.

When you give a name or fill in an input field, the input is validated continually. Wrong input is marked with a red cross.

### 3.1.9 Defaults and naming

The system comes with prefix and suffixes for all the names. You can change these to your preferences at any point via **Tools/Customise**:



All modelling following change of prefix or suffix will receive these names. Active use of customising names may ease the work to make sets and to sort in the browser.

Observe that names cannot contain characters that are recognised by J-script as operators. This means you cannot use e.g. +, -, \*. In addition you cannot use a blank (or space in a name).

Instead you may use \_ like e.g. *My\_beam\_1*.

Note also that you are limited to 8 characters if you want to transfer concept names to Framework or set names to other SESAM programs.

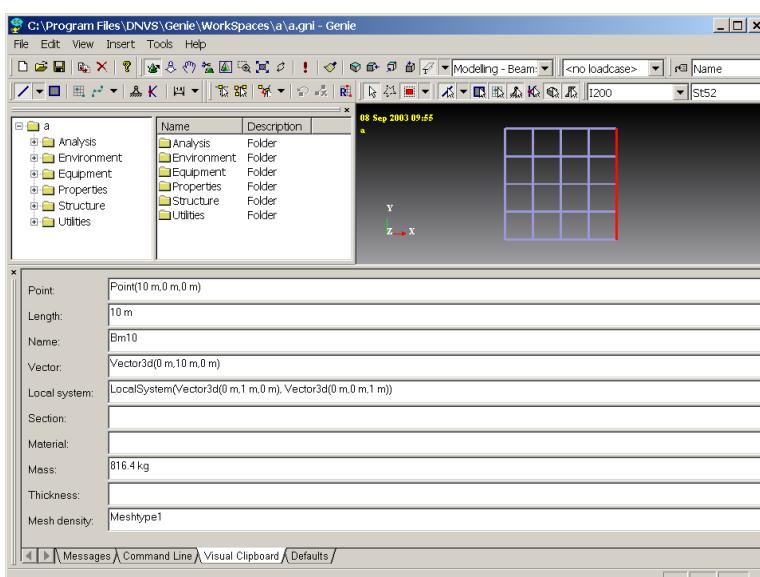
Default long names will give longer names, for example Prefix <Beam>.

Copy clone name rule gives a naming scheme referring to original objects when copying. For example Beam1\_1 is the first copy of Beam1. This naming scheme was used for GeniE version1 releases. You need to use this option when you read in journal files created by such version.

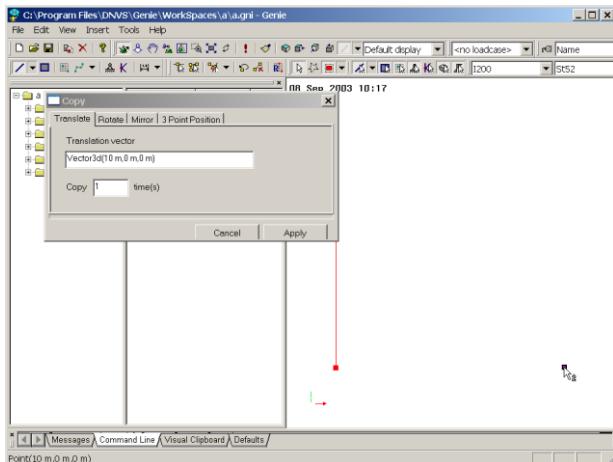
### 3.1.10 Using the clipboards

There are two ways of finding information from your model and use it in input fields. You can either use the visual clipboard or do it on the fly. The picture below shows that information has been found in the visual clipboard for the selected beam. The way to find the information is to click on the information area and then click the desired object. For example, to find the length between two points you click in the information field, then click first point and finally click second point.

The information can now be pasted into other input field or to the command line interface area.



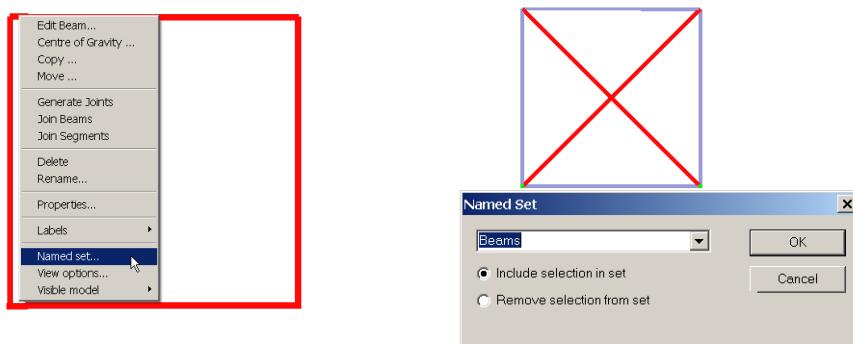
You may also want to find information directly when you are in a modelling modus. For example you want to copy a beam from a point to another point. You then start your copy operation, and find the copy vector graphically, see example below.



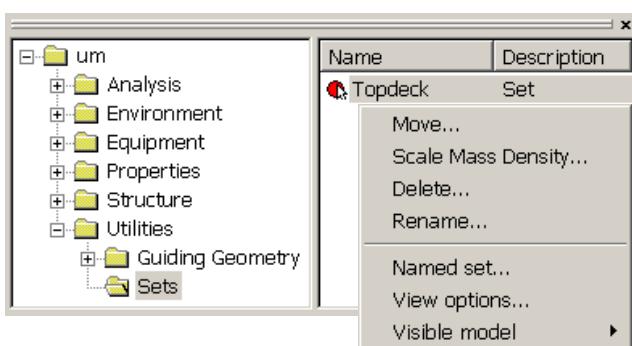
Select the beam you want to copy, then activate the copy command (either from RMB or pull-down menu), then click in the input field before clicking on the ‘from point’ before finally clicking on the ‘to point’. In this case a copy vector of 10 m is found and automatically inserted into the vector field.

### 3.1.11 Making a named set

There are several reasons why you should work with sets. You may refer to a set while modelling, when documenting, and when investigating results in GeniE or in other SESAM programs (in these programs they are termed a *Set*). The way to make a named set is to select the relevant objects and to force the context sensitive menu by RMB where you get access to command Named Set. You will find the named set in your browser under Utilities. The example below shows how to make a named set <Beams> consisting of four beams and also how to include a new object into the same set.



When selecting a set from the browser and push RMB, you have access to the context sensitive menu.

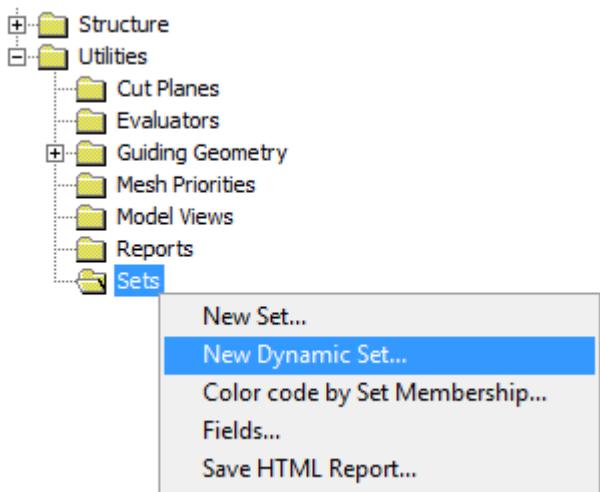


Note that when you use the option Move from this menu, the set name is referred to when scripting the move operation on the journal file, for example `autoMSet.moveTranslate(Vector3d(0 m,0 m,10 m),geUNCONNECTED);`. If you select the members of the set graphically and select Move from the context sensitive menu, all members of the set is scripted.

The same apply when assigning properties to a set.

See also Chapter 3.2.7, *Edit/Rules/Sets*.

### 3.1.12 Making a dynamic set



To create a new dynamic set, rightclick Sets under Utilities in the browser and select “New Dynamic Set”.

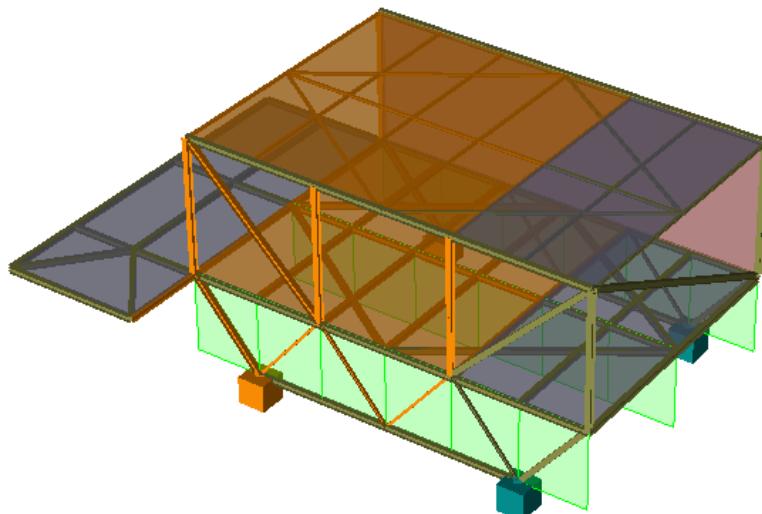
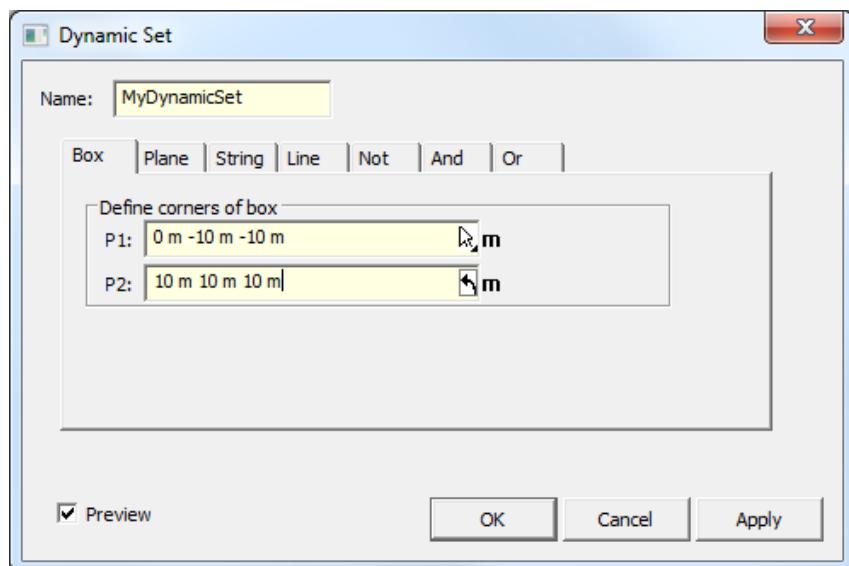
There are different options available for creating a dynamic set. They are explained in the following paragraphs.

Note that the entire objects must be within the limits to be included. If parts of for example a plate or a beam is outside the limits for the dynamic set, that plate or beam will not be included.

#### 3.1.12.1 Dynamic set - Box

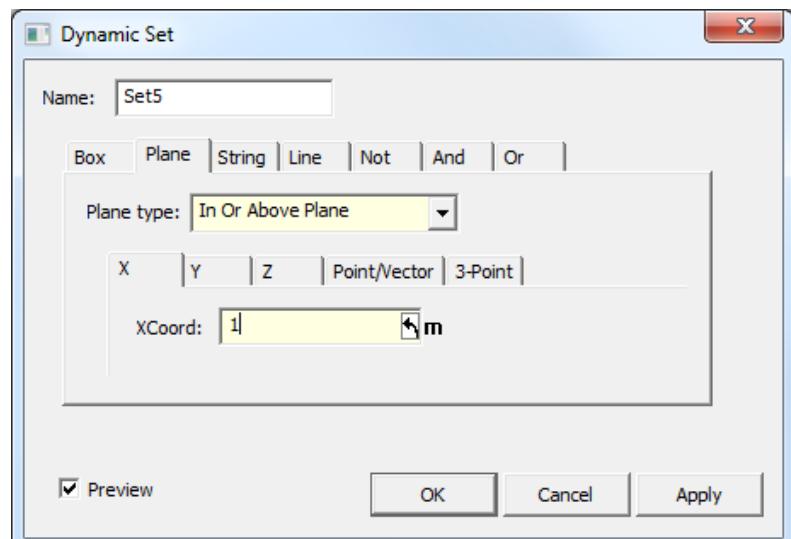
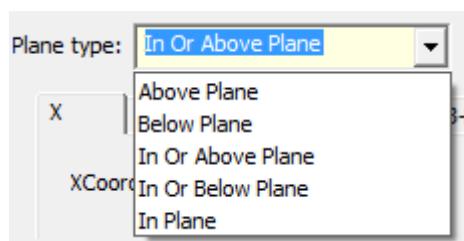
Choose “Box” to create a dynamic set limited by a bounding box. To create the bounding box, type in the corner points of the box or select them in the graphical display.

Note the “Preview” checkbox. If this is checked, the objects being included in the dynamic set are highlighted in orange. See an example of this in the illustration below. Also note that the corner points of the bounding box are highlighted. One of them can be seen as a red dot in the illustration below.



### 3.1.12.2 Dynamic set – Plane

Choose “Plane” to create a dynamic set limited by a plane. “Plane type” gives you a selection of options for how the plane should limit your dynamic set.



#### Above Plane

Objects above the specified plane are included in the dynamic set.

#### Below Plane

Objects below the specified plane are included in the dynamic set.

#### In Or Above Plane

Objects in or above the specified plane are included in the dynamic set.

#### In Or Below Plane

Objects in or below the specified plane are included in the dynamic set.

#### In Plane

Objects exactly in the specified plane are included in the dynamic set.

### There are 5 different methods for specifying a plane

#### X, Y or Z

These can be used to specify a YZ-Plane at the X value specified, an XZ-plane at the Y value specified or a XY-plane at the Z-value specified.

#### Point/Vector

Specify a point in the plane and a normal vector to identify a plane.

#### 3-point

Specify three points to identify a plane.

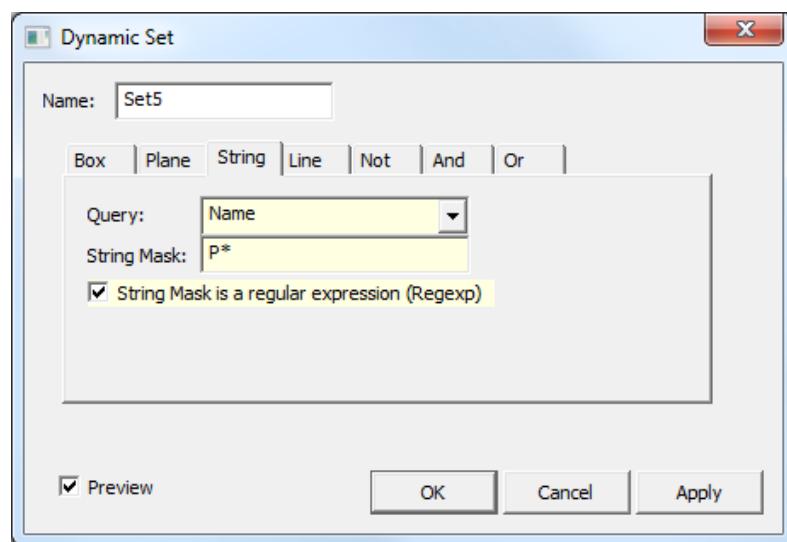
### 3.1.12.3 Dynamic Set – String

Choose “String” to create a dynamic set limited by a string. There is a wide selection of options available under “Query”. The value of the selected option will be compared with the value you specify under “String Mask”.

You can use regular expressions to include objects where part of the value of the option specified under “Query” is identical to the value specified under “String Mask”.

#### Example:

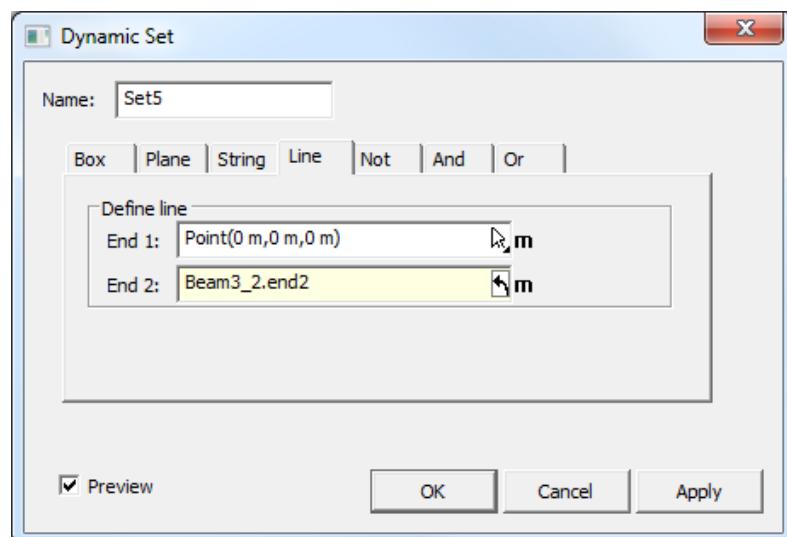
In the illustration to the right, all objects having a name starting with “P” will be included in the dynamic set.



### 3.1.12.4 Dynamic Set - Line

Choose “Line” to create a dynamic set limited by two points. The set will contain every object located on the line between these two points.

Note that as always the entire object has to be within the limits to be included. This implies that plates never can be included in a dynamic set limited by a line.

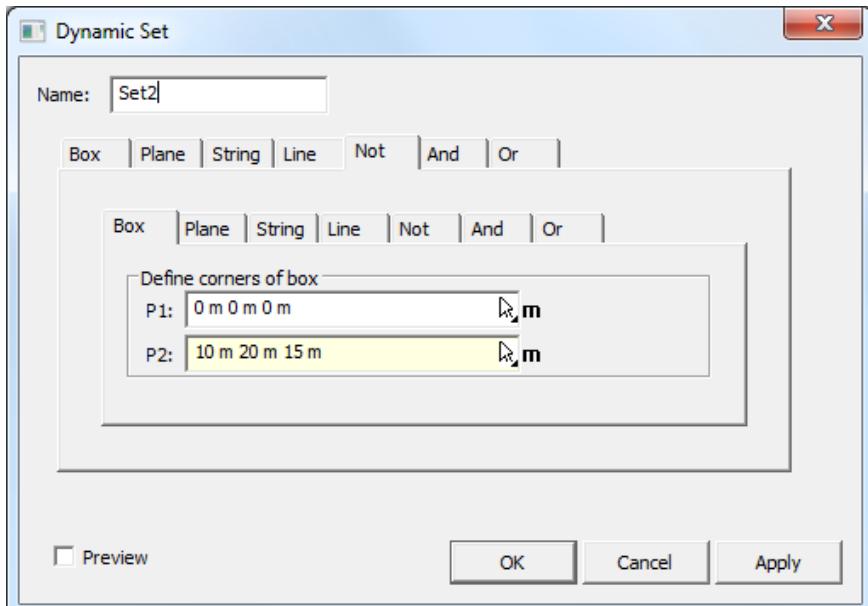


### 3.1.12.5 Dynamic Set – Not

“Not” implies that all objects *except* the ones within the limits specified will be included in the dynamic set.

This is convenient to use if you want almost the entire model to be included in the dynamic set, and only want to exclude some specific objects.

In the example illustration to the right, all objects will be included except those inside the bounding box specified.

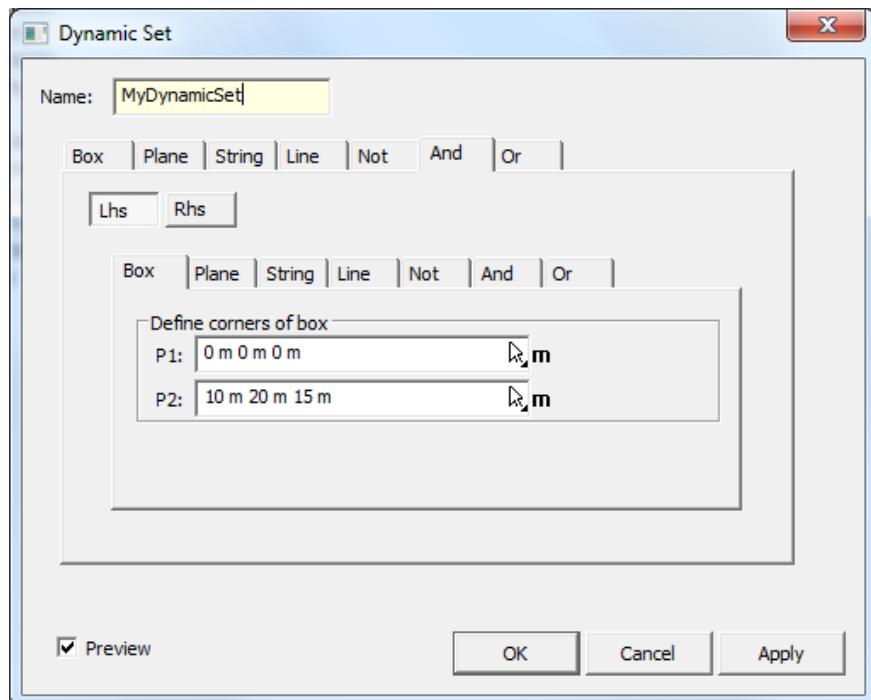


### 3.1.12.6 Dynamic Set – And

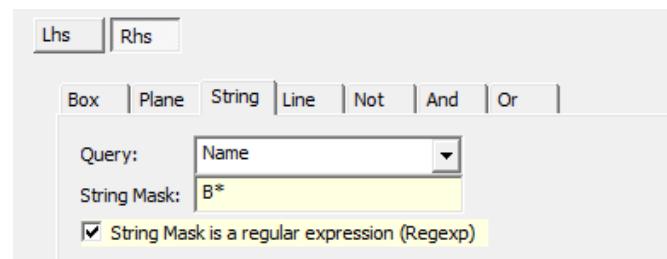
“And” will let you combine two separate specifications to create a more complex dynamic set.

The illustration to the right shows the dialog with the left hand side of the expression.

The button “Lhs” means “Left hand side”.



The illustration to the right shows a part of the dialog with the right hand side of the expression. The button “Rhs” means “Right hand side”.

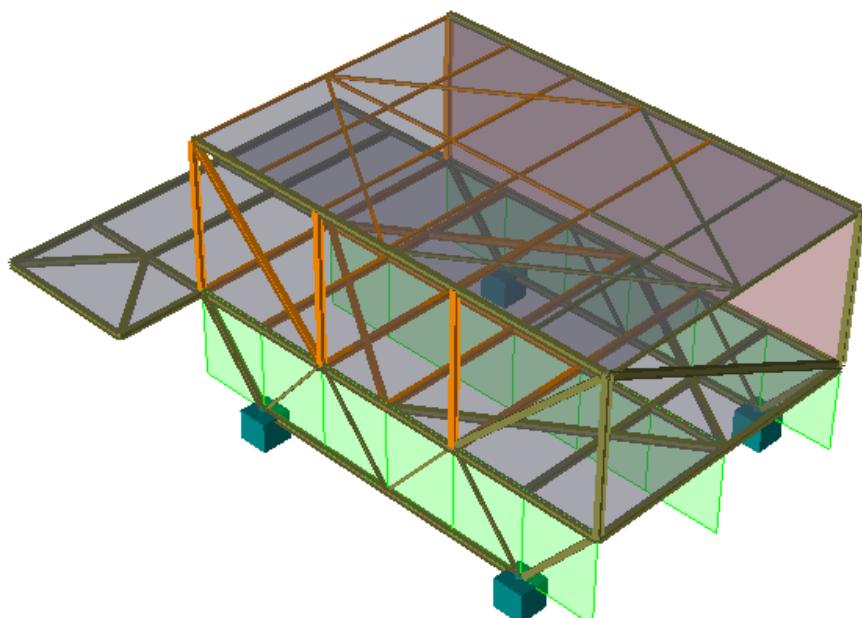


In the illustration to the right, the objects included in the dynamic set defined above are highlighted. Note that to be included in the dynamic set the objects must:

Be inside the bounding box specified under “Lhs”.

And

Have a name that starts with a “B” as specified under “Rhs”.

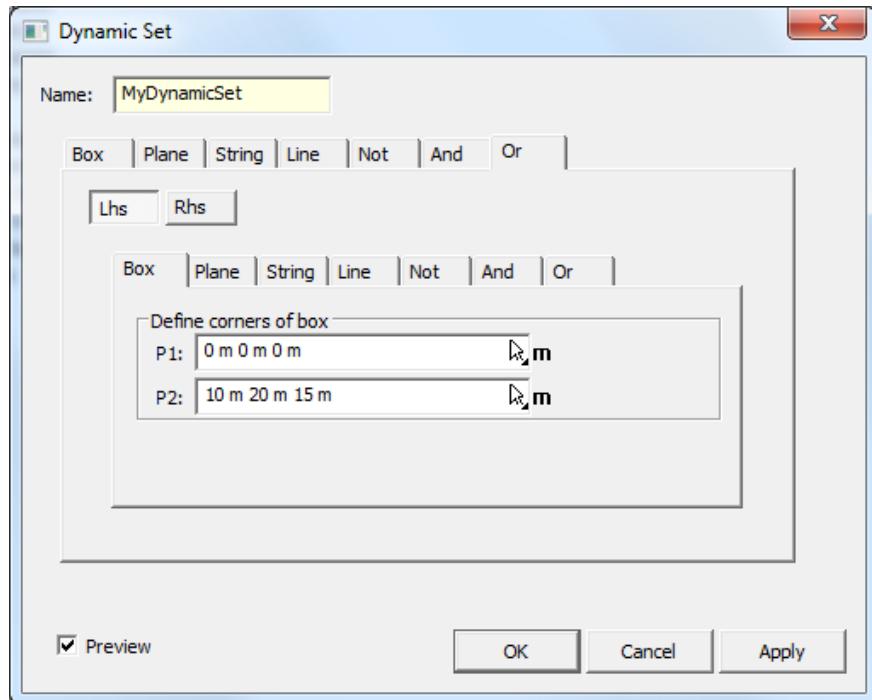


### 3.1.12.7 Dynamic Set – Or

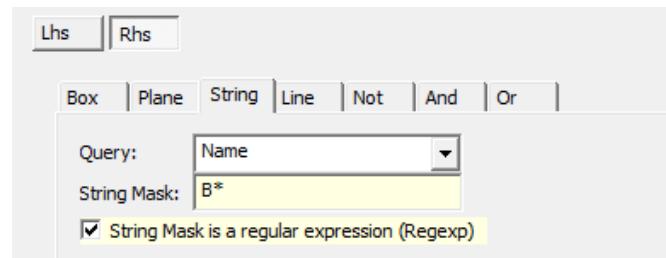
“Or” will let you combine two separate specifications to create a more complex dynamic set.

The illustration to the right shows the dialog with the left hand side of the expression.

The button “Lhs” means “Left hand side”.



The illustration to the right shows a part of the dialog with the right hand side of the expression. The button “Rhs” means “Right hand side”.

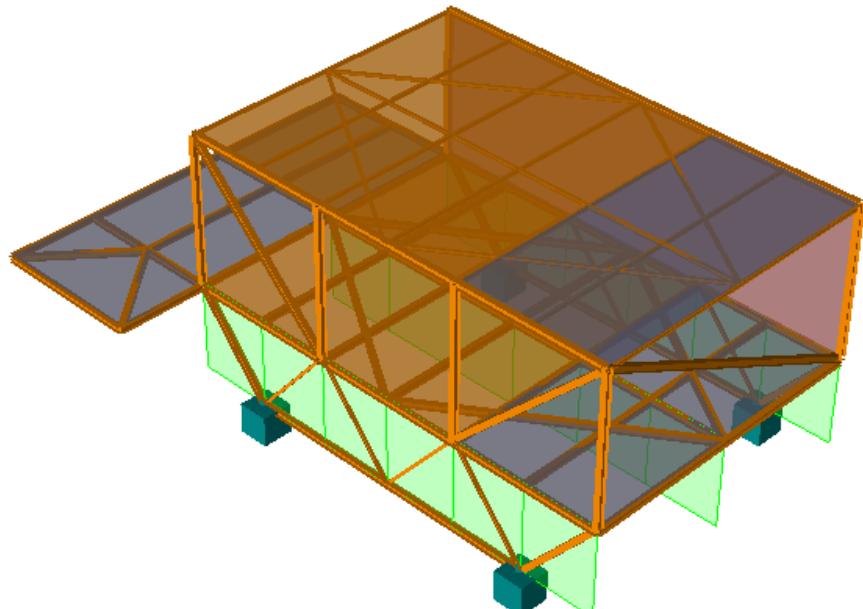


In the illustration to the right, the objects included in the dynamic set defined above are highlighted. Note that to be included in the dynamic set the objects must:

Be inside the bounding box specified under “Lhs”.

Or

Have a name that starts with a “B” as specified under “Rhs”.



Note the difference between “And” and “Or” by comparing this illustration with the one in the previous paragraph.

### **3.1.12.8      *Dynamic Set – More about using “And” and “Or”***

When using the operators “And” and/or “Or” it is possible to select “And” or “Or” on the left hand side or right hand side.

This implies you can combine up to four different specifications for creating a dynamic set. Using Javascript you can combine even more specifications.

### 3.1.13 Making a report

The command *File / Save report* will let you create a report file.

The report file is saved in the workspace directory.

The file can be saved in five different formats. The format can be selected under *Report format*:

**CSV:** Comma Separated Values

**ExcelXML:** XML format that can be read into Excel.

**Html:** Html-format for viewing in a browser.

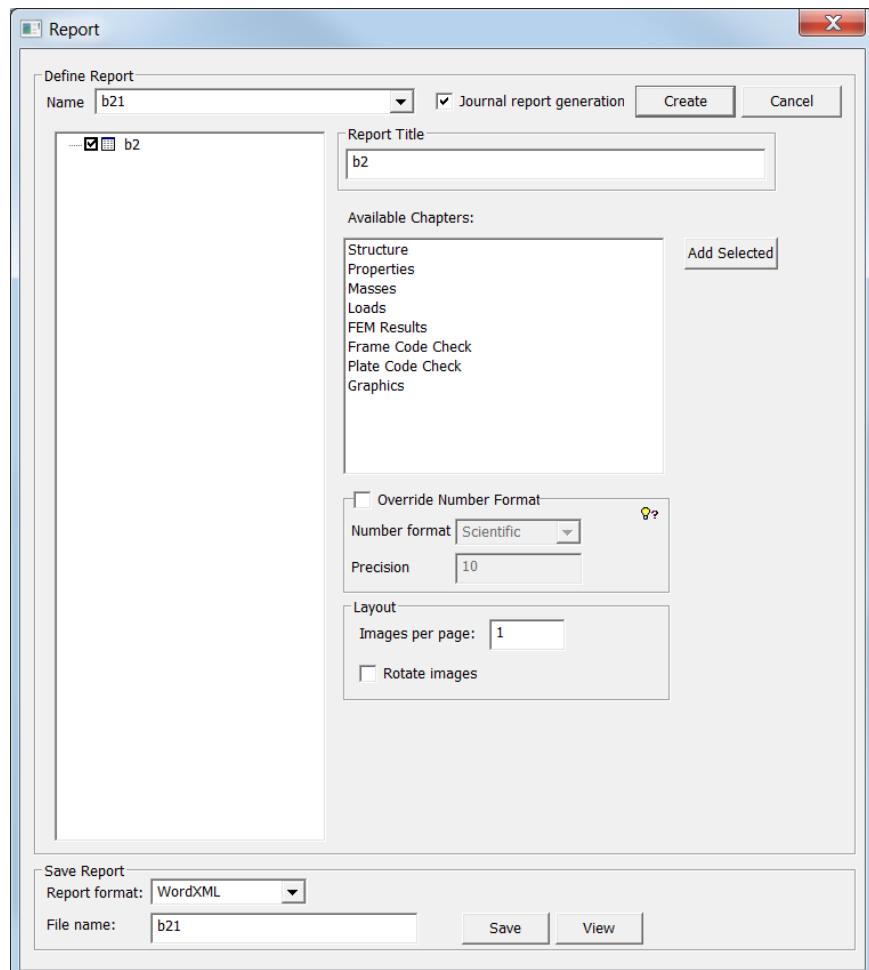
**Text:** Plain .txt file. Can be viewed with a text editing application, like Microsoft Notepad.

**WordXML:** XML format that can be read into Word.

#### Image layout

**Images Per Page:** One or two images per page can be specified

**Rotate Images:** By default images are laid out in portrait mode. Select this option to use landscape mode.



### 3.1.13.1 Looping Loads and Sets In Chapters

Some chapters allow a subset of loadcases, result cases and sets to be specified.

For some chapter children this will result in the table or image appearing once per specified combination of loadcase/resultcase and set. In other cases the child will only appear once in the report but the contents will be restricted to the selected items.

#### Loop Sets / Loop Loadcases / Loop Resultcases / Worst Loadcase

The following description is generic and applies to all chapters where looping is available. Please refer to specific chapter descriptions for variations.

Check one or more of these to loop over selected named sets and/or loadcases or resultcases. When both loadcases/resultcases and sets are selected the loadcases/resultcases represent the outer loop and the sets the inner loop.

When *Loop Sets* is unchecked it is equivalent to choosing all structure as a single set. When *Loop Loadcases* it is equivalent to choosing all loadcases for a single child i.e. there is no looping.

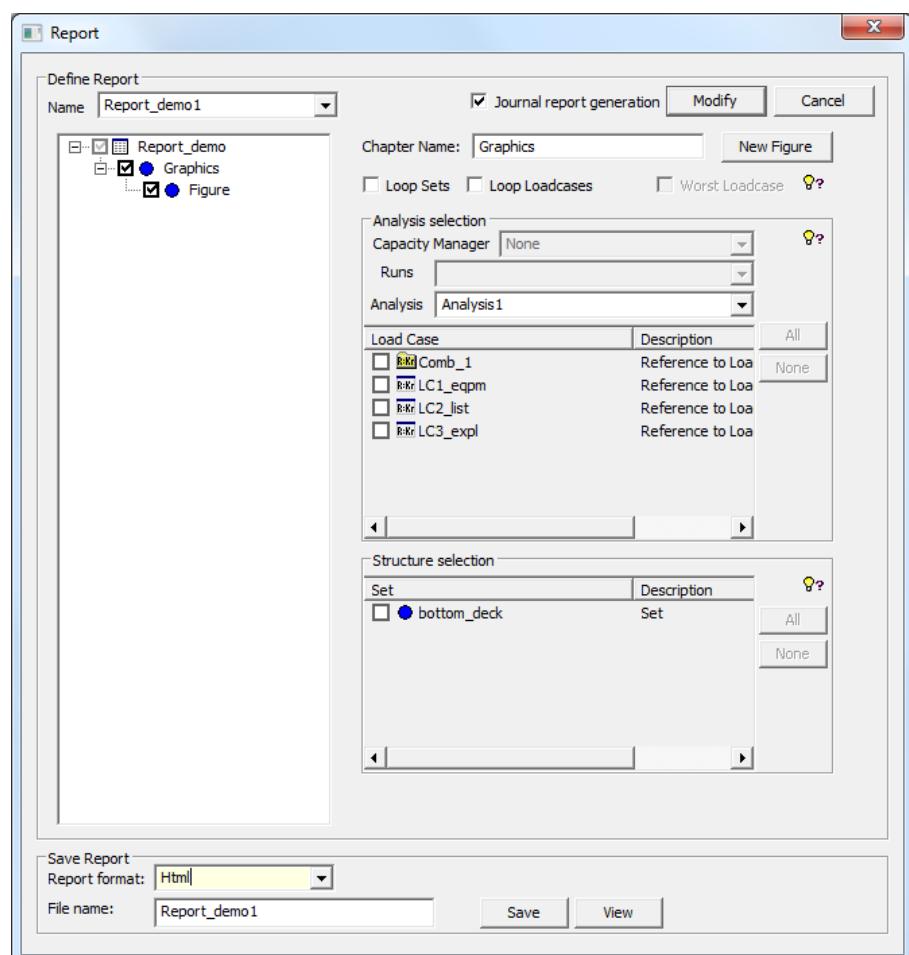
*Worst Loadcase* and *Loop Loadcases* are mutually exclusive i.e. they act as radio buttons. A Capacity Manager must be selected to enable *Worst Loadcase*.

#### Analysis Selection

*Capacity Manager*: Optional except if *Worst Loadcase* is desired. If selected, the associated analysis will also be set automatically.

*Runs*: Select the run from current capacity manager. The available loadcases will be updated.

*Analysis*: Select the analysis you want to report.

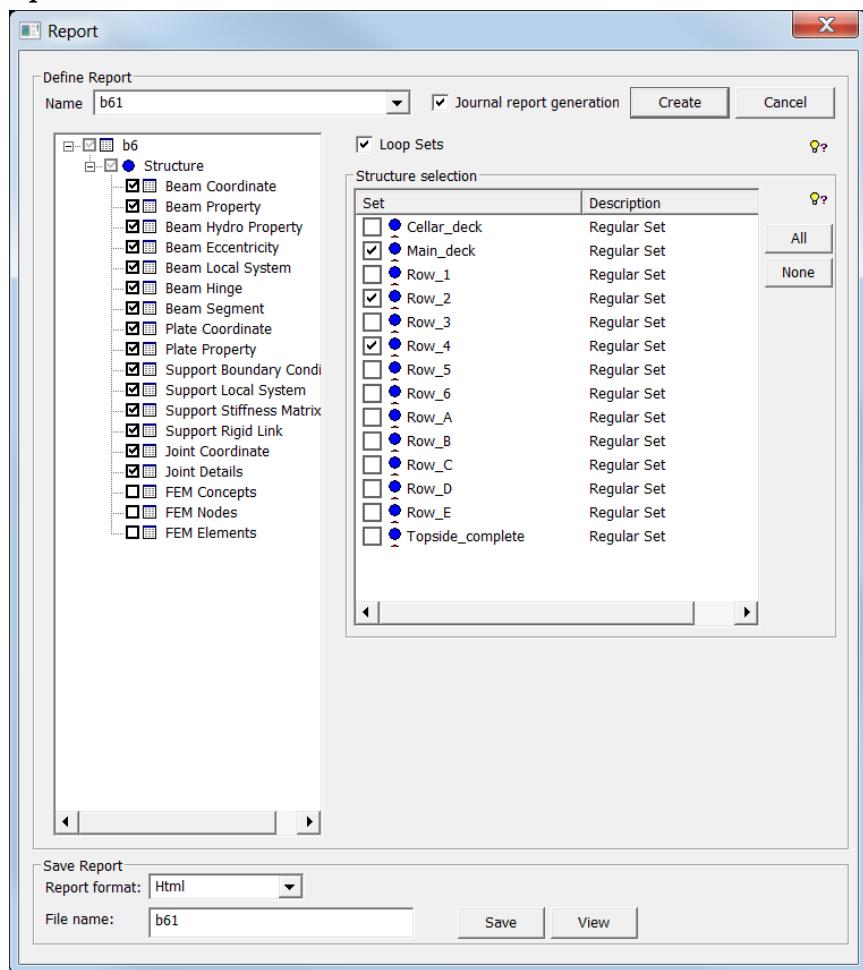


Note that the loadcase selections are stored per analysis whilst Runs act as a filter on this list. Therefore loadcase selections will persist across multiple runs that all share the same analysis.

### 3.1.13.2 The Structure Chapter

The structure chapter support tables of values relating to structural information.

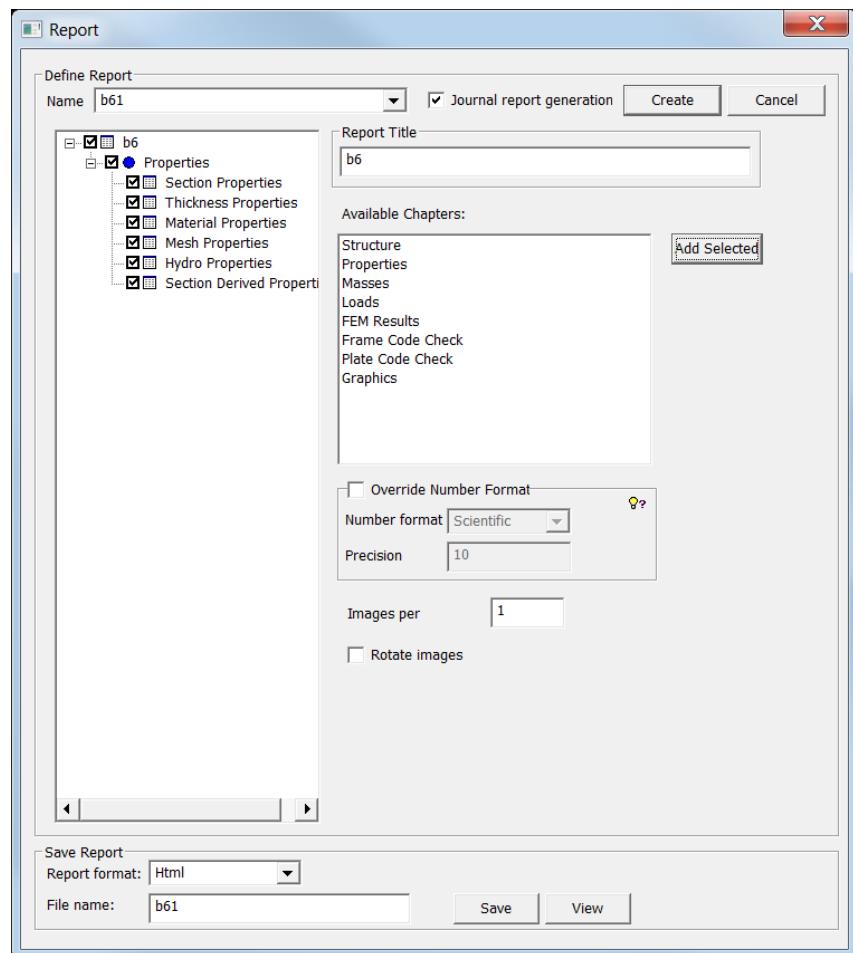
The structure included in the report can be restricted to the sets selected on the structure chapter tab. This includes both regular and dynamic sets.



### **3.1.13.3      *The Properties Chapter***

The properties chapter support tables of values relating to particular properties that are specified in the model.

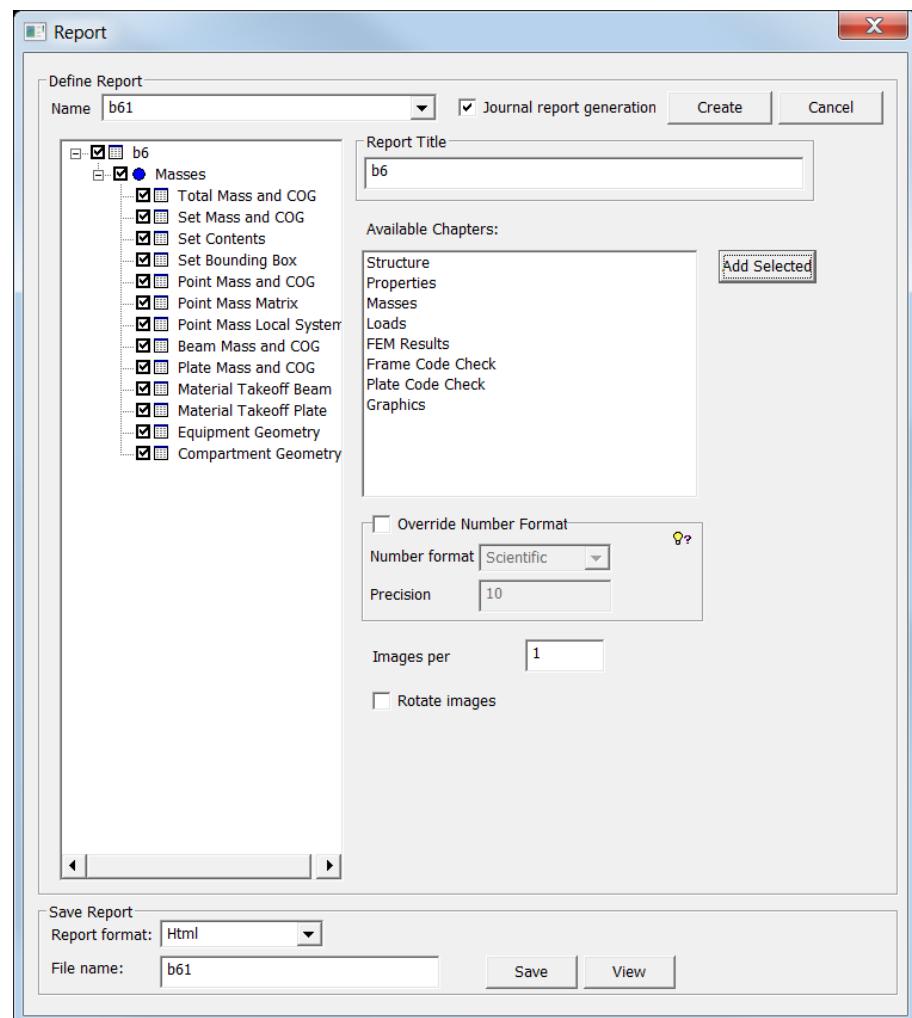
There is no support for restricting sets or loadcases in this chapter.



### 3.1.13.4 The Masses Chapter

The masses chapter support tables of values relating to mass and centre of gravity properties for the model.

There is no support for restricting sets or loadcases in this chapter.

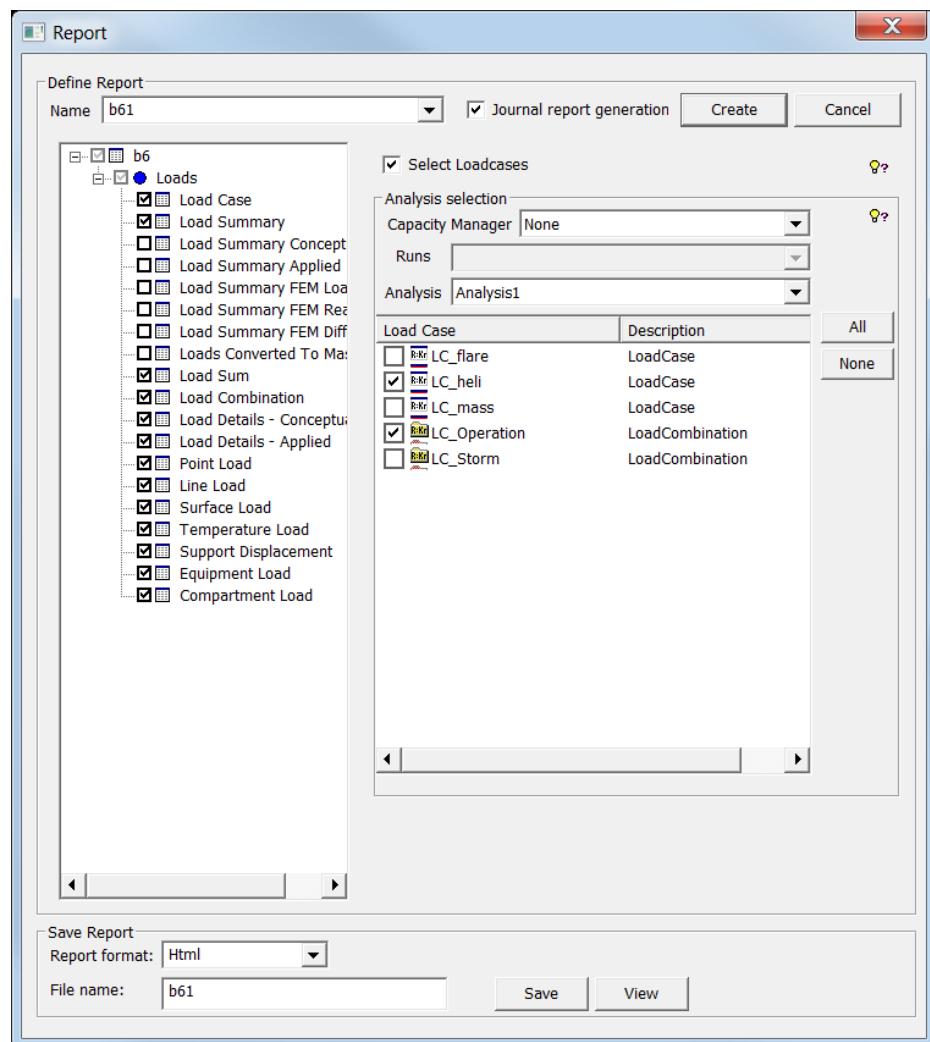


### 3.1.13.5 The Loads Chapter

The loads chapter supports tables of values relating to a variety of loads that are applied to the model.

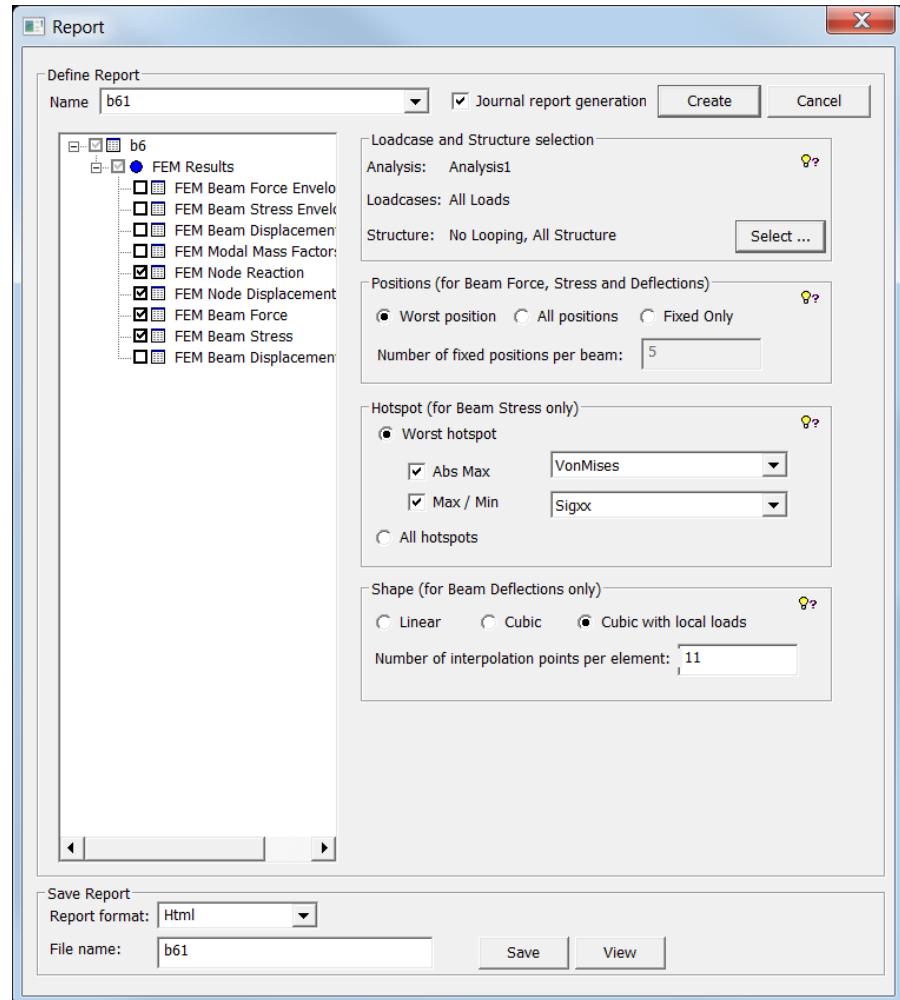
The loadcase included in the report can be restricted to those selected on the Load Chapter tab.

Resultcases are not supported by this chapter.



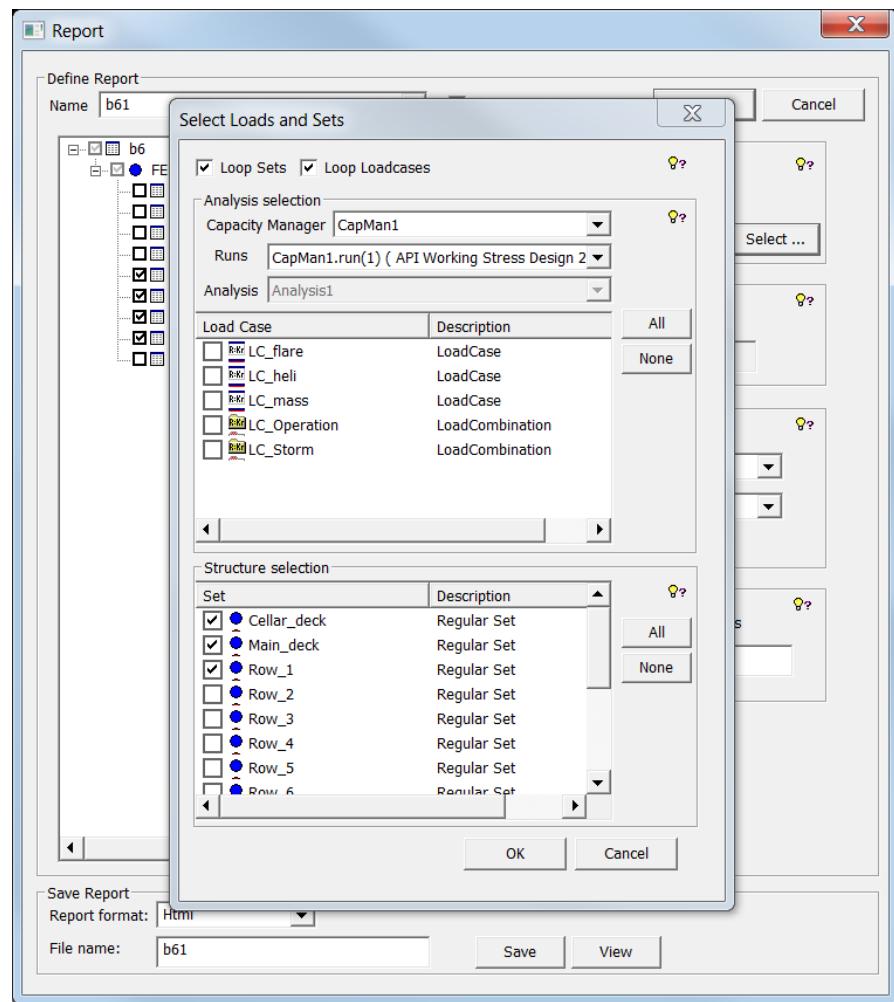
### 3.1.13.6 The FEM Results Chapter

The FEM Results chapter support tables of values relating to Finite Element Mesh Results.



The structure and resultcases included in the report can be restricted to by clicking the **Select...** button on the FEM Results Chapter tab, which raise the **Select Loads and Sets** dialog. Only Resultcases are displayed in this dialog.

The other options in the FEM Results tab can be used to further refine the report as appropriate.

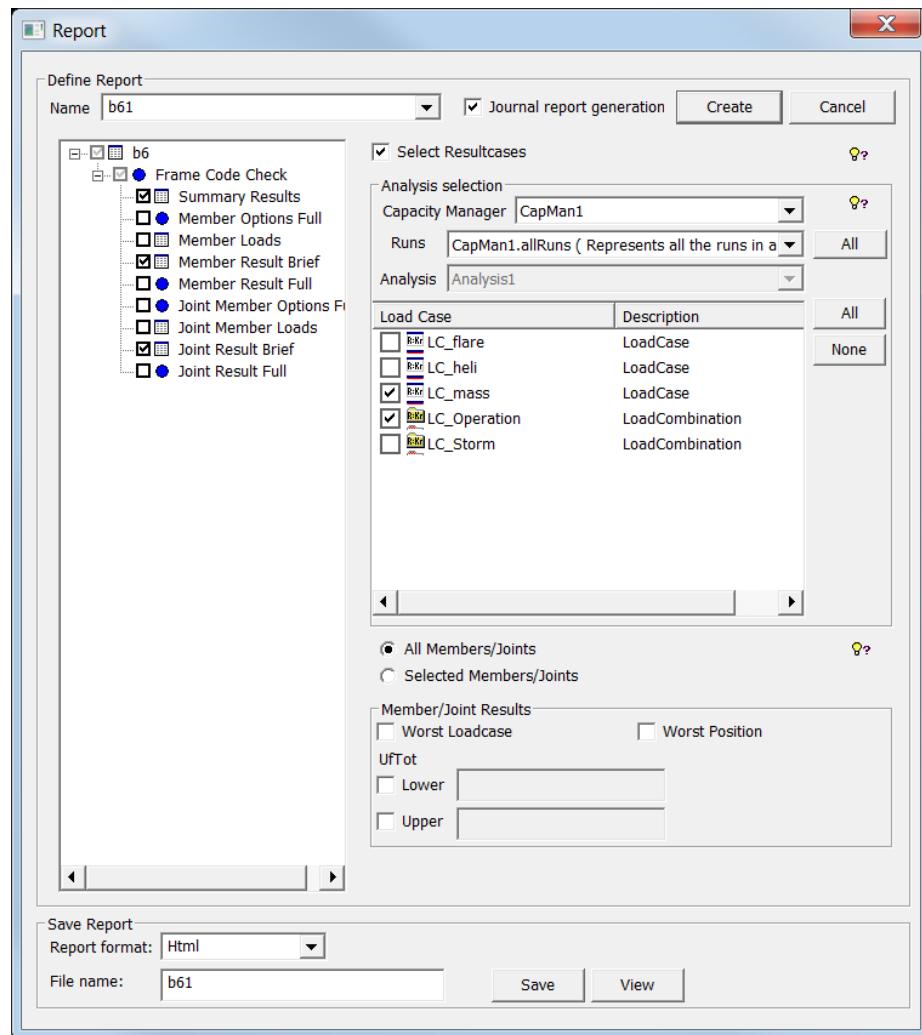


### 3.1.13.7 The Frame Code Check Chapter

The Frame Code Check chapter supports tables of values relating to the code check of members and joints.

The resultcases included in the report can be restricted to the selected resultcases on the chapter tab. Loadcases are not supported by this chapter.

Structural items can be restricted by selecting them in the graphics view *before* the report dialog is raised, and then selecting the *Selected Members/Joints* radio button. Once the *Create/Modify* or *Save* buttons have been clicked this object selection will persist with the report until further modified.

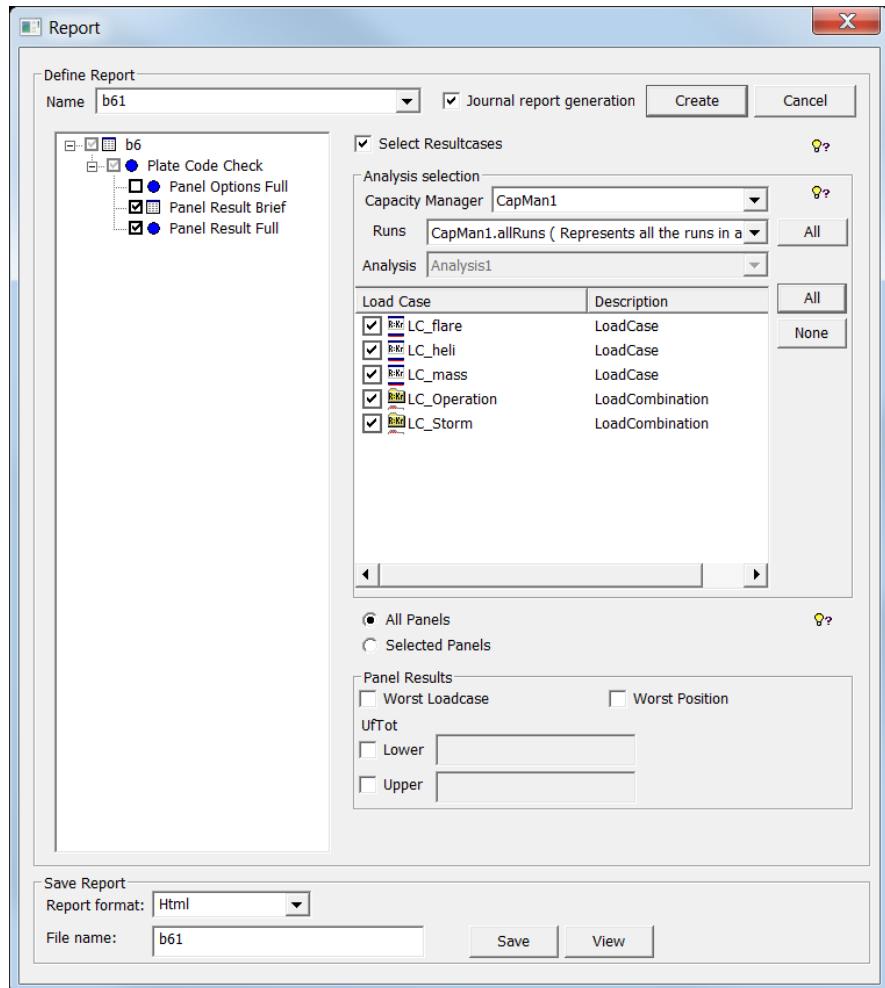


### 3.1.13.8 The Plate Code Check Chapter

The Plate Code Check chapter supports tables of values relating to the code check of panels.

The resultcases included in the report can be restricted to the resultcases selected on the chapter tab. Loadcases are not supported by this chapter.

Structural items can be restricted by selecting them in the graphics view *before* the report dialog is raised, and then selecting the *Selected Panels* radio button. Once the *Create/Modify* or *Save* buttons have been clicked this object selection will persist with the report until further modified.



### 3.1.13.9 The Graphics Chapter

The Graphics chapter lets you add graphical illustrations to your report. You can add multiple Graphics chapters and each chapter can have multiple figures.

Additionally images from external sources can be incorporated into the report.

The different options are explained in the following paragraphs.

Note that changes are not saved until Create, Modify or Save is clicked.

#### 3.1.13.9.1 Parametric Image Definitions

With the exception of importing an external image, the Report dialog allows you to define a set of parameters that represent how an image will be created. The image is not created until the report itself is generated by pressing the *Save* button.

Note that the parameters are not comprehensive i.e. only a subset of the parameters that affect GeniE's graphical representation are included. Others are taken from the GeniE settings at the time the report is generated. Therefore the same report definition can generate different images when run in different GeniE sessions with different settings.

Image sizes are currently fixed and in the case of the WordXML report are laid out with one or two images per page.

Images are stored in a specific subdirectory for each report under the directory for the workspace. This directory is deleted and recreated each time a new report with the same name is generated.

#### 3.1.13.9.2 Supported Report Formats

Only the "Html" and "WordXML" report formats include images. If other formats are used the images will still be generated but not included in the report.

#### 3.1.13.9.3 Configuring the Chapter

##### Chapter Name

Choose a name for the graphical chapter. Default is Graphics. After pressing Create/Modify or Save the name change will be reflected in the report tree.

**New Figure** Adds a new figure definition for this chapter.

**Loop Sets / Loop Loadcases / Worst Loadcase**

Check one or more of these to loop over selected named sets and/or loadcases. One image will be generated for each combination of selected loadcase and selected set for each child figure definition. When both loadcases and sets are selected the Loadcases represent the outer loop and the sets the inner loop i.e. an image for each set is generated for each loadcase in turn, and this layout is reflected in the report.

When Loop Sets is unchecked the entire visible model will be included. When Loop Loadcases is unchecked the model will be displayed with no loadcase active.

Worst Loadcase and Loop Loadcases are mutually exclusive i.e. they act as radio buttons. A Capacity Manager must be selected to enable Worst Loadcase.

When selected the loadcase name and set name are appended to the figure title in the report.

### **Analysis Selection**

*Capacity Manager:* Optional except if Worst Loadcase is desired. If selected, the associated analysis will also be set automatically.

*Runs:* Select the run from current capacity manager. The available loadcases will be updated.

*Analysis:* Select the analysis you want to report.

Note that the loadcase selections are stored per analysis whilst Runs act as a filter on this list. Therefore loadcase selections will persist across multiple runs that all share the same analysis.

### 3.1.13.9.4      Figure Definitions

When clicking on *Figure* in the browser, the choices for your illustration show up.

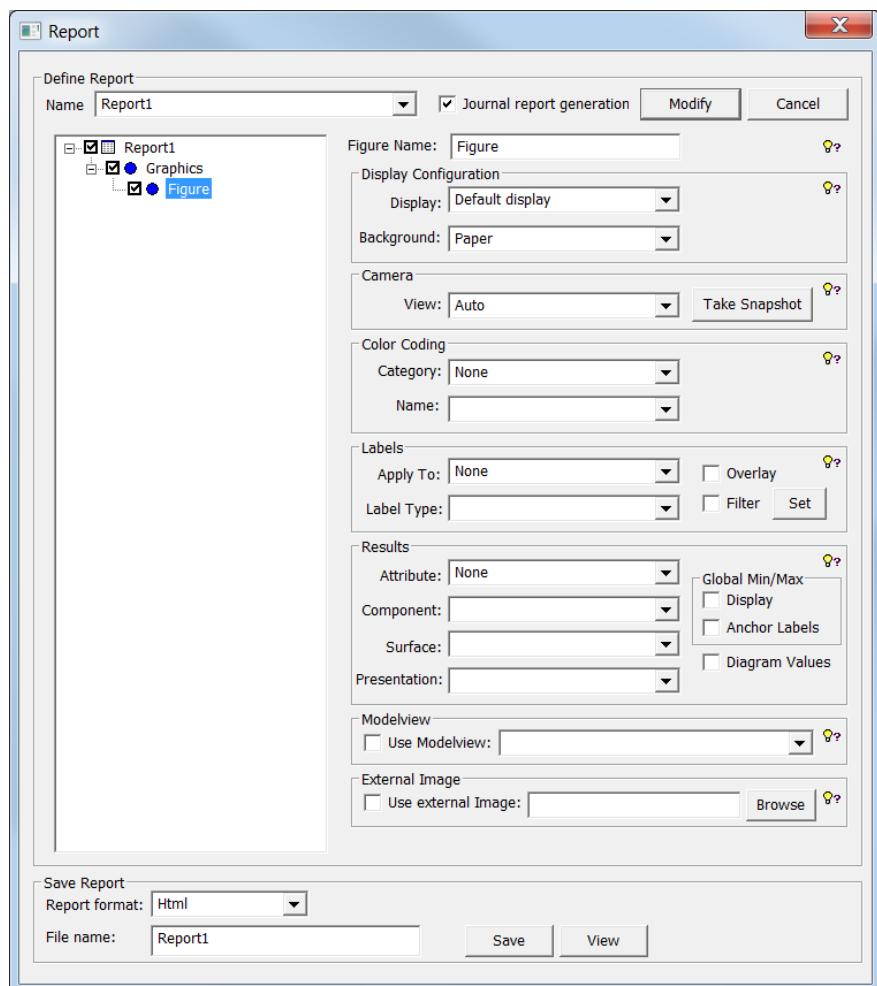
#### Figure Name

Choose a name for the figure definition. Default is *Figure*. After pressing *Create/Modify* or *Save* the name change will be reflected in the report tree.

#### Display Configuration

**Display:** Select the display you would like to use. All the displays available in the graphics window can be selected.

**Background:** The default is “Paper”, which is, by default, white. The “Paper” background setting can be changed in the General tab of the GeniE View Options Dialog. Choose “Screen” for the current background color used by the graphical display in GeniE. The default is black.



#### Camera

**View:** You have several options for view direction:

##### Auto

The default. This option will attempt to orientate the model to an axis-aligned camera such that “flattest” side is facing the viewer. If there is no “flattest” side then an ISO view is chosen. This is a useful setting when generating images of sets consisting of one web frame, one deck etc.

##### ISO

Gives an isometric view of the model.

##### From X

Along the X axis seen from positive towards negative direction of the X-axis

##### From -X

Along the X axis seen from negative towards positive direction of the X-axis. Equivalent options are available for viewing along the Y- and Z-axes.

##### Snapshot

Takes a snapshot of the current GeniE camera parameters. The snapshot is taken either when this option is selected or when the *Take Snapshot* button is clicked. If a snapshot has already been stored then it can be updated using the *Update Snapshot* button.

Note the snapshot is not stored when the selection in the View combo is changed from “Snapshot” to another option.

### Color Coding

*Category:* The categories available are: None, Mesh, Panel Geometry, Properties, Property Values, Results and Sets.

*Name:* Gives you the available options for color coding based on the setting under Category.

These options are the same as provided in the Color Coding Tab in the View Options Dialog.

### Labels

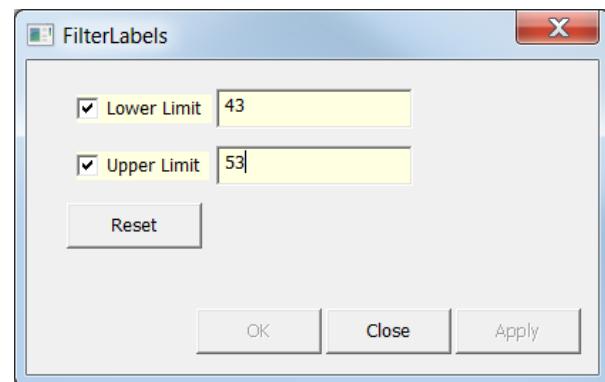
*Apply To:* Labels can be applied to Beams, Concepts, Equipment, Line Loads, Plates, Point Loads, Supports and Surface Loads.

*Label Type:* Gives you the available label types based on the setting under Apply To.

If a label category from *Apply To* or a *Label Type* is not present in the model the option will not appear in the dialog.

*Overlay:* If this is checked the labels will sit on top of the entities in the scene, rather than be obscured by entities in the scene. If the Camera View *Auto* option is chosen and an axis-aligned view results labels will automatically be overlaid.

*Filter:* Some labels that display numerical values can be filtered using a minimum and maximum value. To set these values click the *Set* button. The following dialog will be displayed:



On initial display the upper and lower limits will be set to the minimum and maximum value in the current scene. These can be changed by the user. Use the *Reset* button to reset them to these initial values. The check-boxes enable no, one or both limits.

Note that currently label filtering is not supported for all numerical labels. If not supported the *Filter* checkbox and *Set* button will be greyed out on the Figure dialog. The values are reset each time the *Apply To* or *Label Type* combo boxes are changed.

### Results

The results options are only available if you have selected *Results All* or *Results with Mesh* under *Display Configuration/Display*.

*Attribute:* Attributes available are None, Beam Forces, Displacements, G-Stress and Reaction Forces

*Component:* The available components depend on the selection under *Attribute*.

*Surface:* If you have selected *G-Stress* under *Attribute*, you can select Top, Middle or Bottom. For other values of Attribute, this is not used.

*Presentation:* The results can be presented as *Contour Plot* or *Numeric Annotation*.

*Global Min/Max:* When checked labels showing the location of the global minimum and maximum values of the selected attribute will be displayed.

*Anchor Labels:* When checked the minimum label will be anchored to the top of the screen and the maximum label will be anchored to the bottom of the screen. When unchecked the labels appear in 3D scene space.

**Diagram Values:** To use *Diagram Values*, *Attribute* must be set to *Beam Forces* and *Presentation* set to *Beam Diagram*. When *Diagram Values* is checked, a numerical value for the diagrams will be shown in the graphics together with the diagrams themselves.

### Modelview

**Use modelview:** You can reference an existing modelview by selecting from the dropdown. This option is mutually exclusive with other options. Note that as modelviews do not capture all GeniE graphical state it is possible the final image in the report may vary depending on current workspace settings.

### External Images

**Use external Image:** You can include external images by declaring the filename here. Click Browse to locate the image.

Only PNG images are supported.

The filename is relative to the directory of the current workspace. Note that if a workspace is recreated by importing javascript from another workspace the relative path of the external image is not adjusted.

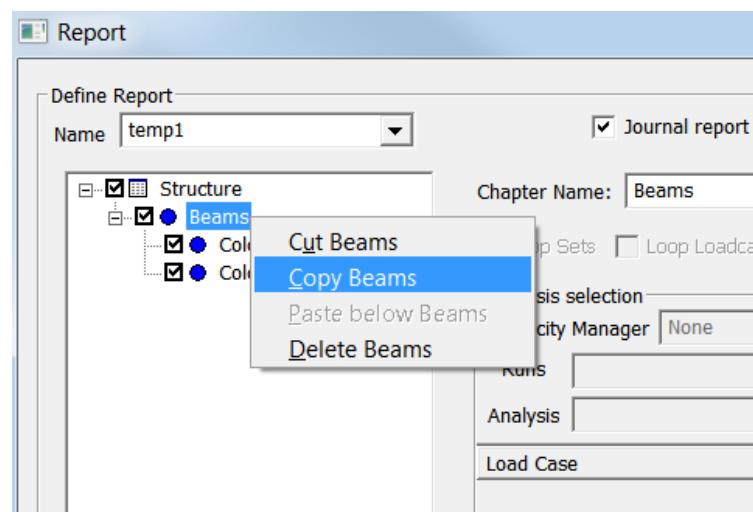
In the case of the WordXML format images will be displayed one per page.

### 3.1.13.10 Cut Copy and Paste Chapters in the report

Complex reports can be created more easily by using cut, copy and paste to duplicate or move individual chapters, tables and figure definitions.

Cut and paste is useful if you want a chapter to be placed at another location in the report. Copy and paste is particularly useful on the Graphics chapter, if you have a lot of figures that should be presented with small variations.

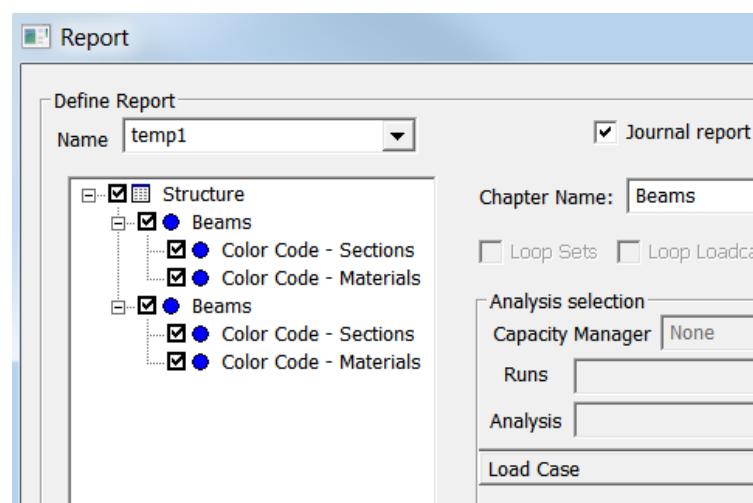
These options can be accessed using the Right Mouse Button context menu whilst selecting items in the report tree:



Cut or Copied items are pasted below the selected target item:

If a chapter is cut or copied, its children will also be cut or copied.

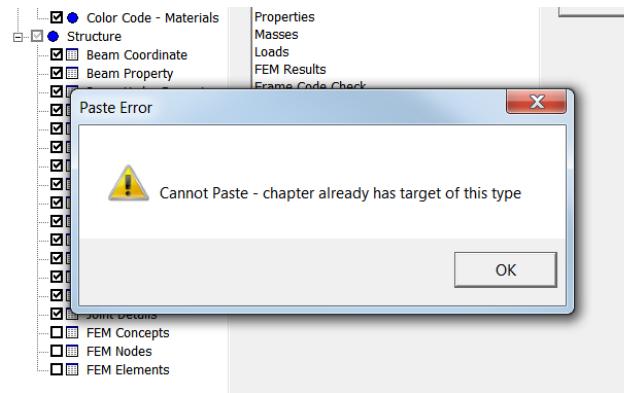
There are some rules regarding where an item can be pasted:



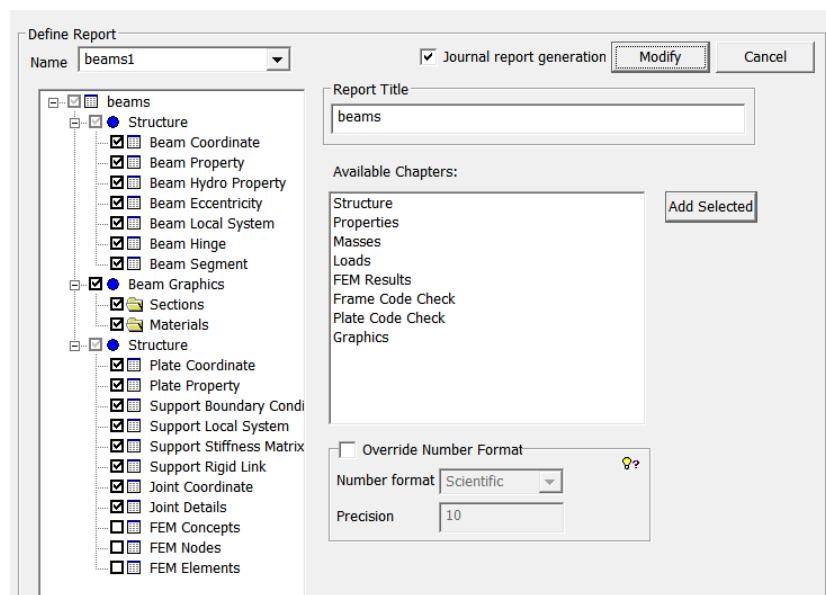
Items must be pasted at the same level i.e. chapters cannot be nested, nor can tables or figure definitions be on the same level as the chapters. If an incorrect paste is attempted the following message will be displayed:



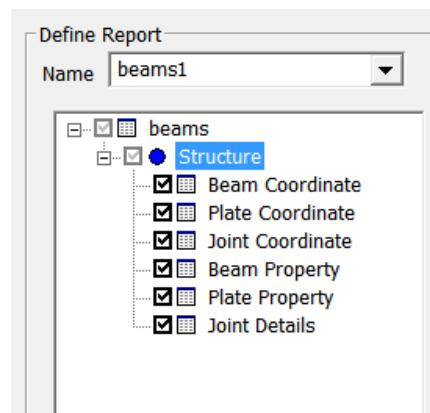
Chapters can be duplicated. However table types cannot be duplicated within chapters. For example an attempt to copy the Beam Coordinate table into a Structure chapter will lead to the following dialog being raised:



As Chapters can be duplicated it is possible to split a table-based chapter by creating two copies and removing the un-needed tables from each copy. Graphics chapters can then be inserted inbetween the two parts of the table chapter e.g:



Similarly cut and paste can be used to reorder tables within a chapter:



## Note

When chapters and their children are pasted the names in the tree will not be updated until the *Create/Modify* or *Save* buttons are pressed. Similarly the results of cut, copy and paste operations will not persist after the *Save Report* dialog is closed unless *Create/Modify* or *Save* is pressed.

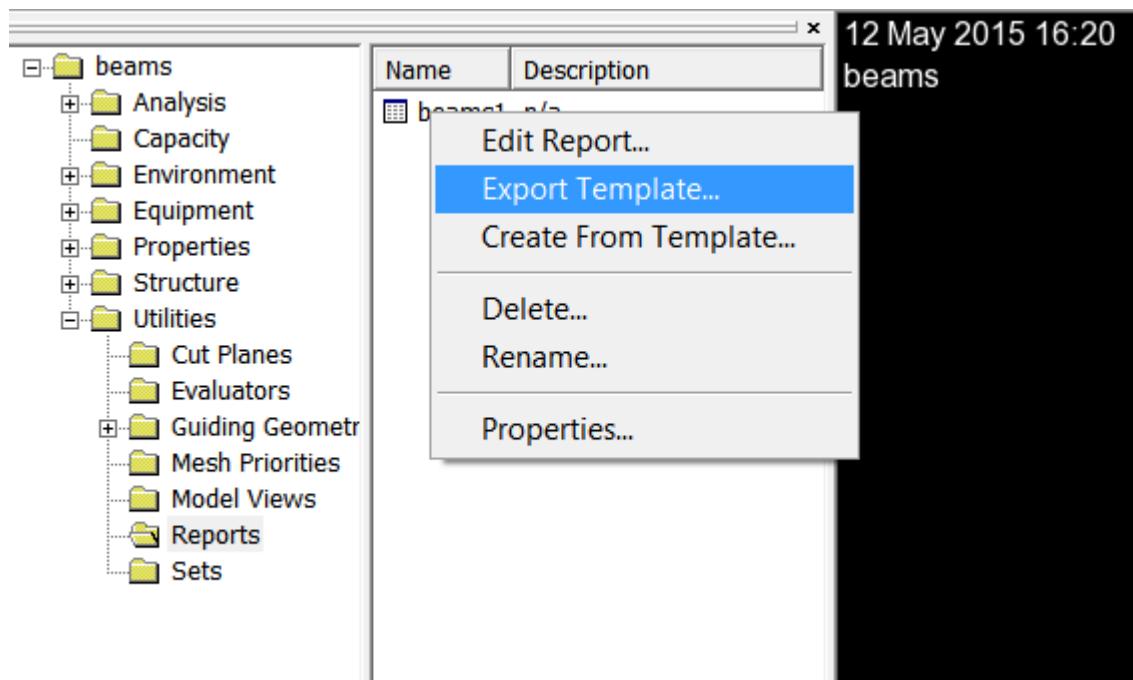
### **3.1.13.11 Report Templates**

Report Templates provide a mechanism to rapidly create new reports for specific tasks. Report templates are model independent and store details of report structure but do not store specifics such as names of loads or sets.

Report templates are in the XML file format. Manual editing of report templates via XML editors is encouraged for advanced users only.

Report templates for standard reports are provided on the GeniE installation.

The easiest way to create a report template is to choose *Export Template* from the Right Mouse Context Menu on an individual report in the browser window. The Create Report Template dialog is then raised and the report template can be given a filename.



To create a new report from a report template use the *Create From Template* option from the same menu. The Create Report from XML Template dialog will be raised to allow you to select a report template. The new report will be given a default name in the browser, and can be edited using *Edit Report*, which will raise the Save Report dialog with the selected report active.

If the report title has been specified in the template then the new report will be given this title. If there is a single Analysis and/or Capacity Model in the GeniE session in which the new report has been created then these will be set as the current options for chapters which require them. If there is no relevant object in the workspace, or there are more than one, then the chapter options will be left blank and a warning will be issued if an attempt is made to create the report without first correctly setting them.

#### **3.1.13.11.1 Specifying Set Names Filters in a report template**

Because report templates are designed to be model independent, no names are stored. However set name filters can be added which can be useful when projects have a consistent naming structure.

Set name filters can include standard DOS wildcards such as \* and ?, so for example:

Row\_\* would include Row\_1 and Row\_21

whereas

Row\_? would include Row\_1 but not Row\_21.

Set filters should be specified between the <chapter tag and the <figure\_definition tag e.g.

```

<chapter_graphics title="Whole Model">
<set_filters name_filter1="Row_?" name_filter2="*_deck" />
| <figure_definitions>
|   <figure_definition title="Model View">

```

### 3.1.13.11.2 Specifying Load Name Filters in a report template

As with sets, loads can also be filtered by name. The syntax is similar as for set names and the wildcard options are the same e.g.

```

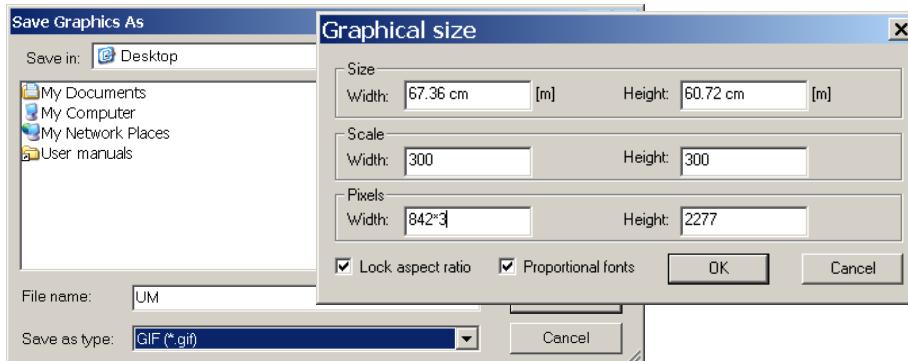
| <chapter_graphics title="Whole Model">
| <load_filters name_filter1="load_?" name_filter2="*_wind" />
|   <figure_definitions>
|     <figure_definition title="Model View" external_image="">

```

## 3.1.14 Saving a plot

Using the command **File/Save Graphics as** will store a file containing the active display. The file format can be of types \*.gif, \*.jpg, \*.eps, \*.bmp, \*.tga, \*.tif, \*.ps, \*.vrml, \*.dfile.

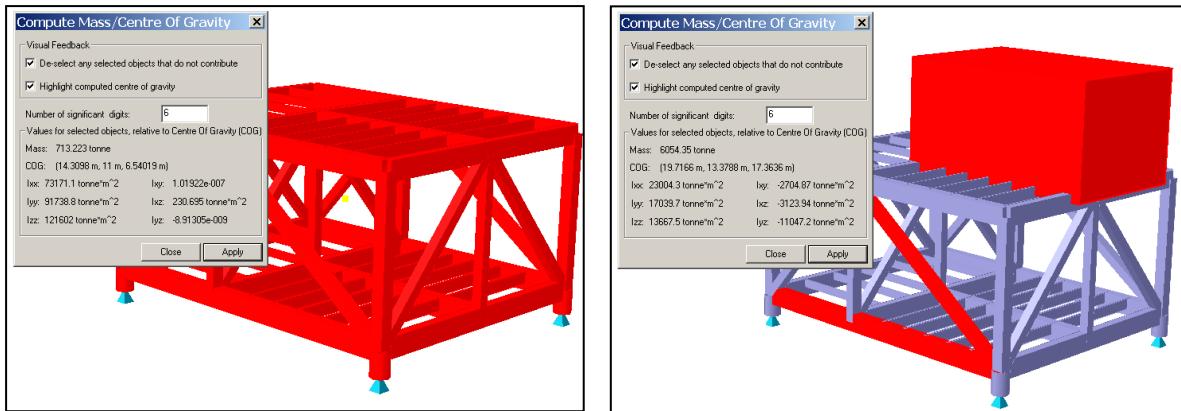
It is also possible to increase the graphics resolution of a plot. When saving a graphics file, you are prompted for such. The example below shows that the graphics resolution is increased with a factor of 3 (it is sufficient to multiply with 3 in this case, since the aspect ratio is locked).



When activating the proportional fonts option, the fonts will be scaled accordingly to increase in graphics resolution.

### 3.1.15 Documenting mass and centre of gravity

GeniE has a special function to calculate the mass of your structure and locating centre of gravity (the feature is available from the context sensitive menu). You can document mass and cog for the whole structure and you can do the same for selected structural parts and equipments. When calculating the masses, beam eccentricities are accounted for.



*The mass, moment of inertia and centre of gravity for entire structure are computed. The support points do not contribute to mass and are de-selected*

*The mass, moment of inertia and centre of gravity for one equipment and two beams are computed*

You may also find information about total mass and mass per individual objects from a printed report. The example below shows how to find such information when viewing from MS Excel.

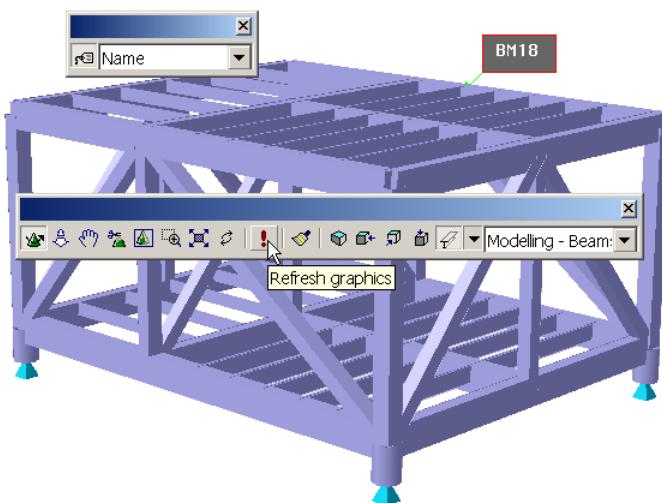
Group	Mass [tonne]	About	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	IXX [tonne*m^2]	IYY [tonne*m^2]	IZZ [tonne*m^2]	Count
2 StraightBeams	7.1322E+02	Origin	14.310	11.000	6.540	1.8998E+05	2.6829E+05	3.5395E+05	61
3		ModelCOG	0.000	0.000	0.000	7.3171E+04	9.1739E+04	1.2160E+05	
4		SubsetCOG	0.000	0.000	0.000	7.3171E+04	9.1739E+04	1.2160E+05	
5 CurvedBeams	0.0000E+00	Origin							0
6		ModelCOG							
7		SubsetCOG							
8 FlatPlates	0.0000E+00	Origin							0
9		ModelCOG							
10		SubsetCOG							
11 CurvedShells	0.0000E+00	Origin							0
12		ModelCOG							
13		SubsetCOG							
14 PointMasses	0.0000E+00	Origin							0
15		ModelCOG							
16		SubsetCOG							
17 SupportPoints	0.0000E+00	Origin							4
18		ModelCOG							
19		SubsetCOG							
20 SupportCurves	0.0000E+00	Origin							0
21		ModelCOG							
22		SubsetCOG							
23 Joints	0.0000E+00	Origin							0
24		ModelCOG							
25		SubsetCOG							
26 Total	7.1322E+02	Origin	14.310	11.000	6.540	1.8998E+05	2.6829E+05	3.5395E+05	65
27		ModelCOG	0.000	0.000	0.000	7.3171E+04	9.1739E+04	1.2160E+05	
28		SubsetCOG	0.000	0.000	0.000	7.3171E+04	9.1739E+04	1.2160E+05	
29 Total-FEM	0.0000E+00	Origin							0
30		ModelCOG							
31		SubsetCOG							

Name	Mass [tonne]	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	Length [m]	FlexLen [m]	Type
2 BM1	3.4289E+01	14.000	0.000	0.000	28.000	28.000	Straight Beam
3 BM10	2.5915E+00	5.500	14.660	0.450	11.000	11.000	Straight Beam
4 BM11	2.5915E+00	5.500	18.330	0.450	11.000	11.000	Straight Beam
5 BM12	6.7754E+00	14.650	11.000	0.300	22.000	22.000	Straight Beam
6 BM13	6.7754E+00	17.150	11.000	0.300	22.000	22.000	Straight Beam
7 BM14	6.7754E+00	19.650	11.000	0.300	22.000	22.000	Straight Beam
8 BM15	6.7754E+00	22.150	11.000	0.300	22.000	22.000	Straight Beam
9 BM16	6.7754E+00	24.650	11.000	0.300	22.000	22.000	Straight Beam
10 BM17	4.0812E+01	14.000	0.000	0.000	12.500	28.000	28.000
11 BM18	4.0812E+01	14.000	22.000	12.500	28.000	28.000	Straight Beam
12 BM19	2.8669E+01	0.000	11.000	12.500	22.000	22.000	Straight Beam
13 BM2	3.4289E+01	14.000	22.000	0.000	28.000	28.000	Straight Beam
14 BM22	3.1867E+00	5.500	3.670	12.900	11.000	11.000	Straight Beam
15 BM23	2.4779E+01	19.500	11.000	12.500	17.000	17.000	Straight Beam
16 RM24	3.1867E+00	5.500	7.340	12.900	11.000	11.000	Straight Beam

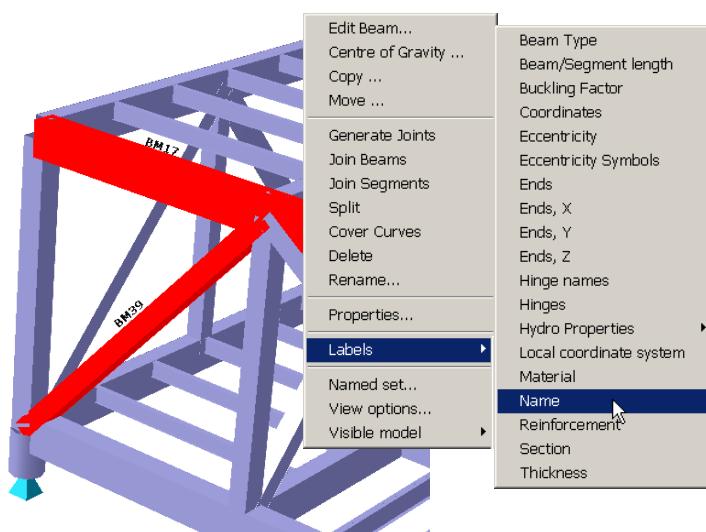
### 3.1.16 Labelling

Labelling is performed on selected objects. There are two ways of doing it, either using the predefined pulldown list or from the context sensitive menu.



In this case <name> from the name Toolbar has been selected. The name tag appear when clicking on the desired beam. The tag disappears when refreshing the graphics either graphically as shown or from pulldown **View|Refresh Graphics**.

Make sure the tag mark on the Toolbar has been activated prior to labelling.



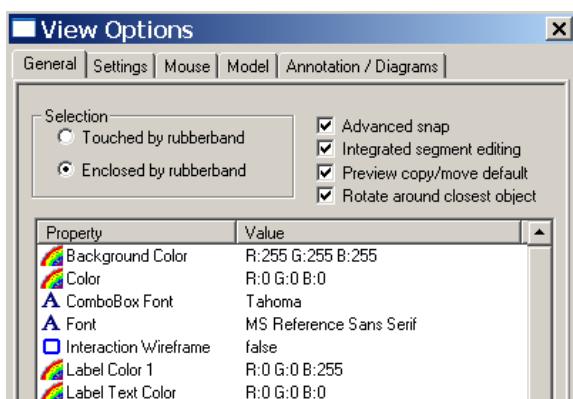
To the left, labelling is activated from the context sensitive menu by selecting an object and then RMB. The picture illustrates the available labels for a beam.

Labelling disappears upon refresh graphics.

Note that any labels are part of the picture when saving a graphic file.

Label orientation along beam axis applies for *Name*.

See ***View/Options/Annotations/Diagrams***.

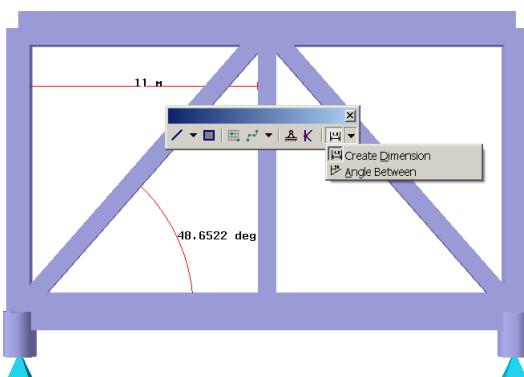


You control the appearance of labels from the ***View/Option/General*** menu.

- The size of the labels is the same as defined in the *Font*
- The colour of the label text is defined in *Label Text Color*
- The colour of label symbols (e.g. eccentricity symbols) is controlled from *Label Color 1*.

### 3.1.17 Show length and angle

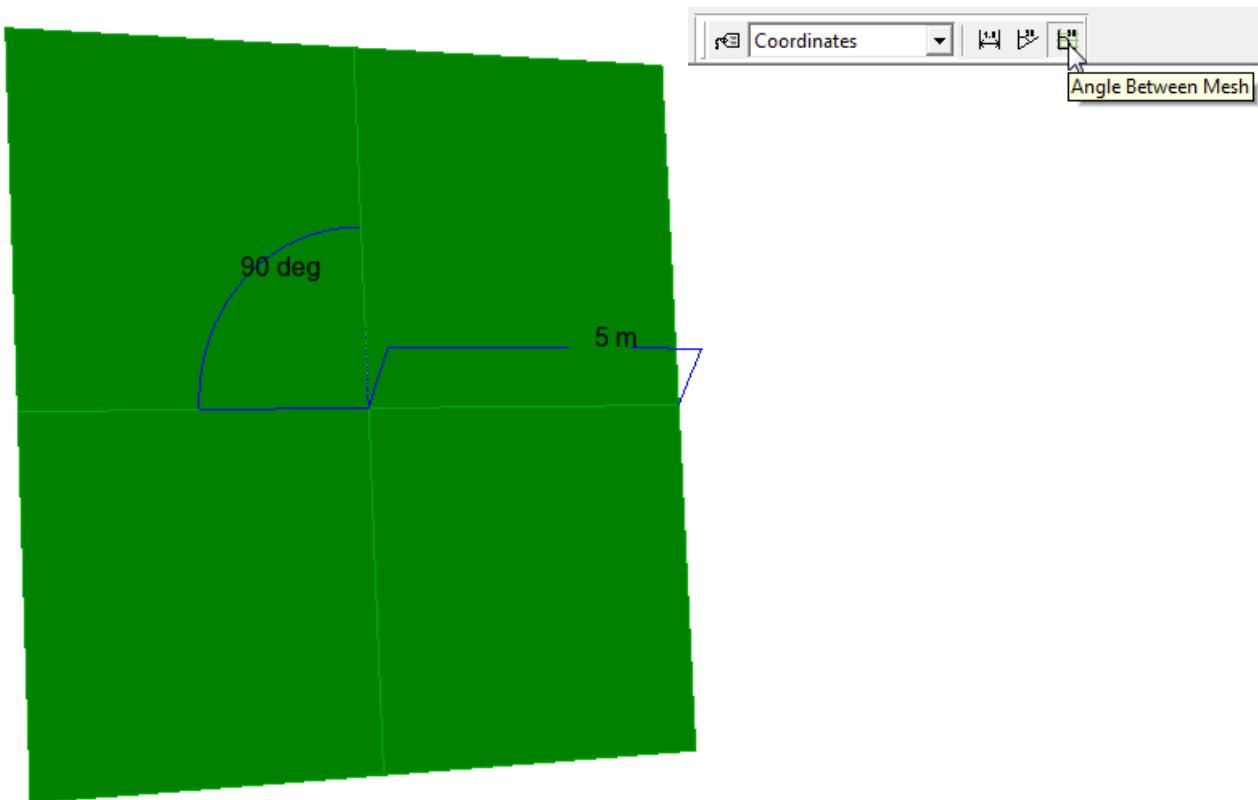
You may show length and angle graphically.



In this example length is found by activating *Create Dimension* and click on the two end points of the actual beam and drag the mouse to where you want the result to occur.

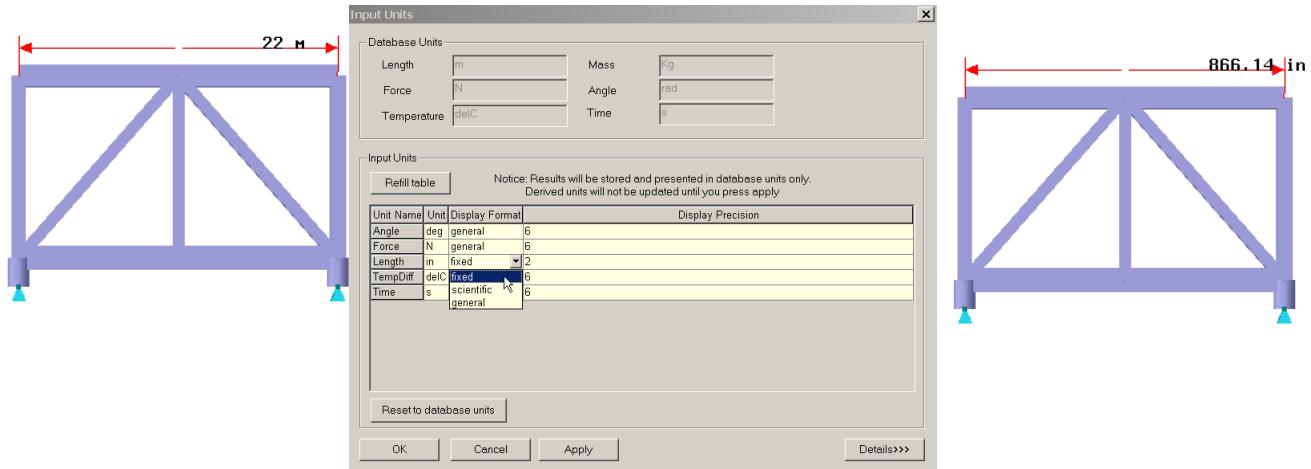
Similarly, the angle is found by activating *Angle Between* and click on the two beams – the result is automatically placed on the screen.

It is important to note that *Create Dimension* also works for the Mesh View. User can visualise distances between nodes in the FEM. However, for displaying angles, one has to use *Angle Between Mesh* feature.



### 3.1.18 Working with units

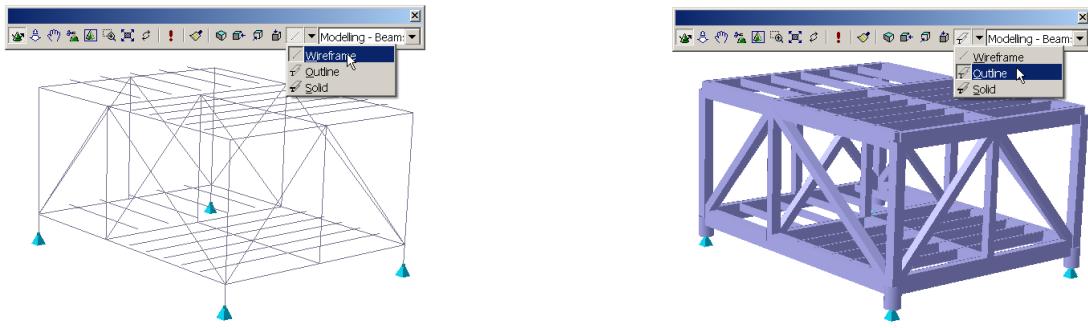
The database units are set when creating a new workspace. The interface files and results created are always given in the database units. The database units can not be changed later for a given workspace. You may, however, choose to work with different input units at different times. For example you start modelling with meters and change to inches later. GeniE will automatically convert units to database units, but they will be stored on the journal file according the input unit. Below is an example where input units are changed from meters to inches and how to visualise the dimension (**Edit|Rules|Units**). As can be seen 22 meters is automatically shown as 866.14 inches.



If you change your mind wrt. database units you can change the parameters in the command `GeniERules.Units.SetDatabaseUnits` and re-run the command input file. Note that when recreating a model based on an existing command input file you must skip setting the database units from the File|New workspace dialog because they are already defined on the command input file.

### 3.1.19 Switch between wireframe, outline, solid view

You can visualise your model in a number of ways, from the Toolbar *View Manipulations* you may change between Wireframe, Outline, and Solid view. The example below shows wireframe and outline views.

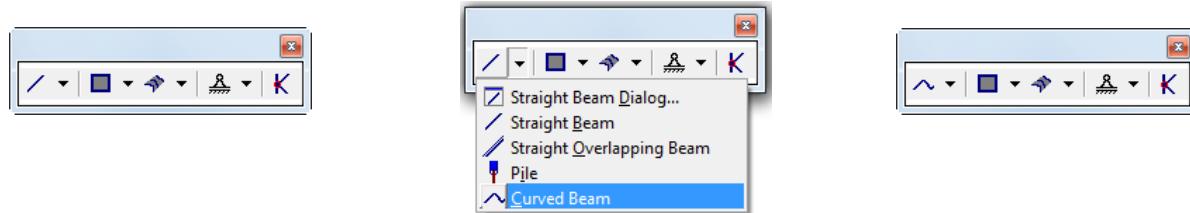


For wireframe view, you control the width of the lines from the *Line Width* on the dialogue **View/Options/General**.

### 3.1.20 Customise the tool buttons

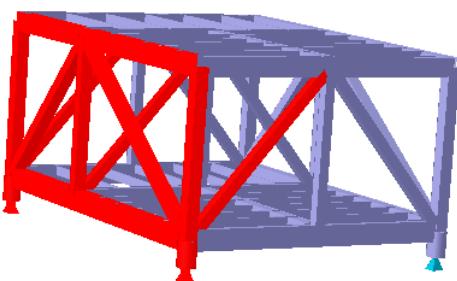
Several of the buttons on the Toolbars may be customised to your needs. They are all visualised with a down-arrow, when you click the arrow you get access to the available configurations.

In the example below default beam modelling assumes insert straight beams. The pictures show an example on how to change the default modus to curved beams.



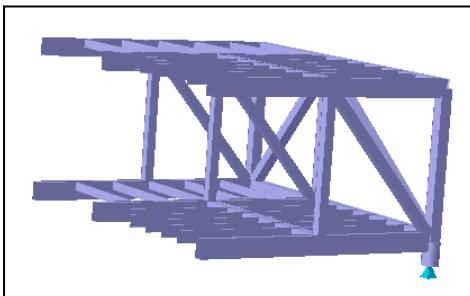
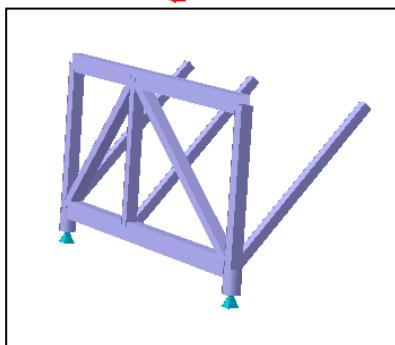
### 3.1.21 Work with visible model

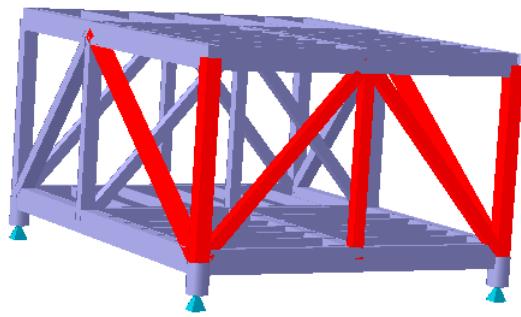
You may choose to view parts of the model when modelling or interrogating the model. You may choose to use Named Sets or make up your selection from time to time. The examples below show how you can manipulate your view, either by manual or global selection/deselection.



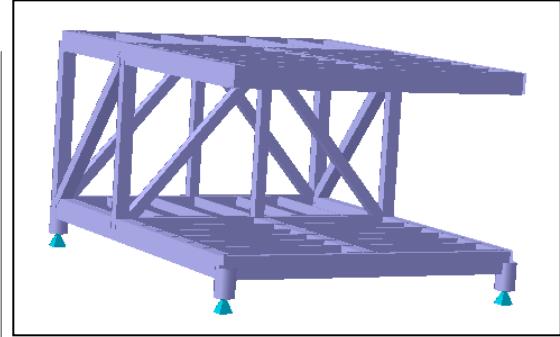
A number of beams have been selected to form the view to the left. By forcing the context sensitive menu, choose **Visible Model>Show selection only** to produce the picture down to the left.

The picture down to the right has been generated by choosing **Visible Model>Show complement**. You view the complete model by **Visible Model>Show all**.



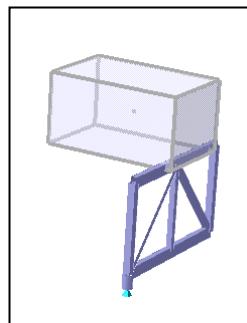
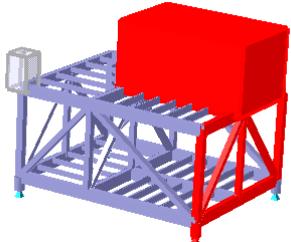


By using the commands showed below (**Visible model|Remove selection**) the picture down to the right is created.

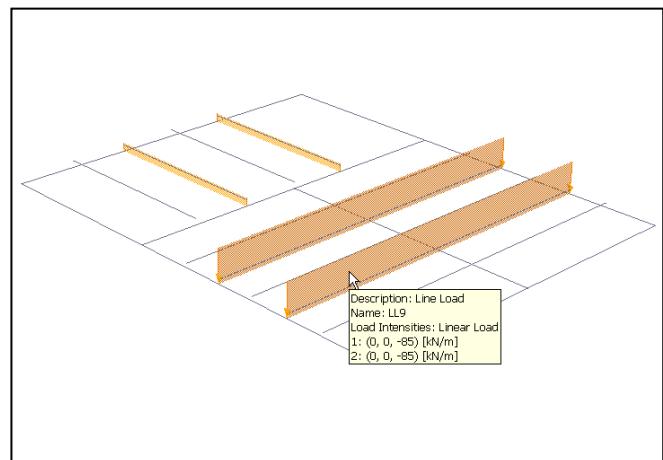
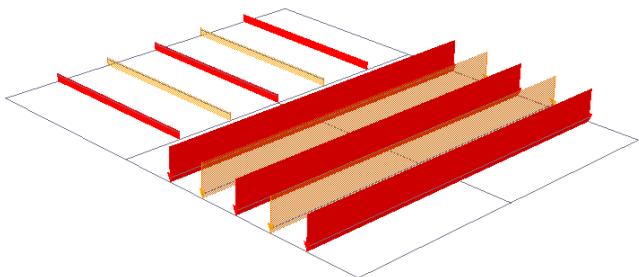


You should observe the short commands for these commands. Please note that when using a laptop you may need to change to mode *num lock* to be able to do *Alt+Plus* and *Alt-Minus*.

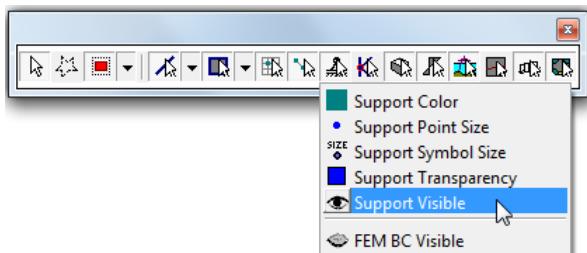
The features for visible model work for all types of concepts. The example below shows selection of structure, equipments and support points (**Visible Model>Show selection only**).



It is also possible to decide which explicit loads you want to visualise for a particular loadcase. In the example below two of the explicit line loads have been selected and by using **Visible Model|Remove Selection** they are not shown.

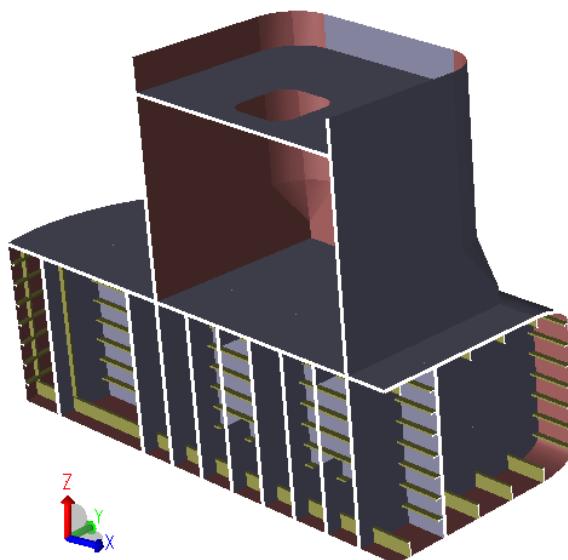


It is also possible to select which concepts you want to see on a global level. This can be done via the predefined views (see Chapter 3.1.3.2) or by using the pulldown menu features for the Selection Toolbar. The pulldown menu is activated by clicking RMB on the actual tool button.



In this example boundary conditions will be shown except when displaying the finite element mesh. Observe that changing settings here is the same as changing from *View/Option/Settings*.

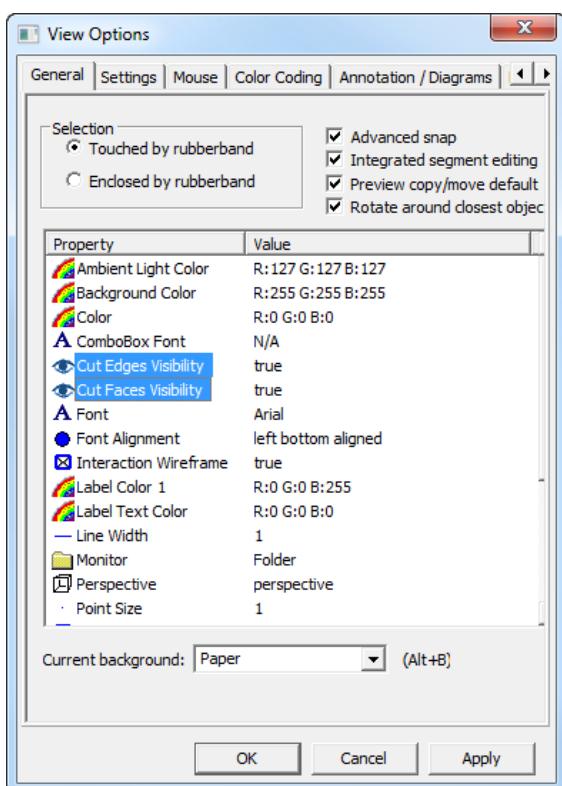
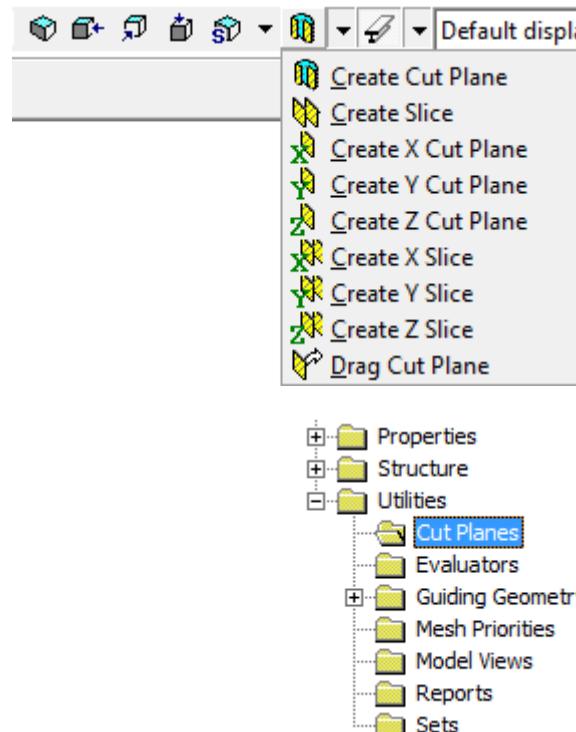
### 3.1.22 Cut planes



Cut planes are used to hide parts of your model, to easier be able to see parts that are hidden behind other parts of the structure.

In the illustration to the left two cut planes have been created, one X cut plane (perpendicular to the X-axis) and one Y cut plane. Internal structure is visible.

Cut planes are available from the drop down menu or from the browser as shown below.



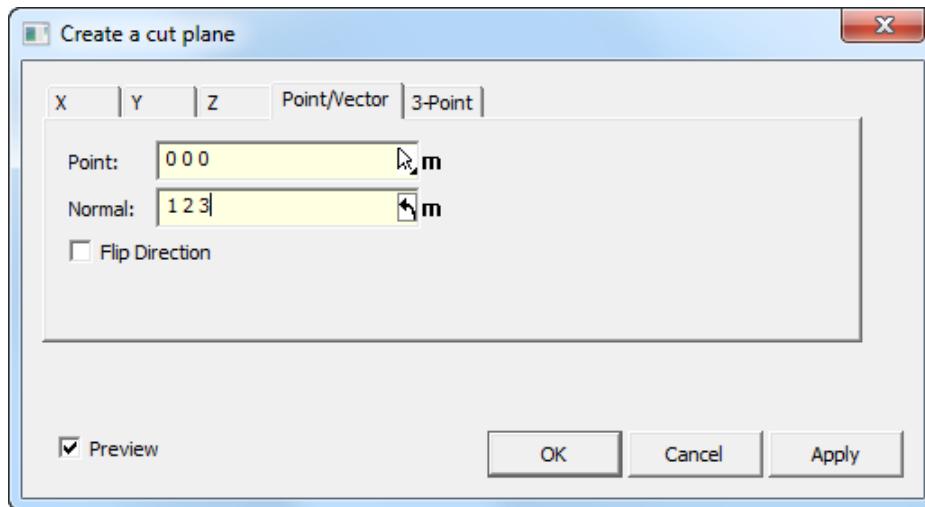
Under the General tab in View | Options you can select if you want the cut edges and/or cut faces to be visible. If you set these options to true, then cut edges and faces are highlighted with white lines. This can make it easier to see the cut edges and faces.

Note that for a large model you may experience graphical performance loss when cut edges and/or faces visibility is set to true.

Calculation of cut edges and faces relies on the model geometry being topologically closed, which is not always the case for model data in GeniE. In such cases it is possible that slightly incorrect edges and faces can be generated.

## Create Cut Plane

This opens the Create Cut Plane Dialog. You can choose between different creation methods as described below.



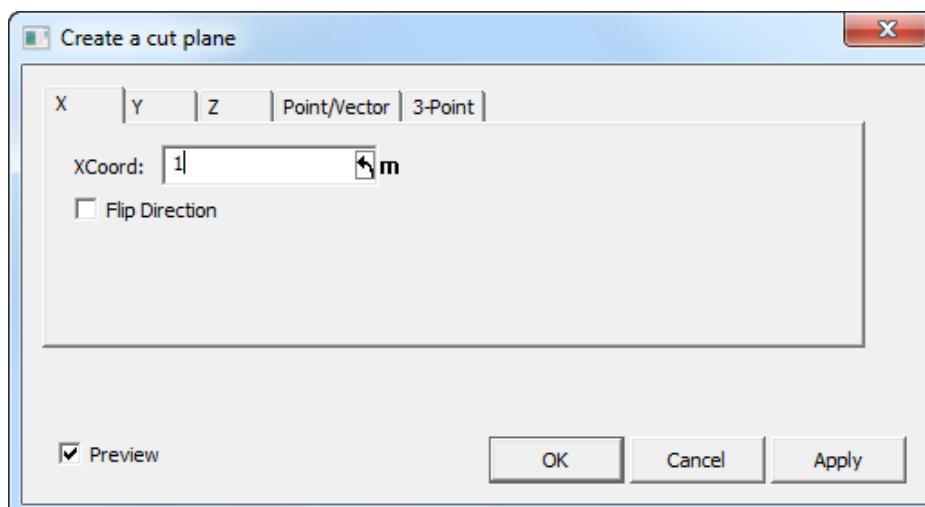
### Create a cut plane by typing in an X, Y or Z coordinate.

By typing in a value for X, Y or Z you can create a cut plane perpendicular to the axis you have selected.

By giving in 1 as in the illustration, a cut plane is created at X=1.

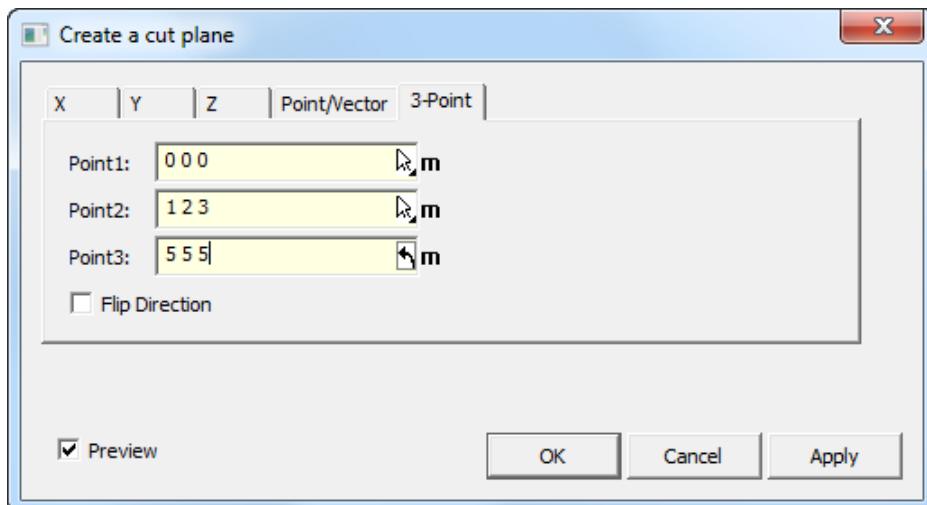
By default everything below X=1 will be displayed.

If you check the “Flip direction” checkbox, everything above X=1 will be displayed.



### Create a cut plane by giving in a point and a normal

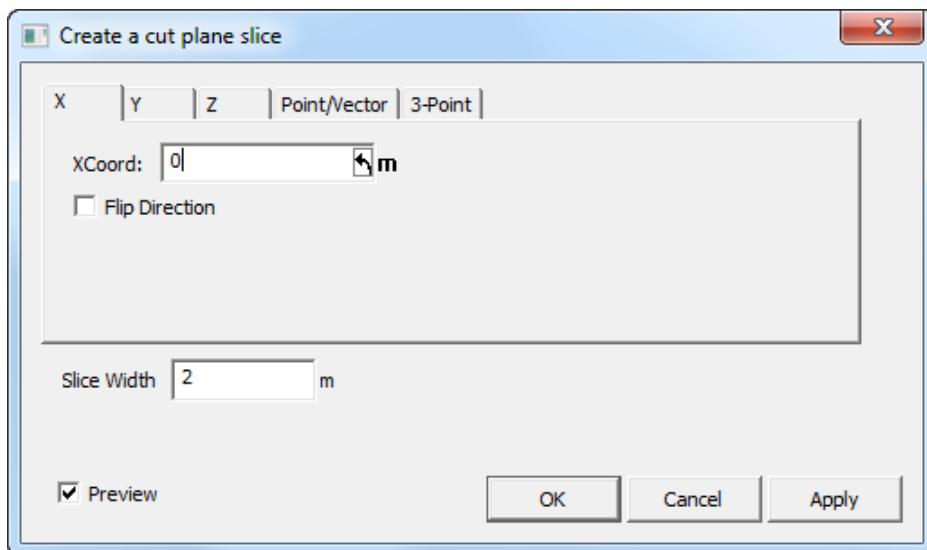
By typing in coordinates for a point and a normal vector a cut plane is created. The point given will be on the plane and the normal vector will be the plane's normal vector. By default everything on the opposite side of the normal vector will be displayed. You can change this by checking the "Flip direction" checkbox.



### Create a cut plane by typing in 3 points

By typing in coordinates for 3 different points a cut plane is created. The points must be different, and they cannot be on the same straight line.

### Create Slice



This works similar to "Create cut plane" described above. Note that in addition to the options for "Create cut plane" you also get to set the slice width.

The example illustration above will create a slice perpendicular to the x-axis, starting at X=-2 and ending at X=0.

## **Create X Cut Plane**

Create a cut plane perpendicular to the X-axis. The plane will be positioned at the middle of the model.

Once a plane has been created the current command is set to "Drag cut plane" so you can drag the new cut plane into position. (This is also the case for Y and Z cut planes and slices)

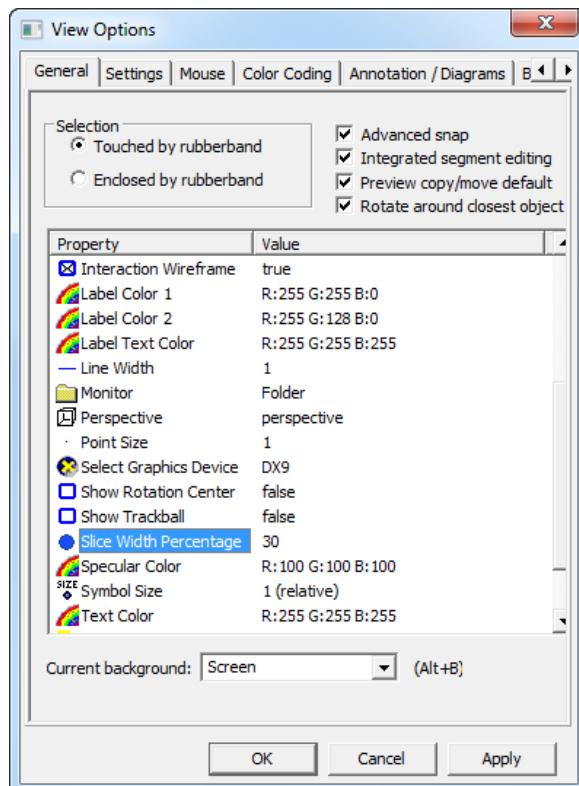
## **Create Y Cut Plane**

Create a cut plane perpendicular to the Y-axis. The plane will be positioned at the middle of the model.

## **Create Z Cut Plane**

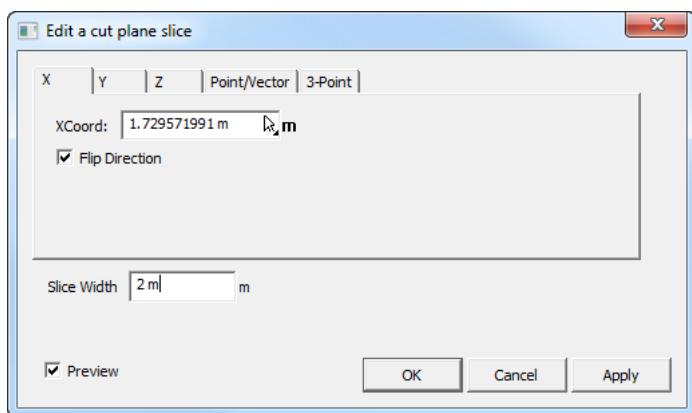
Create a cut plane perpendicular to the Z-axis. The plane will be positioned at the middle of the model.

## Slices



A slice consists of two cut planes and allows you to view a slice of your model. The width of the slice is set under the General Tab in View | Options as shown in the illustration to the left.

You can edit the slice's width by rightclicking one of its cut planes in the browser or in the graphics, selecting "Edit Cut Plane" and changing the value for "Slice Width" as shown in the illustration below.



### Create X Slice

Create a slice perpendicular to the X-axis.

### Create Y Slice

Create a slice perpendicular to the Y-axis.

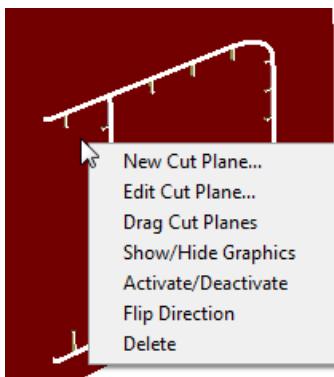
### Create Z Slice

Create a slice perpendicular to the Z-axis.

### Drag Cut Plane

Choose this to drag a cut plane or a slice in the direction of the cut plane's normal vector.

## Cut plane manipulation



Rightclicking a cut plane, either in the graphics or in the browser opens up the rightclick menu. The different menu items are explained below.

### New Cut Plane

Opens the Create Cut Plane dialog explained earlier.

### Edit Cut Plane

Opens the Edit Cut Plane dialog, that behaves in the same way as the Create Cut Plane dialog except you use it to manipulate an existing cut plane

### Drag Cut Planes

Makes it possible to drag the cut planes in the cut plane's normal vector by leftclicking and dragging with the mouse.

### Show/Hide Graphics

Makes the cut plane visible/invisible. To be able to select the cut plane it needs to be visible. To select structure behind the cut plane it needs to be invisible.

### Activate/Deactivate

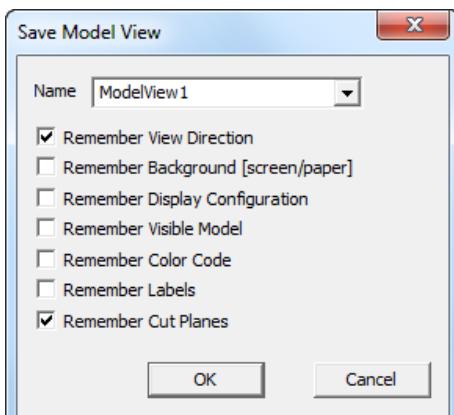
Activates and deactivates a cut plane. If a cut plane is deactivated the graphics are displayed as if the cut plane didn't exist.

### Flip Direction

Flips the direction of what's visible, either everything in front of or behind the cut plane. Note this is generally only useful for single cut planes and not slices. Cut planes act on the complete half space and are not bounded in any way. Therefore if two parallel cut planes point towards each other no graphics will be visible. Therefore flip direction for a slice simply moves the slice along the normal of the planes that make up the slice.

### Delete

Deletes the cut plane. Note that in the browser you can act on an individual cut plane using the RMB menu, or on all cut planes by RMB on Utilities/Cut Planes



When you are saving a model view it's possible to make the model view remember the cut planes by checking the "Remember Cut Planes" checkbox.

## 3.2 The design premise

GeniE may also be customised to ease the modelling activities during a project. This Section explains which parameters that can be adjusted to make modelling as efficient as possible. The settings may be changed at any time during a modelling session. The settings are available from the top dropdown menu; Edit | Rules

- [Beam creation ...](#)
- [Transformation ...](#)
- [Joint creation ...](#)
- [Joint design ...](#)
- [Tolerances ...](#)
- [Connected move ...](#)
- [Geometry ...](#)
- [m/s Units ...](#)
- [Meshing ...](#)
- [Sets ...](#)
- [Compatibility](#)

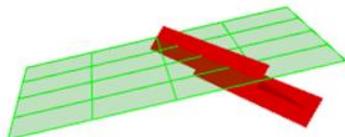
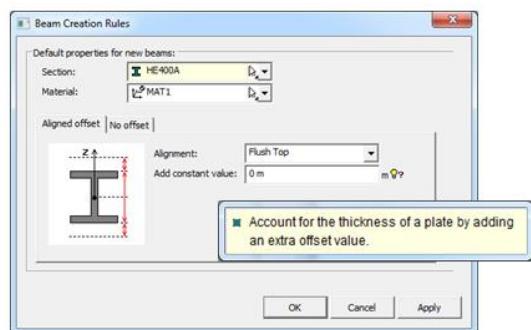
### 3.2.1 Rules for beam creation

The Beam Creation Rules has an option to do Auto flushing when defining a member.

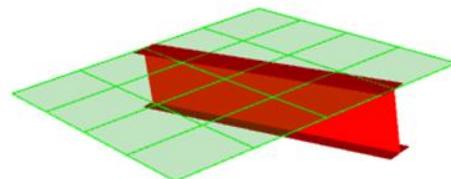
No offset is the default setting, and the user should revert to this option when modelling objects without offsets from the centre line.

Beams created when the Aligned offset option was enabled as default will automatically recompute the offsets when changing the section type or dimensions.

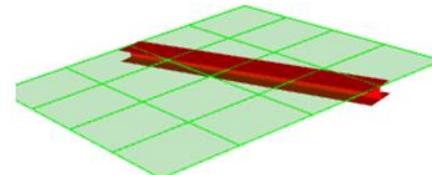
If the user specifies a rotation around local x-axis, this will be in the reference plane and not around the eccentric x-axis.



Rotation around axis in reference plane



Auto flushing when inserting member

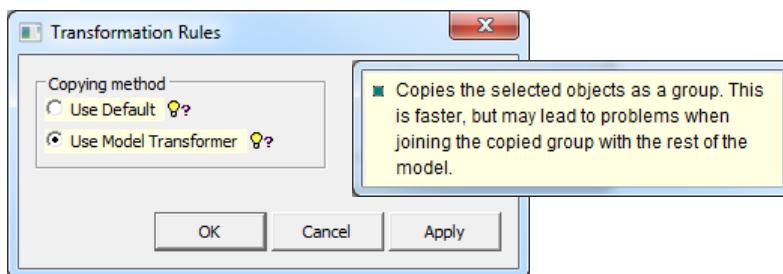


Auto flushing when changing member

### 3.2.2 Rules for transformation

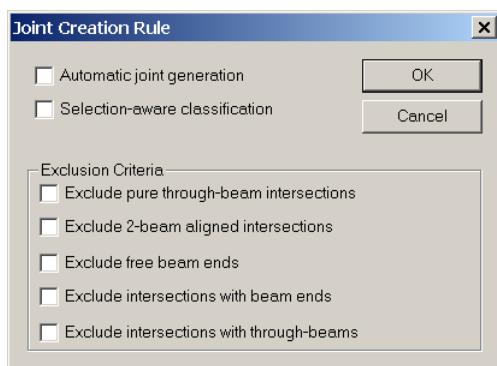
A new method named “Model transformer” has been introduced for copying. Instead of copying structural components one by one they are with the new feature handled as a single body and copied in one operation. For large and complex models this may often give better results, i.e. there is reduced chance for parts not being copied or that errors appear in the copied part.

For small and medium size models, the old copy method is normally a safe method and it is therefore the default option in GeniE for copying.



### 3.2.3 Rules for joint creation

When making the details of a tubular joint it is necessary to define a Joint at the actual point. You may set up GeniE so that such joints are created automatically. If not, you can always insert these manually afterwards from the **Insert|Joint** pulldown menu or by selecting beams, and choose generate Joints from the context sensitive menu.



When automatic joint generation is ticked off, joints will be inserted during modelling according to the exclusion criteria. The exclusion criteria also applies when using the context sensitive menu

The first one excludes joint generation when two beams intersect, for example two braces forming an x.

The second one is when two beams are connected along a straight line.

The third option is at a free beam end (i.e. no other beams connected).

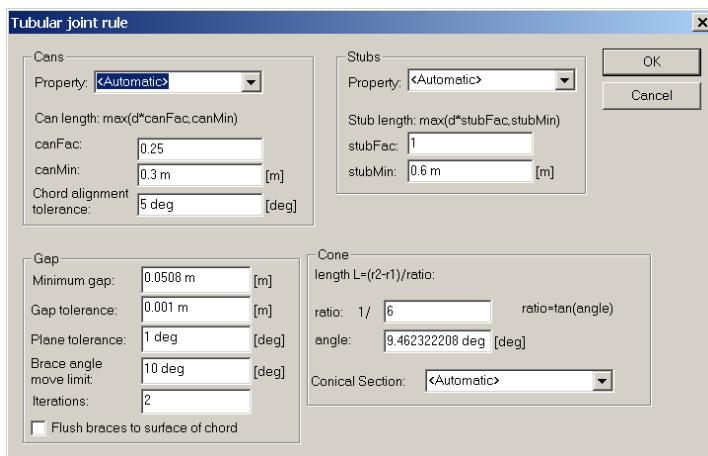
The fourth option excludes joints at all locations where beams start or ends.

The final option excludes joint of type Y/T and K with a through chord beam.

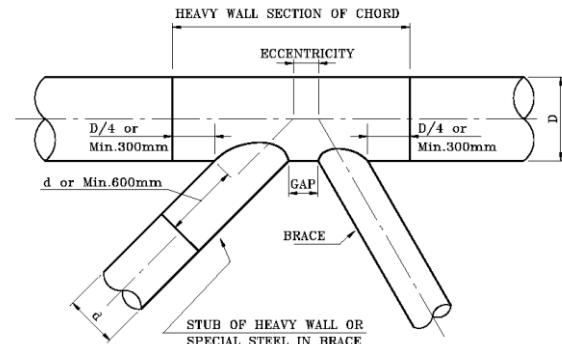
Joints are also used for referencing structural joints (tubular and non-tubular) to have a unique name that will not change when the model is changing. This means you can refer to unique names e.g. when doing code checking in Framework.

### 3.2.4 Rules for joint design

GeniE will automatically create the cans, stubs, cones and do planewise gap calculations based on a set of pre-defined rules. The rules come with default settings found in several standards like e.g. API and NORSO. These settings may be modified by using the command **Edit/Rules/Joint Design**.



The settings in this case are given in SI units and they are all explained in the following.

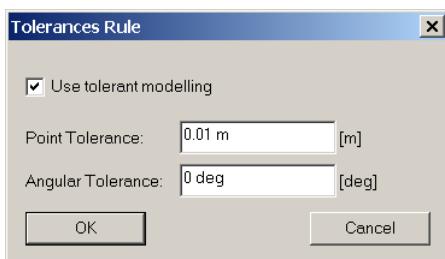


The parameters are – see figure above for references:

Parameter	Explanation
Canfac	Free can length as fraction of can diameter (default = 1/4 of Diameter of chord/can)
Canmin	Minimum free can length (default = 0.3 m)
Chord alignment tolerance	Angle wrt. whether a beam is treated as aligned chord or brace
Stubfac	Free stub length as fraction of stub diameter (default = diameter of stub/brace)
Stubmin	Minimum free stub length (default = 0.6 m)
Minimum gap	Minimum gap (default = 0.051 m)
Gap tolerance	Stop iterating when calculated gap is within minimum gap +/- gap tolerance
Plane tolerance	Tolerance for braces considered being part of a plane (default = +/- 1 degree)
Brace move limit angle	Specifies the smallest angle between a brace and a perpendicular to the chord which will introduce an eccentricity in the brace end when doing planewise gap calculations (default = +/- 10 degrees)
Cone ratio	Cone length (default corresponding to angle 1:6)
Cone angle	Cone angle (default corresponding to 1:6)
Iterations	Number of iteration loops when calculating plane wise gaps (default = 2)
Flush braces to surface of chord	Insert eccentricities along brace length axis. Size of eccentricity equal to distance from centre line to chord surface along brace length axis.

### 3.2.5 Rules for tolerances

You can decide whether to work with tolerant modelling or not. Furthermore, you can decide the level of tolerances. The settings are activated from pulldown menu **Edit/Rules/Tolerances**.



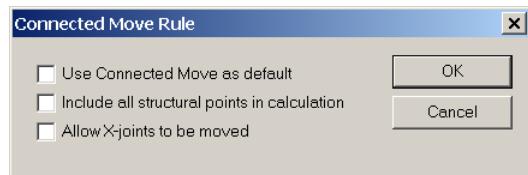
The first option ensures that one topology point is being used when input deviation is less than the tolerance. For example the end of a beam is located at (0m, 0m, 0m) and one end of a new beam is inserted at (0m, 0m, 0.005m) the first point is being used. In other words, the end point of the new beam is snapped to the existing beam.

Tolerant modelling is default option when making a new workspace.

Working with angular tolerances will ensure an aligned beam between several topology points depending if the angle between several beams are within the tolerance.

### 3.2.6 Rules for connected move

When moving structure you may decide to move the selected object(s) only or also to include the connected members. The settings for such are activated from **Edit/Rules/Connected move**.

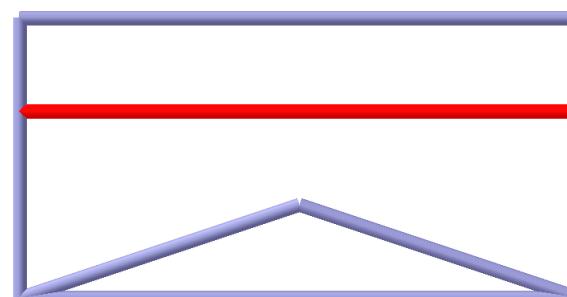
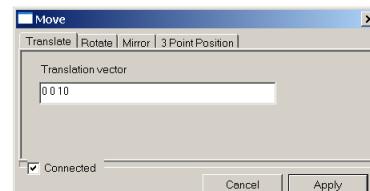
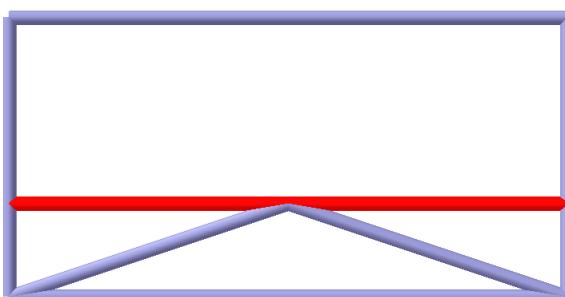
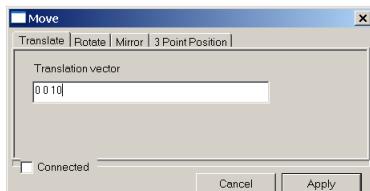
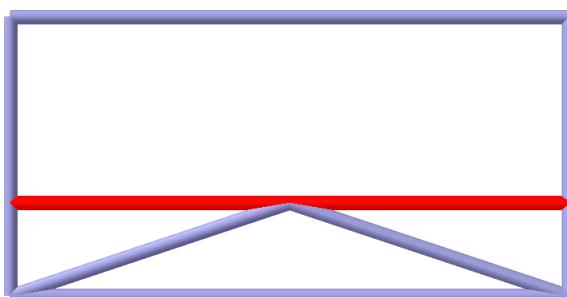


When first selection is ticked off the connected move option in the dialogue for move is always ticked off.

The second option will ensure that belonging parts are part of a move operation, see examples below.

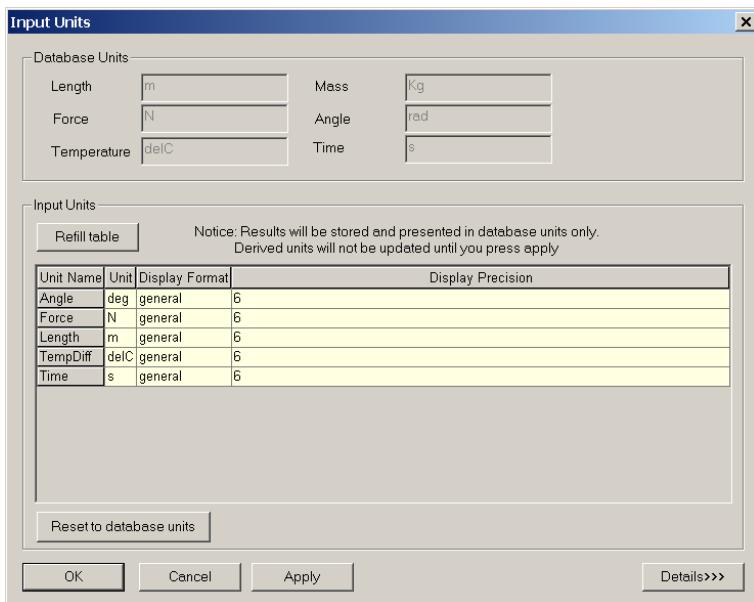
The third option will also include joints in X-joints to move.

The examples below show the difference between a regular move and connected move. The regular move will move the selected beam only, while the connected move will move the associated beams also.



### 3.2.7 Rules for units

GeniE allows modelling in a number of input unit sets – the program will automatically convert to the database units specified when a new workspace was created. The current input unit sets will be shown in any input field, graphic dimensions, and on the reports created. Note that the interface files created for use by other SESAM programs will be according to the database units. Also, when assessing the results in GeniE, they will also be given in database units. The input units are changed from the pulldown menu **Edit/Rules/Units**.



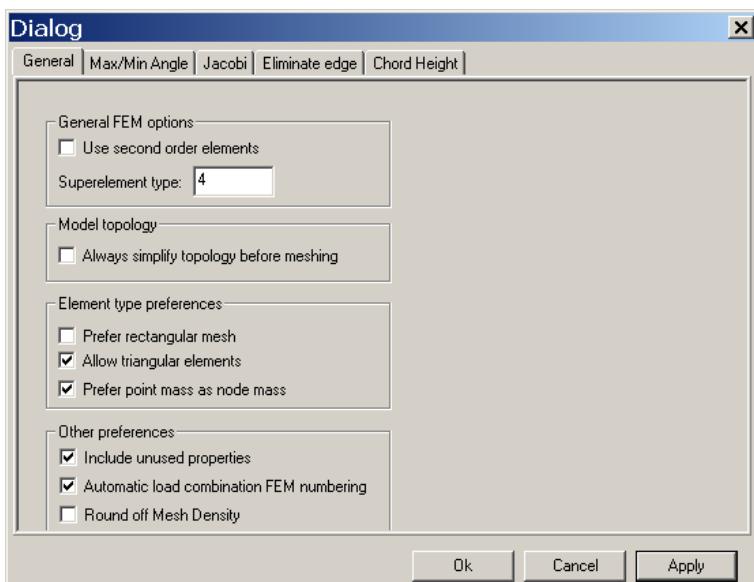
When changing any of these units, the new units will be used in e.g. input fields and on dimensions. They will appear according to the settings chosen.

For example changing length from meters to inches with display format fixed and display precision equal to 3 will result in dimensions with 3 decimals.

You may change several other units from the *Details* option.

### 3.2.8 Rules for meshing

GeniE is capable of generating a satisfactory finite element mesh based on the concept model. However, there may be instances where the user needs to instruct the program on how to create the finite element mesh either because the program cannot create the mesh or the user wants to be in full control of the generated mesh and loadcase numbers. The settings for controlling the quality of the finite element mesh are found under **Edit/Rules/Meshing**. There are five ways of controlling the quality and how to create the mesh, observe that mesh density is specified per objects (or globally).



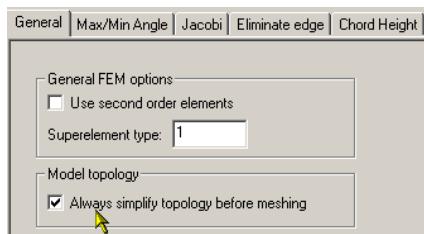
You may also specify that you want to use 2<sup>nd</sup> order finite elements when creating the finite element model. 1<sup>st</sup> order elements are the default.

Per default the super element type is set to 1 which is also being used when running a direct analysis (running Sestra and no super element analysis).

When you create a finite element model that is intended for use in a superelement analysis you can specify the superelement number, in this case number 4. The file to be read by Presel is then T4.FEM.

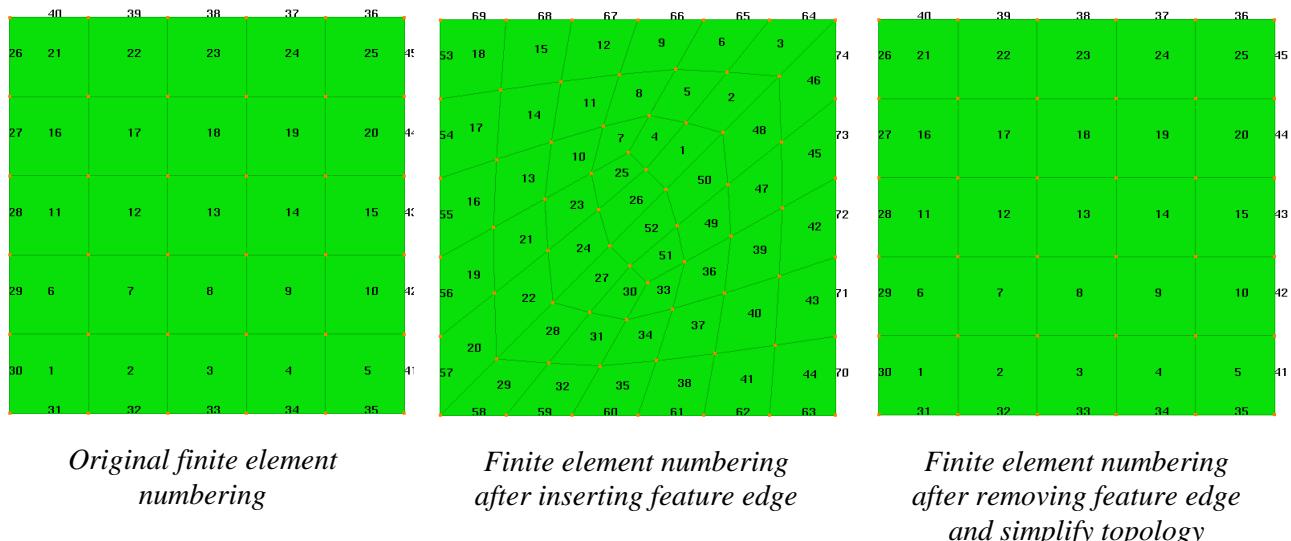
For more details on loadcase numbering see Chapter 3.8.5.

Notice that it is also possible to decide triangular elements or not. If you want to transfer unused properties (e.g. materials or sections) you must tick off the option as shown above. This is important in the case you want to carry all the properties defined inside GeniE to e.g. a re-design in Framework.



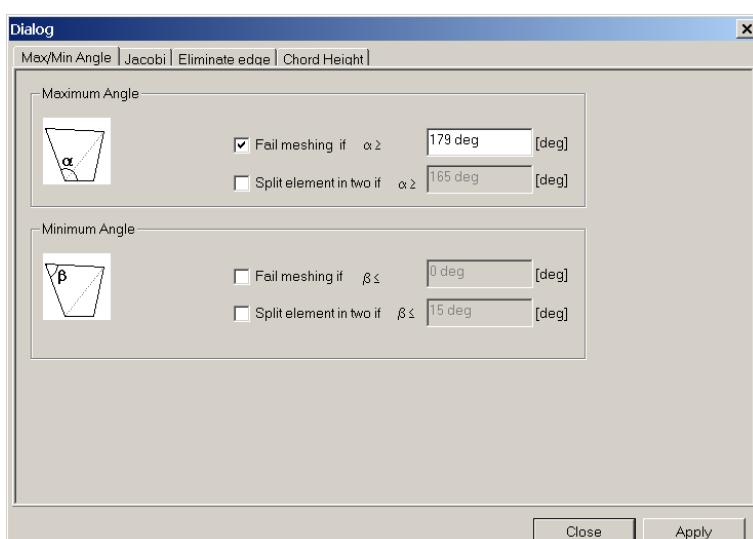
You may also instruct GeniE to always perform a simplify topology prior to making a finite element mesh, you may also do simplify topology manually from **Tools/Structure/Geometry/Simplify Topology**.

The operation will remove unnecessary topology points and lines in the model that will simplify the finite element model. It works for both straight beams and planar plates. In the example the finite element mesh is showed for a plate surrounded with beams, then intersected with a feature edge from lower left corner to upper right corner, and finally the feature edge is removed and simplify topology is performed.



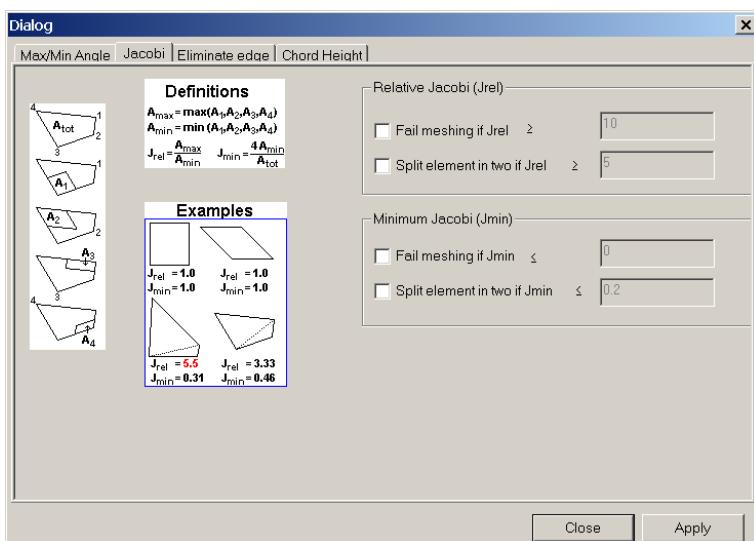
As can be seen, the simplify topology operation leads to a finite element mesh identical to the original mesh.

When you have specified a certain mesh density GeniE will ensure that the FE mesh created contains elements less than that size. The option *Round Off Mesh Density* instructs GeniE to create a mesh where the maximum size is approximate to the specified mesh density.

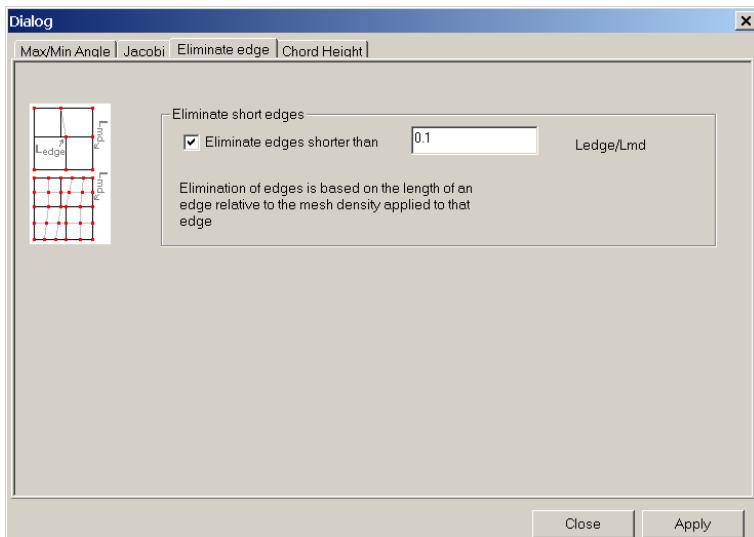


The settings here will instruct the program to fail meshing and inform you if Fail Meshing Angle is larger or smaller than given criteria.

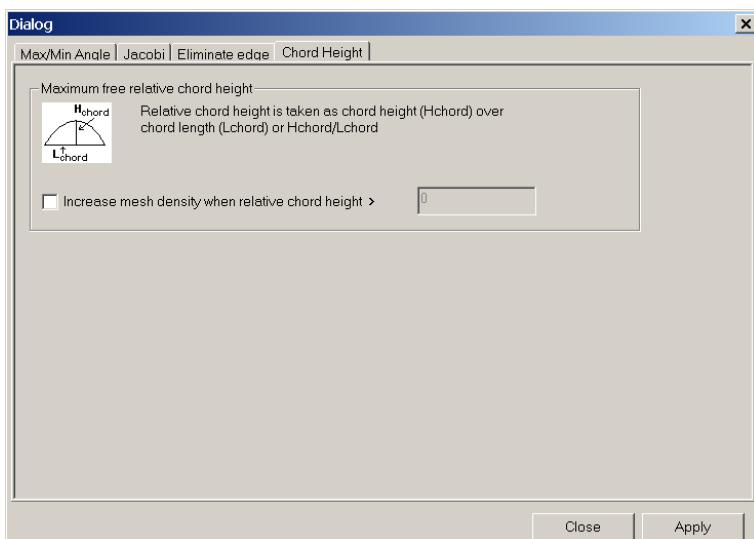
Similarly, you can instruct the program to split a quadrilateral element into 2 triangular elements if the element angle is larger or smaller than given criteria.



You may also perform a Jacobi matrix test of the finite element mesh. You can specify the failing criteria and whether the program shall insert triangular elements if the failing criteria are exceeded.



If you have modelled without tolerant modelling and not been accurate you may end up with very short edges that will lead to very small finite elements. You may bypass this problem by using the feature for eliminating edges as shown to the left. This feature should be used with care since it may simplify the structure that has been modelled.



If there are curves in your model, e.g. when you have used a punch operation, the default mesh density on the face may not be fine enough to get a finite element mesh that properly represents your model.

You may use the chord height option to force the mesh to be closer to the curves. The relative chord height is a measure of the curvature you allow each element to represent.

The length of the edge of the element will be shortened to make the chord height smaller than the maximum set.

### 3.2.9 Rules for scripting of sets

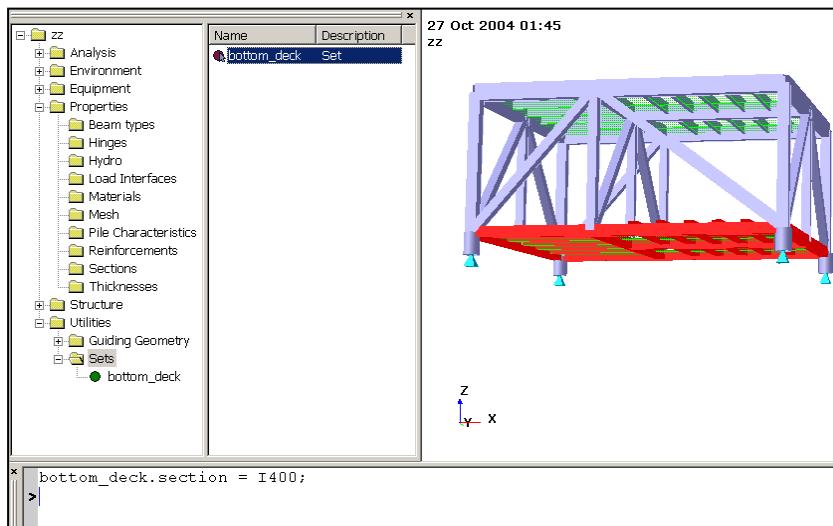
It is possible to instruct GeniE on how to script certain operations of sets. This is available from the *Edit/Rules/Sets*.



The *Compact Scripting* causes a single scripting command to be journalled, like e.g. `bottom_deck.section=I400;`. In this case all beams in the set named *bottom\_deck* receive section type I400. This is the default option.

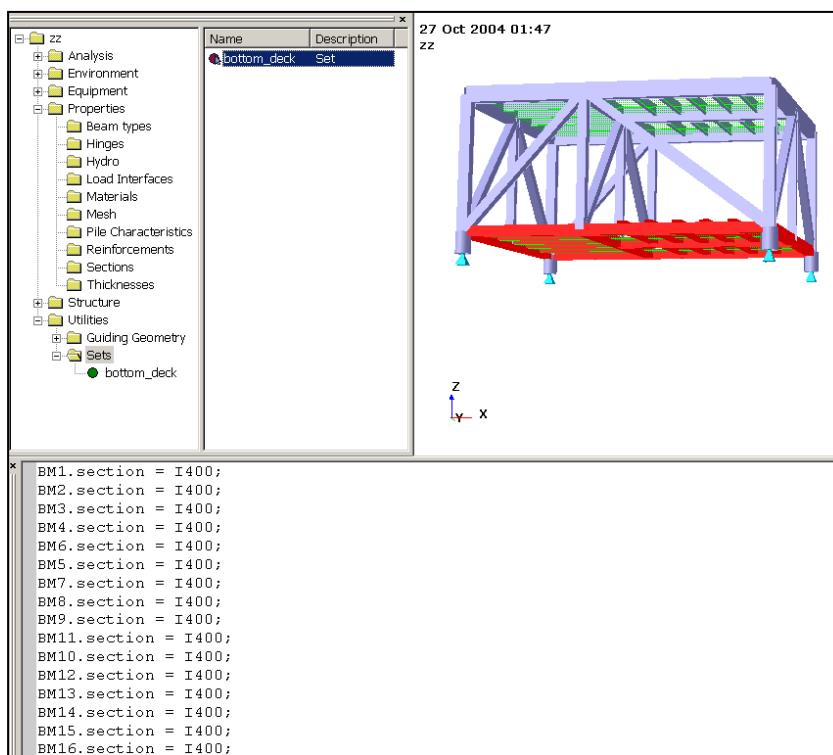
The *Verbose Scripting* enforces one scripting command per objects in the selection.

*Compact Scripting* works for operations like *Move* and *Add/Change properties*.



The *Compact Scripting* is chosen.

When selecting the set *bottom\_deck* and applying the section type *I400* creates one scripting command in the journal file.

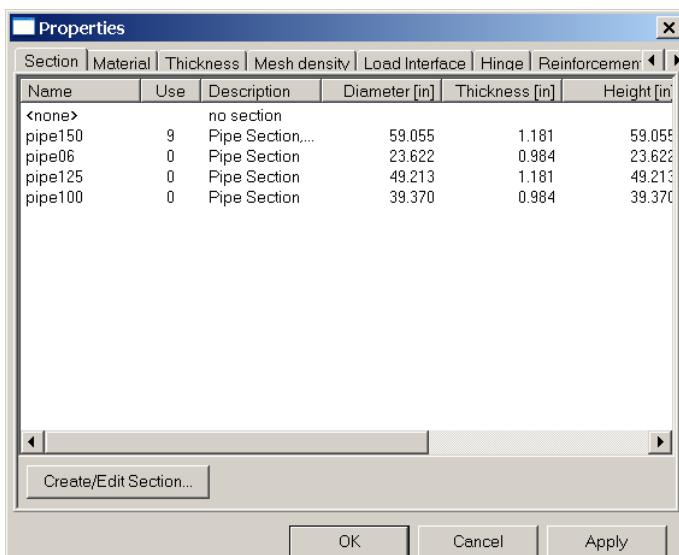


The *Verbose Scripting* is chosen.

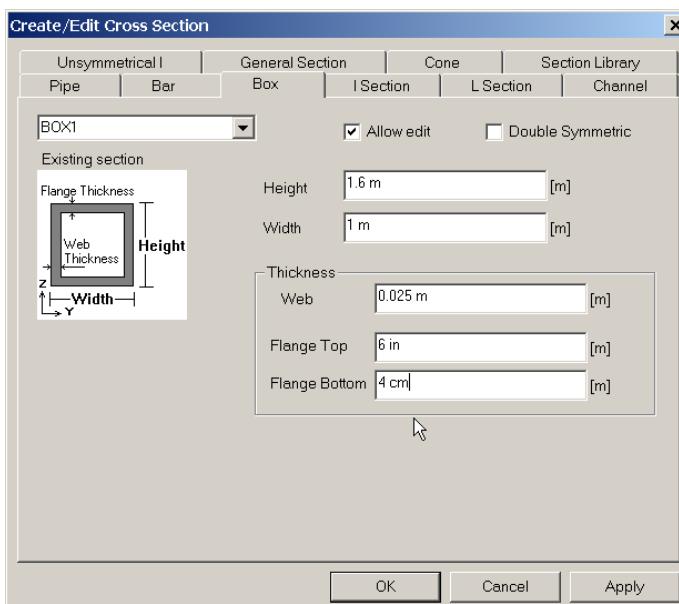
When performing the same operation as above, a scripting command is generated for all beams part of the set.

### 3.2.10 Property libraries

Libraries for sections, materials, and plate thicknesses are made from the pulldown menu **Edit/Properties**.



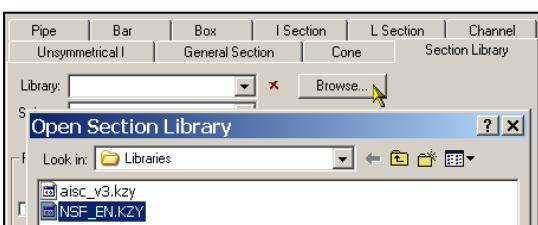
The example to the left shows that a total of 4 section types have been created. A common way of defining libraries is to start creating the properties fro the GUI and continue adding the remaining properties in a journal file. This journal file can be imported to new workspaces to create your library for sections, plates, and materials.



The supported cross section types are Pipe, Bar, Box, I-section, L-Section, Channel, Unsymmetrical I, General Section, and Cone. In this example a Box profile has been created. Observe that input units may be given in a unit different from the specified input unit.

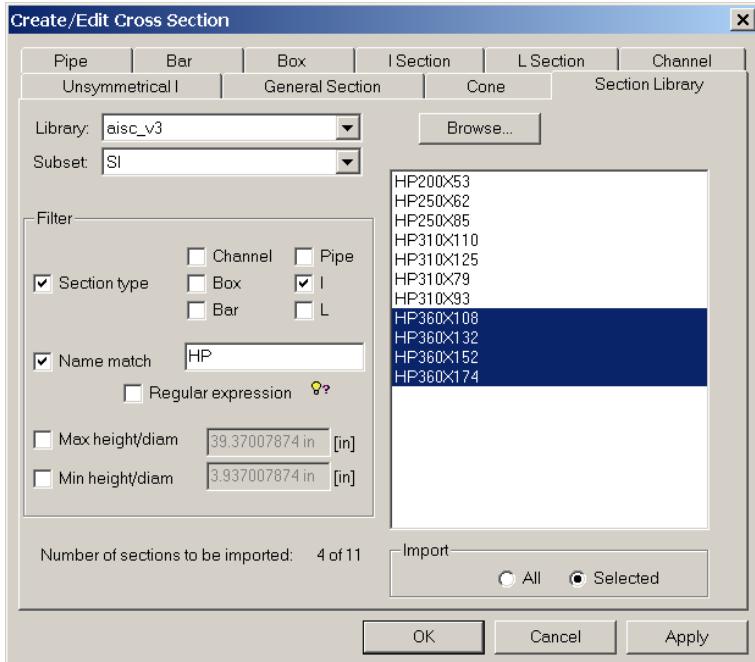
It is also possible to vary the flange thickness for I-section and Box, to the left is shown a Box with different flange thicknesses.

You may also change the shear factor for all section types except for the general profile. This feature is currently available from the Command Line Interface by using e.g. the commands **BOX1.shearFactorY=0.5;** and **BOX1.shearFactorZ=0.8;**. The y and z notations refer to the beam's local coordinate system.



GeniE also supports the AISC and the Euronorm & Norwegian Standard section libraries. You may import the whole or parts of the library for later use in your workspace – in most cases you import those you need (the library consists of more than 1000 profile types). There is also a filter option that will help you when selecting the actual section types.

When selecting profiles from these libraries, these are logged on the journal file. This means that you don't need to import the profiles every time you recreate your model.



The example to the left shows an import of 4 section types using filtering options.

The corresponding commands logged on the journal file are shown below.

```
// Importing sections from library
HP360X174 = ISection(361mm,378mm,20.4mm,20.4mm);
HP360X152 = ISection(356mm,376mm,17.9mm,17.9mm);
HP360X132 = ISection(351mm,373mm,15.6mm,15.6mm);
HP360X108 = ISection(346mm,370mm,12.8mm,12.8mm);
```

You may also import the section library directly from the pulldown menu **File/Import/Section library**.

You may also easily get access (change, delete, insert new) to all the properties defined from the *Browser* under *Properties*.

### 3.2.11 Compatibility

The user should refrain from changing these settings.

Compatibility options are automatically selected based on the specified version in the JS file read into GeniE, where for example : `GenieRules.Compatibility.version = "V6.9-05";` means that a JS file was originally created by and should be compatible with the mentioned version of GeniE.

### 3.3 Guiding geometry

Guiding geometry is a mean to ease modelling whereby known points and curves are defined for later usage. The guiding geometry is not connected to any structural part and/or the finite element mesh.

When modelling a guiding point or guiding curve graphically, we use only snapping points, i.e. significant points on the previously created geometric entities (e.g. the start and end positions of a beam or the four corners of a plate or the vertices of a polygonal arc). A snapping point is also automatically created where two beams or two guiding curves intersect. We can explicitly define snapping points on guiding geometry curves by using the GeniE command-line window and the java-script command:

```
Curve.spacings (Array(double));
```

Then, snapping points appear on the curve at positions proportional to the elements in the given array.

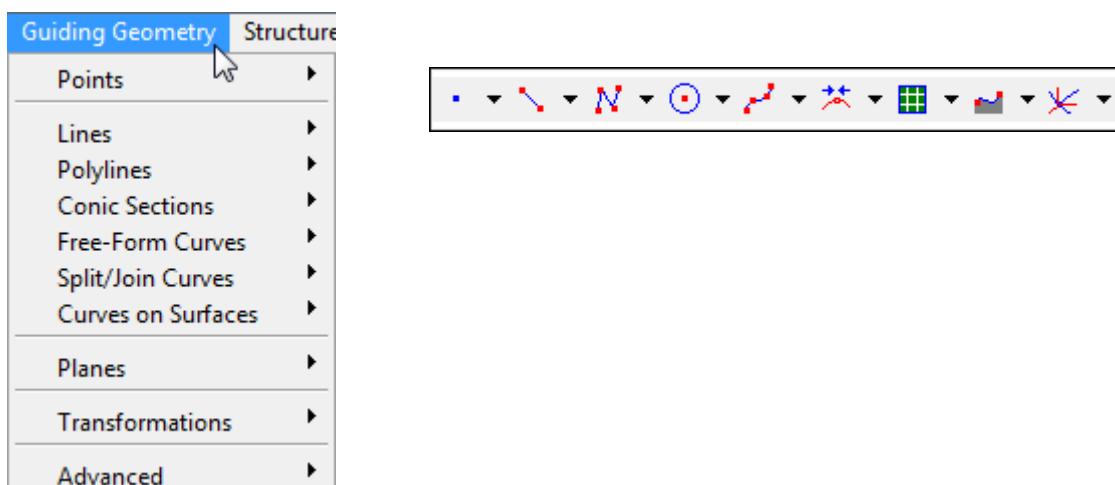
Alternatively, one can create guiding geometric entities through java-scripting commands. That is why in what follows we give in parallel the graphical usage and the js-commands.

This chapter lists the available guiding geometric entities and the methods for creating them. All types are available from the *Browser* under *Utilities* and *Guiding geometry*. Using the default settings, the guiding geometries are visible using the predefined views *Default* and *Modelling – All*. In what follows we are using the following conventions:

- a. <Entity> means the name of a new entity, such as Point, Curve or Guide-Plane.
- b. Rename (tmpArrayOfPoints[i], "<Point>") refers to a sequence of N-points implying:  
Rename (tmpArrayOfPoints[0], "<Point1>");  
Rename (tmpArrayOfPoints[1], "<Point2>");  
...  
Rename (tmpArrayOfPoints[N-1], "<PointN>");
- c. Script variables marked with red are default-constants, which currently cannot be modified in the GUI.

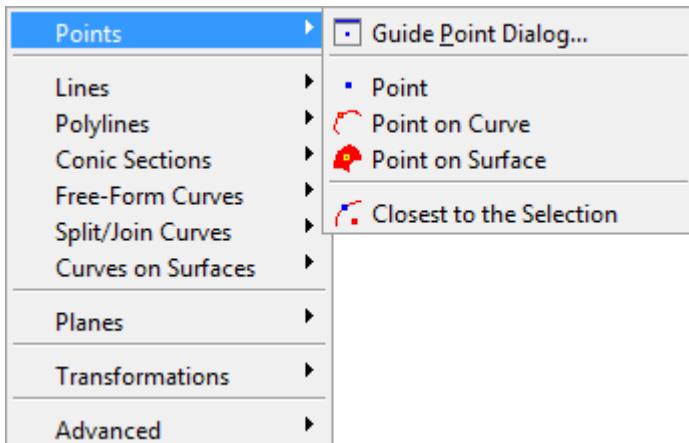
Inserting guiding geometric entities graphically is made available from the pull-down menu and the toolbar (see figures below).

Note that the menu item called “Advanced” is only available from the pull-down menu.

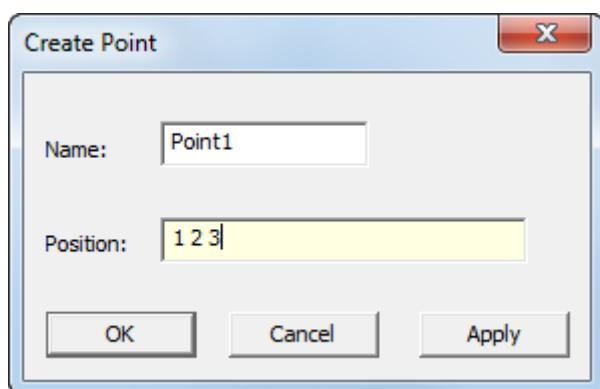


### 3.3.1 Guide points

You can insert guide point(s) the following ways:



**Explanation:** Opens the Create Point dialog. Here you can give the name and position of the guide point that you want to create. A guide point is a snap point which can be used later to create structure or other guiding geometry.

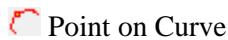


**Explanation:** The command creates a guide point. The guide point is a named snap point which can be used later to create structure or other guiding geometry.

**Graphically:** Select a snap point and create a single guide point out of it.

**JS-command:**

```
<Point>=Point(x, y, z);
```

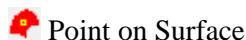
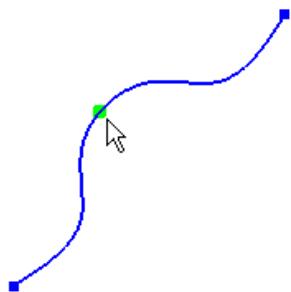


**Explanation:** Creates a guide point on a curve, nearest to the selected position.

**Graphically:** Select a point on a curve.

**JS-command:**

```
<Point>=Point (x,y,z);
```

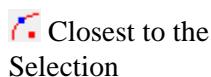
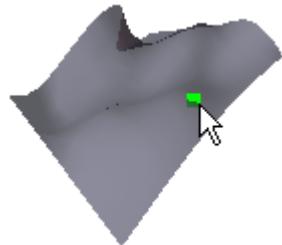


**Explanation:** Creates a guide point on a surface, nearest to the selected position.

**Graphically:** Select a point on a plate/shell.

**JS-command:**

```
<Point>=Point (x,y,z);
```



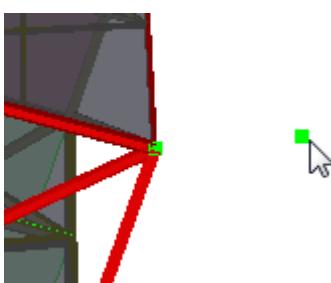
**Explanation:** The command creates a guide point on a pre-selected set of objects that is the closest to the given point.

**Graphically:** Before pressing the button, select a group of objects (Selection). Select a point.

**JS-command:**

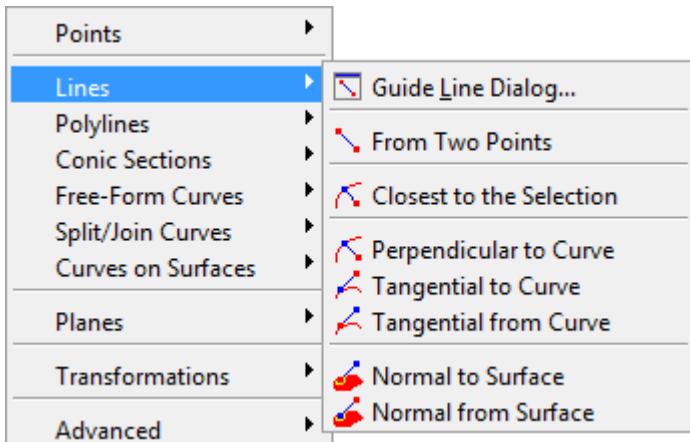
```
<Point>=Point (Point, Array(objects));
```

A guide point is created on the selection on the closest possible location to the point you select. In the example to the left, the green point on the left, on the red selection, is created when the user clicks on the green point to the right.

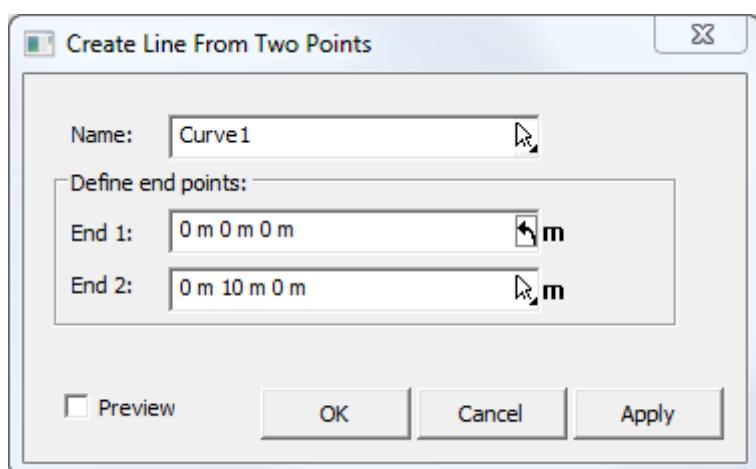


### 3.3.2 Guide lines

You can insert guide lines the following ways:



**Explanation:** Opens the Guide Line dialog. Here you can give the name and start and end position of the straight guide line that you want to create.



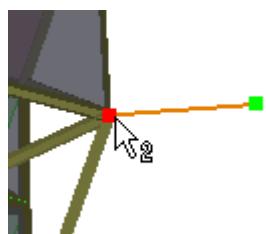
**Explanation:** Creates a straight guide line between two points. The points can be guide points or any other snapping point like parts of the structure.

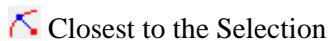
**Graphically:** Select the two end points of the straight line.

#### JS-command:

```
<Curve> = CreateLineTwoPoints(Point1, Point2);
```

In the example to the left a guide line is created from a guide point to a snapping point in the structure.



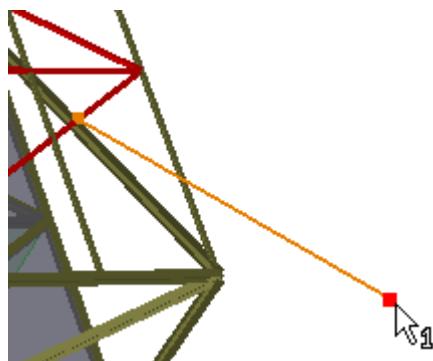


**Explanation:** Creates a straight guiding line between a pre-selected set of objects and the given point. The selection should be done prior to calling this function.

**Graphically:** Before pressing the button, select a group of objects (Selection). Then, select a snapping point on the screen.

**JS-command:**

```
<Curve>      =      CreateLineClosestToSelection(Point1,  
Array(objects));
```



The selection marked in red is selected before you use the function. When you click on a point a guide line is created between the selection and the point that you selected. The guide line hits the selection in such a location that it is as short as possible.

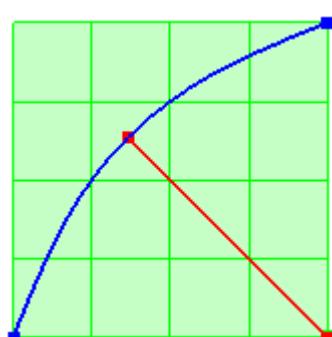


**Explanation:** Creates a straight guide line between the given point and a point on the selected curve so that the guide line created is perpendicular to the curve.

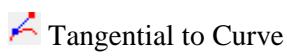
**Graphically:** Select a point and then select a curve.

**JS-command:**

```
<Curve>  
CreateLineFromPointPerpendicularToCurve(Point,  
Curve);
```



In the illustration to the left the point at the bottom right is clicked first. Then the guide curve, shown in blue, is selected. A straight line, shown in red, from the point to the guide curve is created. The line is perpendicular to the guide curve.



### Tangential to Curve

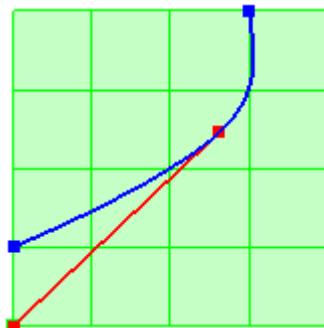
**Explanation:** Creates a straight guiding line from the given point a point on the selected curve so as to be tangential to the curve.

**Graphically:** Select a point and then select a curve.

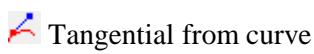
**JS-command:**

```
<Curve>=
```

```
CreateLineFromPointTangentialToCurve (Point, Curve);
```



In the illustration to the left the point at the bottom left is clicked first. Then the guide curve, shown in blue, is clicked. A straight line, shown in red, from the point to the guide curve is created. The line is tangential to the guide curve.

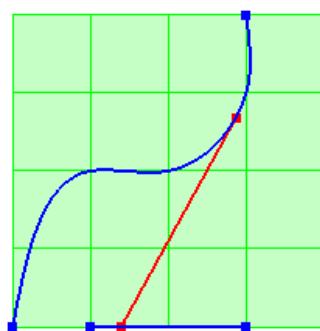


**Explanation:** Creates a straight line tangential from a curve.

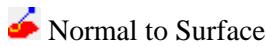
**Graphically:** Select a snapping point on a curve. Then, select a point on the line which is tangential to the curve.

**JS-command:**

```
<Curve> = CreateLineTwoPoints (...)
```



Example of a line tangential from a curve. In this example a point on curve at top is selected first. Then the horizontal line on the bottom is selected. The function creates a tangent from the first curve selected down to the second curve selected.



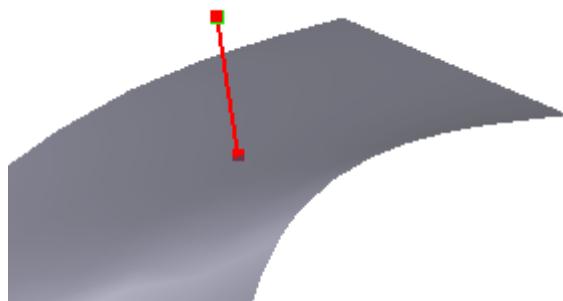
Normal to Surface

**Explanation:** Creates a straight guiding line between the given point and a point on the selected surface (plate/shell) so as to be normal to the surface.

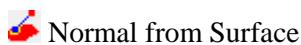
**Graphically:** Select a point on the screen and then select a plate/shell.

**JS-command:** <Curve> =

```
CreateLineFromPointNormalToSurface(Point, Surface);
```



In the illustration to the left the point at the top is clicked first. Then the curved surface is clicked. A straight line is created between the point and the surface. The line will hit the surface in the location where it is normal to the surface.



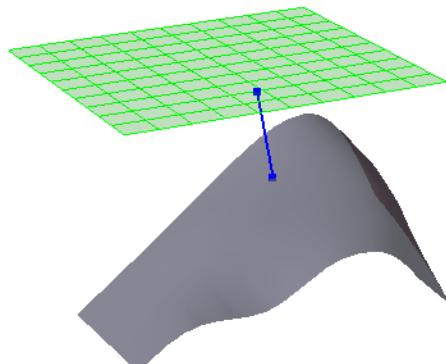
Normal from Surface

**Explanation:** Function creates a normal from a point on a surface to another object.

**Graphically:** Select a snapping point on a surface. Then, select a point on the line which is normal to the surface.

**JS-command:**

```
<Curve> = CreateLineTwoPoints(...)
```

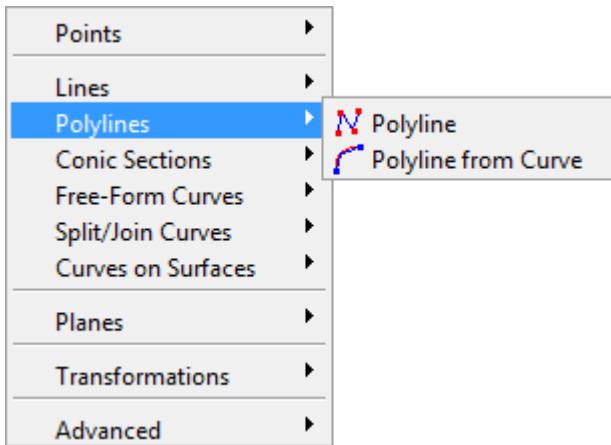


First you click on the surface at the point where you want the normal to be created. Then you click on another object, in this case a guideplan.

A normal will be created from the surface to the object you selected.

### 3.3.3 Guiding poly-lines

You can insert a guiding poly-line (a polygonal arc) using the following ways:

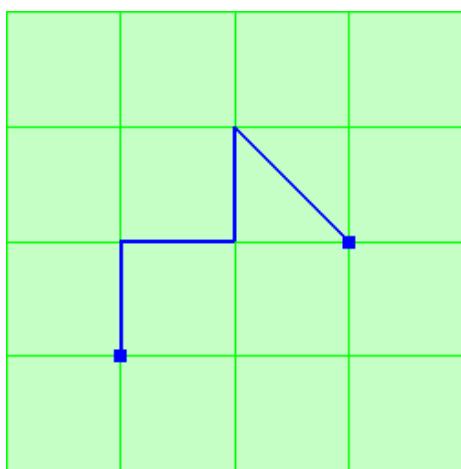


**Explanation:** The function creates a polyline from a set of ordered selected points.

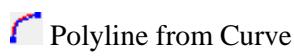
**Graphically:** Select a number of snapping points. Then, a polyline will be created connecting the points, in the same order as you clicked them.

**JS-command:**

```
<Curve> =  
CreatePolylineFromASetOfPoints (Array (Points)) ;
```



Example of a polyline. A guideplane has been used to snap the points of the polyline.

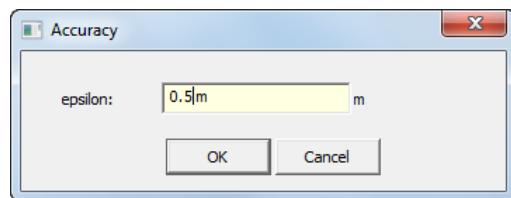


**Explanation:** This function creates a polyline approximation of a curve. When clicking on the curve the Accuracy dialog shows up. Here you can set the accuracy of the polyline. If you set the accuracy to a large value, like 0.5 m in the example below, you can observe that the polyline (shown in red) is an approximation to the curve (shown in blue).

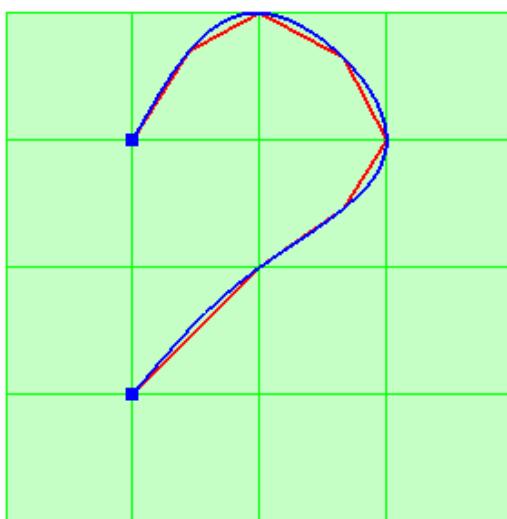
**Graphically:** Select a NURBS Curve. Then, give the tolerance in the dialog that appears.

**JS-command:**

```
<Curve> = CreatePolylineFromCurve(Curve, tol=0.01 m);
```

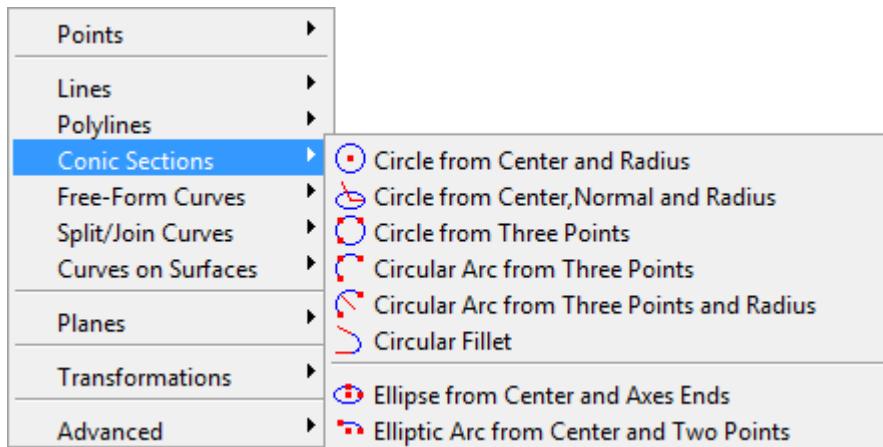


The illustration shows a polyline, shown in red, created from a curve, shown in blue. The accuracy of the polyline has been set to 0.5 m. A smaller accuracy will create a polyline being a closer approximation to the original curve.



### 3.3.4 Guiding conic sections

You can insert guiding circle or a guiding ellipse the following ways:



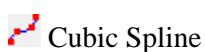
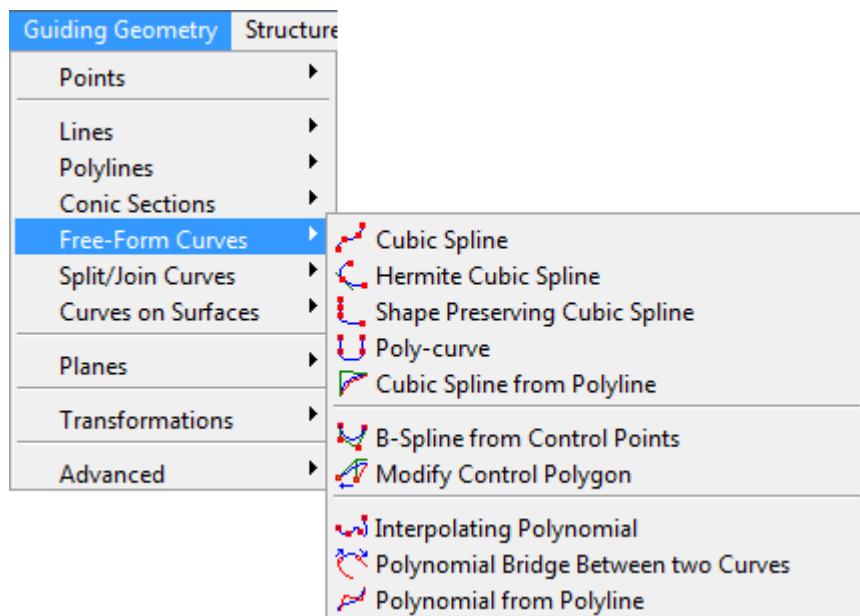
Circle from Center and Radius	<p><b>Explanation:</b> Creates a guiding circle from its center, radius and a third point determining the plane on which the circle is.</p> <p><b>Graphically:</b> Select a point to be the center of the circle. Select a point to be on the circle perimeter (along with the first one defines the radius). Select a point to define (along with the other two) the plane of the circle.</p> <p><b>JS-command:</b></p> <pre>&lt;Curve&gt; = CreateCircleFromCenterAndRadius (CenterPoint, RadiusPoint, PlanePoint);</pre>
Circle from Center, Normal and Radius	<p><b>Explanation:</b> Creates a guiding circle from its center, a normal and a typed in radius.</p> <p><b>Graphically:</b> Select a point to be the center of the circle. Select another point to define the normal to the circle's plane. Type in the radius of the circle.</p> <p><b>JS-command:</b></p> <pre>&lt;Curve&gt; = CreateCircleFromPlaneAndRadius (Point, Vector3d, Radius);</pre>
Circle from Three Points	<p><b>Explanation:</b> Creates a guiding circle interpolating three points.</p> <p><b>Graphically:</b> Select three points (not co-linear).</p> <p><b>JS-command:</b></p> <pre>&lt;Curve&gt; = CreateCircleFromThreePoints (Point, Point, Point);</pre>

 Circular Arc from Three Points	<p><b>Explanation:</b> Creates a guiding circular arc interpolating three points.</p> <p><b>Graphically:</b> Select three points (not co-linear). The function computes the circular arc interpolating them.</p> <p><b>JS-command:</b></p> <pre>&lt;Curve&gt; = CreateCircularArcFromThreePoints (Point, Point, Point);</pre>
 Circular Arc from Three Points and Radius	<p><b>Explanation:</b> Creates a guiding circular arc from two points, radius and a third point determining the plane of the circular arc.</p> <p><b>Graphically:</b> Select two points for the arc. Select a third point to define the arc's plane. Type in the radius.</p> <p><b>JS-command:</b></p> <pre>&lt;Curve&gt; = CreateCircularArcFromThreePointsAndRadius (Point, Point, Radius, Point);</pre>
 Circular Fillet	<p><b>Explanation:</b> Creates a circular arc –of the given radius- connecting two linear segments (from the same or from two different curves) with C1 continuity and returns the joined guiding curve.</p> <p><b>Graphically:</b> Select two points on two curves. Then give the radius in the appering dialog. The function joins all curve segments into one curve and cuts out the remaining segments.</p> <p><b>JS-command:</b></p> <pre>&lt;Curve&gt; = CreateCircularFilletBetweenLinearSegments (Curve1, Point1, Curve2, Point2, Radius);</pre>
 Ellipse from Center and Axes Ends	<p><b>Explanation:</b> Creates an ellipse from the center and end points on the major/minor axes.</p> <p><b>Graphically:</b> Select the center of the ellipse. Then, select the end of the major axis and the end point of the minor axis.</p> <p><b>JS-command:</b></p> <pre>&lt;Curve&gt; = CreateEllipseFromCenterAndAxesEnds (CenterPoint, MajorAxisEndPoint, MinorAxisEndPoint);</pre>
 Elliptic Arc from Center and Two Points	<p><b>Explanation:</b> Creates an Elliptic Arc from Center and Two Points</p> <p><b>Graphically:</b> Select first the center and then two end points to create an elliptic arc.</p> <p><b>JS-command:</b></p> <pre>&lt;Curve&gt; = CreateEllipticArcFromCenterAndEndPoints (CenterPoint, Point1, Point2, LeastArc = true);</pre>

### 3.3.5 Guiding free-form curves (splines)

The menu items for the free-form curves can be split into two parts: the creation/insertion functions and the operations on the curves. Note that all free-form curves are represented in the NURBS form, so we use the term guiding-spline.

You can insert a guiding spline the following ways:



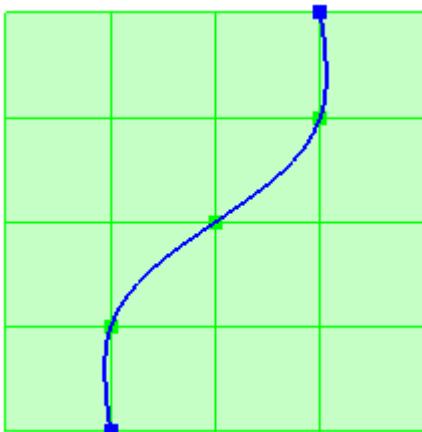
**Explanation:** The function creates a C<sub>2</sub> cubic spline, which interpolates a list of ordered points. The spline may satisfy tangent-vector, natural or periodic boundary conditions. The data parameterization may be chord-length, equidistant or centripetal.

**Graphically:** Select a list of points.

**JS-command:**

```
<Curve> = CreateCubicSpline(Array(Points), IsPeriodic,  
TangentVector1 = Vector3d(0 m,0 m,0 m),  
TangentVector2 = Vector3d(0 m,0 m,0 m),  
Parameterization = sptChordLength);
```

The zero-length tangent vectors at the end points tell the function to employ natural boundary conditions. The IsPeriodic variable tells the function to employ periodic boundary conditions.



An example of a cubic spline, which interpolates the points shown as green dots.

### Hermite Cubic Spline

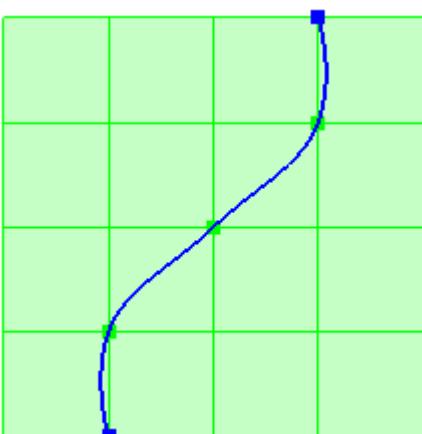
**Explanation:** The function creates a C1 Hermite cubic spline, which interpolates a list of ordered points. The spline may satisfy tangent-vector or periodic boundary conditions. The data parameterization may be chord-length, equidistant or centripetal. The tangent vectors at the intermediate points is computed automatically.

**Graphically:** Select a list of points.

**JS-command:**

```
<Curve> = CreateHermiteCubicSpline(Array(Points),
IsPeriodic,
TangentVector1 = Vector3d(0 m,0 m,0 m),
TangentVector2 = Vector3d(0 m,0 m,0 m),
Parameterization = sptChordLength);
```

The zero-length tangent vectors at the end points tell the function to compute an estimation of them. The IsPeriodic variable tells the function to employ periodic boundary conditions.



An example of a Hermite cubic spline, which interpolates the points shown as green dots.

Note that the tangent vectors at the intermediate points is calculated based on the adjacent points. (For example, note that the tangent vector at the middle point is collinear with its neighboring points).

## Shape Preserving Cubic Spline

**Explanation:** The function creates the C1 shape-preserving Hermite cubic spline, which interpolates a list of ordered points. The spline may satisfy tangent-vector or periodic boundary conditions. The data parameterization may be chord-length, equidistant or centripetal. The tangent vectors at the intermediate points is computed automatically.

With the term “shape-preserving” we simply mean a curve which “*follows the shape of polygonal-arc defined by the data points*”, in the sense that if they form a convex set, then the curve is also convex and if they are co-planar or co-linear, the curve is also planar or the exact straight line interpolating them, respectively.

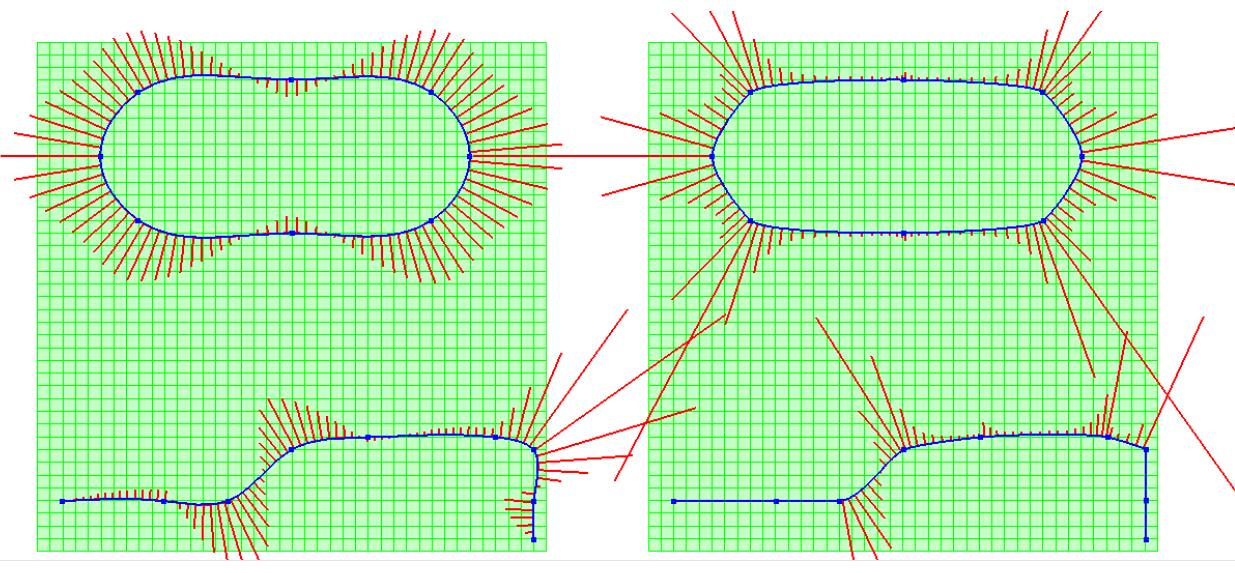
**Graphically:** Select a list of points.

**JS-command:**

```
<Curve> =
CreateCubicShapePreservingSpline(Array(Points),
IsPeriodic,
TangentVector1 = Vector3d(0 m,0 m,0 m),
TangentVector2 = Vector3d(0 m,0 m,0 m),
Parameterization = sptChordLength);
```

The zero-length tangent vectors at the end points tell the function to compute an estimation of them. The IsPeriodic variable tells the function to employ periodic boundary conditions.

The following illustrations show –in blue color- the curves interpolating a closed (periodic) set of data points and an open a set of data points. The figure on the left shows the C2 cubic spline and the figure on the right the C1 cubic Hermite shape-preserving spline. The vector-distributions -in red- show the curvature plots on the curves. The closed data set is convex and the open one is linear near the beginning and near the end, and changes the convexity in one segment. The examples show how the shape-preserving spline achieves to preserve the convexity of the data-sets, while the cubic spline fails in these cases.



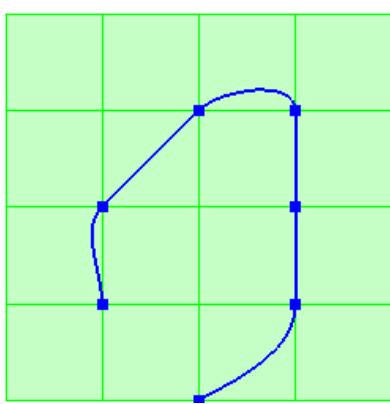
## Poly-Curve

**Explanation:** A poly-curve is a C1 continuous curve, which consists of alternate curved (C2 cubic spline segments) and straight segments.

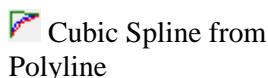
**Graphically:** Select a list of points. The function creates the C2 cubic spline but it has the ability to set explicitly one of its pieces to be a straight-line. By default, the GUI selects one segment and sets it as linear. Alternatively, this can be done by selecting the curve and “edit” it through the menu on the RMB.

**JS-command:**

```
<Curve> = PolyCurve(Array(Points));  
<Curve>.modifyCurveType(i-segment, ggStraight);  
<Curve>.rebuild();
```



To the left is an illustration of a Poly-Curve. Some of the segments are straight lines.



### Cubic Spline from Polyline

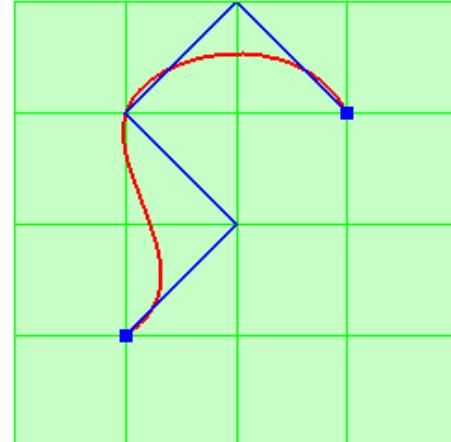
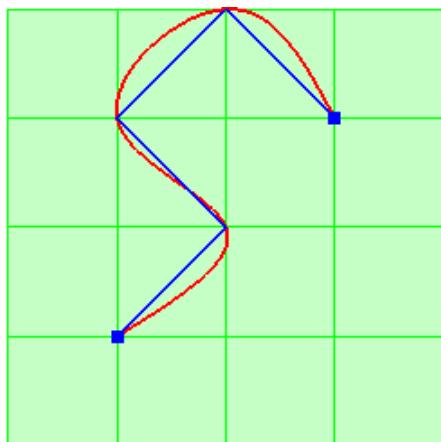
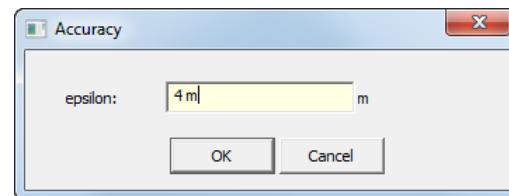
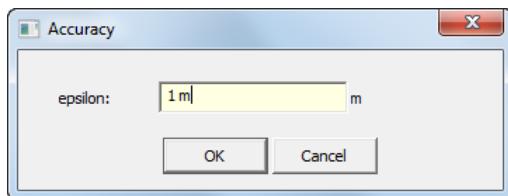
**Explanation:** This function creates a cubic spline approximation of a polyline. When clicking on the polyline the Accuracy dialog shows up. Here you can set the accuracy of the cubic spline.

**Graphically:** Select a poly-line. The function creates a C2 cubic spline, which approximates the selected poly-line.

#### JS-command:

```
<Curve> =  
CreateCubicSplineFromPolyline (Curve, TangentVector1  
= Vector3d(0 m,0 m,0 m), TangentVector2 = Vector3d(0  
m,0 m,0 m) , Tolerance = 1 m);
```

The tolerance is a radius around each of the given vertices of the polygon restricting the curve to lie within these circles. A zero tolerance makes the curve to be the interpolating cubic spline. The zero-length tangent vectors at the end points tell the function to employ natural boundary conditions.



The higher the tolerance is, the less the curve touches the polyline vertices.



## B-Spline from Control Points

**Explanation:** The function creates a cubic B-Spline with uniform knot-vector taking the selected points to be the control points of the curve.

**Graphically:** Select a list of points.

**JS-command:**

```
<Curve> =  
CreateBSplineUniformKnotVector(Array(Points),  
Degree = 3);
```

In case one needs a spline of different degree, he has to change manually the default value. The knot vector used here is the uniform knot vector. For the most general case of creating a B-Spline curve one can use the function:

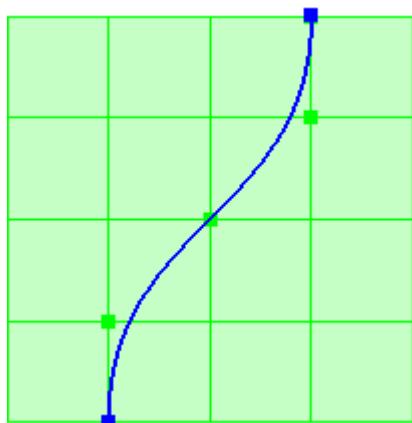
**JS-command:**

```
<Curve> =  
CreateBSpline(Array(Points), Array(doubles), Degree);
```

In case one needs to create a general NURBS curve he should use the function:

**JS-command:**

```
<Curve> = CreateNURBS(Array(Points), Array(doubles),  
Array(doubles), Degree);
```



To the left is an illustration of a B-Spline from Control Points. The control points used to create the B-Spline are (in addition to the end points) shown as green dots.



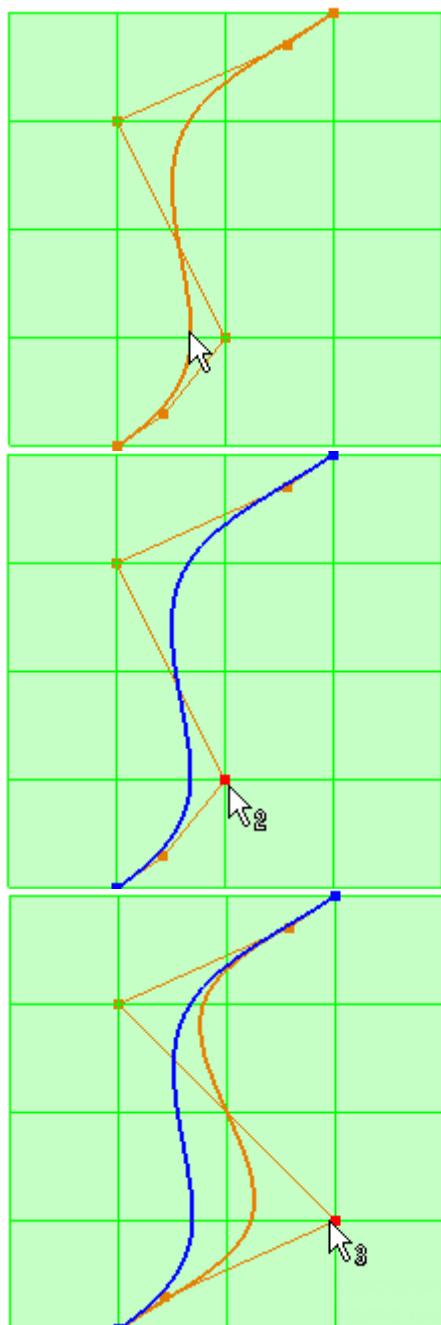
## Modify Control Polygon

**Explanation:** The function lets you modify the control polygon for a curve and thereby modify the shape of the curve.

**Graphically:** Select a curve. The control polygon of the curve appears on the screen. Select one control point of the curve. Click another point on the screen and the selected control point changes its position and the shape of the underlying curve accordingly.

### JS-command:

```
Curve.SetControlPoint(control-point index, new-position);
```



To the left is an illustration of how to modify the control polygon of a curve.

1. Click Modify Control Polygon, and select the curve. The control polygon for the curve shows up.

2. Click on the control point that you want to move

3. Click where you want to move the control point. The shape of the curve will change to fit the modified control polygon.



## Interpolating Polynomial

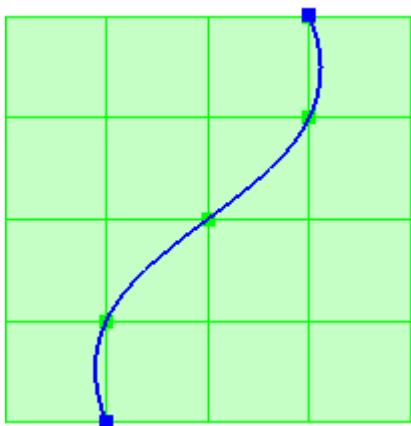
**Explanation:** The function creates the polynomial which interpolates the list of the selected points.

**Graphically:** Select a list of points.

**JS-command:**

```
<Curve> =  
CreateInterpolatingPolynomial(Array(Points),  
Parameterization = sptChordLength);
```

The degree of the polynomial is equal to the number of given points minus one. The user should be aware of the fact that high degrees do not result in visually pleasing solutions. The method should be used for relatively small number of data points (less than ten).



Interpolating polynomial. The points given to create the interpolating polynomial are (in addition to the end points) shown as green dots.

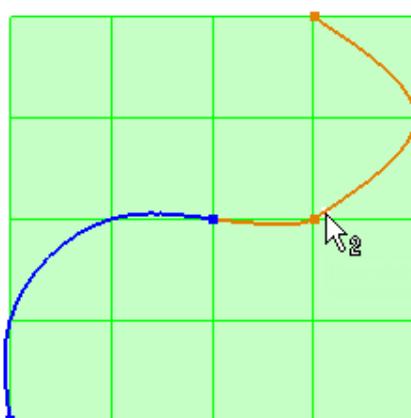


## Polynomial Bridge between two Curves

**Explanation:** Creates a polynomial curve which smoothly connects two disconnected curves.

**Graphically:** Select a point near the end point of each of the two curves to be connected.

**JS-command:** <Curve> =  
CreateBezierJoiningTwoCurves(Curve1, PointOnCurve1,  
Curve2, PointOnCurve2, Parameterization =  
sptChordLength);



The illustration to the left shows two disconnected curves being connected by using a polynomial bridge between two curves.

Note that the two curves can be connected in several different ways depending on which end of the curves you are selecting.

## Polynomial from Polyline

**Explanation:** The function will create a polynomial approximating a polyline. The tangent vectors at the ends may be given explicitly.

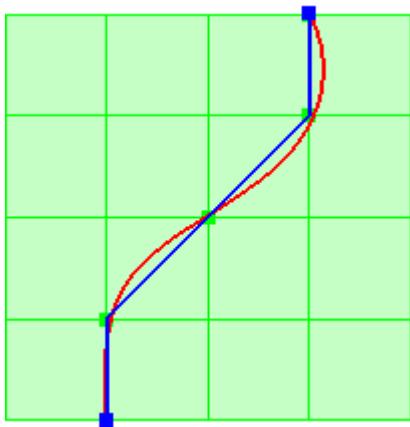
**Graphically:** Select a poly-line. Then set the degree of the polynomial.

**JS-command:**

```
<Curve> =  
CreatePolynomialFromPolyline(Curve, Degree,  
TangentVector1 = Vector3d(0 m, 0 m, 0 m),  
TangentVector2 = Vector3d(0 m, 0 m, 0 m));
```

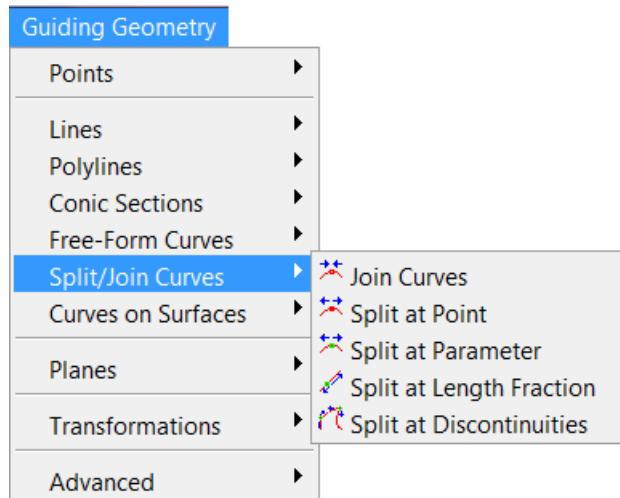
The zero-length tangent vectors at the end points tell the function to compute an approximation of them.

A polynomial (shown in red) is created from a polyline (shown in blue).



### 3.3.6 Split/Join Curves

You can manipulate one or more guiding splines the following ways:



**Explanation:** Function will try to join two or more curves in the order they are given at their common end-points.

**Graphically:** Select a list of curves.

**JS-command:**

```
<Curve1> = Join(CurveList[0], CurveList[1]);  
<Curve2> = Join(<Curve1>, CurveList[2]);  
...  
<CurveN> = Join(<CurveN-1>, CurveList[N]);
```

It is preassumed that any two consecutive curves have a common endpoint. The shape of the curves does not change after the operation. The original curves are deleted.



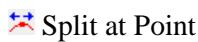
## Join Multiple Curves

**Explanation:** Function will try to join two or more curves into one or several curves.

**Graphically:** Select a list of curves.

**JS-command:**

```
tmpArrayOfCurves =  
JoinMultipleCurves(Array(Curve1,Curve4,Curve2,Curve3,  
...), false);  
  
Rename(tmpArrayOfCurves[0], "Curve5");  
Rename(tmpArrayOfCurves[1], "Curve6");  
...
```



**Explanation:** Splits a curve at a snap point on the curve.

**Graphically:** Select a snap point on a curve.

**JS-command:**

```
<Curve> = Curve.SubdivisionNearPoint(Point);
```

The shape of the curves does not change after the operation. The original curve is cut.



**Explanation:** Splits a curve at the parameter you select.

**Graphically:** Select a curve and a local parametric value (i.e. in [0,1]).

**JS-command:**

```
<Curve> = Curve.SubdivisionAtParametricValue(double);
```

The function splits it at this value. The shape of the curves does not change after the operation. The original curve is cut.



**Explanation:** The function splits the curve at the point where the length of the curve from the starting point of the curve over the total length of it equals this value. The shape of the curves does not change after the operation. The original curve is cut.

**Graphically:** Select a curve and a length fraction (i.e. a value in [0,1]).

**JS-command:**

```
<Curve> = Curve.SubdivisionAtLengthFraction(double);
```



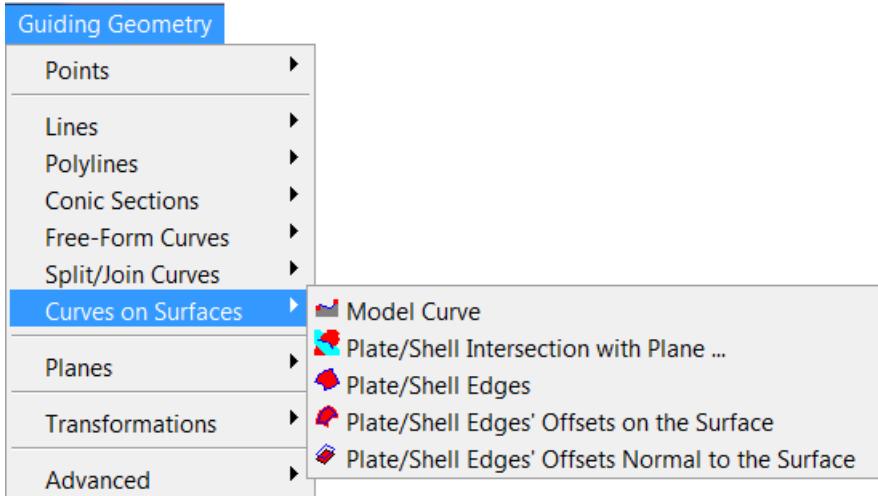
**Explanation:** The function splits the curve at all points where the multiplicity of the knot-vector equals the degree of the spline. The original curve remains the same.

**Graphically:** Select the curve to be split.

**JS-command:**

```
tmpArrayOfCurves = Curve.SplitAtDiscontinuities();  
Rename(tmpArrayOfCurves[i], "<Curve>");
```

### 3.3.7 Guiding curves on surfaces



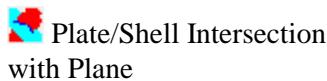
Model Curve

**Explanation:** The function creates a curve on the model between the two endpoints that you have selected.

**Graphically:** Select two points on the model.

**JS-command:**

```
<Curve> = CreateModelCurveFromTwoPoints(Point1,  
Point2);
```



Plate/Shell Intersection  
with Plane

**Explanation:** The function returns the edges of the intersection between the selected plates/shells and the plane as an array of guiding curves.

**Graphically:** Select any number of plates/shells. Then, define the plane through the floating dialog.

**JS-command:**

```
tmpArrayOfEdges = IntersectSurfacesWithPlane(Array of  
Plates/Shells, Plane3d);  
Rename(tmpArrayOfEdges[i], "<Curve>");
```



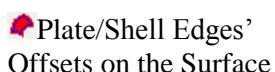
Plate/Shell Edges

**Explanation:** The function returns the edges of the selected plate/shell as an array of guiding curves.

**Graphically:** Select a plate plate/shell.

**JS-command:**

```
tmpArrayOfEdges = CreatePlateShellEdges(Plate/Shell);  
Rename(tmpArrayOfEdges[i], "<Curve>");
```



Plate/Shell Edges'  
Offsets on the Surface

**Explanation:** The function returns the offsets at a given distance of the edge-loops of the selected plate/shell as an array of guiding curves.

**Graphically:** Select a plate plate/shell. Then set the offset distance in the next dialog. Note that if the distance is negative, then the boundaries are offset towards inside the shell. Positive distance offsets the boundaries outwards, i.e. it is only recommended when we want to offset an inner loop.



## Sesam User Manual | GeniE V7.3 | www.dnvgi.com/software

## 3.3.7 Plate/Shell Edges' Offsets

**JS-command:**

```
tmpArrayOfEdges =  
CreatePlateShellOffsetEdges(Plate/Shell);  
Rename(tmpArrayOfEdges[i], "<Curve>");
```

**Explanation:** The function offsets at a given distance the edge-loops of the selected plate/shell in the direction of the normal vector of the surface as an array of guiding curves.

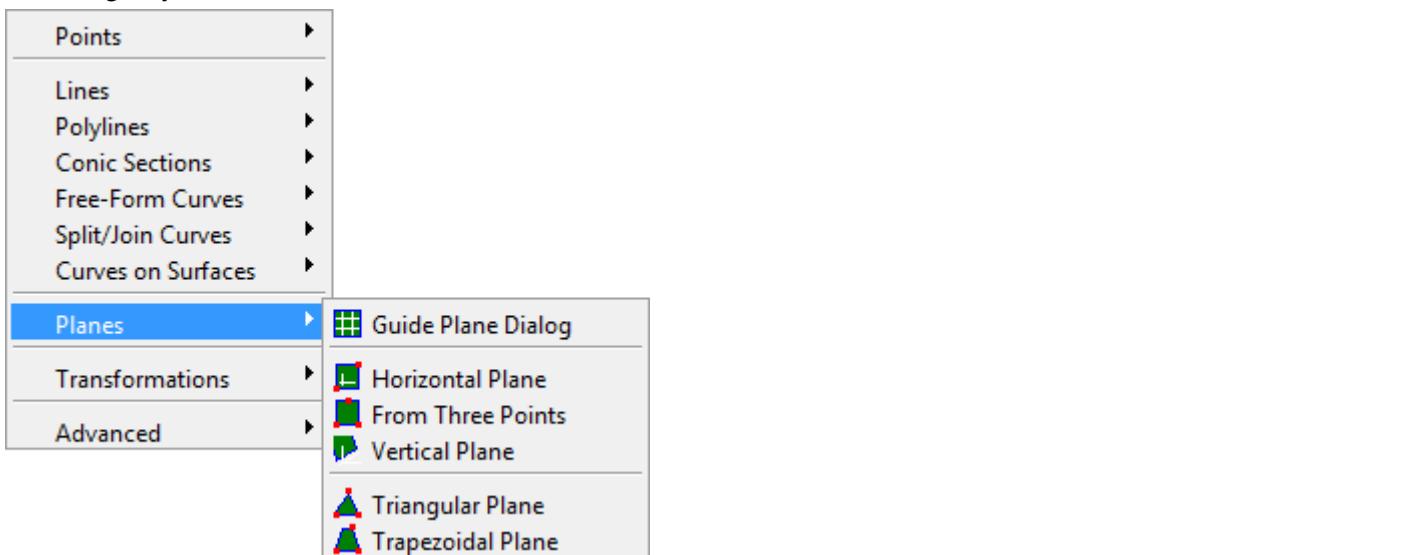
**Graphically:** Select a plate plate/shell. Then set the offset distance in the next dialog.

**JS-command:**

```
tmpArrayOfEdges =  
CreatePlateShellOffsetEdgesNormalToSurface(Plate/Shel  
l);  
Rename(tmpArrayOfEdges[i], "<Curve>");
```

### 3.3.8 Guiding planes

The reason for inserting guiding planes is to create a snapping grid. You can insert a guiding plane the following ways:



**Guide Plane Dialog**

**Explanation:** Opens the Guide Plane dialog.

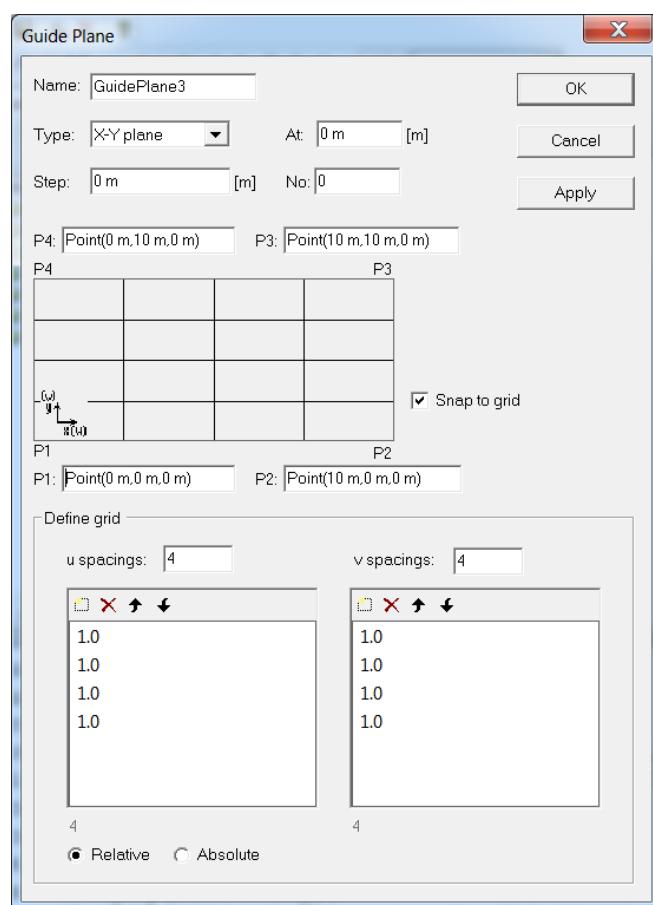
**Graphically:** Opens the dialog shown below, which helps determining the parameters of a guiding-plane. You have the opportunity to change these parameters for any guiding plane by choosing "Properties" on the RMB, after selecting it.

### Guide Plane Dialog

You specify the corner coordinate values (given according to the global axis system) and number of grid-lines in local x and y direction.

The spacings between gridlines are relative, meaning that the program will automatically calculate the positions. You may also specify exact spacings if you want.

A guide-plane may be oriented in any plane and it may very well have a trapezoidal form. It is required that it consists of four corner points.



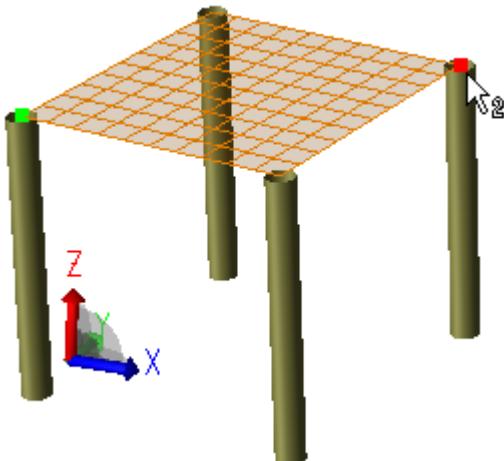
## Horizontal Plane

**Explanation:** Function creates a horizontal guide plane from two points.

**Graphically:** Select two points having different x and y coordinates. The function creates the horizontal rectangular plane z equal to the z-coordinate of its bottom-left point.

### **JS-command:**

```
<GuidePlane> = GuidePlane(BottomLeftPoint,  
TopRightPoint, nx=4, ny=4);
```



A horizontal guide plane is created between two points, shown as a green and a red dot in the illustration.

If the second point you are clicking is not at the same z-level as the first, the second point will snap to the x and z coordinates of the point you are clicking, while the z coordinate will always be the same as for the first point.

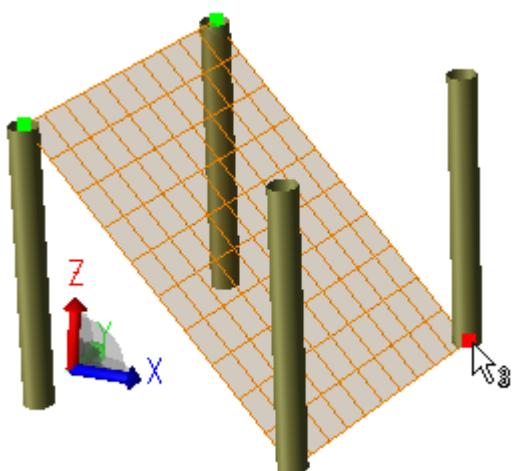
## From Three Points

**Explanation:** The function creates a rectangular guide plane based on the three points chosen.

**Graphically:** Select two points to form a straight edge. Then select a third point to determine the height of the rectangular plane.

### **JS-command:**

```
<GuidePlane> = GuidePlane(tpRectangular, EdgePoint,  
EdgePoint, Point, n=4, m=4);
```





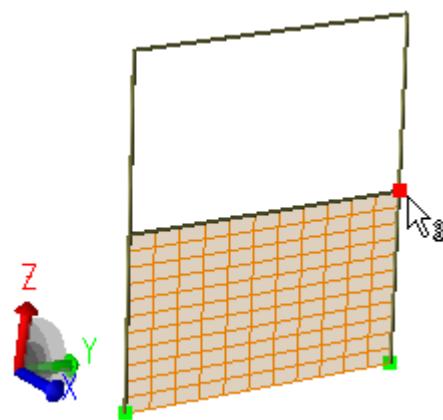
## Vertical Plane

**Explanation:** The function creates a vertical plane based on three points chosen by the user. The two first points form the “baseline” edge while the third point gives the height of the plane. If the two first points are not on the same z-level, the plane will not be rectangular, but it will still be vertical, see illustrations.

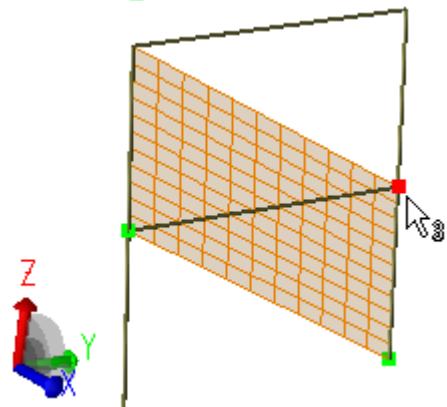
**Graphically:** Select two points to form a straight edge. Then select a third point to determine the height of the tetrahedral –but not rectangular necessarily– plane with a normal vector of type (nx,ny,0).

### JS-command:

```
<GuidePlane> = GuidePlane(tppVertical, EdgePoint,  
EdgePoint, HeightPoint, n=4, m=4);
```



The two first points clicked are marked as a green dot in the two illustrations to the left.



If the two first points are not on the same z-level, you'll get a vertical guide plane that is skewed like in the illustration to the left.



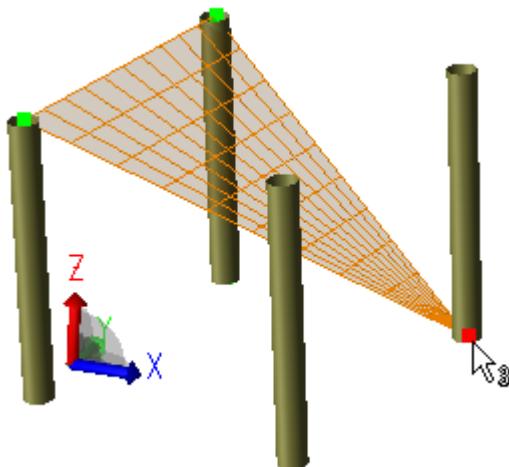
**Explanation:** The function creates a triangular guide plane based on three points.

**Graphically:** Select two points to form a straight edge. Then select a third point to determine the third vertex of the triangular plane.

**JS-command:**

```
<GuidePlane> = GuidePlane(tppTriangular, EdgePoint,  
EdgePoint, Point, n=4, m=4);
```

Example of a triangular guide plane.



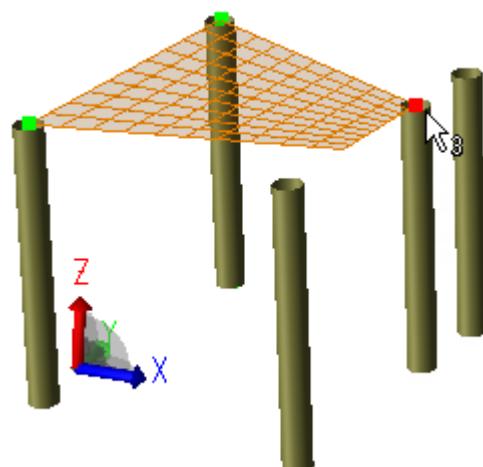
**Explanation:** The function creates a trapezoidal guide plane based on three points.

**Graphically:** Select two points to form a straight edge. Then select a third point to determine the height and the width of the opposite edge of the symmetric trapezoidal plane.

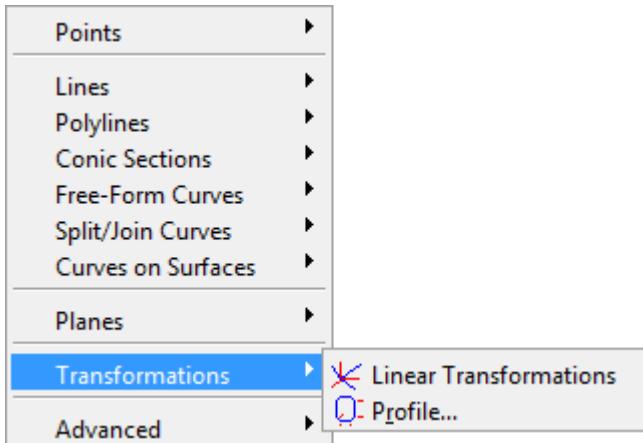
**JS-command:**

```
<GuidePlane> = GuidePlane(tppTrapezoidal, EdgePoint,  
EdgePoint, Point, n=4, m=4);
```

Example of a trapezoidal guide plane.



### 3.3.9 Transformations



✖ Linear Transformations

**Graphically:** Opens the window shown below. The function creates a linear transformation (translation, rotation, scaling, mirroring or any combination of them).

**JS-commands:**

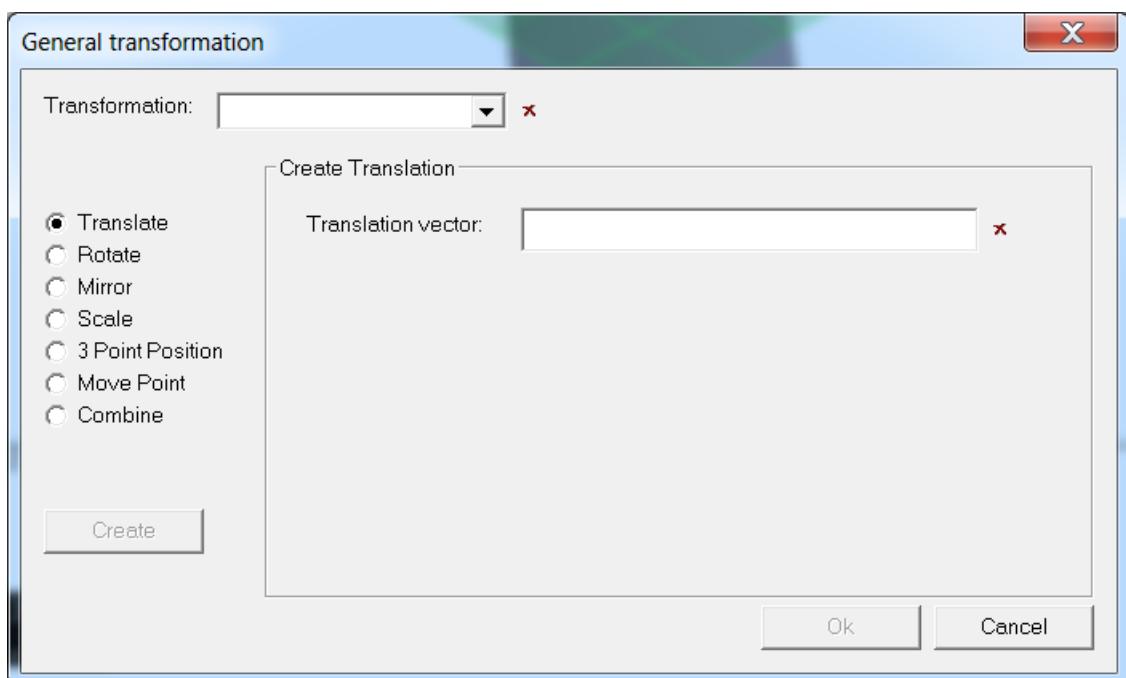
```
<translation> = Translate(Vector3d);  
<rotation> = Rotate(Point,Vector3d,angle);  
<mirroring> = Mirror(Point,Vector3d);  
<scaling> = ScaleC(factor,Point);  
<transformation>.multiply(other transformation);
```

◻ Profile

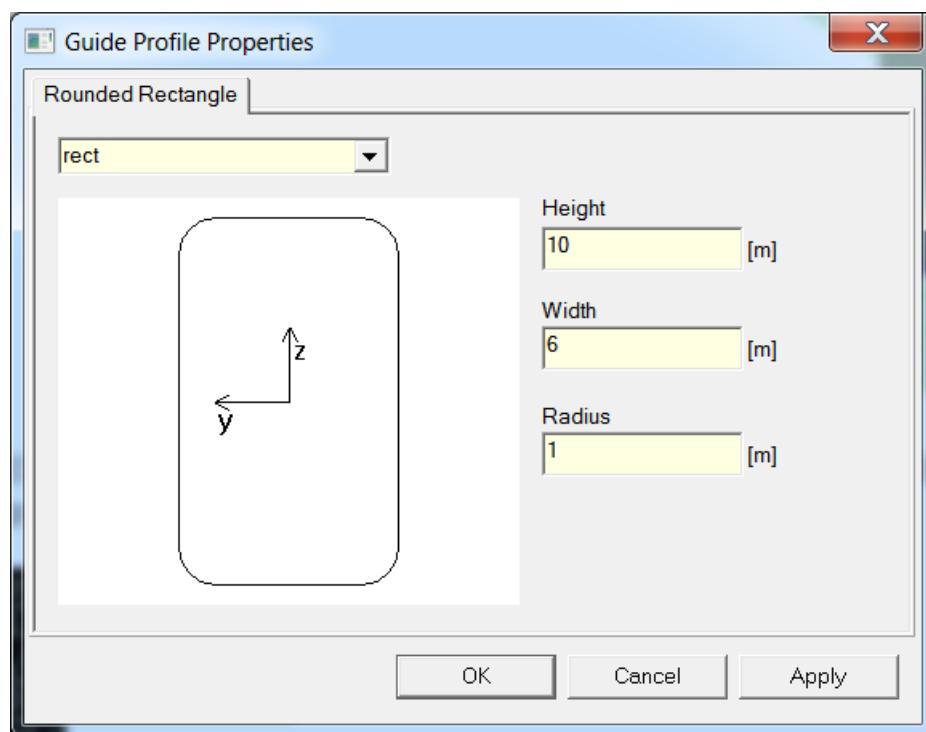
**Graphically:** Opens the window shown below. The function creates a rounded rectangle, which can be used to cut or punch (trim) plates.

**JS-command:**

```
<RoundRectangle> = ProfileRR(Height,Width,Radius);
```

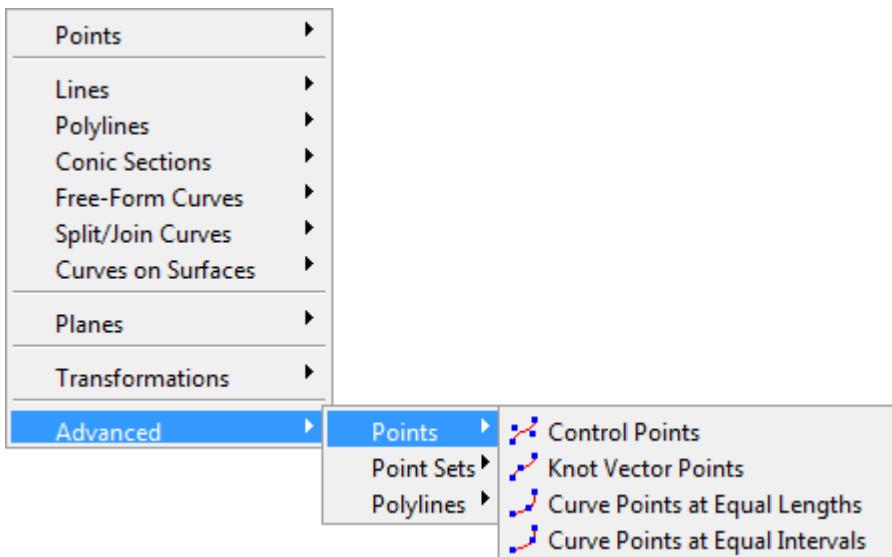


**Linear Transformations Dialog**



**Profiles Dialog**

### 3.3.10 Advanced



**Explanation:** The function computes the control points of a curve.

**Graphically:** Select a NURBS curve.

**JS-command:**

```
tmpArrayOfPoints = Curve.GetControlPointsArray();  
Rename (tmpArrayOfPoints[i], "<Point>");
```



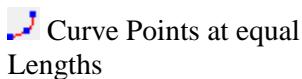
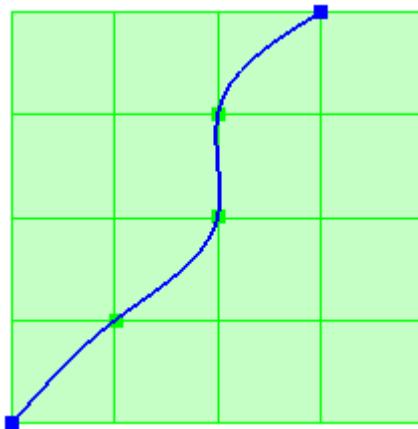


**Explanation:** The function computes the curve points which correspond to the parametric values of its knot vector.

**Graphically:** Select a NURBS curve.

**JS-command:**

```
tmpArrayOfPoints = Curve.GetNodalPointsArray();  
Rename(tmpArrayOfPoints[i], "<Point>");
```

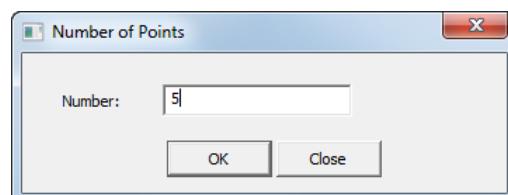
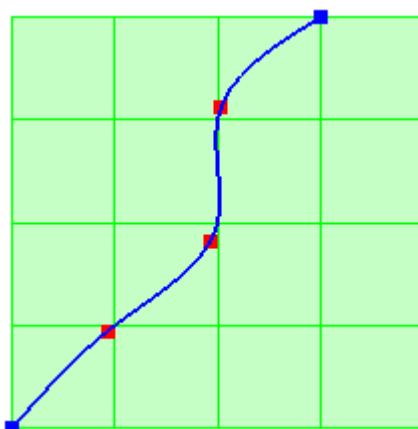


**Explanation:** The function creates the given number of points distributed evenly along the curve's length. The number of points includes the endpoints.

**Graphically:** Select a NURBS curve. Then, give the number of points.

**JS-command:**

```
tmpArrayOfPoints =  
Curve.GetPointsOnEquidistantLengths(N=5);  
Rename(tmpArrayOfPoints[i], "<Point>");
```



An example of curve points at equal lengths.



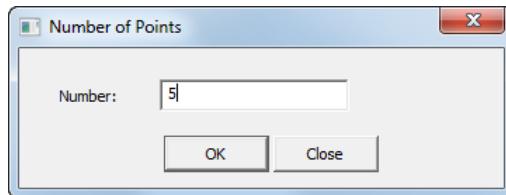
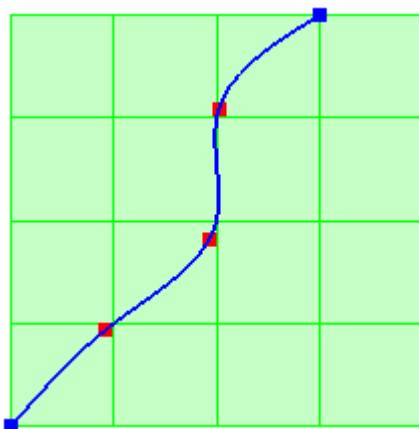
## Curve Points at Equal Intervals

**Explanation:** The function creates the given number of points distributed evenly along the curve's parametric interval. The number of points includes the endpoints.

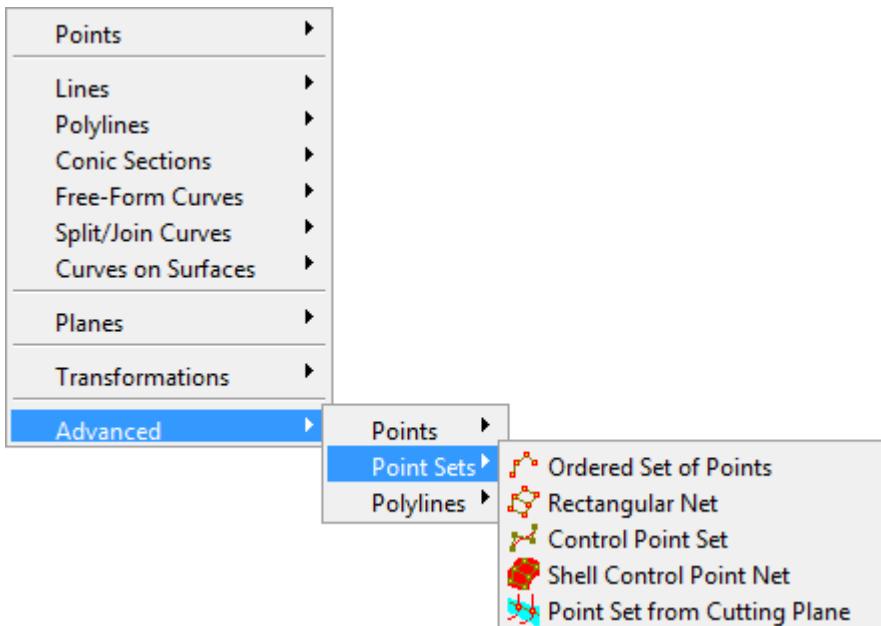
**Graphically:** Select a NURBS curve. Then, give the number of points.

**JS-command:**

```
tmpArrayOfPoints =  
Curve.GetPointsOnEqualIntervals(N=5);  
  
Rename(tmpArrayOfPoints[i], "<Point>");
```



An example of curve points at equal intervals.

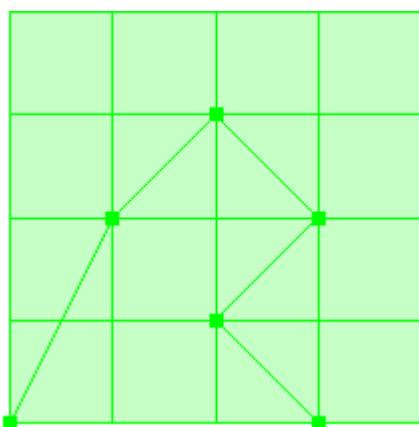


**Explanation:** The function creates an ordered set of points (represented graphically as a polygon) from the selected points. It is displayed with lines connecting the points. The points are ordered. This implies that the order you create the points will be the order of the points in the set.

**Graphically:** Select a list of points.

**JS-command:**

```
<PointSet> = CreatePointSet(Array(Points));
```



Example of an ordered sets of points created on a guide plane.

## Rectangular Net

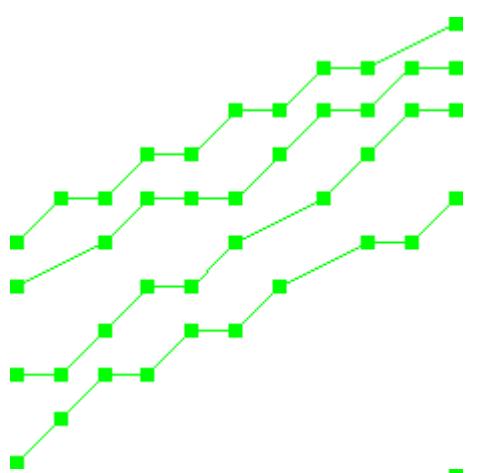
**Explanation:** The function creates a rectangular net i.e. a 3D grid of points (represented graphically as a net of polygons) out of a collection of ordered sets of points. All the point-sets must have the same number of points.

**Graphically:** Select a list of point-sets.

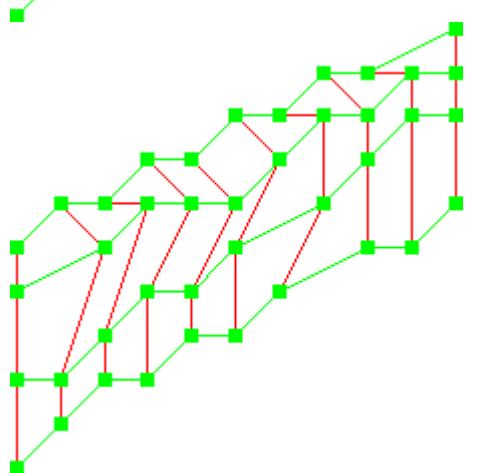
**JS-command:**

```
<PointSet> = CreateRectangularNet(Array(PointSets)) ;
```

Note that the number of points of a point-set and the direction of it cannot be changed.



The illustration to the left is showing four ordered sets of points. All the ordered sets of points have been created from the left to the right, and they all have the same number of points. The leftmost point is the first, and the rightmost point is the last point created.



The four ordered sets of points are used to create a rectangular net. The rectangular net is created in such a way that all the first points of the point sets are connected, all the second points of the point sets are connected and so on. It is important that all point sets have the same number of points and that their directions match so you get the net that you want to create.

## Control Point Set

**Explanation:** The function returns the control polygon points of the selected curve as an ordered point-set (represented graphically as a polygon).

**Graphically:** Select a curve.

**JS-command:**

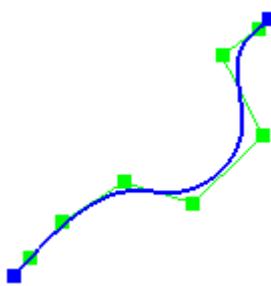
```
<PointSet> =  
CreatePointSet(Curve.GetControlPolygonArray());
```

Note than in order to change the direction of the control polygon, one should change first the direction of the curve, by using the command

```
Curve.Reverse();
```

on the given curve before creating the point-set.

An example of a control point set (shown in green) created from a curve (shown in blue).



## Shell Control Point Net

**Explanation:** The function creates a point-net i.e. a 3D grid of points (represented graphically as a net of polygons) by selecting a point on a surface (plate/shell).

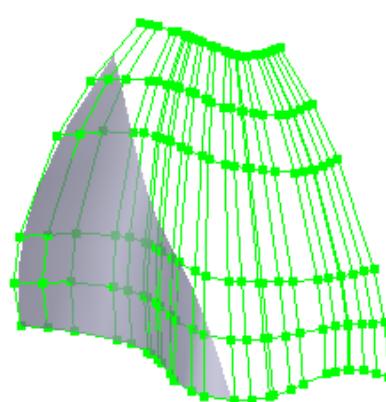
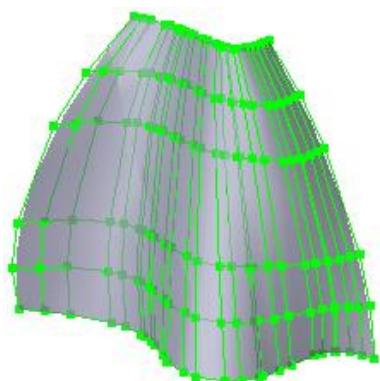
**Graphically:** Click on a surface (plate/shell) point.

**JS-command:**

```
<PointSet> =  
CreateRectangularNetFromShell(Plate/Shell, Point);
```

Note that a plate/shell may consist of more than one surface-patches, and each one of those will return a different point-net. The point is given for distinguishing between different surfaces of the plate/shell.

Example of a shell control point net. Note that the shell control point net does not change after deleting parts of a surface.



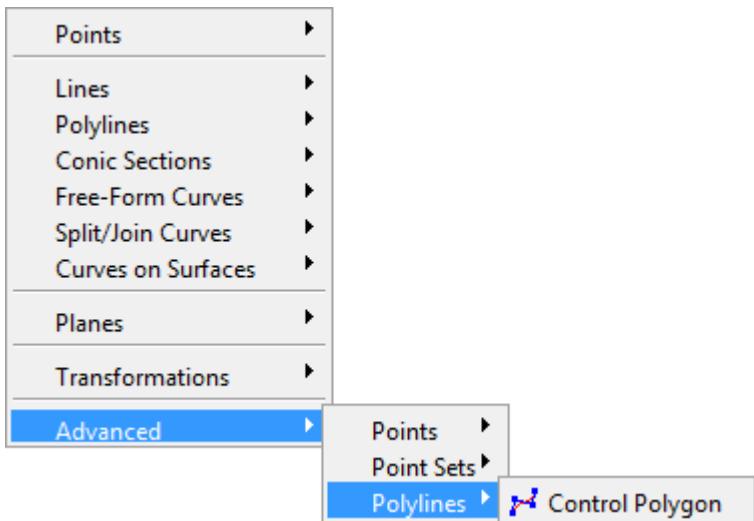
## Point Set from Cutting Plane

**Explanation:** The function creates an ordered point-set (represented graphically as a polygon) by selecting an ordered set of curves and the plane, which cuts them.

**Graphically:** Select a list of curves. Then, select a plane, which cuts them.

**JS-command:**

```
<PointSet> =
IntersectionPointSet(Array(Curves), Plane3d);
```

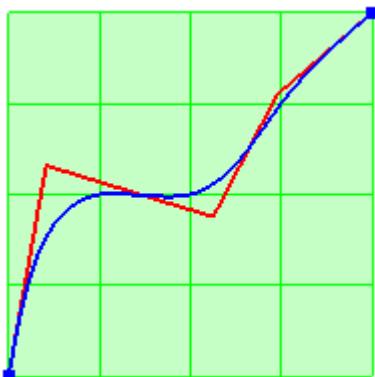


**Explanation:** The function creates the control polygon of the selected curve, as a polyline.

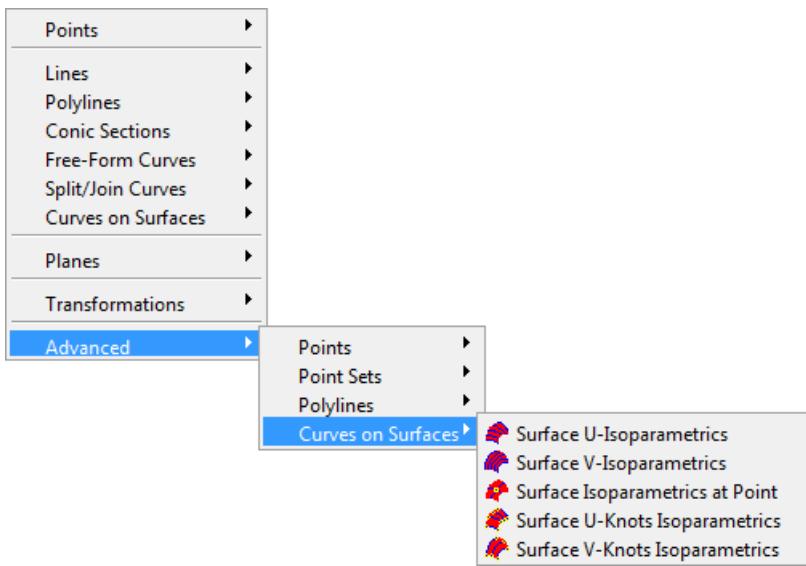
**Graphically:** Select a NURBS Curve.

**JS-command:**

```
<Curve> = Curve0.GetControlPolygon();
```



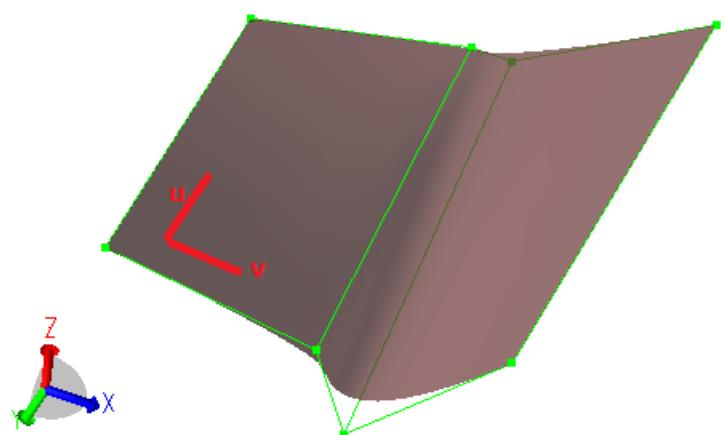
The control polygon, shown in red, are created when you select a curve, shown in blue.



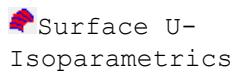
To make the explanation of these commands clearer, let us introduce the following surface patch as a working example.

```
GuidePointSet1 =
CreatePointSet(Array(Point(0,7.5,10),Point(5,6,10),Point(6,6,10),Point(10,2.5,10)));
GuidePointSet2 =
CreatePointSet(Array(Point(0,2.5,0),Point(4,4,0),Point(5,7,0),Point(7.5,2.5,0)));
GuidePointSet3 = CreateRectangularNet(Array(GuidePointSet2,GuidePointSet1));
SRF = CreateBSplineSurface(GuidePointSet3,Array(0,0,1,1),Array(0,0,0,0.2,1,1,1),1,2);
```

The surface SRF is shown in the next figure; note the  $u$  and  $v$  parametric directions drawn on the surface (the surface is linear along the  $u$  direction while it is quadratic in the  $v$  direction). The  $u$  iso-parametric curves of the surface are curves on the  $v$  direction and the  $v$  iso-parametric curves of the surface are curves on the  $u$  direction of it. Now with this convention, we get the results shown in the explanation of the particular commands. Note that regardless of this example one has to keep in mind that the surface of the plate/shell may consist of more than one patch (the actual patch is determined by the selected point). The patches may be trimmed, i.e. the underlying surface may represent a greater area than the one shown on the screen.



The Surface "SRF"

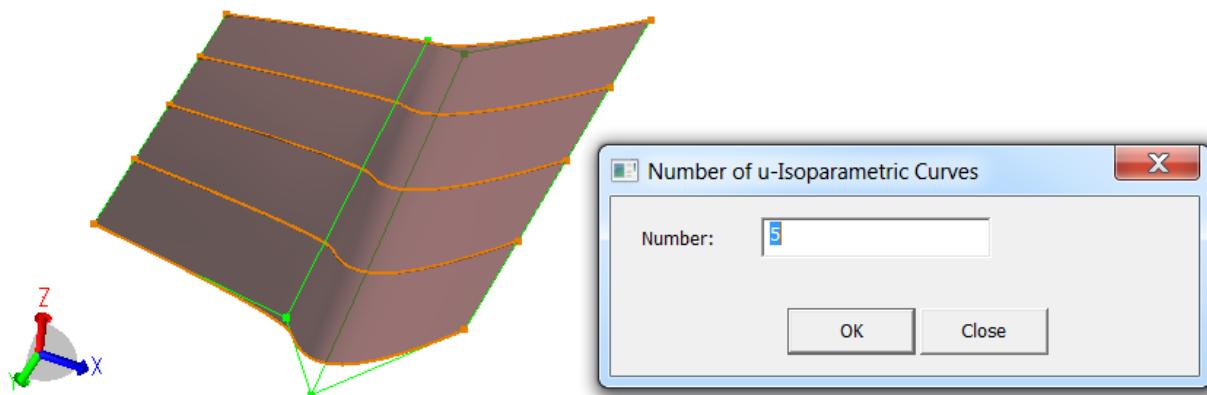


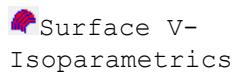
**Explanation:** Creates a user defined number of u iso-parametric curves on the surface; The curves are equally distributed along surface's u-direction.

**Graphically:** Select Surface U-Isoparametrics from the menu. Click on a point on the plate/shell. Select number of curves that you want to create in the dialog.

**JS-command (When creating 5 curves. The point needs to be on the surface of SRF):**

```
tmpArrayOfIsoparametrics=
CreateSurfaceIsoparametricsU(SRF, Point, 5);
Rename (tmpArrayOfIsoparametrics[i],<Curve>");
```



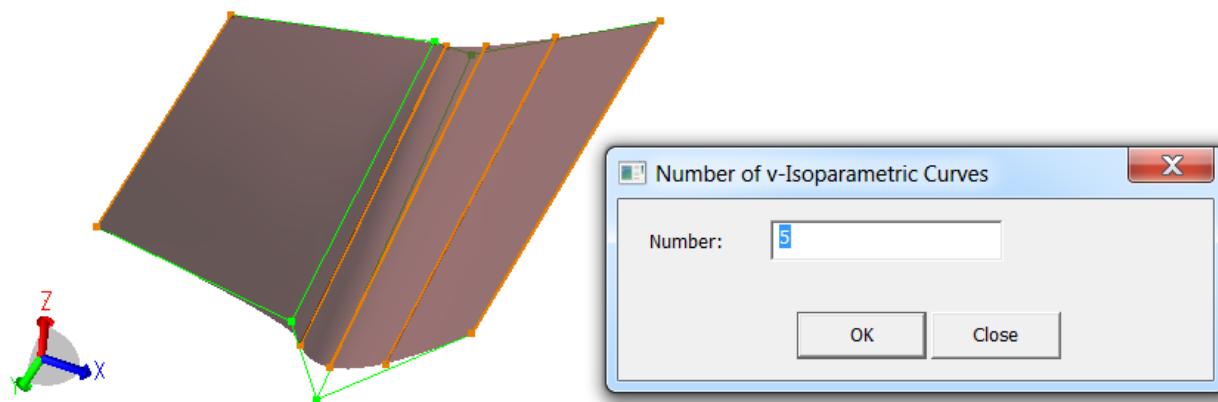


**Explanation:** Creates a user defined number of v iso-parametric curves on the surface; The curves are equally distributed along surface's v-direction.

**Graphically:** Select Surface V-Isoparametrics from the menu. Click on a point on the plate/shell. Select number of curves that you want to create in the dialog.

**JS-command (When creating 5 curves. The point needs to be on the surface of SRF):**

```
tmpArrayOfIsoparametrics=
CreateSurfaceIsoparametricsU(SRF, Point, 5);
Rename (tmpArrayOfIsoparametrics[i],<Curve>);
```



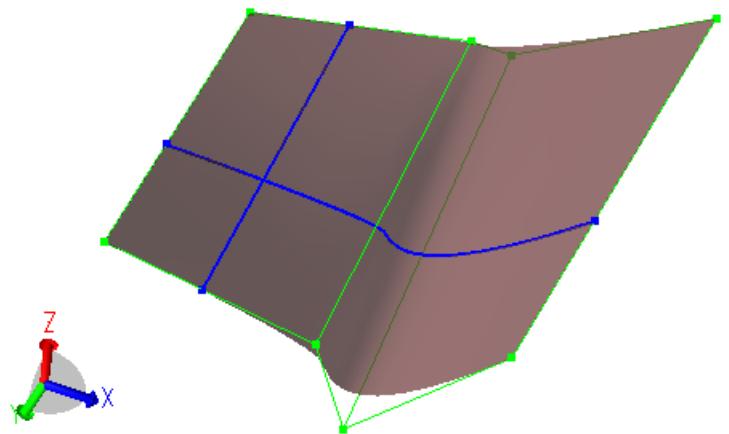
 Surface  
Isoparametrics at  
Point

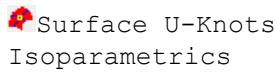
**Explanation:** Creates the two iso-parametric curves which pass through the selected point on a surface. One is in the u-direction and the other is in the v-direction of the surface.

**Graphically:** Select Surface Isoparametrics at Point from the menu. Click on a point on a surface.

**JS-command (The point needs to be on the surface of SRF):**

```
tmpArrayOfIsoparametrics =  
CreateSurfaceIsoparametricsAtPoint(SRF, Point);  
Rename(tmpArrayOfIsoparametrics[0], <Curve>);  
Rename(tmpArrayOfIsoparametrics[1], <Curve>);
```



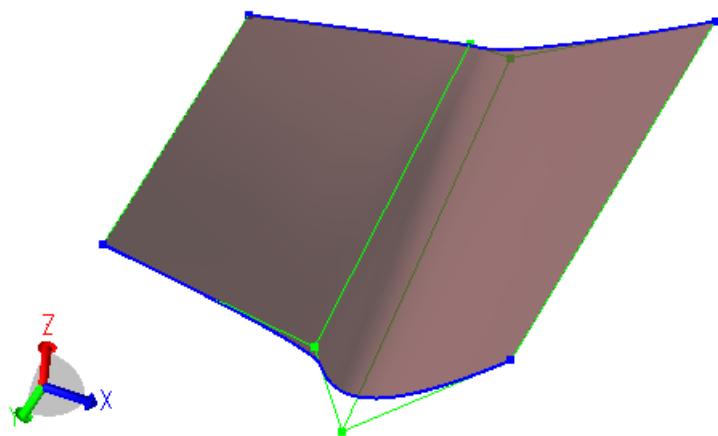


**Explanation:** Creates the iso-parametric curves which correspond to the discrete values of the knot vector along the u-direction.

**Graphically:** Select Surface U-Knots Isoparametrics from menu. Click on a point on the surface.

**JS-command (The point needs to be on the surface of SRF):**

```
tmpArrayOfIsoparametrics =  
CreateSurfaceNodalIsoparametricsU(SRF, Point);  
Rename(tmpArrayOfIsoparametrics[0], "Curve1");  
Rename(tmpArrayOfIsoparametrics[1], "Curve2");
```



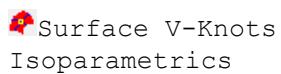
The following command returns the definition parameters of the surface SRF:

```
print(SRF.GeometryToString());
```

The last lines returned are the following:

```
BSplineSurface{Name[SRF_0],Layer[],ControlNet[  
Point(0.0000000000:2.5000000000:0.0000000000)  
Point(4.0000000000:4.0000000000:0.0000000000)  
Point(5.0000000000:7.0000000000:0.0000000000)  
Point(7.5000000000:2.5000000000:0.0000000000) |  
Point(0.0000000000:7.5000000000:10.0000000000)  
Point(5.0000000000:6.0000000000:10.0000000000)  
Point(6.0000000000:6.0000000000:10.0000000000)  
Point(10.0000000000:2.5000000000:10.0000000000)],  
KnotVectorU[ 0.0000000000 0.0000000000 1.0000000000 1.0000000000],  
KnotVectorV[ 0.0000000000 0.0000000000 0.0000000000 0.2000000000  
1.0000000000 1.0000000000 ], DegreeU[1],DegreeV[2],Boundary[4]];
```

Which show that the knot vector in the u-direction has 2 knots, hence the 2 created curves.

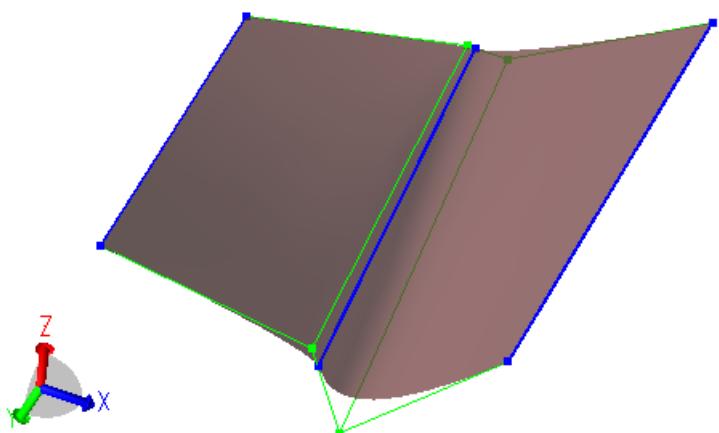


**Explanation:** Creates the iso-parametric curves which correspond to the discrete values of the knot vector along the v-direction.

**Graphically:** Select Surface V-Knots Isoparametrics from menu. Click on a point on the surface.

**JS-command (The point needs to be on the surface of SRF):**

```
tmpArrayOfIsoparametrics =  
CreateSurfaceNodalIsoparametricsV(SRF, Point);  
  
Rename(tmpArrayOfIsoparametrics[0], "Curve1");  
Rename(tmpArrayOfIsoparametrics[1], "Curve2");  
Rename(tmpArrayOfIsoparametrics[2], "Curve3");
```



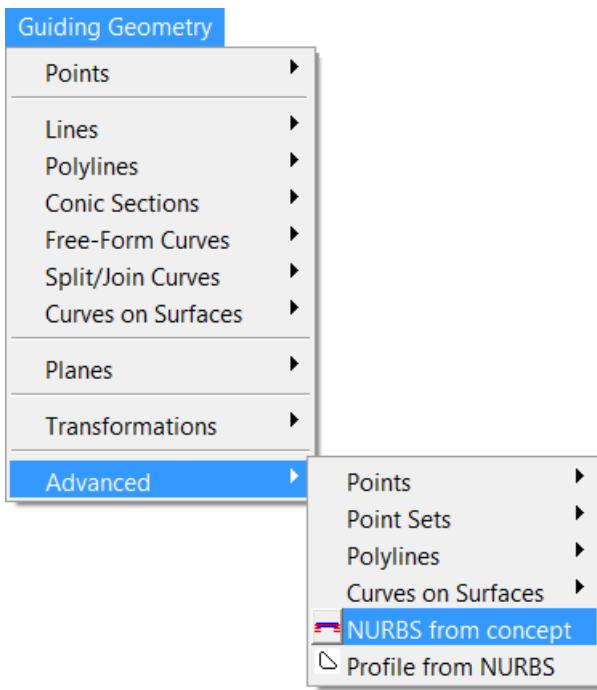
The following command returns the definition parameters of the surface SRF:

```
print(SRF.GeometryToString());
```

The last lines returned are the following:

```
BSplineSurface{Name[SRF_0],Layer[],ControlNet[  
Point(0.0000000000:2.5000000000:0.0000000000)  
Point(4.0000000000:4.0000000000:0.0000000000)  
Point(5.0000000000:7.0000000000:0.0000000000)  
Point(7.5000000000:2.5000000000:0.0000000000) |  
Point(0.0000000000:7.5000000000:10.0000000000)  
Point(5.0000000000:6.0000000000:10.0000000000)  
Point(6.0000000000:6.0000000000:10.0000000000)  
Point(10.0000000000:2.5000000000:10.0000000000)],  
KnotVectorU[ 0.0000000000 0.0000000000 1.0000000000 1.0000000000],  
KnotVectorV[ 0.0000000000 0.0000000000 0.0000000000 0.2000000000  
1.0000000000 1.0000000000, DegreeU[1],DegreeV[2],Boundary[4]]};
```

Which show that the knot vector in the v-direction has 3 knots, hence the 3 created curves.



 NURBS from Concepts

**Explanation:** Creates a guiding curve from a beam, a support curve etc.

**Graphically:** Select the 1D concept you wish to have a copy of its geometry as a guiding curve.

**JS-command:**

```
<curve> = CreateNURBSFromObject( 1D concept );
```

 Profile from NURBS

**Explanation:** Creates a profile curve out of any 2D guiding curve.

**Graphically:** Select the guiding curve you wish to have a copy of its geometry as a profile curve.

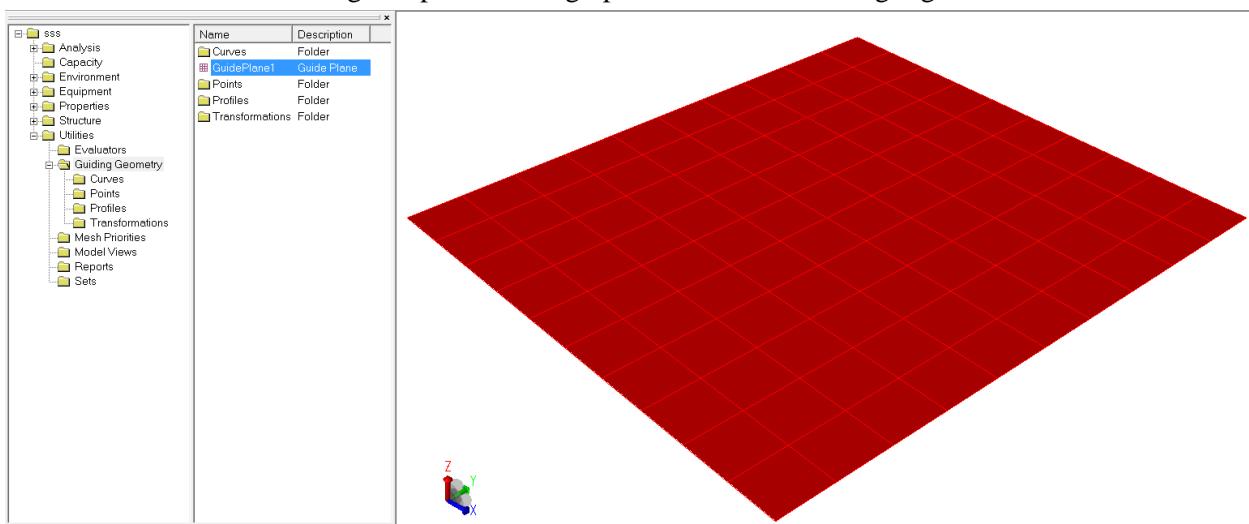
**JS-command:**

```
<profile curve> = ProfileNURBS( Curve );
```

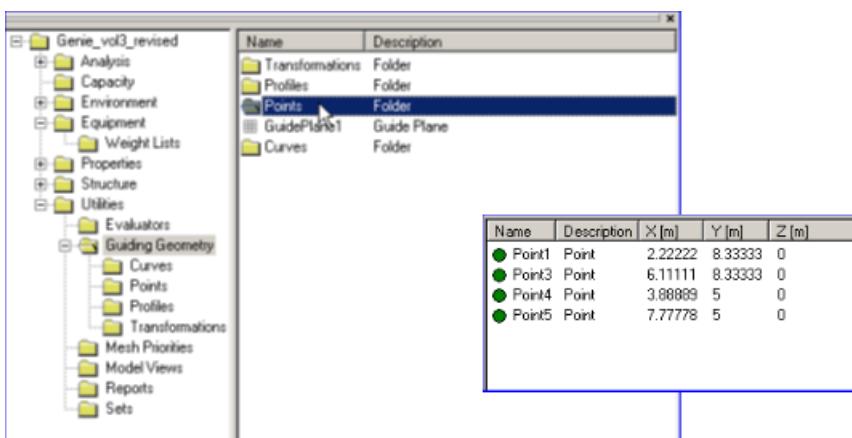
### 3.3.11 Find, select and display guiding geometry

There are two ways of finding and selecting guiding geometry entities (as well as other objects such as beams, plates, equipment, support curves etc).

- When selecting from the graphic window you click on the object you want to select. It is now highlighted and if you have the right browser pane open, the same object is highlighted in the browser.
- It is also possible to select from the graphics screen. Guiding geometry can be found under the browser tab *Utilities* and then *Guiding Geometry*. You can now find all of your guiding geometry whether it is a guide plane, a guide point, a guide curve and so on. If you click on the GuidePlane1 as shown below, the guide plane in the graphic window will be highlighted.



Note that it is also possible to drill down the browser also from the right pane browser window. If -for example- you double click the browser tab *Points* below you will open the content.



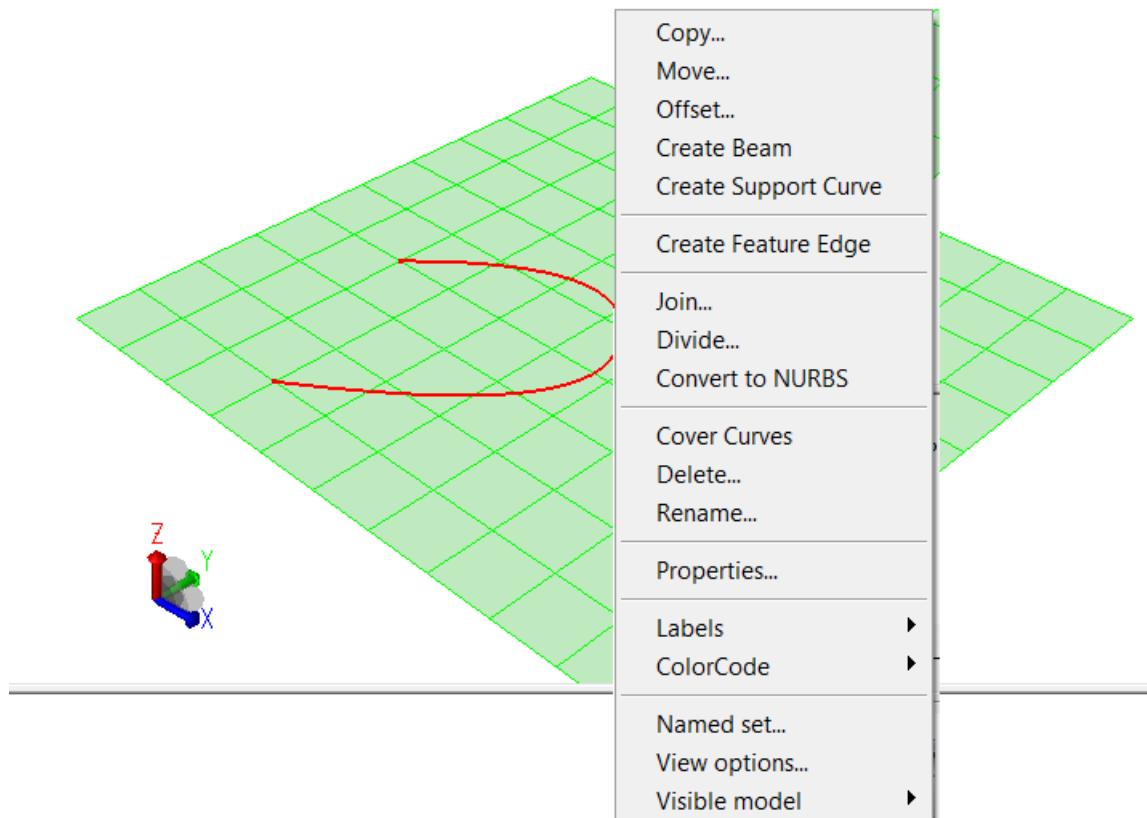
You may also benefit from limiting, or adding, objects to see in the graphics window. There are powerful features for doing so and they are available by selecting an object(s) or named set(s), **RMB** and *Visible Model*. You can also use the short commands, typically **Alt+S** (show selected only), **ALT+Plus** (add selection), **ALT-Minus** (remove selection) and **ALT+A** (show all). Some examples on how to do this is shown on the next pages.

In the example below the graphic window contains a display of a guide plane and it is shown how to add and remove the guide points Point1 -> Point5 in the graphic view. Notice that these operations are not the same as insert or delete.

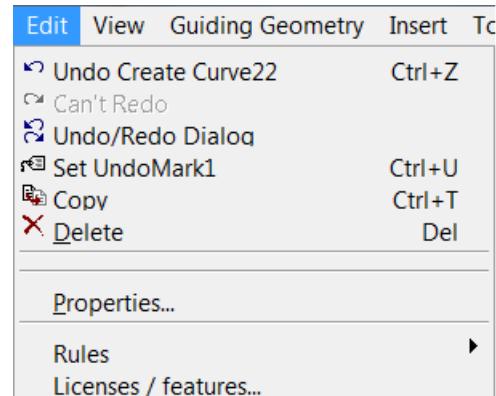
In the example below the graphic window contains a display of a guide plane and it is shown how to add the guide curves Curve3 -> Curve6 in the graphic view. It is also shown how to visualise one of these.

### 3.3.12 Move and Copy guiding geometry entities

All these operations are done by selecting one or more guiding geometry entities from the graphics screen or from the browser, then click the **RMB** and after this select *Copy* or *Move*. *Move* is identical to *Copy* except that no new objects are created; the objects subjected to move are modified.



Alternatively you can select the object and use the pull-down menu as shown to the right. It is also possible to use short commands like **CTRL+T** (for copy). Once you have performed a copy, move or delete operation they are permanent. You may use the undo features (**CTRL+Z** for undo and **CTRL+Y** for redo) to step back or forward. Alternatively you can use the undo/redo dialog and directly go several steps back or forward; the undo/redo history is closed when saving the workspace. In the following there are examples on how to do delete, copy translation, copy rotation, copy mirroring, copy 3 point positioning and copy using a scale factors.

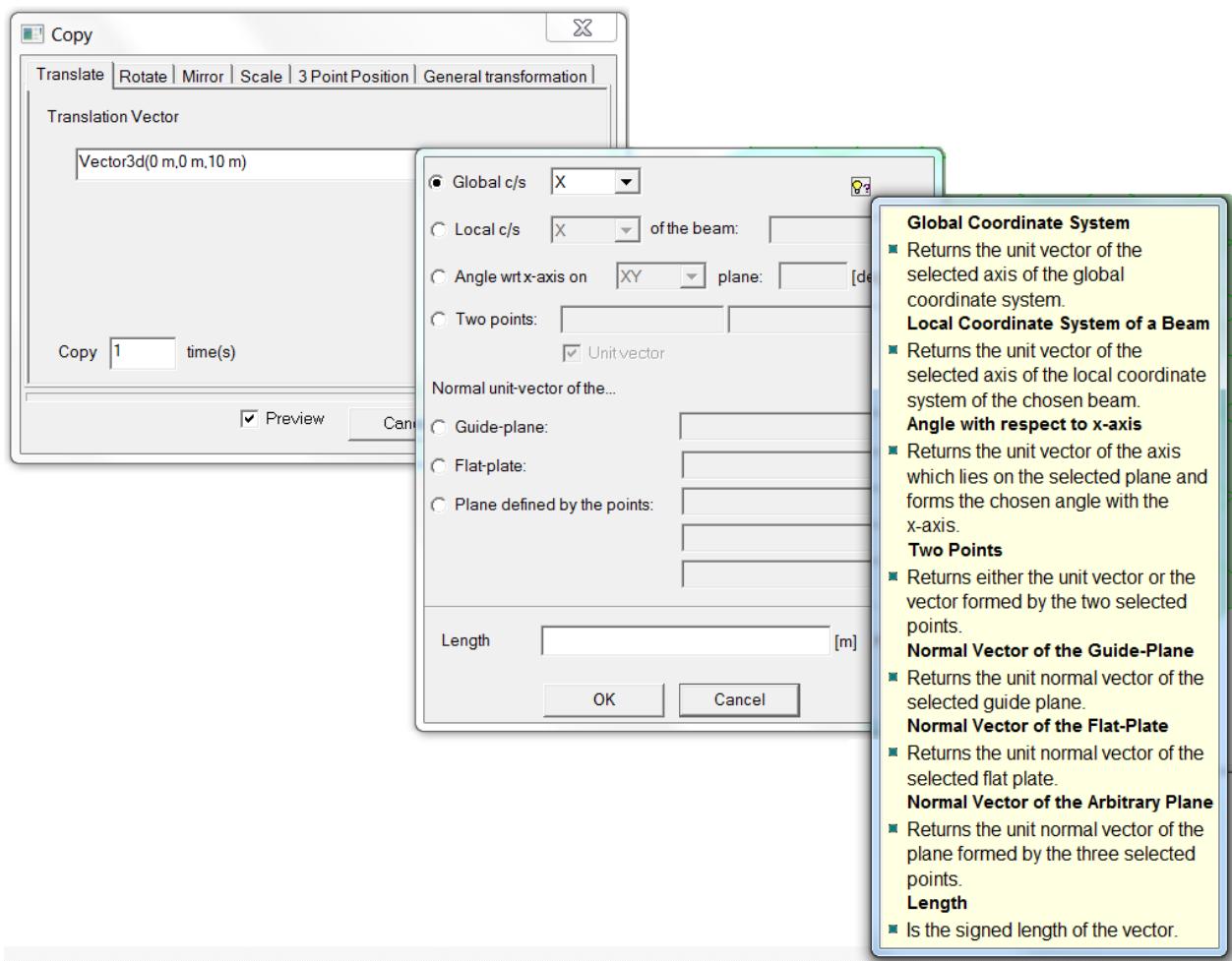


#### *Copy/Move using affine transformations*

When copying or moving guiding geometry entities, we apply an affine transformation to the entity. In order to define the transformations, data points, vectors etc need to be given. These can be written explicitly by

the user or they can be given by picking snapping points from the graphics screen. The possible choices offered in GeniE are the following:

- **Translation along a vector:** the vector can be given explicitly by the user or by moving the mouse over the button at the right-hand side of the vector text-box, the user has special choices, which are given in the following figure.

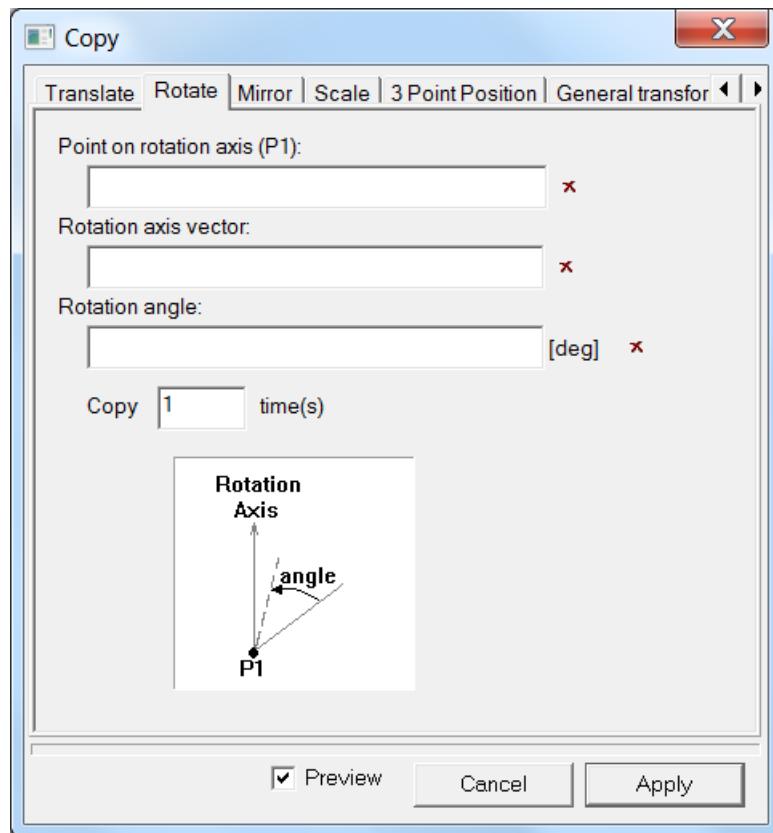


#### JS-commands :

**Copy :**              `<Curve> = entity.copyTranslate(translation_vector);`

**Move :**              `autoMSet = Set();  
autoMSet.clear();  
autoMSet.add(entity);  
autoMSet.moveTranslate(translation_vector);  
Delete(autoMSet);`

- **Rotation around an axis in the 3D space:** The axis is determined by a point and a vector and the angle can be given only explicitly.



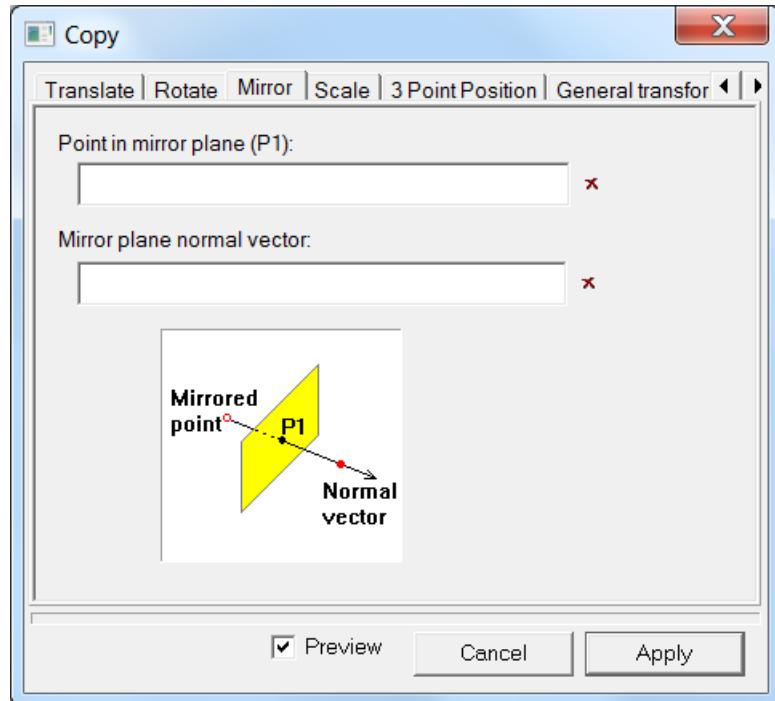
**JS-commands :**

**Copy :**              `<Curve> = entity.copyRotate(point_of_rotation_axis,  
                                 vector_of_rotation_axis, rotation_angle);`

**Move :**

```
autoMSet = Set();  
autoMSet.clear();  
autoMSet.add(entity);  
autoMSet.moveRotate(point_of_rotation_axis,  
                         vector_of_rotation_axis, rotation_angle);  
Delete(autoMSet);
```

- **Mirror with respect to a plane:** the plane is determined by a point and its normal vector.



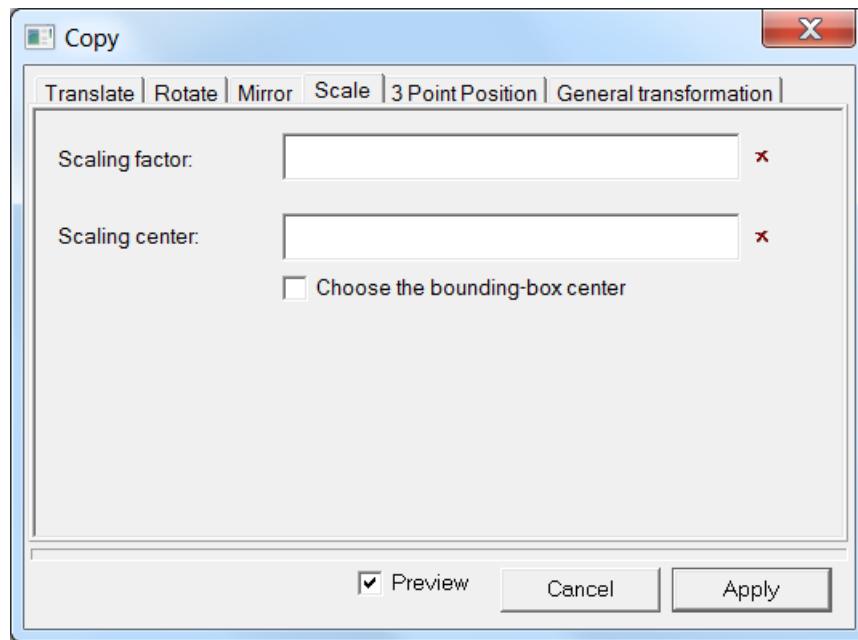
#### **JS-commands :**

**Copy :**              `<Curve> = entity.copyMirror(point_on_mirror_plane,  
                          normal_vector_of_mirror_plane);`

#### **Move :**

```
autoMSet = Set();
autoMSet.clear();
autoMSet.add(entity);
autoMSet.moveMirror(point_on_mirror_plane,
                          normal_vector_of_mirror_plane);
Delete(autoMSet);
```

- **Uniform scaling with respect to a point:** The scaling factor is to be given explicitly, while the center of the scaling can be automatically computed as the center of the entity's bounding box.



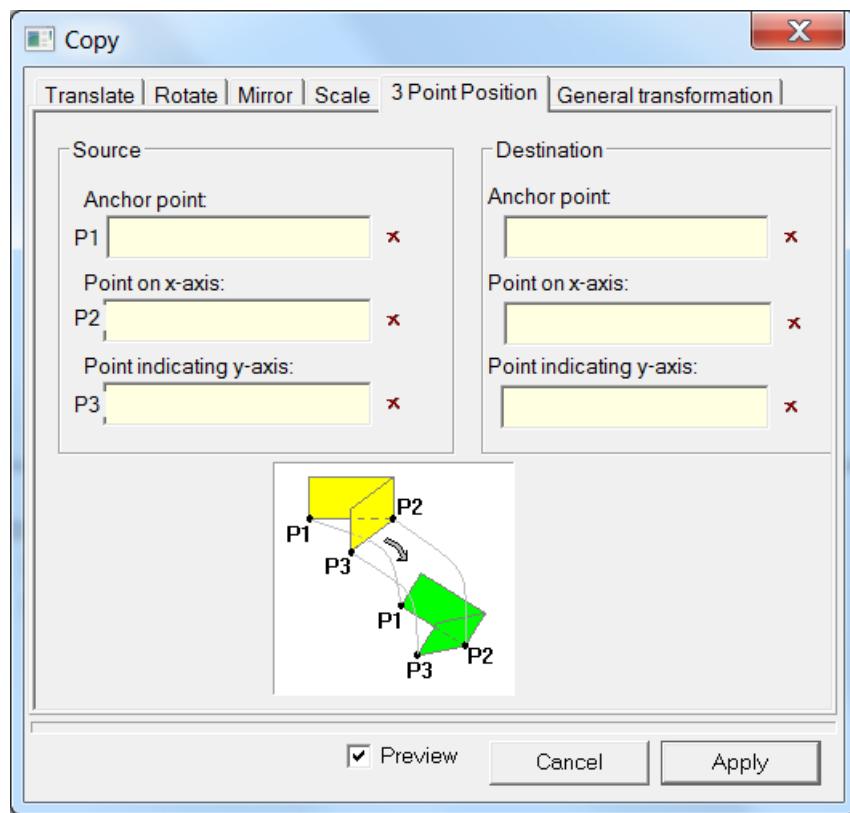
**JS-commands :**

**Copy :** <Curve> = entity.copyScale(scaling\_factor, scaling\_center);

**Move :**

```
autoMSet = Set();
autoMSet.clear();
autoMSet.add(entity);
autoMSet.moveScale(scaling_factor, scaling_center);
Delete(autoMSet);
```

- **3 point transformation:** A 3 point position copy/move operation requires 6 input values; it is necessary to specify 3 reference points and 3 target points. This is illustrated by copying the vertical, yet skew, guide plane from one side to the other. The source points in this case are denoted P1, P2 and P3 while the destination points are named D1, D2 and D3 on the picture below. This copy/move technique is particularly of advantage when modelling battered jackets.



**JS-commands :**

```

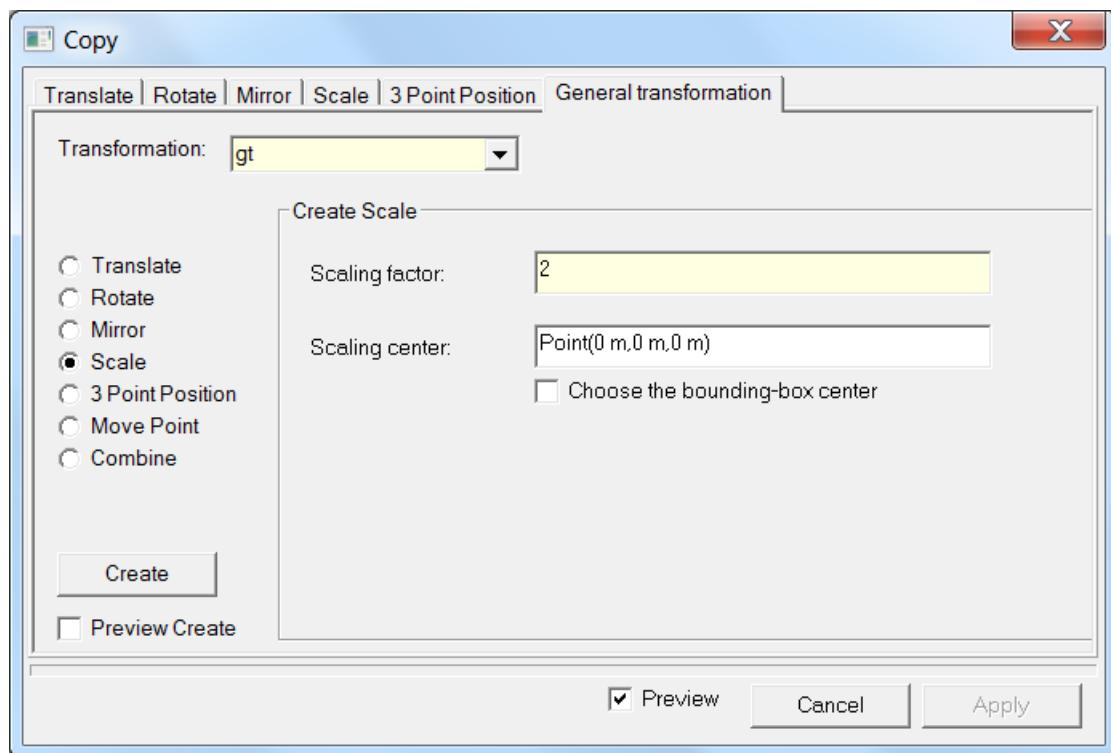
Copy : <Curve> = entity.copy3Point(source_anchor_point,
                                         source_point_on_x, source_point_on_y,
                                         destination_anchor_point, destination_point_on_x,
                                         destination_point_on_y);

Move :

autoMSet = Set();
autoMSet.clear();
autoMSet.add(entity);
autoMSet.move3Point(source_anchor_point, source_point_on_x,
                     source_point_on_y, destination_anchor_point,
                     destination_point_on_x, destination_point_on_y);
Delete(autoMSet);

```

- **General affine transformation:** The objective of this functionality is to define a transformation which is a combination (multiplication) of the basic affine transformations and save it with the name “tr”. The next two figures show how combined scale and mirroring can be defined. First, define the scaling and create/save it by pressing the “Create” button and then define the mirroring and multiply it with the previous one by pressing the “Multiply” button.

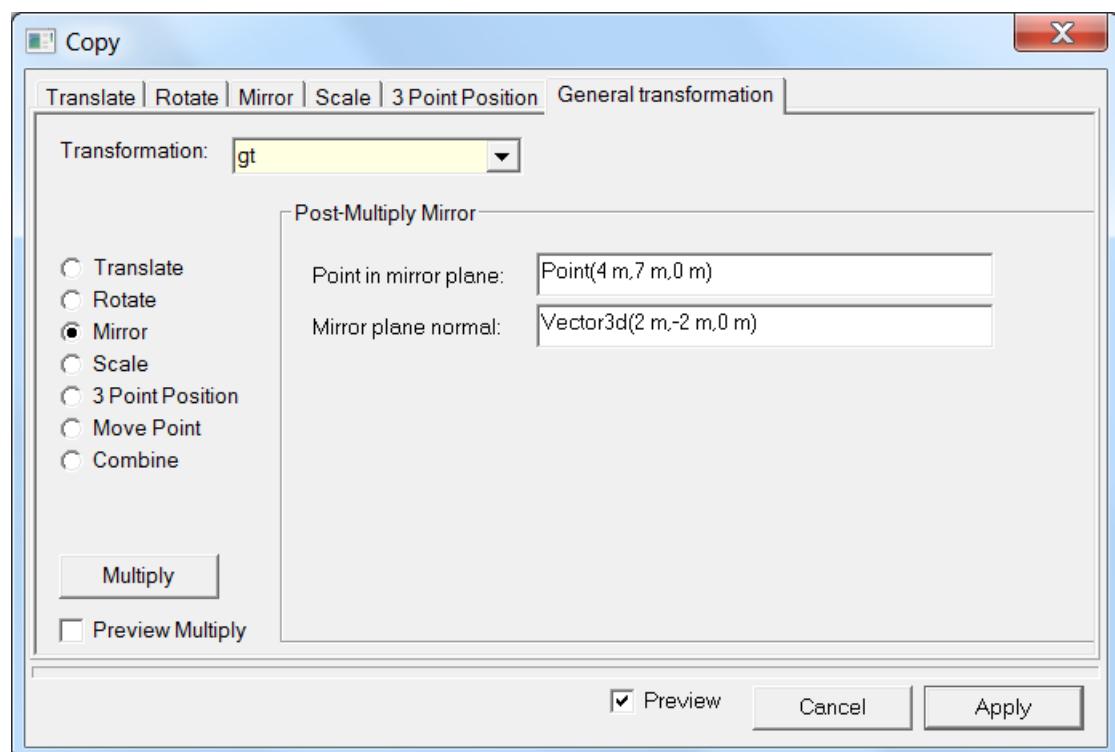


**JS-commands:**

```
Transformation: gt = ScaleC(scale_factor, scaling_center) ;
               gt.mirror(point_on_mirror_plane,
                           normal_vector_of_mirror_plane) ;
```

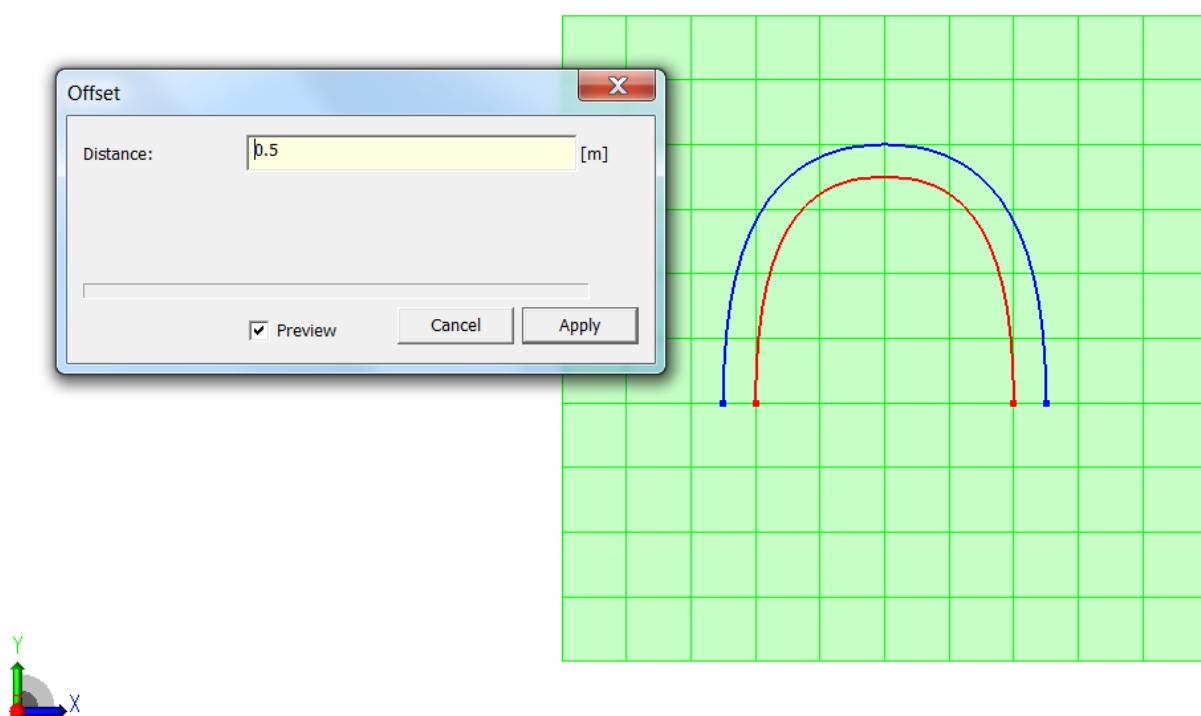
**Copy:** <Curve> = entity.copyTransform(gt);

```
Move: autoMSet = Set();
      autoMSet.clear();
      autoMSet.add(entity);
      autoMSet.moveTransform(gt, geUNCONNECTED);
      Delete(autoMSet);
```

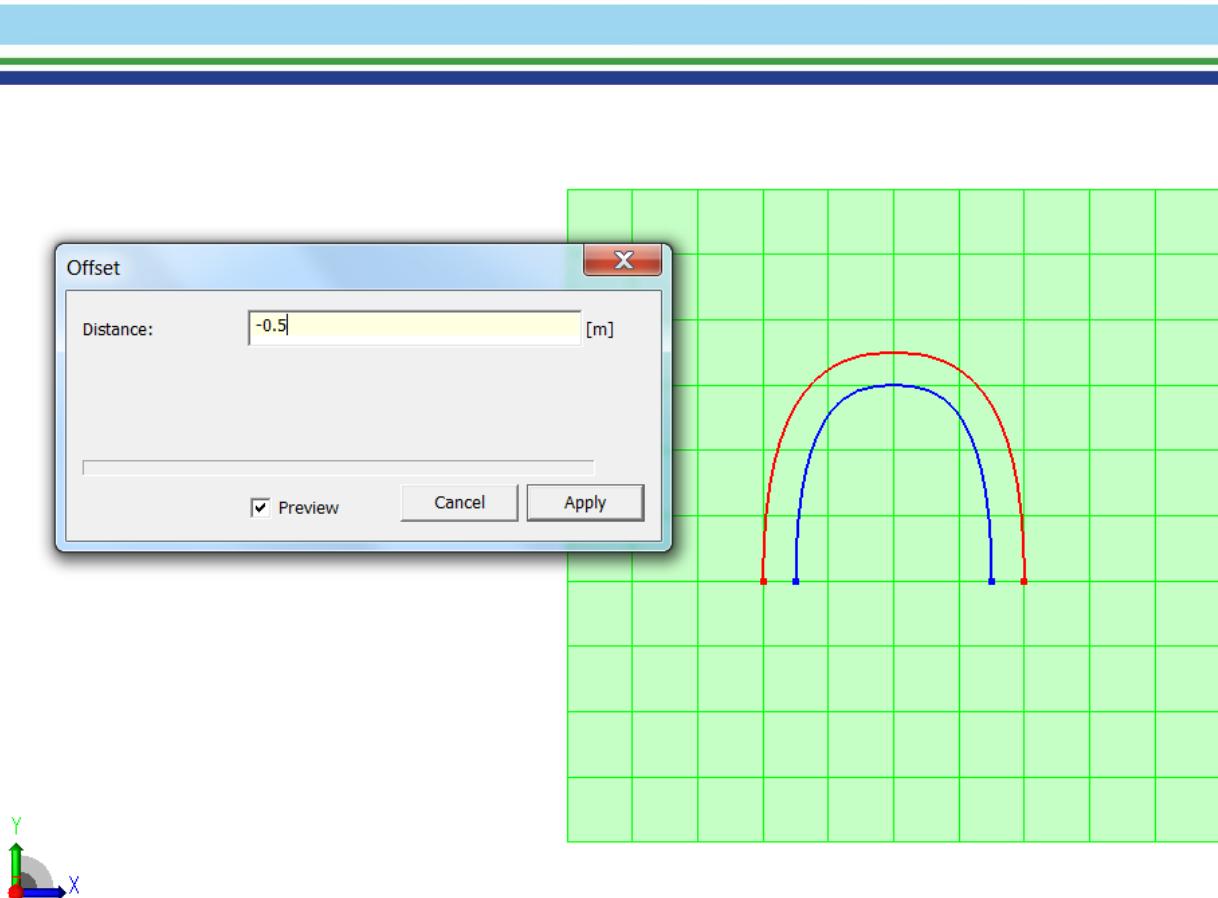


### 3.3.13 Offsetting a guiding curve

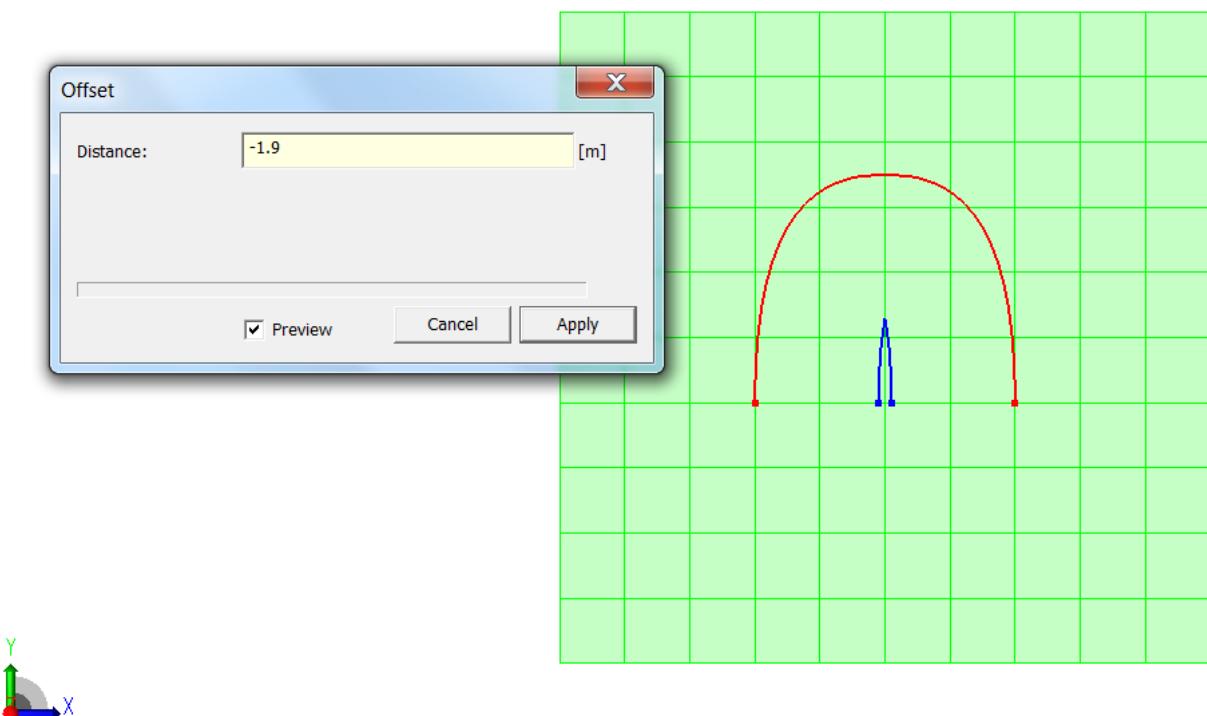
An **offset** of a curve is the envelope of a family of congruent circles centered on the curve. It generalises the concept of parallel lines. It can also be defined as a curve whose points are at a fixed normal distance of a given curve. Selecting one or more guiding curves on the RMB we choose “Offset”.



The given distance can also be negative. In this case the offset-curve is on the other side of the original one.



Offsetting may lead from smooth curves to non-smooth ones. It can also lead to self-intersecting curves. In this case GeniE returns an error and does not show the preview.



**JS-command:** <Curve> = Curve.offset(signed\_distance);

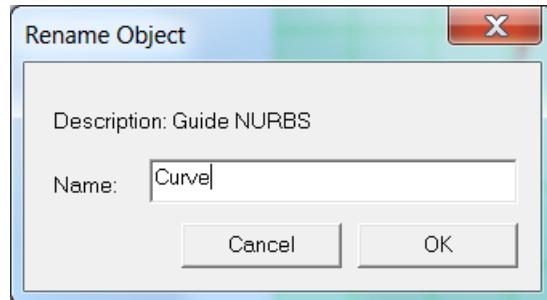
### 3.3.13 Converting a guiding curve from its old-format to NURBS

The format of the guiding curves has been changed since Version 6. When selecting one or more guiding curves and then on the RMB choosing “Convert to NURBS” converts them to the new format.

**JS-command:** <Curve> = GuideNURBS (old\_format\_curve);

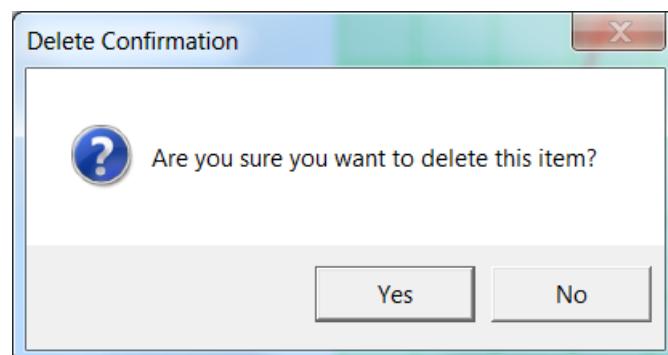
### 3.3.14 Rename and Delete a guiding curve

In general, all objects can be renamed and deleted after they have been created. Typically this is done by selecting one or more objects from the graphics screen or from the browser and by selecting the corresponding commands on the **RMB**. The “Rename” item shows up the following dialog, giving the prompt to rename the selected object:



**JS-command:** Rename (Curve, "NewCurve");

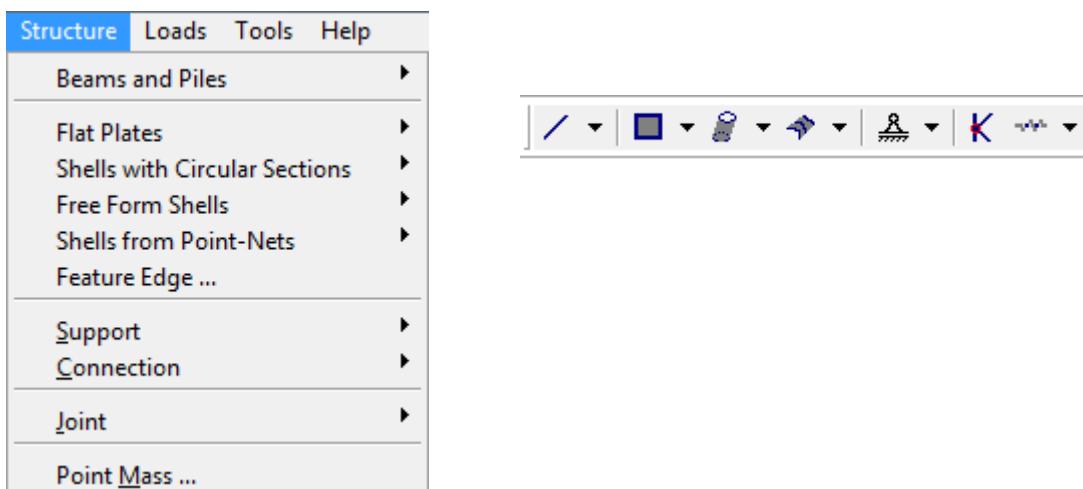
The “Delete” item can be applied to many entities simultaneously, after asking for a verification from the user:



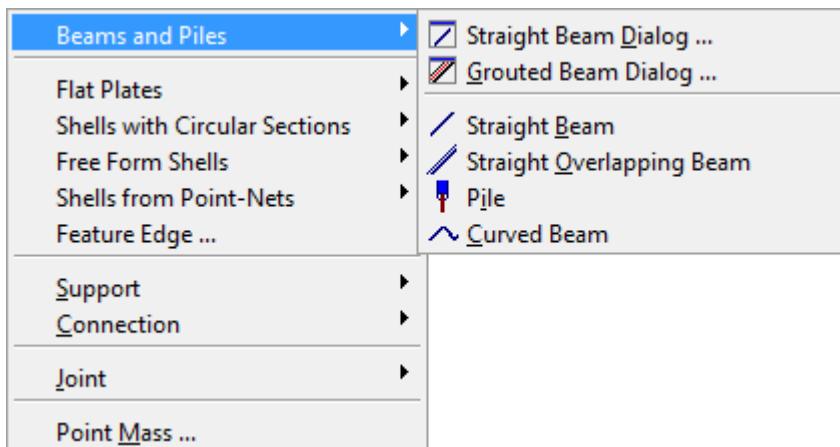
**JS-command:** Delete (Curve);

## 3.4 Structure

The menu “Structure” provides methods to create structural parts of the model. Most of the methods are geometrical and make use of the guiding geometry entities. Inserting structural entities graphically is made available from the pull-down menu and the toolbar shown below.

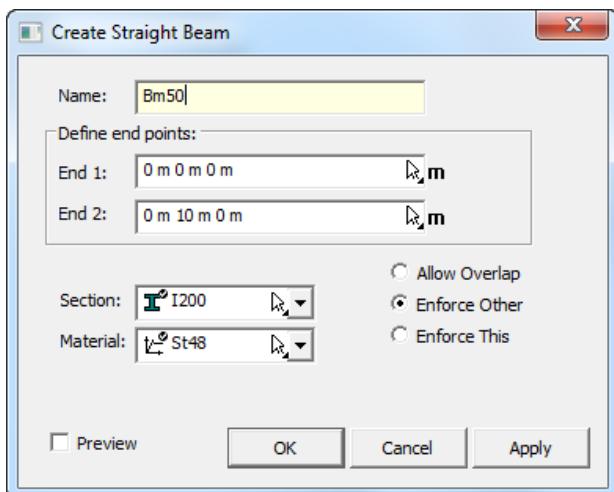


### 3.4.1 Beams and Piles



#### Straight Beam Dialog

**Graphically:** Opens a dialog for creating a straight beam.



**Name:** Specify the name of the beam, or use the default name.

**Define end points:** Enter the two endpoints for the beam.

**Section:** Specify section for the beam.

**Material:** Specify material for the beam.

The following three radio buttons decide how to treat overlapping of an existing beam:

**Allow Overlap:** Allow this beam to overlap an existing beam.

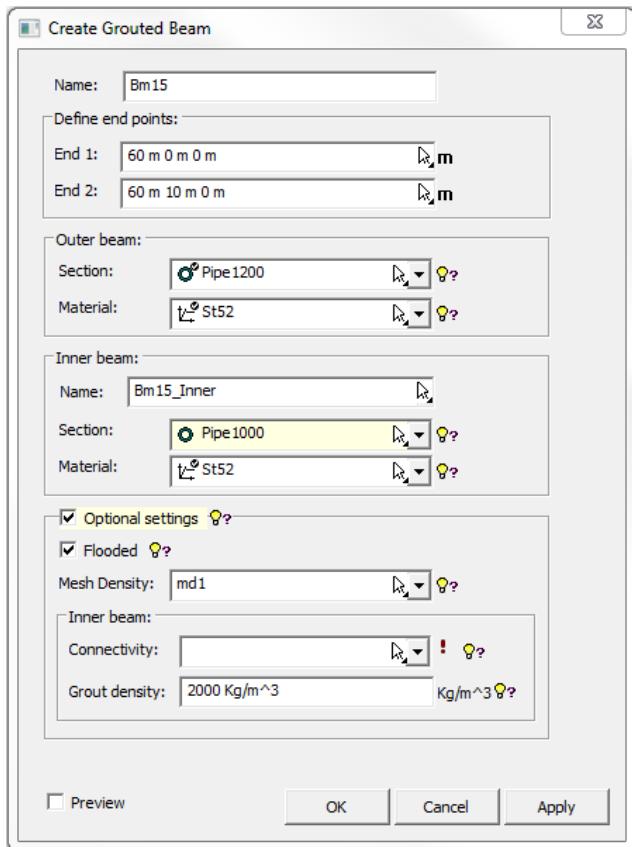
**Enforce Other:** This beam will be trimmed, existing overlapping beam will not be trimmed.

**Enforce This:** This beam will not be trimmed, existing overlapping beam will be trimmed.



## Grouted Beam Dialog

**Graphically:** Opens a dialog for creating a grouted beam.



**Name:** Specify the name of the outer beam, or use the default name.

**Define end points:** Enter the two endpoints for the grouted beam.

### Outer beam

**Section:** Select a section for the outer beam. This needs to be a pipe section with diameter larger than the inner beam.

**Material:** Select a material for the outer beam.

### Inner beam

**Name:** Specify the name of the inner beam, or use the default name.

**Section:** Select a section for the inner beam. This needs to be a pipe section with diameter smaller than the inner beam.

**Material:** Select a material for the inner beam.

**Optional settings:** Check this if you want to change any of the optional settings.

**Flooded:** By default it is assumed that the grouted beam is flooded. The flooding condition is assigned to both inner and outer beam.

**Mesh Density:** A mesh density property may be assigned to both inner and outer beam. This gives the spacing between the nodes connecting the inner and outer beam. If no density is assigned, the global default is used. Note that if no global default exists, 1 element per beam segment is created.

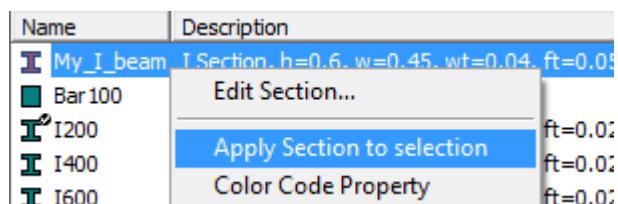
**Connectivity:** Assign a beam type property specifying the connection between inner and outer beam. If no beam type is assigned, then full connections at nodes will be assumed.

**Grout density:** The grout material density must be specified. The default value is 2000 kg/m<sup>3</sup>. The void space between inner and outer beam is assumed to be filled with this material. The increased weight will be assigned to the inner beam.

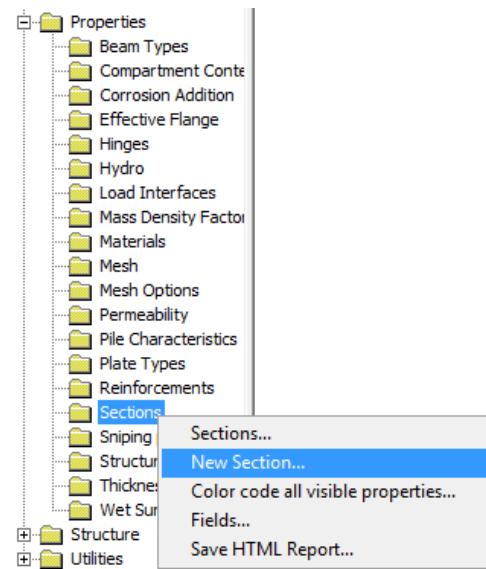
 Straight Beam	<b>Graphically:</b> Select two points to create a straight beam. JS-command: <code>&lt;Beam&gt; = StraightBeam(Point1, Point2);</code>
 Straight Overlapping Beam	<b>Graphically:</b> Select two points to create a straight beam. All or parts of the beam you are creating can overlap an existing beam. <code>&lt;Beam&gt; = StraightBeam(Point1, Point2, geAllowOverlap);</code>
 Pile	<b>Graphically:</b> A pile is easily modelled by the following three step procedure: a) Select the pile top (click a snap point in the structure, e.g. the lower end of a leg or pile sleeve) b) Specify the Z-level of the pile tip in the Snap plane dialog opening up (do not click anywhere in the model) c) Align the pile by clicking an alignment beam in the structure, e.g. a leg or pile sleeve JS-command: <code>&lt;Pile&gt; = Pile(Point1, Point2);</code>
 Curved Beam	<b>Graphically:</b> Select a guiding-curve. The function creates a curved beam, having the geometry of the selected curve. <code>&lt;CurvedBeam&gt; = Beam(Curve);</code>

### 3.4.1.1 The Section dialog

Every beam needs to have a cross section assigned to it



To apply a section to your beams select one/several beam(s), rightclick the section in the browser, and select “Apply Section to selection”.

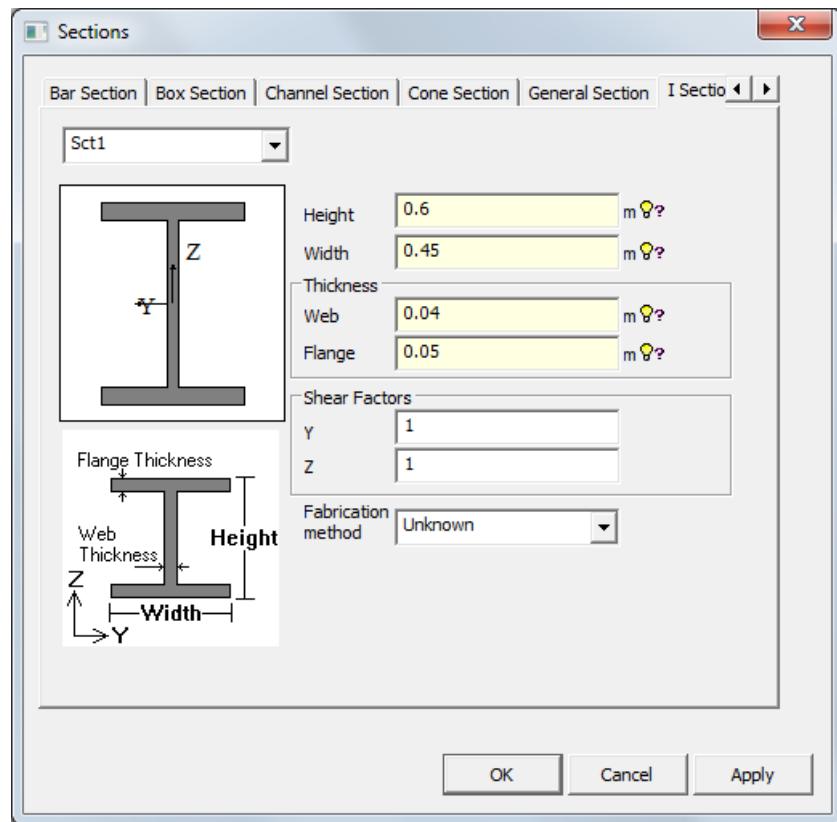


The section dialog is available from the browser. Here you can create a new cross section, or edit an existing one.

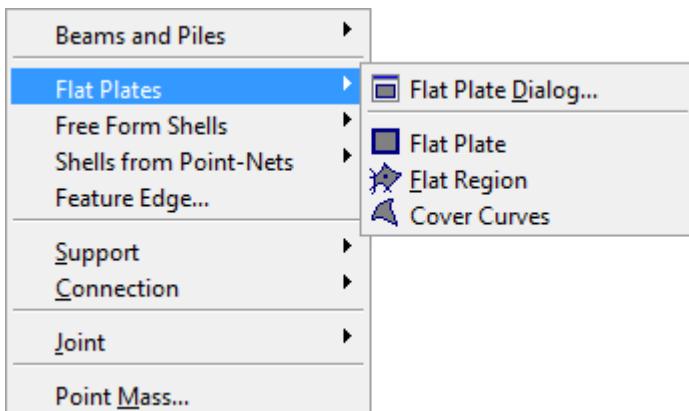
In the section dialog you specify the dimension for a beam cross section.

The upper illustration shows the section in correct scale according to the dimensions you have specified. It will change when you change the input values.

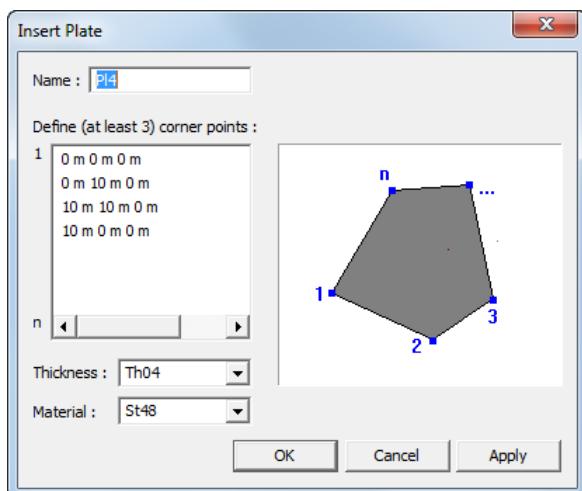
The lower illustration is a graphical explanation showing the measurements of the section.



### 3.4.2 Flat Plates



**Graphically:** Opens a dialog in order to create a Flat Plate.



**Graphically:** Select a set of ordered points to form a closed polygon. The function creates a Plate having this polygon as a boundary.

**JS-command:**

```
<Plate> = Plate(List of Points);
```

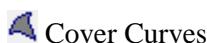
The points selected have to be co-planar and the polygon non self-intersecting.



**Graphically:** Opens a dialog in order to select a plane. Then click the “Select Flat Region in 2D View” button. The plane cuts the visible objects and shows the 2D intersections. Then, the user is able to select one or more 2D closed regions formed by the 2D intersections by clicking one point in each one of them.

**JS-command:**

```
<Plate> = Plate(Plane3d, Array(objects), Point);
```

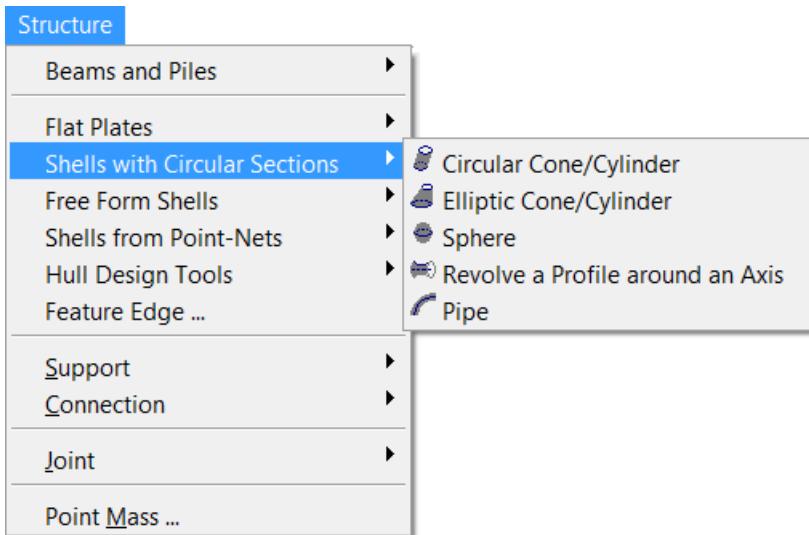


**Graphically:** Select a set of coplanar curves which form a closed 2D region, i.e. have common boundary points.

**JS-command:**

```
<Plate> = CoverCurves(Array(Curves));
```

### 3.4.3 Shells with circular sections



#### Circular Cone / Cylinder

**Graphically:** Click a position to define the first end of your cylinder or cone. Then type in the radius you want for the first end. Click on another position to define the second end of your cylinder / cone. Type in the radius you want for the second end. If this radius is different than the radius for the first end, a cone is created. If the two radii are identical a cylinder is created.

#### **JS-command:**

```
<Plate> = CreateShellCircularConeCylinder(Point1,  
Radius1, Point2, Radius2, from_angle, to_angle);
```

*from\_angle* and *to\_angle* decides where to start and stop the rotation. Both the angles are expressed in degrees. The default is “0,360”. The constraints are: -360 <= *from\_angle*, *to\_angle* <= 360 and *to\_angle*-*from\_angle*<=360



## Elliptic Cone / Cylinder

**Graphically:** Select the position of the start and the end point of the major axis of the cone/cylinder base.

Click a position to define the plane of the base of the cylinder. Enter a value for the ratio of the minor axis/major axis. If you set this to 1, a circle is created.

Click a position in the plane of the ellipse in the top of the cone/cylinder.

Enter a value for the ratio of the radius for the second ellipse/first ellipse. If you set this to 1 an elliptic cylinder is created, otherwise an elliptic cone is created.

### **JS-command:**

```
<Plate> = CreateShellEllipticConeCylinder(Point1,  
Point2, Point3, ratioMinorMajor, Point4,  
ratioSecondFirstEllipse, from_angle, to_angle);
```

*ratioMinorMajor* is the ratio of minor axis/major axis for the first ellipse.

*ratioSecondFirstEllipse* is the ratio of the size of the second ellipse/size of the first ellipse.

*from\_angle* and *to\_angle* decides where to start and stop the rotation. Both the angles are expressed in degrees. The default is “0,360”. The constraints are: -360 <= from\_angle, to\_angle <= 360 and to\_angle - from\_angle <= 360



## Sphere

Graphically: Select the centre of the sphere and then give its radius.

### **JS-command:**

```
<Plate> = CreateShellSphere(Center, radius,  
poles_axis, from_long_angle, to_long_angle,  
from_lat_angle, to_lat_angle);
```

*From\_long\_angle, to\_long\_angle, from\_lat\_angle and to\_lat\_angle* determine where to start and stop the rotation of the sphere. All the angles are expressed in degrees. The default limits are “-90,90” for the longitude angle and “-180,180” for the latitude angles. The constraints are for the longitude: -90 <= from\_long\_angle, to\_long\_angle <= 90 and to\_long\_angle - from\_long\_angle <= 180 and for the latitude angle: -180 <= from\_lat\_angle, to\_lat\_angle <= 180 and to\_lat\_angle - from\_lat\_angle <= 360



## Revolve a Profile around an axis.

### **JS-command:**

```
<Plate> = CreateShellSurfaceOfRevolution(Curve1,  
Point1, Point2, from_angle, to_angle);
```

*from\_angle* and *to\_angle* decides where to start and stop the rotation. Both the angles are expressed in degrees. The default is “0,360”. The constraints are: -360 <= from\_angle, to\_angle <= 360 and to\_angle - from\_angle <= 360



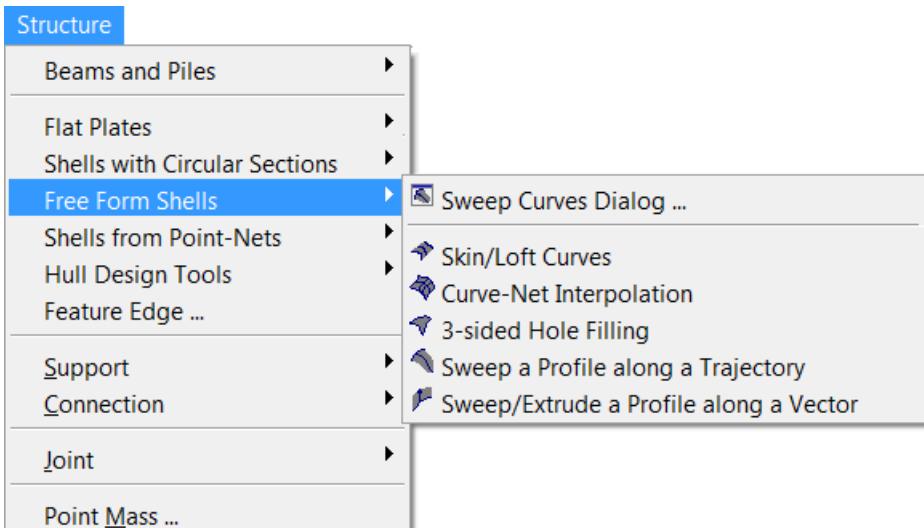
**Graphically:** Select a guiding curve to use as the spine curve of the pipe. Set a radius for the pipe. Note: the spine curve has to be planar.

**JS-command:**

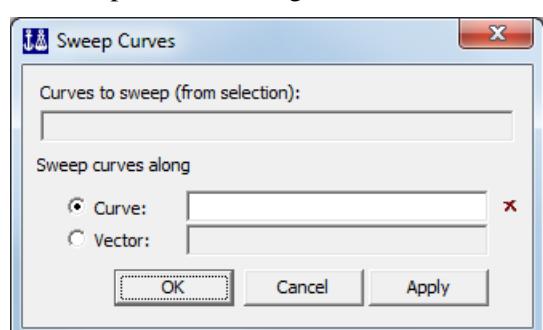
```
<Plate> = P16 = CreateShellPipeSurface(Curve1,  
radius, from_angle, to_angle);
```

*from\_angle* and *to\_angle* decides where to start and stop the rotation. Both the angles are expressed in degrees. The default is “0,360”. The constraints are:  $-360 \leq \text{from\_angle}, \text{to\_angle} \leq 360$  and  $\text{to\_angle} - \text{from\_angle} \leq 360$

### 3.4.4 Free Form Shells



**Graphically:** Opens a dialog in order to create a plate using sweep curve.



In the dialog you can select the curve to sweep and you can choose between sweeping along a curve or a vector.

The dialog covers the same as the two last menu choices for curved shells.



**Graphically:** Select a set of ordered curves in one direction. The function creates a curved-shell, which interpolates them.

**JS-command:**

```
<CurvedShell> = SkinCurves(Array(Curves));
```



**Graphically:** Select a set of ordered curves in two directions, which form a curve-net. The function creates a curved-shell, which interpolates them.

**JS-command:**

```
<Plate> = Plate(List of Points);
```

The curves selected have to form a net, i.e. each curve in one direction must intersect all the curves in the other direction. In case the curves in the first direction have end points which are not interpolated by a curve, the function constructs a selectable guiding curve, by interpolating the end points.

#### 3-sided Hole Filling

**Graphically:** Select three curves, which form a hole in the 3D space, i.e. every couple of them has one intersection point.

**JS-command:**

```
<CurvedShell>=CreateShellLinearThreeSidedHoleFilling  
(Curve1, Curve2, Curve3);
```

#### Sweep a Profile along a Trajectory

**Graphically:** Select the profile curve. Then select a trajectory curve in order to create a curved-shell by sweeping the profile along the trajectory.

**JS-command:**

```
<CurvedShell> = SweepCurve(Profile, Trajectory);
```

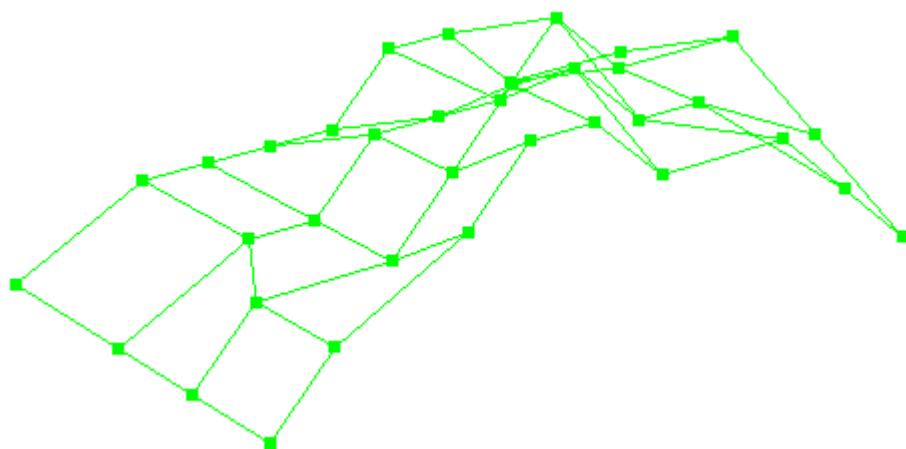
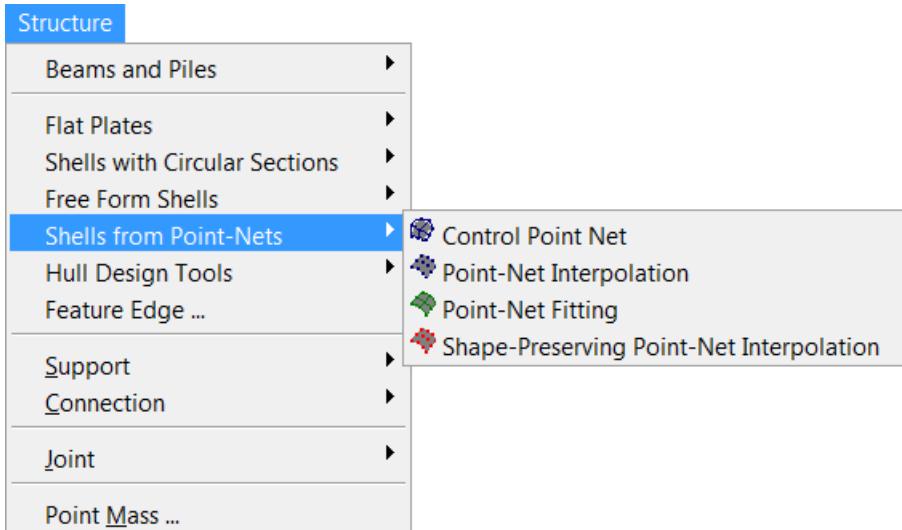
#### Sweep/Extrude a Profile along a Vector

**Graphically:** Select the profile curve. Then select a vector in order to create a curved-shell by sweeping the profile along the vector.

**JS-command:**

```
< CurvedShell> = SweepCurve(Profile,Vector);
```

### 3.4.5 Shells from Point-Nets



Control Point Net

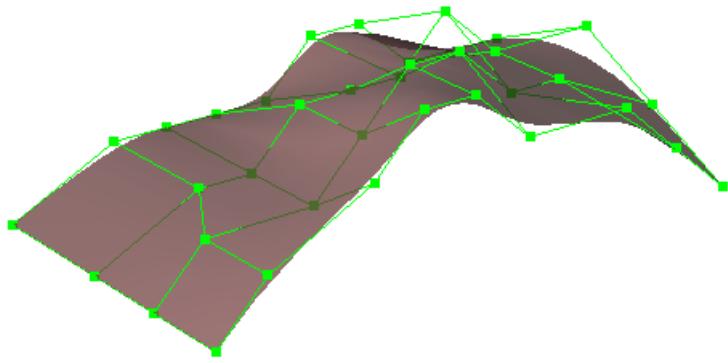
**Explanation:** The function creates a bi-cubic B-Spline surface, which uses the selected point-net as the control polygon net.

**Graphically:** Select a point-net. The function creates a bi-cubic B-Spline surface, which uses the selected point-net as the control polygon net. The shell will typically not intersect all the points in the control polygon net.

**JS-command:**

**JS-command:**

```
<CurvedShell> =  
CreateBSplineSurfaceUniformKnotVectors(GuidePointSet,  
Degree-u = 3, Degree-v = 3);
```



 Point Net  
Interpolation

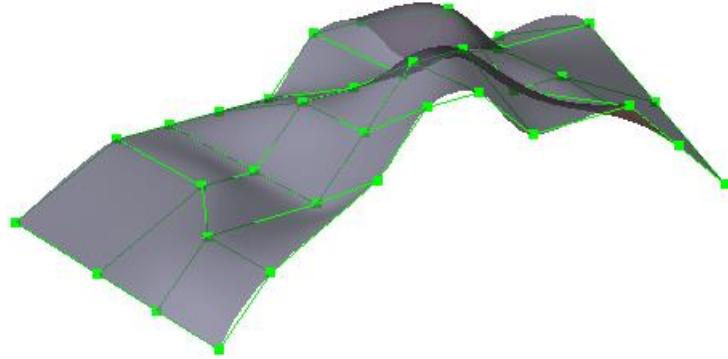
**Explanation:** Select a point-net. The function interpolates the point-sets in u and v directions, thus forming a curve-net. Then, interpolates the curves thus creating a curved shell.

**Graphically:** Select a rectangular net. The function will create a curved shell based on this rectangular net. The shell will intersect all the points in the rectangular net.

**JS-command:**

```
<CurvedShell> =
InterpolateRectangularNet(GuidePointSet);
```

Note that the interpolation curves in u and v directions use the shape-preserving interpolation scheme and they are C1 cubic Hermite splines.

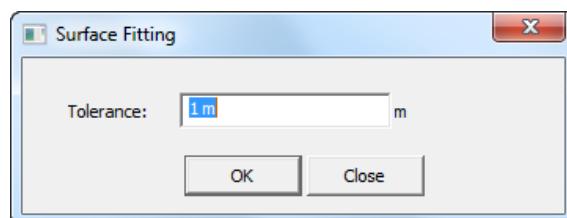


 Point-Net  
Fitting

**Explanation:** The function returns the bi-cubic surface which best fits the given point-net within the given tolerance.

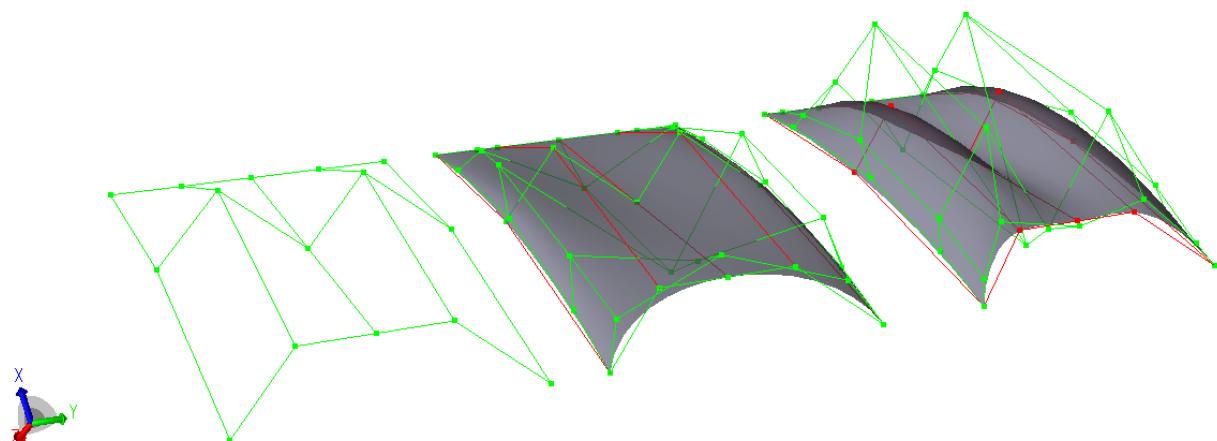
**Graphically:** Select a point-net and set the required tolerance.

**JS-command:** <Shell> = FitSurfaceToRectangularNet(Point-Net, tolerance);



To create a shell using point-net fitting you first select Point-Net Fitting. Then click on a point-net and enter the tolerance in the dialog that shows up.

Click “OK” and a shell is created.



The illustration to the left shows an example point-net.

The illustration in the middle shows a surface created by applying Point-Net-Fitting to the example point-net with a tolerance of 1 m.

The illustration to the right shows a surface created by applying Point-Net-Fitting to the example point-net with a tolerance of 0.1 m.

The surface to the right is expected to be less flat since the smaller the tolerance, the closer the surface is to the data.

The control-net of both the surfaces are shown in green while the initial data set is shown in red.

#### Shape Preserving Grid Interpolation

**Explanation:** The function returns a bi-cubic surface which interpolates the given point-net vertices.

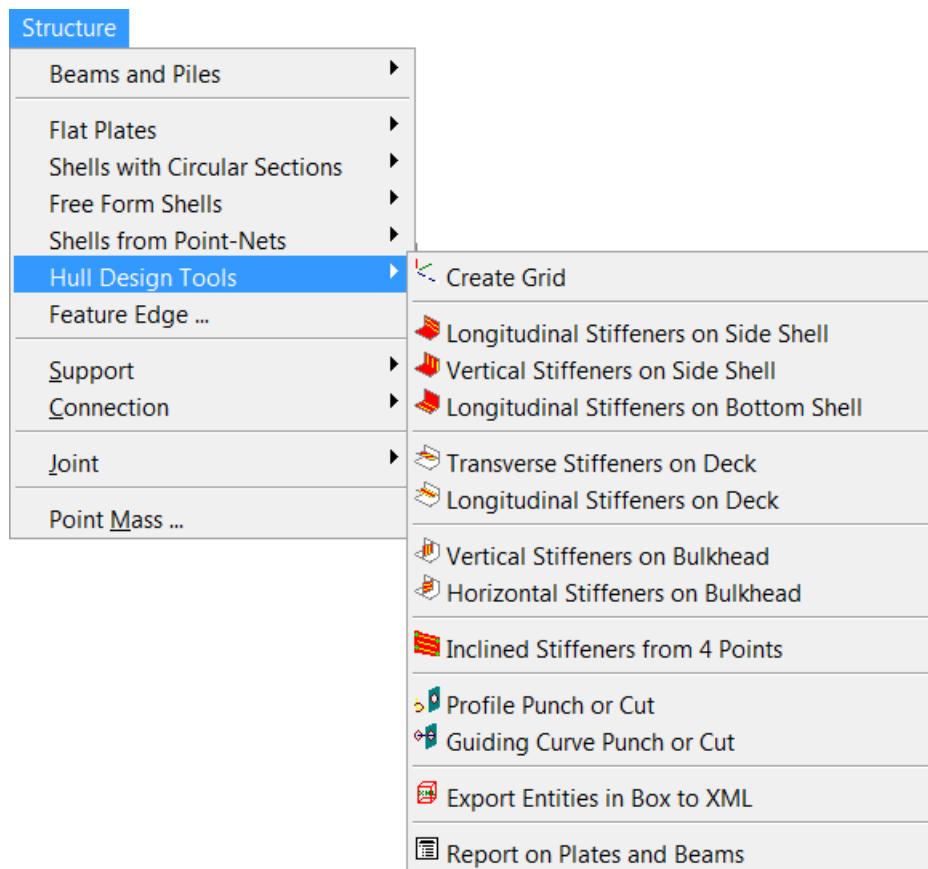
**Graphically:** Select a point-net.

**JS-command:** `<Shell> =  
InterpolateRectangularNetShapePreserving(Point-  
Net, RepresentationContinuity);`

RepresentationContinuity is an enumeration taking the values: `rocC0Continuity` (default) or `rocC1Continuity`

### 3.4.6 Hull Design Tools

The “Hull Design Tools” contain features specialized in modelling of the inner structure of a ship-hull.



 Create Grid

**Explanation:** The function returns a GuidingPointSet with NamedPoints, which discretizes in one of the main directions. The user has to supply the Prefix, a string, choose the direction, v, the starting point, p, and give the Indices of the nodes, N, where the length of the discretization changes. The Indices are given as a list of integers, I, separated by commas and the lengths of each part of the discretization (spacings) as a list of doubles, separated by commas. Note that if the number of elements in I is n, then the number of elements in S is n-1. The position of the nodes, where the discretization rule changes, is meant to be given by:

$$N[1] = p$$

$$N[i] = N[i-1] + v * (I[i]-I[i-1]) * S[i-1], i=2,\dots,n$$

The rule for the discretization of each part is the following:

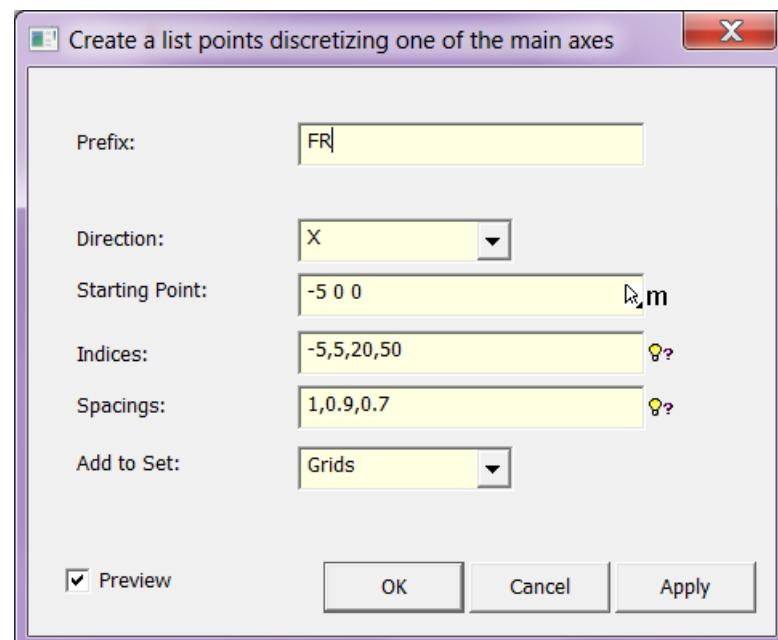
$$\text{Guiding Point-Set Vertex } [k] = N[i]+j*S[i],$$

$$\text{where } k=I[i]+j, \text{ for } j=0,\dots,I[i+1]-I[i] \text{ and } i=1,\dots,n-1.$$

Then, each vertex of the point set is supplied with a string, which can be used in order to obtain it from the point-set. The rule for setting the key strings is the following:

$$\text{Key-string of Vertex } [k] = \text{Prefix} + k$$

**Graphically:** The following dialog opens.

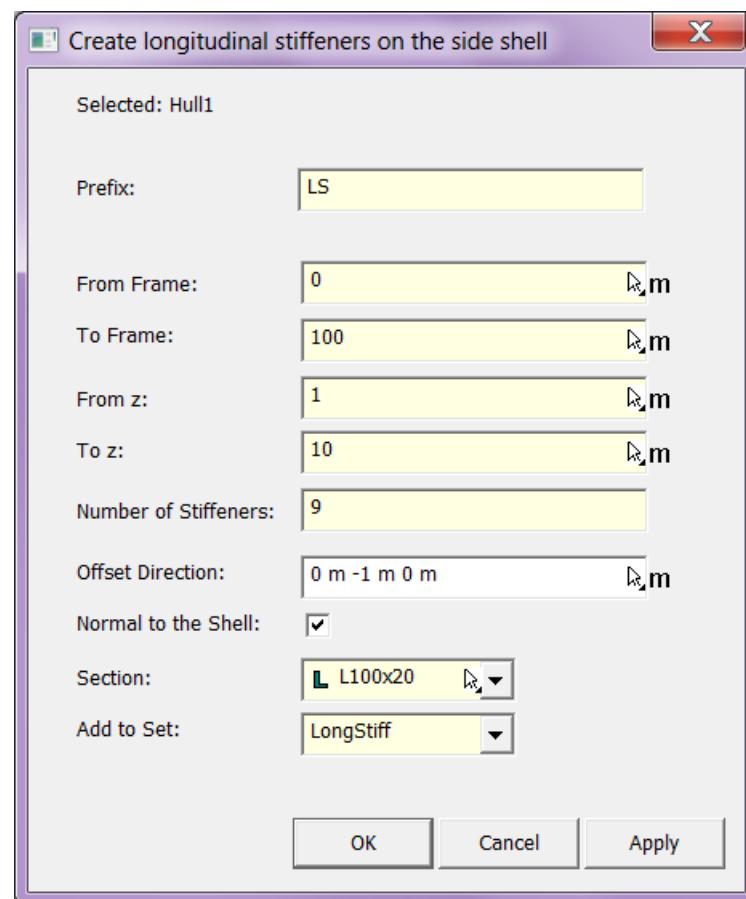


**JS-command:** <GuidingPointSet> = Create1DGridPointSet(  
Direction Vector3d, Starting Point, Array of Indices,  
Array of Spacings, Prefix);

 Longitudinal  
Stiffeners on the side  
shell

**Explanation:** The function places the given number of parallel stiffeners, n, on the selected shell. The stiffeners lie on the planes:  $z[i] = z1 + i * (z2 - z1) / n$  and they are accepted if their middle point is in [From Frame, To Frame].

**Graphically:** The following dialog opens



**JS-command:** `tmpArrayOfBeams = CreateLongitudinalBeamsOnSideShell (Shell, From Frame, To Frame, From Z, To Z, Number of Stiffeners);`

`Rename (tmpArrayOfBeams[i], "<Prefix>");`

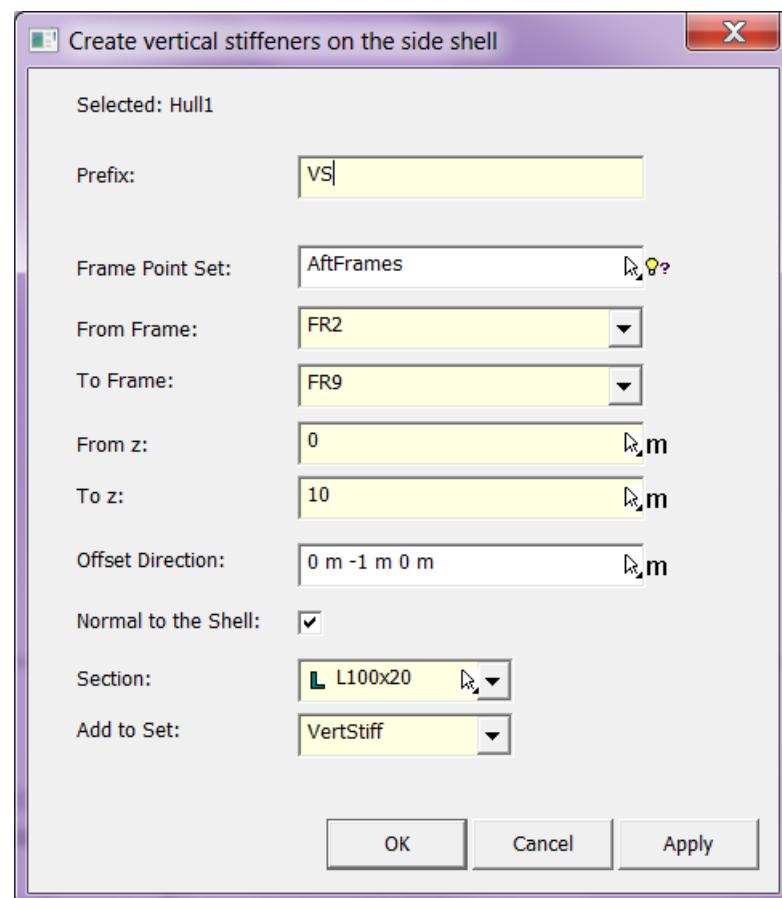
For each newly created beam the function applies the selected section, the selected set, the chosen offset vector and places the stiffener on the plate, if selected. For the above example we get for the first stiffener (index 0):

```
LS0.section(L100x20);
LongStiff.add(LS0);
LS0.rotateLocalX(90);
LS0.rotateLocalX(180);
LS0.setBeamOffsetLocal(Vector3d(0 , 0,-0.540909));
LS0.localSystemRule = RelativeToPlate(Hull1,0);
```

## Vertical Stiffeners on the Side Shell

**Explanation:** The function places a number of parallel stiffeners, on the selected shell. The stiffeners lie on the planes:  $x[i]=\text{frame}$ , where frame is in the discrete set {From Frame,...,To Frame} and they are accepted if their middle point is in [From Z,To Z].

**Graphically:** The following dialog opens



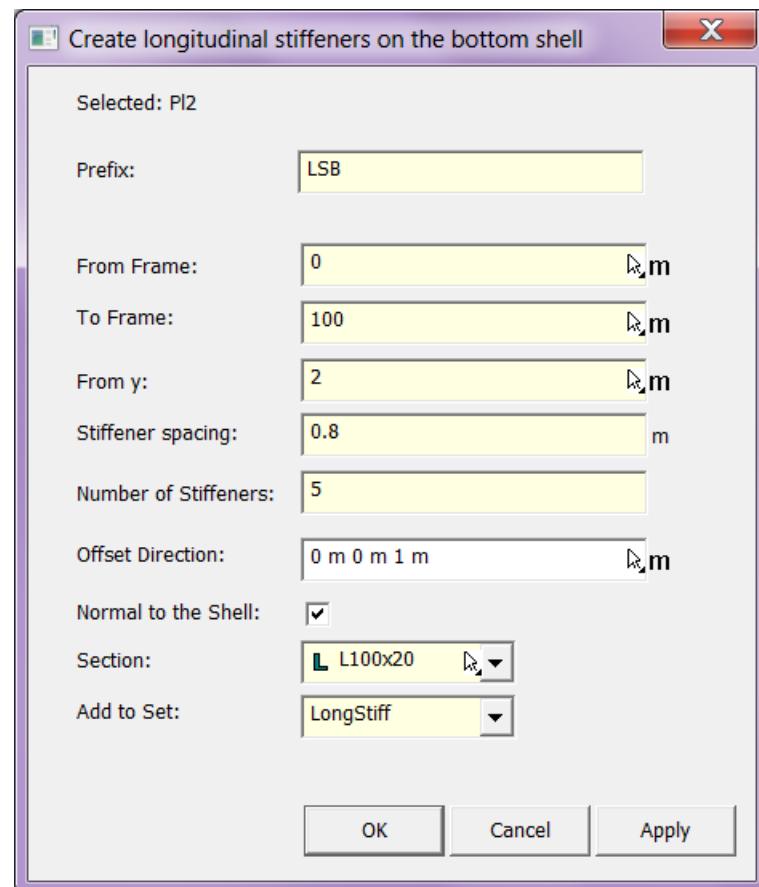
**JS-command:**

```
tmpArrayOfBeams =  
CreateVerticalBeamsOnSideShell(Shell, Guiding PointSet  
with NamedPoints, From Frame Name, To Frame Name, From Z,  
To Z);  
  
Rename(tmpArrayOfBeams[i], "<Prefix>");  
  
For each newly created beam the function applies the  
selected section, the selected set, the chosen offset  
vector and places the stiffener on the plate, if selected.  
For the above example we get for the first stiffener  
(index 0):  
  
VS0.section(L100x20);  
VertStiff.add(VS0);  
VS0.setBeamOffsetLocal(Vector3d(0 , 0,-0.540909));  
VS0.localSystemRule = RelativeToPlate(Hull1,0);
```

 Longitudinal  
Stiffeners on Bottom  
Shell

**Explanation:** The function places the given number of parallel stiffeners, on the selected shell. The stiffeners lie on the planes:  $x[i]=\text{frame}$ , where frame is in the descrete set {From Frame,...,To Frame}.

**Graphically:** The following dialog opens



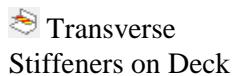
**JS-command:**

```
tmpArrayOfBeams =
CreateLongitudinalBeamsOnBottomShell(shell, From Frame, To
Frame, From y, Spacing, Number of stiffeners);

Rename(tmpArrayOfBeams[i], "<Prefix>");

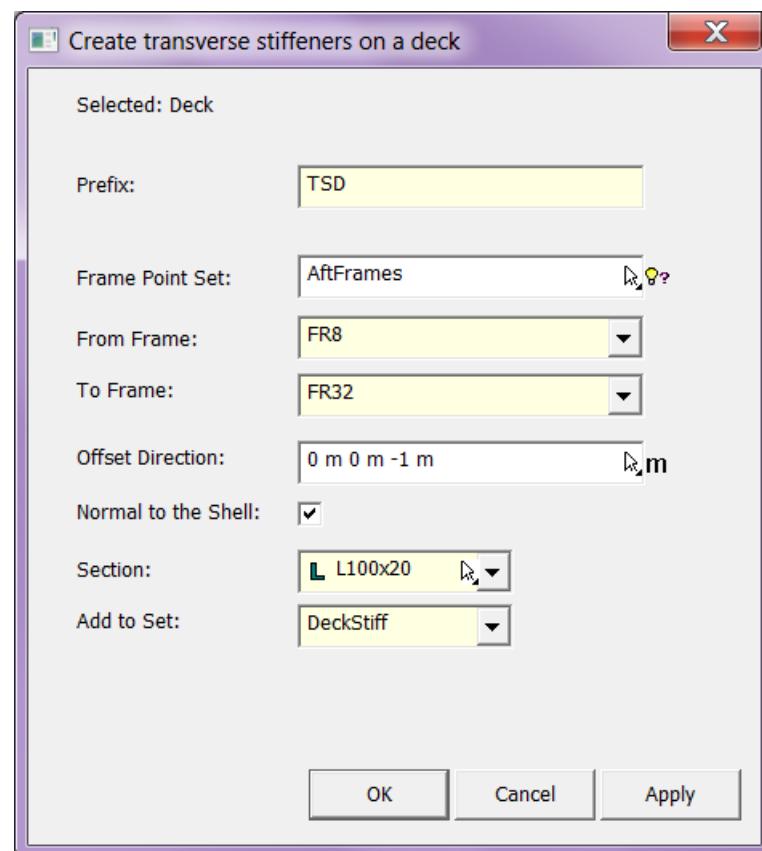
For each newly created beam the function applies the
selected section, the selected set, the chosen offset
vector and places the stiffener on the plate, if selected.
For the above example we get for the first stiffener
(index 0):

LSB0.section(L100x20);
LongStiff.add(LSB0);
LSB0.rotateLocalX(180);
LSB0.setBeamOffsetLocal(Vector3d(0 , 0,-0.540909));
LSB0.localSystemRule = RelativeToPlate(Pl2,0);
```



**Explanation:** The function places the given number of parallel stiffeners, on the selected shell. The stiffeners lie on the planes:  $y[i] = \text{From } y + s * i$ , where From y is the distance from the center line, and s is the stiffener spacing. A stiffener is accepted if its end points are both in [From Frame, To Frame].

**Graphically:** The following dialog opens

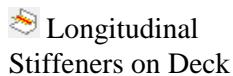


**JS-command:** tmpArrayOfBeams =  
CreateTransverseBeamsOnDeck(Shell, Guiding PointSet with  
NamedPoints, From Frame Name, To Frame Name);

Rename (tmpArrayOfBeams[i], "<Prefix>");

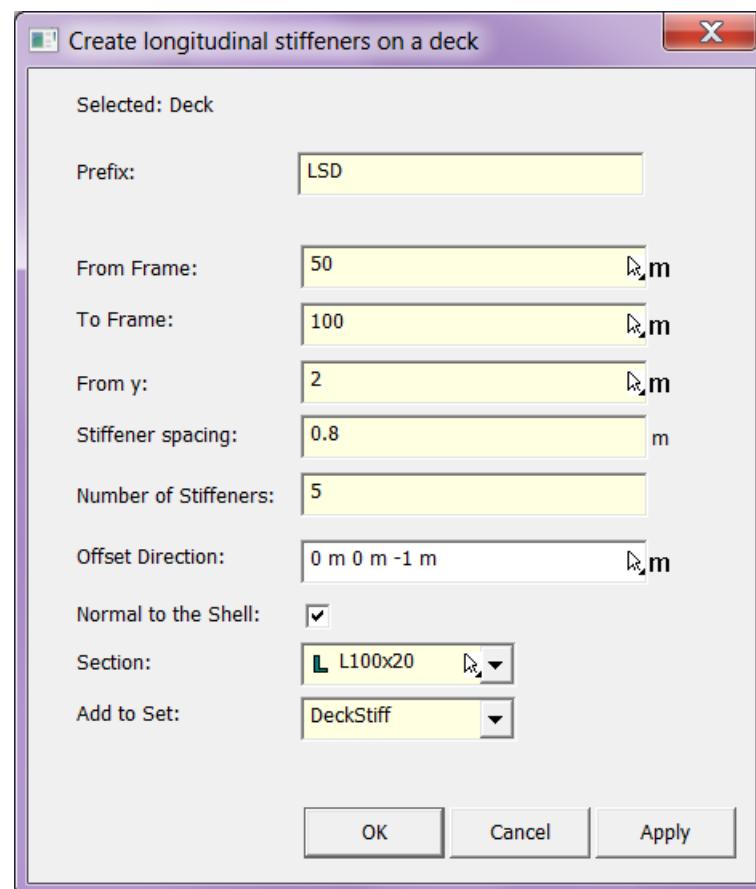
For each newly created beam the function applies the selected section, the selected set, the chosen offset vector and places the stiffener on the plate, if selected. For the above example we get for the first stiffener (index 0):

```
TSD0.section(L100x20);
DeckStiff.add(TSD0);
TSD0.setBeamOffsetLocal(Vector3d(0 , 0,-0.540909));
TSD0.localSystemRule = RelativeToPlate(Deck,0);
```



**Explanation:** The function places the given number of parallel stiffeners, on the selected shell. The stiffeners lie on the planes:  $y[i] = \text{From } y + s * i$ , where From y is the distance from the center line, and s is the stiffener spacing. A stiffener is accepted if its middle point is in [From Frame, To Frame].

**Graphically:** The following dialog opens

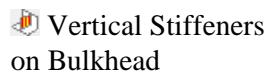


**JS-command:** `tmpArrayOfBeams = CreateLongitudinalBeamsOnDeck(Shell, From Frame, To Frame, From y, Spacing, Number of stiffeners);`

`Rename(tmpArrayOfBeams[i], "<Prefix>");`

For each newly created beam the function applies the selected section, the selected set, the chosen offset vector and places the stiffener on the plate, if selected. For the above example we get for the first stiffener (index 0):

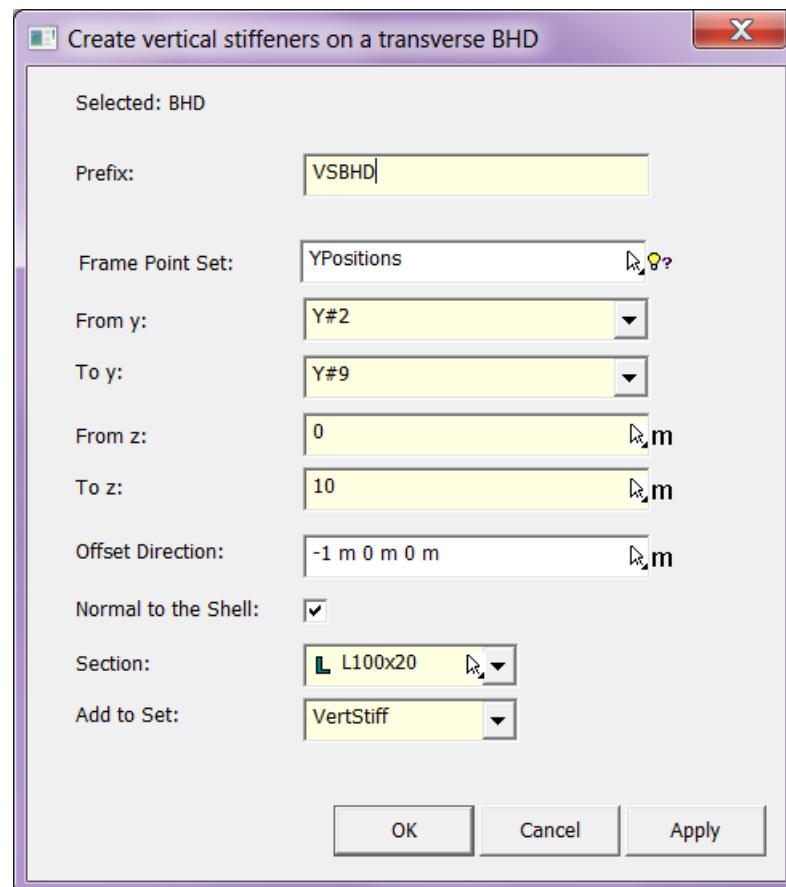
```
LSD0.section(L100x20);
DeckStiff.add(LSD0);
LSD0.setBeamOffsetLocal(Vector3d(0 , 0,-0.540909));
LSD0.localSystemRule = RelativeToPlate(Deck,0);
```



## Vertical Stiffeners on Bulkhead

**Explanation:** The function places the given number of parallel stiffeners, on the selected shell. The stiffeners lie on the planes:  $y$ , where  $y$  is in the discrete set {From  $y$ ,...,To  $y$ }. A stiffener is accepted if its middle point is in [From  $Z$ ,To  $Z$ ].

**Graphically:** The following dialog opens

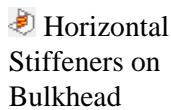


**JS-command:** tmpArrayOfBeams = CreateVerticalBeamsOnTransverseBHD(Shell, Guiding PointSet with Named Points, From Y Name, To Y Name, From Z, To Z);

Rename(tmpArrayOfBeams[i], "<Prefix>");

For each newly created beam the function applies the selected section, the selected set, the chosen offset vector and places the stiffener on the plate, if selected. For the above example we get for the first stiffener (index 0):

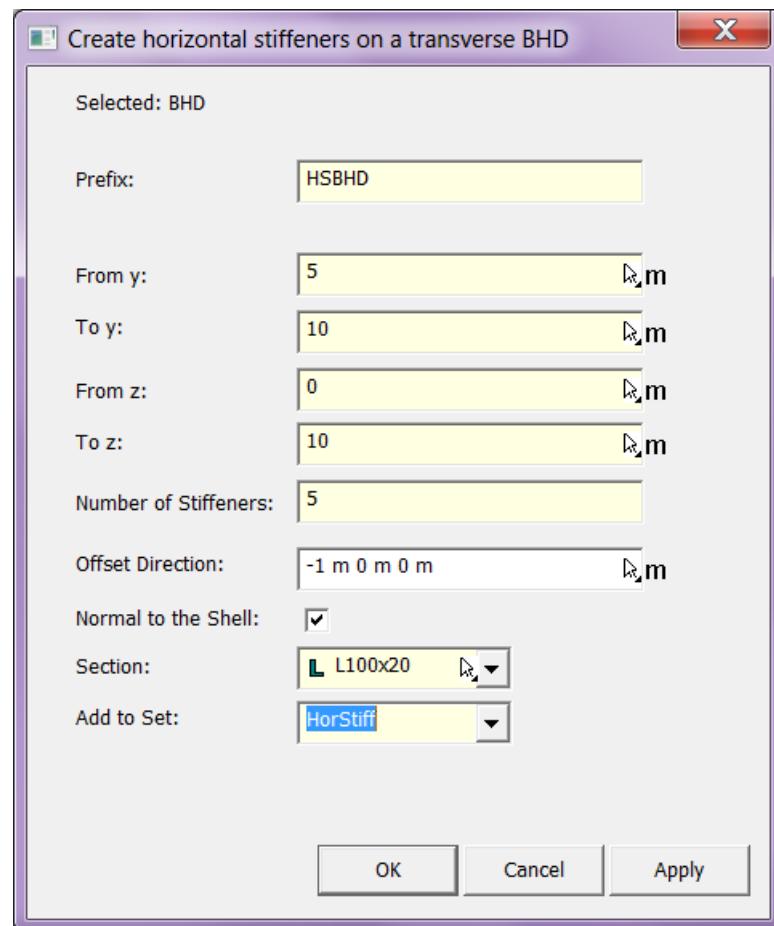
```
VSBHD0.section(L100x20);
VertStiff.add(VSBHD0);
VSBHD0.rotateLocalX(90);
VSBHD0.rotateLocalX(180);
VSBHD0.setBeamOffsetLocal(Vector3d(0 , 0,-0.540909));
VSBHD0.localSystemRule = RelativeToPlate(BHD,0);
```



## Horizontal Stiffeners on Bulkhead

**Explanation:** The function places the given number of parallel stiffeners, on the selected shell. The stiffeners lie on the planes:  $z[i] = z1 + i * (z2 - z1) / n$  and they are accepted if their middle point is in [From Y, To Y].

**Graphically:** The following dialog opens



**JS-command:**

```
tmpArrayOfBeams =
CreateHorizontalBeamsOnTransverseBHD(Shell, From y, To y,
From Z, To z, Number of stiffeners);

Rename(tmpArrayOfBeams[i], "<Prefix>");

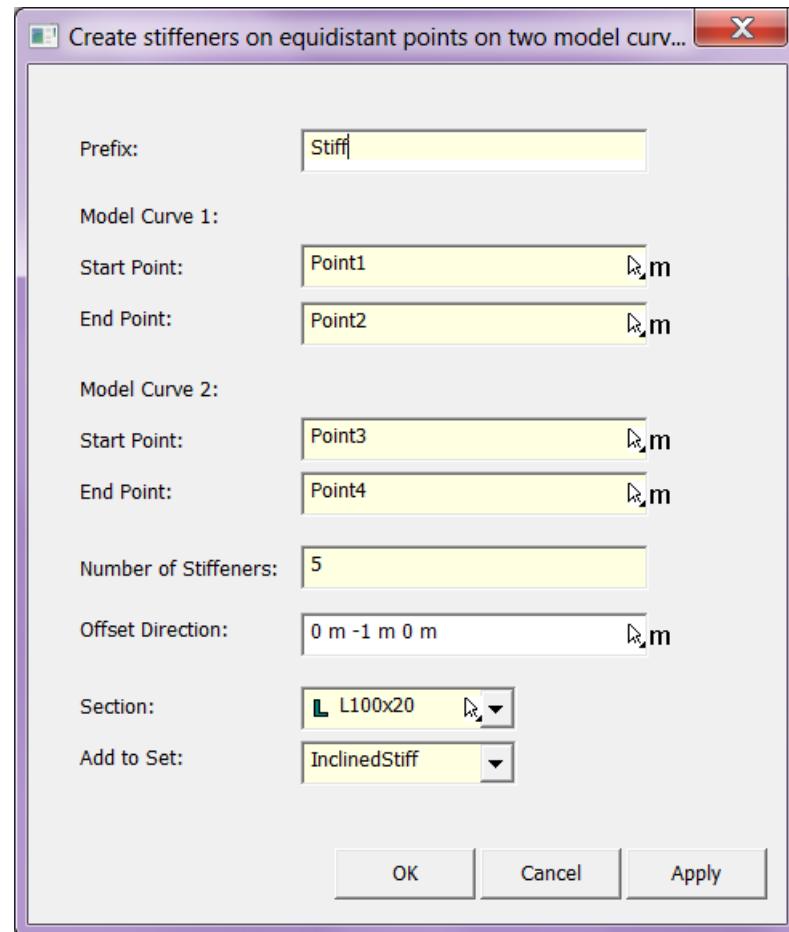
For each newly created beam the function applies the
selected section, the selected set, the chosen offset
vector and places the stiffener on the plate, if selected.
For the above example we get for the first stiffener
(index 0):

HSBHD0.section(L100x20);
HorStiff.add(HSBHD0);
HSBHD0.rotateLocalX(90);
HSBHD0.rotateLocalX(180);
HSBHD0.setBeamOffsetLocal(Vector3d(0 , 0,-0.540909));
HSBHD0.localSystemRule = RelativeToPlate(BHD,0);
```

 Inclined  
Stiffeners from 4  
Points

**Explanation:** The function places the given number of stiffeners, on the selected shell. The stiffeners are created with the following rule: (a) Genie creates two model curves connecting Point1 with Point2 and Point3 with Point4. (b) Genie computes the given number of points at equidistant lengths on the previous model curves. (c) Genie creates model curves connecting each couple of points of the step (b). (d) The model curves created in (c) are turned into beams.

**Graphically:** The following dialog opens



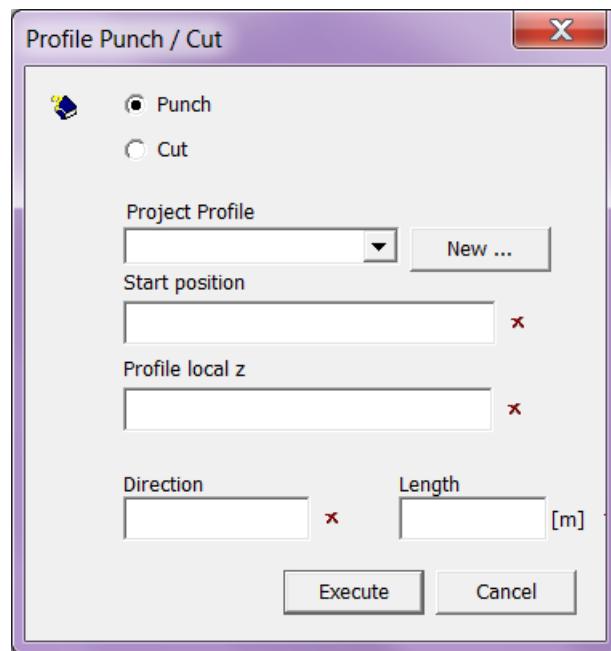
**JS-command:**

```
tmpArrayOfBeams =  
CreateInclinedBeamsFrom4Points(Point1, Point2,  
Point3, Point4, Number of Stiffeners);  
  
Rename(tmpArrayOfBeams[i], "<Prefix>");  
  
For each newly created beam the function applies the  
selected section, the selected set, the chosen offset  
vector. For the above example we get for the first  
stiffener (index 0):  
  
Stiff0.section(L100x20);  
InclinedStiff.add(Stiff0);  
Stiff0.rotateLocalX(90);  
Stiff0.rotateLocalX(180);  
Stiff0.setBeamOffsetLocal(Vector3d(0 , 0,-0.540909));
```

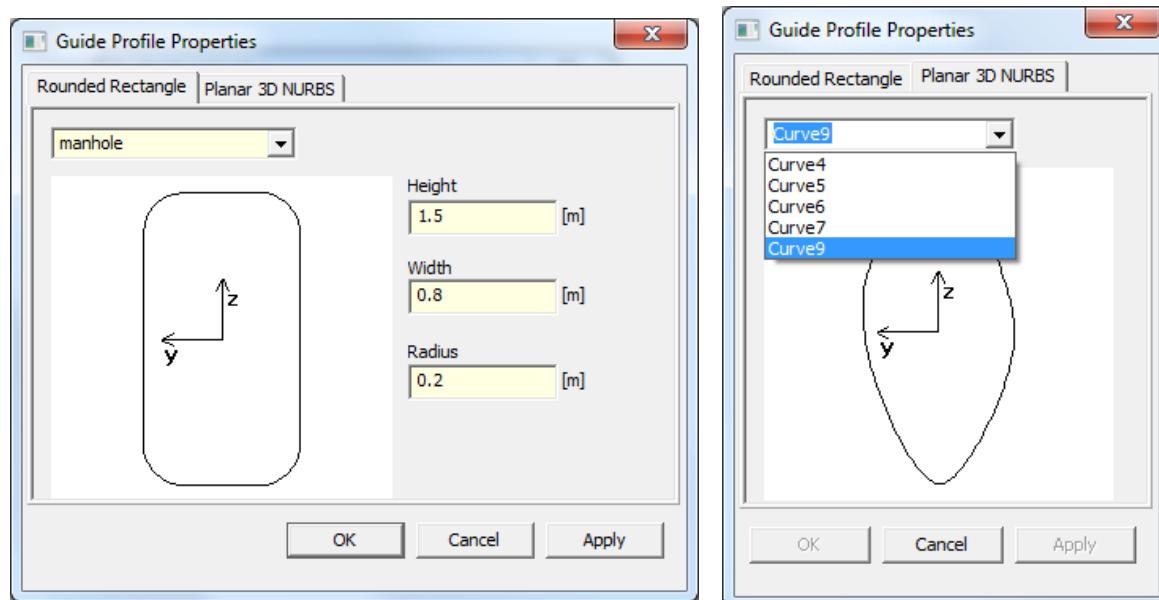
 Profile Punch or Cut

**Explanation:** the function cuts or punches the model, using a 2D profile.

**Graphically:** The following dialog opens:



Clicking the “New” button opens the the dialog below. You can choose between using a rounded rectangle and a planar 3D NURBS. The planar 3D NURBS can be created under Guiding Geometry | Advanced | Profile from NURBS.



Note: When selecting a planar 3D NURBS, just press “Cancel” in the dialog to use it. The purpose of having this as a separate dialog is for you to be able to see the planar 3D NURBS with its axes.

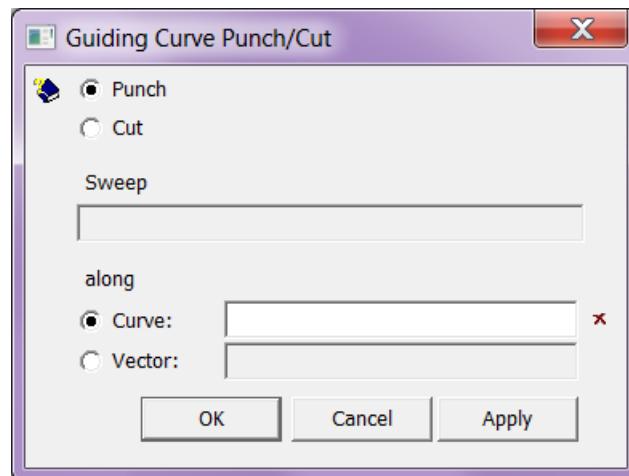
### JS-commands:

```
Profile2D.punch(start position,direction,profile  
normal vector,length);  
Profile2D.cut(start position,direction,profile normal  
vector,length);
```

#### Guiding Curve Punch or Cut

**Explanation:** the function cuts or punches the model, using a guiding curve. Both operations are based on sweeping the selected curve along a profile or a vector.

**Graphically:** The following dialog opens:

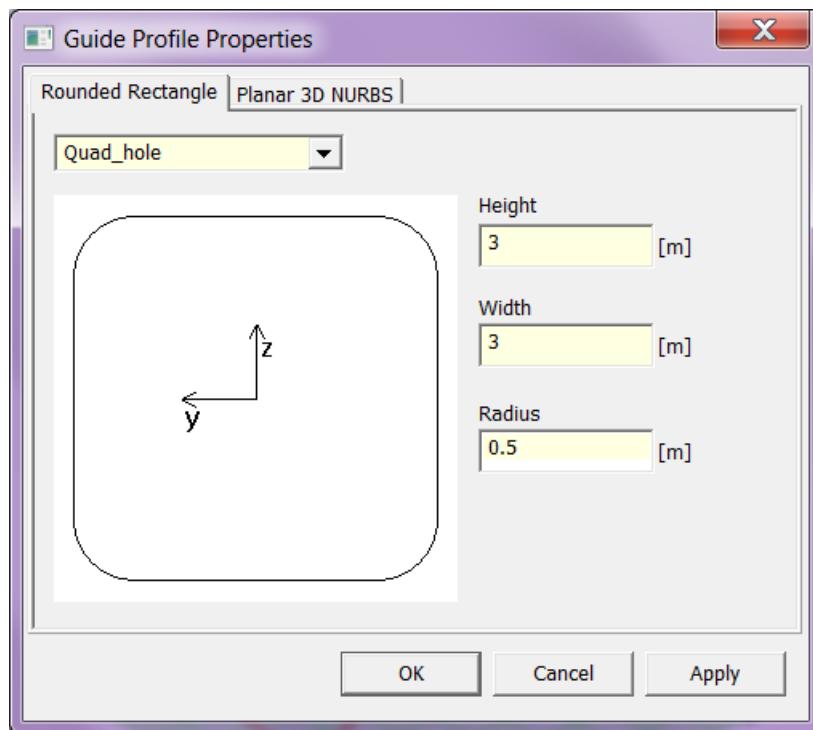


### JS-commands:

```
PunchWithCurves(Array of profiles, path curve or  
vector);  
CutWithCurves(Array of profiles, path curve or  
vector);
```

### 3.4.6.1 2D Profiles:

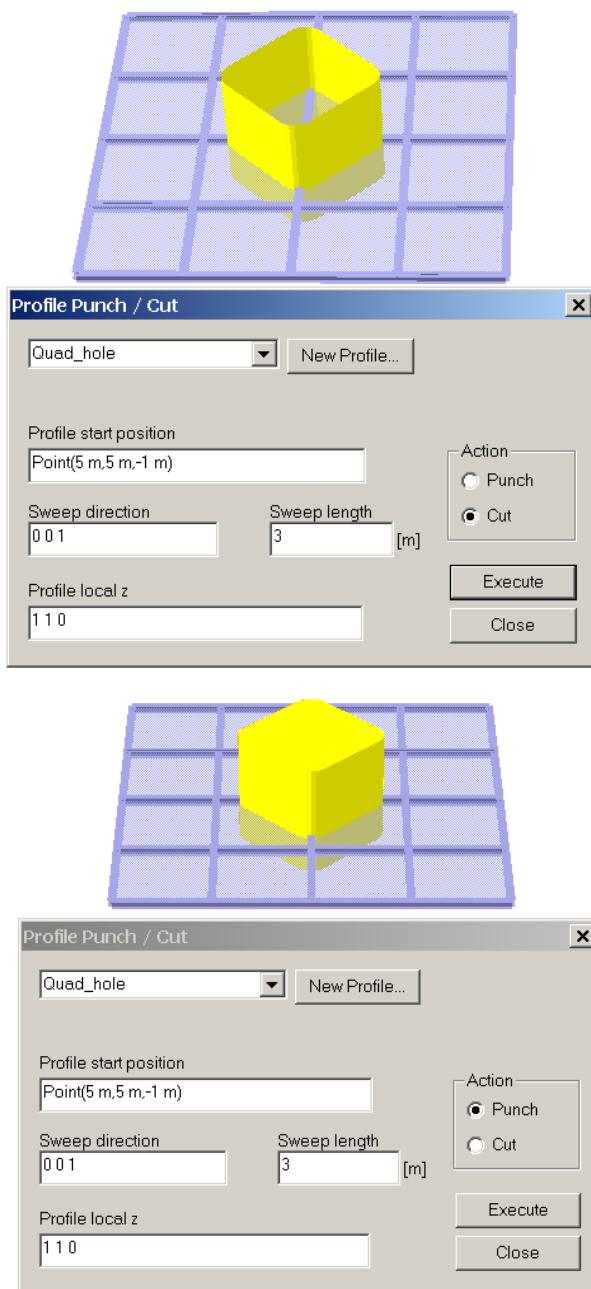
2D Profiles are entities, which are used by the cut and punch operations. The profiles can be created by selecting menu entry: **Guiding Geometry/Advanced/Profile from NURBS**. Alternatively, one may create a rectangular profile curve (specialized in ship applications) from **Tools/Structure/Punch** and clicking the button “**New Profile...**”. Then the following dialog shows up.



In the first tab the user has the possibility to create a rounded rectangle while in the second he may only see the 2D NURBS curves which have been converted to profiles. In the above example, a profile named Quad\_hole has been created. Observe that the y and z-axis refer to a local co-ordinate system. The centre of the local coordinate system is the centre of its bounding box. You may also create a circular profile by entering same values for Height and Width, the radius must be half the Height value.

When performing a cut operation, a cut line defined from the intersection between the 2D profile and the structure. Similarly, when performing a punch operation all material inside the cut line is removed (both for beams and plates).

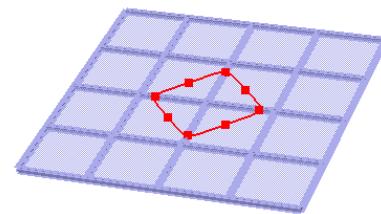
The example below shows the difference between cut and punch, using the created Quad\_hole profile. When the cut line has been created, a model curve or a feature edge may be inserted so that it may be referenced when applying boundary conditions or finite mesh details.



The cut operation is started from the pull down menu *Tools/Structure/Punch*.

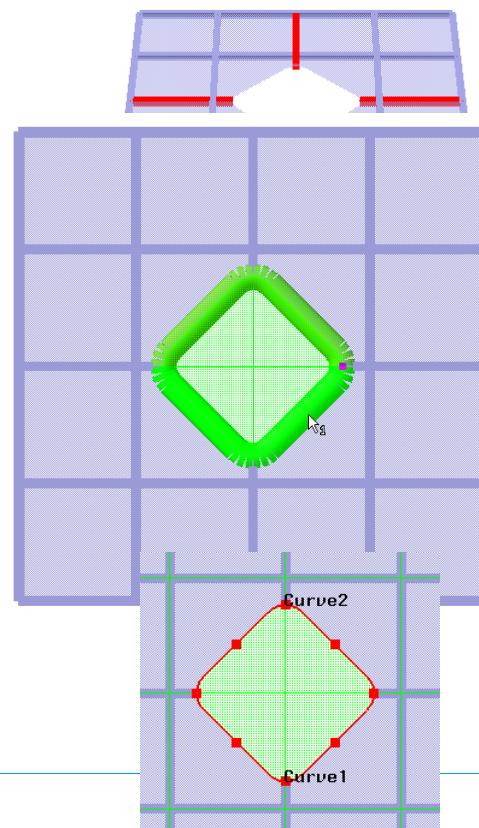
The input parameters will prompt you for where to start, in which direction, this may be in any direction, length of cut operation, and orientation of profile local z-axis in the global co-ordinate system.

The input parameters to the left result in the following cut line on the model:

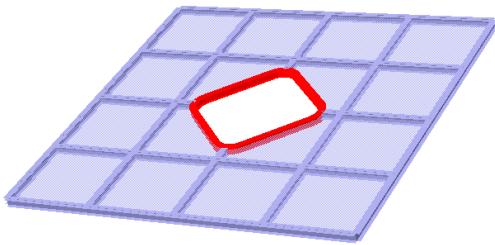


The punch operation is done in the same manner, but now *Punch* has been specified.

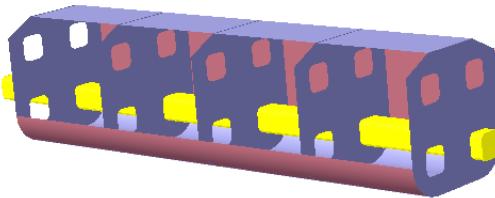
All material inside the volume defined by the profile and the height is removed. The plate now has a hole, and the beams are split in minor beams.



The cut line can now be used to e.g. insert beams. The first step is to create a model curve and make sure the view specified shows guiding geometry. When clicking between two points the program prompts for which part to create a model curve – click on the part to perform operation (GeniE cannot create a closed model curve).



The second step is to add beams to the model curves Curve1 and Curve2. Select the curves, RMB and *Add Beams*.

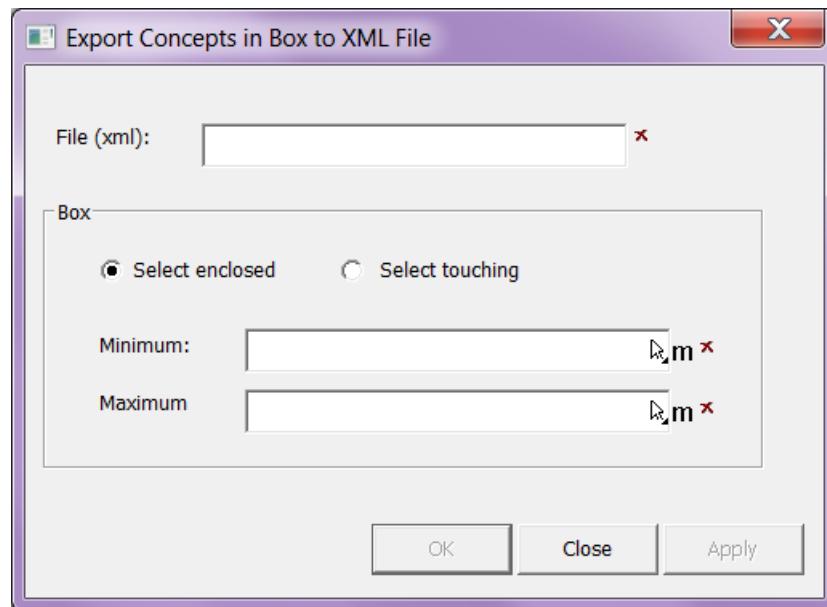


Cut and punch may be used to create cut lines (and punch) through several planes. This example shows how punch has been used to create the same hole in all planes.

#### Export Entities in Box to XML

**Explanation:** The function saves in the given xml file (under the workspace folder) all Genie entities which are enclosed or touching the specified box. The box is determined by the minimum and maximum points.

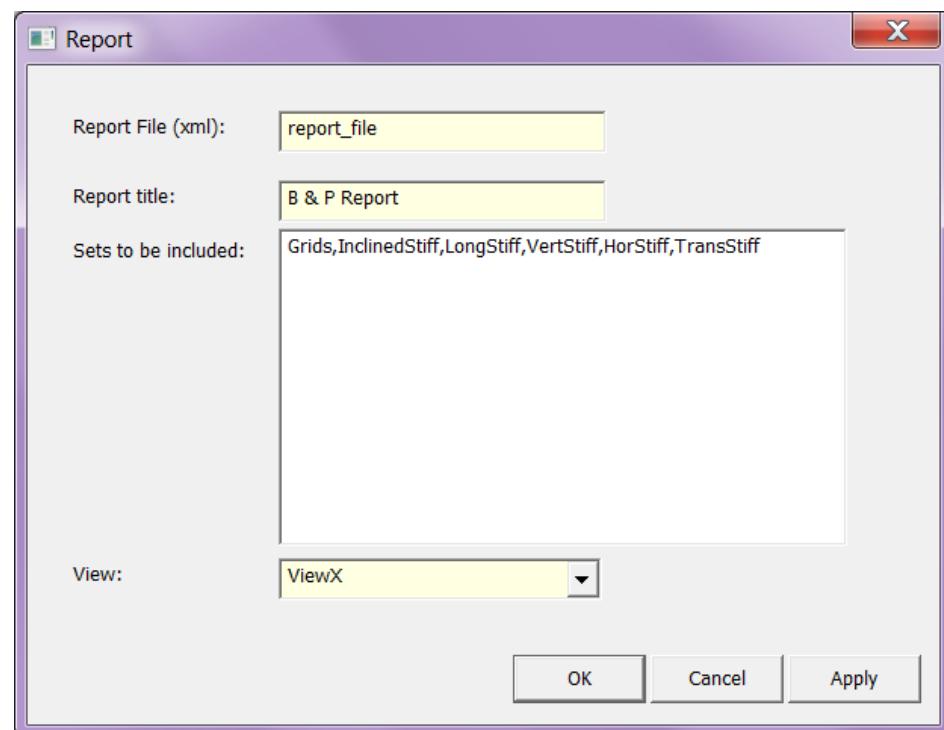
**Graphically:** The following dialog opens:



## Report on Plates and Beams

**Explanation:** The function creates a report of the properties of the plates and beams in xml format. The user may select the sets to be included and a view for the graphical output.

**Graphically:** The following dialog opens



**JS-command:** Initially Genie creates the report:

```
tmpReport = Report("B & P Report");
```

Then, for each Set, Genie searches for all objects of type Beam and Plate. After this, it adds their properties as labels. Genie, finally, projects them to the selected view and produces an image, which is added to the report. In the end it saves the report:

```
tmpReport.saveAs("report.xml",mrWordXML);
```

### 3.4.7 Holes

#### 3.4.7.1 *Create Hole from Guiding Curve*

Create Hole from Guiding Curve is available from the Top Dropdown Menu:

Structure | Features and Holes | Create Hole from Guiding Curve

#### Graphically:

Select Create Hole from Guiding Curve and then click on a guiding curve. The selected guiding curve has to be closed, planar, and to lie entirely on the assembly.

#### Explanation:

The selected guiding curve is imprinted onto the assembly. The imprinted edges have to form a closed loop on the assembly and the hole-concept contains the faces of the underlying plates. These faces need to cover the entire area inside the imprinted edges.

Density can be assigned to feature holes.

Currently beams crossing or connected to the hole are not supported.

#### JS-command:

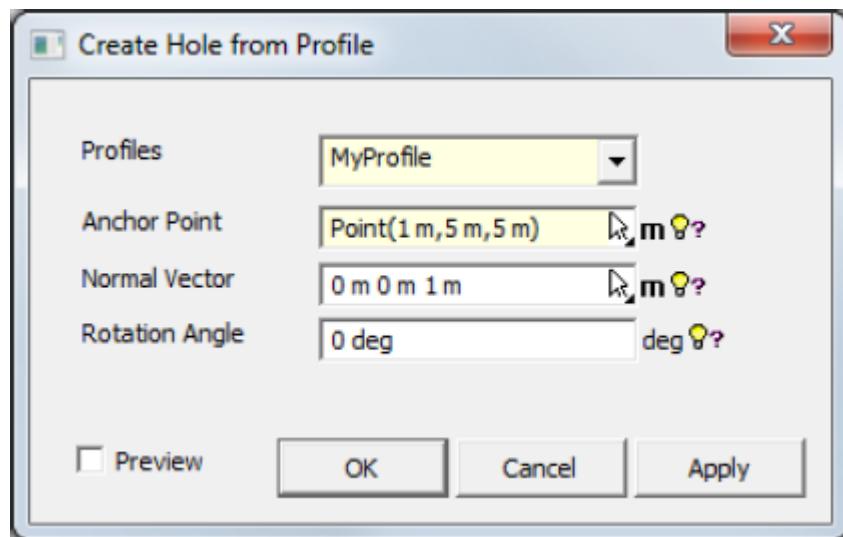
```
Hole1 = FeatureHole(Curve1);
```

### 3.4.7.2 Create Hole from Profile

Create Hole from Profile is available from the Top Dropdown Menu:

Structure | Features and Holes | Create Hole from Profile

**Graphically:**



In the dialog, select a Profile.

**Explanation:**

*Anchor Point:* The point of the model where the profile-center will be placed. The profile center is the center of gravity for the control polygon.

*Normal Vector:* The normal vector at the anchor-point. It takes automatically its value from the model. In case there are more than one plate at it, the user may manually set the correct one.

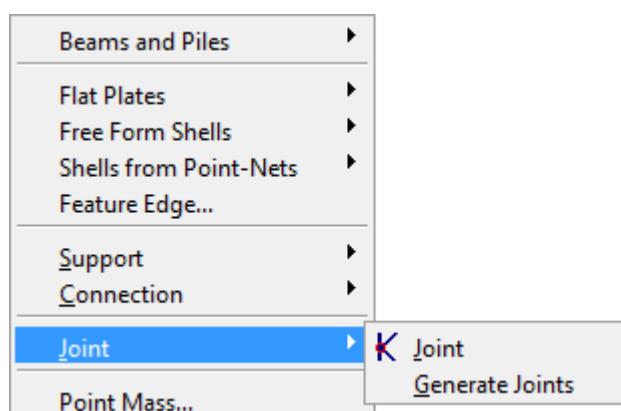
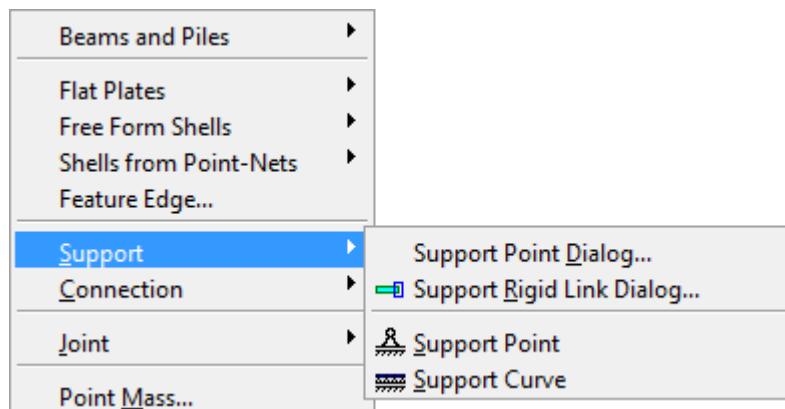
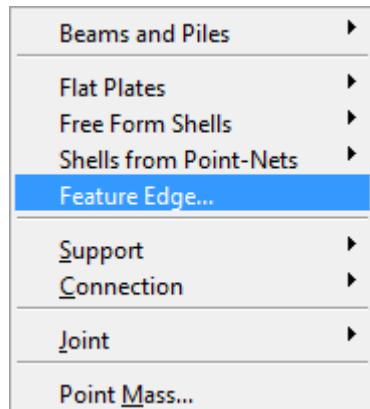
*Rotation Angle:* The angle of rotation (in degrees) of the profile around the normal vector.

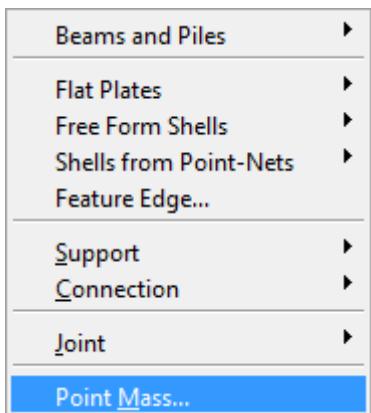
**JS-command:**

```
Hole1 = CreateHoleFromProfile(MyProfile, Point(1 m, 5 m, 5 m), Vector3d(0 m,0 m,1 m), 0 deg);
```

## Other Structural Entities

The following menu items are not new, they have just been moved . Explanations on all of these can be found otherwhere in the documentation.





### **3.4.8 Reference documents on Computer-Aided Geometric Design:**

Web-course by K.C. Shene:

<http://www.cs.mtu.edu/~shene/COURSES/cs3621/NOTES/>

References on Computer-Aided Geometric Design, provided by C. deBoor:

<http://pages.cs.wisc.edu/~deboor/bib/bib.html>

Lecture Notes by T. Sederberg:

<http://tom.cs.byu.edu/~557/text/cagd.pdf>

<http://syllabus.byu.edu/view/SnfjQFxAdIyY.html#scheduleContainer>

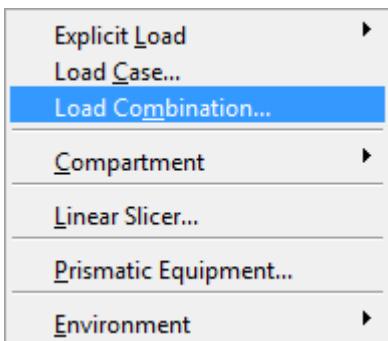
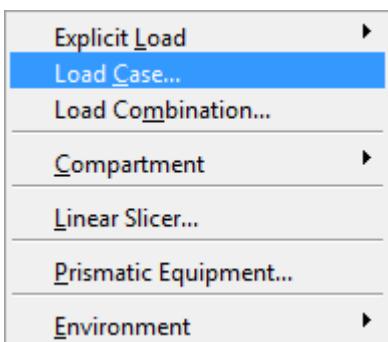
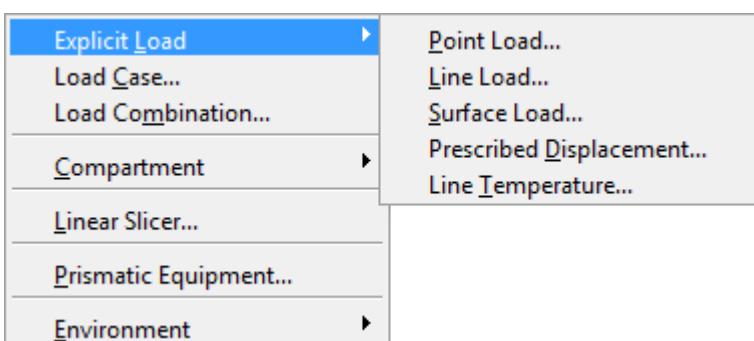
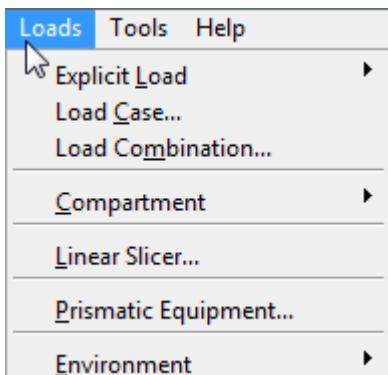
Lecture Notes by T. Lyche and K. Mørken:

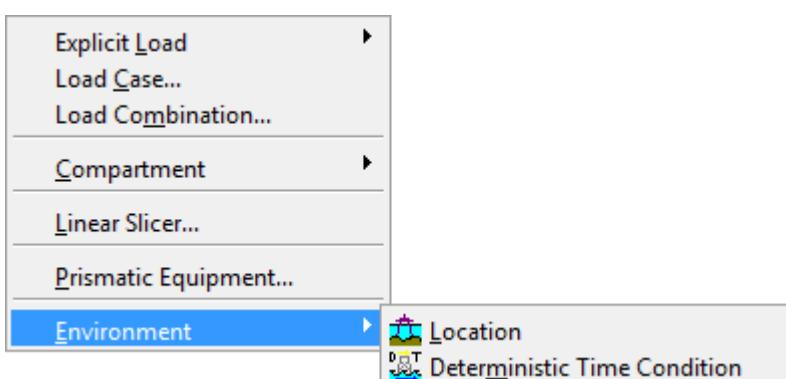
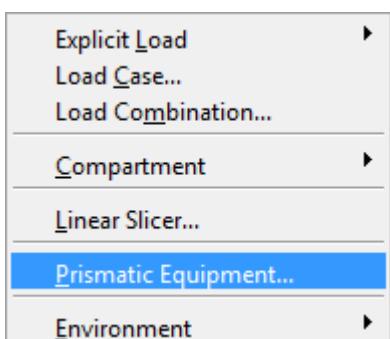
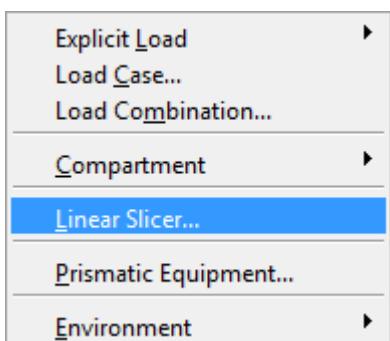
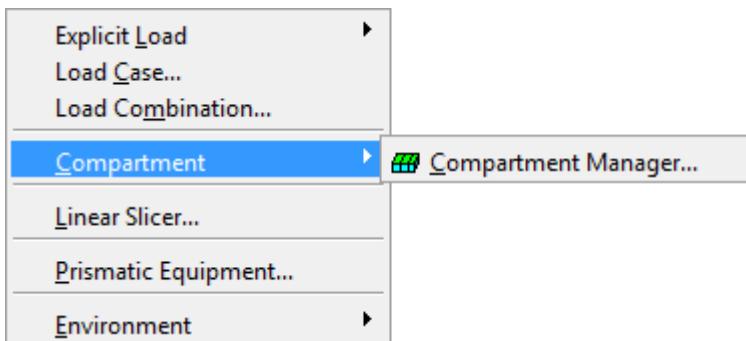
<http://www.uio.no/studier/emner/matnat/ifi/INF-MAT5340/v10/undervisningsmateriale/book.pdf>

Classic text-books in the area of Computer Aided Geometric Design:

- L. Piegl and W. Tiller, “*The NURBS Book*”, Springer Publisher, Berlin-Heidelberg-New York, 1997.
- G. Farin, “*Curves and Surfaces for Computer Aided Geometric Design - A Practical Guide*”, Academic Press, New York, 1997.
- G. Farin, “*NURBS for Curve and Surface Design*”, AK Peters Publisher, Boston, 1999.’
- G. Farin and D. Hansford, “*The Geometry Toolbox*”, A.K. Peters Ltd, 1998.
- I. Stroud, “*Boundary Representation Modelling Techniques*”, Springer-Verlag London Ltd, 2006.

## 3.5 Loads, new menu



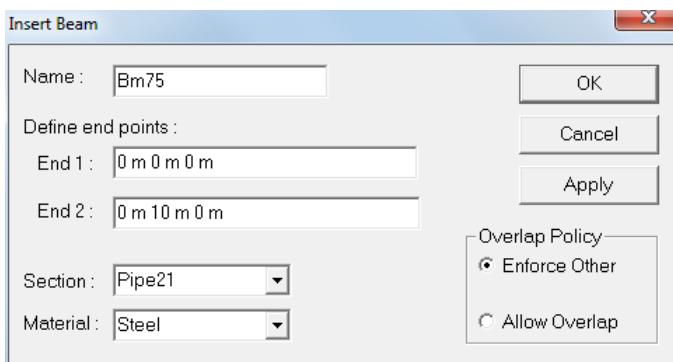


## 3.6 Basic structure modelling of beams

Basic structure modelling of beams covers regular modelling and associated tasks (like for example edit, change, delete) of straight beams. Prior to modelling structure it is assumed that libraries for sections, materials, and plate thicknesses have been defined. When modelling, the default settings are automatically applied to all new structures created. The defaults may be changed at any time, and it is also easy to change properties associated with a particular structural member.

### 3.6.1 Regular Straight Beam

There are two ways of inserting beams. The first option assumes explicitly modelling a beam, while the second option assumes graphic modelling by clicking snap points to define the beam start and end positions.

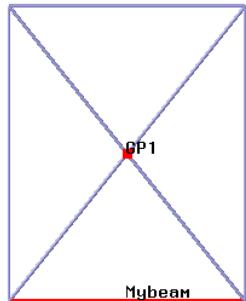


The pulldown menu **Insert/Beam/Straight Beam Dialog** opens up the dialog shown to the left. Here you may specify manually all required input, but you may also refer to existing snap points. You do this by clicking in e.g. input field for end 1 and then click graphically on the desired snap point. Relevant information is automatically filled in.

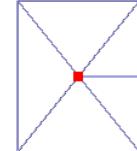
#### Overlap Policy

**Enforce Other:** If the new beam is overlapping an existing beam, the new beam will be cropped.

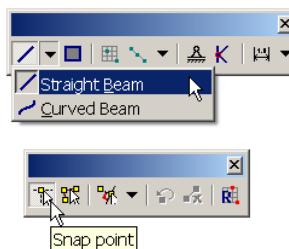
**Allow Overlap:** If the new beam is overlapping an existing beam, it will not be cropped. The existing beam and the new beam will overlap.



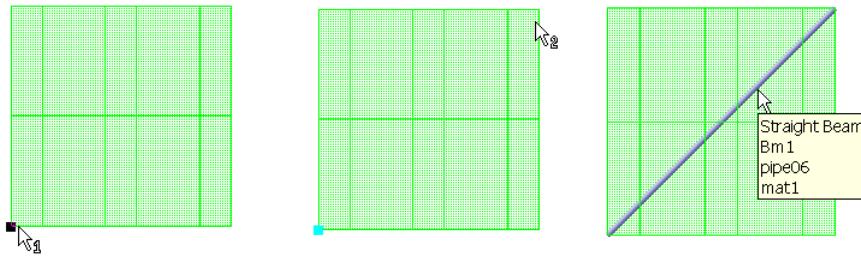
The figure at left shows Mybeam that was manually created together with 5 other beams. The outer beams and the X-beams have been inserted using existing snap points. Note that where the X-beams intersect, a new snap point is automatically created. In this case a guide point has been inserted to the same point. This point can now be used for connecting other members.



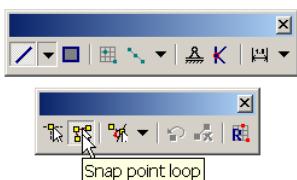
Provided you have snap points available, either from guiding geometry or existing structure, the most efficient way of defining beams is by graphic modelling. You enter graphic modelling of beams either from pulldown menu **Insert/Beam/Straight Beam** or from the toolbar **Object Types**.



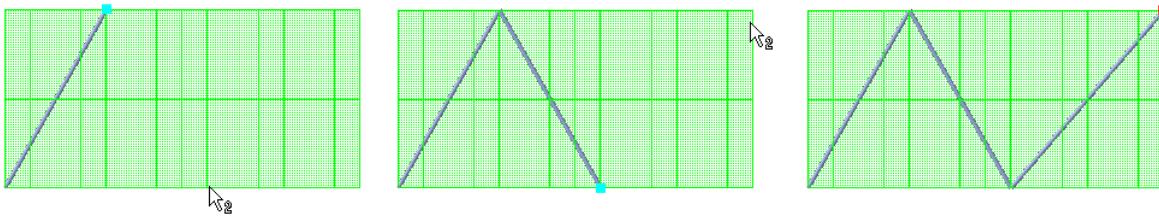
Make sure that the Straight Beam option is the current selection and that the Snap point button is activated. You may now start to model straight beams graphically.



The above pictures show the sequence when inserting a beam between two snap points. When moving the mouse over the graphic window the cursor indicates that the insert beam command is expecting the first input /the program prompts you for which point to be inserted). After positioning the first point, the mouse tooltip now indicates that next position is expected. The beam appears graphically (and in the browser) as seen above.

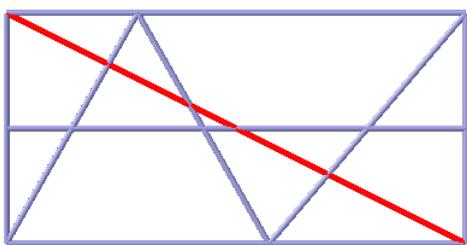


If you have regular modelling you may use a snap to point loop. In this case the end point of a beam is start point of next beam. This means you click end points for the second beam and onwards (or the program will prompt for end points only). The pictures below show an example of this.



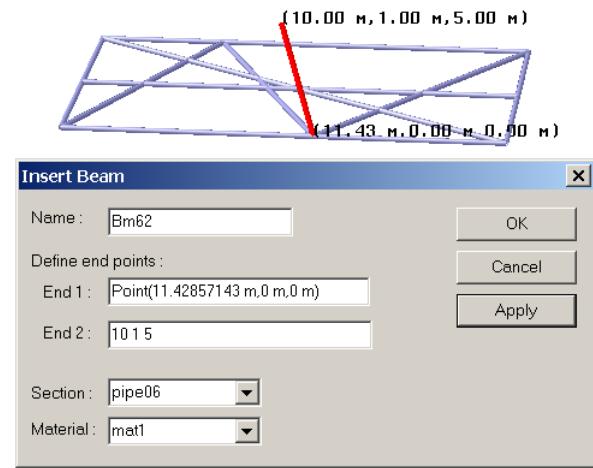
When working with intersecting beams, GeniE will calculate all intersection points and will manage the topology. You can therefore do top-down modelling, i.e. start with the rough and make details afterwards. If not, you may do bottom-up modelling or a combination of both approaches. It is possible to join beams.

The example below shows a number of intersecting beams. Observe that even though a beam intersects with others, it is still one beam (highlighted in red) unless it is divided into smaller parts during a split operation.



A continuous beam intersected by several beams. A topology point (or snap point) is automatically created by the program where beams intersect each other. A finite element node will always be inserted where topology points.

Often there is a need to model a beam extending from a given point or a guideplane to a known position. This is best achieved by using the **Insert/Beam/Straight Beam Dialogue** where you a) find information for end 1 from the graphic window by clicking on the point and b) fill in the known co-ordinate values for end 2.



The beam to the left has been generated using the dialogue for inserting a straight beam by extracting coordinate values for End 1 from the existing structure and specifying co-ordinate values for End 2

When you have a model with beams, plates, guide planes you can use the selection filters explained in Chapter 5.1.2 under the Selection Toolbar to avoid snapping to e.g. plates and guide planes.

### 3.6.2 Overlapping beams

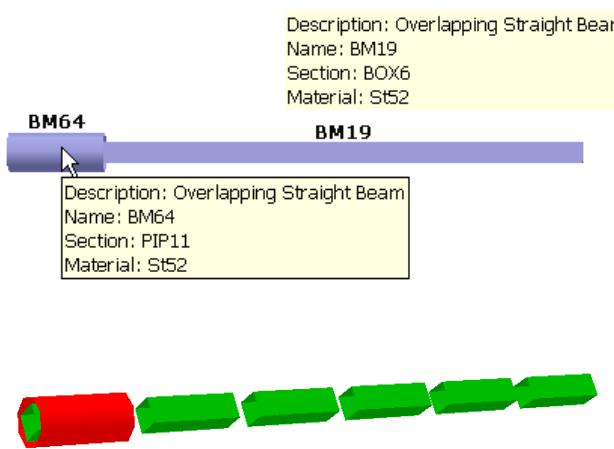
When inserting an overlapping beam you do the following steps:

1. Insert a regular straight beam
2. Switch modelling modus from regular straight beam to straight overlapping beam
3. Insert the overlapping beam



You may have more than two overlapping beams.

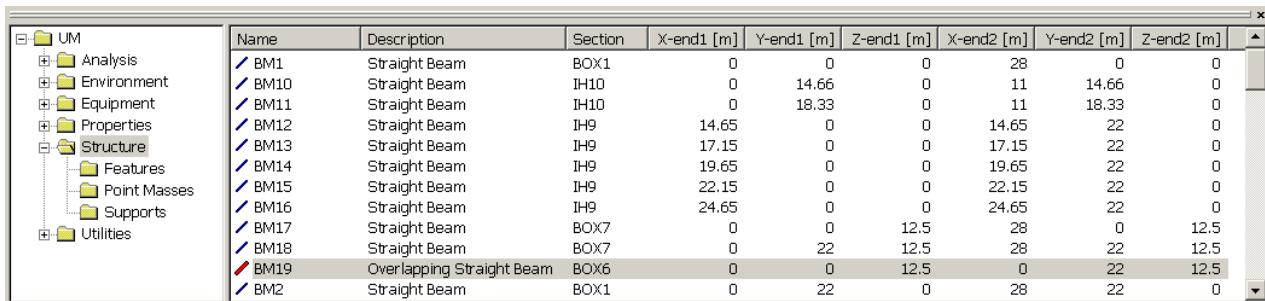
In the example below a beam with a box section is present prior to inserting a beam with pipe section. Furthermore, the mesh density is equal to both the inner and outer beam unless you specify otherwise. See Chapter 3.12.2 on how to make a finite element model.



Notice that the original straight beam (BM19) also becomes an overlapping straight beam when the additional beam (BM64) has been added.

The picture to the left shows the finite element mesh. There are two finite beam elements covering the beam overlapping area. These finite beam elements are connected at the finite element nodes (number of nodes depend on the mesh density).

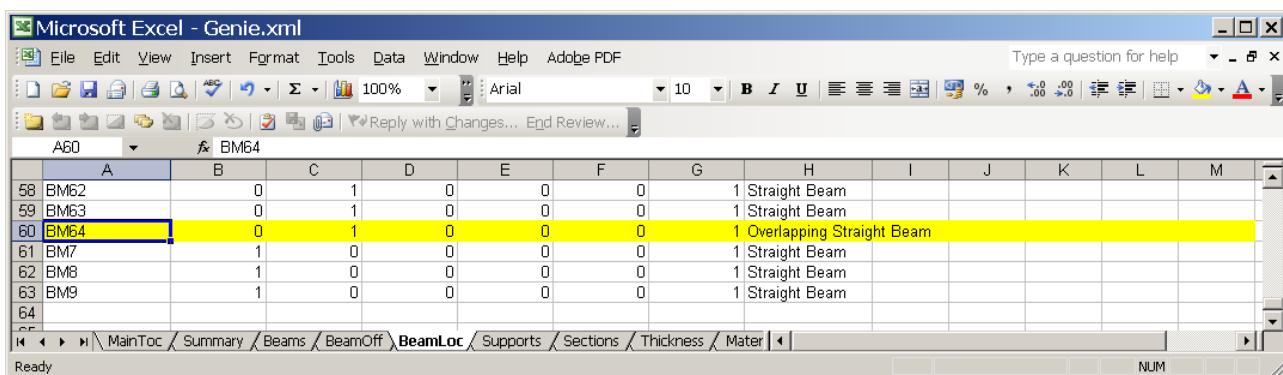
To verify the overlapping beams, you may either use the browser functionality or from the saved report. Both options are showed below.



The screenshot shows the 'Structure' folder expanded in the left-hand tree view. A table to the right lists beam properties:

Name	Description	Section	X-end1 [m]	Y-end1 [m]	Z-end1 [m]	X-end2 [m]	Y-end2 [m]	Z-end2 [m]
BM1	Straight Beam	BOX1	0	0	0	28	0	0
BM10	Straight Beam	IH10	0	14.66	0	11	14.66	0
BM11	Straight Beam	IH10	0	18.33	0	11	18.33	0
BM12	Straight Beam	IH9	14.65	0	0	14.65	22	0
BM13	Straight Beam	IH9	17.15	0	0	17.15	22	0
BM14	Straight Beam	IH9	19.65	0	0	19.65	22	0
BM15	Straight Beam	IH9	22.15	0	0	22.15	22	0
BM16	Straight Beam	IH9	24.65	0	0	24.65	22	0
BM17	Straight Beam	BOX7	0	0	12.5	28	0	12.5
BM18	Straight Beam	BOX7	0	22	12.5	28	22	12.5
BM19	Overlapping Straight Beam	BOX6	0	0	12.5	0	22	12.5
BM2	Straight Beam	BOX1	0	22	0	28	22	0

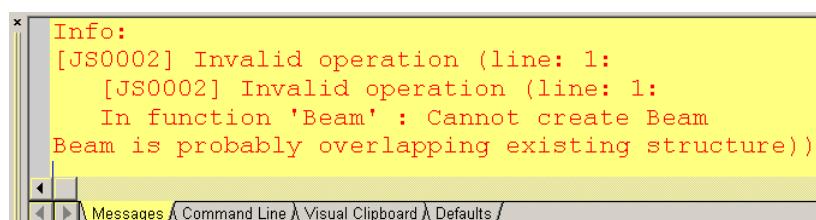
Notice to see the additional fields like coordinate values, you need to add the relevant fields to the view by selecting the folder *Structure*, push RMB and select *Fields*.



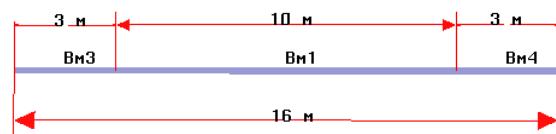
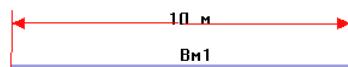
The screenshot shows an Excel spreadsheet with data in columns A through M. Row 60 contains the highlighted cell BM64, which corresponds to the beam BM4 in the table above. The table includes columns for beam ID, X, Y, Z coordinates, and a description column.

A	B	C	D	E	F	G	H	I	J	K	L	M
58	BM62	0	1	0	0	0	1	Straight Beam				
59	BM63	0	1	0	0	0	1	Straight Beam				
60	BM64	0	1	0	0	0	1	Overlapping Straight Beam				
61	BM7	1	0	0	0	0	1	Straight Beam				
62	BM8	1	0	0	0	0	1	Straight Beam				
63	BM9	1	0	0	0	0	1	Straight Beam				
64												

GeniE will not accept overlapping beams unless you have changed modelling modus. If you accidentally insert a beam over an existing beam, GeniE will inform you that the insert operation is not performed. You will get a warning in the command line interface window: Cannot create Beam, Beam is probably overlapping existing structure. The same message will appear in the Message area, see below:

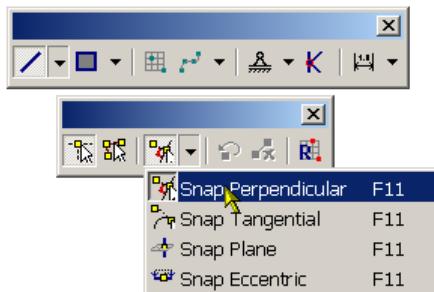


If you insert a beam that partially covers an existing beam, the delta part is inserted as new beams. The example below shows that Bm2 is inserted over Bm1 (Bm2 is 6 meters longer than Bm1). As can be seen, two new beams (Bm3 and Bm4) are created at each end of Bm1.



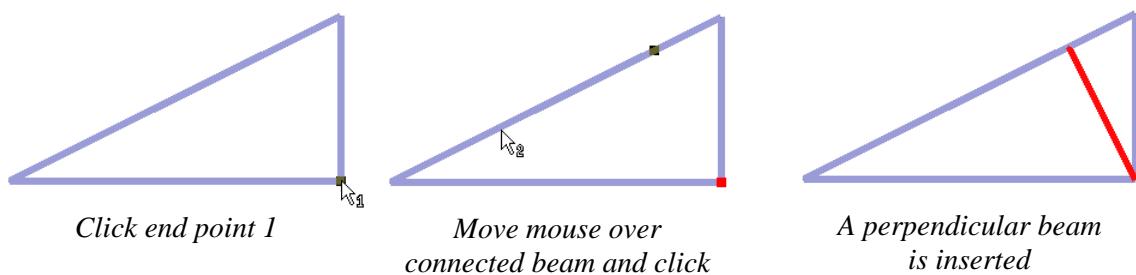
### 3.6.3 Modelling perpendicular beams

GeniE comes with a feature that allows for efficient modelling of beams perpendicular to another beam.

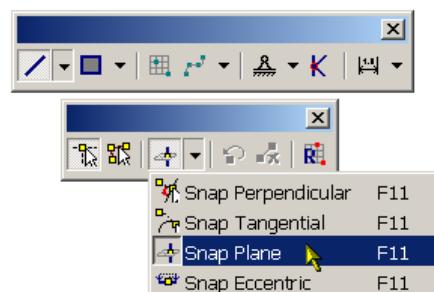


Make sure the Snap Perpendicular button is activated. You insert a perpendicular beam to another beam as follows:

1. Click a snap point
2. Move the mouse over the beam you want to connect to and click
3. The beam is now inserted always perpendicular to the connected beam

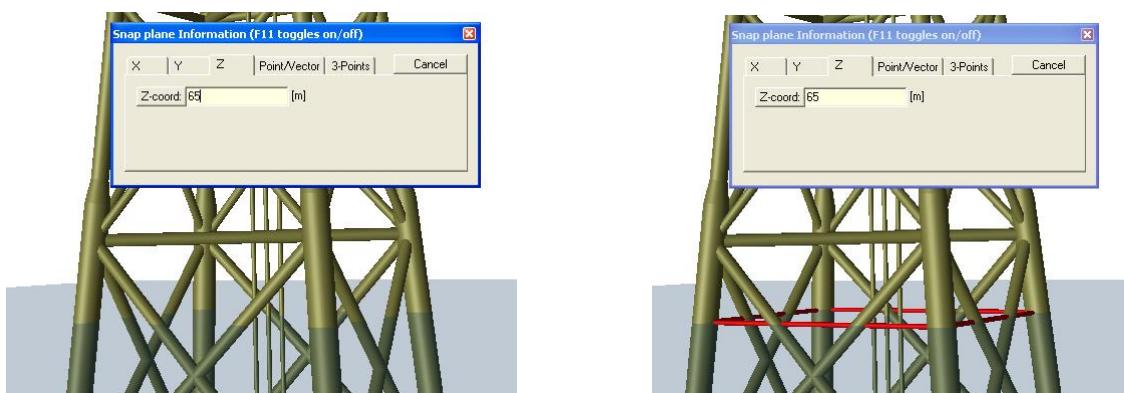


### 3.6.4 Snap planes



Often it may be convenient to connect beams to existing structure in a given plane. A typical example may be that you have modelled inclined jacket legs and you want to insert horizontal beams between the legs at given planes. Make sure that the Snap Plane button is activated. Please also note that the still water level (see Vol. 2 for details on how to define) also acts similar to a snap plane. An alternative method is to use *Insert/Split/Points*.

The example below shows how a snap plane can be used to insert horizontal beams on a jacket. To insert horizontal beams at z-level 65, you click the Snap Plane button and you are prompted to insert values for a plane. The plane set up now is a temporary guideplane, and where it intersects with existing beams temporary snap points are established. You then insert beams as before, but the snap plane must be present during modelling. The selected beams (highlighted in red) are all created using this method.

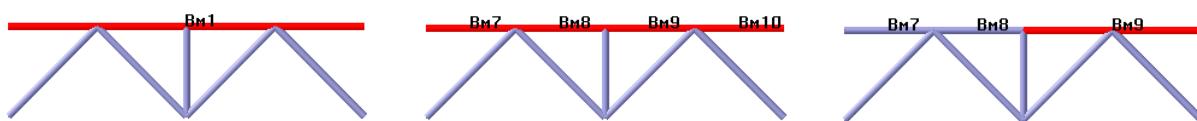


### 3.6.5 Split, join and divide beams

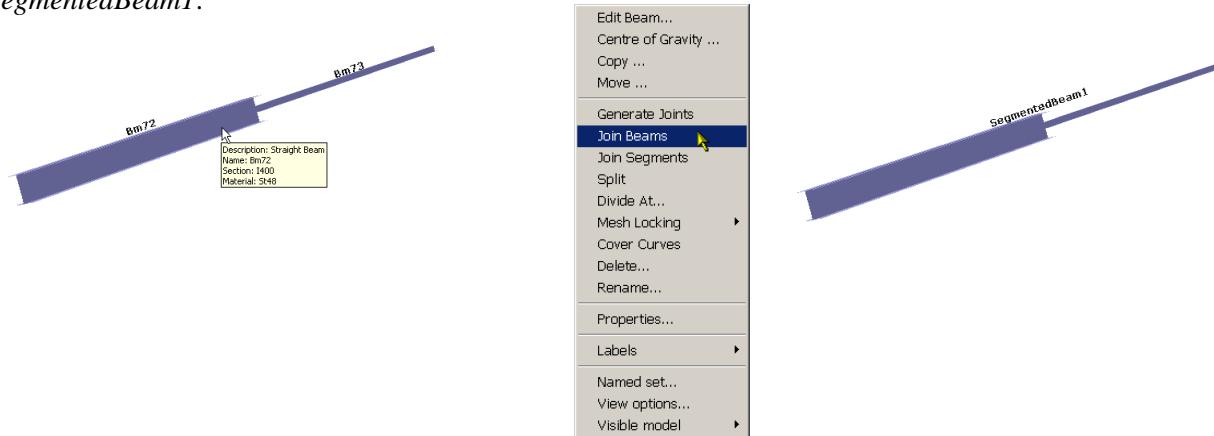
A continuous beam may be intersected by several beams. There may be reasons why you want the beams between intersections to be individual beams. To split such beam, you select the beam and use the command **Tools/Structure Split** from the pulldown menu. Alternatively you may do this from the context sensitive menu and choose *Split*. Similarly if you want to join beams along a line, you select the beams, force the context sensitive menu, and choose *Join Beams*. When the joined beams have different section or material properties, a segmented beam will be created.

These features open up for a flexible modelling strategy, either you want to work with top-down or bottom-up modelling approach or a combination of both approaches.

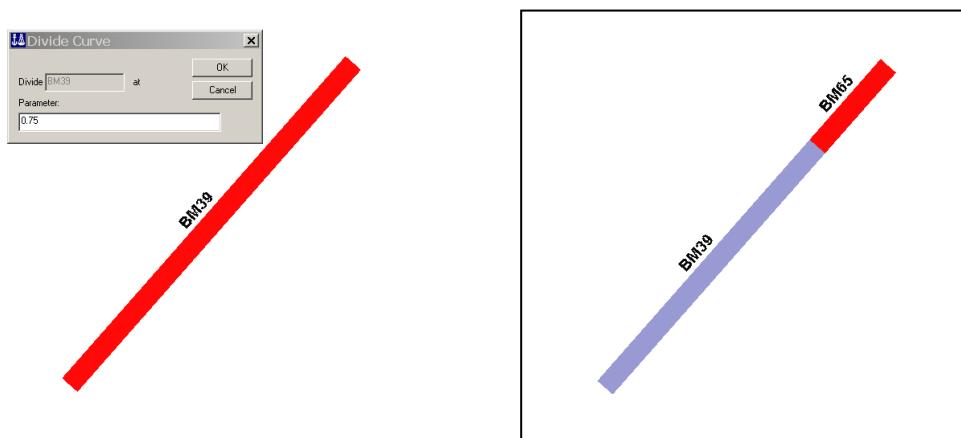
The example below shows the beam Bm1 split into minor beams (Bm7-Bm10), before Bm9 and Bm10 are joined.



If you joint two beams with different section (or material) properties, the joined beam automatically becomes a segmented beam. In the example below the two beams *Bm72* and *Bm73* are joined to *SegmentedBeam1*.

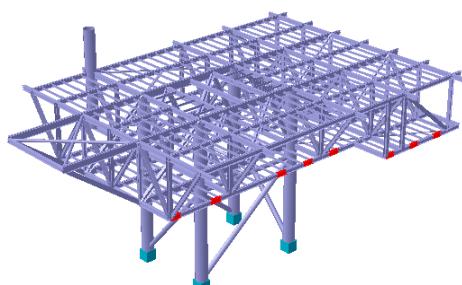


You may also split a beam at a known position by using the *Divide At* feature. The split position is given either as a length factor referring to the start point of the beam or by clicking an existing topology point. In the example below the beam is split  $\frac{3}{4}$  of the length from end 1 and BM65 is created.

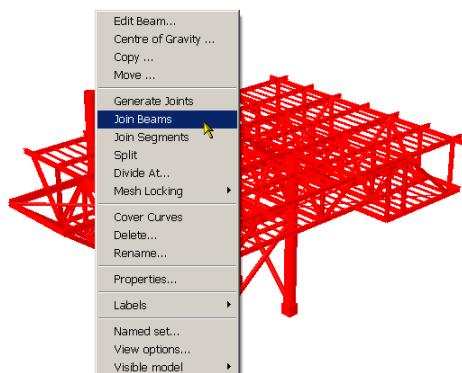


When joining beams GeniE will use the tolerant settings as specified under **Edit/Rules/Tolerances**. Furthermore, it will look at those beams along the same line and create a beam (or segmented beam when differences in section or material properties) if tolerance criteria is met.

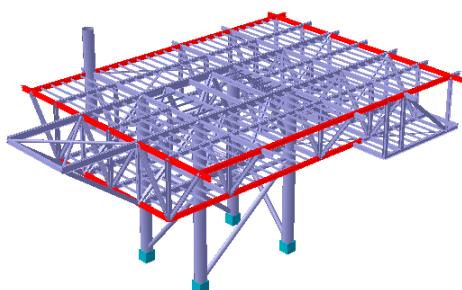
This means you can select many beams at the same time, and GeniE will automatically detect which beams that can be joined. This may be beneficial if you have used bottom up modelling or have imported a model from a FEM or CAD system – these are normally not based on concept modelling but geometry or finite element modelling and has a much smaller discretisations than necessary. GeniE works with double number precision and when importing data from other systems where single number precision have been used it may be necessary to use a high angular tolerance when joining the beams.



Imported model from FEM format (created by previous SESAM modeller Preframe). When importing, each finite element is translated into a concept. Some of these concepts are highlighted to show the density of the model.



All beams have been selected and **Join Beams** is activated.



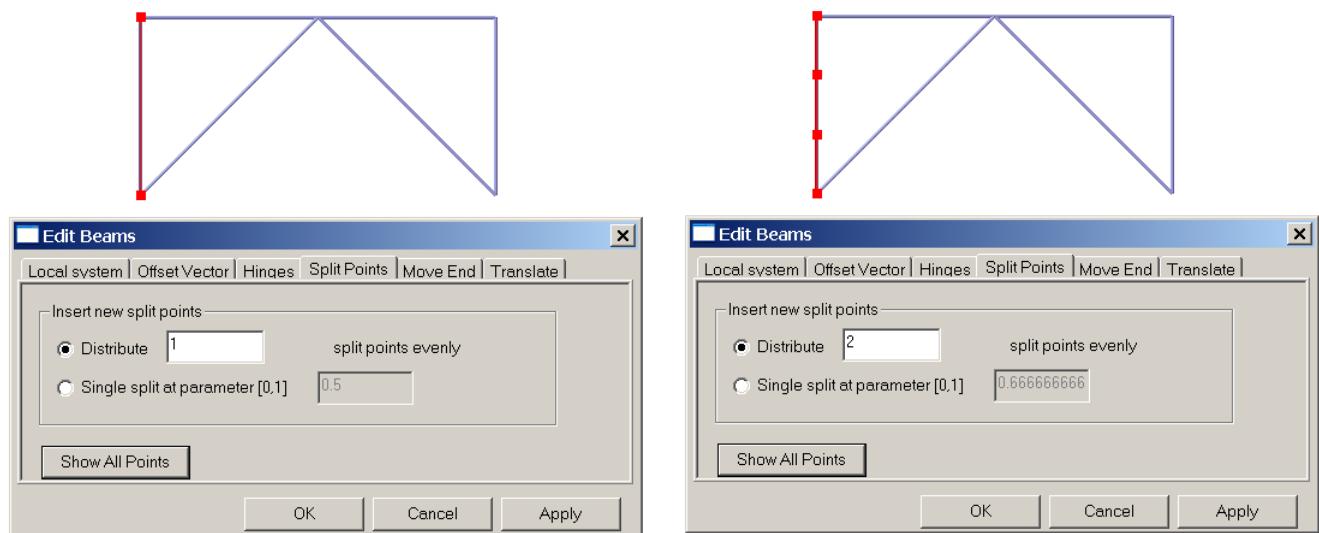
All beams along a line and satisfying the tolerance criteria are joined and a new concept model is established. As can be seen the small concepts along one of the deck edges are joined into one beam only.

For this particular example, number of beams (or beam concepts) was reduced from 959 to 256.

You should also be aware that join and split beams do not account for eccentricities assigned to beams. This means you need to re-assign eccentricities to the generated beams.

### 3.6.6 Insert split points

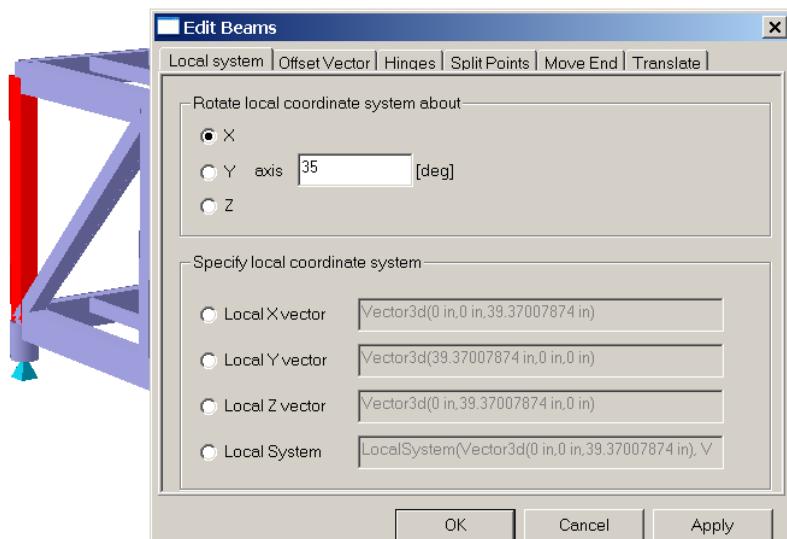
When creating a beam it has snap points (or topology points) at its ends or where it is intersecting with other beams. You may insert snap points in other positions by using the feature *Edit Beam/Insert Split Points* found on the context sensitive menu. In the example below, split points are inserted one beam to create additional snap points.



Snap points (or topology points) that are not used may be removed by using the command **Tools/Structure/Geometry/Simplify Topology** from the pulldown menu. A finite element node will be inserted at each topology point, hence a more clean finite element mesh can be achieved by using this feature.

### 3.6.7 Beam local co-ordinate system

The local x-axis vector is from first to second modelling point of the beam, while the local y-axis will sweep in the global XY-plane until the beam is vertical (then local y-axis is along global X-axis). For a plate the right hand rule applies, and the local coordinate system is defined according to how the plate was defined.

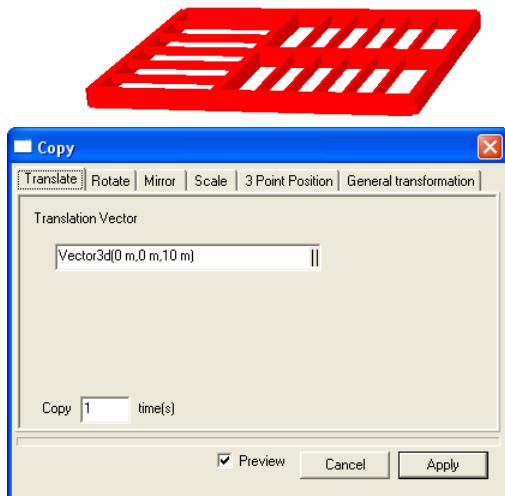


The example to the left shows how to rotate the local coordinate system 35 degrees around local x axis.

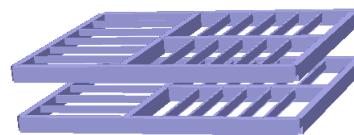
The command is available from the context sensitive menu, select Edit Beams and then Local System.

### 3.6.8 Delete, move and copy beams

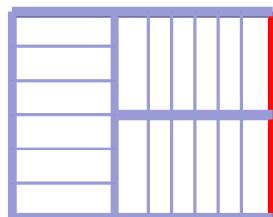
A beam may be deleted, moved or copied. The easiest way is by forcing the context sensitive menu and choose the relevant options. When a beam has been deleted you may re-use the name for a new beam.



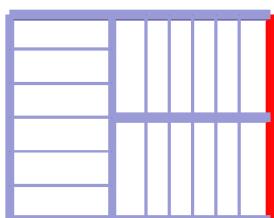
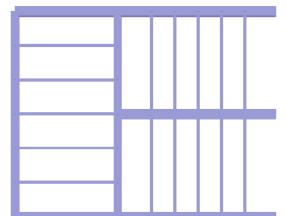
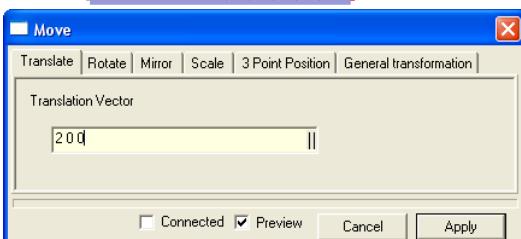
There are six options when performing a copy operation. They are translate (which is the most common), rotate, mirror, scale, 3 point positioning and general transformation. In this example a complete grid has been selected for a copy translate operation with vector 5 meters in vertical z-direction.



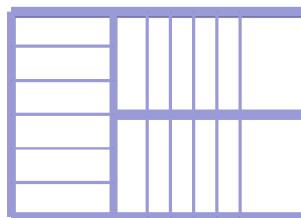
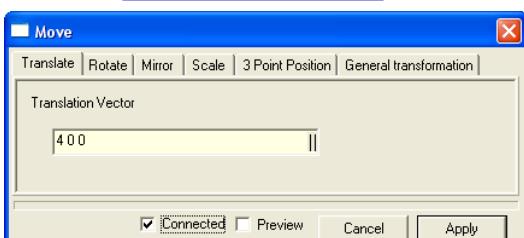
When using the feature for Move, you may move an entire beam or the end(s) of the beam. Both methods are explained below.

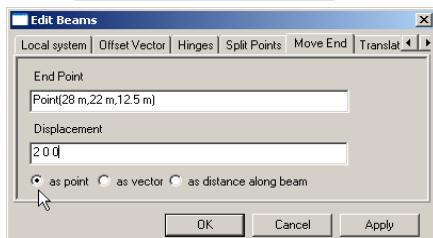
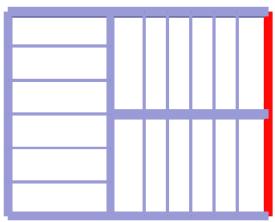


The move command has been activated by forcing the context sensitive menu and using *Move*. The translation vector 2 meters in x-direction will move the actual beam and de-connect it from the rest. Moving back again will automatically connect the beam to the rest of the model.

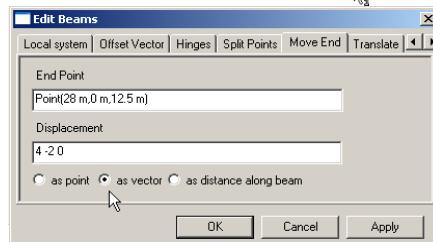
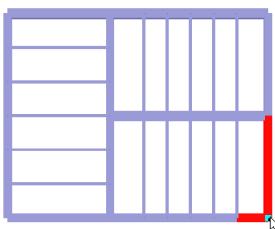
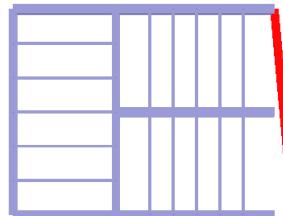


In this example the beam is moved 4 meters in x-direction. Observe that connected move option has been activated. All the connected beams will move accordingly (they are highlighted in red).

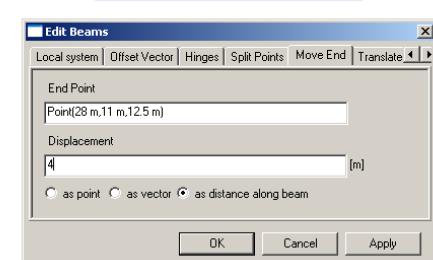
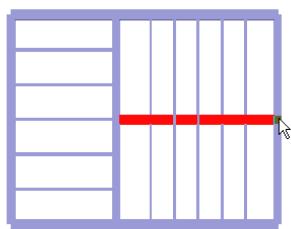
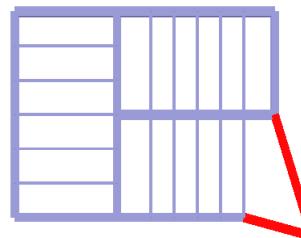




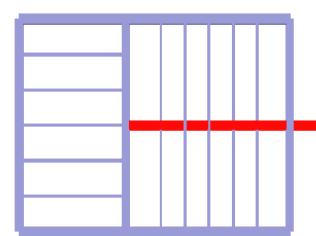
The context sensitive menu has been forced from where you select *Edit Beams/Move End*. In this example the end point at bottom has been selected to move 2 meters in x-direction (you may also move it to a given point). The highlighted beam is now connected only at top.



In this example common point of two beams is selected (note that the beams have been split compared to the previous examples). Moving this point a vector 4 meters in x-direction and -2 meters in y-direction will move the two connected beams as shown below.

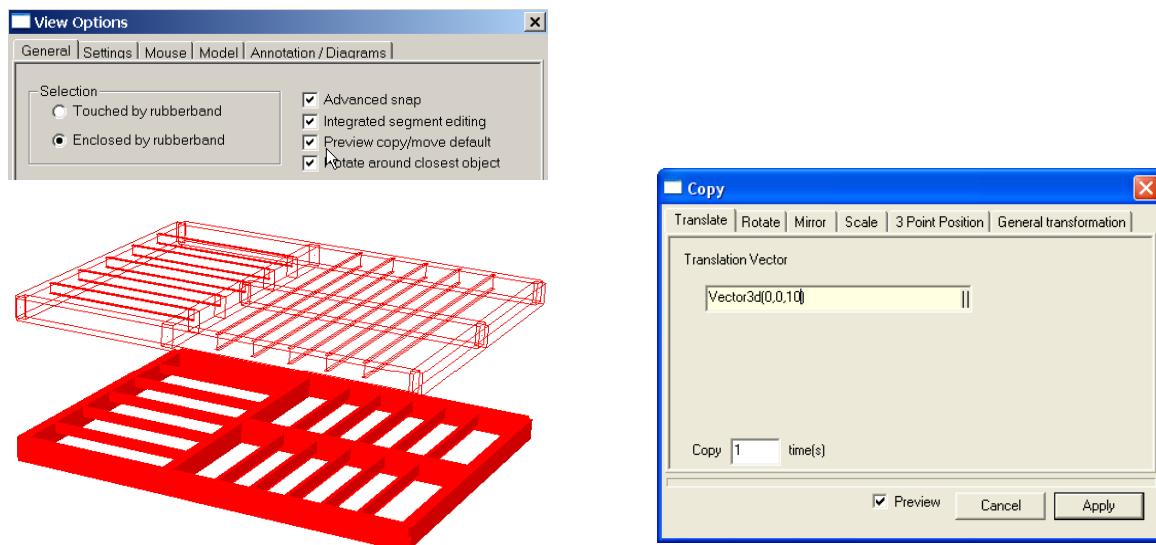


The example to the left shows how to extend (or shorten) a beam along the beam local x-axis. By giving a positive value the beam is extended; a negative value will shorten the length.



When activating the preview option it is possible to get a graphical preview of the actual move or copy before the operation is performed. It is thus easy to see if you have specified a wrong vector.

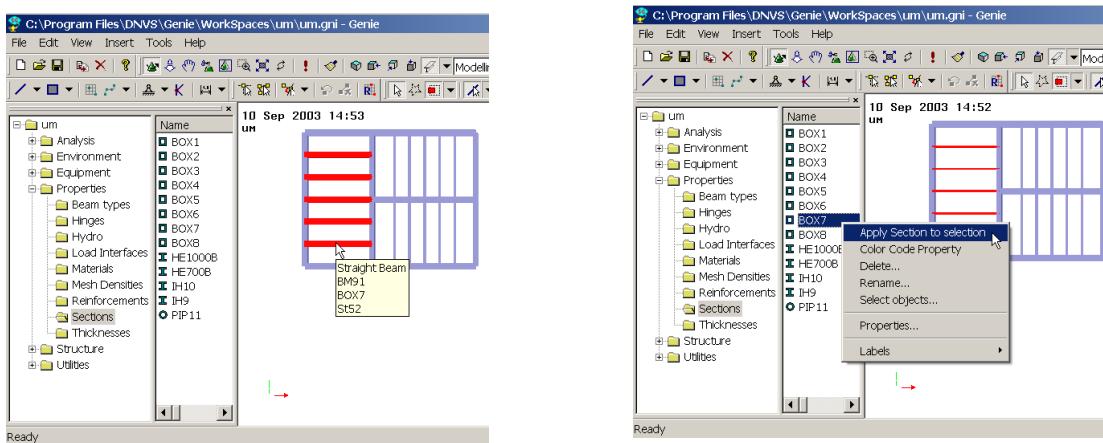
You may set preview to be the default option from the **View/Option/General** or tick off for each time you make a copy/move operation.



### 3.6.9 Change beam properties

Properties connected to a beam may be changed in several ways. One alternative is to selecting the beam(s), then forcing the context sensitive menu and choose *Properties* and do changes from herein.

You may also select the beam(s) and do the change from the Property browser area. The example below shows a number of beams that have section property IH10 and how these are changed to for example BOX6.



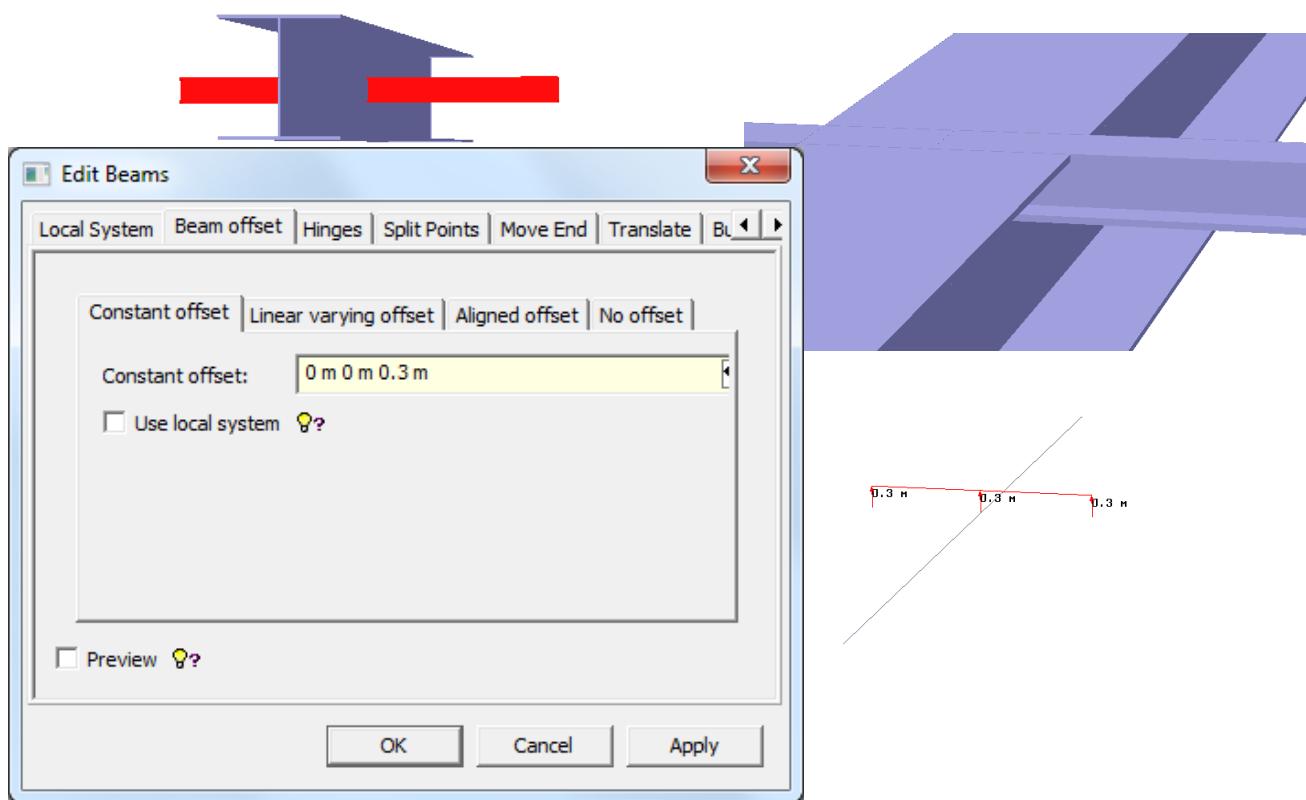
### 3.6.10 Beam eccentricities

Eccentricities in the SESAM system are applied to the beams and not the plates. This means that for a stiffened panel the plate is modelled at top of steel level. Similarly, if you have beam with different section properties and want to align them at e.g. top of steel, it is necessary to apply eccentricities to beam with the least section height.

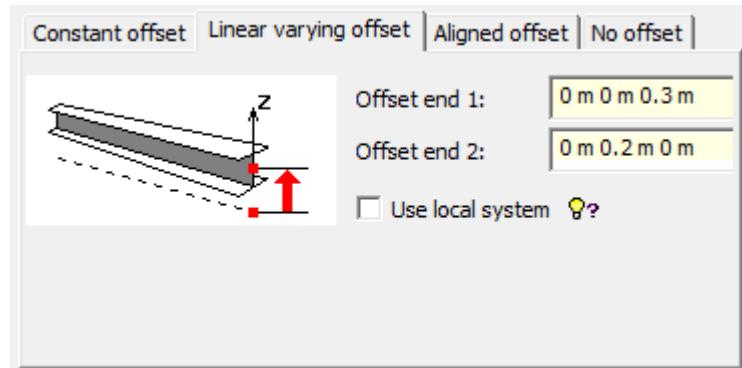
The example below shows two beams with different section profiles, I800 and I200. To align these at top of steel it is necessary to add an eccentricity to the I200 beam that equals  $(1/2 \text{ height of I800}) - (1/2 \text{ height of I200})$ . Using real values this gives an eccentricity  $(0.4 \text{ m}) - (0.1 \text{ m}) = 0.3 \text{ m}$ .

The features for inserting eccentricities are found from the context sensitive menu Edit Beams|Beam offset. You may insert offsets to whole beam or to one of the beam ends.

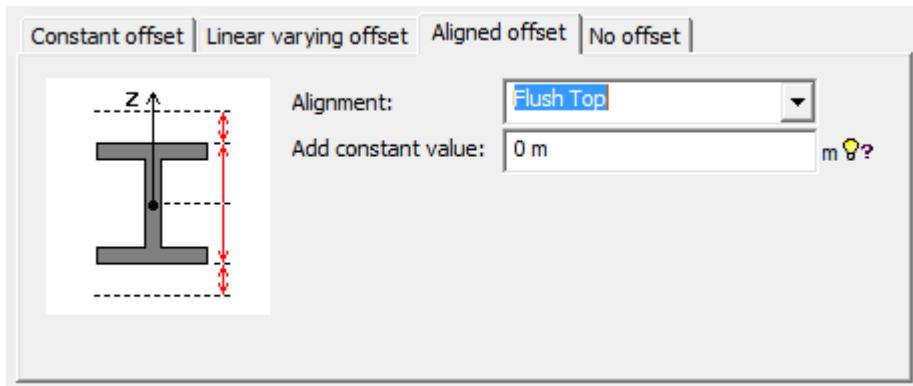
To verify the eccentricities, select the beams, force context sensitive menu, then Labels|Eccentricities (best viewed in wireframe view).



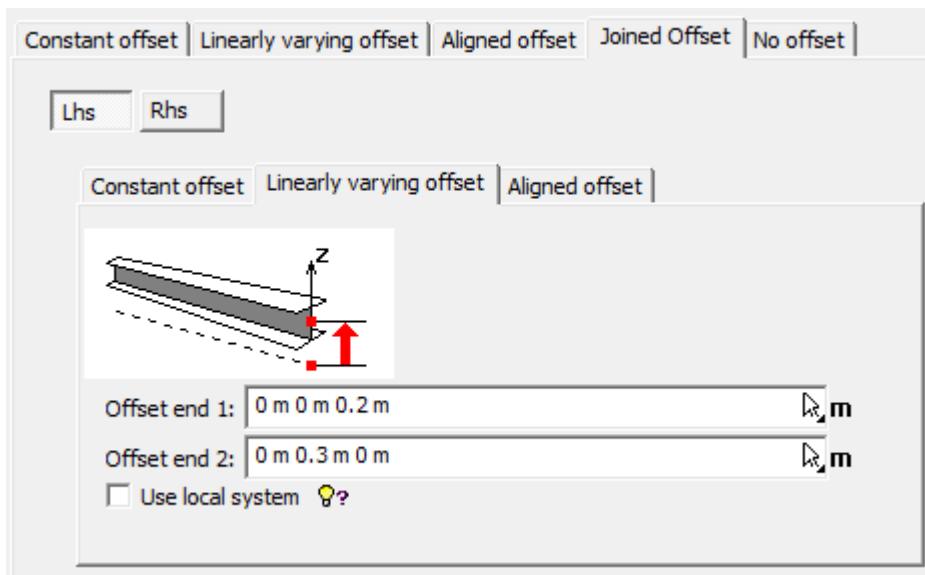
Choose *Linear varying offset* to apply different offsets to the beam's ends.



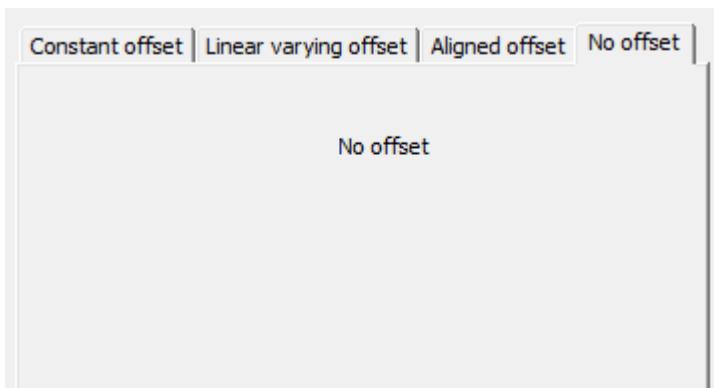
Choose *Aligned offset* to make the beam flush with a plate or with other beams. You then select *Flush Top* or *Flush Bottom*. It's also possible to apply a constant value.



Choose *Joined offset* to combine two of the above mentioned offsets on a beam.



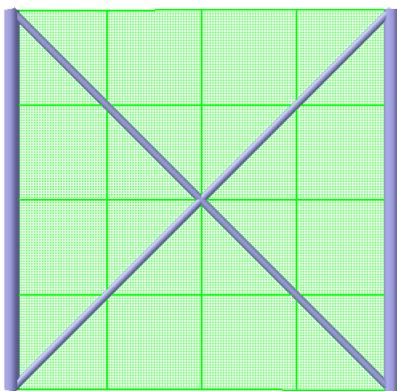
Choose *No offset* if you want to remove all offsets for a beam.



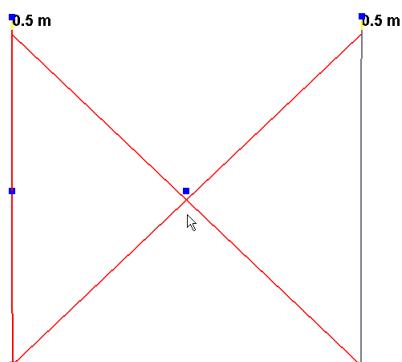
See also stiffened panels for further references on how to label and visualise eccentricities.

### 3.6.11 Connecting beams to an eccentric beam end

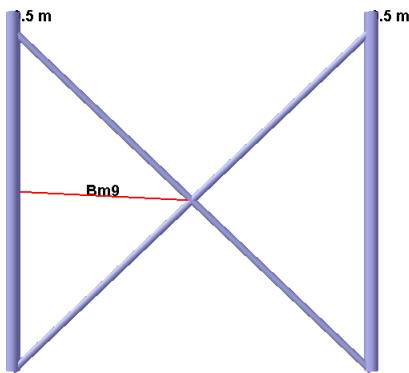
When using the feature for *Snap Eccentric* it is possible to connect a beam to another beam end using its eccentric position. This is typical for an X-brace in a jacket where the beam ends have eccentricities due to e.g. gap calculations and you want to connect a beam to the eccentric position of the X. The example below shows how to do this.



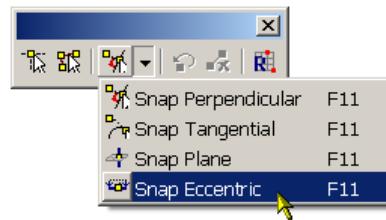
1. A regular X-brace is created. Please note that the vertical beam to the left is split in two



2. The top ends of the inclined beams are given a vertical offset. This leads to an automatic offset of the X-joint itself related to the topology point (the blue dots indicate topology points).



3. When activating the *Snap Eccentric* feature it is now possible to create a beam that will snap to the eccentric position of the X-joint. Choose *Snap Eccentric* as shown and insert a beam by clicking first at the vertical beam and then to the X-joint.

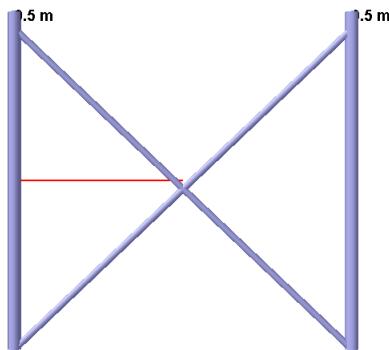


When inserting Bm9 as shown above, this will generate the following journal file (lines 95 and 96 are created by the program, the rest of the lines are edited).

```

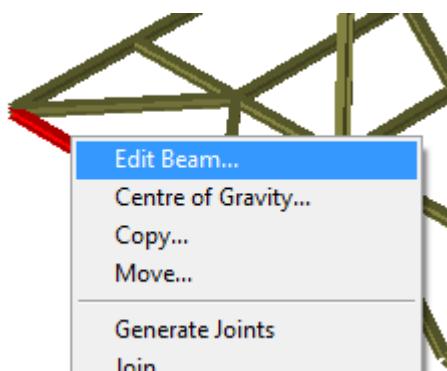
90 //
91 // The following commands are automatically created:
92 // First inserting the beam to the topology point of the X joint
93 // Then inserting the same eccentricities as for the X joint itself
94 //
95 Bm9 = Beam(Point(0 m,5 m,0 m), Point(5 m,5 m,0 m));
96 Bm9.setEndOffset(2, Vector3d(-8.881784197e-016 m,-0.25 m,0 m));
97 //

```



If you are not using the *Snap Eccentric* feature, you create a horizontal beam where the end point is connected to the topology point without any eccentricities.

### 3.6.12 More about the Edit Beam dialog



Rightclick a beam to get access to the Edit Beam dialog.

The following paragraphs explain the tabs under the Edit Beam dialog that are not described other places in this volume;

“Offset Vector” is explained in paragraph 3.6.10

“Hinges” is explained in paragraph 3.8.3

“Split Points” is explained in paragraph 3.6.6

“Move End” is explained in paragraph 3.6.8

The purpose of this dialog is to modify beam specific parameters like local system, offset vector, hinges, split points, move ends, translate and buckling factors.

#### 3.6.12.1 Local System

Purpose: To modify the local system of a beam

##### Guide local system

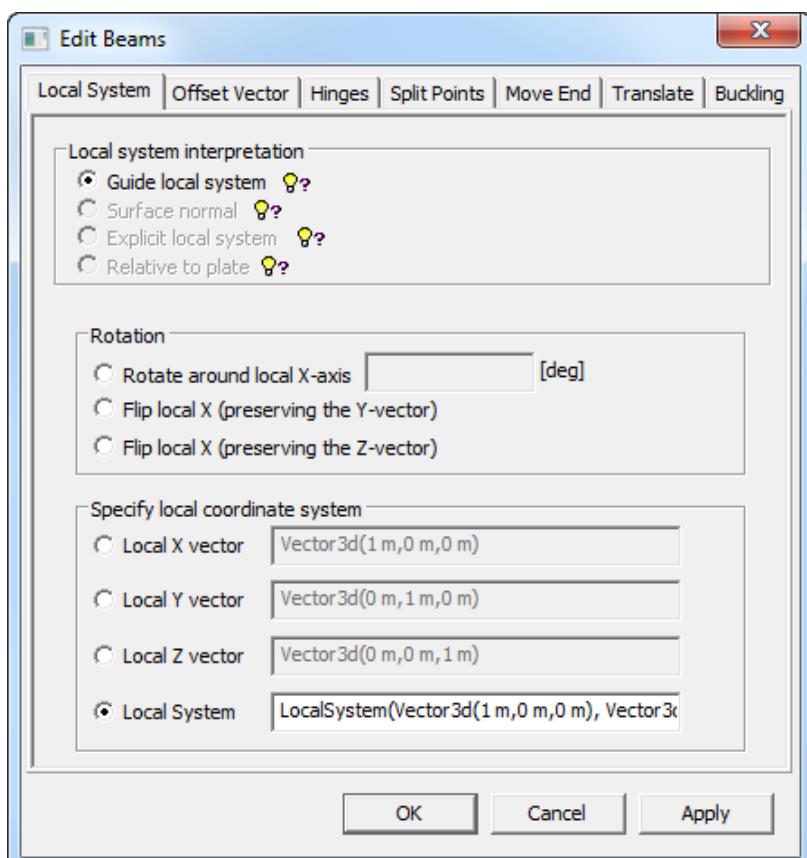
The local system you specify under “Specify local coordinate system” at the bottom of this dialog is projected onto the curve.

$X_{local}$  = Projection of  $X_{global}$  on curve.

$Y_{local}$  =  $Z_{global}$  cross  $X_{local}$

$Z_{local}$  =  $X_{local}$  cross  $Y_{local}$

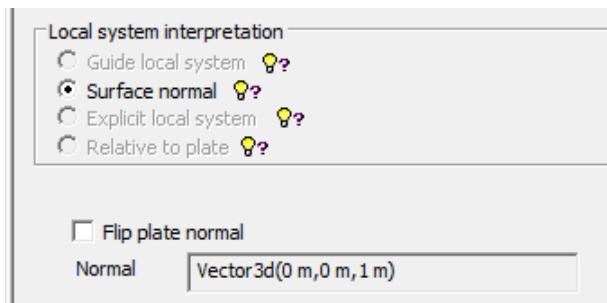
Applicable for beams and support curves.



##### Surface normal

Click on a plate to enable this choice.

This allows you to inspect and flip the surface normal of one or more plates and/or shells.



## Explicit local system

The local system you specify is applied directly to the concept (if applicable).

This is applicable for support points, support curves and placed equipment.

Local system interpretation

Guide local system

Surface normal

Explicit local system

Relative to plate

Rotate local coordinate system about

X

Y axis  [deg]

Z

Specify local coordinate system

Local X vector

Local Y vector

Local Z vector

Local System

Global System

## Relative to plate

Force the local system's Z-value to follow the normal of a plate.

This is applicable for support curves and curved beams.

Local system interpretation

Guide local system

Surface normal

Explicit local system

Relative to plate

Rotation

Rotate around local X-axis  [deg]

Flip local X (preserving the Y-vector)

Flip local X (preserving the Z-vector)

Relative to plate

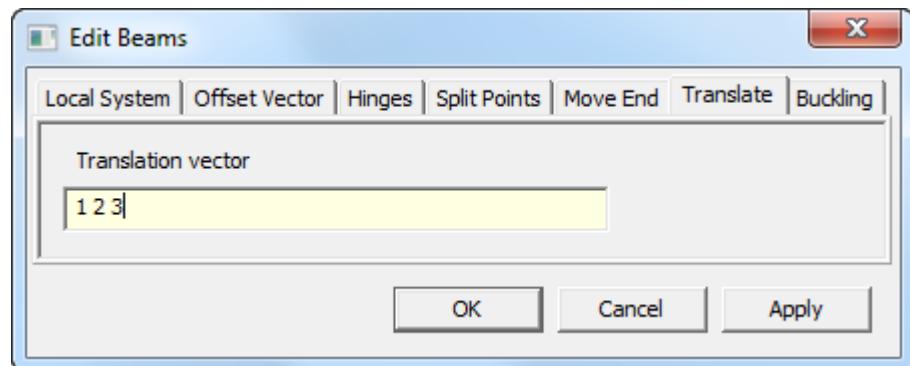
Align local z-axis  [deg]

relative to the normal of

### 3.6.12.2 Translate

The purpose of this dialog is to move a beam with a translation vector.

Type in the coordinates of the vector to move the beam.



### 3.6.12.3 Buckling

#### Buckling Length

Choose if you want to use the beam's length or type in manually the buckling length you want.

#### Effective length factor

Default value is 1. You can insert another effective length factor if you want.

#### Moment amplification

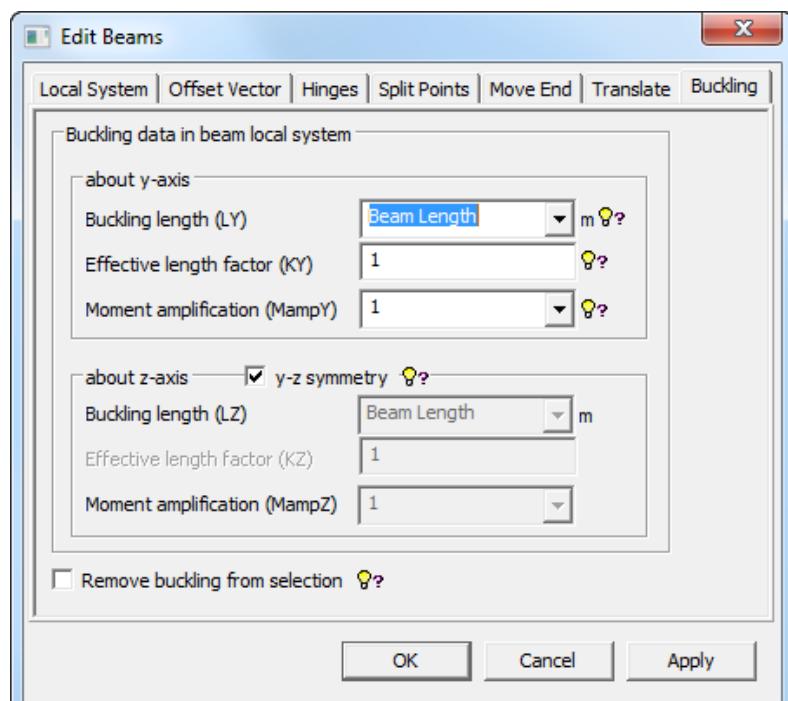
Default value is 1. You can insert another value or select one of the following alternatives:

Formula A

Formula B

Formula B C

Formula C



#### Remove buckling from selection

Check this if you want to remove the buckling factor property for the selection.

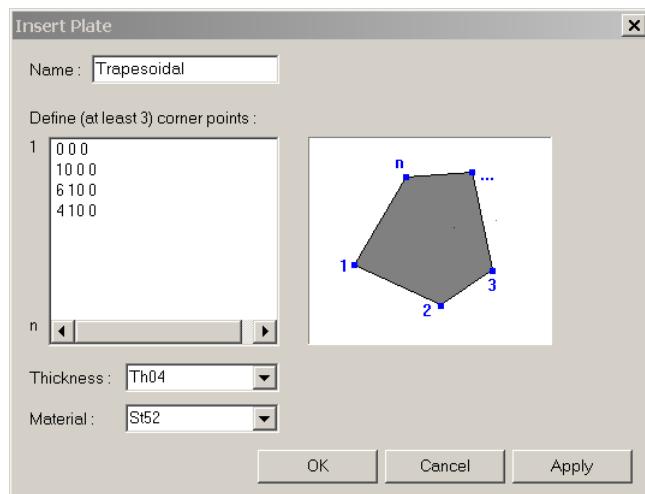
## 3.7 Basic plate modelling

Basic structure modelling of plates covers regular modelling and associated tasks (like for example edit, change, delete) of plates in a plane (planar plates). Prior to modelling structure it is assumed that libraries for sections, materials, and plate thicknesses have been defined. When modelling, the default settings are automatically applied to all new structures created. The defaults may be changed at any time, and it is also easy to change properties associated with a particular structural member

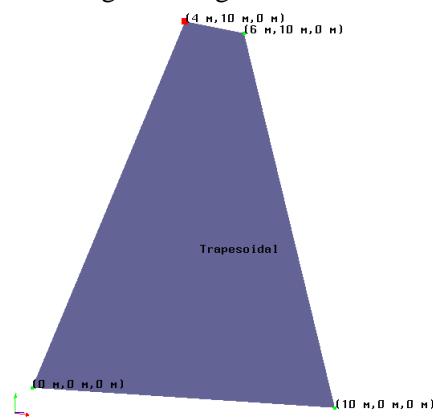
A plate is modelled using the same methodology as for a beam, in other words by explicit modelling of plate end co-ordinates or by clicking to existing snap point in a guide geometry or existing structure.

### 3.7.1 Regular plate in a plane

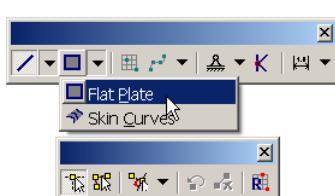
Explicit modelling of the plate is done from the pulldown menu Insert|Plate|Flat Plate Dialogue.



This example shows how to create a trapezoidal plate. Other plate shapes may be triangular, rectangular, or n sides. The only requirement is that the sequence of snap points (or topology points) form a closed circuit. The local z-axis is always perpendicular to the plate, positive direction according to the right hand rule.

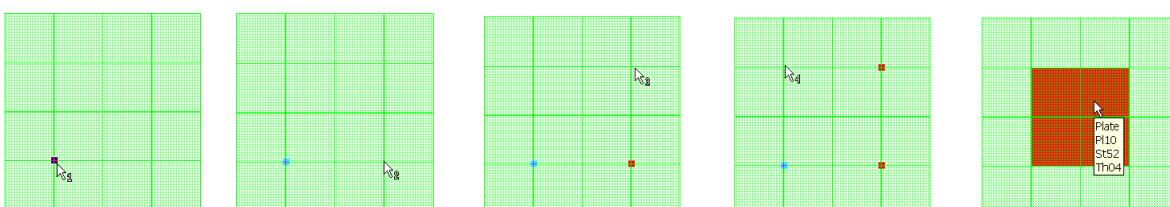


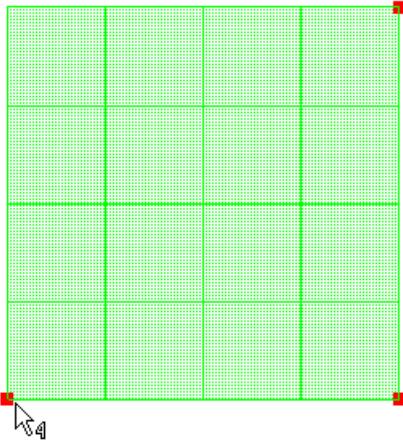
The other alternative of inserting a plate is by using the snap to point method.



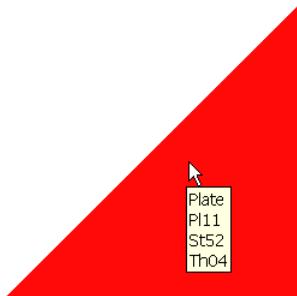
Make sure that the Flat Plate option is selected and that the insert mode button has been activated.

The example below shows the sequence when modelling a rectangular plate using the snap points in a guide plane. The program will prompt for input during the modelling sequence.

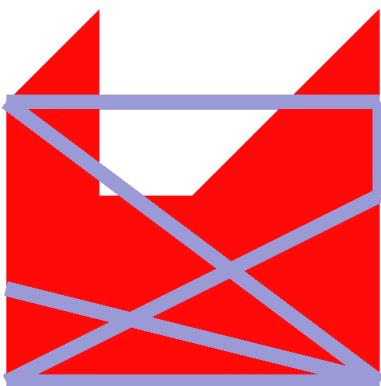
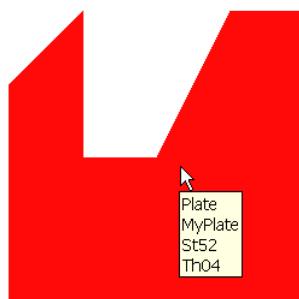
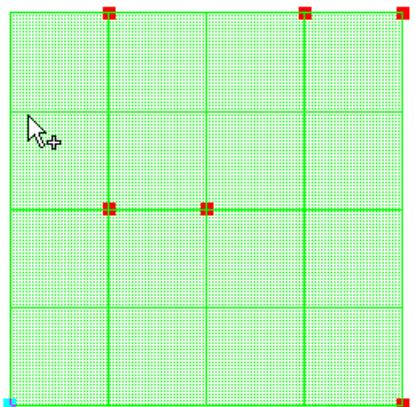




To insert a triangular plate the same technique is used. The difference is that the prompted 4<sup>th</sup> snap point is the same as the first snap point used when creating the plate.



You may also insert a plate consisting of n edges. To do this you must use the snap point loop method and ensure that a closed circuit is generated by the snap points. This means that the last snap point is the same as the first snap point. The mouse tooltip indicates a + to prompt for new snap points. The example below shows a plate with 8 edges.

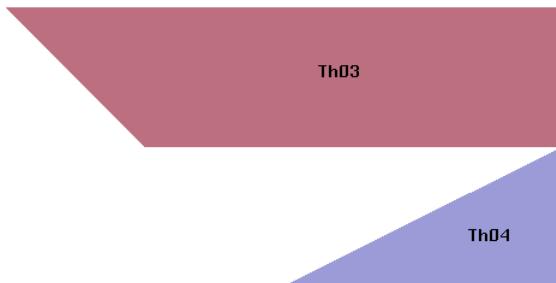


Beams and plates may be inserted at any time, GeniE calculates all intersections and manages a consistent topology. If you expect many changes to your model during structure modelling, it may be of advantage to do most of beam modelling prior to inserting plates. The reason for this is that the topology model then becomes as clean as possible. If you choose to model plates initially, you should use the feature **Tools/Structure/Geometry/Simplify Topology** frequently to remove unnecessary topology points and lines.

The example to the left shows one plate intersected by many beams.

### 3.7.2 Overlapping plates

You cannot insert a plate in the same position as an existing plate. The program will give a warning on this.

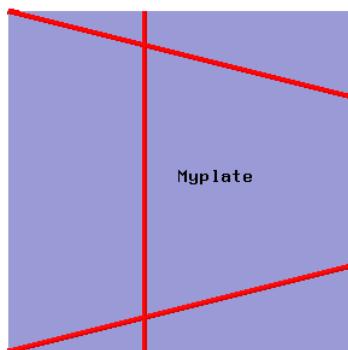


If you insert a plate larger than the original one, the program will insert the delta between these plates. See the example to the left where a larger plate is moved over a smaller plate.

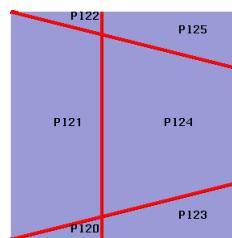


### 3.7.3 Split and join plates

The command **Tools/Structure/Split** will split the selected plate(s) in minor plate parts in between intersections with beams (or feature edges). You may also select the plate, RMB and use the *Split* option.



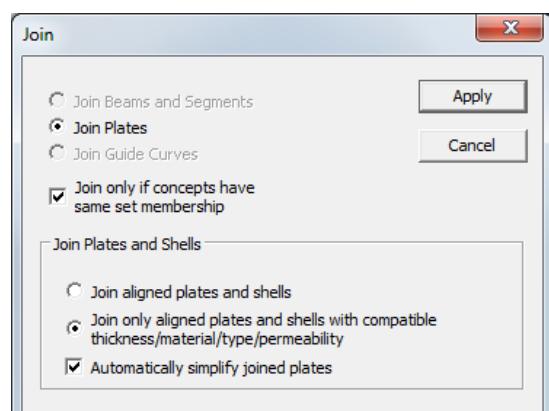
This example shows MyPlate intersected by 3 beams before and after a split operation. As can be seen, MyPlate is split into six smaller plates.



Plates can be joined by selecting two or more plates, rightclick and choose “Join”.

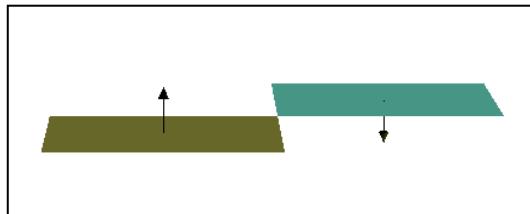
In the Join dialog you have some options to control the rigidness of the joining criteria.

Plates can also be joined from the command line area. Joining P1124 with P1121 has the command  
*P1121.join(P1124);*



### 3.7.4 Plate local co-ordinate system

The local co-ordinate system of a plate is defined according to the right hand rule, i.e. positive x-axis along thumb, positive y-axis along pointer finger, and positive z-axis along middle finger. The positive x-axis is in the direction from first snap point to second snap point.



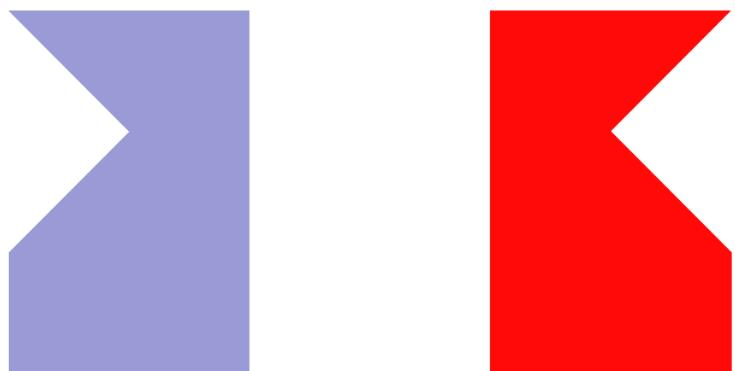
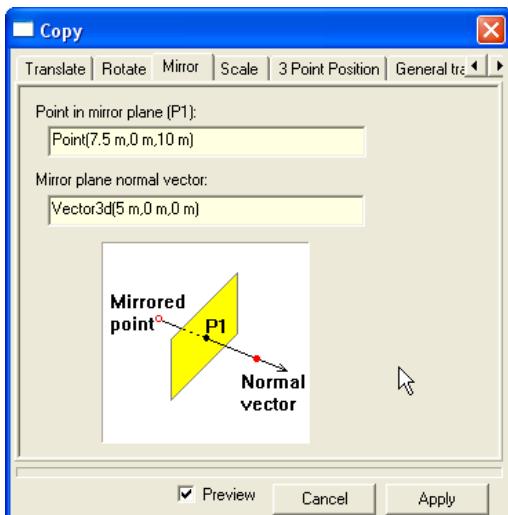
The example below show that two plates have been modelled in opposite directions, hence local z-axes in different directions (context sensitive menu and *Labels/Local coordinate system*).



You change directions of plate normals by selecting the plate, RMB and *Flip Normal*.

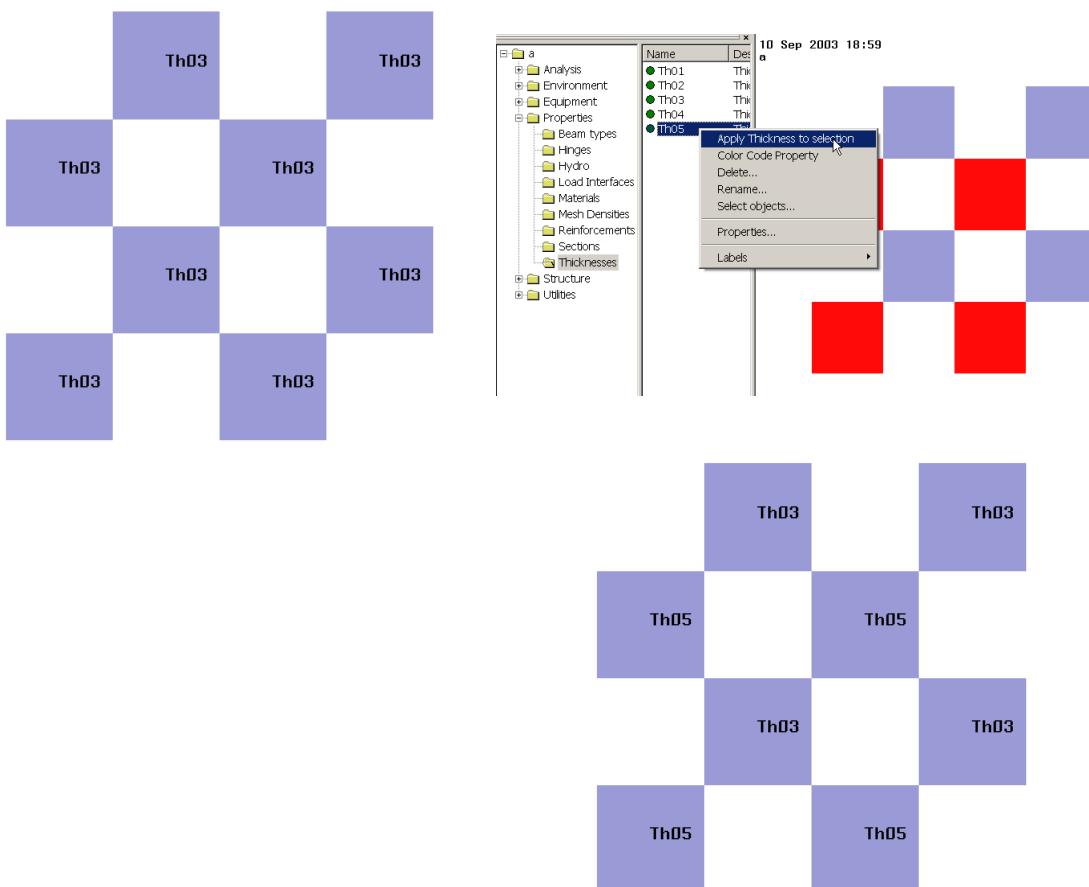
### 3.7.5 Delete, move and copy plates

These operations work the same as for beams. The example here shows a plate (the left plate) that is copied using the mirror option.



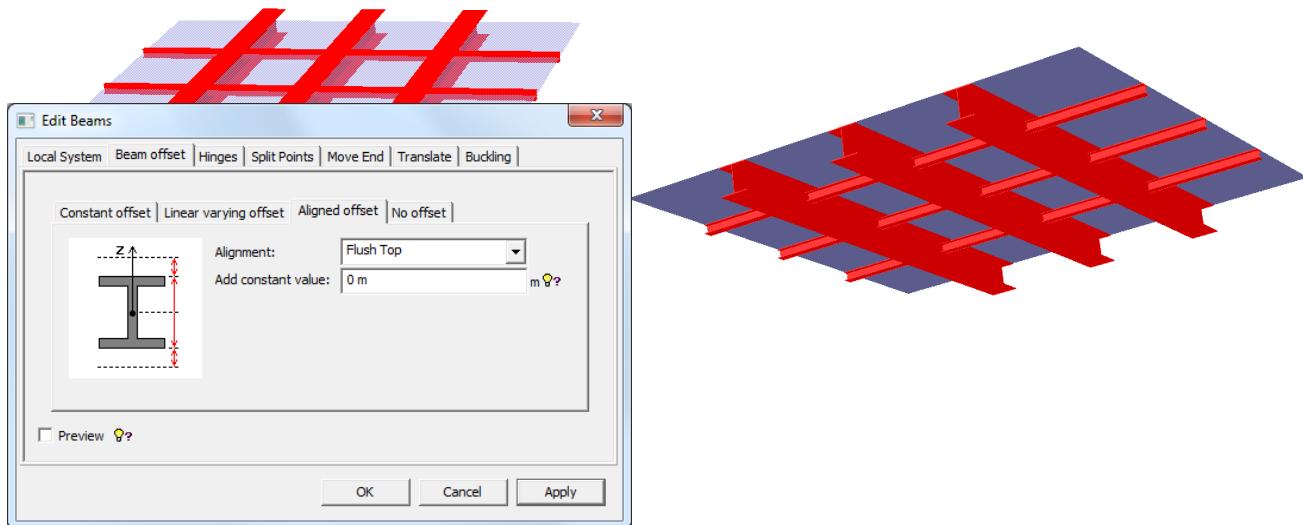
### 3.7.6 Change plate properties

Plate properties may be changed from context sensitive menu Properties or from the browser, both methods require that a plate(s) is selected first. The example below shows how to change plate thickness for a given plate.



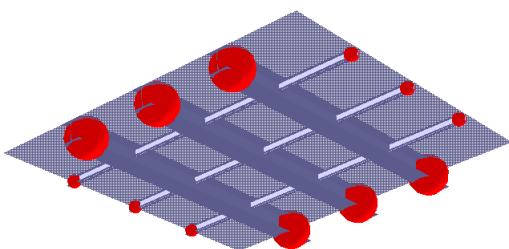
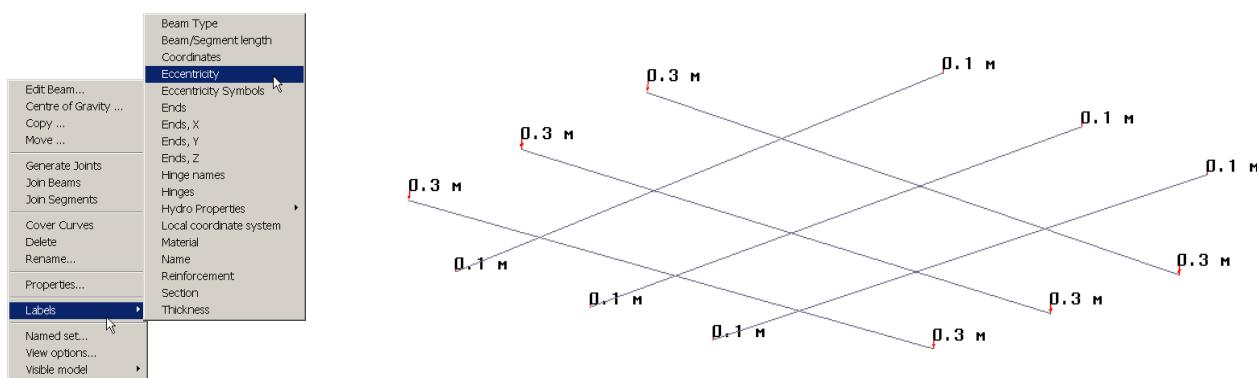
### 3.7.7 Stiffened panels

GeniE can do flushing (insert eccentricities to beams related to a plate) of multiple beams at the same time. The example below shows beams with two different section types. When modelling these they are inserted at the same level (system level or top of steel) as the plate. In most cases, the beams are flushed either above or below the plate. Select the beams, force the context sensitive menu and choose *Edit Beams/Offset Vector*.



In the example above flush to top of beam was used (flush top). This means the beam is placed below the plate. Similarly flush bottom will place the beam on top of the plate. Observe that the flush operation will use eccentricities calculated along the beams local z-axis. This means that flushing also may be done on beams attached to e.g. vertical walls.

Eccentricities may be graphically verified (or from the printed report) selecting Labels from the context sensitive menu – either showing real values or symbols.



## 3.8 Advanced structure modelling

This Section describes the more advance modelling features in GeniE. They are special features related to topside structures (segmented modelling, hinges), jacket structures (tubular joint modelling, local joint flexibility, shim elements), plated structures such as topsides, semi-submersibles and spar buoys (curved structure).

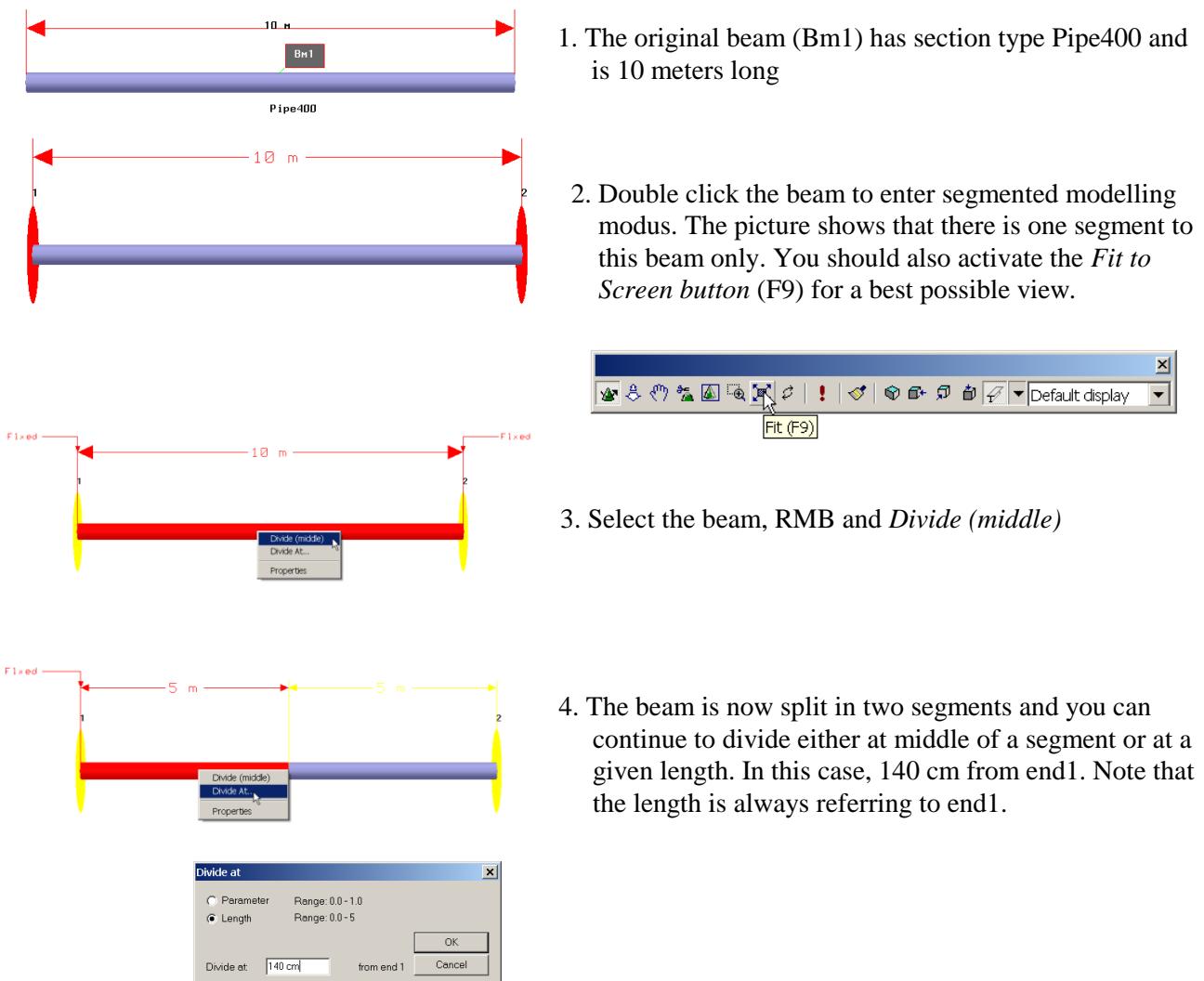
Emphasis has been put on describing the steps necessary to arrive at the desired model. There are several modelling approaches, this document describes some of them. Focus has also been put on the importance on how to document and verify your model.

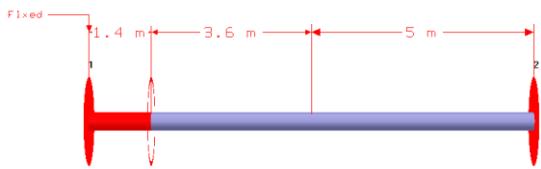
### 3.8.1 Segmented beam modelling

A segmented beam is a continuous beam consisting of several segments. A segment is normally inserted when there is a change of section or material property. It will act in the same way as several beams aligned, the main difference is that the segmented beam is a higher level concept compared to a set of beam members. Instead of referring to several beam names you can refer to only one name of a segmented beam and still get the same results.

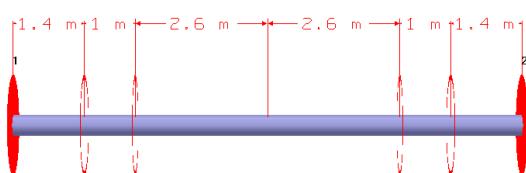
In the following is explained the steps on how to create a segmented beam from a straight beam until the segments are joined to form a straight beam again.

#### Create segments



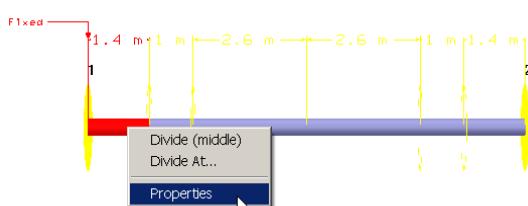


5. The beam is now split in 3 segments and the individual lengths are automatically calculated ensuring that the total beam length is maintained.

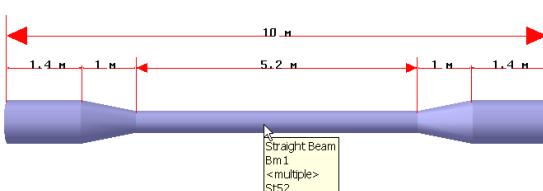


6. The final configuration of segments prior to assigning new section properties. Note that you can change individual lengths by clicking the relevant segment, type the new length from your keyboard, and complete the operation by pressing Enter. You may be prompted to fix one of the segment ends by using the TAB button.

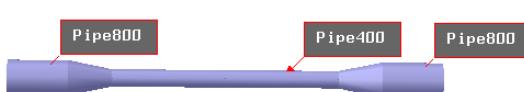
### Assign new section properties



7. Select a segment(s), RMB and assign new section or other material. In this example a cone section has been used in the transition between larger and smaller section profiles.



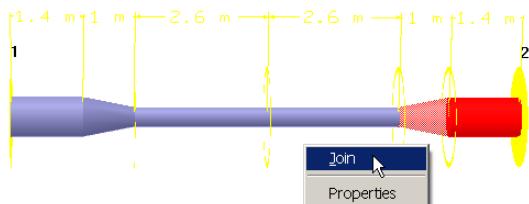
8. The final configuration. Dimensions have been added using (remember to activate the *Filter Segment*) the dimension tool *Create Dimension*.



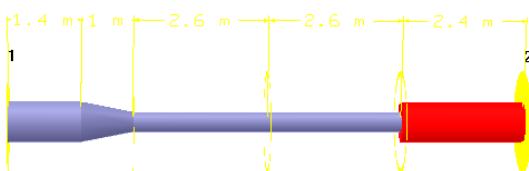
9. Verifying the model by applying labels, this is only possible when the *Filter Segment* is activated. You may also verify from Tab BeamSeg on a saved report

	A	B	C	D	E	F	G	H	I	J	K	L
1	Name	SegNo	Section	Material	Length [m]	FlexLen [m]						
2	Bm1	1 of 6	Pipe800	St52	1.400	1.400						
3		2 of 6	Cone	St52	1.000	1.000						
4		3 of 6	Pipe400	St52	2.600	2.600						
5		4 of 6	Pipe400	St52	2.600	2.600						
6		5 of 6	Cone	St52	1.000	1.000						
7		6 of 6	Pipe800	St52	1.400	1.400						

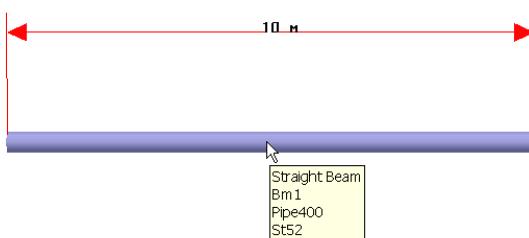
## Join segments



10. Make sure that segment modelling modus is enabled (double click the beam) and select the members you want to join. The first segment chosen will decide properties for the new segment. In this case Pipe800 is chosen.

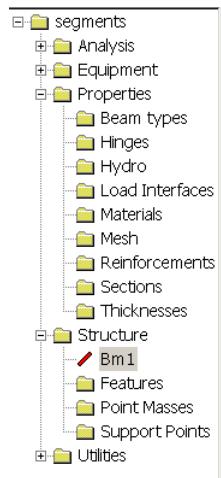


11. The two segments joined to one. You may join two segments or more at the same time.



12. Joining the five remaining segments (and selecting one of the middle ones first). The segmented beam is now back to the original straight beam

## Edit from browser



Name	Description	X [m]	Y [m]	Z [m]	Section	Length [m]	Material
End1	Apex	0	0	0			
End2	Apex	0	10	0			
Segment1	Segment				Pipe800	1.4	St52
Segment2	Segment				Cone	1	St52
Segment3	Segment				Pipe400	5.2	St52
Segment4	Segment				Cone	1	St52
Segment5	Segment				Pipe800	1.4	St52

When double clicking on Bm1, the details of the segmented beam appear in the browser.

You can change details of each segment by RMB.

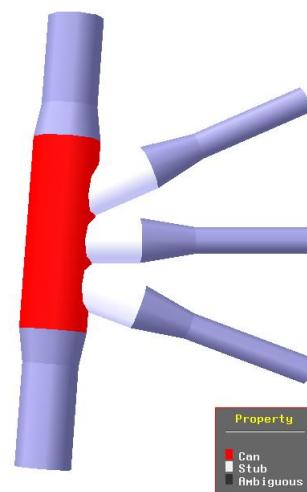
### 3.8.2 Tubular joint modelling

The purpose of tubular joint modelling is normally to enhance a beam model with information about cans, stubs, cones, and gaps to improve code checking results.

There are two ways of doing tubular joint design in GeniE. The first approach assumes you create segments and add information manually to the segments. Examples of such are manual assign can details to a segment, or manual offset of a brace member end along chord axis. Alternatively, all such information may be added in the code checking program Framework.

The most efficient way is, however, to use the automatic approach based on the rules specified by you (or using the default rules). This approach is described in the following:

- Step 1. Define rules for joint creation
- Step 2. Define rules for lengths of cans, stubs, cones and also gaps
- Step 3. Assign joints
- Step 4. Automatic assignment of cans and stubs
- Step 5. Change sections for cans and stubs – all joints in one operation if possible
- Step 6. Do automatic assignment of conesDo planewise gap calculation – automatic and all joints in one operation if desired



**Step 1 – See Rules Chapter 3.2**

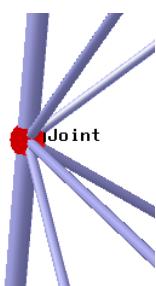
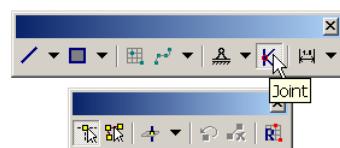
**Step 2 – See Rules Chapter 3.2**

**Step 3 – Assign joints**

In the following it is assumed that joints are inserted manually. A joint can be inserted from the pulldown menus

- Insert|Joint|Joint Dialogue. You are prompted for input
- Insert|Joint|Joint. Assumes graphic modelling and use of default names.
- Insert|Joint|Generate Joints. Will create joints on selected beams according to the rules

You may also insert a joint from the toolbar Object Types using the *Joint* button. Make sure the program is set to input modelling mode.



The example to the left shows that a joint has been inserted and renamed by selecting it, RMB and *Rename*. It is required to define a Joint at each tubular joint where you want to do automatic assignment of cans, stubs, cones, and gaps. It may be of advantage to specify 50% transparency of the joint symbol (*View Options/Settings/Structure/Joint/Transparency*). To be able to see the joints graphically, ensure that the Joint selection is activated.

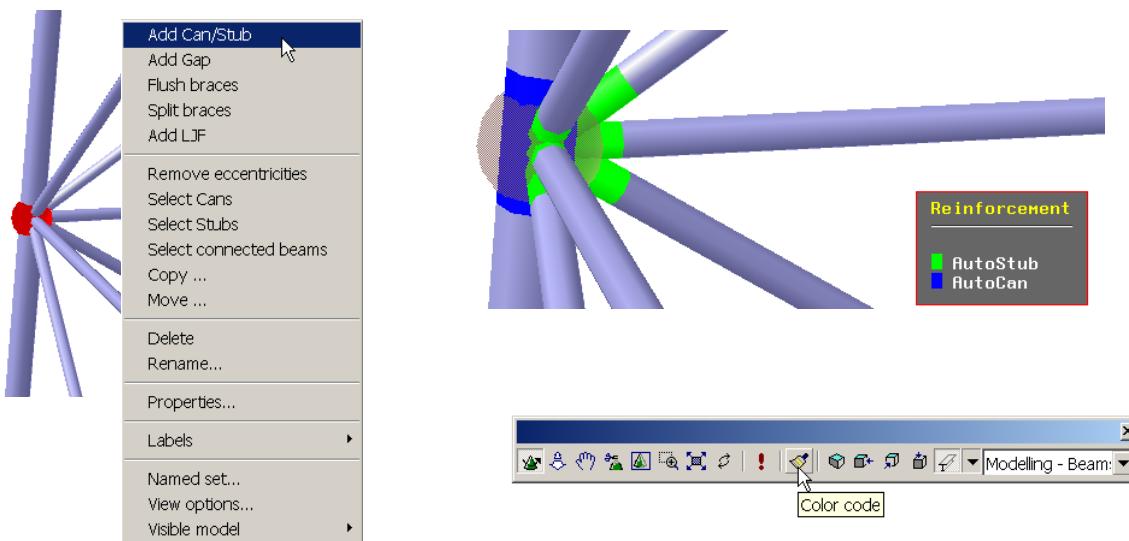
Note that by default, joints are visible, but the joint selection button is not active. You must click the joint selection button to be

able to manipulate joints.

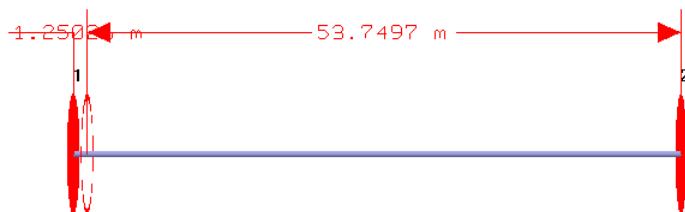


#### Step 4 – Assign cans and stubs

Cans and stubs may easily be assigned – again based on the rules specified - when a joint (or joints) have been selected. The example below shows assignment of can and stubs by the *Add Can Stub* option in the RMB menu for the selected joint. The cans and stubs have been colour coded by selecting them from the browser *Properties/Reinforcements*, RMB and then *Colour Code Property*. To view beams connected to a particular joint, select the joint, RMB and then *Select Connected Beams*. The colour coding is switched off by de-activating the *Colour Code* button.

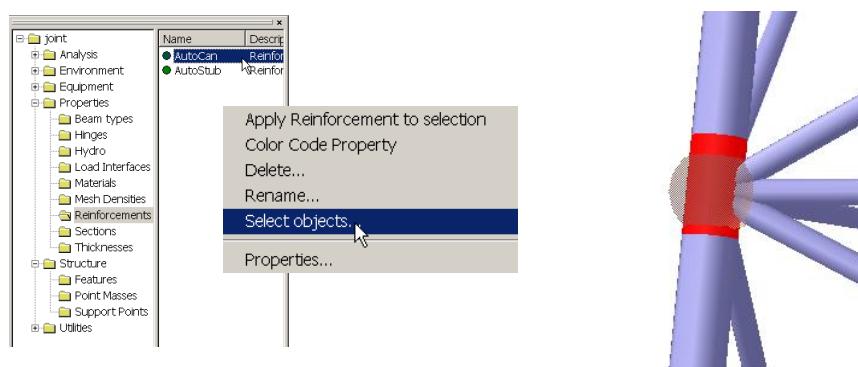


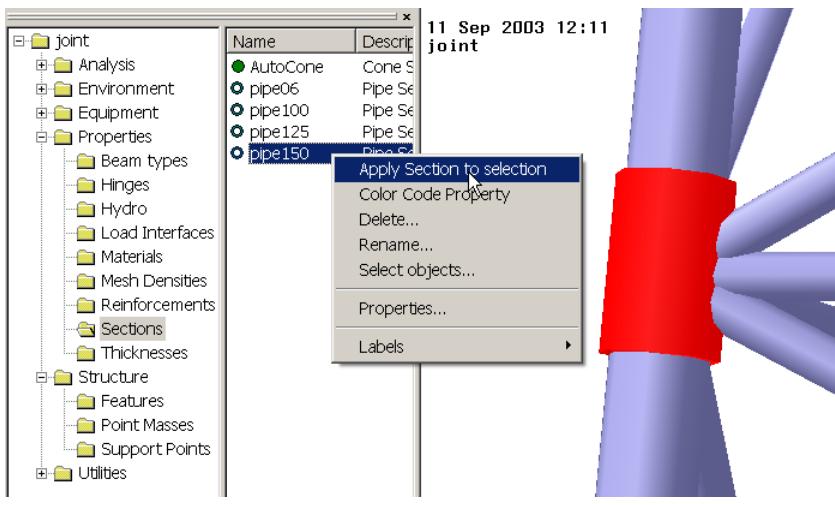
When assigning cans and stubs, the program makes the respective straight beams into segmented beams, where the segment lengths are according to the can and stub lengths specified in the rules for such. This command will also assign concept information (cans and stubs) that will be read by the code checking program Framework. The picture below shows the segmented members of the horizontal brace.



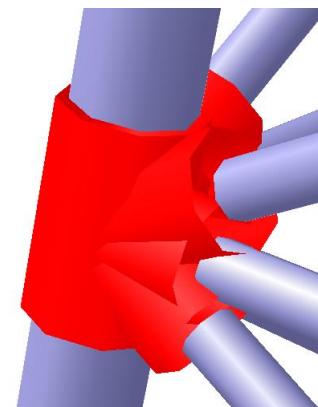
#### Step 5 – Assign new properties to cans and stubs

The final task to do with cans and stubs is to assign other section or material properties. The easiest way is to select these objects from the browser, choose properties from browser and apply. The sequence below shows how to select the cans and how to change section properties.





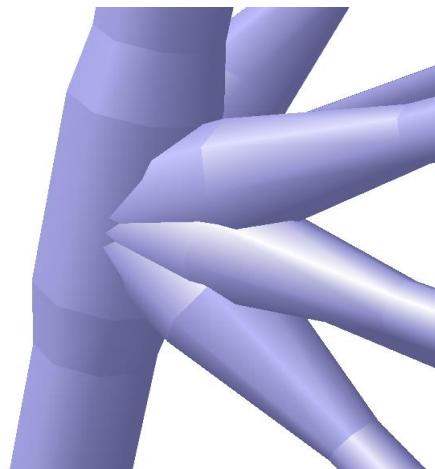
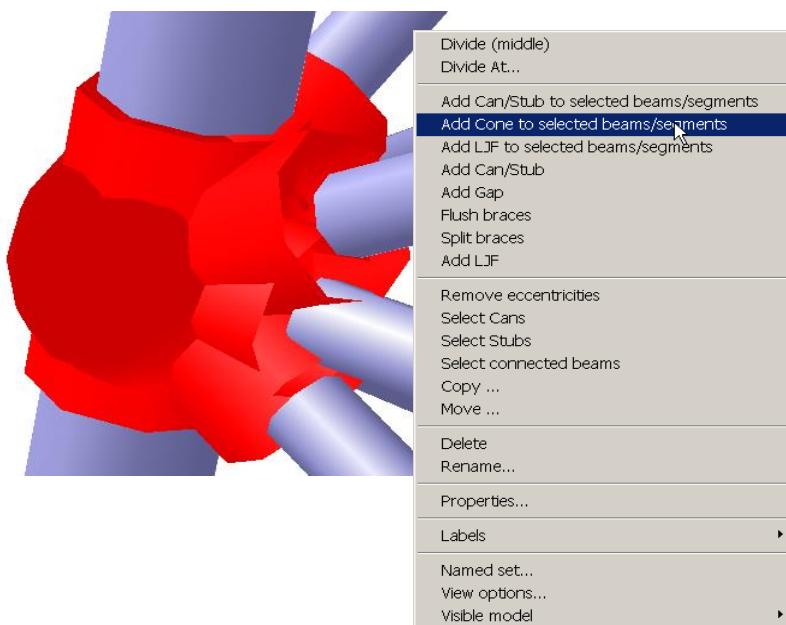
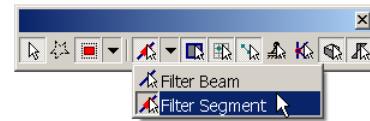
The cans are assigned new section property pipe150. Picture below shows both cans and stubs reinforced.



Note that if you do not want to add gaps you need to recalculate can and sub lengths (same as Step4) due to the updated geometry. The gap operation will adjust the can and stub lengths.

#### Step 6 – Assign cones

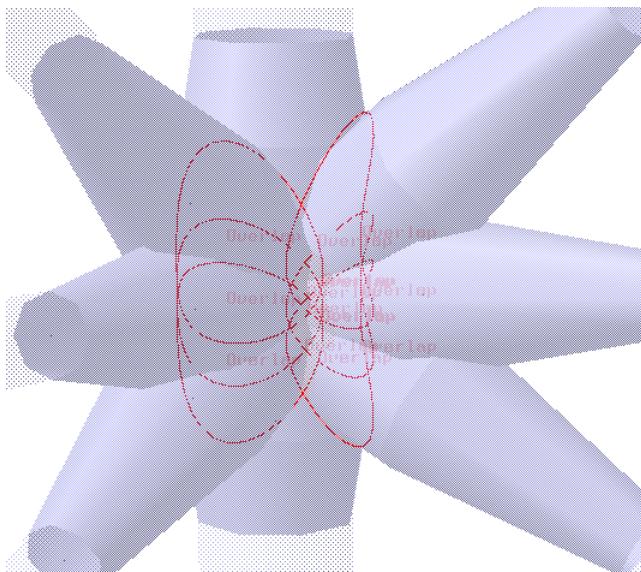
To insert cones you need to select a joint and the cans and stubs to which you want to insert a transition (cone) to the chord or the brace. The easiest way to select these is to use rubberband selection around a joint. Make sure that the joint is visible and that the *Filter Segment* is activated. When the objects are selected, then RMB and *Add Cone to selected beams/segments*.



You may choose to not insert cones to all the cans or stubs, you then deselect the relevant cans or stubs and no cones will be calculated.

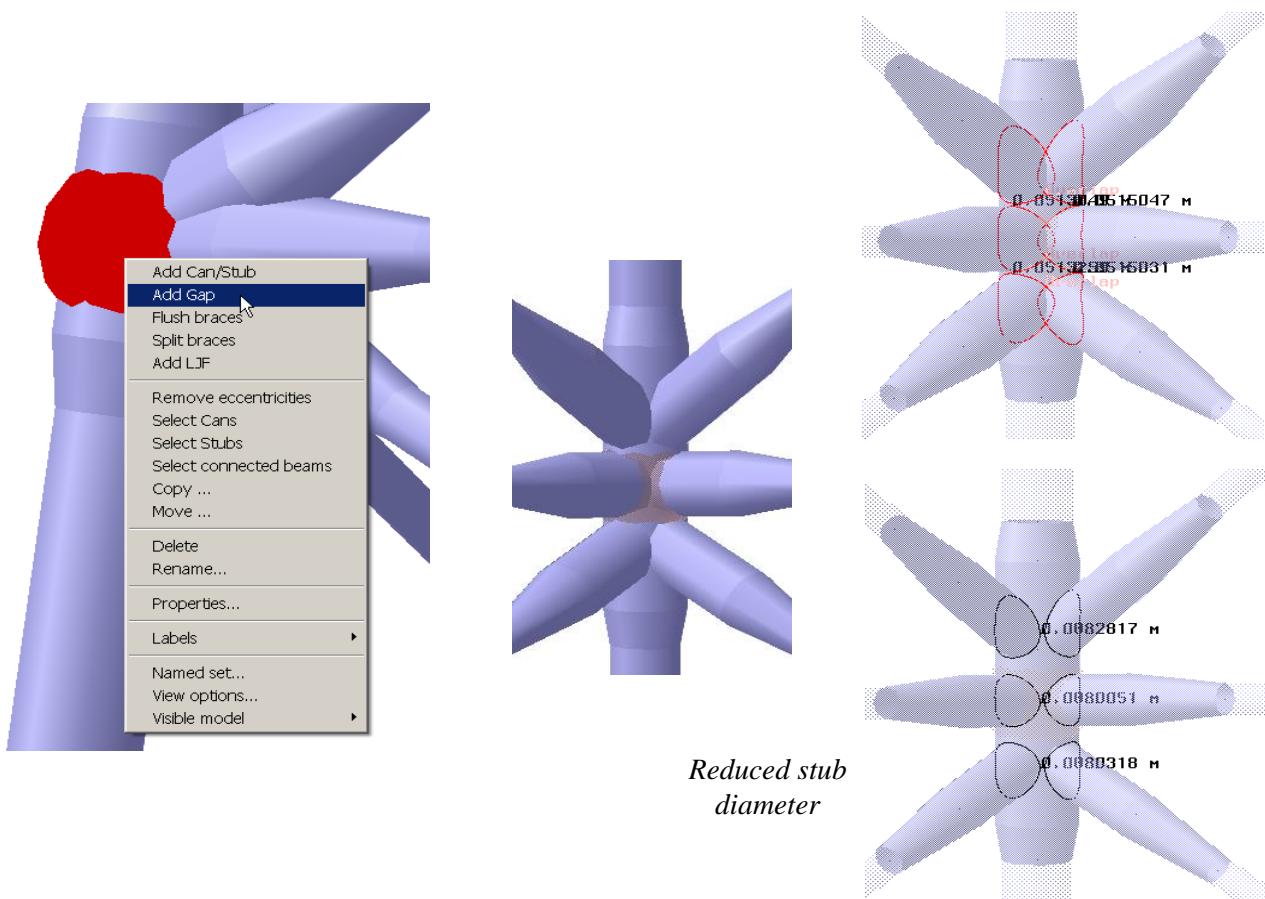
## Step 7 – Perform planewise gap calculations

The gap calculations are performed by selecting the joint (graphically or from browser), RMB and *Add Gap*.



You may find the existing gap values before a gap calculation by double-clicking the joint. Gap values shown in red colour indicates that they are larger than the minimum gap value specified in the rules.

The example below shows the actual gap calculation. It also shows that for this case that a) the diameter of the can is not large enough to satisfy minimum gap requirements (the stubs are overlapping each other) and b) by changing the stub diameter satisfactory gap requirements are maintained. Note that gaps (due to existing geometry) which are larger than minimum required gap are not reduced.



The program has now inserted eccentricities to the respective incoming brace ends. The eccentricity vectors are in line with the chord axis. The brace(s) perpendicular to the chord will not receive eccentricity vectors.

The beam end eccentricities will be written out to the FEM file and used in the structural linear analysis.

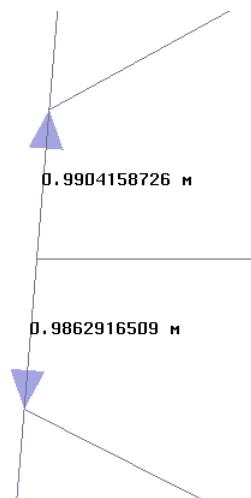
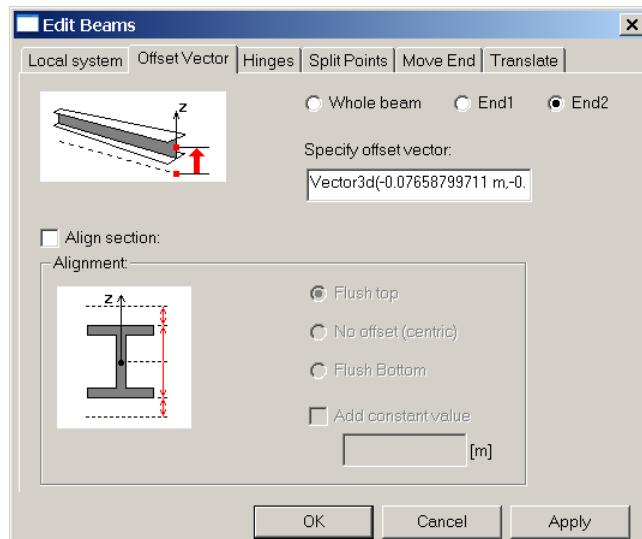
For the lower brace, an eccentricity vector  $\delta = 0.98629$  m has been inserted. For more details about  $\delta_x$ ,  $\delta_y$ ,  $\delta_z$  this may be found by opening the beam editing function from the context sensitive menu.

In this case they are:

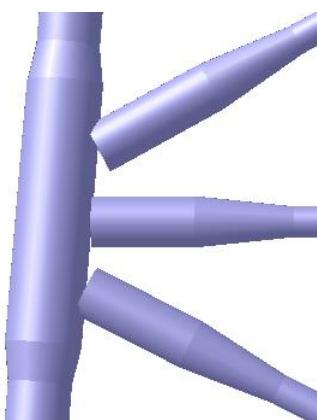
$$\delta_x = -0.0765879971 \text{ m}$$

$$\delta_y = -0.0765879971 \text{ m}$$

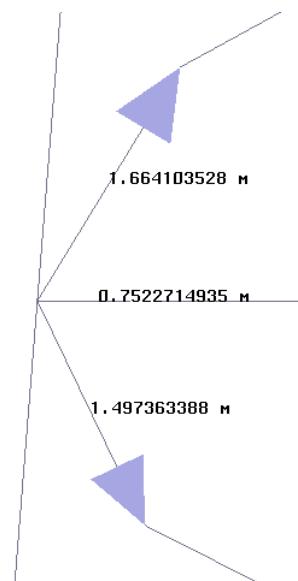
$$\delta_z = -0.9803263631 \text{ m}$$



You may also flush the braces to the chord wall by selecting the joint, RMB and Flush Braces. In this case eccentricities are first calculated along the chord axis and then moved out to the chord wall. Horizontal braces will also receive eccentricities, see example below.



*Braces flushed to chord wall*



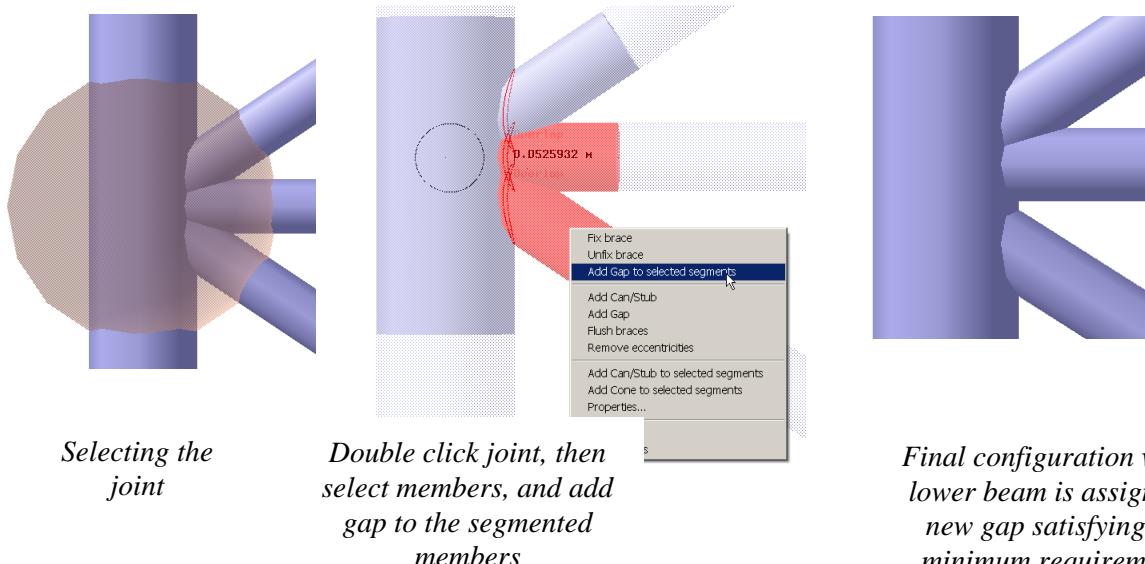
*Eccentricities out to chord wall,  
horizontal brace also receives  
eccentricity*

To remove eccentricities assigned, select a joint(s), then RMB and *Remove Eccentricities*.

There may be cases where you want to perform other special planewise gap operations. Several options are available and in the following some are documented.

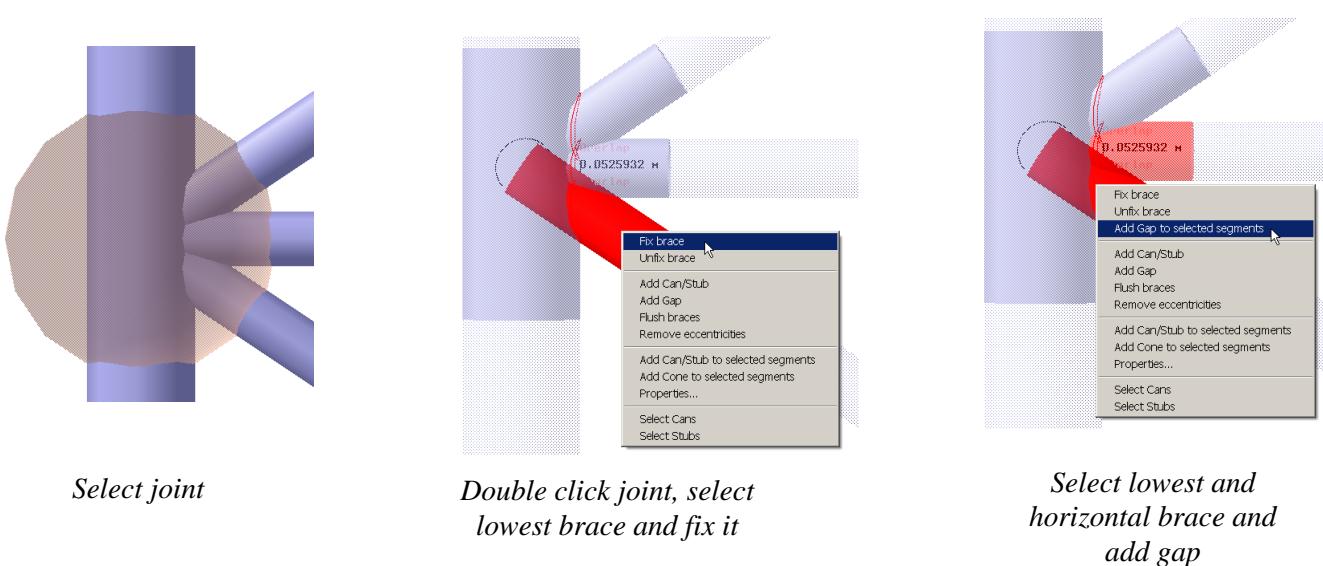
#### **Option 1 – Add gap to one brace only**

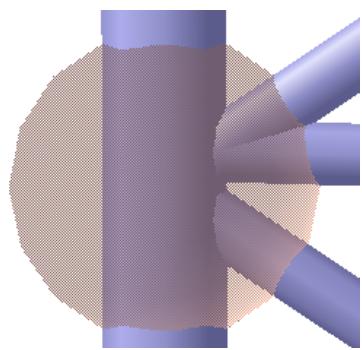
You need to create cans and stubs prior to calculating the actual gaps (double click the joint). Per default the modelling is switched to segmented modelling modus. You can now select the horizontal brace and the inclined brace to calculate new gaps, RMB and *Add gap to selected segments*. The figures below show the steps to add gap to the lower beam only.



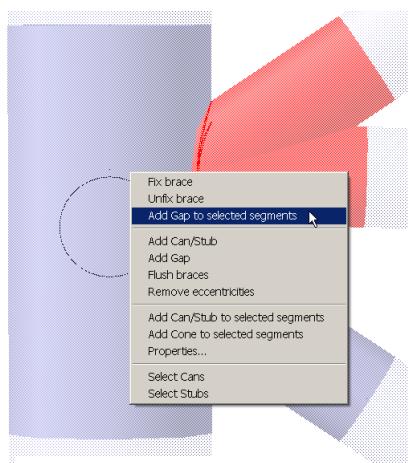
#### **Option 2 – Move the T in a KT-joint up**

This example shows how you can move the T (or the horizontal brace) in a KT-joint up. It also shows how to ensure that the minimum gap requirements are met on both sides of the T.

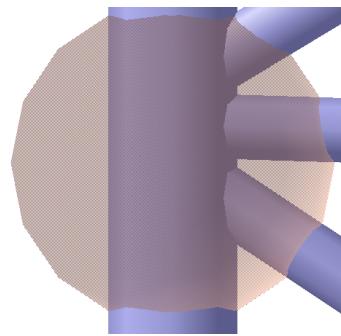




*Configuration before gap calculations on upper brace*



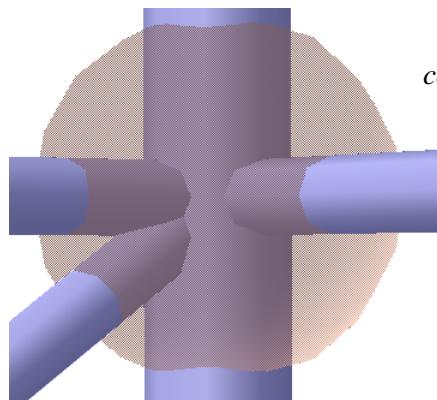
*Double click joint, select upper braces and add gap*



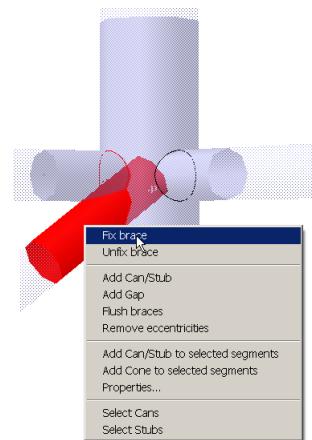
*A KT-joint where T is moved upwards satisfying the minimum gap requirements*

#### **Option 3 – Move the T part of K-joint (Y+T)**

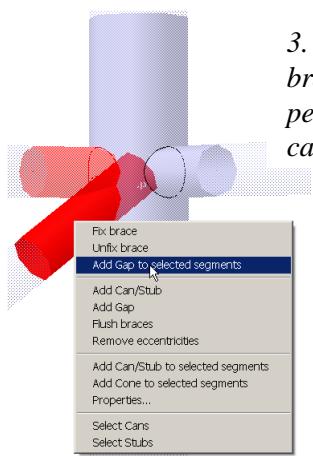
This example illustrates how to move a T (perpendicular to chord) in a Y+T joint. The steps are basically as showed before, the differences lie in which members to select.



*1. Initial configuration*

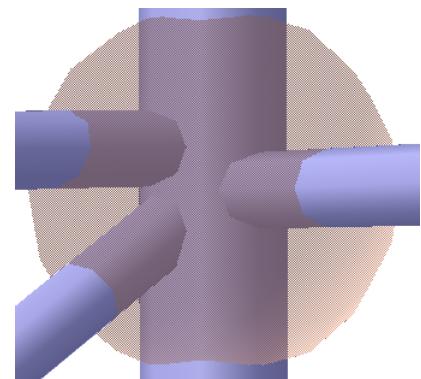


*2. Double click joint, select, RMB and fix brace*



*3. Select both braces, RMB, perform gap calculation*

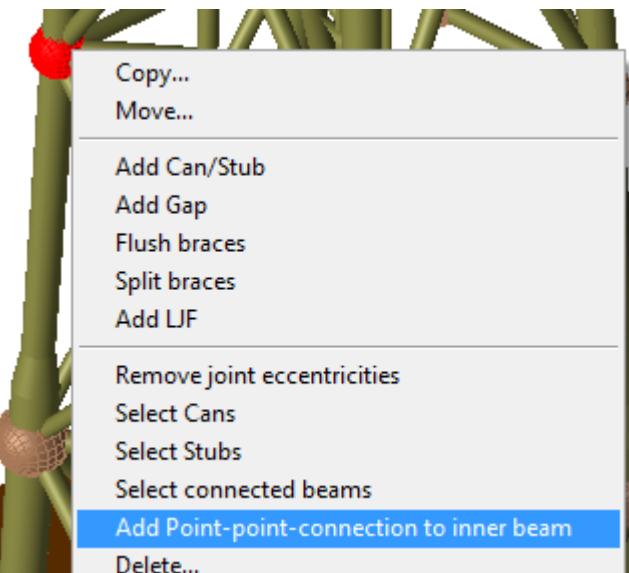
*4. Final configuration where the upper horizontal beam has been moved upwards*



### 3.8.2.1 Add point-point-connection to inner beam

This operation will create a point-point connection between outer and inner beam at the location of the selected joint(s). The inner beam must have beam type Inner Beam Disconnected assigned.

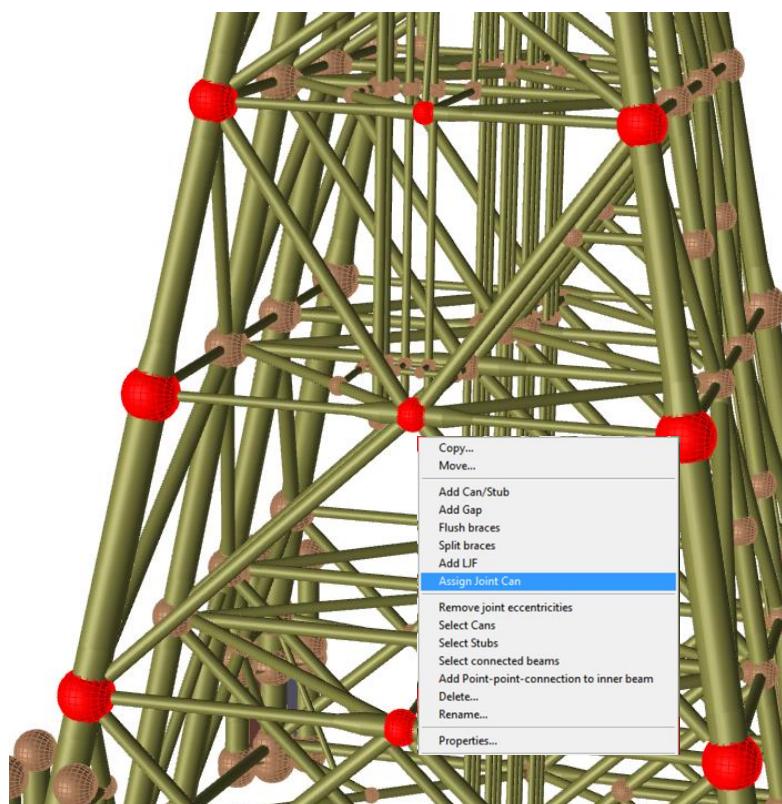
Connection point management will be done, to ensure that the inner and outer beams are disconnected at the selected joint(s). The point-point connection spring will be aligned with the inner beam local system.



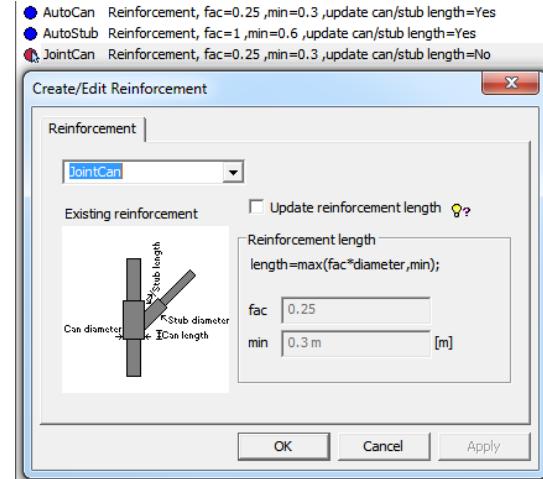
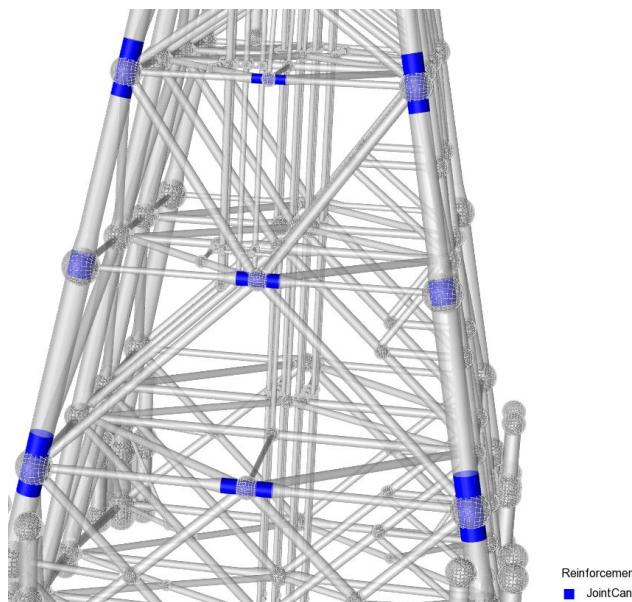
### 3.8.2.2 Assign Joint Can

To automatically create and assign Joint-Can reinforcement to joints, do the following:

- Select relevant joints and run the “Assign Joint Can” command from the context sensitive (RMB) menu



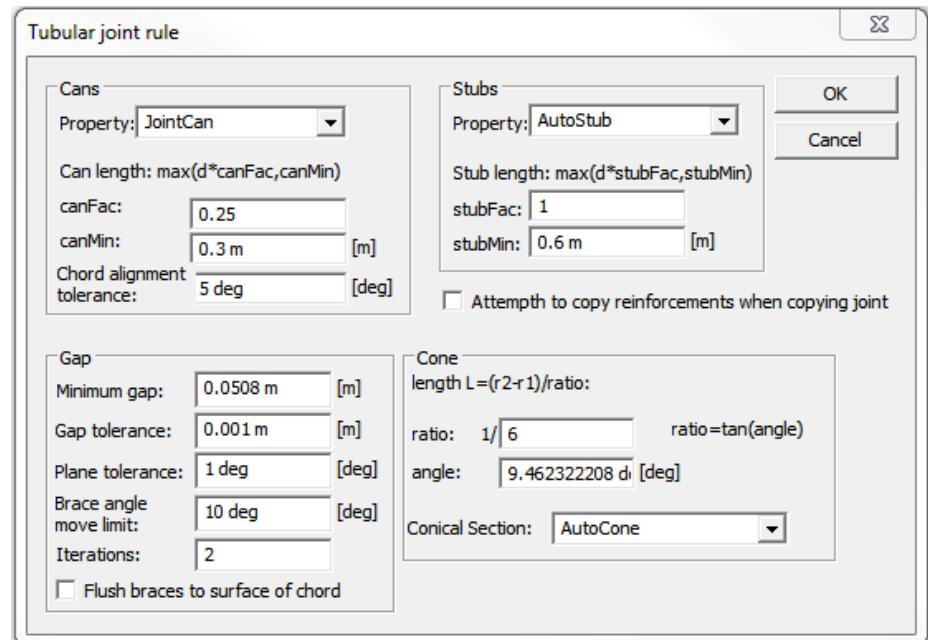
A Reinforcement property named “JointCan” is created.



Joint Cans are assigned.

Note that if a JointCan reinforcement is assigned to a chord that is illegal in a Joint Code Check, calculations are skipped for that joint/chord.

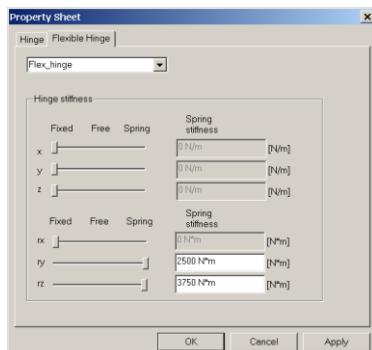
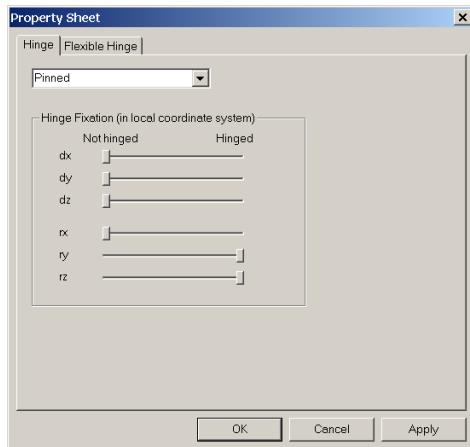
If not already done Joint Design (Tubular joint) rule is established.



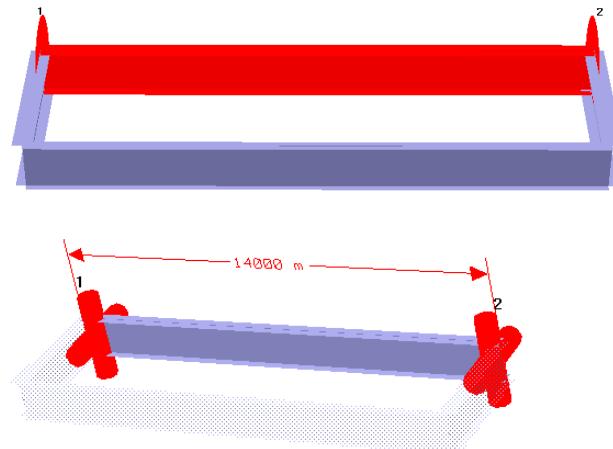
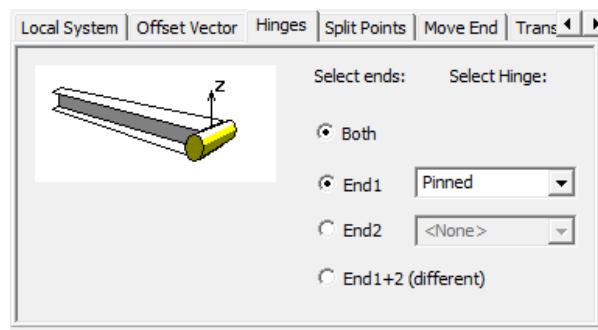
### 3.8.3 Hinges

Hinges may be inserted at end of beams with the purpose to release local degrees of freedom. For example, a typical hinge is fixed from deflections in all translation DOF and torsion DOF. The two other rotation DOF ( $r_y$  and  $r_z$ ) are free. The below example shows how to insert and verify hinges. You can also represent a flexible hinge if you know the spring stiffness for each DOF.

The steps are first to define a hinge property(ies), select beam(s) end(s), and apply the property to beam end.



A hinge property may be generated from **Edit/Properties/Hinge/Create/Edit Hinge**. You may choose *Hinge* or *Flexible Hinge*, in the following Hinge has been used.



Select beam, RMB, Edit Beams|Hinges.

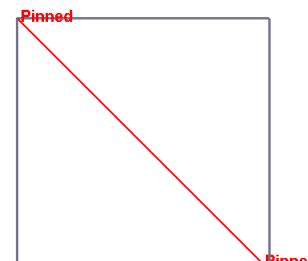
Apply hinge property to one or both beam ends.

To verify, select same beam, RMB, *Edit Beams/Hinges* or double-click beam to enter segmented modelling modus. Both methods will show the hinges graphically. You may also save a report to do a tabular verification.

	A	B	C	D	E	F	G	H	I	J	K
1	Name	X-End1 [m]	Y-End1 [m]	Z-End1 [m]	X-End2 [m]	Y-End2 [m]	Z-End2 [m]	Section	Material	Hinge-End1	Hinge-End2
192	H277	194830.000	224000.000	47000.000	205170.000	224000.000	47000.000	H200A	S355		
193	H278	202540.000	214000.000	46650.000	202540.000	219200.000	46650.000	H240A	S355		
194	I120	176500.000	210000.000	44000.000	176500.000	224000.000	44000.000	I12050FJ	S355	Pinned	Pinned
195	I121	180500.000	210000.000	44000.000	180500.000	224000.000	44000.000	I12030DG	S355		

Finally, it is possible to label hinge symbols and names as shown to the right. You may also label hinge symbols when displaying a finite element mesh.

Note that hinges on a 3-node beams are not applicable. You will not get any response from GeniE on this when meshing.



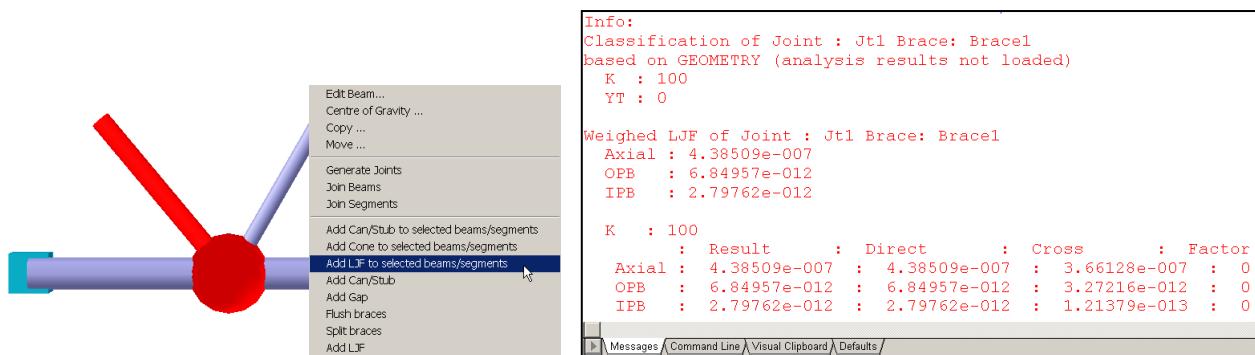
### 3.8.4 Local joint flexibility

You may also insert local joint flexibility (LJF) according to Buitrago et. al. (1993). It is presumed that you are familiar with the theory behind introducing flexible braces to chord connections.

LJF may be assigned based on a pure geometry configuration or a load path.

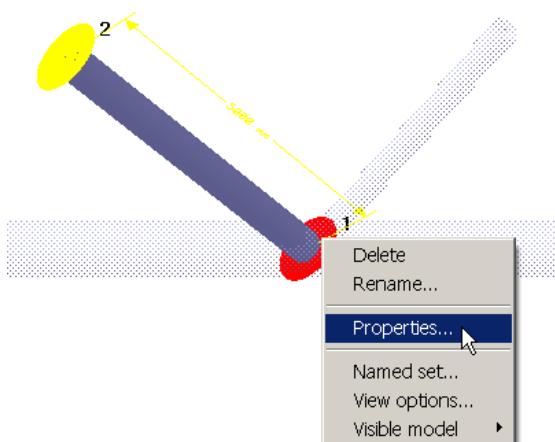
- Pure geometry configuration only. Select joint(s), RMB, *Add LJF*. Local joint flexibility is assigned to all braces connected to the joint.
- Based on loadpath. Run a structural analysis. Select one loadcase and a joint(s), RMB and *Add LJF*. Note that the LJF assigned refer to one loadcase only when you run the final analysis covering several loadcases.
- The LJFs can be calculated for individual braces or for all braces entering the selected joints.
- The LJFs (Axial, Out of Plane Bending<sub>OPB</sub> and In Plane Bending<sub>IPB</sub>) are stored on the journal file (you may verify details here) and in the model referring to the brace ends, and they are independent of brace local co-ordinate system (they are calculated referring to a plane defined by the chord and brace).
- When generating a finite element model, the LJFs are meshed as flexible hinges ( $k=1/LJF$ ) referring to the brace local x-axis and a plane defined by the brace and the chord. This means that LJF<sub>OPB</sub> and LJF<sub>IPB</sub> are independent of beam local co-ordinate system.

The example below shows how LJF is assigned to one beam end based on a pure geometry approach.

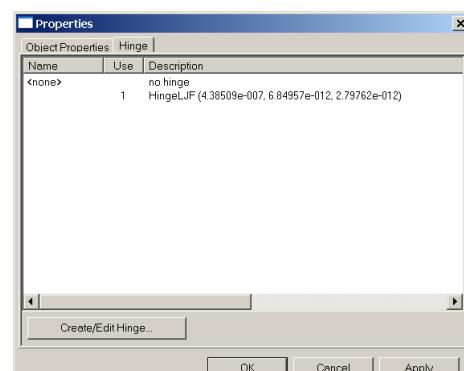


1. Select joint and beam, RMB, Add LJF to selected beam/segments

2. Verify details as found under the Message tab (see also the journal file)



3. Double click beam to enter segmented modelling modus, select joint, RMB and Properties



4. Verify from Property sheet the LJF details

The top screenshot shows a table titled 'Brace1' with columns: Name, X-End1 [mm], Y-End1 [mm], Z-End1 [mm], X-End2 [mm], Y-End2 [mm], Z-End2 [mm], Section, Material, Hinge-End1, and Hinge-End2. The bottom screenshot shows a table titled 'Brace1-End1' with columns: Name and Type, containing one entry for Brace1-End1.

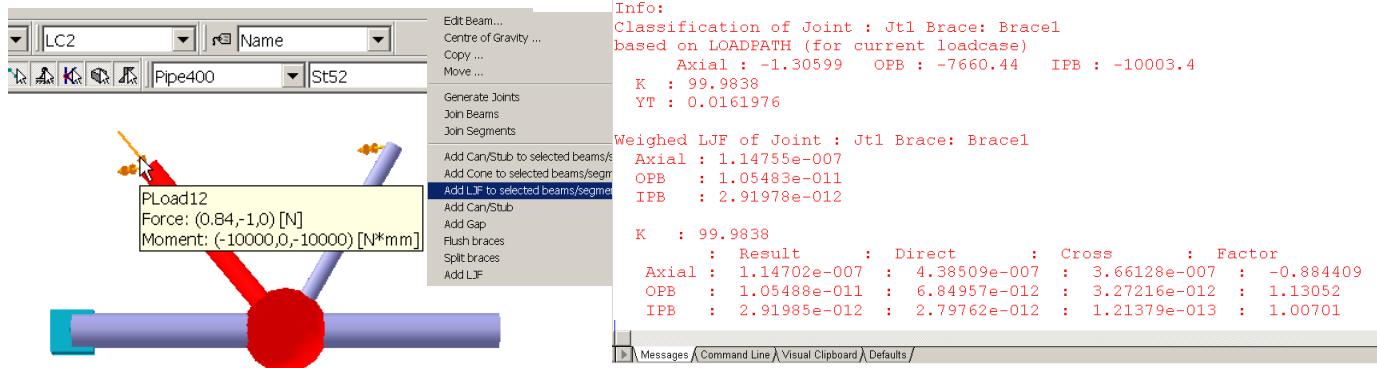
	A	B	C	D	E	F	G	H	I	J	
1	Name	X-End1 [mm]	Y-End1 [mm]	Z-End1 [mm]	X-End2 [mm]	Y-End2 [mm]	Z-End2 [mm]	Section	Material	Hinge-End1	Hinge-End2
2	Bm1	0.000	0.000	0.000	5000.000	0.000	0.000	Pipe800	St52		
3	Bm2	5000.000	0.000	0.000	10000.000	0.000	0.000	Pipe800	St52		
4	Brace1	5000.000	0.000	0.000	1786.060	3830.220	0.000	Pipe500	St52	Brace1-End1	
5	Brace2	5000.000	0.000	0.000	7500.000	4330.130	0.000	Pipe400	St52		

	A	B	C	D	E	F	G
1	Name	Type					
2	Brace1-End1	HingeLJF (4.38509e-007, 6.84957e-012, 2.79762e-012)					
3							
4							
5							

## 5. Verify from saved report

The example below shows how LJF is assigned to one beam end based on load path approach. It is assumed that structural analysis has been performed prior to the calculation of LJF and that you have selected a governing loadcase (in the example below LC2 has been selected). To verify, use the methods described above.



1. Run analysis, select loadcase, select joint and beam, RMB, Add LJF to selected beams/segments

2. Verify details under the Messages tab.

The following should be noted:

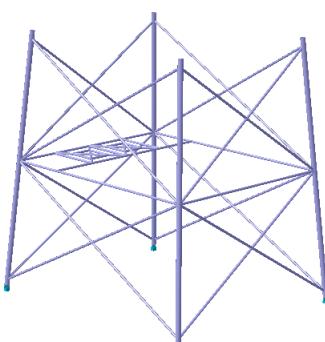
- When running an analysis after load path based LJFs has been created, the effect of these LJFs will be included in the analysis. Hence, if new LJFs are calculated based on an “updated” load path the LJF values will change accordingly.
- LJFs can only be assigned to start or end of a beam connected to a joint

- The default LJF range limitations for resulting LJF values (resulting LJF = direct\_term + loadratio x cross\_term) are as follows:
  - Minimum LJF = 0.1 x direct term
  - Maximum LJF = 5.0 x direct term
    - The direct term relates to the reference brace and the cross term is the effect of loading the second brace. The loadratio (factor) is the ratio of load action in the second brace divided by load action in the reference brace.
- When outside the range of applicability of geometric values, i.e. the  $\gamma$ ,  $\tau$ ,  $\beta$ ,  $\theta$  ratios, the limit values are used when deriving the LJFs.
- The minimum and maximum LJF range limitations can be reset from the command language interface by the commands:
  - GeniERules.LJF.setLimitMin(param, value);
  - GeniERules.LJF.setLimitMax(param, value);
  - GeniERules.LJF.setLimit(param, minvalue, maxvalue);
    - Available parameters are ljfAxial, ljfIPB, ljfOPB
  - Example – change Min values to 0.2 and Max values to 9.0 for all three components
    - GeniERules.LJF.setLimit(ljfAxial, 0.2, 9.0);
    - GeniERules.LJF.setLimit(ljfIPB, 0.2, 9.0);
    - GeniERules.LJF.setLimit(ljfOPB, 0.2, 9.0);
- LJFs are typically used in connection with planewise gap calculations (flush to chord wall). If you want to apply LJF to a X-joint, remember to split the braces (Select joint, RMB, *Split Braces*) after gaps have been added at the end of X-braces.

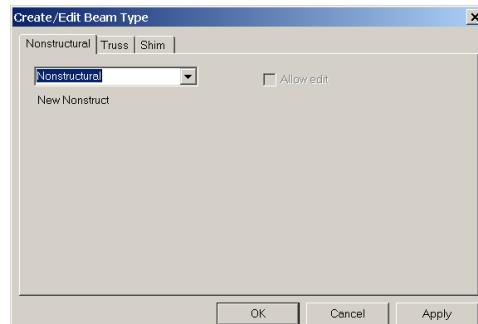
### 3.8.5 Non-structural beams, shims, and truss elements

These element types are special for jacket and topsides. The truss element (an element with no bending stiffness) is mainly used to model slings when lifting structure. The non-structural beams are not contributing with stiffness and they are normally used to compute wave loads. The shim elements are special connections between conductors and conductor frames whereby the conductor is free to move in vertical direction. How to apply these properties is described in the following.

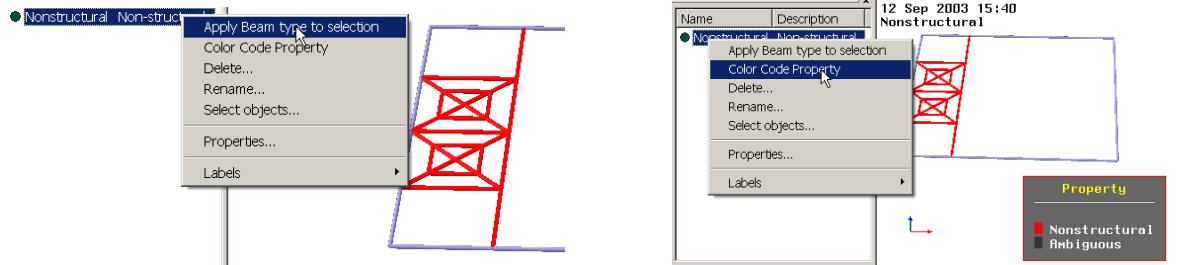
#### Non-structural beams



In this example the “conductor frame” is assigned as non-structural beams. This is achieved by a) making a beam type property called Nonstructural (*Edit/Properties/Beam Type*) and b) apply this to the relevant beams.

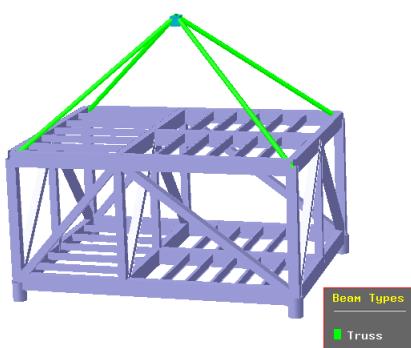


To assign the Nonstructural beam type to the relevant beams select the beams, RMB then *Properties* and assign the right property. Alternatively, select the beams, open up the Beam Type browser area, select Nonstructural and *Apply beam to selection*. To verify, use normal colour coding from browser or from the save report under Tab Beams.



	A	B	C	D	E	F	G	H	I	J	Flood
1	Name	X-End1 [m]	Y-End1 [m]	Z-End1 [m]	X-End2 [m]	Y-End2 [m]	Z-End2 [m]	Section	Material	Beam Type	Flood
34	Bm35	-24.063	8.438	30.000	-17.188	8.438	30.000	pipe06	mat1	Nonstructural	
35	Bm36	-24.063	2.813	30.000	-17.188	2.813	30.000	pipe06	mat1	Nonstructural	
36	Bm37	-24.063	8.438	30.000	-24.063	2.813	30.000	pipe06	mat1	Nonstructural	
37	Bm38	-24.063	-2.813	30.000	-17.188	-2.813	30.000	pipe06	mat1	Nonstructural	
38	Bm39	-24.063	-8.438	30.000	-17.188	-8.438	30.000	pipe06	mat1	Nonstructural	
39	Bm40	-24.063	-2.813	30.000	-24.063	-8.438	30.000	pipe06	mat1	Nonstructural	
40	Bm45	-17.188	8.438	30.000	-17.188	2.813	30.000	pipe06	mat1	Nonstructural	
41	Bm48	-17.188	-2.813	30.000	-17.188	-8.438	30.000	pipe06	mat1	Nonstructural	

## Truss



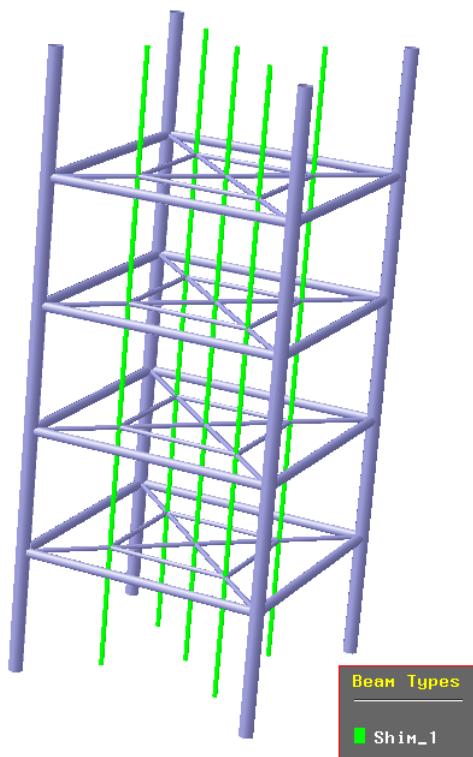
The lifting arrangement has been modelled with truss beam types. It is necessary to define Truss (*Edit/Properties/Beam Type*) as a separate property before they are applied to the beams. Such beam types should be used with care as they may lead to singularity during analysis if used wrongly.

To verify you may use colour coding from browser or from *View/Options/Model/Use property colour coding* or from the saved report under Tab Beams.

	A	B	C	D	E	F	G	H	I	J
1	Name	X-End1 [m]	Y-End1 [m]	Z-End1 [m]	X-End2 [m]	Y-End2 [m]	Z-End2 [m]	Section	Material	Beam Type
58	BM62	11.000	14.660	12.500	11.000	18.330	12.500	BOX8	St52	
59	BM63	11.000	18.330	12.500	11.000	22.000	12.500	BOX8	St52	
60	BM64	0.000	0.000	12.500	14.000	11.000	25.000	PIP10	St52	Truss
61	BM65	0.000	22.000	12.500	14.000	11.000	25.000	PIP10	St52	Truss
62	BM66	28.000	0.000	12.500	14.000	11.000	25.000	PIP10	St52	Truss
63	BM67	28.000	22.000	12.500	14.000	11.000	25.000	PIP10	St52	Truss
64	BM7	0.000	3.670	0.000	11.000	3.670	0.000	IH10	St52	
65	BM8	0.000	7.340	0.000	11.000	7.340	0.000	IH10	St52	

## Shim elements

This is a special variant of a spring that will allow for e.g. horizontal fixation between a conductor and its support. Prior to assigning shim functionality to the connections between beams it is necessary to define a shim element property with a given spring stiffness.

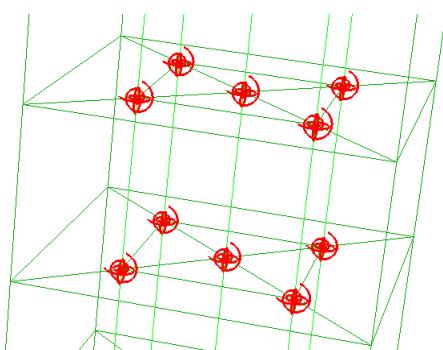


The conductors to the left are assigned a beam type Shim\_1. When a finite element model is created spring elements (with stiffness perpendicular to the conductors) are automatically assigned at all connections between the conductors and the structure. The exception is when the end of a conductor intersects with structure.

See next page for more details.

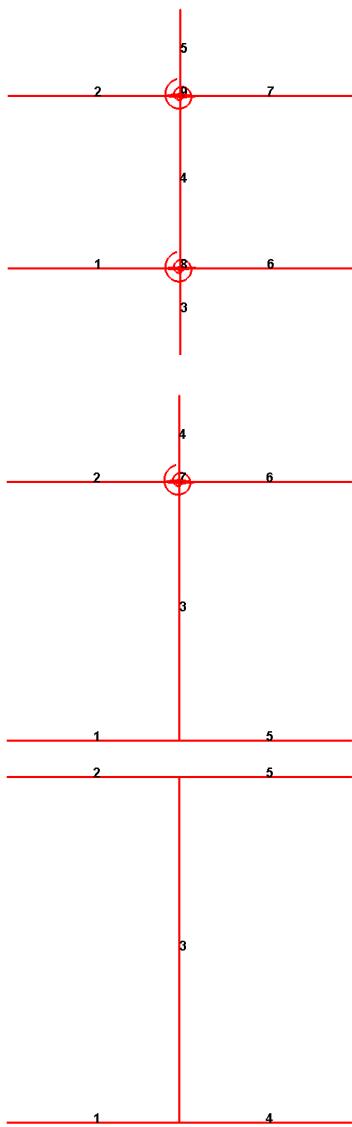


A shim property is manually created with stiffness 1.0E08 N/m.



You do verification of shim elements from regular colour coding, from the saved report Tab Beams, or after you have created a finite element mesh where the spring elements are shown with own symbols.

	A	B	C	D	E	F	G	H	I	J
1	Name	X-End1 [m]	Y-End1 [m]	Z-End1 [m]	X-End2 [m]	Y-End2 [m]	Z-End2 [m]	Section	Material	Beam Type
40	Bm46		7.500	7.500	20.000	2.500	7.500	20.000	Pipe200	St52
41	Bm47		2.500	2.500	0.000	2.500	2.500	25.000	Pipe200	St52
42	Bm48		7.500	2.500	0.000	7.500	2.500	25.000	Pipe200	St52
43	Bm49		5.000	5.000	0.000	5.000	5.000	25.000	Pipe200	St52
44	Bm5		0.000	0.000	5.000	10.000	0.000	5.000	Pipe400	St52
45	Bm50		2.500	7.500	0.000	2.500	7.500	25.000	Pipe200	St52
46	Bm51		7.500	7.500	0.000	7.500	7.500	25.000	Pipe200	St52
47	Bm6		10.000	0.000	5.000	10.000	10.000	5.000	Pipe400	St52



The vertical beam is assigned *Shim* and spring elements are assigned at each connection between structure and the shim structure. Hence two spring elements are created.

The numbers and symbols refer to the finite element model.

The same configuration as above, but the bottom point of the Shim element is coinciding with the lower horizontal beam. Hence one spring element is created.

In this case, both ends of the Shim coincide with regular beams. Hence no spring elements are created.

## **Inner Beam**

### **Fully coupled**

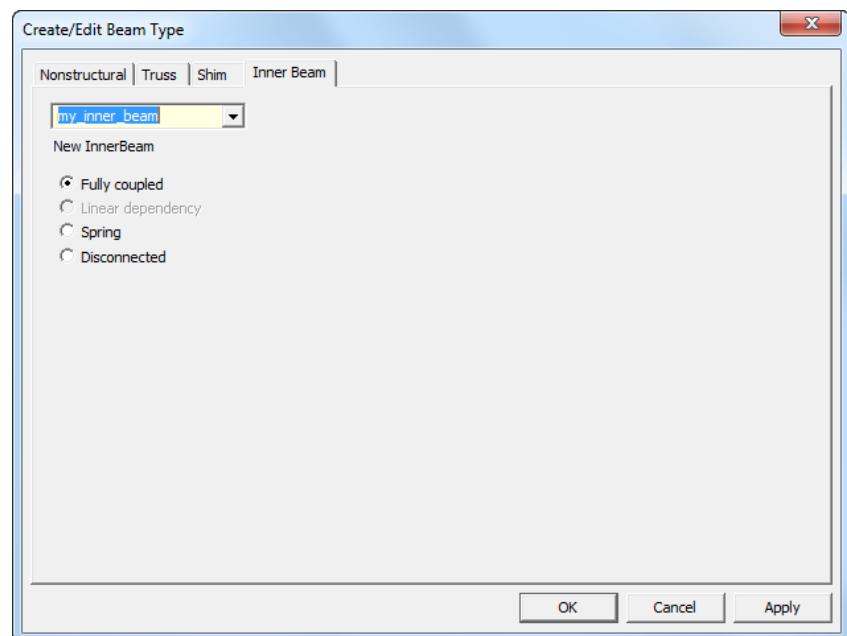
All nodes are fully coupled in six degrees of freedom

### **Linear dependency**

Feature is not available

### **Spring**

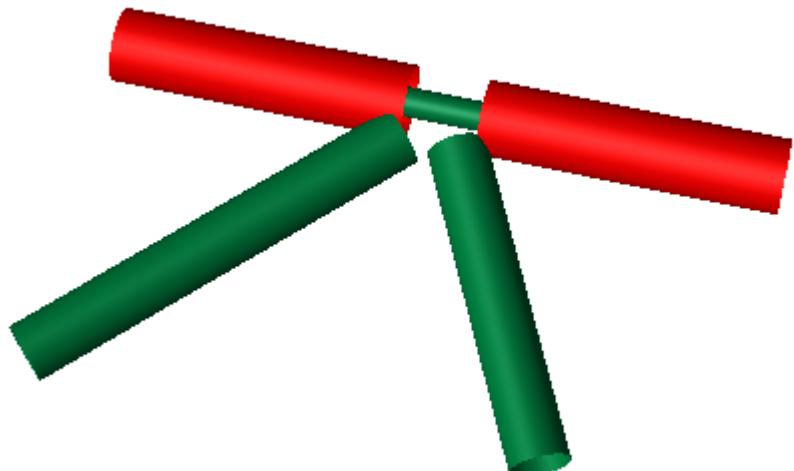
All nodes in the inner beam are connected to corresponding nodes in the outer beam, using the specified spring stiffness per length unit.



### **Disconnected**

Inner beam is not connected to nodes in outer beam.

In the illustration to the right the inner beam is disconnected from the outer beam. This is clearly visible in mesh view.

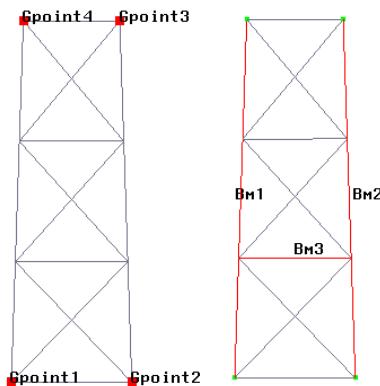


### 3.8.6 Reference point modelling of beam structures

Using reference point modelling is different from normal modelling in the way that the technique always refer to an object in the logging of a command, for example a beam definition `Bm1 = Beam(GPoint1, GPoint2);` where `GPoint1` and `GPoint2` have been defined as guiding points. The normal logging is typically `Bm1 = Beam(Point(0 m,0 m,0 m),Point(10 m,0 m,0 m));`, in other words the co-ordinate values are referred to.

Reference point modelling may be used when establishing a model that you want to change by editing the journal file. The example in the following shows how a small frame will adjust based on changes of two of the reference points. Observe that there are no mechanisms for priority settings when changing a model like this, when GenIE can not create a new model based on changes it will give a warning on this.

To activate the reference point modelling make sure that the *Reference point modelling* in the Toolbar *Object Types* is activated.

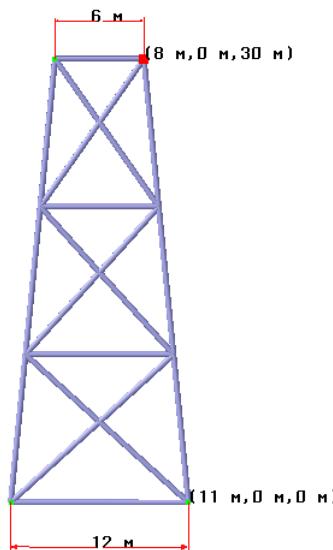


The frame to the left has been created as follows:

`Gpoint1 (0,0,0)`  
`Gpoint2 (10,0,0)`  
`Gpoint3 (9,0,30)`  
`Gpoint4(1,0,30)`

The beams have been graphically modelled and are typically logged as follows:

`Bm1 = Beam(Gpoint1, Gpoint2)`  
`Bm3 = Beam (Bm1.intersect(ZPlane3d(10)),Bm2.intersect(ZPlane3d(10)));`  
 and always referring to the name of the object (Bm1&Bm2 in this case).



When editing the journal file and changing the values as shown below, the model to the left is automatically created when reading in the journal file to a new workspace.

```
/// Creating Guiding Points for use in parametric modelling
/// Changing x value for all guiding points
///
Gpoint1 = Point(-1,0,0);
Gpoint2 = Point(11,0,0);
Gpoint3 = Point(8,0,30);
Gpoint4 = Point(2,0,30);
///
/// Creating beams using reference point modelling
///
Bm1 = Beam(Gpoint1,Gpoint4);
Bm2 = Beam(Gpoint2,Gpoint3);
Bm3 = Beam(Bm1.intersect(ZPlane3d(10)),Bm2.intersect(ZPlane3d(10)));
Bm4 = Beam(Bm1.intersect(ZPlane3d(20)),Bm2.intersect(ZPlane3d(20)));
```

Reference point modelling is primarily intended for frame modelling. It is not possible to journal plate reference points. This means that a plate must be referred to reference points in guide planes and beams. It also requires that the journal file is edited and read into a new workspace to gain effects from the changes.

## Curved structure

Curved structure is often created using guide geometry, but can also be created using existing linear or curved structure as basis. For example, an existing curved or straight beam can be used as an implicit guide curve. The explicit guiding curves (lines, splines, elliptic arc, circle, polycurve and model curve) are referenced from two alternative methods for creating curved shells, i.e. the skin and cover operations. The skin and cover operations typically generate spline surfaces.

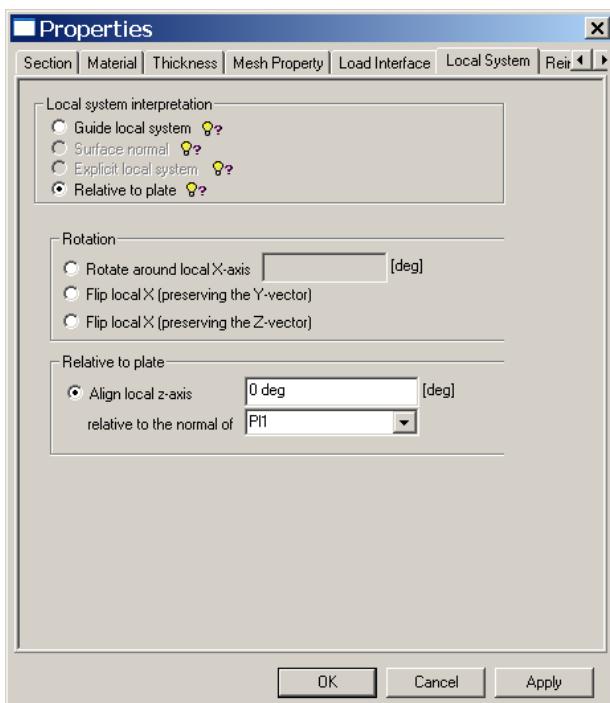
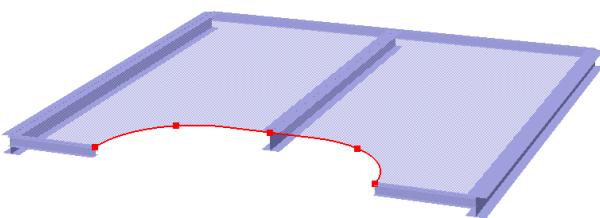
The *skin* operation assumes a series of curves that may be open or closed. A surface is adapted to stretch over the curves in the specified sequence; much like a skin can cover the ribs in a canoe. Notice that, in GeniE, the skin operation leaves a continuous surface, there are no sharp bends over the ribs.

The *cover* operation assumes a closed curve where the "hole" is to be filled. Again, a surface is adapted to stretch across the hole much like the skin on a drum. If the closed curve is non-planar, a spline surface will be used to interpolate.

In the following a number of practical examples are given on how you can create curved geometry. Additional examples may also be found in Chapter 3.20.

Observe that if you do not have access to the *Curved Geometry Extension* in GeniE you can not see any of the commands related to curved geometry.

### 3.8.6.1 Curved beams and eccentricities



A model curve has been defined along the curved plate edge. To insert a curved beam you select the model curve, RMB and *Create Beam*. A curved beam is now inserted in line with the plate (i.e. the origin of the local co-ordinate system is at same elevation as the plate).



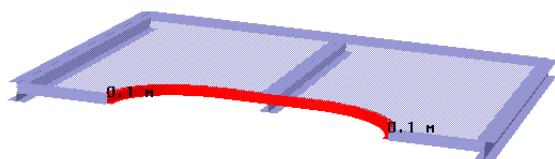
You may also control the orientation of the curved beam. By selecting the beam, RMB and then select *Properties/Local system* you have access to the dialogue showed to the left.

In this case the beam has been chosen to have a local coordinate system relative to a plate. In this case the local z-axis is 0 degrees relative to the normal of the plate P11.

To insert eccentricities, select the curved beam Bm1 and type the following command in the Command Line Interface window:

```
Bm1.setBeamOffsetLocal(Vector3d(0,0,-Bm1.section.cogZ));
```

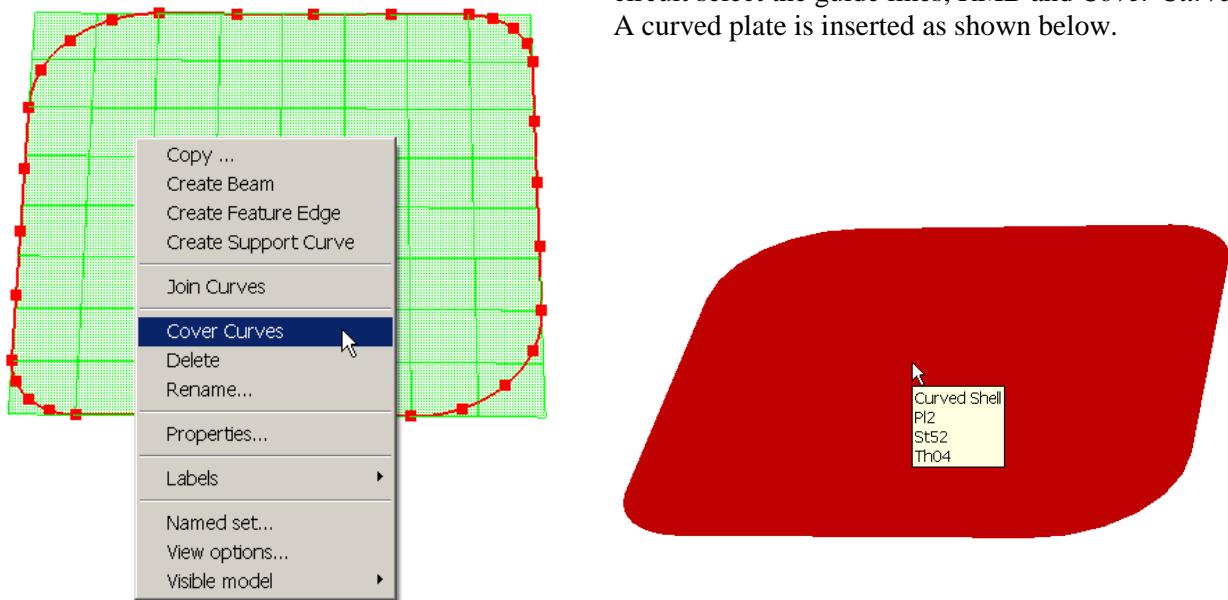
This command will insert an offset equal to the distance from local centre of gravity to the beam edge (or the half height of the beam). In this case an offset of 0.1 m is applied since the profile height is 0.2 m. If you want to move below the plate, then add half plate thickness to the offset vector.



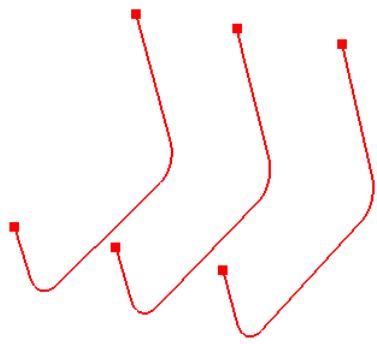
The curved beam Bm1 is now flushed to the plate.

### 3.8.6.2 Fill a plate inside a closed guide curve circuit

A closed circuit has been created by guide lines and guide arc elliptic. To insert a curved plate inside the circuit select the guide lines, RMB and *Cover Curves*. A curved plate is inserted as shown below.

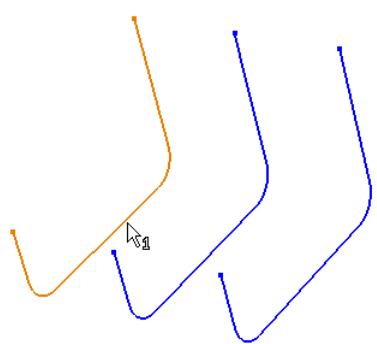
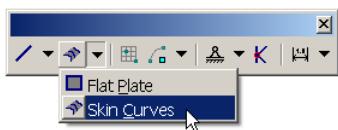


### 3.8.6.3 Insert a curved beam and plate between guide geometries

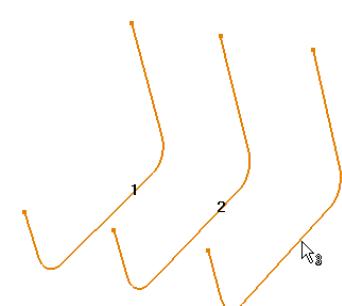


Three guide lines have been established from a) **Insert Guiding Geometry/Guide Line** and **Guide Arc Elliptic** and b) selecting lines, RMB and **Join Curves**.

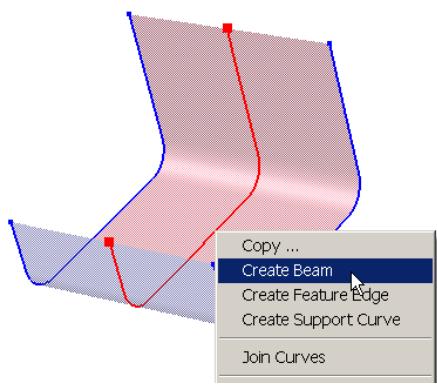
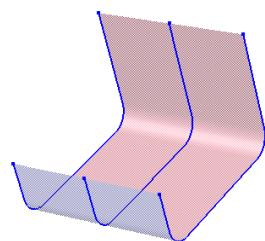
When inserting a plate between the guide lines, the skin feature must be used. Make sure that the *Skin Curves* is activated from Toolbar Object Types.



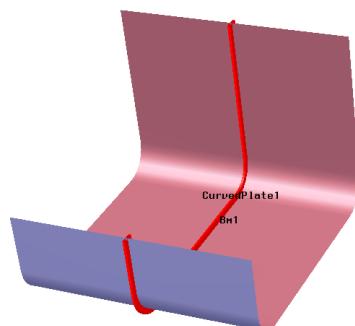
You are prompted for input when modelling in the graphic window. In the picture to the left, the first line is prompted for. Click the line and the mouse tooltip prompts for line number two. If you have other objects than guiding lines, it may be advantageous to use filters to help you pick lines only.



Guide lines one and two are already given, and GeniE waits for definition of line number 3. When this has been done, you need to click line number 3 again to complete the command input. A curved plate is then inserted between the lines.



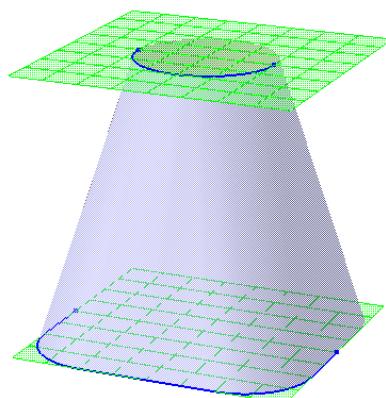
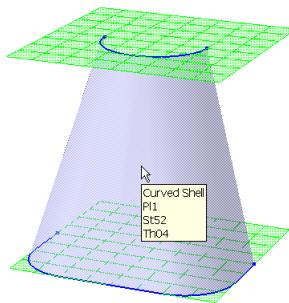
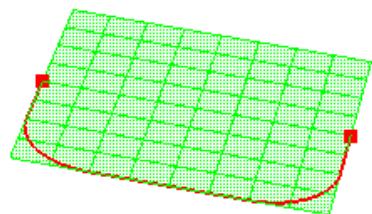
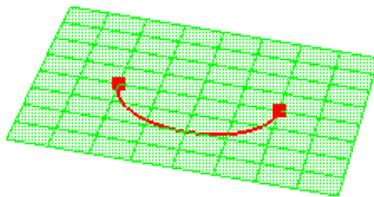
To insert a beam select the guide line, RMB and *Create Beam*.



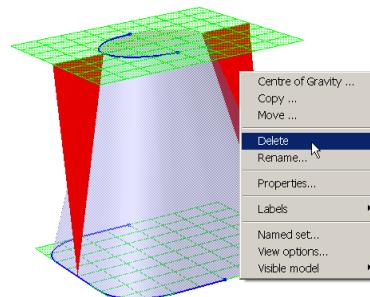
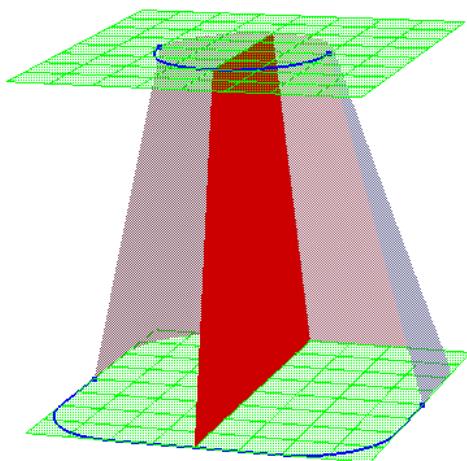
### 3.8.6.4 Transition between a rectangular and circular shape

This example shows how to make a transition between a rectangular and a circular shape like in the transition from the pontoon to the column in a semi-submersible. Focus is on creating the hull, but inside plate structure has also been modelled. To insert curved beams, model curves need to be inserted so that they can be referred to when creating beams. Eccentricities may be applied from the Command Line Interface as described above.

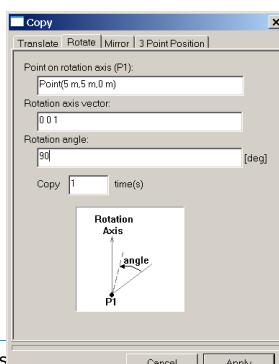
Two guide curves have been inserted to represent half the transition area. This example assumes that the structure is symmetrical in the vertical plane. A curved plate is inserted when using a skin operation and referring to the bottom and upper geometry curves.



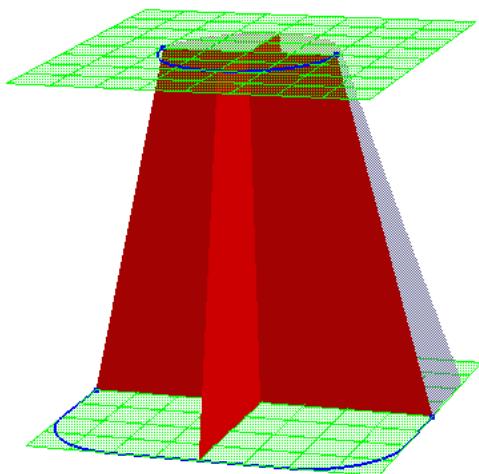
The outer hull plate is now created by copy and mirror operation. A vertical inside plate is inserted by referring to upper and lower guide plane. To remove superfluous material, the plate must be split (Tools|Structure|Split) and delete the parts outside the hull.



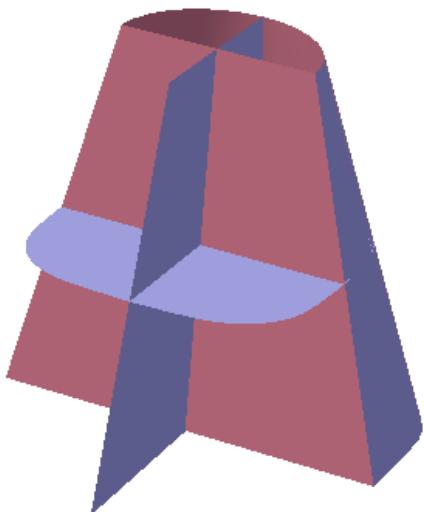
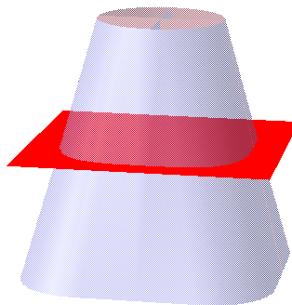
The vertical plate is now trimmed to the hull by a combination of split and delete.



To insert a new vertical plate perpendicular to the first, select the first, copy and rotate 90 degrees.



Both vertical plates are now trimmed to the hull. To insert a horizontal plate a new guide plate has been made and the plate corners snapped to it. To trim it, select the plate, split and delete the superfluous parts.

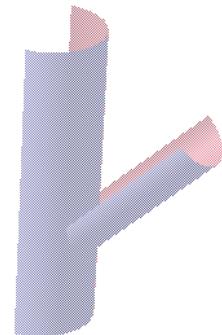
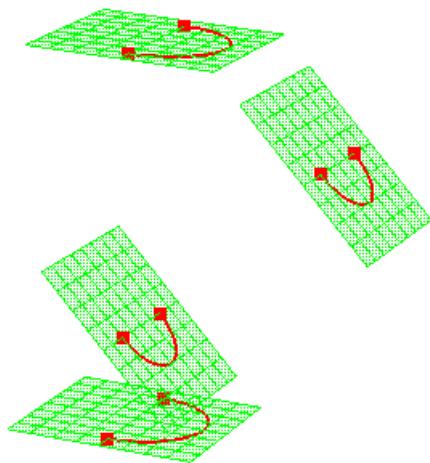


The transition part that includes trimmed vertical and horizontal plates.

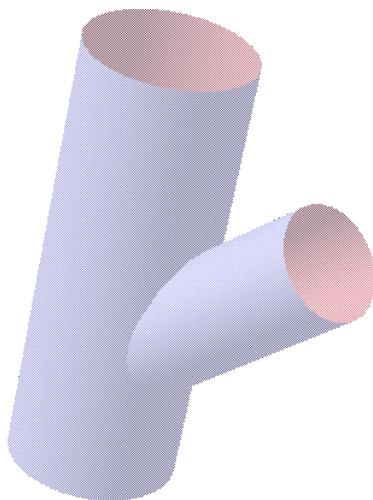
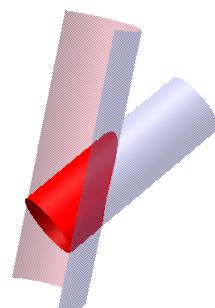
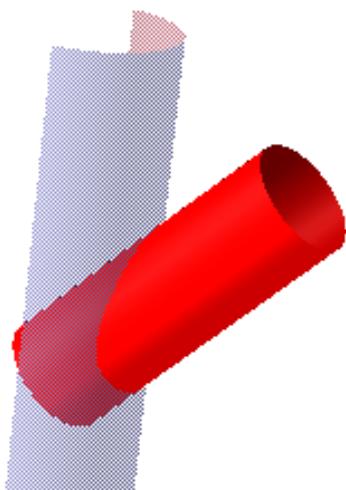
### 3.8.6.5 Two tubes intersecting each other

This example focuses two tubes that intersect each other. Trimming has also been done.

Guide curves have been inserted to define boundaries of each half cylinder. Curved plates are inserted by using skin curve operation.

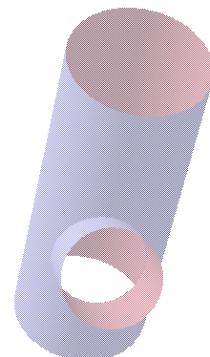


Copy and rotate (180 degrees) the lower part of the inclined tube gives a complete brace. To trim the brace it is necessary to split it and delete interior parts.



The complete vertical chord is now created by copy and rotate (180 degrees) the first chord part.

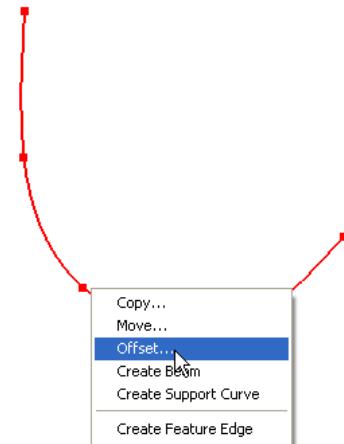
A trimming (split and delete) of the chord will result in a hole as shown on picture to the right.



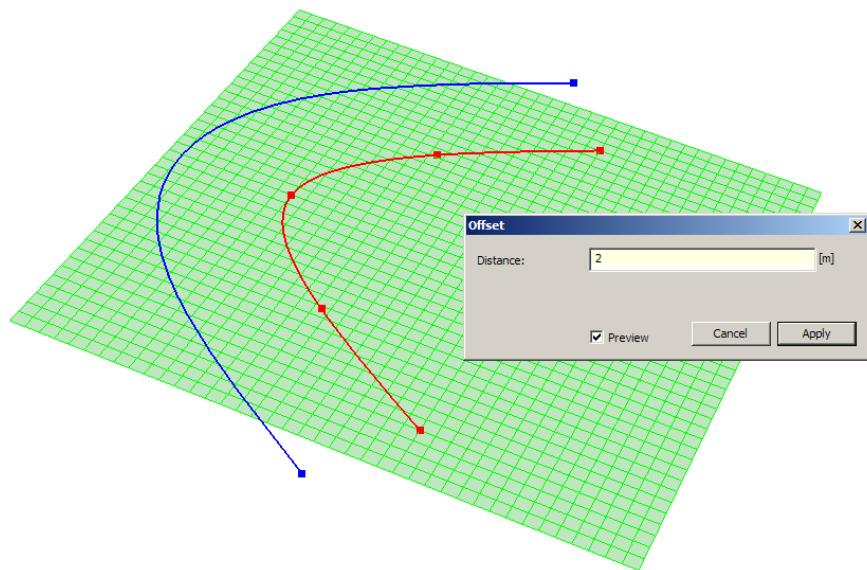
### 3.8.6.6 Offset

Offset lets you create an offset copy of a planar curve or a curved shell.

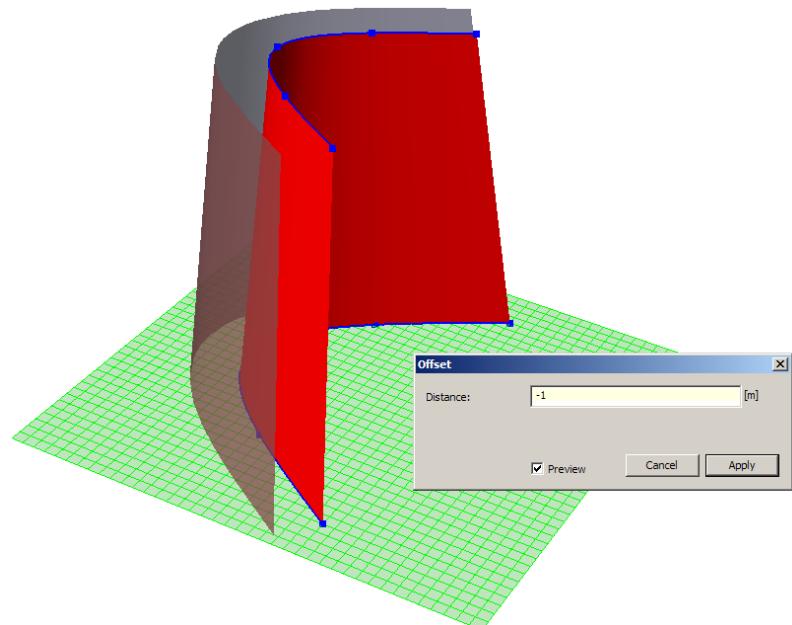
Open the offset dialog by rightclicking on your model and select *Offset*.



By using offset a planar curve is copied 2 m in the direction of the curve's normal vector.



By using offset a curved shell is copied -1 m in the direction of the shell's normal vector.



## Healing models

GeniE comes with several ways of verifying the quality of your model. These features are available from the **Tools|Structure|Verify**. In addition GeniE allows you to heal badly connected beams in your model.

Tolerant modelling is the default behaviour in GeniE, but this may not always remedy all tolerance problems. When creating e.g. a beam, the tolerant modelling will try to snap the start and end positions of the beam to existing structure. Depending on the tolerances you have specified, you may have problems with two beams almost intersecting or where an already modelled beam almost intersects the new beam. Similarly, when importing a structure modelled in other systems you may also have tolerance problems.

Model healing analyses each beam in the model and searches for candidates that should intersect the beam in question. To do this it is necessary to categorize the beams. The healing operation assumes that a beam which beam ends touch the interior of another beam is categorized as less important than the beam it touches.

Model healing is highly time-consuming and run time behaviour of such is not feasible. Therefore, this feature is available either as part when exporting a clean journal file or by manual selection.

- Export a clean journal file (**File|Export|GeniE journal file (JS)**). After categorizing all the beams in the model, the most important beams (typically jacket legs) are exported first. Beams that should snap to the jacket legs are exported next, and so forth until all beams in the model have been exported. When reading the journal file into a new workspace, the model will automatically be snapped together.

Export including model healing is available from the script language. You may do it from the Command Line Interface or when e.g. importing a journal file. The necessary commands are:

`JsExporter.hierarchicExport=true;`

`JsExporter.overrideModelTolerance(0.03);` In this case the new tolerance is set to 0.03m

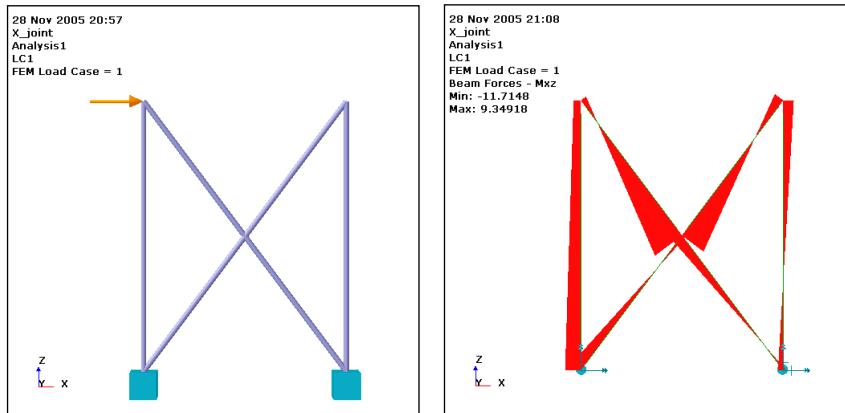
`JsExporter.DoExport(file name);`

The resulting journal file (the clean journal file) now contains new headings (Structure, level *n*) indicating the categorization of beams (i.e. in the order from more important to less important beams).

- Model healing by manual selection. Select the relevant part of your model and choose **Tools|Structure|Geometry|Heal Structure**. After categorizing all the beams in the model, all the categorized beams are unconnected from the model. Then the most important beams are inserted into the model first, and the less important beams until all beams are inserted. Since this is a selection sensitive operation it may not solve all problems; beams may become unconnected in other places of the model (not part of the selection subject to model healing). You should use manual model healing in connection with Tools|Structure|Verify. Typically beams with short edges or unexpected free ends are candidates for model healing.

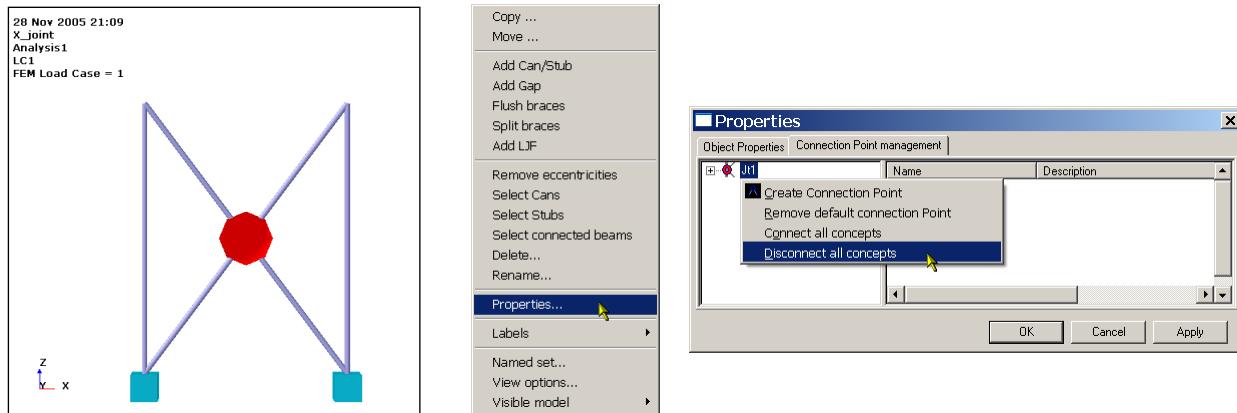
### 3.8.7 Release X-joints

It is possible to release X-joints so that the beams are free to move independently. The example below shows a model prior to releasing the X-joint and how to release the X-joint.

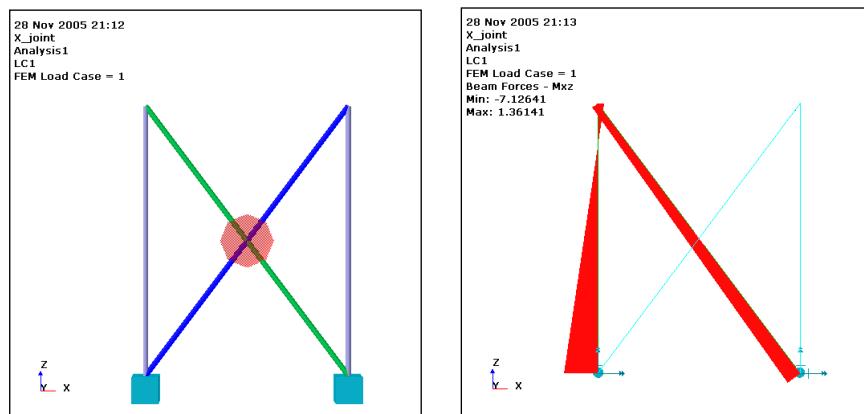


The beams in the X-joint are connected and both beams have displacements and bending moments because of the horizontal point load.

To release the beams in the X-joint the procedure is as follows:



Insert a joint and select *Properties*. From *Connection Point Management* you release the beams in the X-joint by *Disconnect all concepts*. The beams may be connected again by selecting *Connect all concepts*. When you have disconnected the beams, you will see a colour coding of the released beams. If you later double click on the joint you will also see the same (you are then in segmented modelling view).



When the beams are disconnected in the X-joint one of the beams is deflected and has a bending moment due to the horizontal point load.

### 3.8.8 Point-point connection

Point-Point connections give you:

- Full control of connection (proper stiffness and load transfer)
- No need for using super-element modelling
- Easy transfer to non-linear analysis – no need to remodel connections

The pile and the leg are only connected in the joint. In this illustration the pile is shown in red and the leg in green. The yellow point-point connection symbol is in the same location as the joint.



#### 3.8.8.1 Applications of point-point connection

- Conductors and conductor frames (shim elements)
- Overlapping members (double beams)
- Pile in leg - how and where they are connected

#### 3.8.8.2 Importing model data from SACS

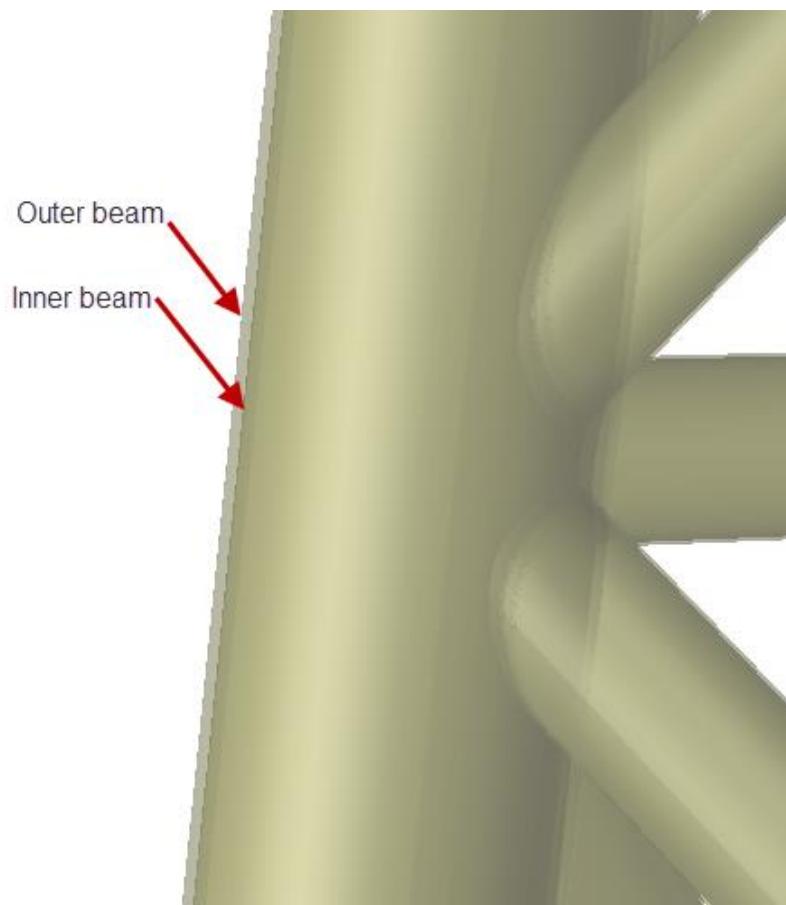
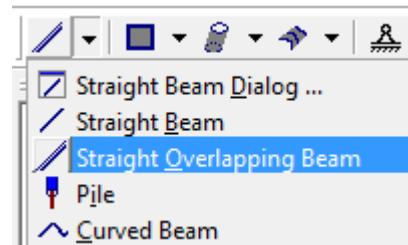
When importing model data from SACS, point-point connections will automatically be created for:

- Connections between leg and inner pile
- Connections between conductor and conductor frame

### 3.8.9 Modelling a pile inside a leg - ungrouted case

The following section will show you how to model an ungrouted pile inside a leg.

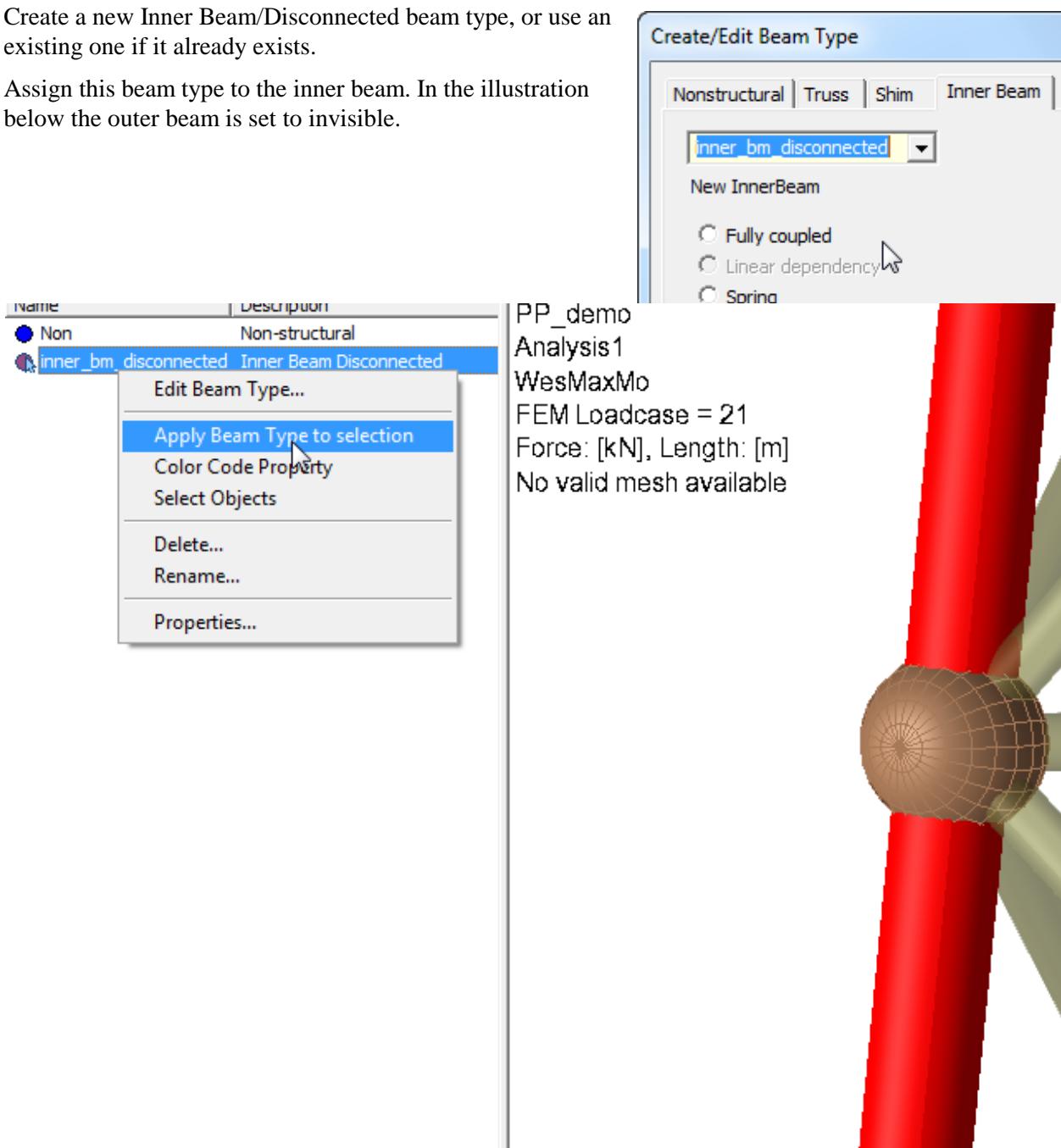
When creating a point-point connection between a leg and an inner beam, the inner beam needs to be created as a straight overlapping beam.



The illustration shows the beams partially visible, and the inner beam inside the outer beam (jacket leg) can be seen.

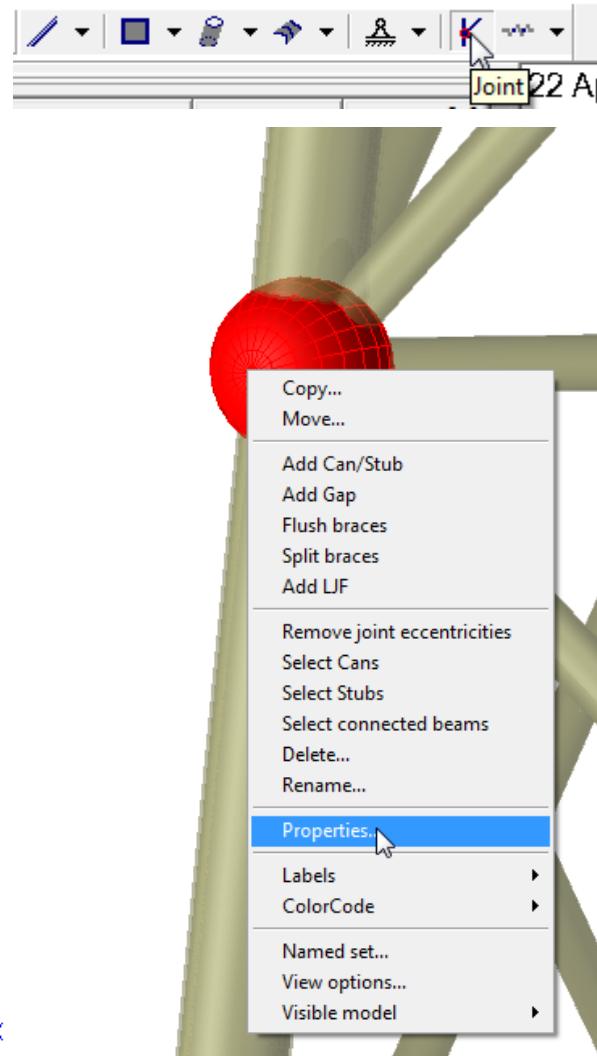
Create a new Inner Beam/Disconnected beam type, or use an existing one if it already exists.

Assign this beam type to the inner beam. In the illustration below the outer beam is set to invisible.



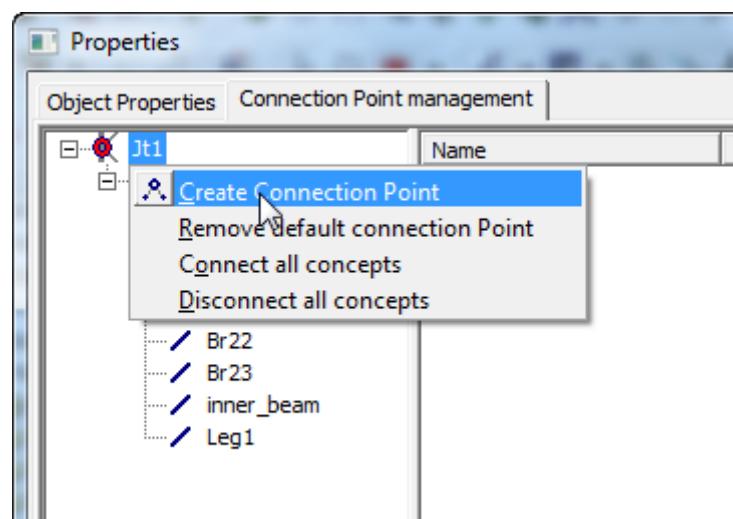
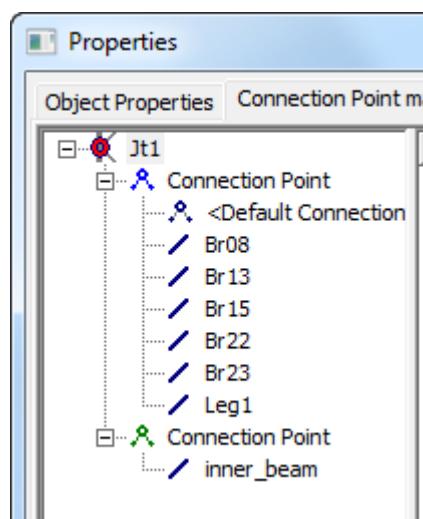


Insert a joint in the position where you want to insert the shim connection, or use an existing joint if there already is a joint in this position.



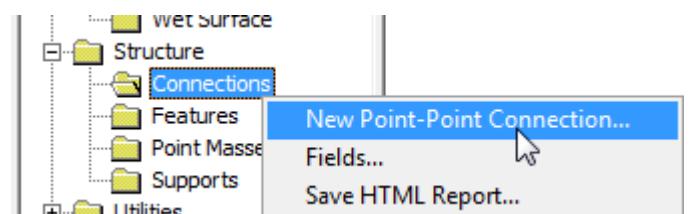
Rightclick the joint and select “Properties”. Make sure that the inner beam is located under a separate connection point.

If necessary a new connection point can be created by right clicking the joint in the dialog, “Jt1” in the example below. Drag the beam you want to connect into the new connection point that you created.

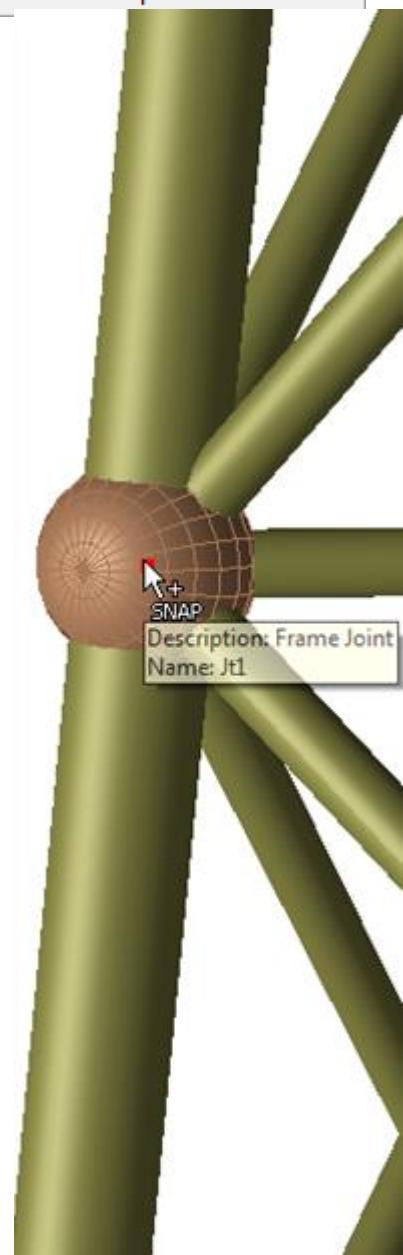
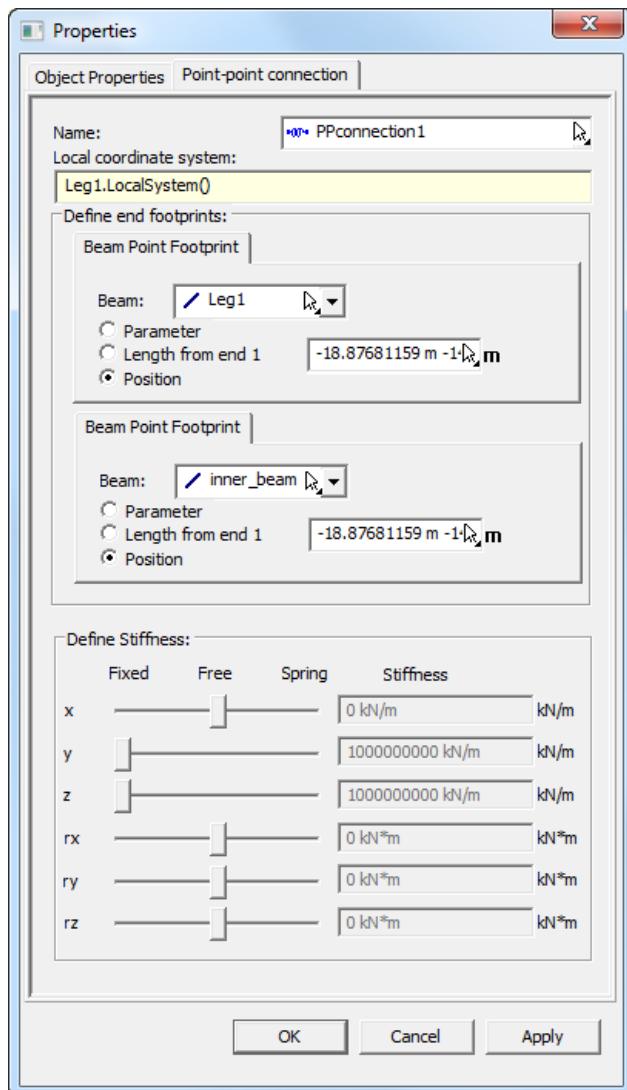


Specify shim springs by selecting “New Point-Point Connection” from the browser, under Structure/Connections.

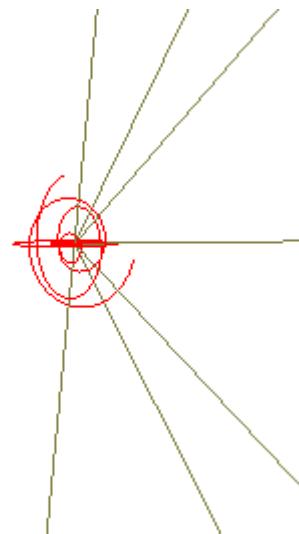
Click in the first “Beam” field, the one containing “Leg1” in the illustration below. By clicking twice on the joint, GeniE will now automatically select the outer beam as the first beam and the inner beam as the second beam.



Remember to use the leg beam’s local systems as “Local coordinate system”, and to release motion in the local X-direction.



By switching to wireframe mode and setting the joint to invisible, you can see the point-point connection symbol.



### 3.8.9.1 Troubleshooting

#### Same node used two times for spring.

- Verify that the outer and inner beams are properly disconnected.
- Verify that the spring is defined from the inner to the outer beam.

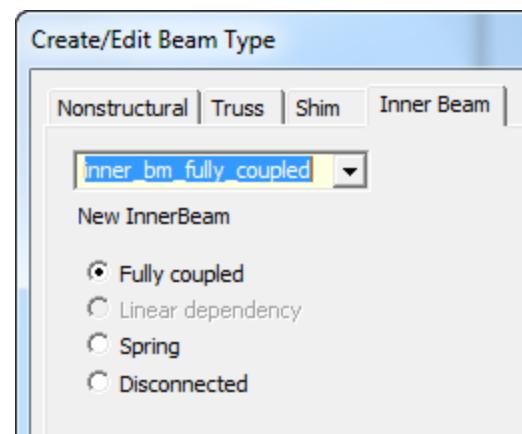
#### Singularity in FEM solution.

- Verify that there is no disconnected nodes or elements in the FEM model.
- Verify that the model is sufficiently constrained:
  - Are conductors constrained from spinning about the local X-axis?
  - Are the piles constrained against motion along the jacket legs at the top of pile?

### 3.8.10 Modelling a pile inside a leg – grouted case.

The following section will show you how to model a grouted pile inside a leg.

Create a new beam type Inner Beam / Fully Coupled and assign it to the pile inside the leg.

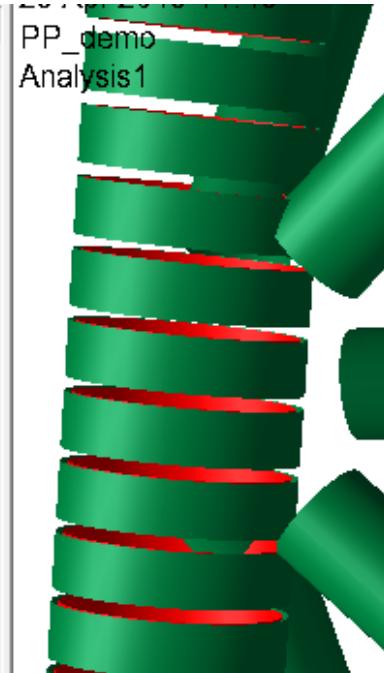


Specify a mesh density of e.g. 1 m for the outer and inner beams.

The outer and the inner beam will be connected at each mesh node.

In the illustration a mesh density of 1 m has been applied to the outer and inner beam. The inner beam is shown in red and the outer beam in green.

Name	Description	Mesh Density [m]
leg_pile_mesh	Mesh Density	1



### 3.8.11 Importing and exporting a model

GeniE can import a number of other finite element models and CAD models. Similarly, it can export to CAD and to other SESAM programs. The features are available from the pulldown menu **File/Import** and **File/Export**.

#### 3.8.11.1 From/to XML

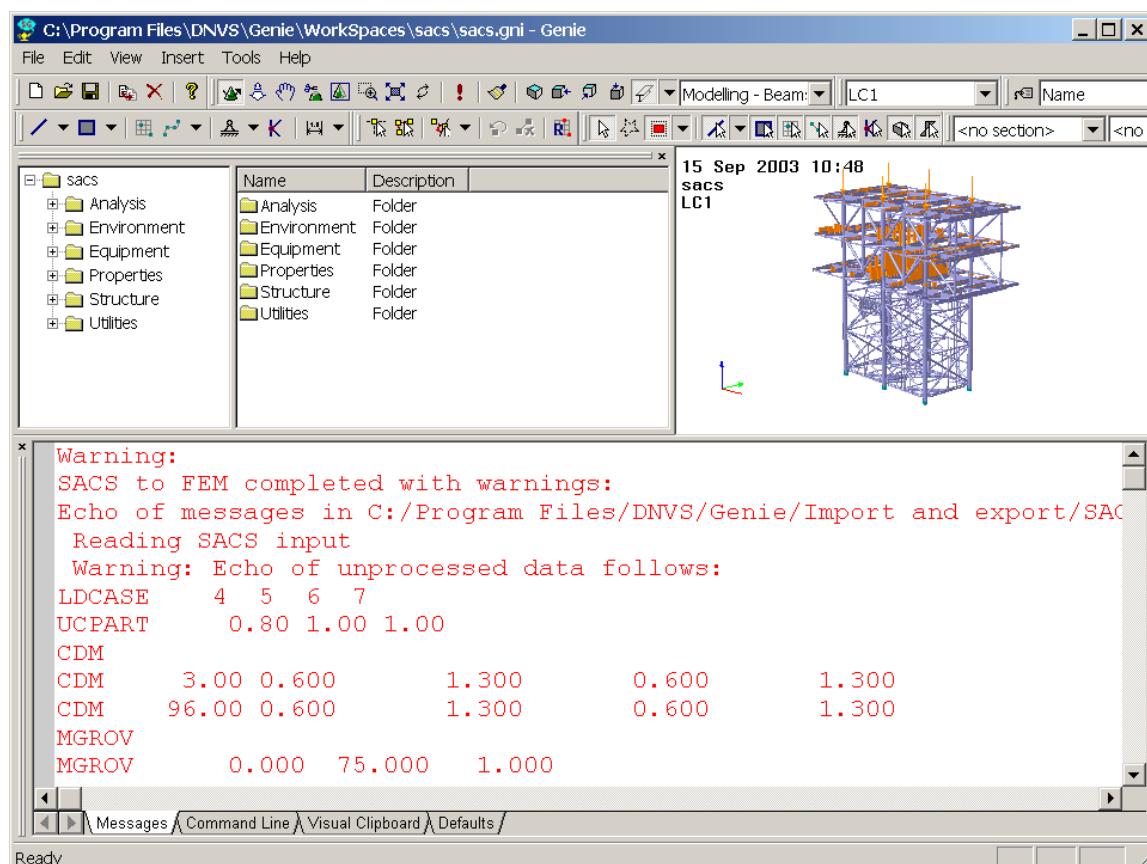
See the Import and Export chapter 10 in Volume 3

#### 3.8.11.2 From/to FEM

See the Import and Export chapter 10 in Volume 3

#### 3.8.11.3 From Sacs and StruCad3D

When importing a model from Sacs or StruCad3D the file formats \*.inp and \*.s3d are used. These models are first translated to a finite element model (temporarily FEM file) that is converted to a concept model. You get a log of items not translated to FEM-format under the Message tab area. This information is also stored in a separate file <import name file.CHK> in the same catalogue as the \*.inp or \*.s3d file (the full file path is given under the Message tab area).



When

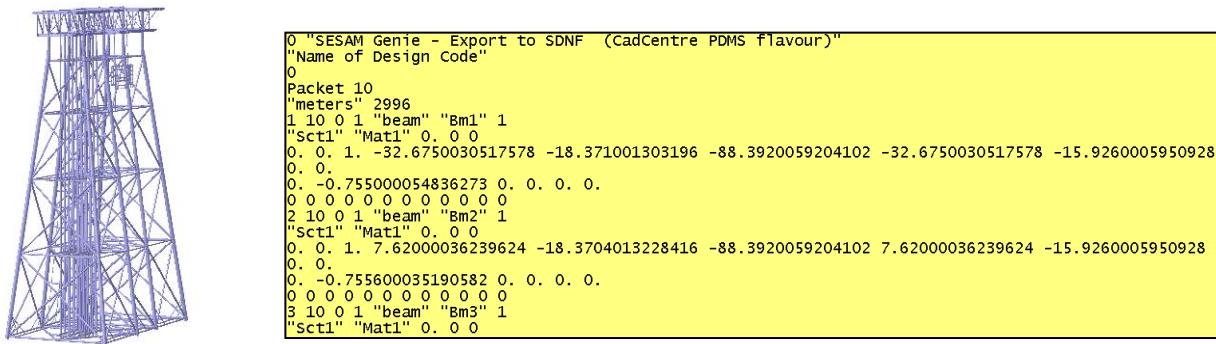
importing a SACS or StruCad3D model you should specify which units is part of the model. The units may be specified when activating the **File/Import/SACS File** or **File/Import/STRUCAD File**. The units are brought over during the FEM-file conversion prior to the actual import into GeniE

### 3.8.11.4 From other CAE systems

It is possible to import models from other systems as well provided they can be converted to a FEM model. Examples of such may be Ansys and Strudl models when these are converted to FEM models in Prepost.

### 3.8.11.5 From/to CAD

GeniE can import and export data from CadCentre PDMS and Intergraph PDS using the file format sdnf (\*.sdn). It is required that there exist a journal file containing the section and material properties if you want an automatic upload of these to the respective beams when importing the data. If not, all beams are visualised as lines since they have no knowledge about the belonging properties. The steps when importing such a model are thus a) **File/Read Command File** and b) **File/Import/Cadcenter PDMS**. Similarly, when exporting a model the command is **File/Export/Intergraph PDS**. The example below shows an imported PDMS model and a part of the model data found on the \*.sdn file.



You may also exchange data with CAD systems supporting the SAT file format from Spatial Technology. Note that ACIS SAT files containing solid models are not generally supported. When such models are encountered, a warning message is issued and the faces in the model are converted to double sided. This should however not be confused with an idealization for FEM analysis. In such cases, the model must typically be manually changed or re-modelled from scratch.

### 3.8.11.6 Create a journal file neutral storage

You may import from other models and create a journal file for later usage in GeniE. You do this by using the command **File/Export/GeniE journal file (JS)**. This is the same as to create a clean journal file. GeniE will now create a journal file that will reproduce the model.

```
0 "SESAM Genie - Export to SDNF (CadCentre PDMS flavour)"
0 "Name of Design Code"
0
Packet 10
"meters" 2996
1 10 0 1 "beam" "Bm1" 1
"Sct1" "Mat1" 0_0
0_0_1_-32.6750030517578 -18.371001303196 -88.3920059204102 -32.6750030517578 -15.9260005950928
0_0_0_0_-0.755000054836273 0_0_0_0
0_0_0_0_0_0_0_0_0_0_0_0
2 10 0 1 "beam" "Bm2" 1
"Sct1" "Mat1" 0_0
0_0_1_7.62000036239624 -18.3704013228416 -88.3920059204102 7.62000036239624 -15.9260005950928
0_0_0_0_-0.755600035190582 0_0_0_0_0
0_0_0_0_0_0_0_0_0_0_0_0
3 10 0 1 "beam" "Bm3" 1
"Sct1" "Mat1" 0_0
```

It is thus possible to import data from several sources (e.g. FEM, SACS, StruCad3D) into the same GeniE model and create a new journal file for the combined model.

For models consisting of straight beams and regular plates the clean journal file is ideal for neutral storage. Please note that any history is disregarded.

The clean journal file is divided into parts each starting with comments; hence it is easy to understand the journal file.

See also Chapter 3.6.8 for exporting a healed model to a clean journal file.

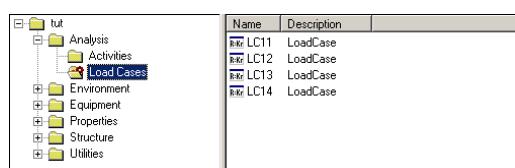
## 3.9 Loadcases and load combinations

Load generation is done in GeniE either by placing equipments on the structure or by modelling explicit loads like point and line loads. Each equipment contains information about mass and footprint so that it can create line loads when subject to a constant acceleration field.

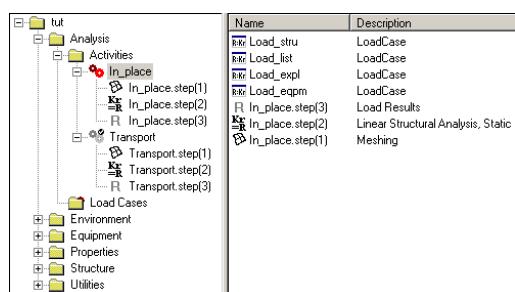
A loadcase may be used in a linear structural analysis where loads are being analysed. For a dynamic analysis where the mass model is of equal importance a loadcase may be treated as an addition to the structural mass. This is further explained in the Chapter describing how to make a finite element model.

It is possible to use the same structure concept model for different analyses in one GeniE workspace (or project). For the different analyses it is possible to have different loadcases, but this version of GeniE does not allow for different boundary conditions or changes in the structure model. If you want to run GeniE using with several analyses you should insert the loadcases per analysis. On the other hand, if you want to run one analysis only, you may define your loadcases in the folder termed loadcase.

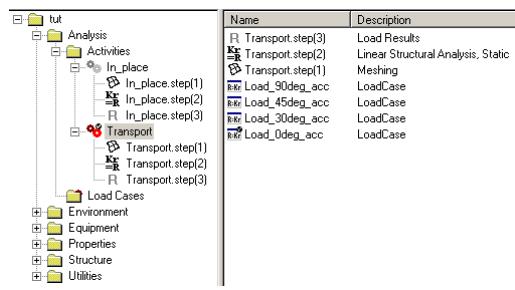
See Chapter 3.13 for more details on multiple analyses.



One analysis only will be used; hence loadcases may be located under folder *Load Cases*. You may also store the loadcases under the activity folder you defined in catalogue *Activities*.



Two analyses will be run. The activities *In\_place* and *Transport* have different loadcases. The figure to the left shows that loadcases for activity *In\_place* are stored under the activity folder.



Similarly, the loadcases for activity *Transport* are stored under this folder.

In the following it is assumed that you will run one analysis only and store the loadcases under the folder *Load Cases*. The loads are inserted and documented the same way if located under the folder *Activities* <your input text>.

### 3.9.1 Some principles for mass and load calculations

GeniE allows you to accurately represent the masses and their position by modelling equipments. In addition GeniE can read in similar information from weight lists in an automated way. Finally, you may decide to use explicit loads like point load, line load, pressure load, temperature load, and prescribed displacements.

There are some differences between these load sources that you should be aware of:

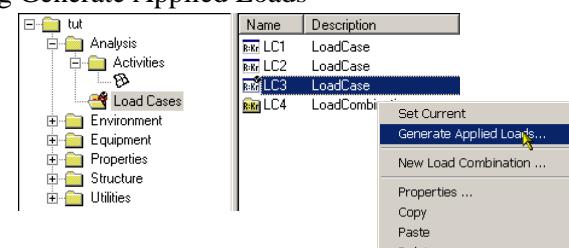
- The equipments will create a load or a mass on the structure. Load and mass will be calculated based on the equipment's footprint, its centre of gravity, the load interface, and whether you want equilibrium of applied mass versus calculated forces.
- Objects from a weight list will create a load or a mass to the structure. These will be calculated based on the object's physical size (the box), its mass, and its position. Wherever the box of the object hits a structural member (beams) a constant line load and mass will be generated. In other words, you may end up not having equilibrium between applied mass and calculated force. Note also that no force couples will be calculated when subjected to horizontal accelerations.
- The explicit loads will only create loads and can not be used to add mass to the mass model. You may apply these loads independently of your structural concepts, e.g. a line load along two beams or a pressure load on a part of a concept plate. Point masses may also be applied.
- The effect of structural weight (density of structure and any explicit modelled point mass) is accounted for in a structural analysis when this has been ticked off for in a loadcase. The effect of this is always part of the mass model for use in a dynamic analysis. Note that if you have ticked off for structure self-weight in many loadcases, the mass from each loadcase is added to the total mass model. Hence, for dynamic analysis you should only tick off this option for one loadcase.

An explicit load or an equipment are considered a concept and loads are applied to the analysis model when there is a intersection between a structure concept and a load concept. Hence:

- Applied Loads* result from applying concept loads to a structure.
- If the sum of concept and applied loads differ, some loads have not been properly applied.

The following example illustrates the effect of proper load application and how this can be verified. For verification of loads generated by equipments, please see Chapter 3.9.5.

One line load and one point load applied to the model. Applied loads calculated when selecting Generate Applied Loads

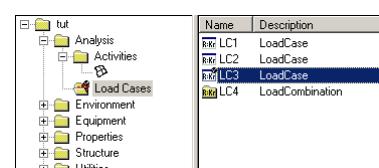


Applied loads have been calculated.



Name	Description	Force X [N]	Force Y [N]	Force Z [N]	Moment X [Nm]	Mom...
Applied Loads	n/a	0	0	-2.50231e+006	-1.37635e+007	1.187
Explicit Loads	n/a	0	0	-2.50231e+006	-1.37635e+007	1.187

To verify whether the concept loads are fully loaded to the structure double click LC3 in the browser.



The reporting of concept and applied loads show they are 100% used:

Name	Description	Usage	Force X [N]	Force Y [N]	Force Z [N]	Moment X [N*m]	Mc
LLoad1	Line Load	100 %	0	0	-1.50231e+006	-8.76349e+006	1.e
↓ PLoad1	Point Load	100 %	0	0	-1e+006	-5e+006	1e

Load	Concept	Description	Usage
LLoad1	Bm53	Applied Line Load	100 %
↓ PLoad1	Bm47	Applied Point Load	100 %

In case some of the concept loads are not intersected with structure the utilisation will be less than 100%. In the example below the point load is not intersecting any structure concept.



The point load will not give applied loads since it is not connected to structure and the reporting of concept and applied loads shows that the concept load is contributing with 0% to applied loads. If you are using the default visualisation options, you can also see from the graphics that no applied loads have been created (transparent colour).

Name	Description	Usage	Force X [N]	Force Y [N]	Force Z [N]	Moment X [N*m]	Mc
LLoad1	Line Load	100 %	0	0	-1.50231e+006	-8.76349e+006	1.e
↓ PLoad1	Point Load	0 %	0	0	-1e+006	-5e+006	1e

Load	Concept	Description	Usage
LLoad1	Bm53	Applied Line Load	100 %

### 3.9.2 Making basic loadcases and account for selfweight

A loadcase is defined by the command **Insert/Loadcase**. From the browser area you may select it, RMB and **Properties** to see the content of it. You can then re-assure that e.g. acceleration field is as expected and that the masses and loads correspond. From this dialog you also decide whether to include structural self-weight (structural mass and point masses) and whether the equipments shall be represented as finite element loads or contribute to the mass matrix for e.g. dynamic analysis.

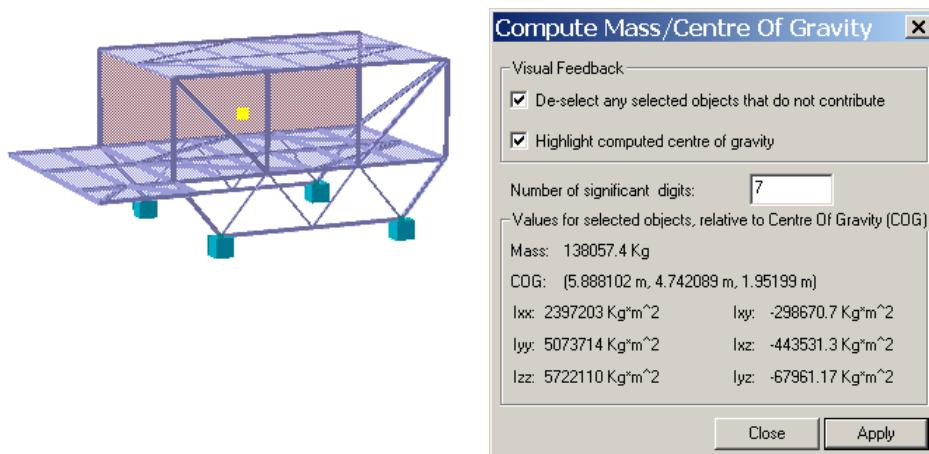
Name	Description
R:R Load_eqpm	LoadCase
R:R Load_expl	LoadCase
R:R Load_list	LoadCase
<b>R:R Load_stru</b>	LoadCase

Load Case Properties: Load_stru	
<b>General</b>   Equipment   Loads   Rotation Field	
Environment	
Acceleration field: <input type="text" value="ctor3d[0 m/s^2, 2.0 m/s^2, -9.80665 m/s^2]"/>	
Structural Analysis Load and Mass management	
<input type="button" value="Delete Explicit Loads"/> <input type="button" value="Generate Applied Loads"/>	
<input type="radio"/> Represent Equipment as loads	<input type="radio"/> Represent Equipment as loadcase-independent mass: <input type="text"/>
<input checked="" type="checkbox"/> Include structure self-weight in structural analysis	
Sum over Equipments	
Mass [kg]:	0
COG [m]:	(0, 0, 0)
Applied load [N]:	Fx=0, Fy=0, Fz=0
Conceptual load [N]:	Fx=0, Fy=0, Fz=0
Sum	
Explicit conceptual load [N]: No loads	
Total applied load [N]: Fx=0, Fy=0, Fz=0	
<input checked="" type="checkbox"/> FEM Loadcase number: <input type="text" value="4"/>	
<input type="radio"/> Display in Input Units <input type="radio"/> Display in Database Units	
<input type="button" value="OK"/> <input type="button" value="Cancel"/> <input type="button" value="Apply"/>	

After a loadcase has been loaded with equipments or explicit loads, the Property dialogue box of the loadcase is one of the important sources for verification in addition to graphic verification or from investigating a saved report.

The Property dialogue box does not document the total structure mass. You do this either by selecting the whole structure (or parts of it), RMB and *Centre of Gravity* or investigate a saved report to verify the mass.



The numbers are shown with 7 significant digits.

Mass, centre of gravity and relevant moments are shown.

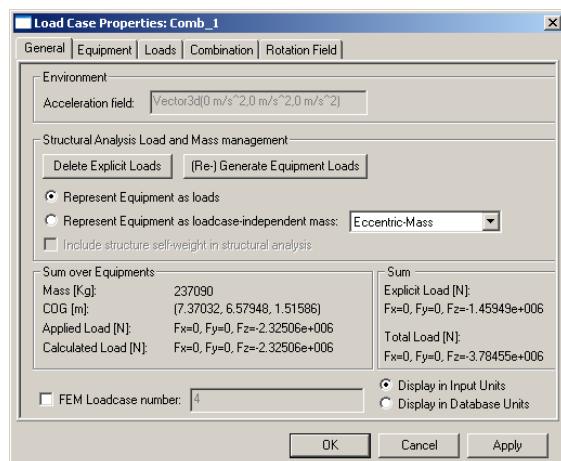
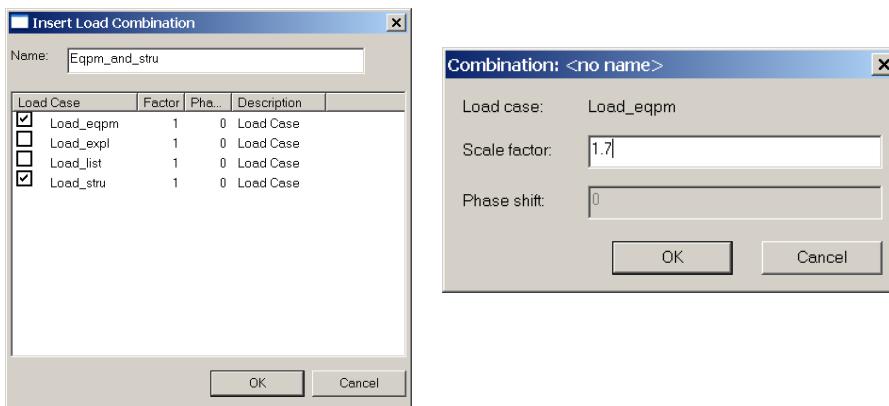
	A	B	C	D	E	F	G	H	I	J	K	L
1	Group	Mass [Kg]	About	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	Ixx [Kg*m^2]	Iyy [Kg*m^2]	Izz [Kg*m^2]	Count		
2	StraightBeams	2.3447E+04	Origin	6.714	4.787	1.449	1.1728E+06	1.9736E+06	2.7005E+06	69		
3			ModelCOG	0.826	0.045	-0.503	5.9220E+05	8.8933E+05	1.1223E+06			
4			SubsetCOG	0.000	0.000	0.000	5.8623E+05	8.6742E+05	1.1063E+06			
5	CurvedBeams	0.0000E+00	Origin							0		
6			ModelCOG									
7			SubsetCOG									
8	FlatPlates	1.1461E+05	Origin	5.719	4.733	2.055	4.8550E+06	8.4126E+06	1.0913E+07	8		
9			ModelCOG	-0.169	-0.009	0.103	1.8050E+06	4.1844E+06	4.5998E+06			
10			SubsetCOG	0.000	0.000	0.000	1.8038E+06	4.1799E+06	4.5965E+06			
11	CurvedShells	0.0000E+00	Origin							0		
12			ModelCOG									
13			SubsetCOG									
14	PointMasses	0.0000E+00	Origin							0		
15			ModelCOG									
16			SubsetCOG									
17	SupportPoints	0.0000E+00	Origin							4		
18			ModelCOG									
19			SubsetCOG									
20	SupportCurves	0.0000E+00	Origin							0		
21			ModelCOG									
22			SubsetCOG									
23	Joints	0.0000E+00	Origin							0		
24			ModelCOG									
25			SubsetCOG									
26	Total	1.3806E+05	Origin	5.888	4.742	1.952	6.0278E+06	1.0386E+07	1.3613E+07	81		
27			ModelCOG	0.000	0.000	0.000	2.3972E+06	5.0737E+06	5.7221E+06			
28			SubsetCOG	0.000	0.000	0.000	2.3972E+06	5.0737E+06	5.7221E+06			
29	Total-FEM	0.0000E+00	Origin							0		
30			ModelCOG									
31			SubsetCOG									

The above report created using *File/Save Report/XML Excel* shows the necessary details. The details are the same if you create a report in another supported format.

You may also select parts of structure and equipments in a loadcase to find the mass of the selection.

### 3.9.3 Making load combinations

A load combination is made by the command **Insert/Load Combination**. Per default it will assume that all loadcases and other load combinations are part of the new loadcase. You need to edit those that are not part of the combination and you need to change the load factor from default value 1.0. You enter the edit mode of a loadcase by double clicking on it. To verify, select the load combination from the browser area, RMB and *Properties*.



The total loads for a load combination may be found either from the load property sheet or from the saved report, both options are shown below.

Total load sum for the load combination Comb\_1 is shown. More details may be found under the tabs *Loads* and *Combination*.

	A	B	C	D	E	F	G	H	I
1	Type	X-For [N]	Y-For [N]	Z-For [N]	X-Mom [N*m]	Y-Mom [N*m]	Z-Mom [N*m]	Count	Note
2	Explicit-Point							0	
3	Explicit-Line							0	
4	Explicit-Surface							0	
5	Explicit-Temperature							0	
6	Explicit-Displacement							0	
7	Explicit-Applied							0	
8	Explicit-Conceptual							0	
9	Equipment-Applied	0.0000E+00	0.0000E+00	-2.4517E+06	-1.6344E+07	1.8388E+07	0.0000E+00	2	
10	Equipment-Conceptual	0.0000E+00	0.0000E+00	-2.4517E+06	-1.6344E+07	1.8388E+07	0.0000E+00	2	
11	Structure-Conceptual	0.0000E+00	0.0000E+00	-1.3539E+06	-6.4202E+06	7.9718E+06	0.0000E+00	77	
12	Total-Applied	0.0000E+00	0.0000E+00	-3.8055E+06	-2.2765E+07	2.6359E+07	0.0000E+00	79	
13	Total-Conceptual	0.0000E+00	0.0000E+00	-3.8055E+06	-2.2765E+07	2.6359E+07	0.0000E+00	79	
14	Total-FEM							0	

Alternatively, you may select (in the browser area) the load cases you want to be part of the load combination, RMB and *New Load Combination*. The load combination now only includes the relevant load cases.

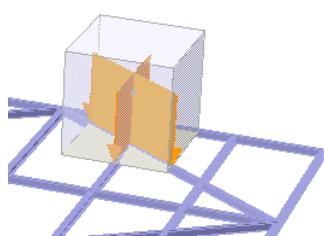
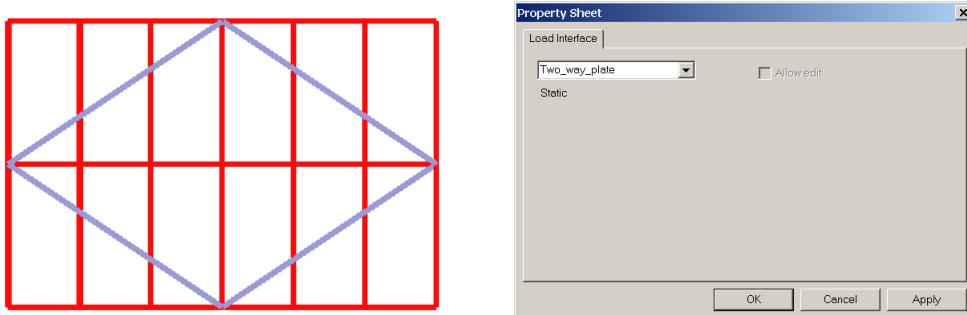
A load combination may include other load combinations, also known as nesting.

### 3.9.4 Making a load interface

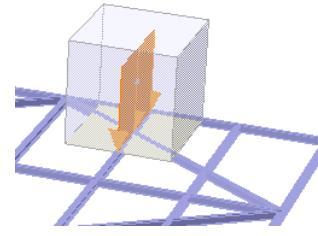
A load interface specifies which beams to receive line loads from equipments or weight list objects. If no load interface is specified all beams are loaded when there is an intersection between beams and equipment footprints.

Observe that load interfaces and equipment footprints (see Chapter 3.9.2 for definitions) do not exclude each other – they work together. This means that loads carried through a footprint can only be loaded to members assigned with a load interface.

You create a load interface by selecting the beams to be part of the interface, RMB and *Properties/Load Interface*. The same load interface must be connected to relevant equipments in each loadcase where it shall apply. The example below shows that only beams in x and y direction shall carry load, the diagonal members are not part of the load interface called *Two\_way\_plate*. Furthermore is visualised the difference in load generation prior and after the load interface has been added to the structure and the equipment.

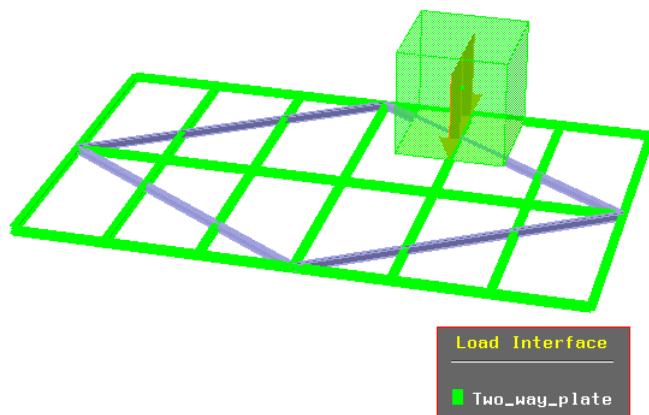


*Line loads before load interface is connected*



*Line loads after load interface is connected*

You can verify which object are part of the load interface by colour coding (select the load interface from browser, RMB and *Colour Coding*).



Joints may also be added to a load interface. The calculated equipment load is then distributed with equal load at each joint (i.e. joint load = total equipment load / number of joints). Joints outside the footprint of the equipment may also be selected. Note that when joint(s) are part of the load interface loads are transferred to joints only, i.e. any beams added to the load interface do not get any loads.

Also see the tutorial *A10 – Wind Loads* for more info.

### 3.9.5 Controlling the loadcase numbering

GeniE comes with a scheme for automatic finite element load case numbering in order to ensure logical loadcase sequencing. The aim of the loadcase sequence is to obtain a compact numbering without holes or overlaps.

You may override the automatic numbering scheme by deselecting the option as found under **Edit/Rules/Meshing**.

The logical loadcase sequence is:

- Basic loadcases: Defaults numbers provided by GeniE, the user may override these
- Wave loadcases: Variable sequence numbering, the first wave loadcase may be defined in the wave load activity. See Vol. 2 for further details.
- Combined loadcase: Automatic numbering, but always after Manual and Variable. May be overridden by user.

It is recommended to use the above loadcase numbering scheme, however, when using GeniE for superelement modelling you may want to have a unique and constant loadcase number also for the load combinations. In this case you need to override by deselecting the automatic loadcase numbering option.

The examples below show how the automatic loadcase numbering scheme works and how the user may override it.

Name	Description	FEM Loadcase	FEM LC Rule
LC1_eqpt	Load Case	1	Manual
LC2_ist	Load Case	2	Manual
LC3_load	Load Case	3	Manual
Comb_1	Load Combination	4	Automatic

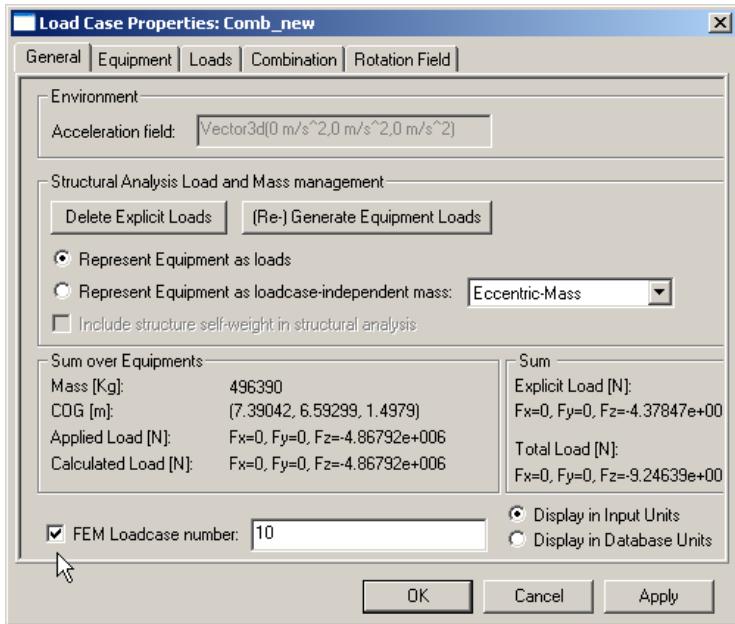
The model has three basic loadcases (manual FEM LC Rule) and one load combination (automatic FEM LC Rule).

Name	Description	FEM Loadcase	FEM LC Rule
LC1_eqpt	Load Case	1	Manual
LC2_ist	Load Case	2	Manual
LC3_load	Load Case	3	Manual
LC5_new	Load Case	4	Manual
Comb_1	Load Combination	5	Automatic

A new basic loadcase has been inserted; hence the load combination Comb\_1 receives FEM loadcase number 5.

Name	Description	FEM Loadcase	FEM LC Rule
LC1_eqpt	Load Case	1	Manual
LC2_ist	Load Case	2	Manual
LC3_load	Load Case	3	Manual
LC5_new	Load Case	4	Manual
Comb_1	Load Combination	5	Automatic
Comb_new	Load Combination	6	Automatic

A new load combination Comb\_new added, it receives FEM loadcase number 6 because it is using the automatic loadcase scheme



The load combination *Comb\_new* has been selected from the browser and RMB to access the context sensitive command.

When selecting Properties, you may change the FEM loadcase number. In this case it has been changed from 6 to 10.

Also observe that when doing this, the FEM loadcase numbering scheme changes from automatic to manual, see picture below.

Name	Description	FEM Loadcase	FEM LC Rule
LC1_eqpt	Load Case	1	Manual
LC2_list	Load Case	2	Manual
LC3_load	Load Case	3	Manual
LC5_new	Load Case	4	Manual
Comb_new	Load Combination	10	Manual
Comb_1	Load Combination	11	Automatic

The load combination *Comb\_1* receives FE loadcase number 11 since *Comb\_new* now follows the manual FEM LC Rule.

Name	Description	FEM Loadcase	FEM LC Rule
LC1_eqpt	Load Case	1	Manual
LC2_list	Load Case	2	Manual
LC3_load	Load Case	3	Manual
LC5_new	Load Case	4	Manual
Comb_new	Load Combination	10	Manual
LC6_new	Load Case	10	Manual
Comb_1	Load Combination	11	Automatic

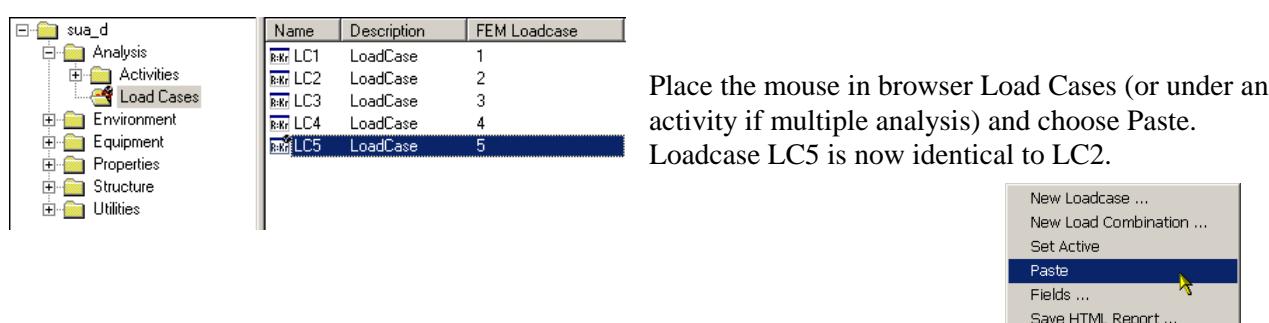
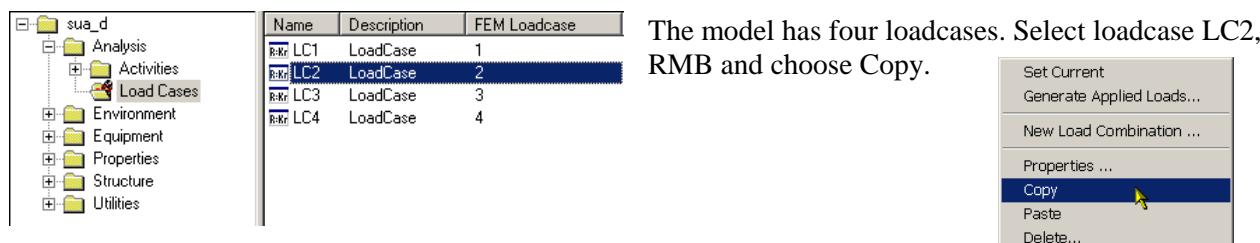
In this example a new loadcase has been added (*LC6\_new*) and the FEM loadcase number has been set to 10. Since the basic loadcases have priority over the combined loadcases, the FEM loadcase number for *Comb\_new* (which is using the manual FEM LC Rule) becomes invalid.



In addition to a graphic warning that the loadcase is invalid you will also be notified when running analysis. In this case GenIE will not perform the analysis, but stop after meshing has been performed. See Chapter 3.13 for more details.

### 3.9.6 Copy a loadcase

You may copy a loadcase and its content (this includes placed equipments) to another loadcase. The example below shows how a loadcase is copied into another loadcase.



### 3.9.7 Result cases

When results are read back into GeniE after an analysis, result cases are created. These correspond to the result cases that are created by Sestra. When GeniE is used for both modelling and analysis, the result cases are automatically linked to the load cases in the analysis when the results are loaded. Result cases are managed by GeniE, and cannot be created or deleted by the user.

Name	Description	FEM Loadcase	FEM LC Rule
KR Analysis1.step(2)	Linear Structural Analysis, Static		
R Analysis1.step(3)	Load Results		
Analysis1.step(1)	Meshing (Always Regenerate)		
RK LC1_eqpm	Reference to LoadCase	1	Manual
RK LC2_list	Reference to LoadCase	2	Manual
RK LC3_expl	Reference to LoadCase	3	Manual
RK Analysis1.resultCase(4)	ResultCase		

If, however, results are imported from external analyses, result cases will be created that are not connected to any load case. They are given names automatically, on the form <analysis name>.resultCase(<result case number>), e.g. Analysis1.resultCase(1). In cases where a FEM file is imported, followed by import of a SIN file, load cases are imported from the FEM file, and result cases created based on the SIN file. In this case, the load cases and result cases are not linked.



Result cases may also be used in scripting, e.g.

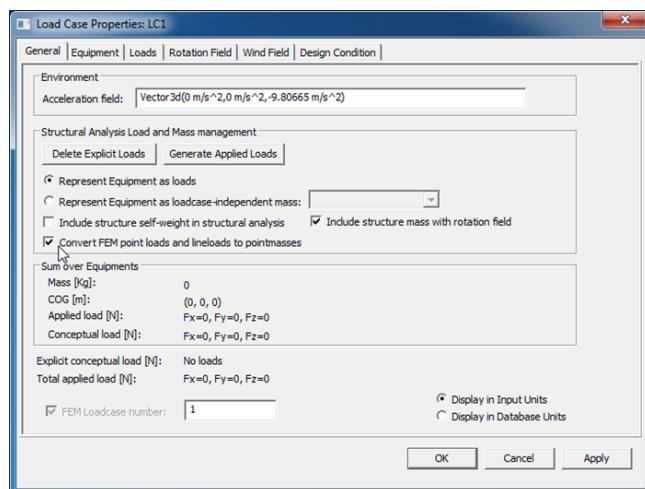
```
Analysis1.resultCase(1).designCondition = lcSeagoing;
```

This can be done even if the result case has been linked with a load case.

Generally, result cases can be used everywhere load cases can be used. This includes both code checking and reporting. However, it is currently not possible to create result combinations using result cases.

### 3.9.8 Converting point and line load to point masses

User can control how loads are generated on the FE level. One of the possibilities is to force the mesher to output Point Masses of adequate mass instead of Point Loads and Line Loads.



Select a load case -> choose Properties.

Option to convert point and line loads to point masses is controlled by the checkbox to the left.

#### Workflow:

When the tick box is checked, during the next meshing all BNLOAD and BELOAD cards on FEM file will be replaced by BNMASS. This change happens on the FEM file level, not on the concept level. Total mass of the BNMASS cards generated this way will equal the total vertical force component of all the applied point and line load. Acceleration field will be used to calculate mass from forces.

When tick box is deactivated all the masses generated by this feature will be removed from the FEM file and loads will be regenerated during next meshing.

#### Limitations:

- 1) Global mass model on the FEM file

In the FEM file we have global mass model, therefore activating this feature for any load case, will influence results of all the other load cases in current analysis. To overcome this limitation, user can create additional analysis. In the below setup, it is possible to overcome described limitation.

```
Analysis1 -> LC1 -> "Convert fem point loads ..." = true
```

Analysis2 -> LC2 -> “Convert fem point loads ...” = true

- 2) Lack of support for 2<sup>nd</sup> order elements. In this scenario no Masses will be generated.

The user should notice that when loads to mass is enabled, then no loads will be written to the FEM file for the corresponding loadcase. Therefore, the user should always ensure that the basic loadcase with the highest FEM number is a self weight case or a loadcase which is NOT converted to mass. Otherwise, the Sestra analysis may fail.

## 3.10 Equipment modelling

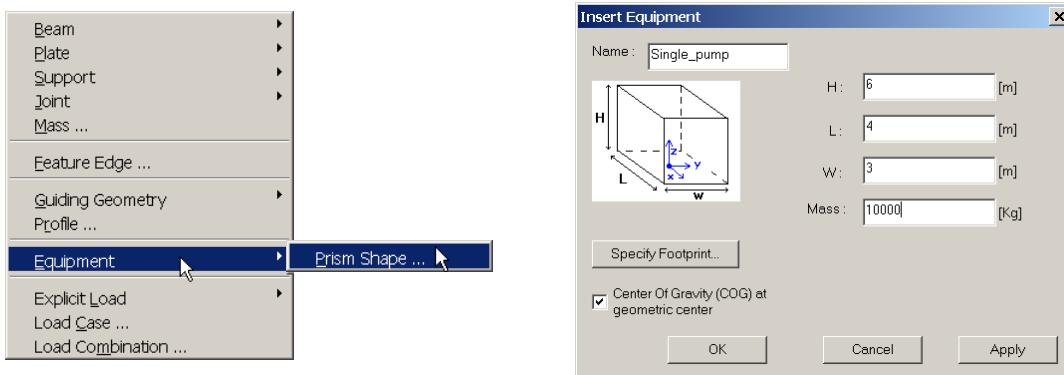
GeniE lets you model equipments and place these on the model. Based on the load calculation rule and load interface you have chosen, GeniE will calculate the reacting forces or masses to the model. In other words, when a finite element model is created, the equipment mass is converted to either a load or a mass.

The normal steps are to create the equipments you would use in an analysis and then place them as needed in the various load cases. You may also build your library of equipments in a journal file. This will allow you to easily generate equipments you need for other projects.

When you have placed the equipments you should carefully check that the loads and masses are correctly calculated.

### 3.10.1 Creating equipments

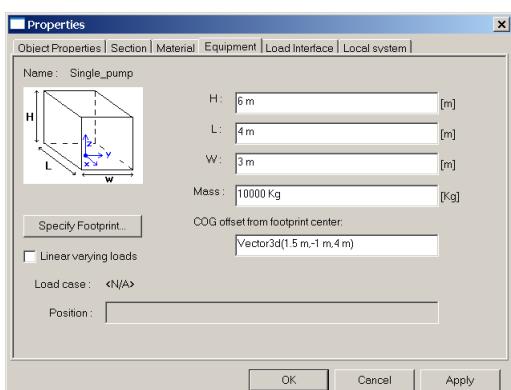
The command **Insert/Equipment/Prism shape** defines the equipment with its mass, size, centre of gravity, and the footprint (or load transfer area). The example below shows the equipment Single\_Pump with its size and mass. The c.o.g. is calculated from the default route which is in the middle of the box. Similarly, the default foot-print is the same as the bottom area of the box.



The equipments may be selected from the browser area (under Equipment) for editing and modifications. The following sections explain how to modify both c.o.g. and the foot-print.

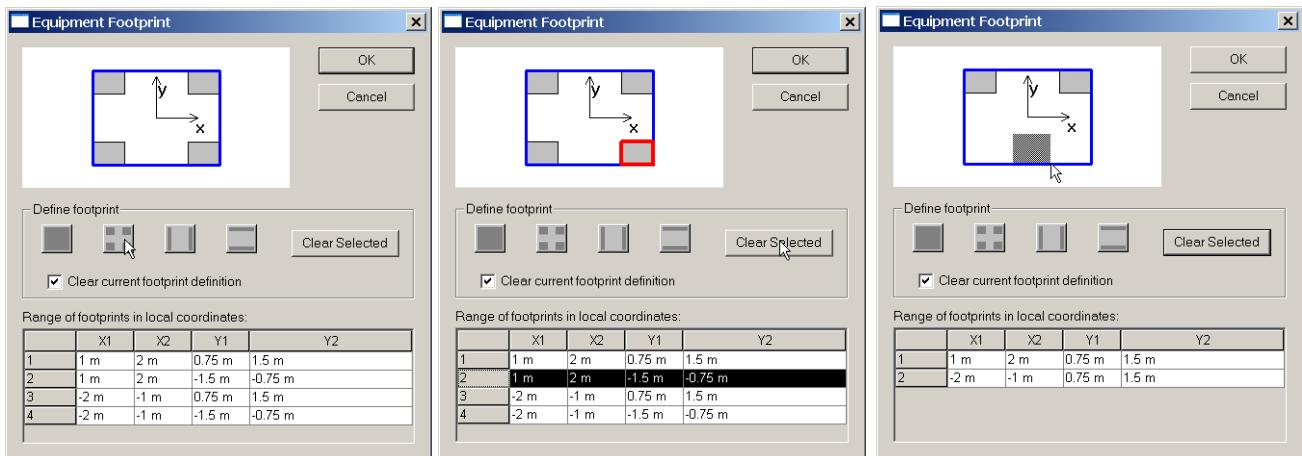
### 3.10.2 Editing the cog and footprint

The cog is per default in the middle of the equipment. To change the centre of gravity select the equipment, RMB, *Properties*, and type in the new position of the cog. Note that this position is relative to the local coordinate system (origin in the middle of the bottom plane and local z-axis upwards). Note that the cog may be outside the equipment box. This origin also acts as a snap point when equipment is placed on structure.



This example shows that the cog is located towards one of the corners and is 4 meters above bottom plane.

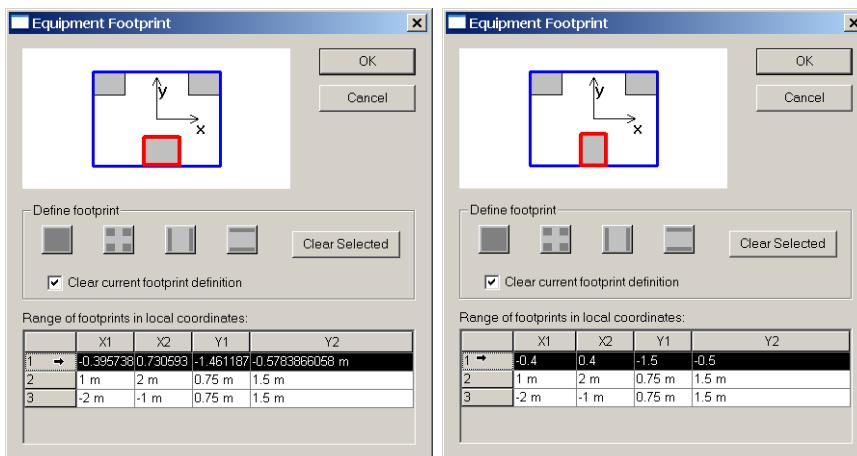
The footprint (or load distribution area) is per default the same as the area defined by the length and the width of the equipment. To change the footprint, select the equipment from the browser, RMB and click *Specify Footprint*. There are 4 predefined footprint layouts, all of these may be edited and changed to meet the requirements. The example below shows how to change a 4 corner footprint may be changed to a 3 point footprint. Note that the footprint must be defined by an area.



1. Select footprint template

2. Click on 2<sup>nd</sup> and 3<sup>rd</sup> footprint and Clear Selected

3. With a “rubberband” select insert new footprint



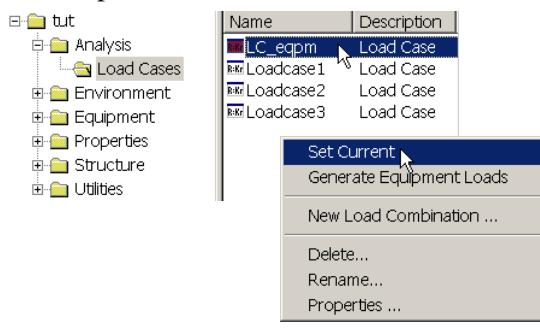
4. Select the new footprint and change local co-ordinate values

5. Final configuration with 3 support points

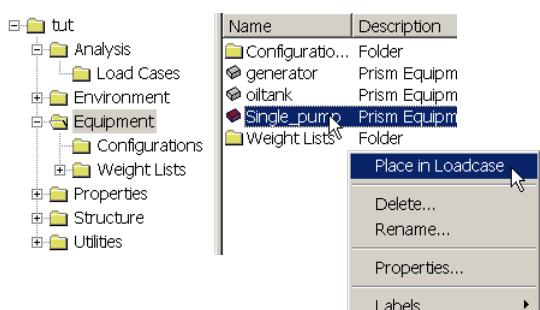
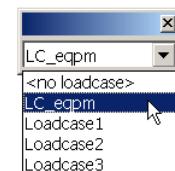
A footprint may also be larger and outside the bottom plane of the equipment. This technique may be used when defining blanket loads (or uniform distributed loads).

### 3.10.3 Placing the equipment

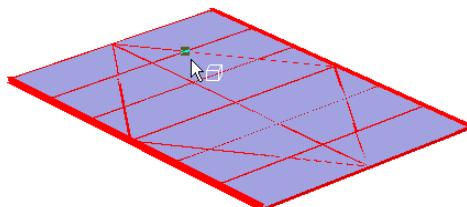
Prior to loading the structural model with equipments, a loadcase must be set to current (the active loadcase). The reason for this is that one equipment may be used in many loadcases at different positions. A loadcase is set to current by selecting it from the browser, RMB and *Set Current*. The example below shows how to place the equipment *Single\_Pump* to deck structure in loadcase *LC\_eqpm* and how to move it to another position.



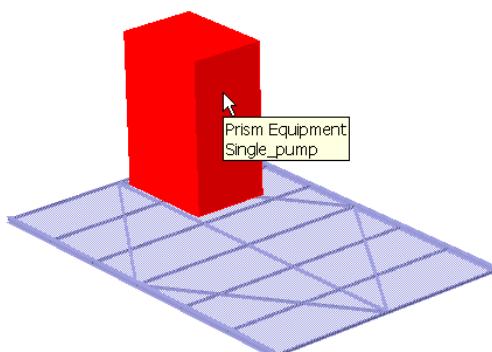
1. Selecting the loadcase and set it to current. The graphic loadcase pulldown list also shows *LC\_eqpm*.



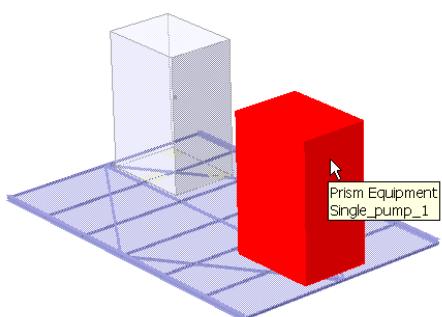
2. Selecting the equipment to be placed.



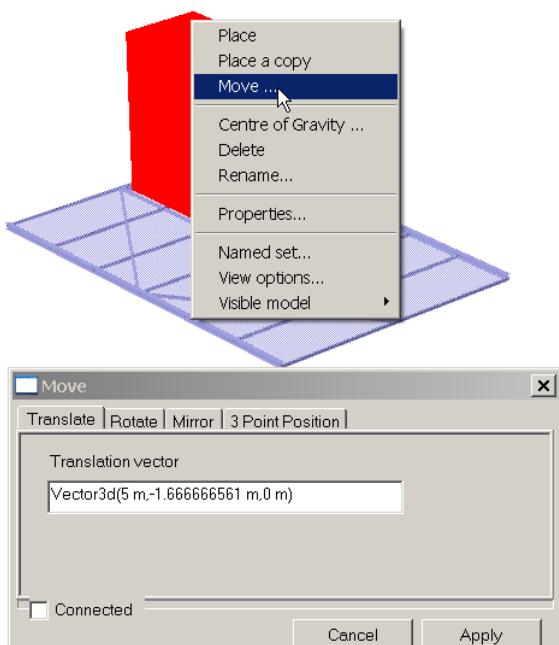
3. When moving the mouse over the structure the mouse tooltip indicates a box symbol. The positioning of the equipment now takes place where the equipment snap point (origin of box at bottom) intersects with a beam structure snap point.



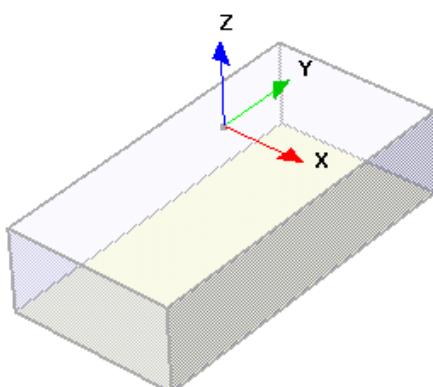
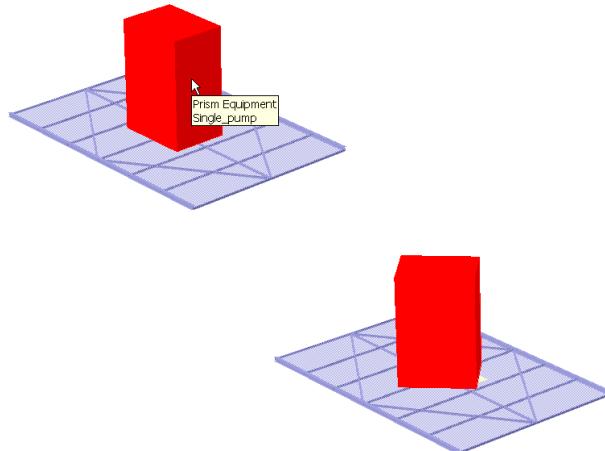
4. The equipment is now placed onto the structure in the desired loadcase.



An equipment can be used only once in a loadcase. A copy of the equipment may be used in the same loadcase but at another position(s). Select the equipment, RMB and *Place a copy*. You place the copy in the same way as described above. This equipment can be renamed from Single\_pump\_1.



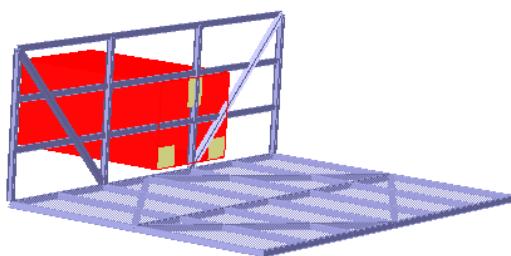
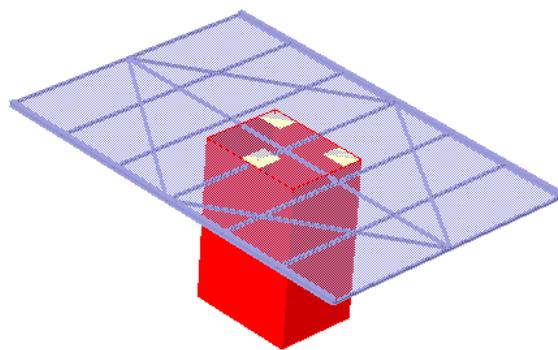
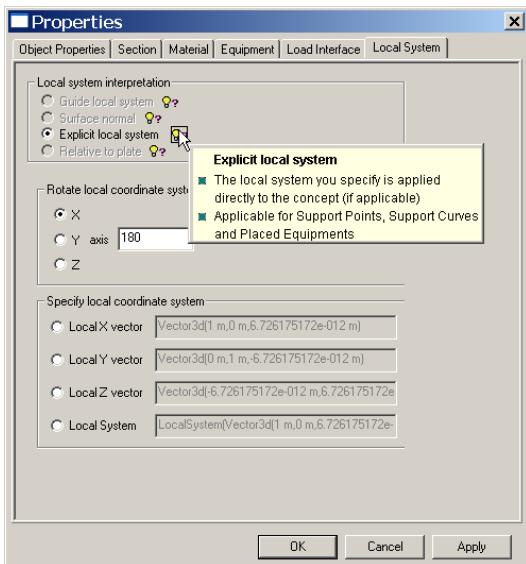
You may place the equipment more accurately by either moving it or by selecting the equipment, RMB and *Move*. The equipment may now be translated, mirrored, or rotated. In the first example it is translated, while the other shows a rotation (45 degrees).



You may also verify the equipments local coordinate system by selecting the equipment, push **RMB** and choose *Labels/Local Coordinate System*.

Origin of the local coordinate system is in the equipments centre of gravity. In this case the centre is moved above the box itself.

The equipment may be placed on top of, below, or along a beam(s). To place it below a horizontal beam(s), select the equipment, RMB and Rotate it 180 degrees. Similarly, to place it along a vertical beam(s) first rotate it 90 degrees. You find out which axis to rotate around from ***View/Options/Annotations***. The rotations are relative to the equipment local coordinate system. The example below shows the equipment rotated 180 degrees and is now placed below the plane.



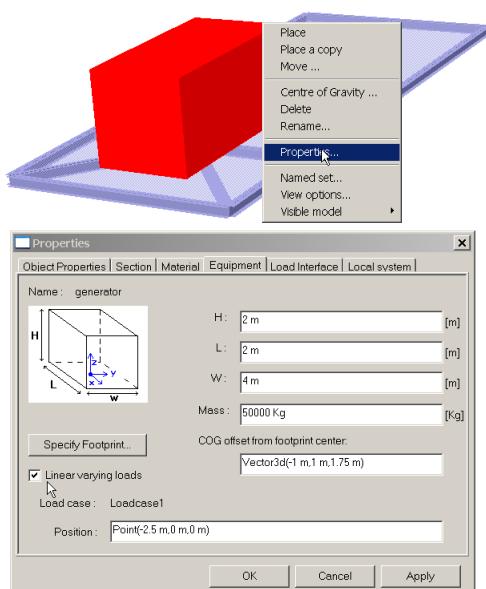
Same equipment placed along a vertical wall and rotated 90 degrees so that the footprints are aligned with the wall plane.

Note that when switching between different loadcases the equipments which are used (placed) in a loadcase are given actual position when looking in the equipment folder in the browser.

### 3.10.4 Creating forces from placed equipments

GeniE allows you to operate with two different types of load calculation rules.

- Linearly varying loads. This method will always ensure equilibrium between the applied mass and calculated force-
- Constant loads. This method will not ensure equilibrium.

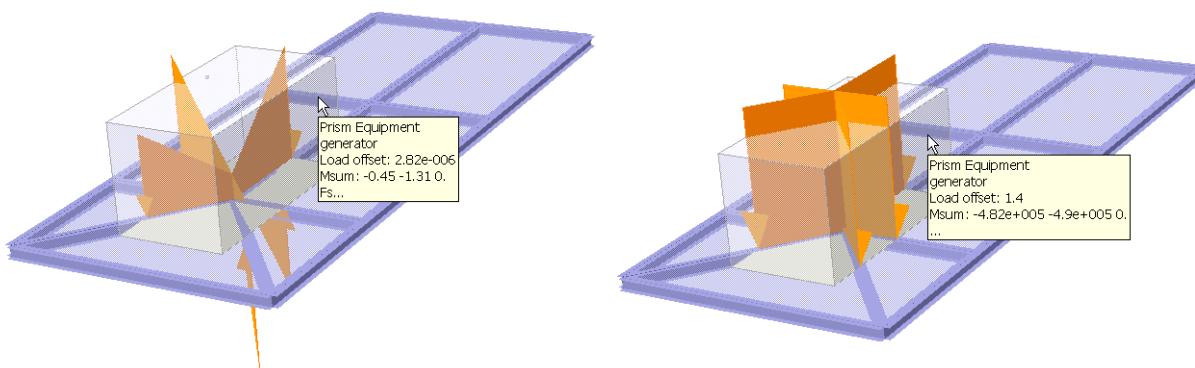


Default option is to calculate linearly varying loads. By deactivating the tick-off for linearly varying loads, the program calculates constant line loads. You may want to do this when e.g. working with uniform distributed loads (UDL).

The equipment loads are always calculated when creating a finite element mesh either manually or as a result of running an analysis.

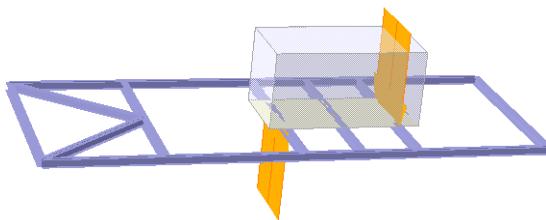
You may also force the program to calculate loads by selecting the loadcase from the browser, RMB and *Generate Equipment Loads*. You normally do this when you make up the loadcases and you want to verify the applied loads.

**Note that curved beams cannot be used to receive loads from equipment.**

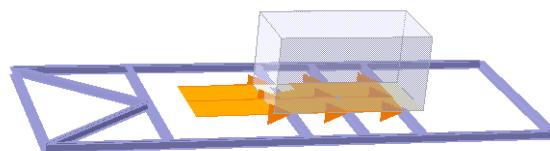


The two examples above show the difference between linear and constant – note that the mouse tooltip called *Load Offset* indicates the horizontal distance between the equipment's mass cog and the calculated force cog. I.e, it documents the error introduced by choosing constant loads. By using linearly varying loads, the load offset is 2.82E-06 m (in other words equilibrium), while the constant line load option yields a load offset 1.4 m (a significant offset).

The above loads assume a constant vertical acceleration field (gravity is one example of such). Adding a horizontal acceleration will introduce shear forces and a force couple. This is shown on the next example where the acceleration field is horizontal only. Note that when placing equipments along a vertical wall, both vertical forces (shear force) and horizontal forces (the force couple) are created.



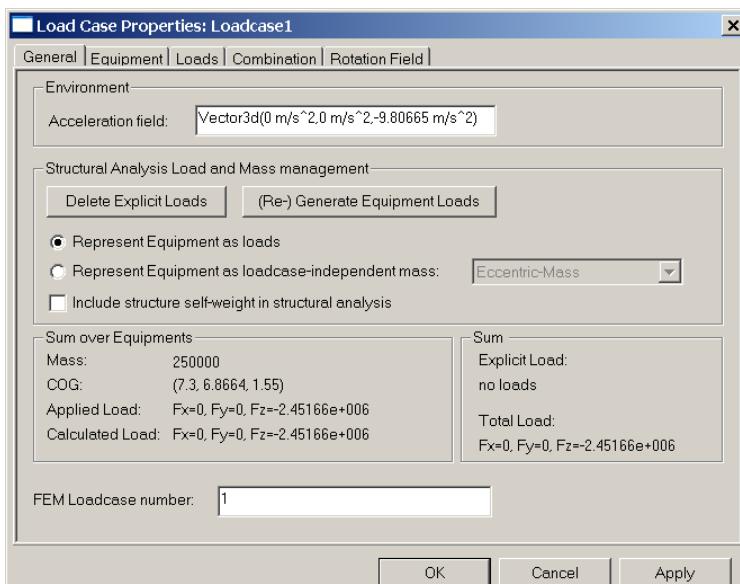
*Force couple due to horizontal acceleration*



*Shear force couple due to horizontal acceleration*

### 3.10.5 Verifying the applied loads

There are several ways of verifying that the equipments are placed and loads calculated as intended. You may graphically verify on the fly by moving the mouse over the equipment where the mouse tooltip will show you the details.

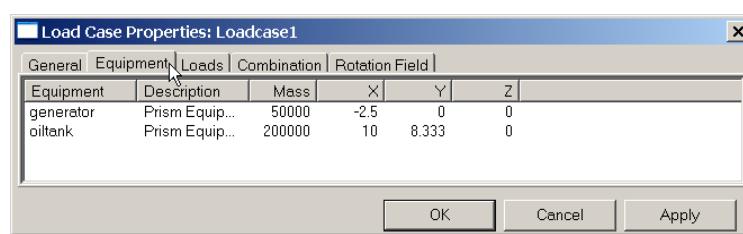


Another option is by selecting a loadcase in the browser, RMB and *Properties*.

Here you can verify that the applied load (line loads calculated by the program) corresponds to the calculated load (mass x acceleration).

If these deviate there is normally equipment footprint(s) that does not intersect with beam structure.

This example shows that the loadcase is loaded with a total equipment mass of 250000 and no explicit loads.



More details about the equipments and where they are placed may be found under the tab *Equipment*, or look in the equipment folder in the browser.

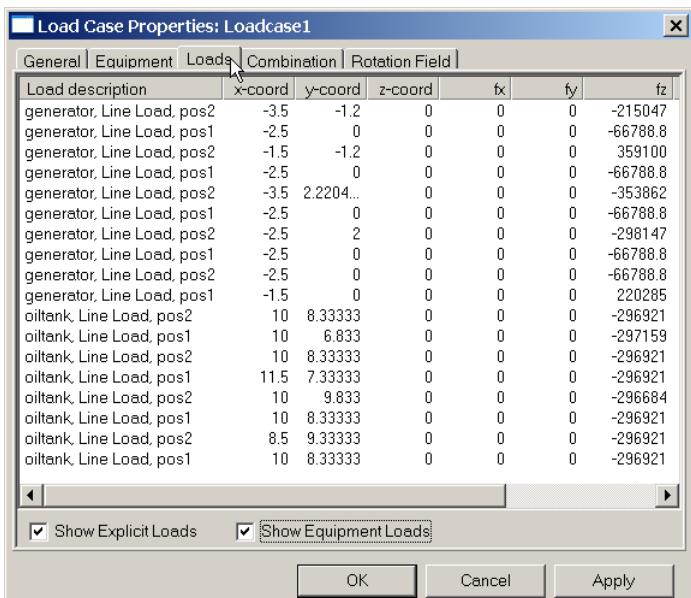
Two equipments (generator and oiltank) are placed in this loadcase (global coordinate values used).

Verifying the loads from browser shows in this case 100% utilisation, i.e. the concept loads from the equipments are fully applied to the structure.

Name	Description	Usage	Force X [N]	Force Y [N]	Force Z [N]	Moment X [N*m]
generator	Prism Equipment	100 %	0	0	-490333	-2.45166e+006
oiltank	Prism Equipment	100 %	0	0	-1.96133e+006	-1.63444e+007

Load	Concept	Description	Usage
generator	Bm55	Applied Line Load	18.7681 %
generator	Bm45	Applied Line Load	41.6426 %
generator	Bm46	Applied Line Load	20.8213 %
generator	Bm53	Applied Line Load	18.7681 %
oiltank	Bm56	Applied Line Load	54.5837 %
oiltank	Bm47	Applied Line Load	45.4163 %



The applied line loads are specified in detail under the tab *Loads*.

Each line load is described as follows:

- from which equipment
- start and stop co-ordinate values
- line loads values ( $f_x$ ,  $f_y$ ,  $f_z$ )

The details may also be found on the saved report

Microsoft Excel - Genie.xml											
1	Cause	X [m]	Y [m]	Z [m]	X-For [N/m]	Y-For [N/m]	Z-For [N/m]	X-Mom [N]	Y-Mom [N]	Z-Mom [N]	Description
2	Eq: oiltank	10.000	8.333	0.000	0.0000E+00	0.0000E+00	-2.9692E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
3	Eq: oiltank	8.500	9.333	0.000	0.0000E+00	0.0000E+00	-2.9692E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2
4	Eq: oiltank	10.000	8.333	0.000	0.0000E+00	0.0000E+00	-2.9692E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
5	Eq: oiltank	10.000	9.833	0.000	0.0000E+00	0.0000E+00	-2.9668E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2
6	Eq: oiltank	11.500	7.333	0.000	0.0000E+00	0.0000E+00	-2.9692E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
7	Eq: oiltank	10.000	8.333	0.000	0.0000E+00	0.0000E+00	-2.9692E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2
8	Eq: oiltank	10.000	6.833	0.000	0.0000E+00	0.0000E+00	-2.9716E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
9	Eq: oiltank	10.000	8.333	0.000	0.0000E+00	0.0000E+00	-2.9692E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2
10	Eq: generator	-1.500	0.000	0.000	0.0000E+00	0.0000E+00	2.2029E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
11	Eq: generator	-2.500	0.000	0.000	0.0000E+00	0.0000E+00	-6.6789E+04	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2
12	Eq: generator	-2.500	0.000	0.000	0.0000E+00	0.0000E+00	-6.6789E+04	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
13	Eq: generator	-2.500	2.000	0.000	0.0000E+00	0.0000E+00	-2.9815E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2
14	Eq: generator	-2.500	0.000	0.000	0.0000E+00	0.0000E+00	-6.6789E+04	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
15	Eq: generator	-3.500	0.000	0.000	0.0000E+00	0.0000E+00	-3.5386E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2
16	Eq: generator	-2.500	0.000	0.000	0.0000E+00	0.0000E+00	-6.6789E+04	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
17	Eq: generator	-1.500	-1.200	0.000	0.0000E+00	0.0000E+00	3.5910E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2
18	Eq: generator	-2.500	0.000	0.000	0.0000E+00	0.0000E+00	-6.6789E+04	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
19	Eq: generator	-3.500	-1.200	0.000	0.0000E+00	0.0000E+00	-2.1505E+05	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2

### 3.11 Weight lists

GeniE can import information from weight list systems, create load objects, place them (all or selected items) in a loadcase(s), and create constant line loads. The load objects being created are termed Bounding Box Equipments as compared to the manually created Prism Equipments. You may import from several weight list reports in the same GeniE model.

The process of importing weight list data is semi-automatic. The first step is to import the weight list data, then to create equipments, and finally to place them. The weight list data normally contain information about the size of each weight item – if so equipments may be automatically created. If not, you need to add dimension data to the weight items before equipments may be generated. Both options are described in the following.

### 3.11.1 Format of weight list data

GenIE can import data from a weight list using two different formats – either XML (EXtendable Markup Language) or CSV (Comma Separated Values). They must at least contain the following information, the order on the data file does not make any difference as long as the headers are the same.

- weight\_item.name : a unique name of the item (mandatory)
- weight\_item.description : additional description (optional)
- weight\_item.weight.dry : the mass of the item (mandatory)
- weight\_item.position.x/y/z : position of the item in global axis system (mandatory)
- weight\_item.dimension.dx/dy/dz : dimension of the item (optional, but recommended)

Note that the weight list is dimension less and the data will be imported according to the current *input unit settings*.

	A	B	C	D	E	F	G	H	I
2	LPumpA	Light pump	500	-2	-3	0.5	2	3	1
3	LPumpB	Light pump	500	-3	1	0.5	3	5	2
4	LPumpC	Light pump	500	-1	2	0.5	4	7	3
5	LPumpD	Light pump	500	-2	3	1.5	6	9	4
6	HPumpA	Heavy pump	1500	1	5	5.5	5	2	1
7	HPumpB	Heavy pump	1500	5	10	4.5	4	4	2
8	HPumpC	Heavy pump	1500	10	2	5.5	7	6	3
9	HPumpD	Heavy pump	1500	13	9	5.5	2	8	4
10	MiscA	Misc.	200	1	5	0.5	3	10	1
11	MiscB	Misc.	200	9	3	0.5	9	1	2
12	MiscC	Misc.	300	5	7	-0.5	7	2	3
13	MiscD	Misc.	300	12	9	0.5	6	3	4
14	GeneratorA	None	2000	-2	-3	0.5	3	4	1
15	GeneratorB	None	1500	-2	3	1.5	3	5	2

This example below shows an example of a weight list data on CSV file format (visualised in MS Excel).

```

<?xml version="1.0" encoding="UTF-8"?>
<weight_report name="dimension">
  <weight_item name="LPumpA" description="Light pump">
    <position x="-2" y="-3" z="1.5" />
    <weight dry="800" />
    <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>
  <weight_item name="HPumpB" description="Heavy pump">
    <position x="5" y="10" z="4.5" />
    <weight dry="1500" />
    <dimension dx="3" dy="3" dz="3" />
  </weight_item>
  <weight_item name="HPumpC" description="Heavy pump">
    <position x="10" y="2" z="5.5" />
    <weight dry="1500" />
    <dimension dx="3" dy="3" dz="3" />
  </weight_item>
  <weight_item name="HPumpD" description="Heavy pump">
    <position x="13" y="9" z="5.5" />
  </weight_item>
</weight_report>

```

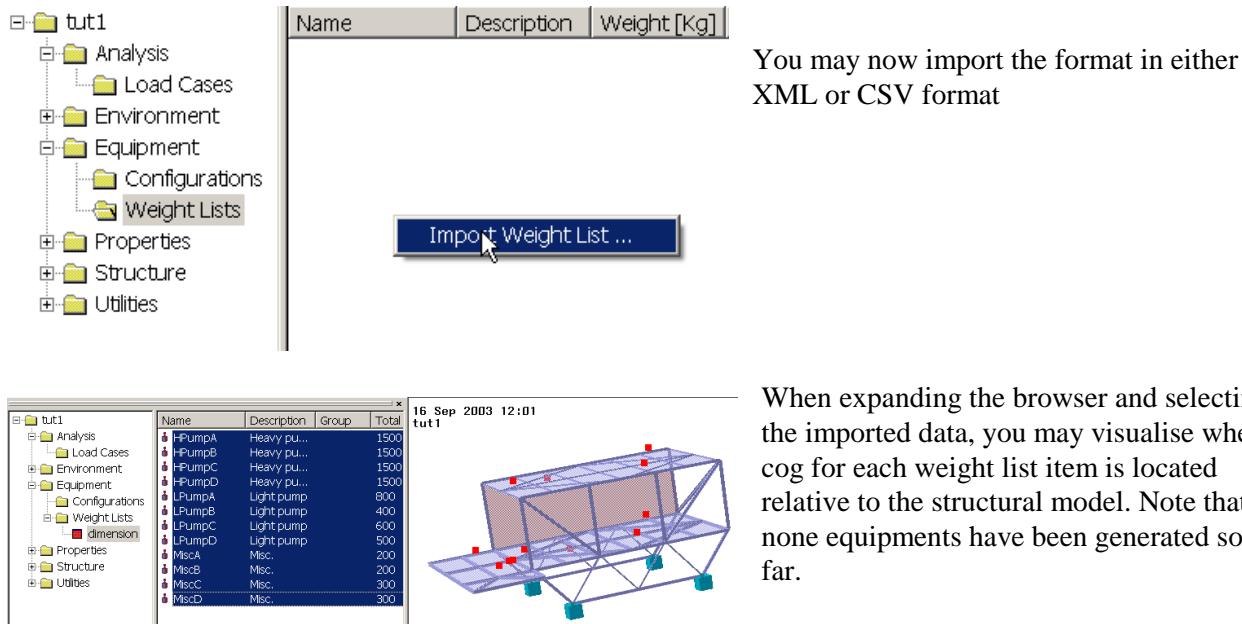
The same weight list data given on XML format.

Visualised in Internet Explorer.

### 3.11.2 Import weight list data

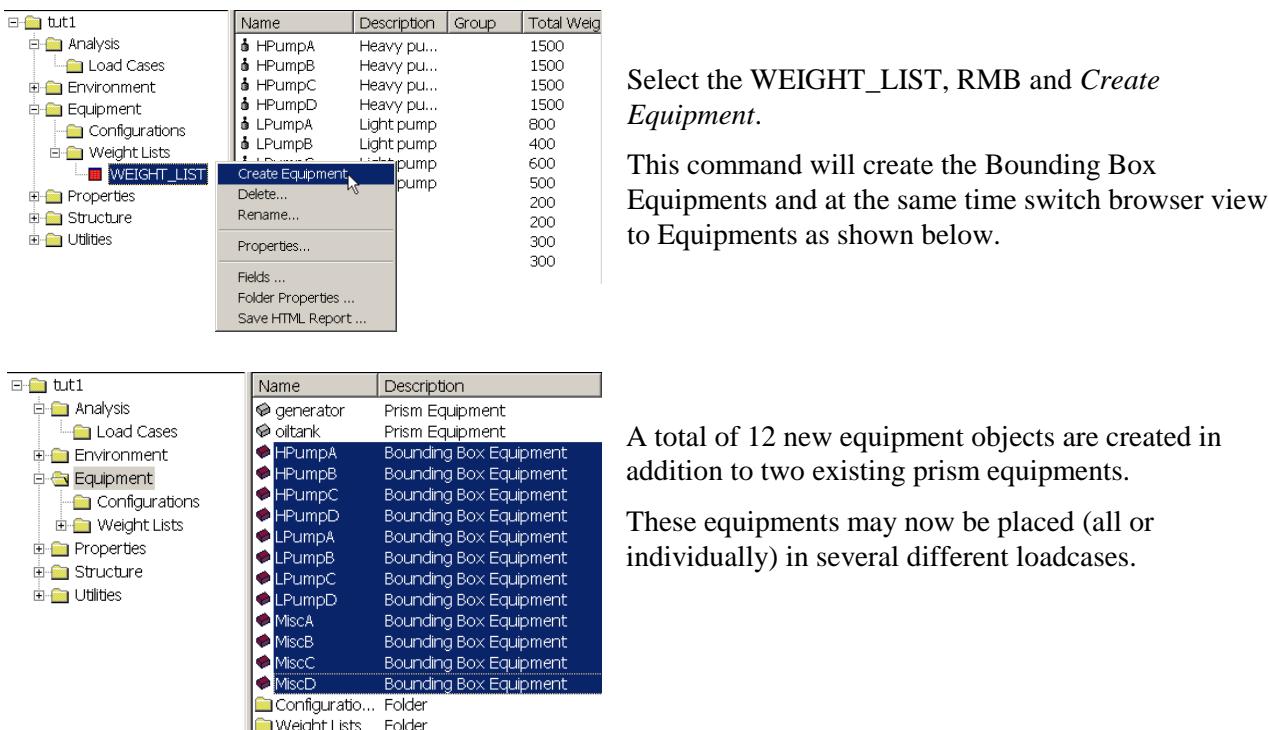
You may import several weight list reports to the same GeniE model and place individual weight list items to different loadcases. To import a weight list may be done from the pulldown menu

**Tools/Equipment/Import Weight List** or from RMB in the browser area as shown on the figure below.



### 3.11.3 Creating and placing equipments from weight lists

The next step is to create Bounding Box Equipments based on the weight list items. Remember that the Bounding Box equipments are different from Prism equipments in the way that they have no information about footprints and local cog. The pictures below show how to create such Bounding Box Equipments based on an imported file WEIGHT\_LIST.

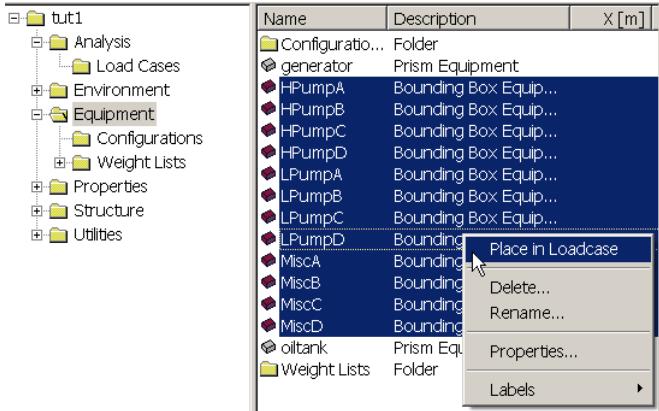


Select the WEIGHT\_LIST, RMB and *Create Equipment*.

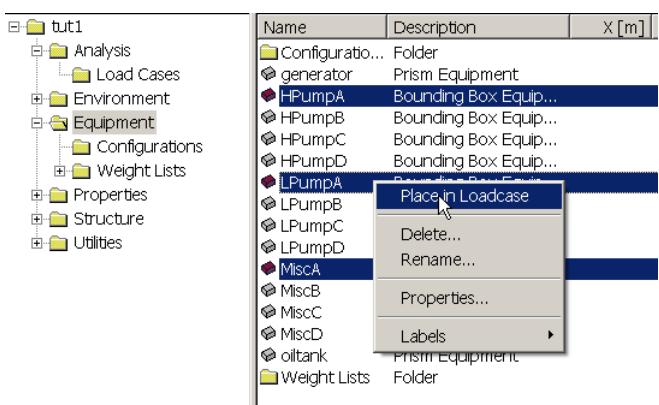
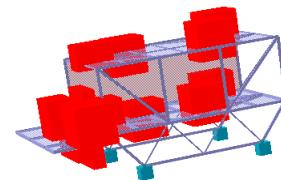
This command will create the Bounding Box Equipments and at the same time switch browser view to Equipments as shown below.

A total of 12 new equipment objects are created in addition to two existing prism equipments.

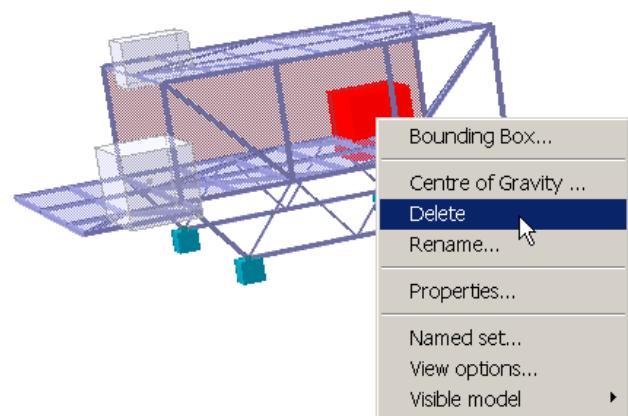
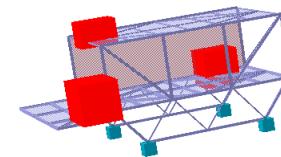
These equipments may now be placed (all or individually) in several different loadcases.



The equipments may now be placed by selecting them, RMB and *Place in Loadcase*. The position is known from before, this means you can not change position of a bounding box object without changing the weight list data file.



In this example three weight list objects are selected, RMB and *Place in Loadcase*.



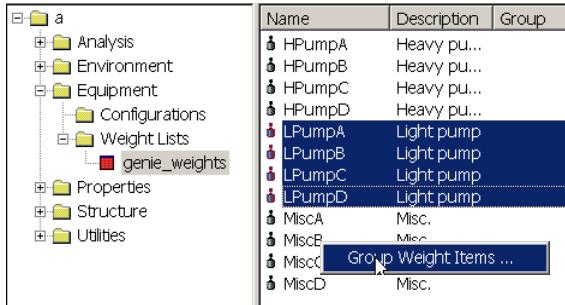
You may also remove a Bounding Box Equipment from the loadcase. By doing this, you do not delete the equipment as such. If the equipment is used in other loadcases it is still loaded for the loadcase in question.

This also applies to regular Prism Equipment.

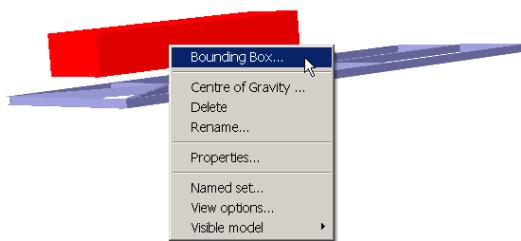
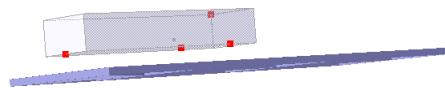
If you delete equipments from the Equipment browser you also remove the equipment from all loaded loadcases.

If the weight list does not contain dimensional information for the weight items it is necessary to create the Bounding Box Equipments manually. This is done by grouping weight items together and let the minimum box they fit into define the dimensions. The example below shows that 4 weight items without dimensions are grouped together with the purpose to become one Bounding Box Equipment termed LP.

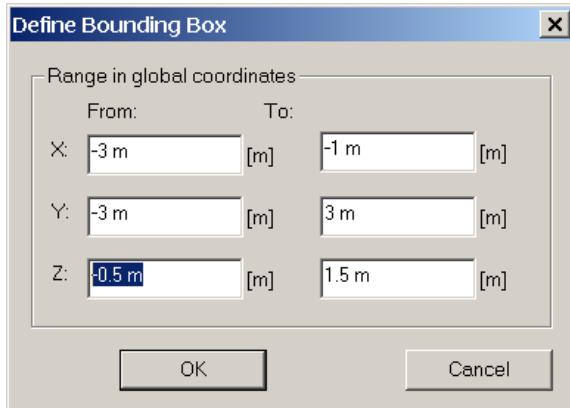
The procedure is thus to create Bounding Box Equipments by grouping selected weight list items. Thereafter the steps are as for regular Bounding Box Equipment (click on weight list in browser, RMB, *Create Equipment*, and place these in the loadcases).



Selecting the weight list objects, RMB and *Group Weight Items*. The group is given the name LP. The Bounding Box Equipment is placed on structure below. As can be seen the equipment does not intersect the structure and hence no loads are calculated.



To ensure that the bounding box equipment intersects structure it is possible to change the size of the manually created bounding box object. Select the equipment, RMB and Bounding Box where the values can be adjusted. This will not change the cog which is important when creating a mass model or the loadcase contains a horizontal acceleration vector.

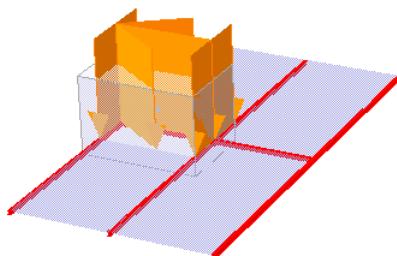


The bottom plane is moved from 0.5 m to -0.5 m and the effect is shown below



### 3.11.4 Calculating the loads

They are calculated as before by setting the actual loadcase as current and then *Generate Equipment Loads*. The major difference from Prism Equipments is that only constant loads are created, also if subjected to horizontal accelerations (i.e. no force couples are established). The constant line load is calculated from total equipment force divided by total beam length intersecting the bounding box equipment.



Constant loads applied to all beams intersecting the bounding box equipment. You may, however, direct the load transfer to beams by using Load Interface properties.

### 3.11.5 Verifying the calculated forces

The loads generated may be verified in the same manner as for Prism Equipments (graphically, from Loadcase Property Sheet, and from a saved report.) Below are some examples from the property Sheet and the saved report (note that only constant line loads are calculated).

Load description	x-coord	y-coord	z-coord	fx	fz	m <sub>x</sub>	
MiscD, Line Load, pos2	10.8333	10	-1	0	0	-425.017	0
MiscD, Line Load, pos1	10.5	10	-0.6	0	0	-425.017	0
MiscD, Line Load, pos2	12.5	10	0	0	0	-425.017	0
MiscD, Line Load, pos1	12.5	7.5	0	0	0	-425.017	0
MiscD, Line Load, pos2	12.5	10	0	0	0	-425.017	0
MiscD, Line Load, pos1	13.5	10	0	0	0	-425.017	0
MiscD, Line Load, pos2	10.5	10	0	0	0	-425.017	0
MiscD, Line Load, pos1	12.5	10	0	0	0	-425.017	0
MiscD, Line Load, pos2	10.5	8	0	0	0	-425.017	0
MiscD, Line Load, pos1	11.25	7.5	0	0	0	-425.017	0
MiscC, Line Load, pos2	5	8.33333	0	0	0	-576.496	0
MiscC, Line Load, pos1	5.25	8.5	0	0	0	-576.496	0
MiscC, Line Load, pos2	3.5	7.33333	0	0	0	-576.496	0
MiscC, Line Load, pos1	5	8.33333	0	0	0	-576.496	0
MiscC, Line Load, pos2	5	8.33333	0	0	0	-576.496	0
MiscC, Line Load, pos1	5	5.5	0	0	0	-576.496	0
MiscC, Line Load, pos2	5	8.5	0	0	0	-576.496	0
MiscC, Line Load, pos1	5	8.33333	0	0	0	-576.496	0
MiscB, Line Load, pos2	10	1.66667	0	0	0	-284.194	0
MiscB, Line Load, pos1	9.75	1.5	0	0	0	-284.194	0

	A	B	C	D	E	F	G	H	I	J	Description
1	Cause	X [m]	Y [m]	Z [m]	X-For [N/m]	Y-For [N/m]	Z-For [N/m]	X-Mom [N]	Y-Mom [N]	Z-Mom [N]	
2	Eq: LP	-2.500	0.000	0.000	0.0000E+00	0.0000E+00	-6.4018E+03	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
3	Eq: LP	-3.000	-0.600	0.000	0.0000E+00	0.0000E+00	-6.4018E+03	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2
4	Eq: LP	-2.500	0.000	0.000	0.0000E+00	0.0000E+00	-6.4018E+03	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos1
5	Eq: LP	-1.000	-1.800	0.000	0.0000E+00	0.0000E+00	-6.4018E+03	0.0000E+00	0.0000E+00	0.0000E+00	Line Load, pos2

## 3.12 Explicit loads

You may define point loads, constant or linearly varying line loads, constant pressure loads, temperature loads or prescribed displacements. Prior to any modelling of explicit loads, it is necessary to set a loadcase current.

An explicit load does not need to be aligned with a snap point (or a structural joint) in your model. As long as there is intersection between the load and the structure GeniE will compute the necessary details to ensure a correct model.

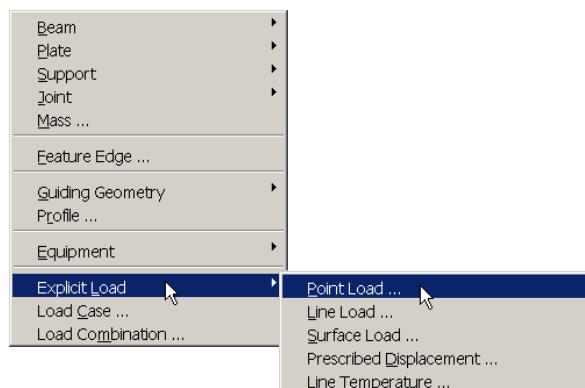
You may delete or change individual loads per loadcase, observe that this must be done graphically (select object, RMB, and *Delete* or *Properties*). Make sure that the filter Diagram selection on/off from the Property Toolbar is activated, if not you are not able to select the loads graphically.



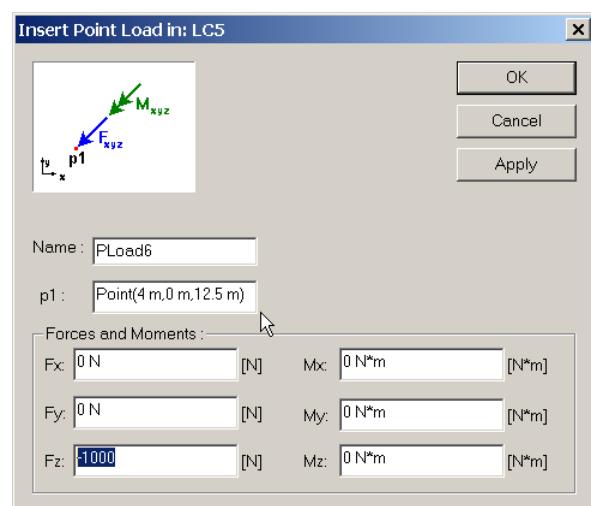
The applied explicit loads may be verified graphically, from the loadcase property sheet or from a saved report. In addition you can do it from the browser as explained in Chapter 3.8.1.

### 3.12.1 Point loads

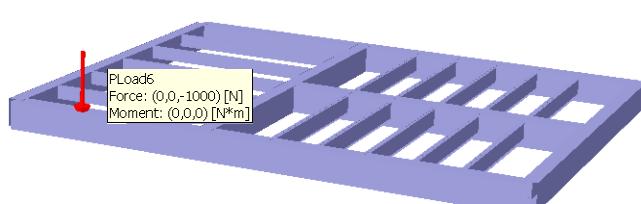
A point load on a beam member is applied by using the command **Insert/Explicit Load/Point Load**.

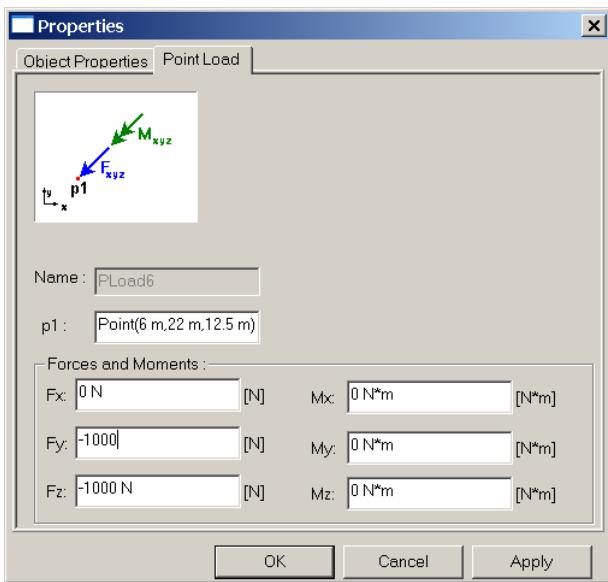


All explicit loads are defined from the pulldown menu  
**Insert/Explicit Load**.

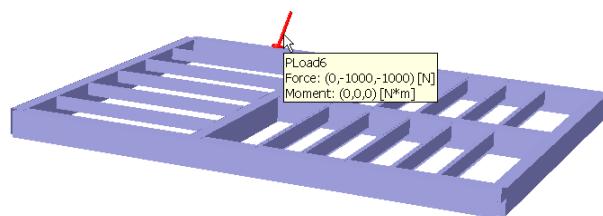


Inserting a point load in the middle of a beam where no snap point exists. Note that when moments are included in the point load special care must be taken with regard to mesh settings. If its position is changed after being meshed the first time the “Regenerate mesh option” must be set to “Always Regenerate Mesh” or else the load will be lost.





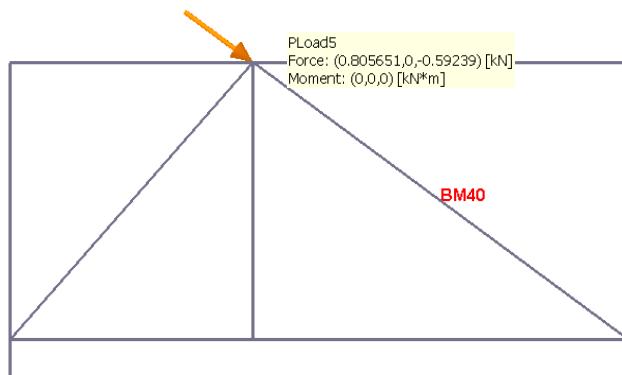
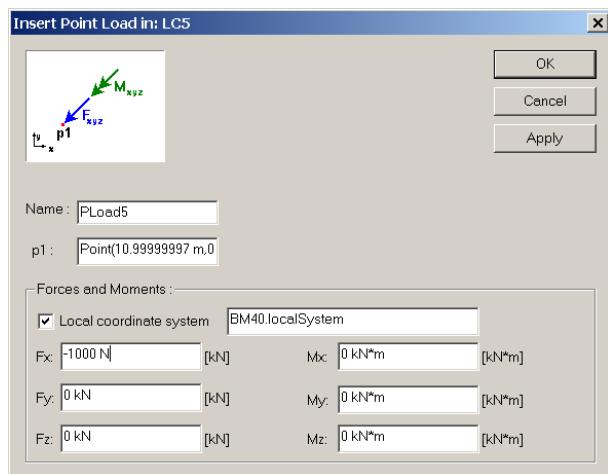
The point load may be changed by selecting the point load, RMB, and Properties. In this case both the location and intensity have been changed.



The load details of the saved report are shown below.

Microsoft Excel - Genie.xml											
1	Cause	X [m]	Y [m]	Z [m]	X-For [N]	Y-For [N]	Z-For [N]	X-Mom [N*m]	Y-Mom [N*m]	Z-Mom [N*m]	Description
2	PLoad6	6.000	22.000	12.500	0.0000E+00	-1.0000E+03	-1.0000E+03	0.0000E+00	0.0000E+00	0.0000E+00	Point Load
3											
4											

You may also refer to a local coordinate system when defining the loads (applies to both point load, line load and surface load). In the example below, the coordinate system has been graphically selected by clicking onto beam BM40. In stead of specifying a point load in global x- and y-directions it is now possible to give the load as is in local x-direction (i.e. along the beam axis).



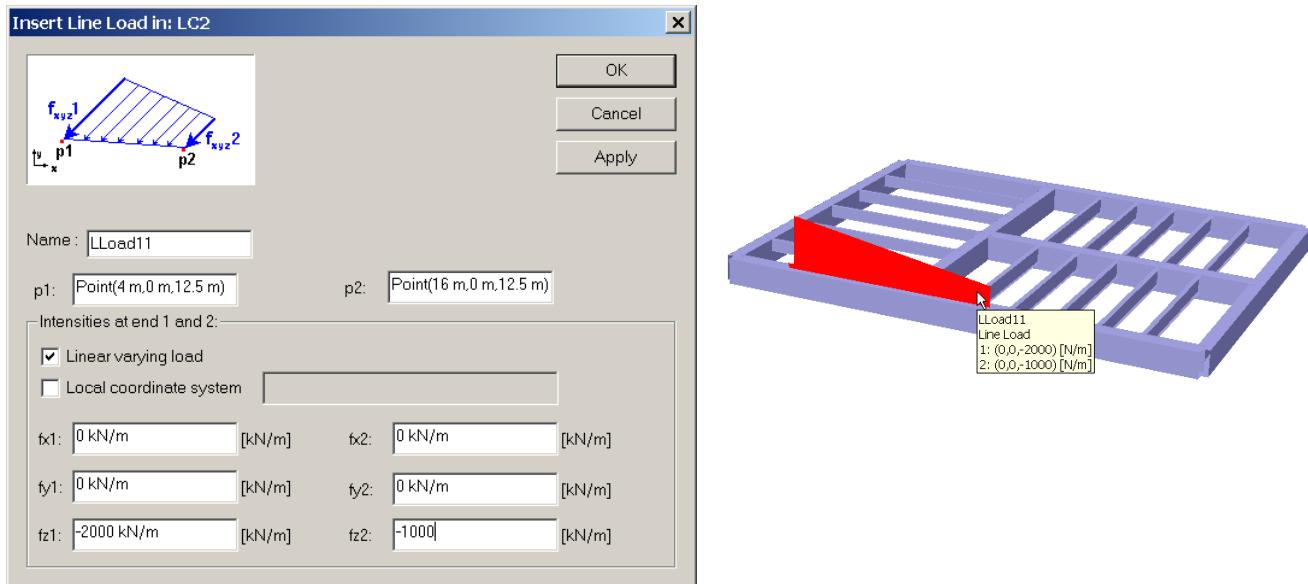
Note that when GeniE reports the load (graphically or tabulated) applied it uses the global coordinate system.

Cause	X [m]	Y [m]	Z [m]	X-For [kN]	Y-For [kN]	Z-For [kN]	X-Mom [kN*m]	Y-Mom [kN*m]	Z-Mom [kN*m]	Description
PLoad5	11.000	0.000	12.500	8.0565E-01	0.0000E+00	-5.9239E-01	0.0000E+00	0.0000E+00	0.0000E+00	Point Load
3										
4										
5										
6										

### 3.12.2 Line loads

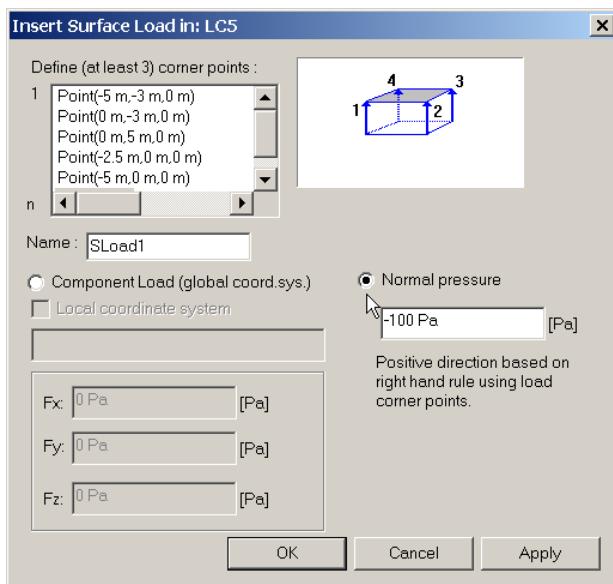
The command **Insert/Explicit Load/Line Load** lets you insert a constant or varying line load along beams.

A linearly varying line load has been inserted with start and end point explicitly defined (no snap points exist at these locations, so the input field has been manually edited).



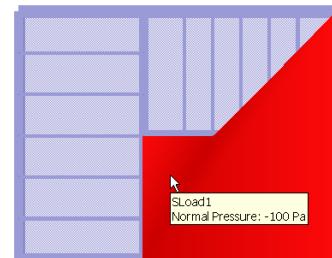
### 3.12.3 Pressure loads

Constant pressure loads are applied to flat plates using the command **Insert/Explicit Load/Surface Load**.

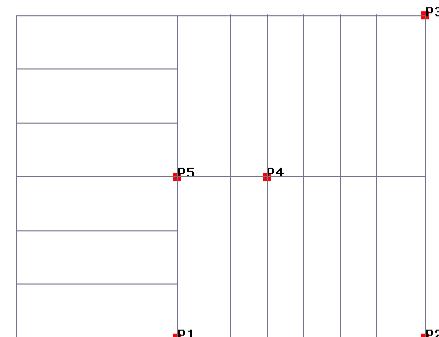


A normal pressure has been applied to parts of the plate. Make sure that the final point inserted forms a closed circuit with the first point.

Observe that the only parameter you can change for this load is the intensity of the pressure. Otherwise you need to delete it and redefine it.

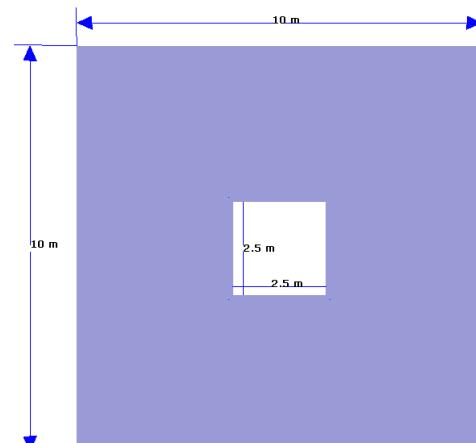


In the example above a total of five positions are used to specify the pressure area. You use the values (by clicking or manual input) for points P1-P5, i.e. you do not close the loop by entering data for P1 following P5.



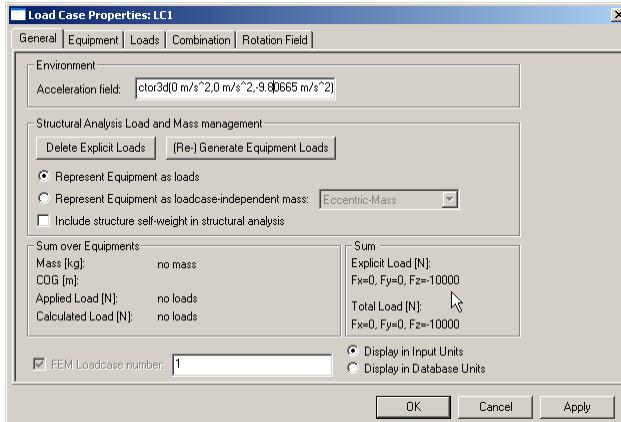
In case you have a plate with holes and you apply a surface load over the entire plate, GeniE will use the entire plate area when reporting the surface total surface load. You find the reported value either when selecting the loadcase and choose *Properties* from the context sensitive menu or from a printed report. The loads transferred to the structural analysis (Sestra) are correct.

The example below shows that a pressure of -100 Pa has been applied to a surface with a hole, the surface area is  $93.75 \text{ m}^2$ . Hence, the total load shall be -9375 N. GeniE will report -10000 N, while the analysis program Sestra computes -9375 N which is the correct value.





A uniform pressure of -100Pa applied over the whole area.



Genie reports -10000 N based on a simplified calculation.

```
C:\Workspaces\UM\20041022_002646_sestra.xls
321                                     RETRACKING MODULE - GLOBAL DATA
322
323
324 SUM OF GLOBAL LOADS AND MOMENTS
325 ****
326
327 LOADCASE (INDEX)      X          Y          Z          RX         RY         RZ
328
329      1      0.0000E+00  0.0000E+00 -9.3750E+03 -4.6875E+04  4.6875E+04  0.0000E+00
330      2      0.0000E+00  0.0000E+00 -9.3750E+03 -4.6875E+04  4.6875E+04  0.0000E+00
331
332 DATE: 22-OCT-2004 TIME: 00:26:50 **** SESTRA ****
333
334                                     RETRACKING MODULE - GLOBAL DATA
335
336
337 SUM OF REACTION FORCES AND MOMENTS
338 ****
339 GIVEN IN THE GLOBAL COORDINATE SYSTEM OF THE TOP LEVEL SUPERELEMENT
340
341 LOADCASE (INDEX)      X          Y          Z          RX         RY         RZ
342
343      1      8.4128E-12 -4.5475E-13  9.3750E+03  4.6875E+04 -4.6875E+04 -6.3665E-11
344      2      8.4128E-12 -4.5475E-13  9.3750E+03  4.6875E+04 -4.6875E+04 -6.3665E-11
345
346 DATE: 22-OCT-2004 TIME: 00:26:50 **** SESTRA ****
347
348                                     RETRACKING MODULE - GLOBAL DATA
349
350
351
352
```

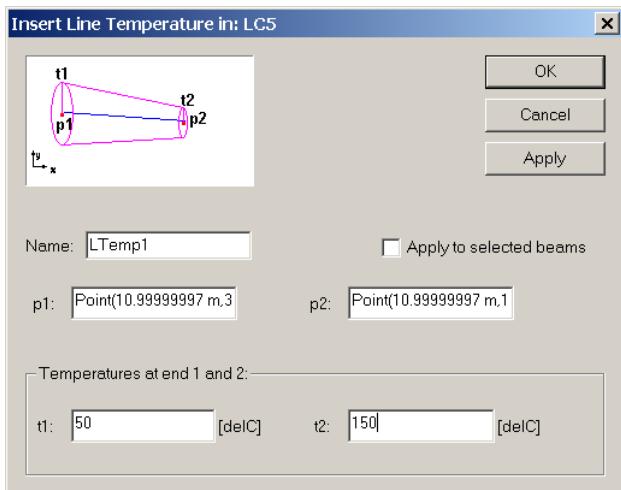
Sestra has computed the correct values, hence results like displacements and stresses are correct.

See chapter 3.13 for further details on how to run analysis.

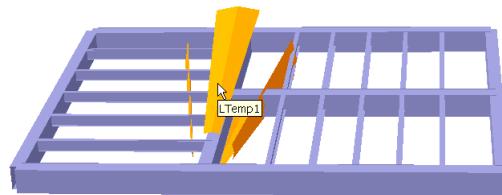
In case you have a pressure load defined by an area that differs from the topology of a plate, the correctness of the loads depend on the mesh density, type of mesh (first or second order elements). See the next Chapter on how to refine and control the finite element mesh.

### 3.12.4 Temperature load

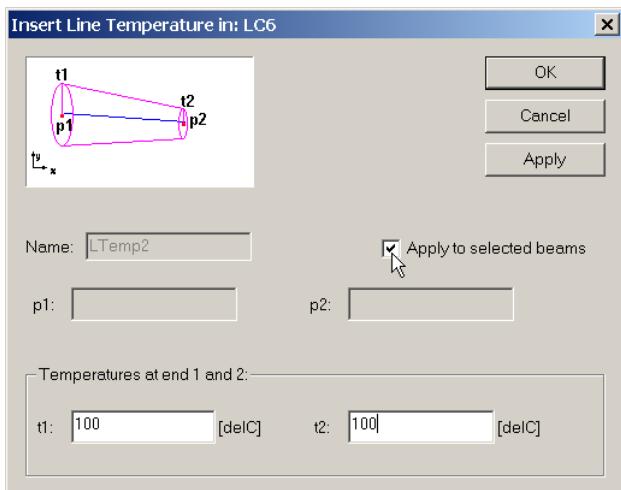
Temperature loads may be applied to beams only. The temperature loads are constant over the beam cross section, but they may vary in intensity along a beam. Temperature may be inserted to one or several beams at the same time. The command for doing so is **Insert/Explicit Load/Line Temperature**.



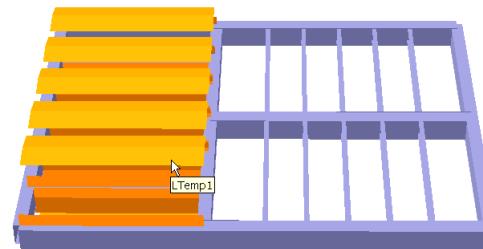
A varying temperature load is inserted along one beam. The temperature and positioning may be changed (or deleted) by selecting the load object, RMB and *Properties*.



Temperature loads may also be applied to a number of beams at the same time. You should be careful of using this option if you have varying temperature since you need to know start and stop ends of the beams. The example on the next page shows the same temperature applied to a set of beams. Make sure that the beams are selected before inserting the temperature load.



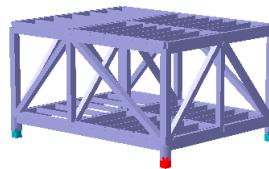
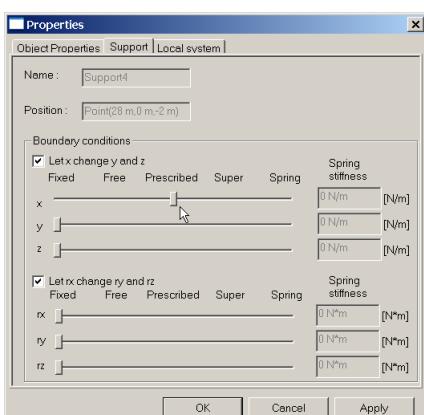
Constant temperature load applied to the beams selected prior to load definition.



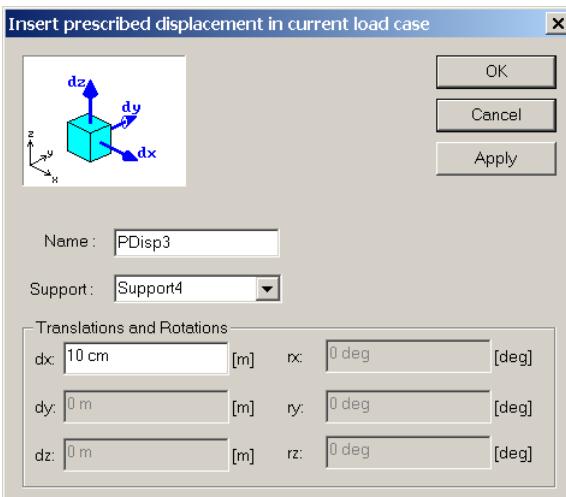
### 3.12.5 Prescribed displacements

A prescribed displacement is a boundary condition combined with the actual displacement or rotation per loadcase. A support point needs to be inserted first before the actual displacement or rotation can be defined, see the next Chapter on how to define boundary conditions.

This means that a structure may have several prescribed displacements to the same support point, but in different loadcases.



Prescribed displacement in x-direction applied to support point *Support4*.



Add 10 cm as the actual prescribed displacement. Note that you can only specify a translation or rotation in the dof specified as a prescribed displacement.

Use ***Insert/Explicit Load/Prescribed Displacement***, select the support point having prescribed displacement defined as boundary condition and give translation or rotation value to be used.

The saved report may be used to verify the prescribed displacements by investigating the boundary conditions under the Tab *Supports* and the translation (or rotation) under the Tab *Loadcases*, see below.

Microsoft Excel - Genie.xml														
File Edit View Insert Format Tools Data Window Help														
Anal														
E5 Prescr.														
1	Name	X [m]	Y [m]	Z [m]	X-Tra	Y-Tra	Z-Tra	X-Rot	Y-Rot	Z-Rot	K	L	M	N
2	Support1	0.000	0.000	-2.000	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed				
3	Support2	0.000	22.000	-2.000	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed				
4	Support3	28.000	22.000	-2.000	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed				
5	Support4	28.000	0.000	-2.000	Prescr	Fixed	Fixed	Fixed	Fixed	Fixed				

Microsoft Excel - Genie.xml													
File Edit View Insert Format Tools Data Window Help													
Anal													
E2 A 0.1													
1	Cause	X [m]	Y [m]	Z [m]	X-Tra [m]	Y-Tra [m]	Z-Tra [m]	X-Rot [c]	Y-Rot [d]	Z-Rot [deg]	Description	K	
2	PDisp4	28.000	0.000	-2.000	0.100						Displacement, Support4		
3													
4													
5													

### 3.12.6 Acceleration and rotational fields

The weight distribution of the model accounting for any point mass or element mass will be computed by the analysis program Sestra. There are thus no weight distribution loads calculated in GeniE, but the effect of it is part of the result file. The element masses are based on

- Density of the material properties connected to structure
- Volume of beam members (Cross sectional area times length – note eccentricities are accounted for)
- Volume of plate (area x thickness)

There are two ways of specifying acceleration loads, either by using a constant acceleration field or a rotational field, please note that the two methods are different (see below for explanations).

For both of them they are activated from the Loadcase Property Sheet and they may be different from loadcase to loadcase.

#### 3.12.6.1 Constant acceleration field

This is specified on the "General" tab. The field is transferred to the FEM model as an explicit field (FEM data type BGRAV) and the local gravity load on the structure is calculated by the solver program (SESTRA) on the basis of local element geometry and material densities (i.e. "self-weight"). The direction of the self-weight load is the same as the direction as the acceleration field.

Self-weight will only be calculated if the "Include self-weight in structural analysis" option is checked (if it is unchecked, the BGRAV data will not be transferred to the FEM model, hence self-weight will not be calculated).

#### Include structure self-weight in structural analysis

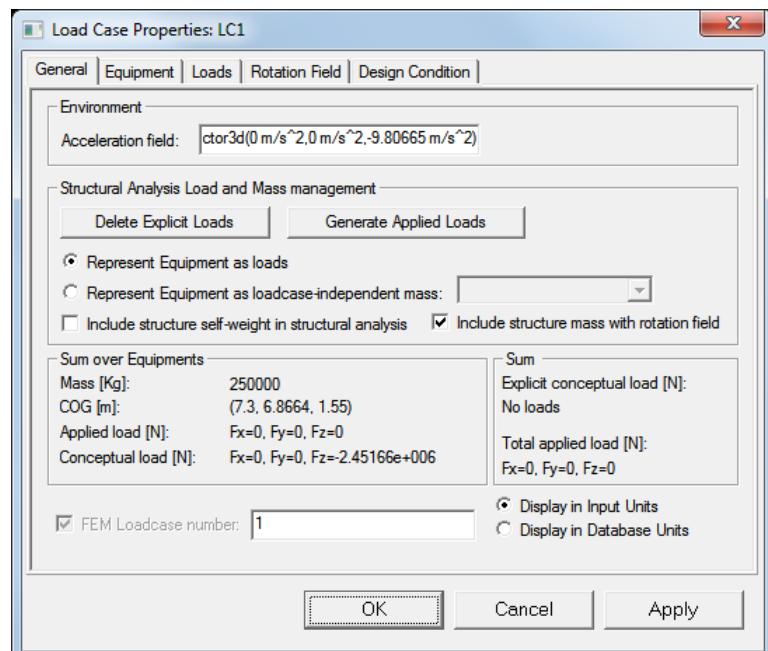
To include constant acceleration make sure that "Include structure self-weight in structural analysis" is activated.

#### Include structure mass with rotation field

This should be ticked if you want to include the structure's mass with the rotation field. If this is not ticked only loads and equipment's mass is included.

Gravity is a special case where the vertical acceleration is  $g = -9.81 \text{ m/s}^2$ .

The acceleration field may be changed from this property sheet.



To verify the acceleration loads you can either look at the loadcase property sheet or the saved report.

The image displays two Microsoft Excel windows side-by-side, both titled "Microsoft Excel - Genie.xml".

**Top Window (Loadcase Properties):**

Name	Description	FEM LC	Self Weight	Equip as Lo:	Equip Rep	X-Acc [m/s^2]	Y-Acc [m/s^2]	Z-Acc [m/s^2]
			1 Yes	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00
LC1	Load Case		2 No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00
LC2	Load Case		3 No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00
LC3	Load Case		4 No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00
LC4	Load Case							

**Bottom Window (Equipment Types):**

Type	X-For [N]	Y-For [N]	Z-For [N]	X-Mom [N*m]	Y-Mom [N*m]	Z-Mom [N*m]	Count
Explicit-Point							0
Explicit-Line							0
Explicit-Surface							0
Explicit-Temperature							0
Explicit-Displacement							0
Explicit-Actual							0
Equipment-Actual							0
Total-Actual							0
Equipment-Calculated							0
Structure-Calculated	0.0000E+00	0.0000E+00	-6.5364E+03	7.1901E+04	-9.3402E+04	0.0000E+00	61

### 3.12.6.2 Rotation field

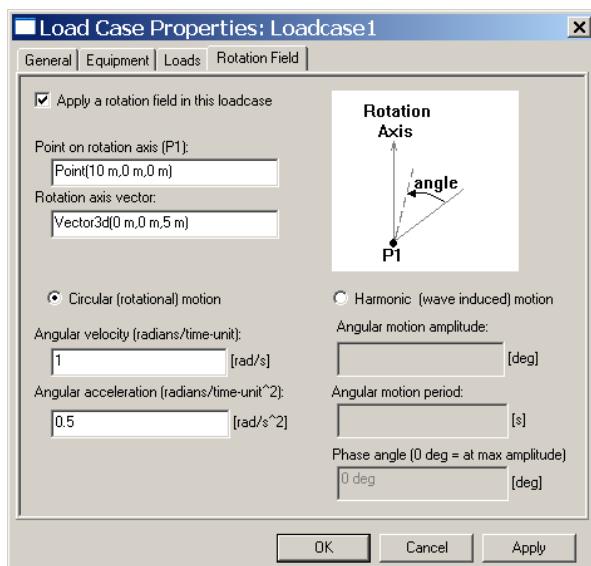
This is specified on the "Rotation Field" tab. A rotation field is transferred to the FEM model as applied nodal accelerations (BNACCLO). The sign of the applied accelerations is such that the resulting loads correspond to inertia forces.

The angular acceleration is a forced rotation and not a rotational acceleration field. This means that the inertia forces due to the angular acceleration have the opposite direction of the direction of rotation. This may be compared with giving the support points in a model upwards acceleration instead of introducing an acceleration field acting downwards. In both cases the inertia forces will act downwards. This is illustrated in the following example:

*Consider a rotating carousel with a passenger, where the carousel's rotational velocity is increasing:*

*The rotation field describes the rotation of the carousel ("the structure"). The carousel feels the inertia force from the passenger, i.e. the applied inertia accelerations are in the opposite direction of the rotational field acceleration. Also, the field rotation velocity is applied as nodal centripetal acceleration, i.e. it is directed away from the rotation centre.*

Notice also that the option "Include self-weight in structural analysis" in the "General" tab has no effect for rotation fields.



A rotational field is applied by specifying by a rotation axis vector.

The rotational field is an acceleration field composed of centripetal acceleration due to angular velocity and tangential acceleration due to angular acceleration about the axis vector.

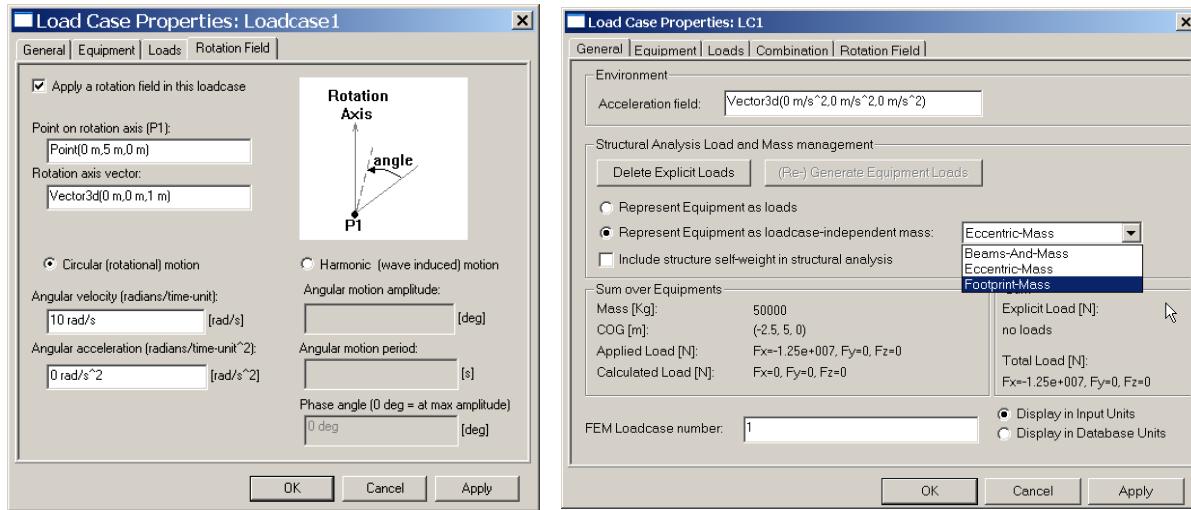
The direction of rotation is according to the right hand rule with respect to the direction of the rotation axis vector.

To verify and change you can use the loadcase property sheet.

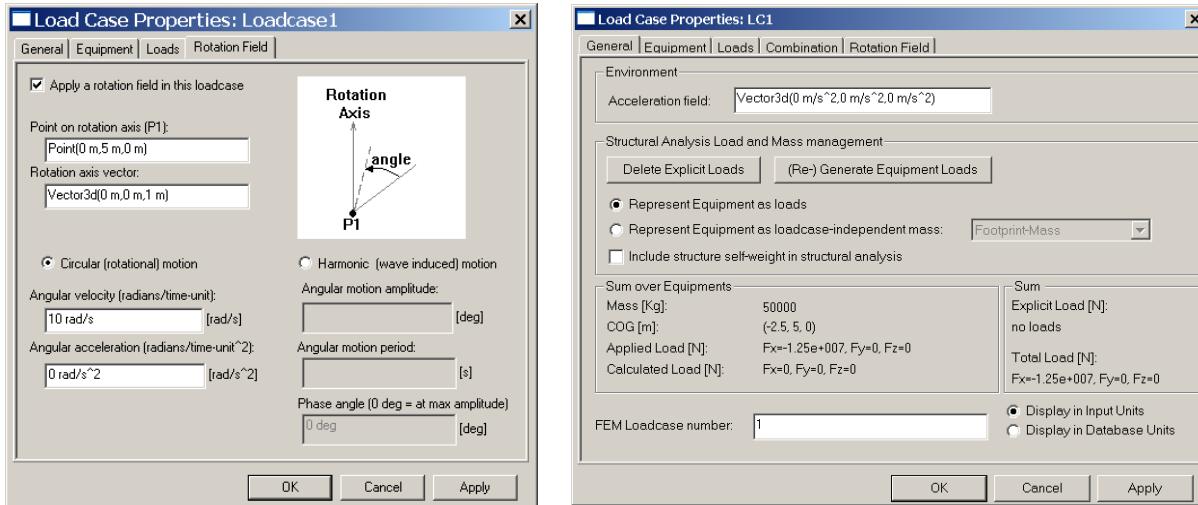
When performing the analysis there are two alternatives:

- The resulting inertia loads are computed by the analysis program Sestra based on the node accelerations calculated by GeniE when generating the finite element model.
- GeniE will compute the equipment loads based on an equivalent acceleration in the equipment's centre of gravity and using the same assumptions as for constant acceleration when calculating the forces. When using this alternative, you must remember to specify the constant acceleration field to (0,0,0) if you want to consider the effect from the rotational field only.

The pictures below illustrate the differences between these options.



The above example will transfer the accelerations to the interface file so that the resulting inertia loads are calculated by Sestra (and results brought back to GeniE for evaluations). Note that when combining two or more loadcases containing Rotational Field, the smart load combination must be used, see chapter 3.15.4 for more details.



Similarly, the example above will give equipment loads in GeniE and structural analysis is carried out using line loads in stead of accelerations and mass. You may verify the loads by looking at the loadcase property sheet or from the saved report.

Note that the angular velocity and acceleration are given in radians.

### 3.12.6.3 Harmonic motion

You may also calculate the maximum angular accelerations as  $\Theta_{acc} = ((2\pi/T)^2) \times \Theta$ , and maximum angular velocity as  $\Theta_{vel} = (2\pi/T) \times \Theta$ .

The input parameters are:

- T is the period in seconds
- $\Theta$  is the maximum roll (or pitch) angle in degrees

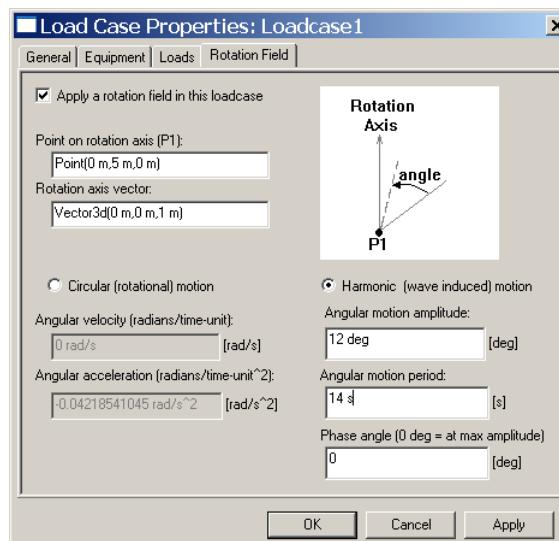
The variation of direction of the gravity vector is also accounted for, and will be:

$$G_y = -G \times \sin(\Theta) \text{ and } G_z = -G \times \cos(\Theta) \text{ for pure roll motion}$$

$$G_x = G \times \sin(\Theta) \text{ and } G_z = -G \times \cos(\Theta) \text{ for pure pitch motion}$$

where

- G is the acceleration of gravity.
- $\Theta$  is the maximum roll (or pitch) angle in degrees



The computations yield for a specific rotation point and a rotation axis vector. For a floating vessel the axis vector (1,0,0) is normally the vessel's longitudinal direction (along the x-axis). The angular motion is then a roll motion.

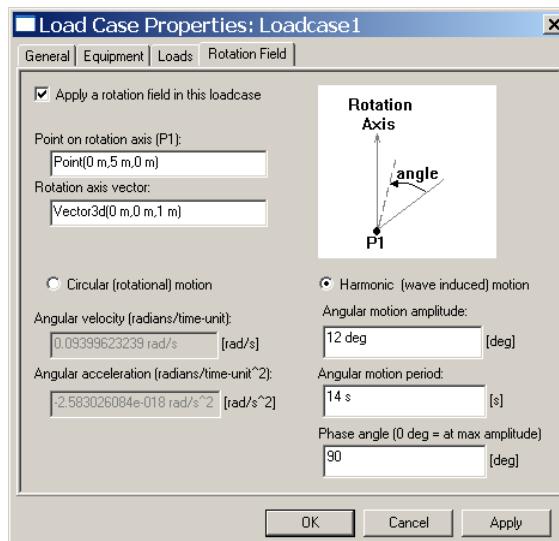
Similarly, using a vector (0,1,0) normally describes a pitch motion.

The example to the left shows the computed values for angular velocity and acceleration when the motion amplitude is 12 degrees and the corresponding period is 14 seconds.

The phase angle is used to compute max values for the angular velocity and acceleration. At phase angle 0 degrees the angular velocity is 0 and angular acceleration has its maximum value.

When the phase angle is 90 degrees the angular velocity has its maximum value and angular acceleration is 0.

At phase angle 180 degrees the values are the same as for 0 degrees, but with opposite sign for the angular acceleration.

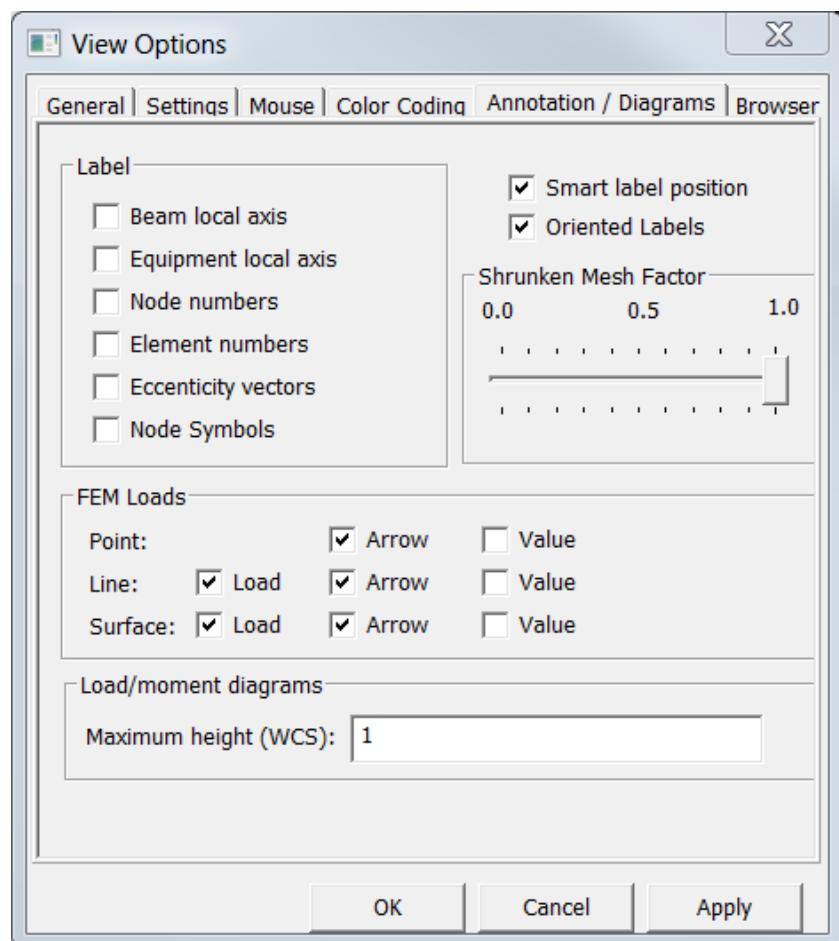


## 3.13 Load Annotations

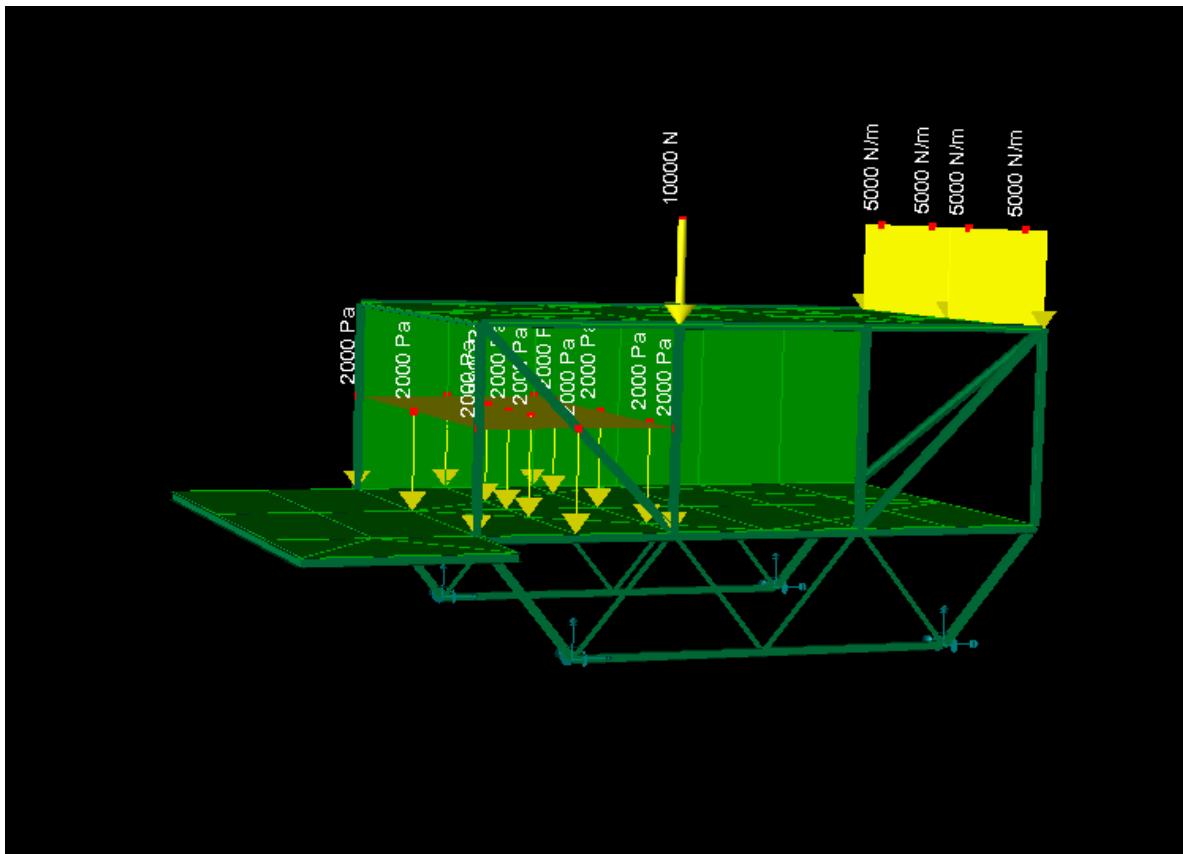
GeniE has a set of labelling options that are load-specific, termed annotations. They are drawn and controlled in a different way to standard labels in GeniE. In particular labelling surfaces can result in many annotations being generated which can affect draw performance and interactivity. There are therefore a number of options to control this.

### 3.13.1 Displaying Load Annotations

Load annotations can be displayed using the check boxes in the *FEM Loads* section of the *Annotation / Diagrams* tab of the *View / Options* dialog.



*Load* specifies whether a visual representation of the load itself should be drawn. This is only available for line and surface loads as point loads are simply represented by a single arrow. *Arrow* specifies whether the load arrows should be drawn for that type of load and *Value* specifies whether the load annotation should be displayed.

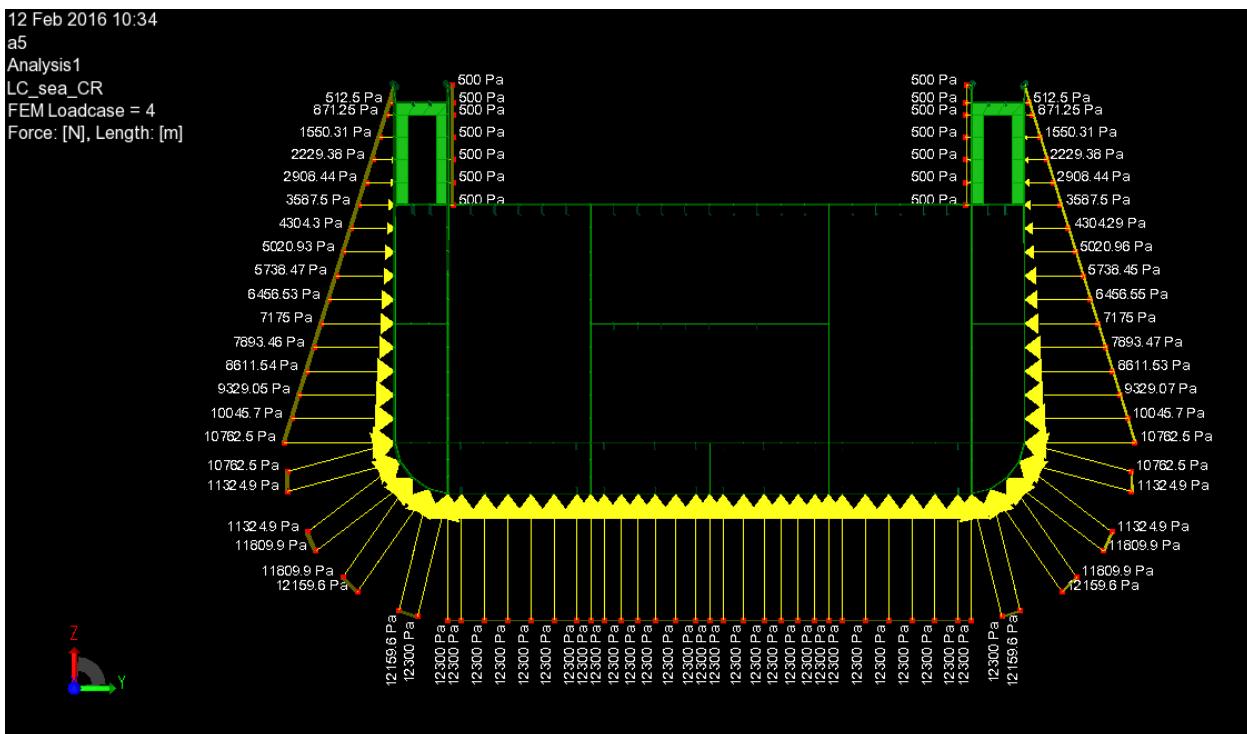


The Load annotation visibility control settings do not persist with the workspace or the GeniE installation.

### 3.13.2 Annotation Layout and the Refresh Graphics button

By default annotations are displayed screen aligned, or billboarded. This means they are always face on to the viewer no matter how the camera is oriented. Similarly their size is constant even as the view zooms in or out.

When inserted into the scene, through pressing *Apply* or *Ok* button on the *View Options* dialog the annotations are positioned relative to the surface they are labelling - be it left-side, right-side, above or below.

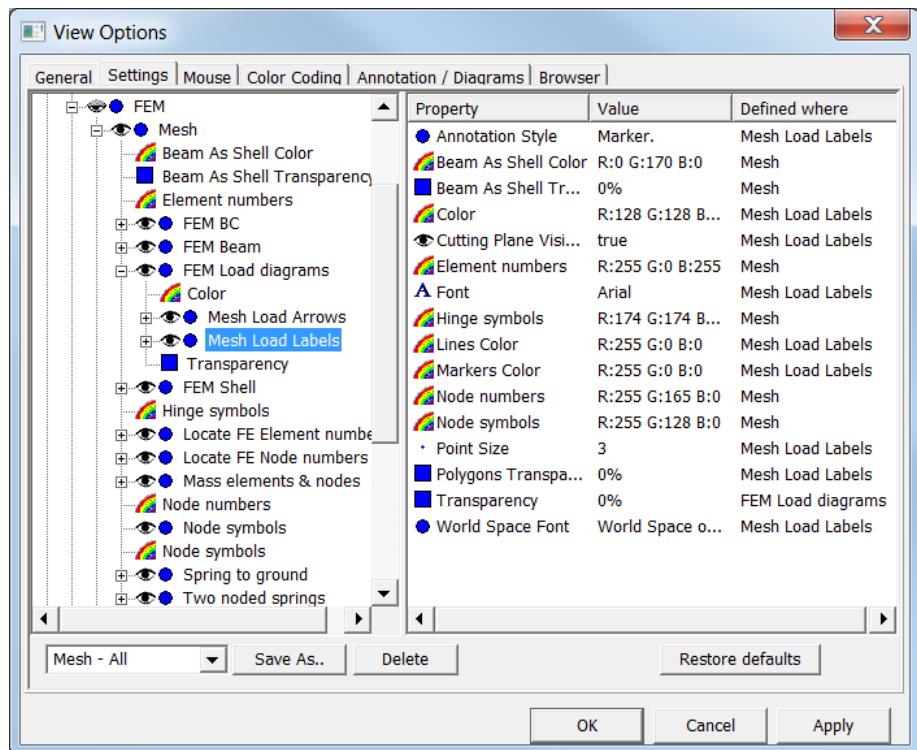


If the camera is rotated so the viewer is looking from the other side of the model the labels may then look incorrect. Click the *Refresh Graphics* button on the toolbar to refresh the layout based on the new camera position.

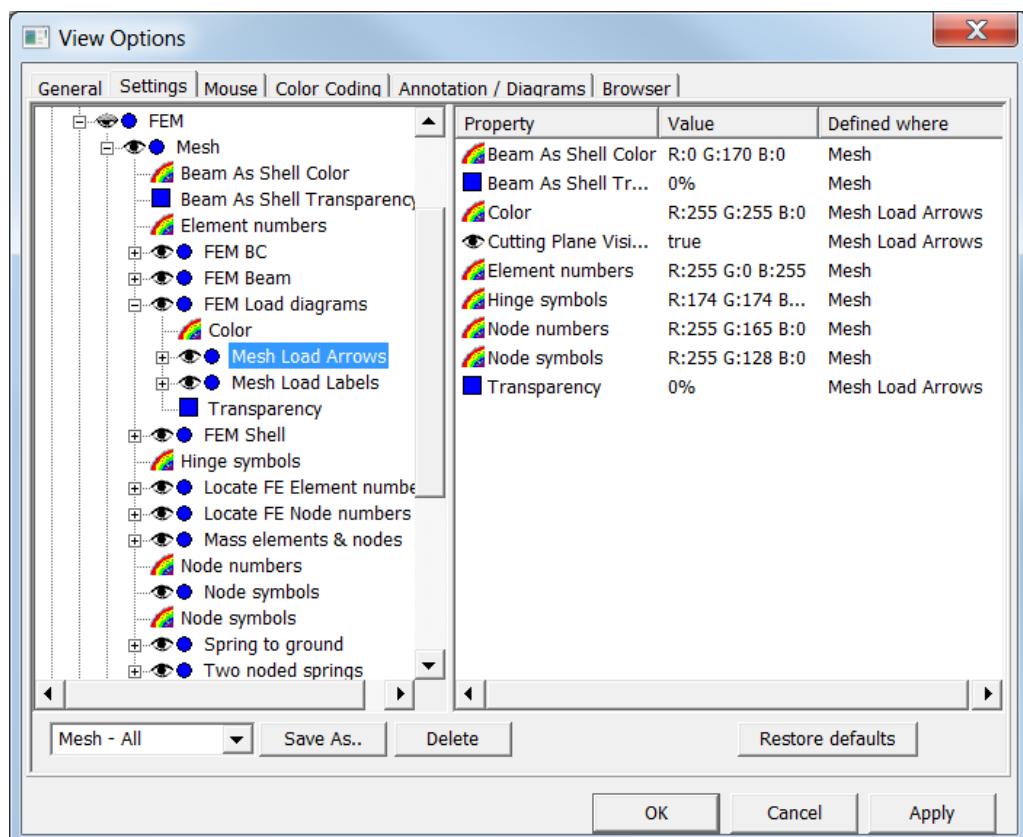
Note that when refresh graphics is clicked the orientation of arrowheads of load arrows is also recomputed such that as far as possible they are aligned with the current camera position.

### 3.13.3 Annotation Settings

There are a number of specific settings available for annotations from the *Settings* tab of the *View / Options* dialog under *FEM / Mesh / FEM Load Diagrams*. These include *font* and *colour* as well as *style* and *world space font*.



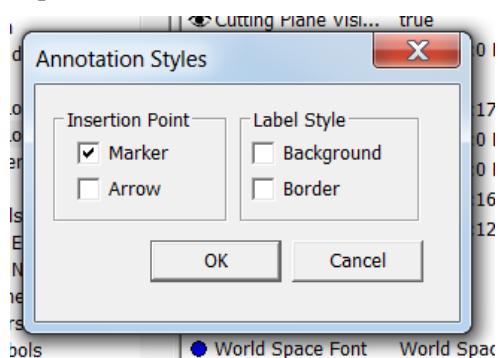
There are also settings to control the display of load arrows:



### **3.13.3.1 Annotation Style**

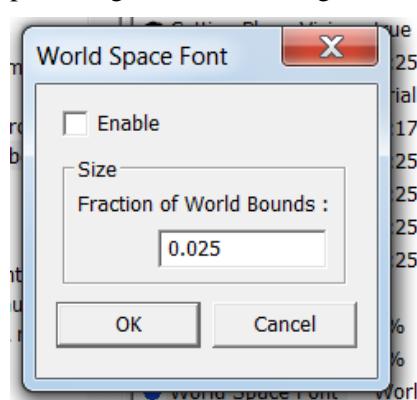
A number of annotation styles are available which include arrows, borders and backgrounds.

Note that there are currently issues with saving annotations to images with different aspect ratios. If image export is required it is recommended to use the default settings of text with no background or border. It is expected this limitation will be removed in future versions.



### **3.13.3.2 World Space Font**

If preferred a world space font can be chosen. The annotations will then align with the nearest major axis i.e. X and Y, and will not change as the camera position and orientation changes. The size is controlled by a percentage of the bounding box of the current model.

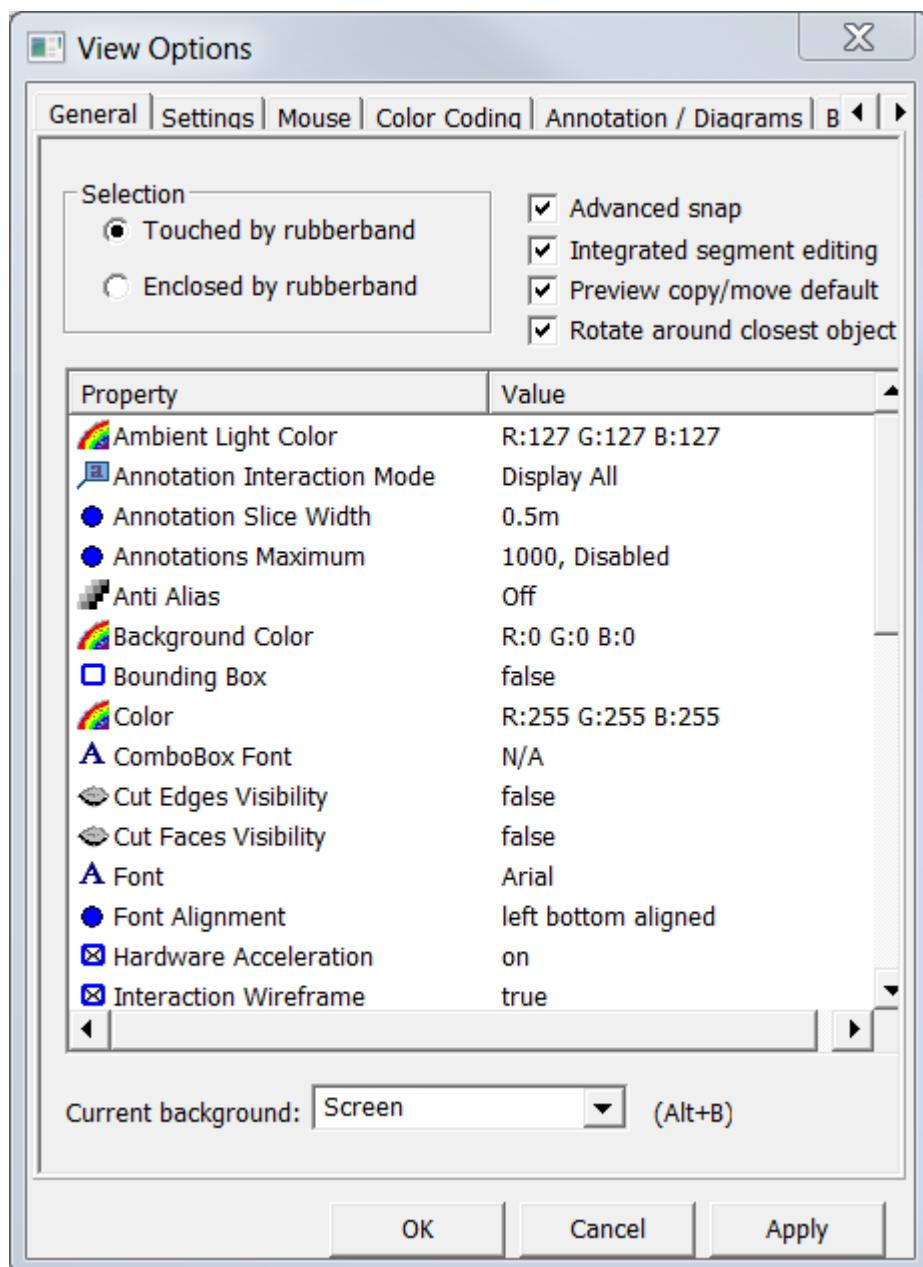


### 3.13.4 Performance options

Currently load annotations are slow to draw and displaying many of them can adversely affect draw performance and interactivity. There are a number of options available to help with this.

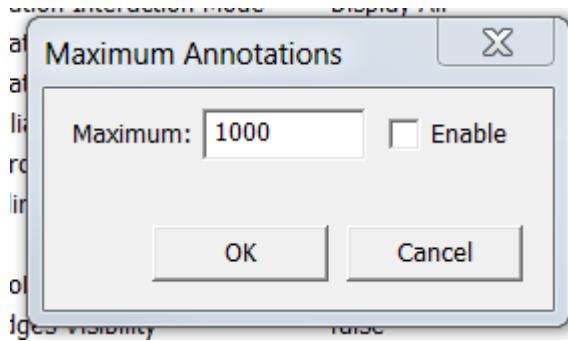
#### 3.13.4.1 Annotation Interaction Mode

This option is available on the *General* tab of the *View / Options* dialog. By default it is set to *Display All*. When toggled to *Hide All* annotations will be hidden when the user is moving the camera and redisplayed when the user stops the motion.



#### 3.13.4.2 Annotations Maximum

Available from the *General* tab of the *View / Options* dialog, when enabled this option will restrict the number of annotations generated to the number specified in the dialog:



Note there is no order guaranteed for the creation of the annotations. This option can be useful safeguard when working with large models and generating annotation slices. If a user inadvertently attempts to create a large number of annotations, e.g. by disabling the annotation slice, then this setting will prevent too many annotations being created.

### 3.13.5 Annotation Slices

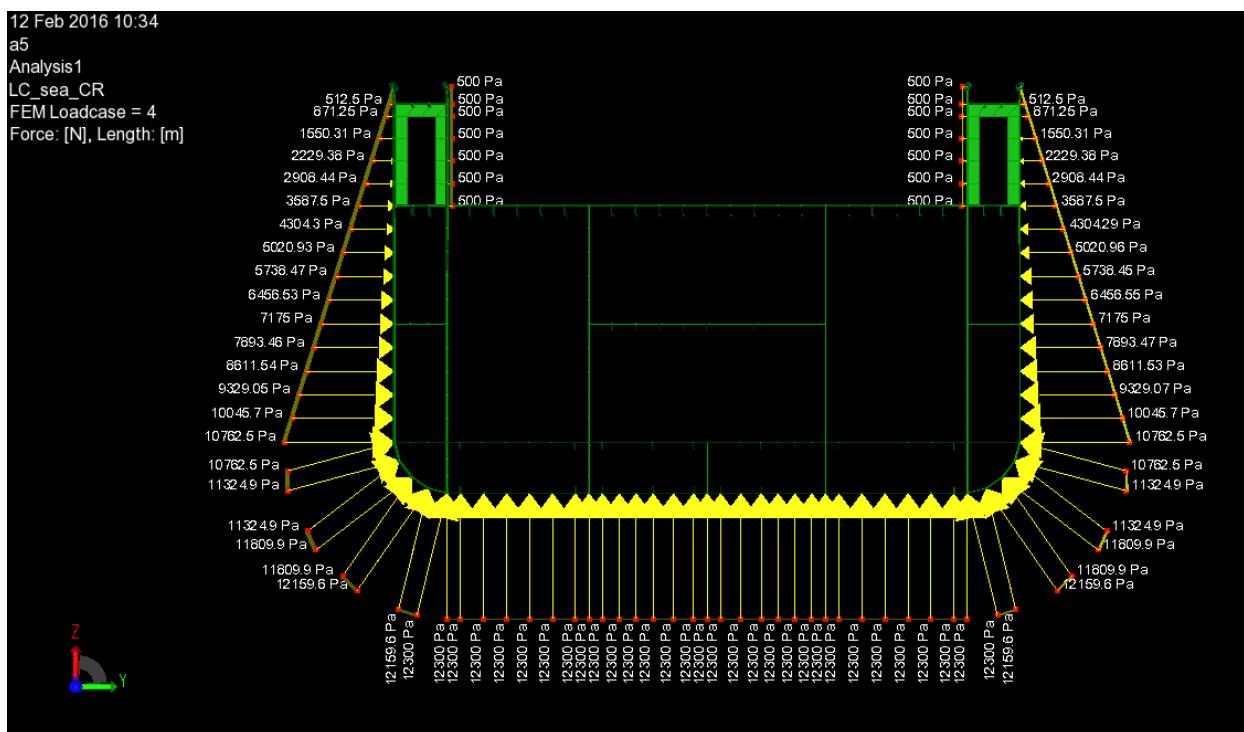
One of the more useful ways to restrict creation and display of annotations is via annotation slices. An annotation slice is a cut plane slice for which the *Clip Annotations* option has been enabled. This restricts display of annotations to only those that appear within the planes of the slice.

Annotation slices can be created via the *Cut Plane* button on the toolbar.

#### 3.13.5.1 Clipping versus Cutting

A cut plane *cuts* the model such that only items on one side of the plane are displayed. This operation is done on the GPU i.e. the entire model is loaded onto the graphics card and the display of *cut* geometry is suppressed at draw time.

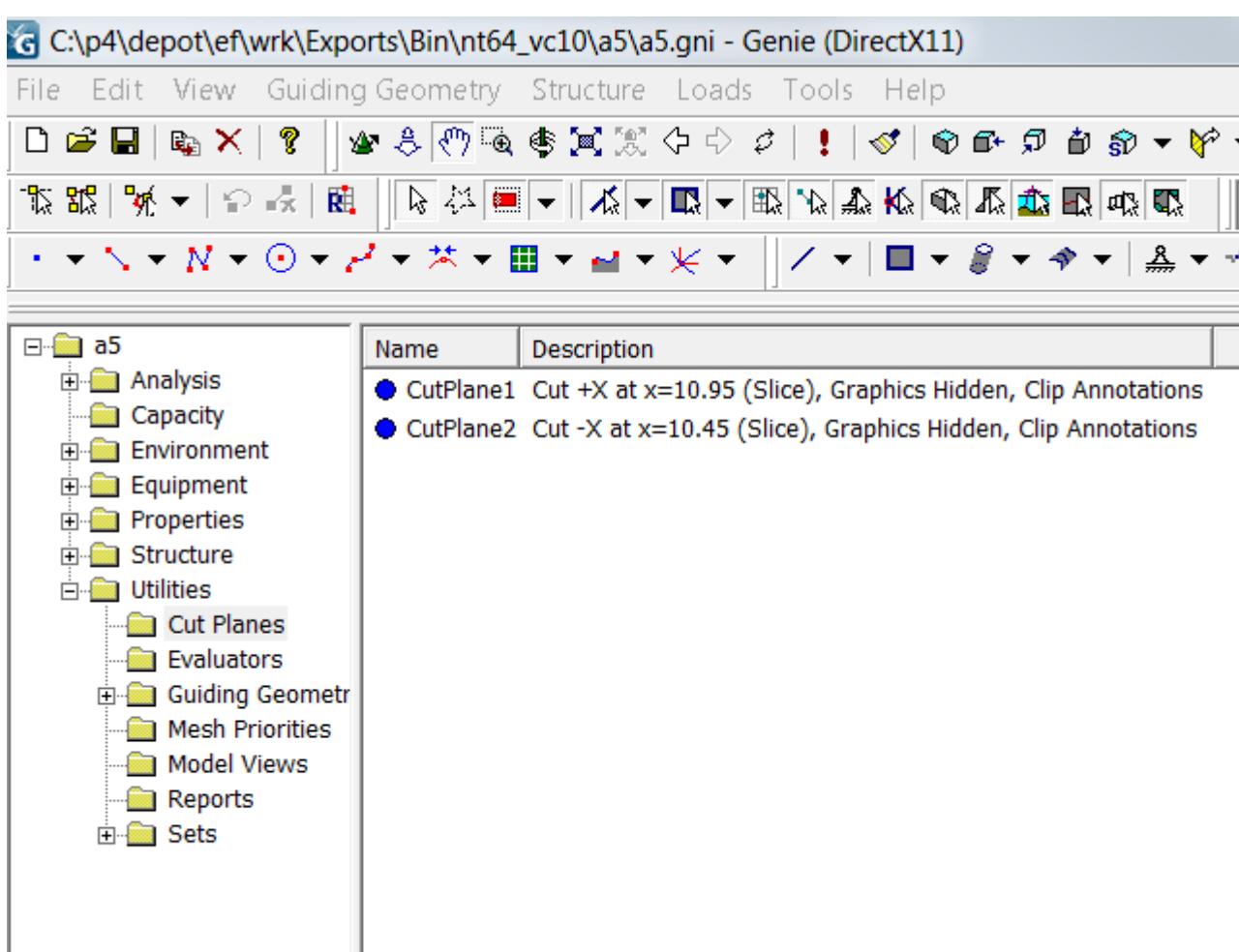
In the context of load annotations *clipping* is defined as an operation where the annotation is only created if it lies on the correct side of the cut plane. Note that the terms *cut* and *clip* are arbitrary and may not have the same usage outside this context.



*Clipping* is offered as an alternative to *cutting* for performance reasons i.e. even if cut planes are in place creating a large number of load annotations is a slow operation.

Any cut plane or slice can have clipping activated. This can be done by selecting the *Clip Annotations* option from the right click context menu for a cut plane or slice, either in the graphics view or the browser. Note that the *Show/Hide Graphics* toggle from the same menu is useful to prevent display of the transparent polygons representing the cut planes or slice.

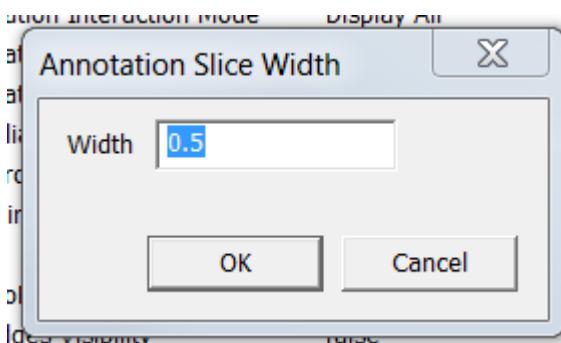
Annotation slices are simply a standard cut plane slice with the *Clip Annotations* option activated.



#### 6.8.4.2. Annotation Slice Width

The default width of slices created using the *Create Slice* variants of the *Cut Plane* toolbar button is controlled by the *Slice Width Percentage* option on the *General* tab of the *View / Options* dialog. This is expressed as a percentage of the bounds of the model and by default is set to be quite wide e.g. 30%.

The default width for annotation slices is controlled separately using the *Annotation Slice Width* option on the same tab.



This is expressed in the default units of the workspace e.g. metres or millimetres, and by default this is set to be very thin. The intention is that annotation slices should allow the user to quickly display a cross section with a single set of load values within the defined plane.

## 3.14 Making a Finite Element Model

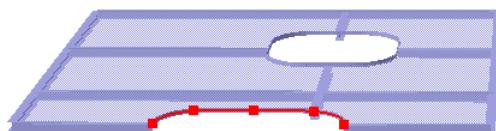
Three types of finite element models may be created in GeniE – either for use in a structural analysis, as a mass model for dynamic analysis (including hydrodynamic analysis followed by a structural analysis), or for the purpose of defining a mass model intended for a hydrodynamic analysis only.

This Section focuses on how to create a finite element model and the three possible representations. Also is described how to apply boundary conditions and how to adjust the finite element mesh.

GeniE comes with built-in knowledge for doing transparent meshing, in other words GeniE is capable of making a satisfactory mesh for most of the structures you can model with GeniE. It is, however, important that you can control how the mesh is generated either by using the mesh settings under **Edit/Rules/Meshing** (the settings here are all global to the model) or by manually control the mesh in local areas.

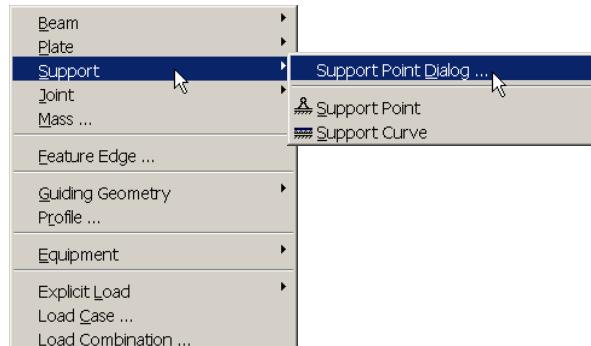
### 3.14.1 Boundary conditions

Boundary conditions may be inserted at support points or along an edge (along a beam, along a plate, inside a plate). The boundary condition may be free, fixed (default), supernode, prescribed displacement or a spring (spring between structure and ground).

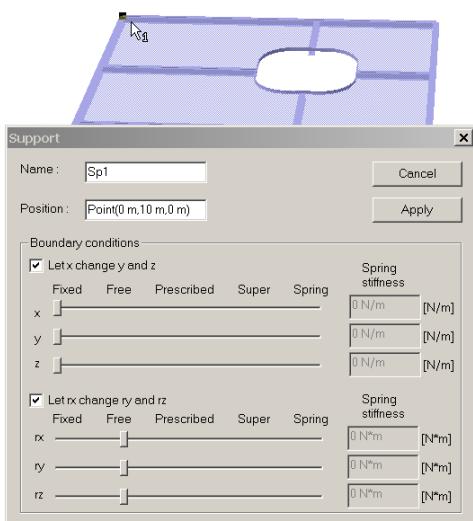


To illustrate the various boundary conditions and how they are applied, a common model (at left) will be used. It has one hole and a cut out along one of the plate edges. There is one model curve (highlighted in red).

To insert a boundary condition, the command **Insert/Support** may be used.

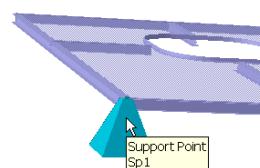


By using the support dialog option, you specify the boundary conditions before they are applied. Otherwise, the default settings are used (fixed in all dof) and you need to change afterwards.

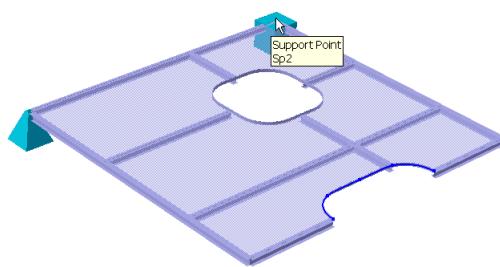


In this case a support point is inserted at the top left corner (all translation dof fixed and all rotational dof free).

When applied the support point looks like a pyramid to reflect that it is free to rotate.

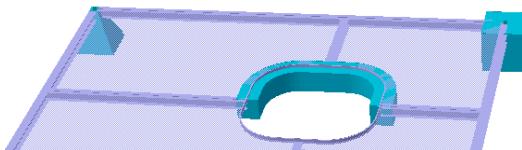


The boundary conditions are changed by selecting the support point, RMB and *Properties*.

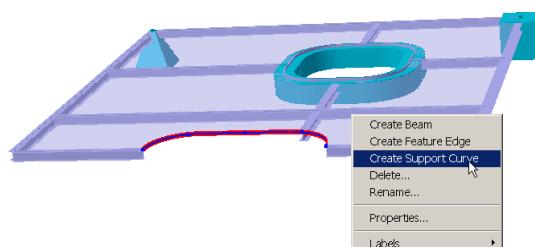


When using the command ***Insert/Support/Support Point*** (or from the toolbar) snap to point is being used and default boundary conditions are applied.

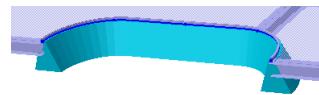
The support point generated will be fixed in all 6 dof and it is visualised as a solid cube. Changes to the support point will also be reflected in the visualisation, e.g. free to move in one direction.



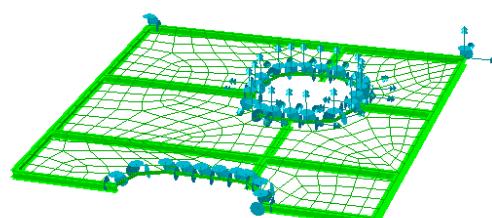
To insert boundary conditions along an edge the command ***Insert/Support/Support Curve*** (or from the toolbar) can be used. The command will prompt you for at least 3 snap points, in this case the first is at the left, second in middle, third and fourth at the right side (the fourth to complete the input). To change from default setting (fixed in all dof), select, RMB and *Properties*.



Alternatively, to insert support curves along a curved edge you may do this form selecting an already define model curve, RMB and *Create Support Curve*. Again default settings are applied which can be changed.

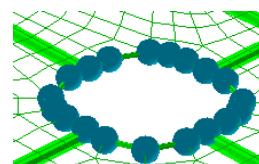


Note that when creating a finite element mesh, all relevant nodes will automatically receive boundary conditions. Typically, one finite element node for support points, and several nodes along a support curve.

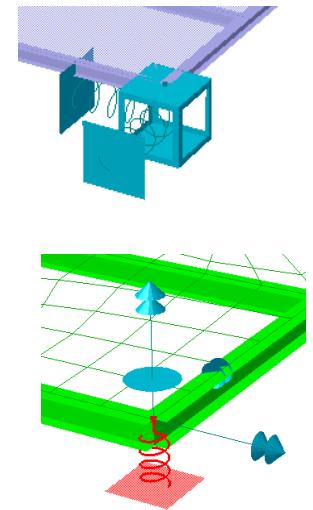
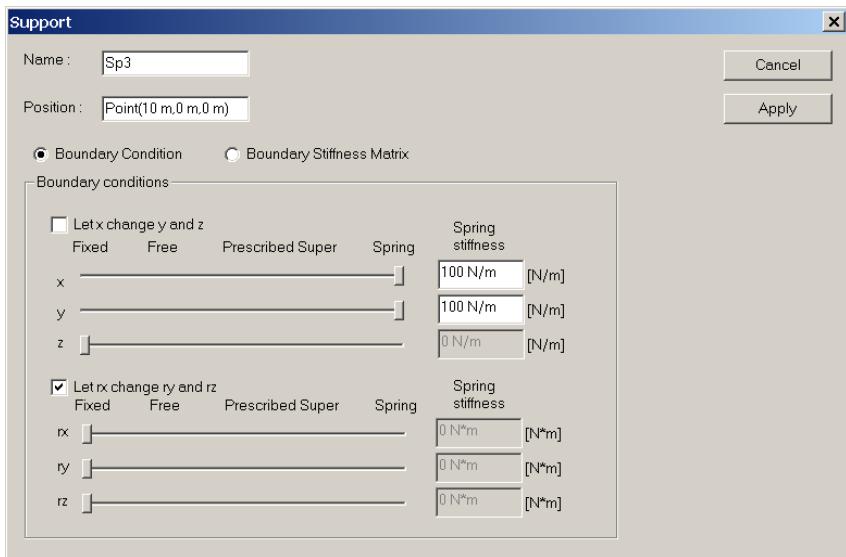


Boundary conditions automatically created in the finite element model based on the support points or curves.

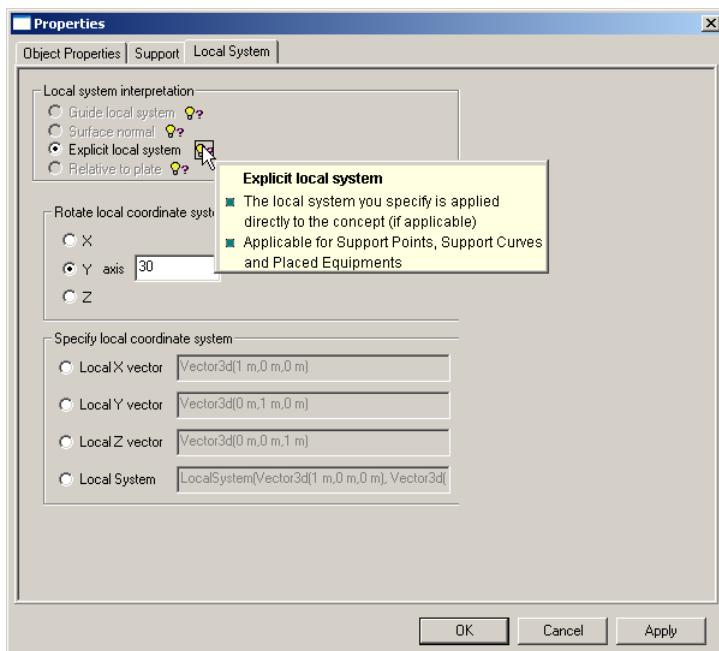
The boundary condition symbol changes depending on which dof is fixed, free, spring or supernode. A supernode is shown as a ball. Changing all dof to supernode along the inner hole leads to:



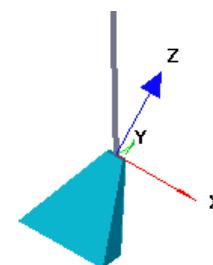
The example below shows a support point is inserted at one of the plate corners with a spring in global x and y directions. Graphical views of the support concept and corresponding finite element are shown.



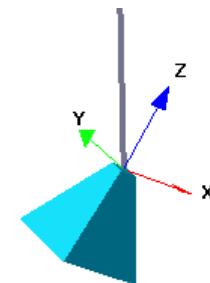
You may also apply a local coordinate system to the boundary conditions. Select a support point and force the context sensitive menu by RMB. When choosing *Properties* you have access to the Tab *Local System*, see below.



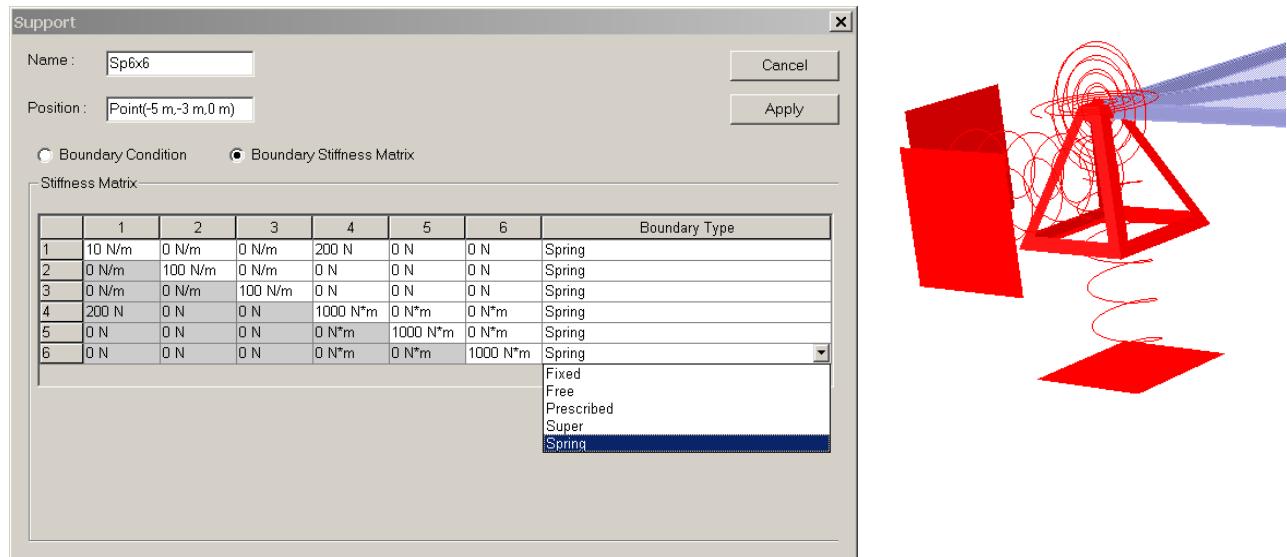
The support point is rotated 30 degrees around local y-axis, see figure below.



Please note that when rotating you do it relative to the local coordinate system of the support point. The picture below shows an additional rotation 45 degrees around local z-axis.



It is also possible to insert a 6x6 matrix where it is sufficient to specify the upper triangle of the matrix, since symmetry is enforced. This boundary condition is also referred to as Spring to Ground. A graphical view of the actual spring is shown.



You may also associate a local coordinate system for a 6x6 matrix in the same way as for a regular support point.

Boundary conditions may also be documented on the saved report.

A screenshot of Microsoft Excel showing a spreadsheet titled 'Microsoft Excel - Genie.xml'. The data is organized into columns: Name, X [m], Y [m], Z [m], BC-Type, X-Tra, Y-Tra, Z-Tra, X-Rot, Y-Rot, and Z-Rot. The rows contain information for three supports: Sp3, Sp3, and Sp6x6. The Sp3 entries are diagonal, while the Sp6x6 entry uses a 6x6 matrix. The data corresponds to the values shown in the 'Support' dialog box above.

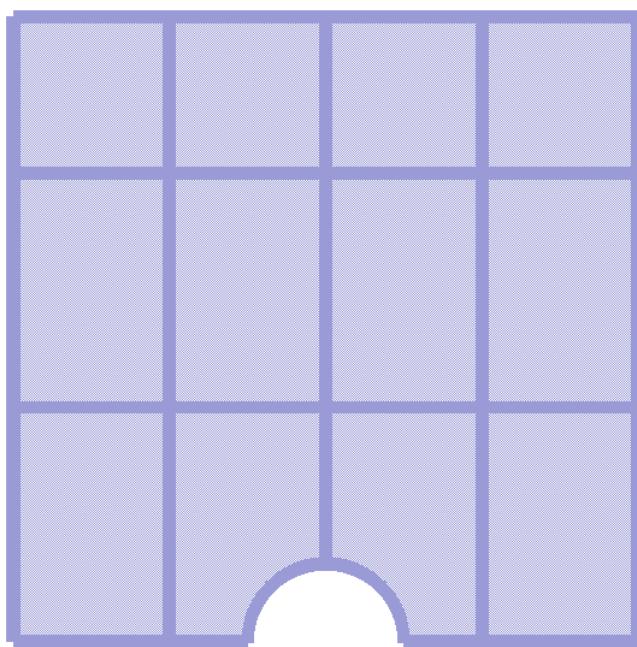
1	Name	X [m]	Y [m]	Z [m]	BC-Type	X-Tra	Y-Tra	Z-Tra	X-Rot	Y-Rot	Z-Rot			
2	Sp3	10.000	0.000	0.000	Diagonal	Spring	Spring	Fixed	Fixed	Fixed	Fixed			
3					Diagonal	100 N/m	100 N/m							
4	Sp6x6	-5.000	-3.000	0.000	Matrix	Spring	Spring	Spring	Spring	Spring	Spring			
5					X-Tra	10 N/m			200 N					
6					Y-Tra		100 N/m							
7					Z-Tra			100 N/m						
8					X-Rot	200 N			1000 N*m					
9					Y-Rot					1000 N*m				
10					Z-Rot						1000 N*m			
11														

The final boundary condition – prescribed displacement - has been described in the previous Chapter about Loads.

### 3.14.2 Create and control a mesh

The meshing may be done automatically using either default settings or settings specified by you. There are global settings you may alter to improve the settings (use the **Edit/Rules/Meshing**) or you can use local settings on how to improved the quality in certain areas of the model.

This Chapter will show how to create a finite element mesh and how each control (or mesh rule) influence the mesh.

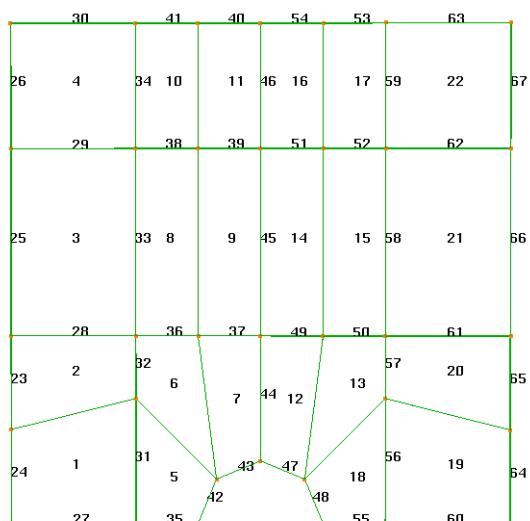


The same model is used to see the effect of the various changes in settings.

It consists of beams, plates, and a cut-out along the bottom plate edge.

There are many ways of creating the finite element mesh and how to visualise it. The following Sections give various ways of making alternative mesh. The finite element technology is used by Sestra when computing the displacements and forces, hence the quality of the finite element mesh is of crucial importance when performing the analysis.

#### 3.14.2.1 Create a default finite element mesh.



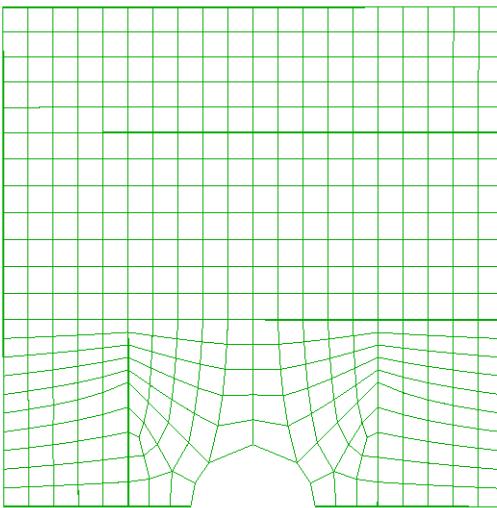
To create a finite element mesh you can use the command **Tools/Analysis/Create Mesh**.

To view the mesh details you must switch view in the combo box to e.g. Mesh All or Mesh Transparent unless you have specified a personal view where you can see the mesh details.

The mesh to the left has been generated without adjusting any mesh parameters.

The picture also shows the node symbols and the element numbers. You may increase the width of the lines from **View/Options/General** and adjust the *Line Width*.

### 3.14.2.2 Adjust global mesh density

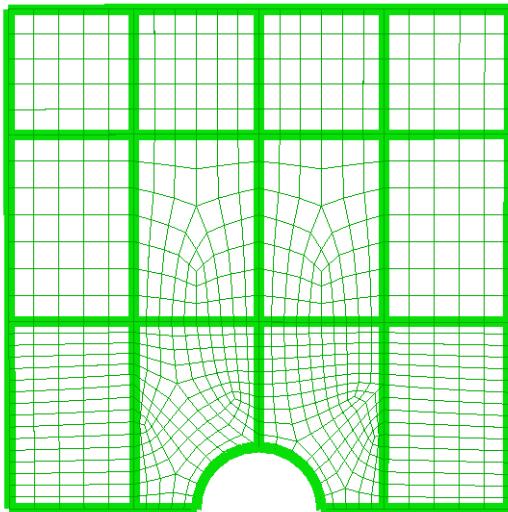


In this case the default global mesh density has been specified to 0.5 meters.

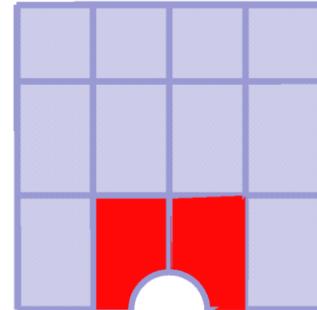
This is done by defining a mesh property from EditProperties|Mesh density. A property MD05 has been created with maximum element length 0.5 meters. This has been set to be the global default.

As can be seen, the mesh is much more dense than the example above.

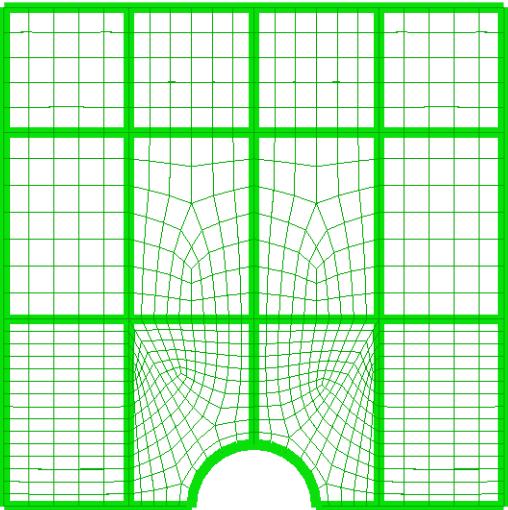
### 3.14.2.3 Adjust local mesh density



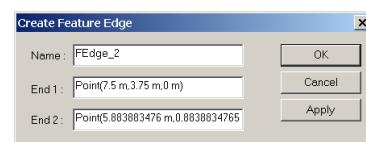
In this case local mesh density MD025 (max element length 0.25 m) has been applied to the selected plates as shown below. Note that the same settings apply to the beams that intersect the plates (i.e. not necessary to select all). The mesh is refined around the cut out and a transition zone towards the coarser mesh is automatically set up.



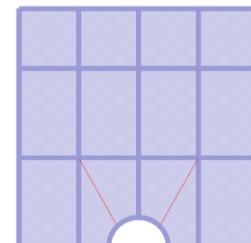
### 3.14.2.4 Adjust mesh with feature edges



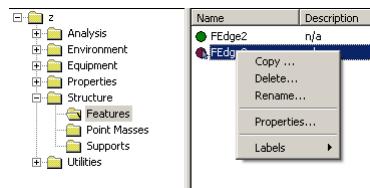
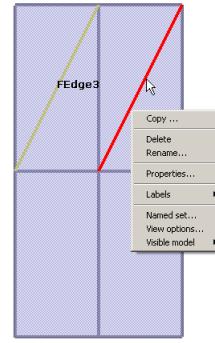
Two feature edges are inserted to improve the mesh close to the cut out. ***They are inserted from the menu Insert/Feature Edge.***



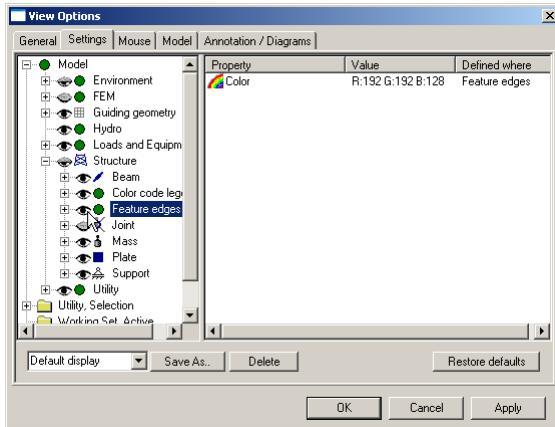
The picture to the right shows two feature edges inserted and the effect these have on the finite element mesh is shown to the left.



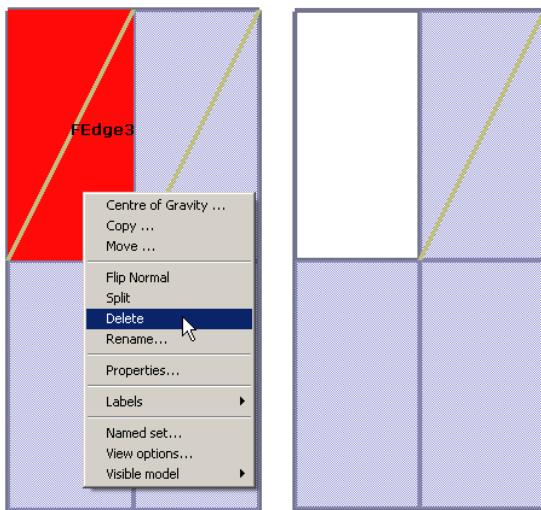
To verify, change or delete the feature edges you may select them from the browser or by graphic selection.



Before you can select them graphically make sure that the right options in the View|Option|Settings are ticked off. Alternatively, you may use the tool button for feature edge selection. See examples below.

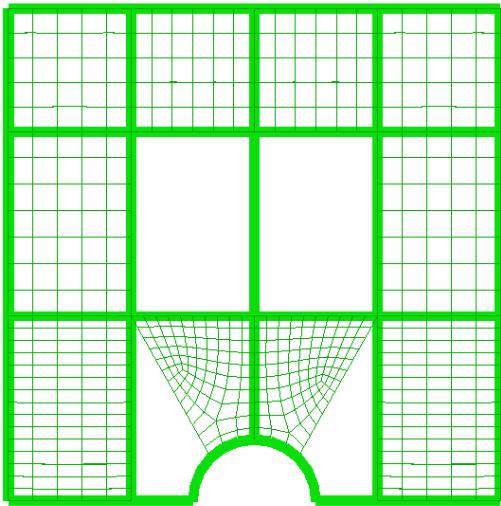


A feature edge will define a topology line (or curve) on a plate between two points. This means that the feature edge belongs to a plate and it is not possible to insert a feature edge when there are no plates. Furthermore, when deleting a plate including a feature edge the feature edge will also be deleted.

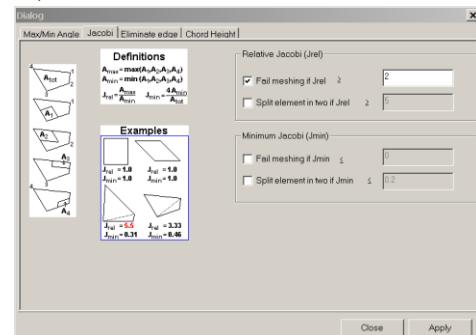


When deleting the plate, the feature edge *Fedge3* is automatically deleted,

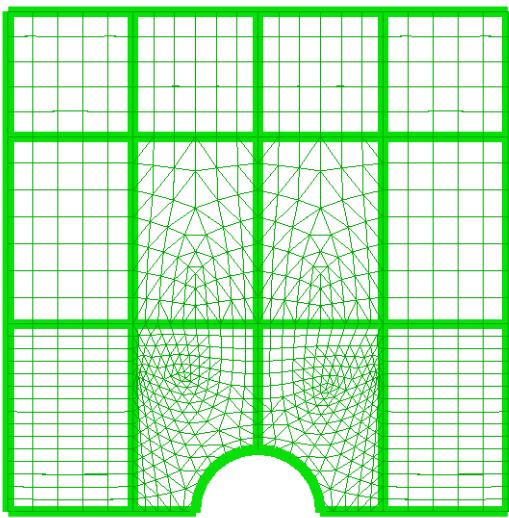
### 3.14.2.5 Check quality of mesh vs. Jacobi Matrix test



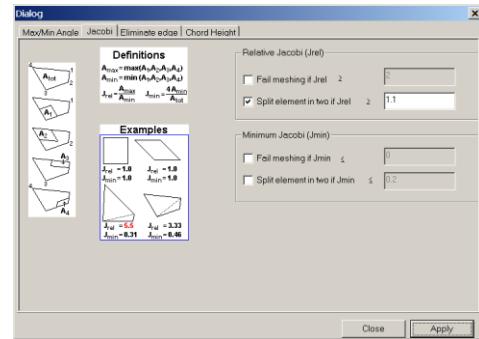
A Jacobi matrix test has been performed specifying to fail if the relative Jacobi matrix is larger than 2.0. As can be seen there are four plates where the mesh failed (GeniE will tell which plates that fail).



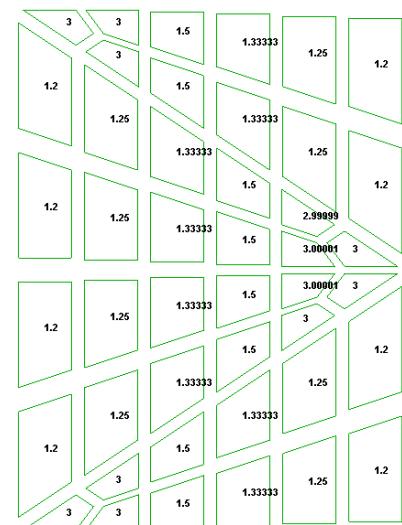
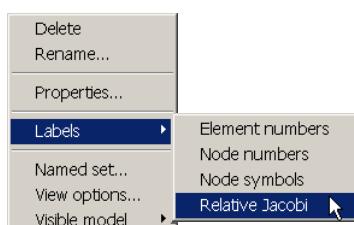
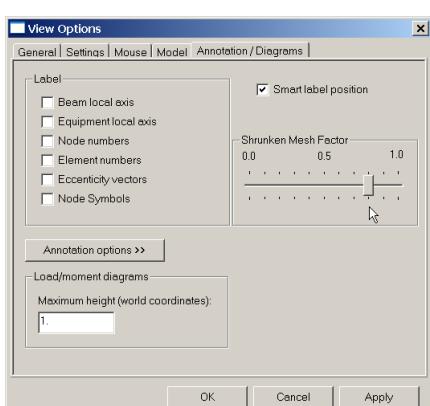
### 3.14.2.6 Improve quality of mesh vs. Jacobi Matrix test



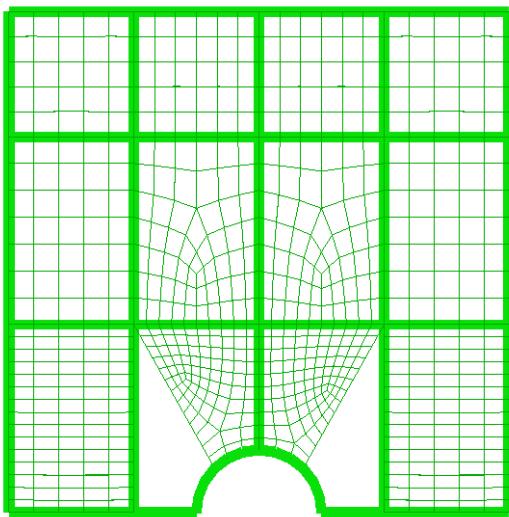
In this case it has been specified to insert triangular elements (split elements in two) when the relative Jacobi matrix is larger than 1.1.



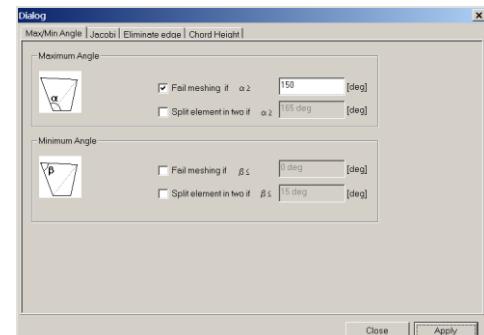
The results from a Jacobi determinant check is possible to visualise for 1<sup>st</sup> order quadrilateral elements. This example also shows how the shrunken mesh factor works.



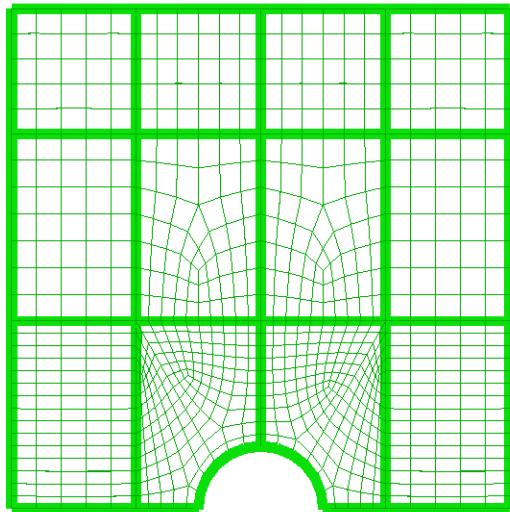
### 3.14.2.7 Check quality of mesh vs finite element angles



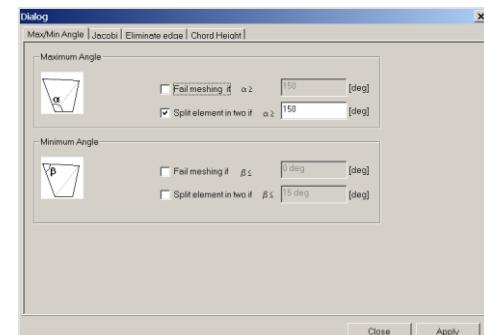
Two parts of the model was not possible to mesh because the element angles were above the failing angle. In this case GeniE is instructed to fail meshing if angles are exceeding 150 degrees.



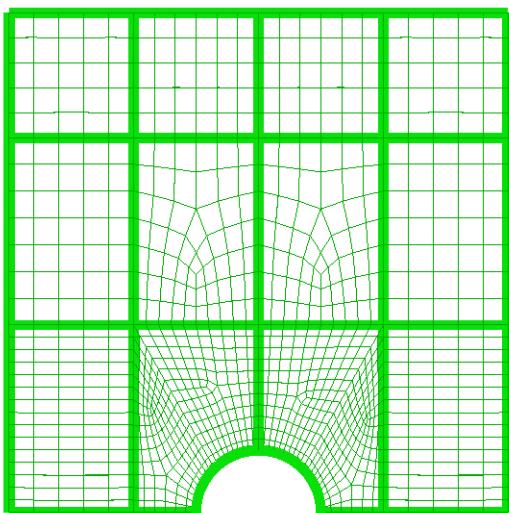
### 3.14.2.8 Improve quality of mesh vs finite element angles



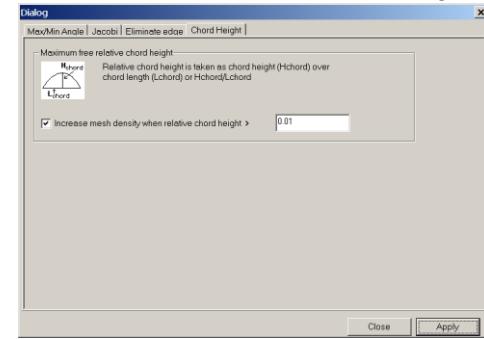
In this case the program is instructed to split elements in two if the mesh angle of the automatic created mesh is above 150 degrees.



### 3.14.2.9 Adjust mesh along the curved edge



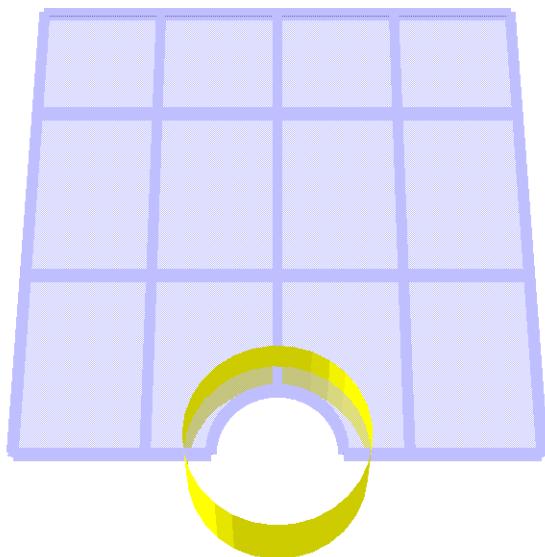
The option of controlling the relative chord height of mesh along a curved edge has been used. The relative chord height has been set to 1.1 and a dense mesh is established along the curve.



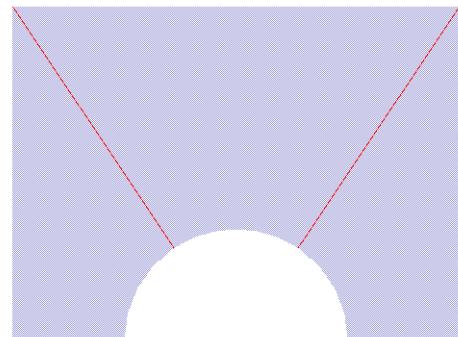
### 3.14.2.10 Controlling the mesh close to the cut out

This example shows how to create a mesh that is perpendicular to the curved edge of the cut out. The procedure is to create a profile that is larger than the original cut out and to do a cut operation to insert a new curved feature edge.

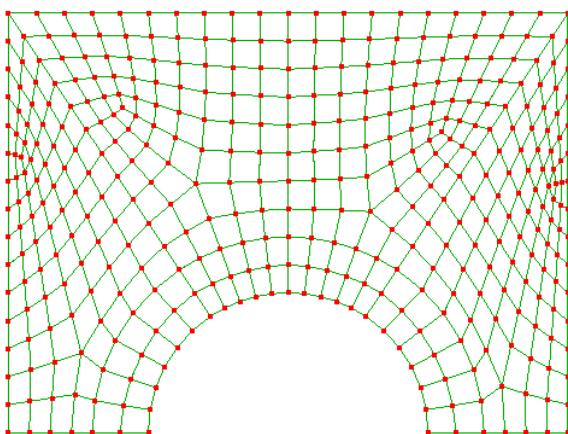
The pictures below explain how this can be achieved.



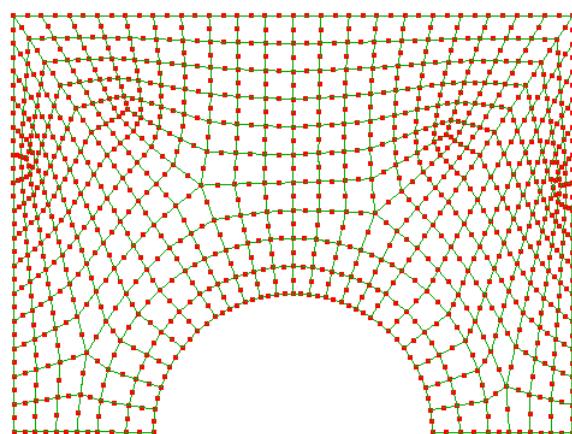
*Make a cut with a profile larger than the cut out*



*Inserting two feature edges in addition to the feature edges inserted at the new curved cut lines*



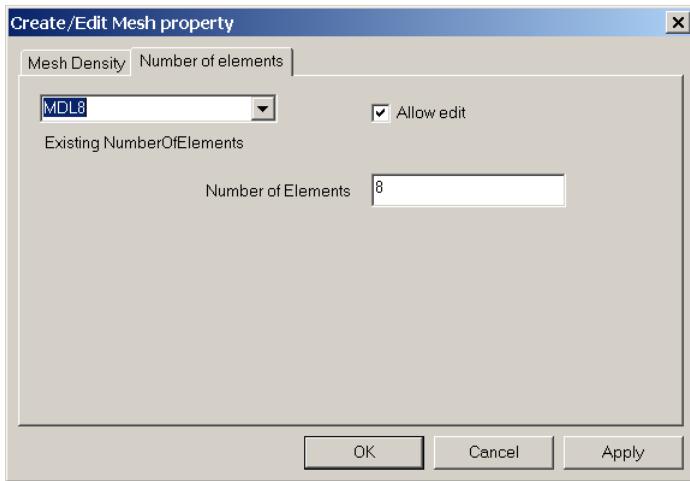
*The finite element mesh using 1<sup>st</sup> order elements perpendicular to the curvature*



*The finite element mesh using 2<sup>nd</sup> order elements perpendicular to the curvature*

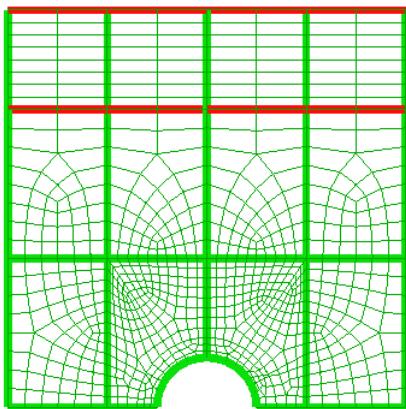
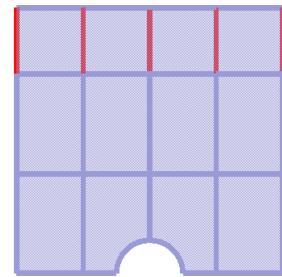
The feature **Tools/Structure/Geometry/Simplify Topology** should be used when you have done changes to the model that influences the topology. This feature is described under Rules setting and works only for planar plates and beams.

### 3.14.2.11 Controlling the mesh along straight lines

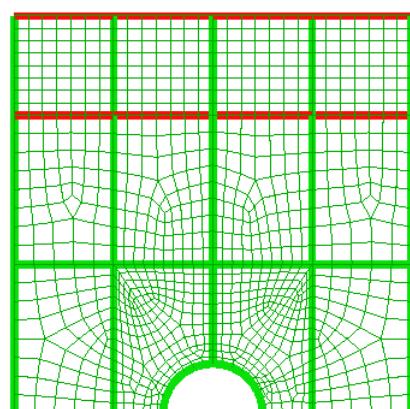


Using this feature it is possible to control number of elements (with equal length) along a straight line. These settings may be applied to beams and feature edges.

In this example the highlighted beams have been assigned 8 equal elements.

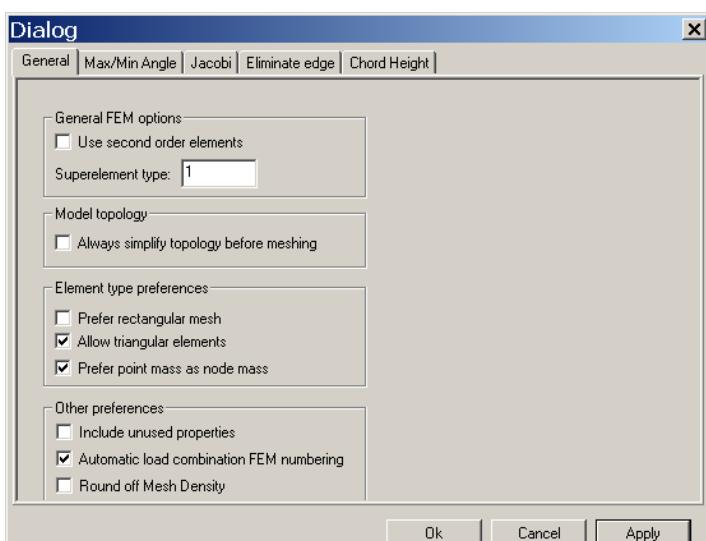


*Number of elements along each segment set to 2*



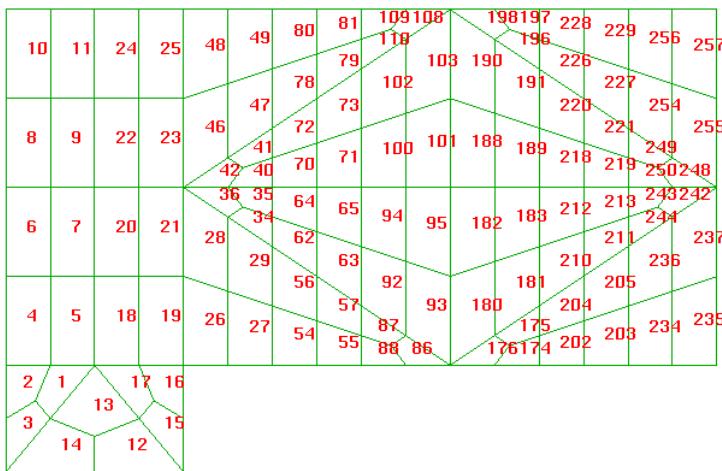
*Number of elements along each segment set to 7*

### 3.14.2.12 Specify type of elements to be used in FE mesh



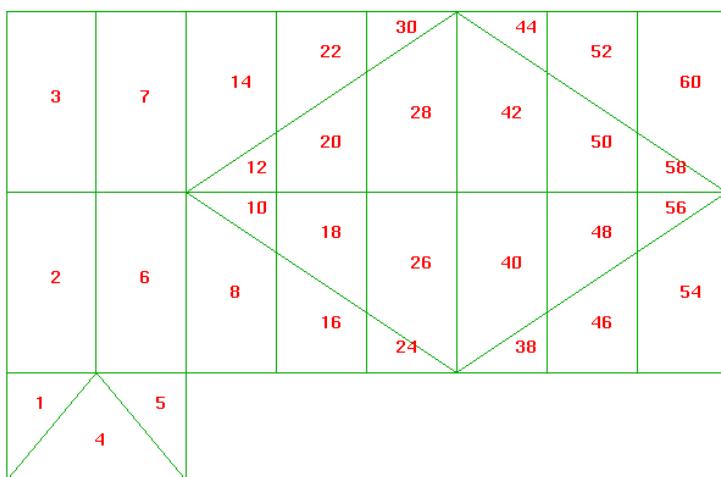
Per default GeniE will create quadrilateral and triangular finite elements. You may override this setting by using the **Edit/Rule/Mesh settings**.

The example below shows the difference in using this rule.

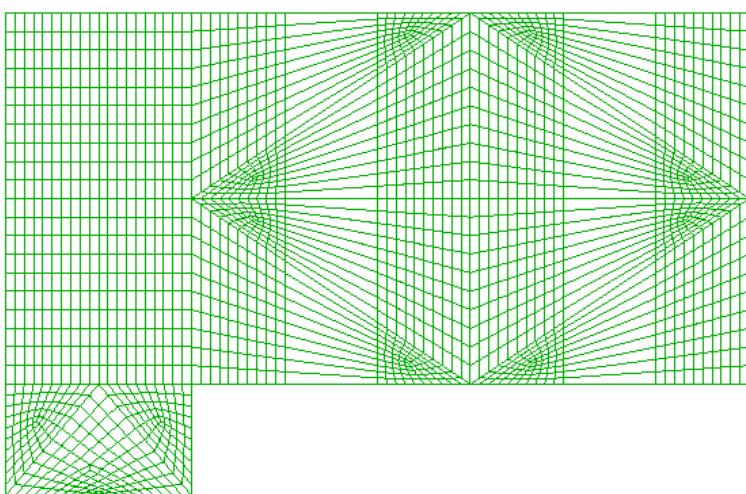


No triangular elements may be used.  
There are no specifications of mesh density.

The finite element mesh consists of quadrilateral elements only.



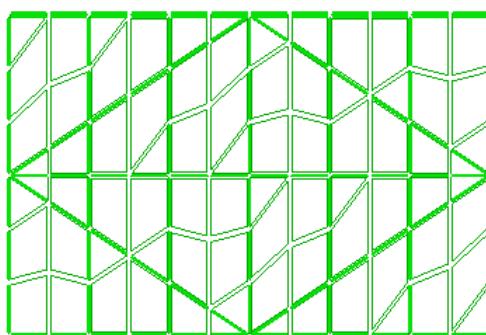
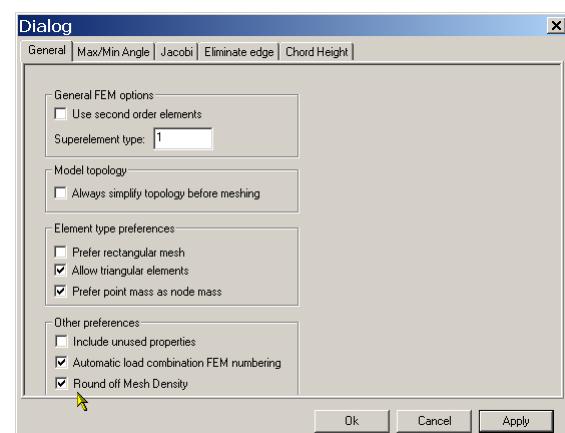
Default mesh rule to allow for triangular elements has been used. There are no specifications of mesh density.



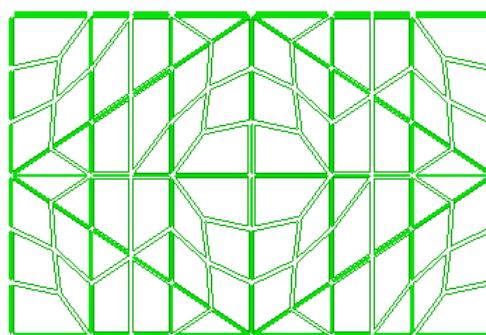
Preferred rectangular mesh and no triangular elements. There are no specifications of mesh density

### 3.14.2.13 Use feature for round off mesh density

When you have specified settings for mesh densities, GeniE will ensure that the maximum mesh densities are according to or smaller than the settings. In some cases this may lead to a more dense mesh than you desire. In such cases you can instruct GeniE to work with approximate mesh density settings, i.e. open up for finite elements having a slightly larger size than the maximum size specified by you. You do this from **Edit/Rules/Mesh settings** using the *Round off Mesh Density*.



A maximum mesh density of 2.0 m has been specified. No finite elements are larger than this size since the *Round off Mesh Density* is not activated.



A maximum mesh density of 2.0 m has been specified. There are elements larger than size 2.0 (but close to) since the *Round off Mesh Density* is activated.

The mesh changes and is in this case a more symmetric mesh.

### 3.14.2.14 Setting mesh priorities

When specifying various types of mesh settings (global from **Edit/Rules/Meshing** or local by applying properties to individual objects), GeniE uses the following priorities when creating the finite element mesh.

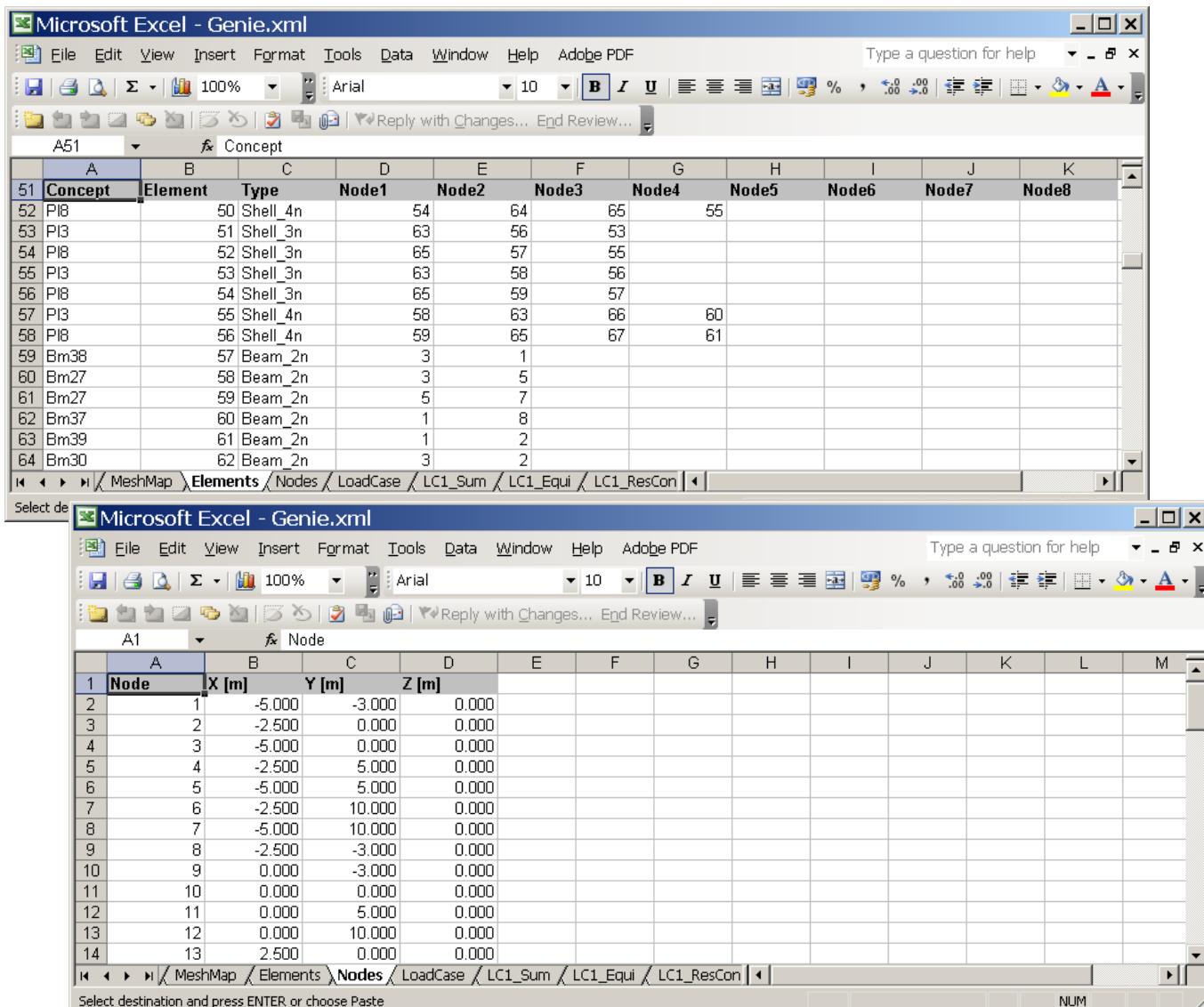
1. Number of elements along a line (beam or feature edge)
2. Mesh density applied to a line (beam or feature edge) and a plate
3. Global settings specified in the mesh rules
4. If there are no settings, the settings specified along edge are inherited

Note also that the create mesh operation is not logged on the journal file. You can, however, add this command manually to the journal file (or run it from the CLI window) using the following command **CreateMesh();**. This means you can re-run your model including meshing of the model.

The mesh settings are logged on the journal file.

### 3.14.2.15 Tabular verification of the mesh

You may find more details about a finite element mesh in a saved report under the Tab Nodes and Elements, remember to tick off for these details when creating the report.



The screenshot shows two Microsoft Excel windows side-by-side. Both windows have the title 'Microsoft Excel - Genie.xml'. The top window is titled 'Elements' and displays a table with columns: A (Index), B (Element), C (Type), D (Node1), E (Node2), F (Node3), G (Node4), H (Node5), I (Node6), J (Node7), K (Node8). The bottom window is titled 'Nodes' and displays a table with columns: A (Index), B (Node), C (X [m]), D (Y [m]), E (Z [m]). Both tables contain numerical data corresponding to the model's geometry and mesh.

A	B	C	D	E	F	G	H	I	J	K	
51	Concept	Element	Type	Node1	Node2	Node3	Node4	Node5	Node6	Node7	Node8
52	P18	50	Shell_4n	54	64	65	55				
53	P13	51	Shell_3n	63	56	53					
54	P18	52	Shell_3n	65	57	55					
55	P13	53	Shell_3n	63	58	56					
56	P18	54	Shell_3n	65	59	57					
57	P13	55	Shell_4n	58	63	66	60				
58	P18	56	Shell_4n	59	65	67	61				
59	Bm38	57	Beam_2n	3	1						
60	Bm27	58	Beam_2n	3	5						
61	Bm27	59	Beam_2n	5	7						
62	Bm37	60	Beam_2n	1	8						
63	Bm39	61	Beam_2n	1	2						
64	Bm30	62	Beam_2n	3	2						

A	B	C	D	E	F	G	H	I	J	K	L	M
1	Node	X [m]	Y [m]	Z [m]								
2	1	-5.000	-3.000	0.000								
3	2	-2.500	0.000	0.000								
4	3	-5.000	0.000	0.000								
5	4	-2.500	5.000	0.000								
6	5	-5.000	5.000	0.000								
7	6	-2.500	10.000	0.000								
8	7	-5.000	10.000	0.000								
9	8	-2.500	-3.000	0.000								
10	9	0.000	-3.000	0.000								
11	10	0.000	0.000	0.000								
12	11	0.000	5.000	0.000								
13	12	0.000	10.000	0.000								
14	13	2.500	0.000	0.000								

There is also a table showing the relation between concepts and finite elements, i.e. which finite elements belonging to a concept (beam or plate).

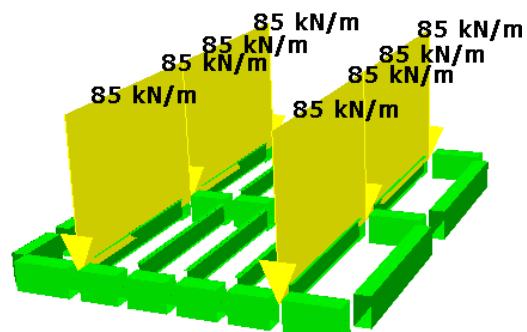
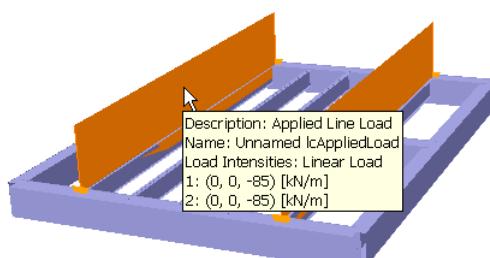
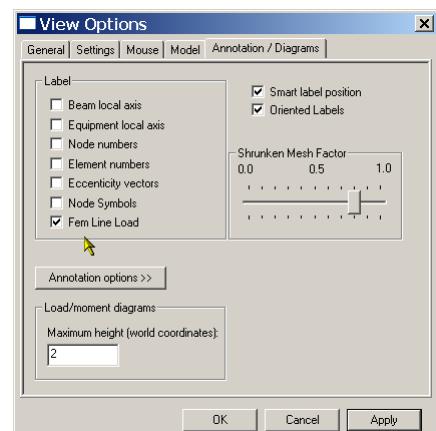
	A	B	C	D	E	F	G	H	I	J	K
1	Name	NumEle	Elem1	Elem2	Elem3	Elem4	Elem5	Elem6	Elem7	Elem8	Node1
2	Bm1	1	79								
3	Bm10	6	89	111	135	151	175	197			
4	Bm11	1	196								
5	Bm12	2	123	163							
6	Bm13	1	150								
7	Bm14	1	134								
8	Bm15	1	110								
9	Bm16	1	174								
10	Bm17	2	74	76							
11	Bm18	4	92	94	96	98					
12	Bm2	2	113	153							
13	Bm20	2	84	106							
14	Bm21	4	115	117	119	121					

### 3.14.2.16 Graphic verification of the finite element loads

You can graphically verify the finite element loads by using the feature for such from **View/Options/Annotation/Diagrams**.

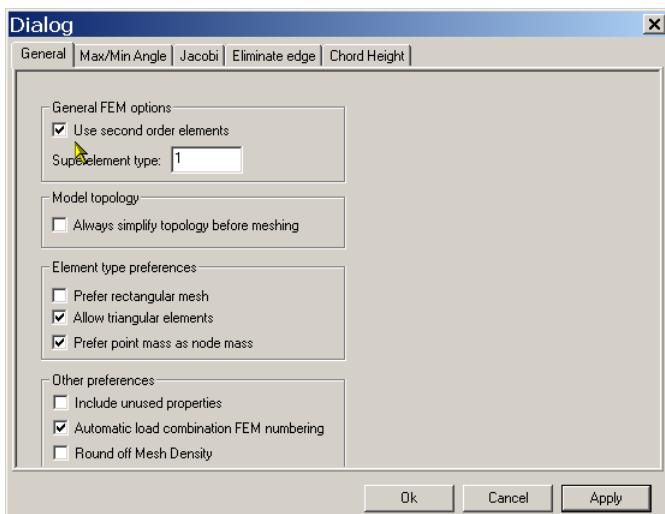
The picture below to the left shows the applied line loads while the finite element loads are shown on the figure down to the right.

As can be seen, the values are in accordance with each other; hence all loads are safely transferred from the concept model to the finite element model.



### 3.14.3 Element types

GeniE can create both 1<sup>st</sup> and 2<sup>nd</sup> order elements when creating a finite element model. You choose which element formulation to use either from the rules setting for mesh generation **Edit/Rules/Meshing**.



When you have specified 2<sup>nd</sup> order elements in the rules settings, this will show on this sheet. You may overrule the settings by deselecting in this sheet.

Please note that element types FQAS and FTAS are specified from the command line interface. Please see chapter 5.3.2 for references.

The following finite element mesh types are generated and exported to the FEM file. More details about each element may be found in Sestra User Manual.

Name	Type	Order	Comments
2-node beam element	BEAS	1 <sup>st</sup>	
3-node beam element	BTSS	2 <sup>nd</sup>	Cannot be used in framecode checking
Triangular flat thin shell element	FTRS	1 <sup>st</sup>	
Quadrilateral flat thin shell element	FQUS	1 <sup>st</sup>	Inserted when adjusting mesh rules to split elements
Quadrilateral sub parametric curved thick shell element	SCQS	2 <sup>nd</sup>	
Triangular sub parametric curved thick shell element	SCTS	2 <sup>nd</sup>	Inserted when adjusting mesh rules to split elements
Quadrilateral flat thin shell with drilling dof	FQAS	1 <sup>st</sup>	Includes the rotational dof around the axis perpendicular to the membrane in the membrane formulation
Triangular flat thin shell with drilling dof	FTAS	1 <sup>st</sup>	- o -
Non-structural 2 node beam element	BEAS	1 <sup>st</sup>	Special variant of BEAS with no contribution of the structural stiffness
Truss element	TESS	1 <sup>st</sup>	Element type with no bending stiffness
Spring to ground	GSPR	1 <sup>st</sup> & 2 <sup>nd</sup>	Includes the 6x6 matrix
Shim element	GLSH	1 <sup>st</sup> & 2nd	Special variant of the 2 node spring element with equal stiffness in two translation directions. No stiffness in other directions.
One node mass element	GMAS	1 <sup>st</sup> & 2nd	May be eccentric if connected to a finite element node with 6 dof.

Within the same model it is not possible to have both 1<sup>st</sup> and 2<sup>nd</sup> order element types.

### 3.14.4 Controlling the FEM Loadcase numbers

Chapter 3.8.5 explains how to control the loadcase names as well as the loadcase numbers to appear on the finite element model. When performing an integrated analysis completely from the GeniE domain, the sequence of FEM Loadcase numbers do not play an important role since you refer to the loadcase names when investigating the results. The exception is when you introduce large holes in the loadcase numbering; this will increase the physical size of the result data file.

However, when making a superelement model or using other postprocessors you need to be more in control of the FEM loadcase numbers. You document the FEM loadcase numbers from the browser or the printout. The examples below show both options; the first picture shows that the FEM Loadcase numbers have been sorted in integer order.

Name	Description	FEM Loadcase	FEM LC Rule
LC1	Load Case	1	Manual
LC2	Load Case	2	Manual
LC3	Load Case	3	Manual
LC4	Load Case	4	Manual
Comb_1	Load Combination	5	Automatic
Comb_2	Load Combination	6	Automatic

The FEM Loadcase number is changed from the loadcase property sheet (select the loadcase in the browser, RMB and choose *Properties*).

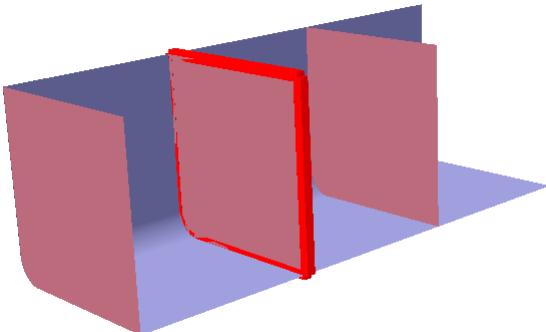
A	B	C	D	E	F	G	H	I	J	K	
1	Name	Description	FEM LC	Self Weight	Equip as Lo	Equip Rep	X-Acc [m/s^2]	Y-Acc [m/s^2]	Z-Acc [m/s^2]	X-For [kN]	Y-For [kN]
2	Comb_1	Load Combination	5	Yes			0.0000E+00	0.0000E+00	-9.8067E+00	-8.0000E+03	0.0000E+00
3	Comb_2	Load Combination	6	Yes			0.0000E+00	0.0000E+00	-1.9613E+01	-1.6000E+04	0.0000E+00
4	LC1	Load Case	1	Yes	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00	0.0000E+00	0.0000E+00
5	LC2	Load Case	2	No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00	0.0000E+00	0.0000E+00
6	LC3	Load Case	3	No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00	-8.0000E+03	0.0000E+00
7	LC4	Load Case	4	No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00	0.0000E+00	0.0000E+00

If you work with loads both in the *Load Case folder* and loads in the *Analysis Activity folder* you should note that when creating load combinations in the *Load Case folder*. One example is when you have wave loads in the *Analysis Activity folder* and regular loads and load combinations in the *Load Case folder*.

- FEM loadcase numbers may overlap.
- In such cases you should manually define the start FEM loadcase number for load combinations. The number must be higher than the largest FEM number for wave loads (loads defined with *Edit/Rule/Meshing* and specify variable numbering of FEM loadcase numbers)

### 3.14.5 Mesh only parts of model

If you want to create superelements out of a complete model you may use a feature to create a finite element model for parts of the model and export the mesh to a FEM file. The principles behind the feature are to create a set (which you want to mesh) and an activity in which you make the mesh.. The example below shows which steps (using the GUI as well as the script features) are involved in such operation.



The plate and the boundary conditions have been added to *MySet* (select the concepts, RMB, *Named set*)

The same model has one activity called *Analysis\_Regular* which will perform meshing and analysis of the entire structure.

The analysis has one loadcase *LC1\_Weight*.

When setting this activity current, a mesh operation will create a finite element model for the entire structure and the loadcases.

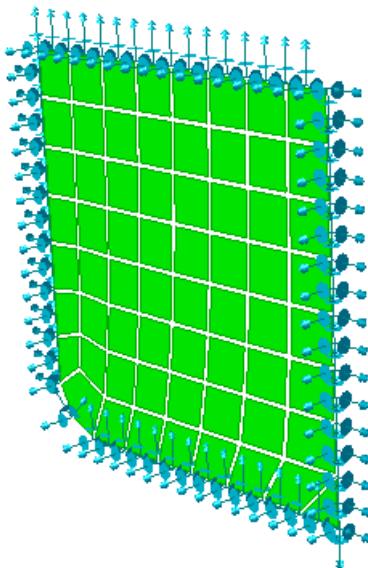
### 3.14.5.1 Mesh part of model using script language

```

119
120 MeshingAct = Analysis();
121 MeshingAct.add(MeshActivity());
122 MeshingAct.add(LoadResultsActivity());
123 MeshingAct.step(1).subset = MySet;
124

```

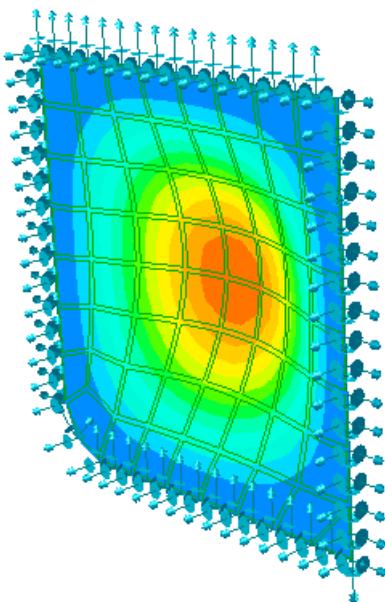
To create an activity that will mesh only the selected part it is necessary to define the script commands as shown to the left. In this case, it is the set called *MySet* that will be meshed in the activity *MeshingAct*.



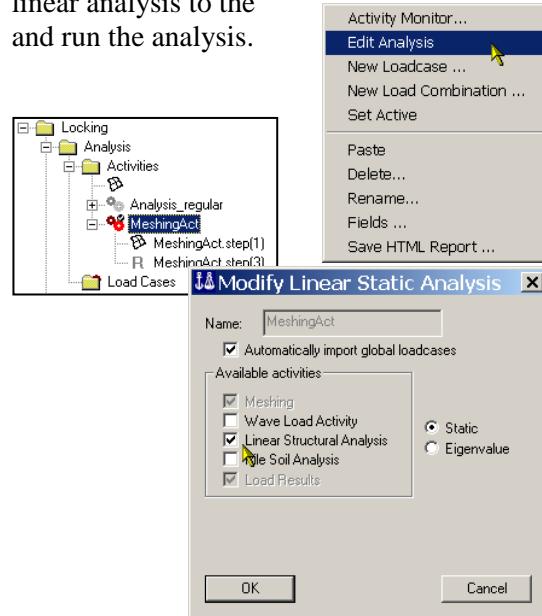
When making a mesh (Alt+M or **Tools/Analysis/Create mesh**) only the selected parts and the loadcases will be part of the model. You need to ensure that *MeshingAct* is the active activity.

	Name	Description	FEM Loadcase	FEM LC Rule
LC1_weight	Reference to LoadCase	1	Manual	
MeshingAct.step(1)	Meshing			
MeshingAct.step(3)	Load Results			

You can now export the FEM file from **Tools/Analysis/FEM file**.

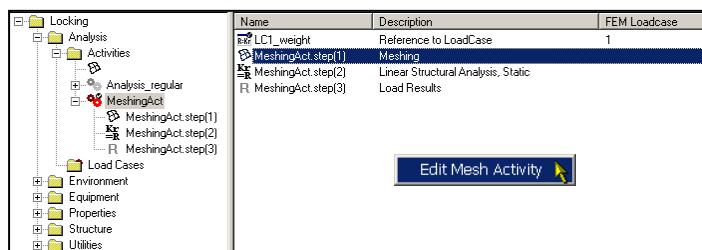


In case you want to run analysis of the selected part you can add linear analysis to the activity *MeshingAct* and run the analysis.

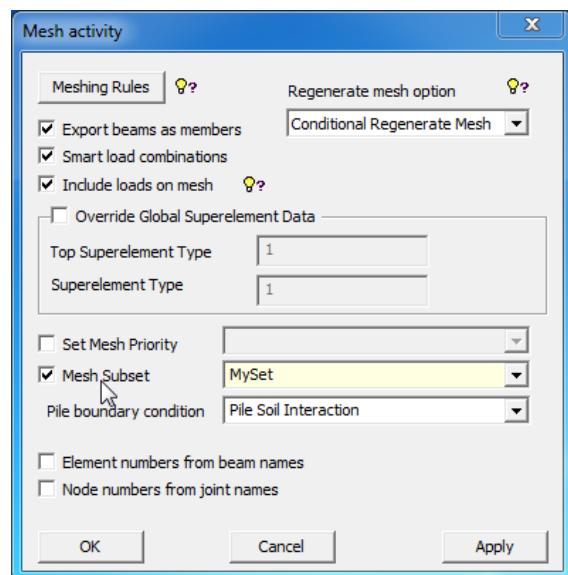


If you want to analyze the whole structure you set the other activity *Analysis\_rgular* to active and run analysis.

### 3.14.5.2 Mesh part of model using GUI



Select the mesh operation in the activity *MeshingAct*, RMB and choose *Edit Mesh Activity*.



You may now decide if you want to mesh and analyse only parts of the model from the Mesh Subset feature.

Please note that you can use one set only. If you have several sets you want to mesh at the same time you can group these into one set.

Also observe that you can access meshing rules as well as change the superelement numbering. This means that you can create several superelements and export these for a subsequent analysis using Presel to build an entire structure, please consult the Presel User Manual for more details.

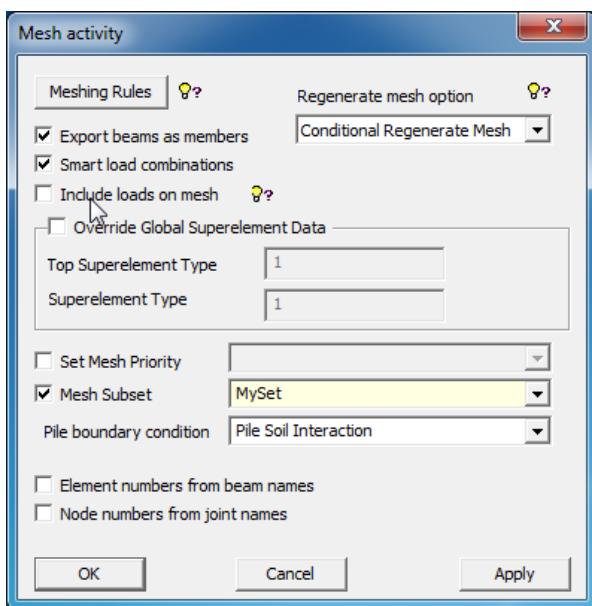
### 3.14.5.3 Exclude loads from the mesh

Sometimes you are only interested in generating a clean mesh file, without any loads on it.

In such scenario you can get considerable performance boost, depending on number of loads.

Such performance gain is extremely helpful while tuning your work on quality of your mesh with features like Partial Meshing, Feature Edges, Mesh Editing and many other.

There is no point to run the analysis with “Include loads on mesh” option turned off, so you are recommended to use ALT+M for meshing purpose. Then you will not need to alter activity setup in the analysis.



To exclude loads from the mesh you have to uncheck “Include loads on mesh” option in the Mesh Activity dialog.

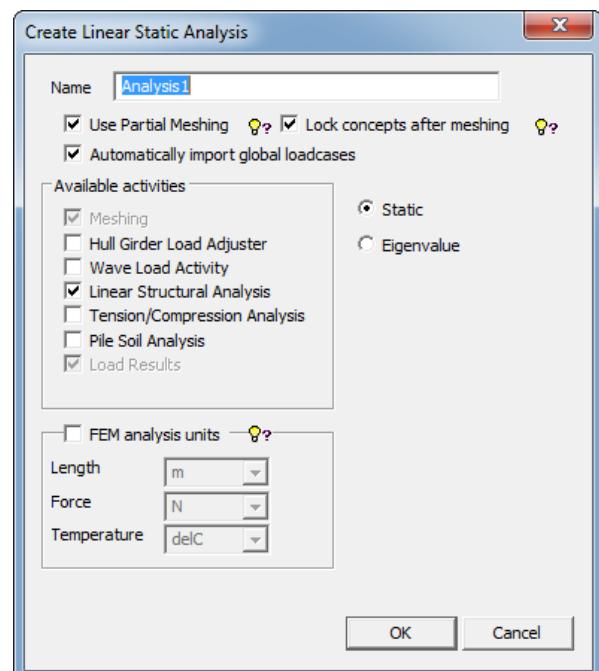
## 3.15 Run Structural Analysis

GeniE comes with predefined work processes for

- Linear static structural analysis
- Eigenvalue analysis
- Wave load analysis
- Pile Soil analysis
- Integrated wave, pile-soil and structural analysis
- Hull Girder Load Adjustment for Ship structures

Linear structural and eigenvalue analysis are covered herein, while Vol. 2 of this user manual documents the other options.

The work processes are available from **Tools/Analysis/Activity Monitor**, Alt+D, or from the Activity folder in the left browser (RMB and select *New Analysis*).



You may also specify several analysis runs, but remember (see Chapter 3.8 for more details).

- Different analysis runs must have same structure and boundary conditions. It is possible to have different loads.
- The load cases should be defined under the respective activity folders, and not under the load case folder.

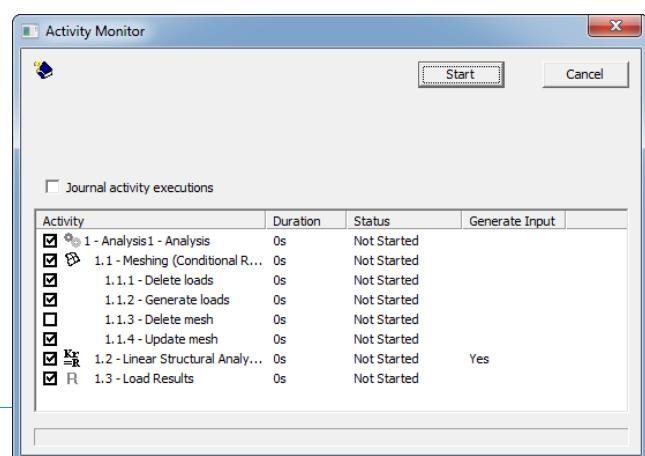
When starting a structural analysis in GeniE (**Tools/Analysis/Activity Monitor**) this implies

- Create a new finite element model to ensure that there is full consistency between the model and the finite element model. A new FEM-file is created.
- Run a direct analysis using Sestra (predefined multifront solver option and a full retracking of results), i.e. no superelement analysis. A result file (SIN) is created in addition to print files containing details about the analysis (the LIS and MNT files).
- Establish the relation between objects (beams, plates), loadcases, and result attributes. It is thus possible to select an object(s) and query for results.

The following example shows how to run a structural analysis and how you verify the details of it. Examples will also be given on how to modify analysis parameters and how to include analysis parameters to the journal file.

You start a linear static structural analysis by

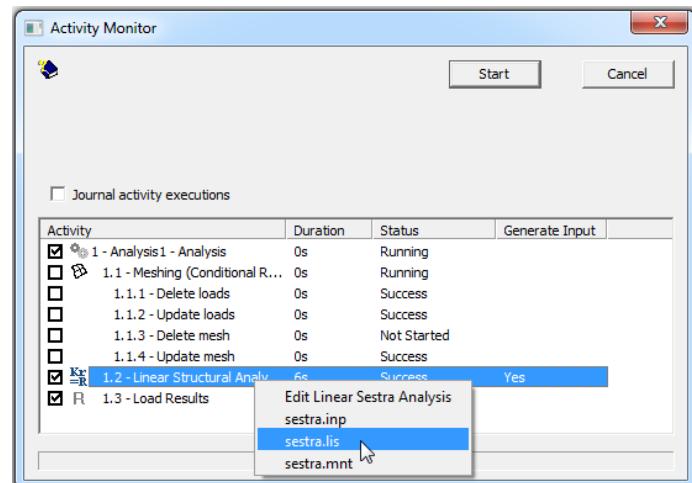
- **Tools/Analysis/Activity Monitor**, specify an analysis name and tick off the right box (see picture above)
- Click start on the *Activity Monitor* that appears in the graphics window



- The *Activity Monitor* reports status on each activity during execution

When the analysis is done, the *Activity Monitor* specifies if the analysis is successful or whether there are warnings or errors in the respective analysis steps. You have instant access to the details from analysis runs. By pushing RMB on an activity you can open listing files (or others) to investigate the details.

The files will be opened in your default editor for file types \*.LIS, \*.MNT and \*.INP.



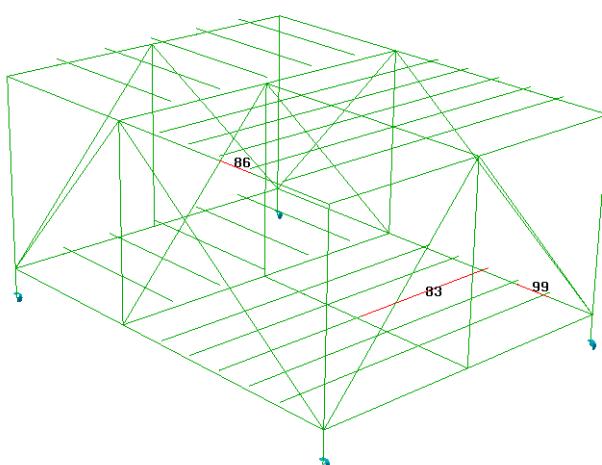
Below are examples of a Sestra listing file viewed in MS Notepad (load sums and difference between load sums and reaction forces).

```
*** SUM OF LOADS AND MOMENTS FOR SUPERELEMENT TYPE 1
X-LOAD = SUM OF GIVEN LOADS IN GLOBAL X-DIRECTION
Y-LOAD = SUM OF GIVEN LOADS IN GLOBAL Y-DIRECTION
Z-LOAD = SUM OF GIVEN LOADS IN GLOBAL Z-DIRECTION
X-MOM = SUM OF LOCAL MOMENTS ABOUT GLOBAL X-AXIS
Y-MOM = SUM OF LOCAL MOMENTS ABOUT GLOBAL Y-AXIS
Z-MOM = SUM OF LOCAL MOMENTS ABOUT GLOBAL Z-AXIS
X-RMOM = SUM OF MOMENTS ABOUT GLOBAL X-AXIS FROM GIVEN LO
Y-RMOM = SUM OF MOMENTS ABOUT GLOBAL Y-AXIS FROM GIVEN LO
Z-RMOM = SUM OF MOMENTS ABOUT GLOBAL Z-AXIS FROM GIVEN LO

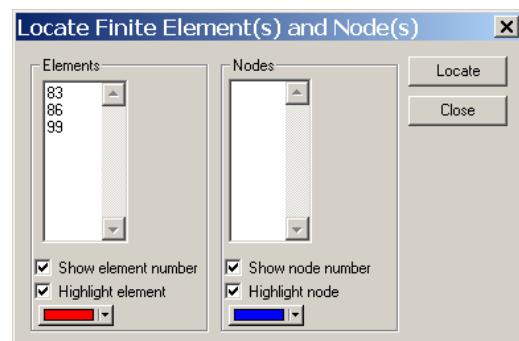
LOADCASE   X-LOAD    Y-LOAD    Z-LOAD    X-MOM
  1  5.6843E-14  0.0000E+00 -6.5364E+03  0.0000E+00
  2  0.0000E+00  0.0000E+00 -5.8840E+04  2.2524E+04
  3 -8.0000E+03  0.0000E+00 -6.0000E+04  2.1199E+04
  4  0.0000E+00  0.0000E+00 -1.0560E+04  0.0000E+00

RETRACKING MODULE - GL
DIFFERENCES BETWEEN SUMMED LOADS AND REACTION FORCES
*****LARGER THAN 0.00E+00 FOR TRANSLATIONAL COMPONENTS AND LARG
LOADCASE (INDEX)   X          Y          Z
  1  3.4106E-13  2.8422E-14 -8.1855E-12
  2  9.0949E-13  7.7307E-12 -7.2760E-11
  3  2.6375E-11  8.1855E-12 -5.0932E-11
  4  6.8212E-13 -7.3896E-13  2.3647E-11
TOTAL TIME CONSUMED IN SESTRA
CPU TIME:
```

If the analysis for some reasons fails, Sestra will report that e.g. element number 99 has failed. To find the exact location of this element you may use the feature **Tools/Analysis/Locate FE**. The example below shows how to find finite element numbers 83, 86, and 99 (FE nodes not shown).



**Tools/Analysis/Locate FE** gives this input dialog

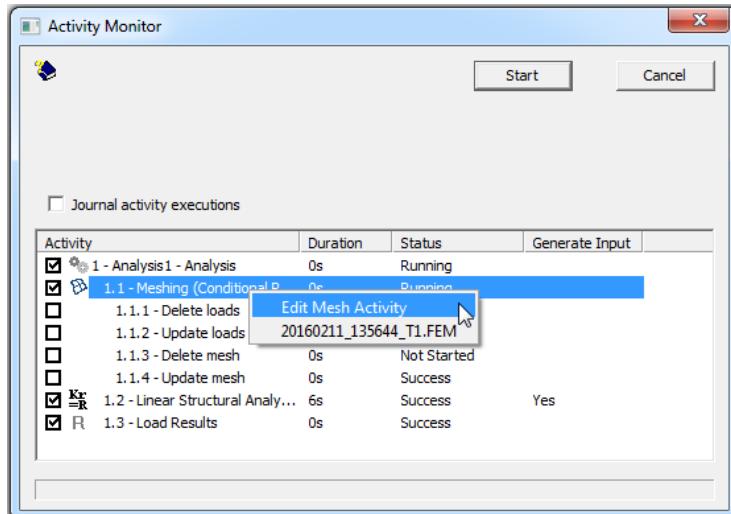


There are several ways to improve the mesh if the analysis fails, for example by use of split element if Jacobi matrix test fails or maximum angle is exceeded, inserting feature edges or by increasing the mesh density. See Chapter 3.12.2 for further details.

### 3.15.1 The Meshing option of the Activity Monitor

The Meshing activity of the Activity Monitor is step 1.1 of an Analysis and has three separate options – *Always Regenerate Mesh*, *Conditional Regenerate Mesh* and *Never Regenerate Mesh*.

The Meshing activity option defaults to *Always Regenerate Mesh*, except when a FEM file is imported. In the latter case the option is set to *Never Regenerate Mesh*. The user may change the option if necessary.



#### Always Regenerate Mesh

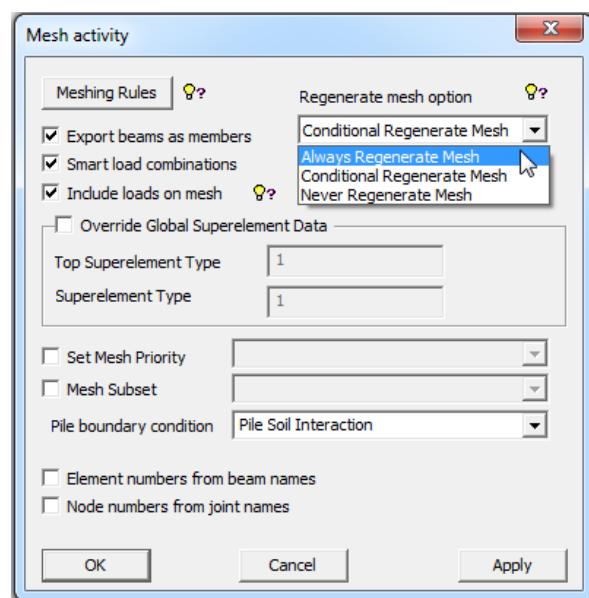
When the Meshing activity option is set to *Always Regenerate Mesh* – then a complete mesh generation is performed and the user may not turn off any of the individual steps, i.e. the steps 1.1.1, 1.1.2, 1.1.3 and 1.1.4 are all executed.

#### Conditional Regenerate Mesh

The user can give Genie mandate to decide the necessary mesh generation steps in a particular analysis by setting the Meshing activity option to *Conditional Regenerate Mesh*. The user may alter the settings, 1.1.1 to 1.1.4 by individual judgement. However, it is necessary to run step 1.1.4 whenever step 1.1.2 is executed as part of the mesh generation since the latter is needed in order to update the analysis model loads. Likewise, if step 1.1.1 and step 1.1.3 are run, then steps 1.1.2 and 1.1.4 should be run to maintain a consistent analysis model.

#### Never Regenerate Mesh

The *Never Regenerate Mesh* Meshing activity option is the default when a FEM-file has been imported for further analysis. In this case no mesh generation steps are run.



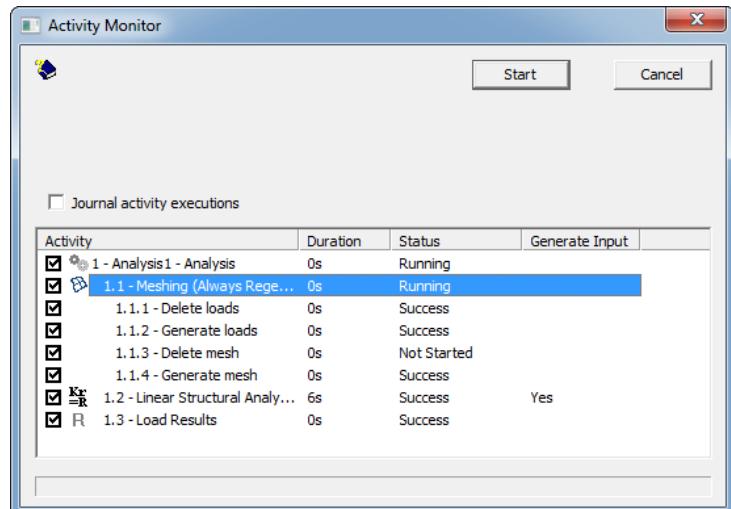
### 3.15.2 About the different meshing steps

#### 1.1 – Meshing

The meshing activity is split into four sub activities. The first time you run the analysis it is necessary to perform all steps.

When you re-run the analysis later it might not be necessary to perform all steps every time. GeniE will automatically suggest which parts of the meshing procedure that needs to be performed if the Meshing activity option is set to *Conditional Regenerate Mesh*, by letting the appropriate checkboxes being checked or unchecked.

Furthermore you can manually check or uncheck the different checkboxes.



##### 1.1.1 - Delete loads

If you have done extensive changes to your model and want to be sure that all loads are recalculated, check this checkbox.

##### 1.1.2 – Generate loads

If you have changed or added loads, “Generate loads” needs to be started again.

##### 1.1.3 – Delete Mesh

If you have done larger changes to your model, like adding or removing structure, this checkbox will be checked by default.

##### 1.1.4 – Generate mesh

If you have changed properties, like changing sections or materials, GeniE will just do a property update if you check this checkbox.

If “Delete Mesh” is also checked, a new mesh will be generated.

### 3.15.3 Modify the analysis parameters

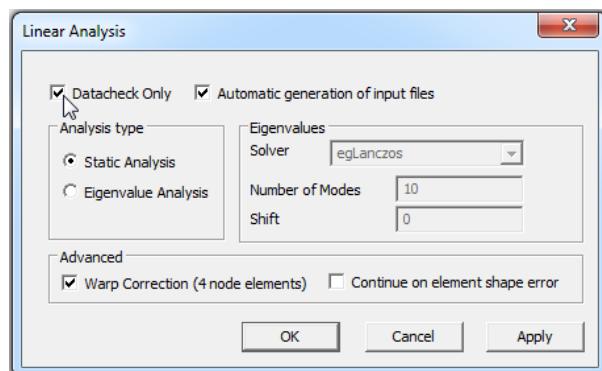
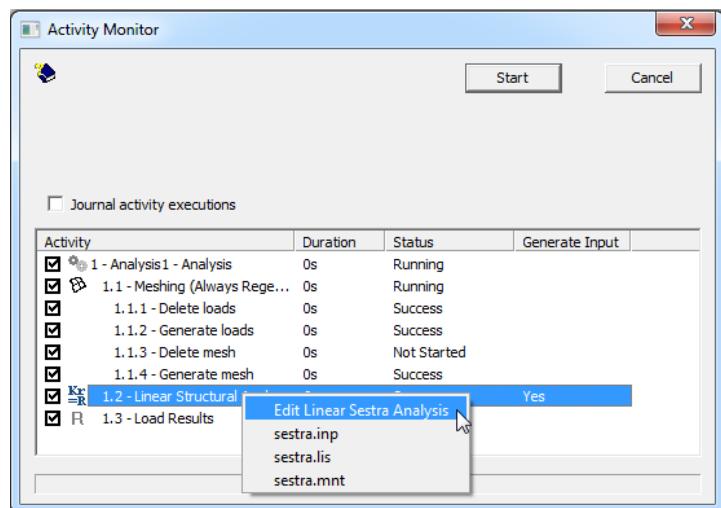
You may also change the analysis parameters to e.g. run a datacheck only or change the Sestra input file manually. To do this you need to open up the *Activity Monitor* and access editing features from the activity *Linear Structural Analysis*.

#### 3.15.3.1 Datacheck only

This example shows how to run a datacheck. Select the *Linear Structural Analysis*, push **RMB** and select *Edit Activity*.

Then check the *Datacheck Only* check-box and push OK.

When the datacheck is done, no results are calculated

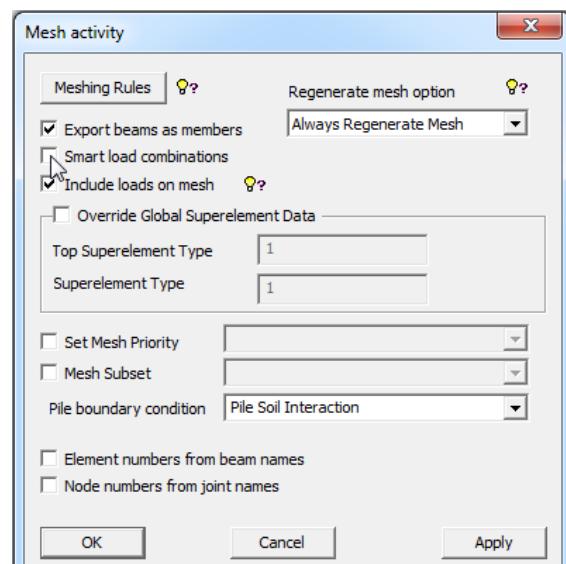


#### 3.15.3.2 De-activate smart load combinations

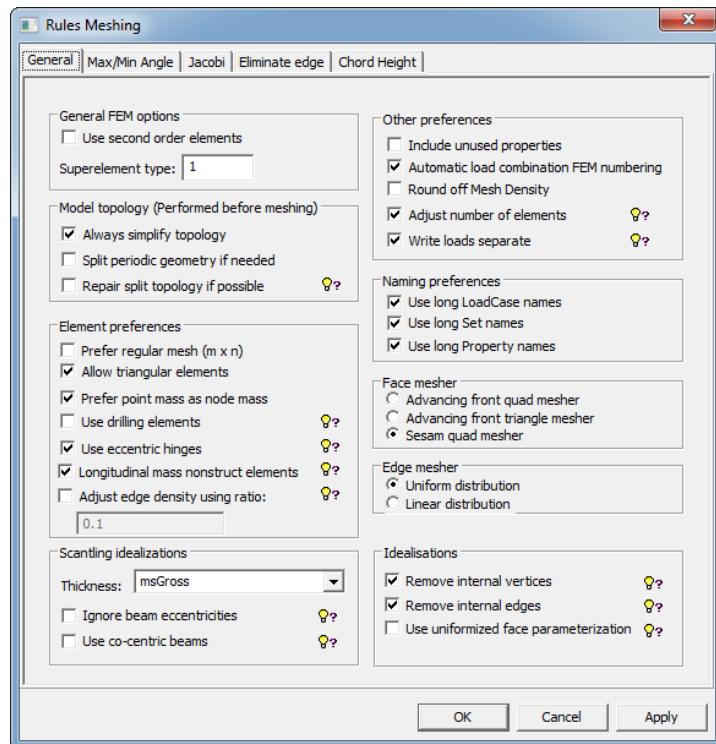
Select the *Meshing*, push **RMB** and select *Edit activity*.

Remember that smart load combinations is the default setting.

De-activate the *Smart load combinations*. When running the analysis the load computations are computed by Sestra Sestra, see Chapter 3.13.2 for more details.

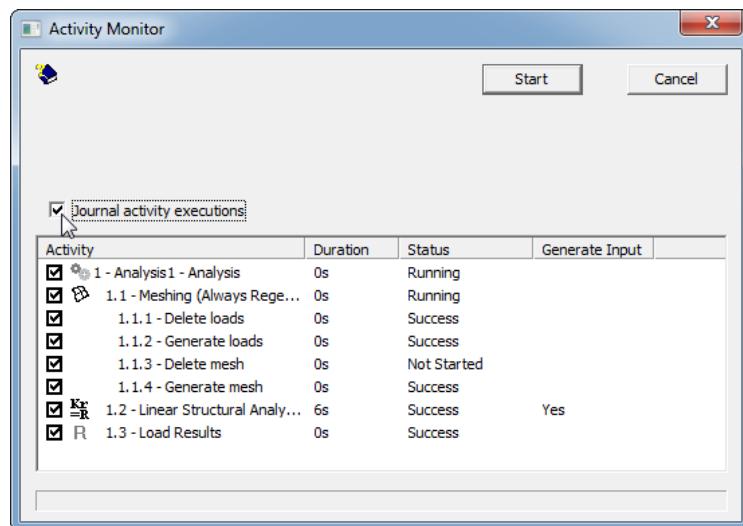


### 3.15.3.3 Accessing the mesh rules from Activity Monitor



By pushing the Meshing Rules (see picture above) you have direct access to the meshing rules (see picture to left). Any changes here are persistent, i.e. the same settings apply if you choose to access the mesh settings from **Edit/Rules/Meshing** (see also Chapter 3.2.6).

### 3.15.3.4 Journal analysis runs

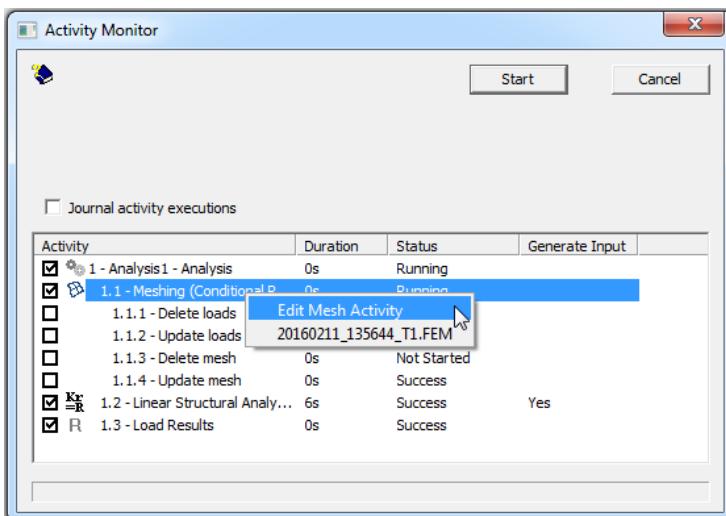


Activate the Journal activity executions. This operation will add the outlined details on the journal file in addition to those generated when defining an activity:

```
Analysis1 = Analysis();
Analysis1.add(MeshActivity());
Analysis1.add(LinearAnalysis());
Analysis1.add(LoadResultsActivity());
Analysis1.execute();
```

### 3.15.4 Smart load combinations

When this feature is activated this option the load combinations are computed when needed inside GeniE when looking at results.



It is of great importance to notice the following when using Smart load combinations:

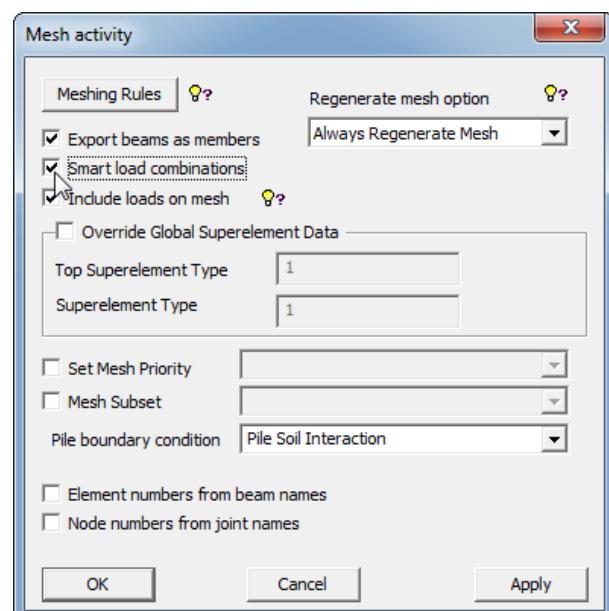
- When combining load cases where two or more load cases contain Rotation Field the Smart load combination option **must** be used. This applies when accelerations are transferred on the interface file.
- The FEM loadcase number of the first load combination must be the largest loadcase number used for the basic load cases +1, and the basic load case having the largest FEM load case number must not be an empty or zero loadcase.
- If the largest FEM loadcase contains equipment only and this is converted to mass then this will become a zero loadcase causing the same problem.

Note that Framework, Platework and Xtract, are also capable of handling load combination defined as smart load combinations.

I.e. the computation of results for the combined loadcases takes place in Framework, Platework or Xtract based on results from basic loadcases computed in Sestra.

When deactivating the smart load combinations GeniE calculates the complete sets of the combined/scaled loads and writes this load information onto the \*.FEM file, and Sestra will calculate results for all basic load cases and load combinations.

Hence, use of smart load combinations reduces the number of result cases to be solved and then reduce the time spent by Sestra.

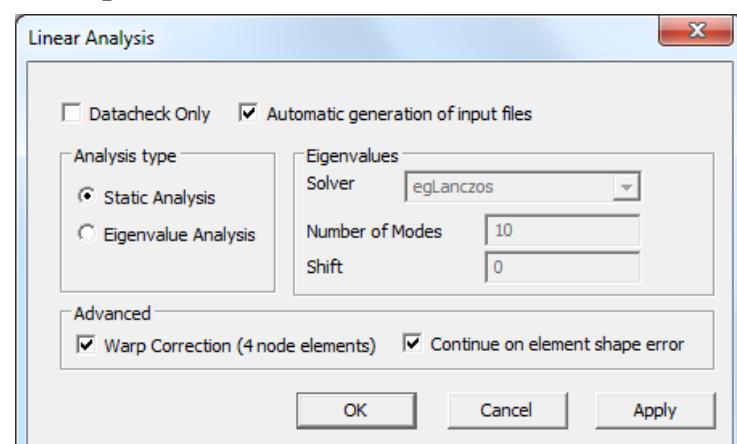


### 3.15.5 Continue on element shape error

When this checkbox is checked, Sestra will continue running even if elements with a bad shape are encountered.

This might in some cases allow you to run the analysis with a poor quality mesh.

Be aware that this might also cause Sestra to stop, as the checking of element shape is turned off.

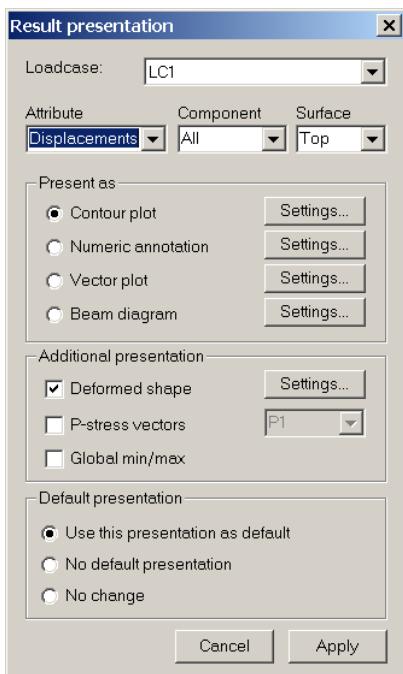


## 3.16 Results

GeniE can present displacements, general and principal stresses on plates, and beam forces. All results presentation is on a concept level, i.e. the results are presented per object (beam or plate member(s)). For all other result attributes, the program Xtract should be used by accessing the result file (\*.SIN) created by GeniE.

### 3.16.1 View preferences

Similarly to how you view your model, you can specify how you want to view the results in GeniE. To change these settings, and also to specify which result attribute to present, the command **Tools/Analysis/Presentation** is used.



From this dialog you switch between result attributes, loadcase, and how to present. The presentation is according to the selected structure (typically individual beams or plates, set names), default is the complete model.

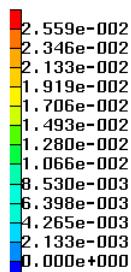
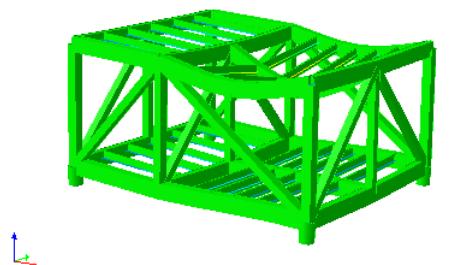
The first time you present results you should set up what is your preferred view option as Default. This view will apply to all new models you create. The settings at left specifies default settings to show a contour plot including deformed shape of displacements (x,y,z combined).

In the following some examples are given on how to present the various result attributes.

Common for them all is that a structural analysis has been run without errors (to create results) and that you have switched (in the combo box for predefined views) to a view that shows results (e.g. *Results – All* or *Results – with Mesh*).

### 3.16.2 Displacements

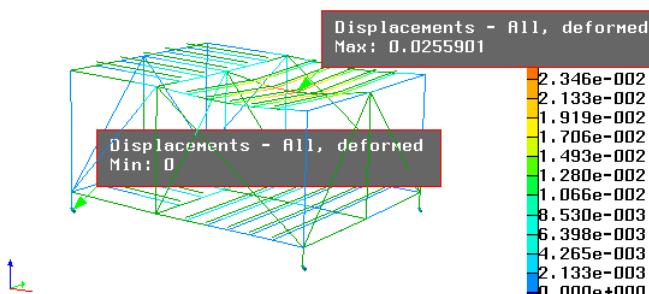
```
18 Sep 2003 12:33
STR
LC4
Displacements - All, deformed
Min: 0
Max: 0.0255901
```



A contour plot of the displacements for loadcase LC4 is presented. The display also contains a deformed shape.

The results are shown on an outlined view of the structure. For many of the results processing tasks, it may be beneficial to switch to a wireframe view.

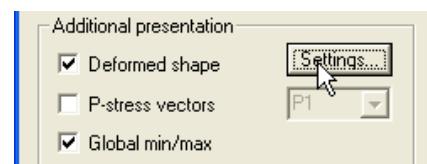
18 Sep 2003 12:37  
STR  
LC4  
Displacements - All, deformed  
Min: 0  
Max: 0.0255901



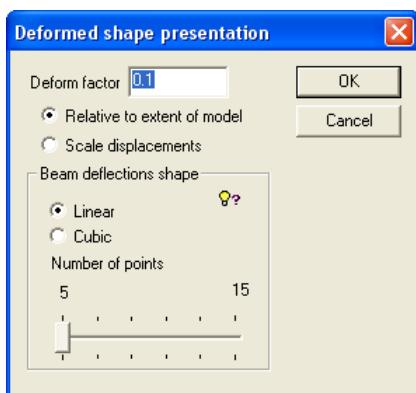
In this case, view has been switched to wireframe.

The displacements have been scaled with a factor of 50 and maximum and minimum values are presented.

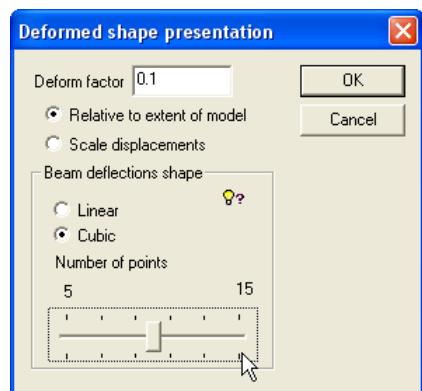
For more details, see the Result presentation sheet under Additional Presentation.



By clicking the “Settings” button under “Additional presentation”, you get extended control on how to present the deformations. The beam to the right is used as an example below. Mesh settings have been applied so the beam consists of two finite elements.



We have switched to the “Mesh – All” view. Under “Beam deflection shape” Linear is selected. The mesh consists of two elements along the beam. The member elements remain straight.

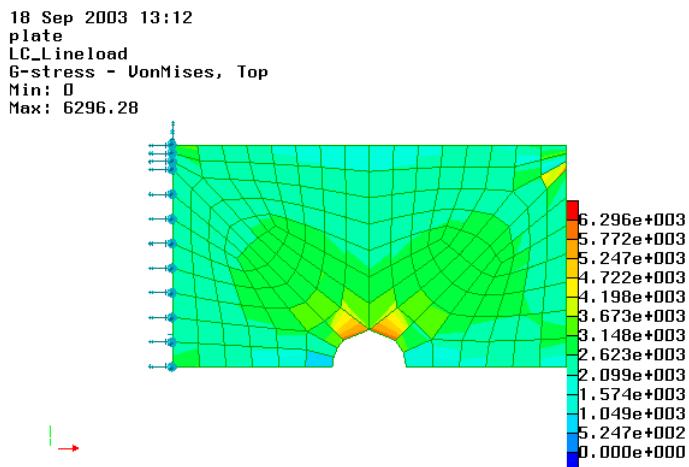


Under “Beam deflection shape” Cubic is selected. Furthermore we have selected 10 points on the slider. You are able to select from 5 to 15 points. As can clearly be seen, we now get to see a deformation which is much closer to real life. No changes have been done to the mesh settings.



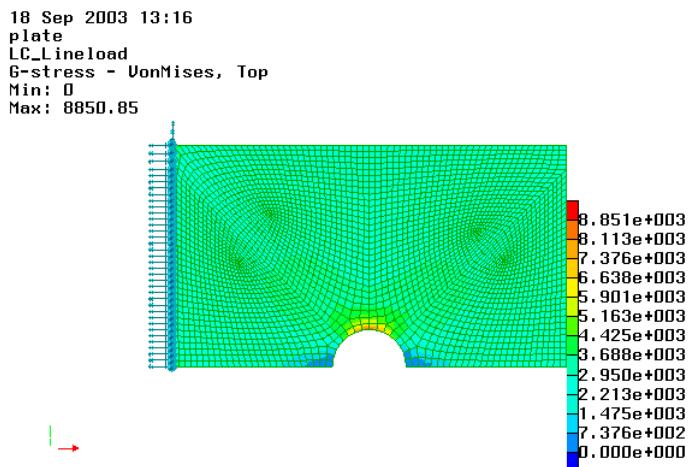
### 3.16.3 Element General Stresses

The Element General Stresses (G-stress) are stresses that are extrapolated from the result points to the element nodes and pertaining to the individual finite elements (plate elements only). In other words there is no averaging of stress components between adjoining elements. As such, large differences between finite element results is one indication of the quality of the finite element model. Ideally, the stress results between finite elements should be a smooth pattern.



In this case the VonMises results are shown for a plate with a cut out (fixed at the left hand side, uniform line load along the right hand side).

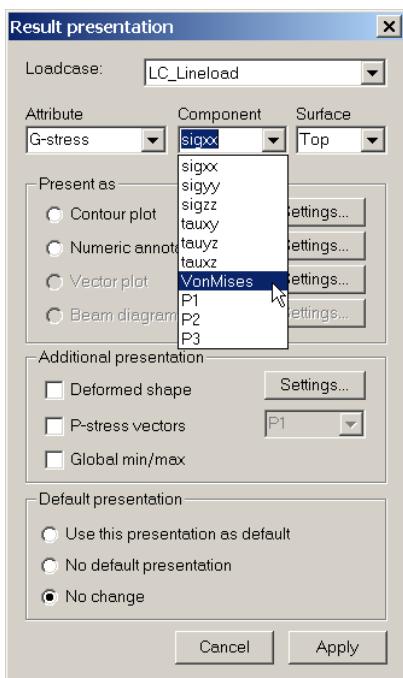
This picture shows that the finite element created is not adequate for representing the real stress pattern due to the forces applied.



There are several ways of improving the mesh quality, in this case the default mesh density has been increased by a factor of 8.

As can be seen the stress pattern is fairly smooth and the finite element model is much better to compute the peak values.

The deviation in peak values between the two models is in this case close to 30%.



GeniE can present several attributes for general stresses  
**SIGXX** – stress in the direction of the local x-axis  
**SIGYY** – stress in the direction of the local y-axis  
**SIGZZ** – stress in the direction of the local z-axis (irrelevant for shell elements)

**TAUXY** – shear stress in the direction of local x/y axes  
**TAUYZ** – shear stress in the direction of local y/z axes (irrelevant for 1<sup>st</sup> order shell elements)  
**TAUXZ** – shear stress in the direction of local x/z axes (irrelevant for 1<sup>st</sup> order shell elements)  
**VONMISES** – von Mises stress

You may also choose which surface you want to present the stresses in, top, middle (neutral plane), or bottom.

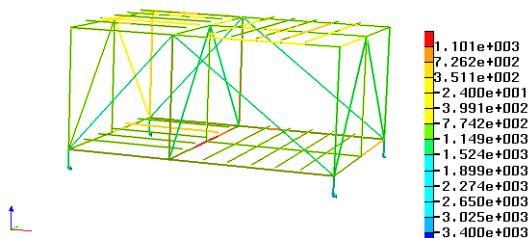
For all above, axis or surfaces refer to a plate local co-ordinate system.

### 3.16.4 Beam forces

Beam forces or moments are normally presented as contour plots, as pure numerical values, or as diagrams. In the following some examples are given on how to do this as well as to present results for selected parts of the model only.

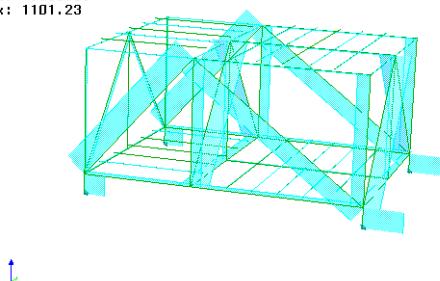
```
18 Sep 2003 14:28
beam
LC4
Beam Forces - Nxx
Min: -3399.69
Max: 1101.23
```

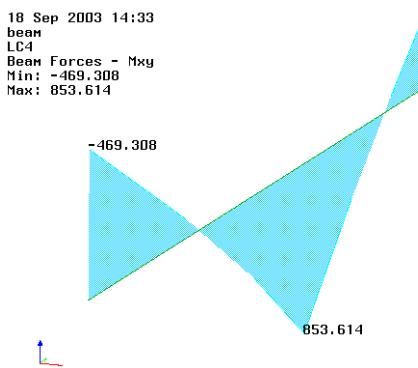
Contour plot of axial beam forces are shown for the whole model.



```
18 Sep 2003 14:41
beam
LC4
Beam Forces - Nxx
Min: -3399.69
Max: 1101.23
```

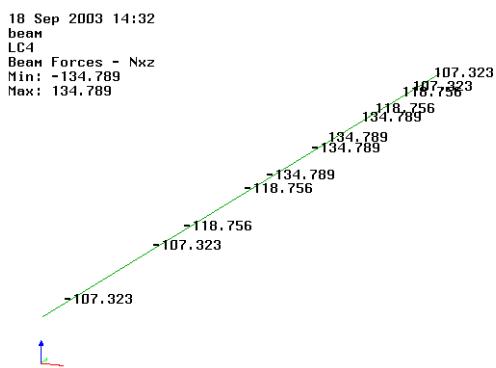
Diagram of axial forces are shown for the whole model



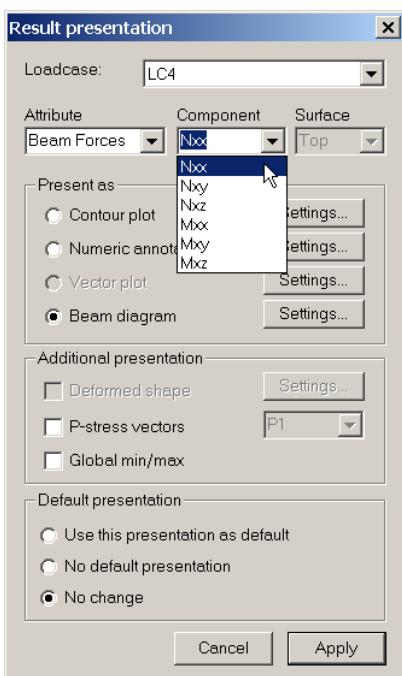


Bending moment diagram presented for one beam only.

Maximum and minimum values also added to plot.



Bending moment diagram presented for same beam as above, but as numericals.



The attributes GeniE can present for beam forces are

**NXX** - Axial force

**NXY** - shear force in the direction of the local y-axis

**NXZ** – shear force in the direction of the local z-axis

**MXX** – torsional moment

**MYX** - bending moment about the local y-axis

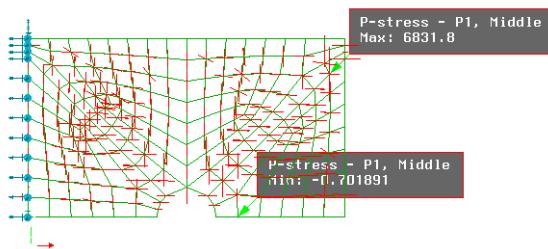
**MXZ** – bending moment about the local z-axis

For all above, axis refer to a beam local co-ordinate system

### 3.16.5 Principal stresses

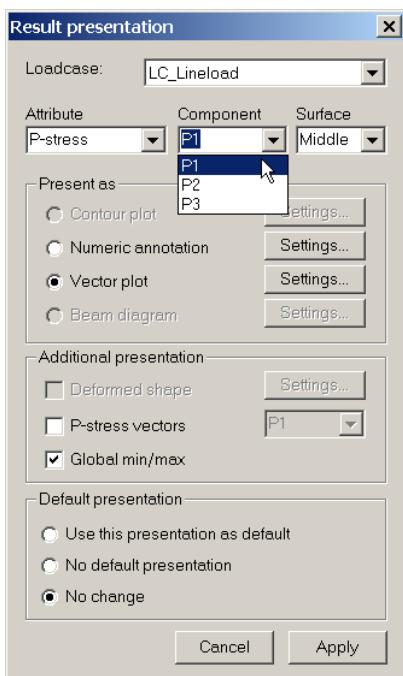
Principal stresses are shown as vector plots, and it is also possible to include principal stresses together with presentation of general stresses.

```
18 Sep 2003 20:38
plate
LC_Lineload
P-stress - P1, Middle
Min: -0.701891
Max: 6831.8
```



The principal stress P1 shown (middle surface) of a plate with cut out.

Maximum and minimum values added to the presentation.  
Colours of the different principal stresses may be changed.



The following principal stresses may be presented:

P1 – The highest principal stress for shell elements

P2 – The second highest (and lowest) principal stress for shell elements

P3 – Irrelevant for shell elements

### 3.16.6 Surface load

Contour plotting of *pressure* (separate load on each side of plate). Note that *traction1* and *traction2* are not in use.

Note the following conventions:

- *Middle* surface: will sum positive and negative pressures.
- *Top* surface: will only sum up positive pressures, that is positive pressure on Front side of plate concept.
- *Bottom*: will only sum up negative pressures, that is positive pressure on Back side of plate concept.

### **3.16.7 Other results presentation**

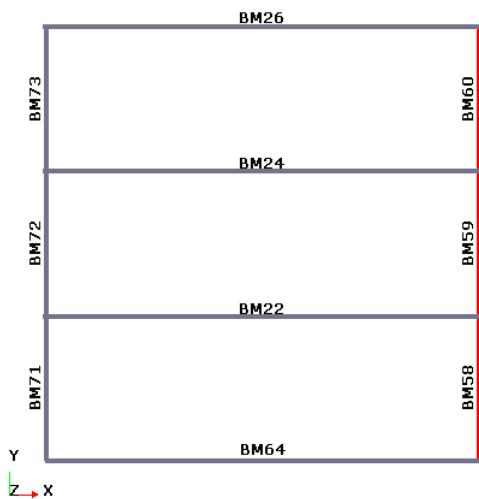
The model and results may be visualised in Xtract which is a general finite element postprocessor. Xtract has many features for presenting other result attributes and has a user interface that is similar to GeniE. When importing a model from GeniE into Xtract, please note that concept information is not read since Xtract is based on finite elements (nodes, elements, and loadcases). Named sets are read by Xtract, this means that if you want to investigate parts of a GeniE model in Xtract these should be part of named sets in GeniE.

### 3.17 Run Code Checking using Framework

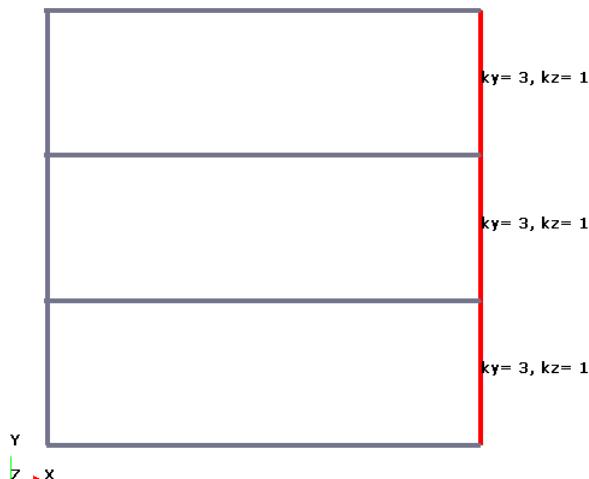
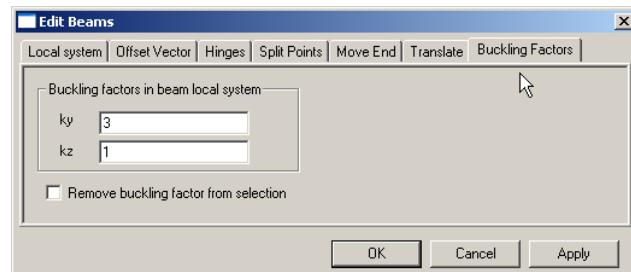
Code checking in GeniE is now normally done inside GeniE. However it is still possible to do code checking in the program Framework. The program may be started from GeniE. When model data is imported in Framework, all concept information is read in. Examples of concept data may be names of objects (beams, sections, materials, load case names, joint names, buckling length factors) and relationship between beam members and the finite elements.

This means that when referring to an object the name in GeniE is used, or you may use the finite element numbering system if this is the preferred solution. Note that when creating a model in GeniE the number of characters in a name should not exceed eight (8) to avoid truncation of names.

Prior to code checking you may want to add buckling length factors to the beams. The buckling length factors are being used by Framework to calculate the beam buckling length when performing the code checks. In the example below beams BM58, BM59, BM60 are given buckling factors to simulate buckling in local y-direction covering the span from beam BM64 to beam BM26. Similarly, the buckling length in local z-direction shall be equal to the beam length.



Select the beams, force the context sensitive menu and choose Edit Beam and Tab Buckling Factors



The buckling factors are defined and applied to the selected beams.

Note that if you split a beam or join beams, the assigned buckling factors will be set to default ( $k_y=-1$  and  $K_z=-1$ ) which means default buckling factors will be used by Framework).

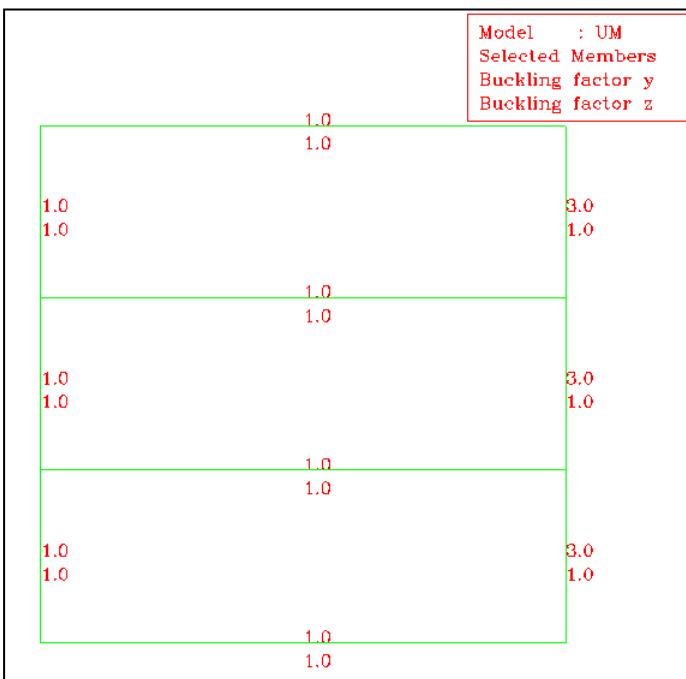
The default values are not shown when you label the buckling factors.

You may also verify the applied buckling factors from the browser (by adding the necessary fields to the structure browser), the saved report or from Framework itself. All options are showed below.

Name	Description	Section	Buckling factor, kx	Buckling factor, ky
BM57	Straight Beam	BOX8		
BM58	Straight Beam	BOX8	1	3
BM59	Straight Beam	BOX8	1	3
BM6	Straight Beam	BOX1		
BM60	Straight Beam	BOX8	1	3
BM61	Straight Beam	BOX8		
BM62	Straight Beam	BOX8		
BM63	Straight Beam	BOX8		
BM64	Straight Beam	BOX7		

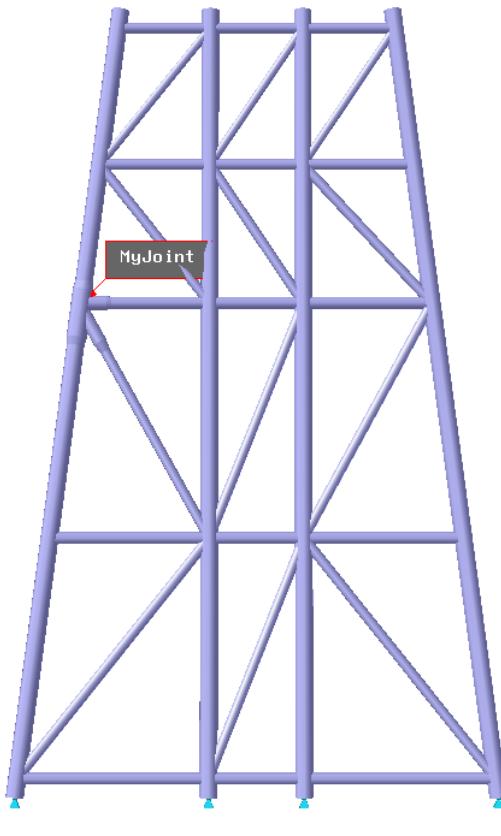
The fields kx, and ky are added to the browser view.

1	Name	X-End1 [m]	Y-End1 [m]	Z-End1 [m]	X-End2 [m]	Y-End2 [m]	Z-End2 [m]	Section	Material	Mesh Beam	Flood Morisc	Hinge-End1	Hinge-End2	Ky	Kz
50	BM57	28.000	11.000	12.500	28.000	22.000	12.500	BOX8	St52					3	1
51	BM58	11.000	0.000	12.500	11.000	3.670	12.500	BOX8	St52					3	1
52	BM59	11.000	3.670	12.500	11.000	7.340	12.500	BOX8	St52					3	1
53	BM6	11.000	11.000	0.000	28.000	11.000	0.000	BOX1	St52					3	1
54	BM60	11.000	7.340	12.500	11.000	11.000	12.500	BOX8	St52					3	1
55	BM61	11.000	11.000	12.500	11.000	14.660	12.500	BOX8	St52					3	1
56	BM62	11.000	14.660	12.500	11.000	18.330	12.500	BOX8	St52					3	1



The buckling factors as reported by Framework.  
See below on how to start the code checking.

The code checking program is started from **Tools/Analysis/Frame Code Check**. This Chapter focus how to set up the code checking model and how to control which data are exported to Framework. For more details on how to do the actual code checking (yield, buckling, punching), fatigue (deterministic, stochastic), or earthquake analysis, reference is made to the Framework User Manual.

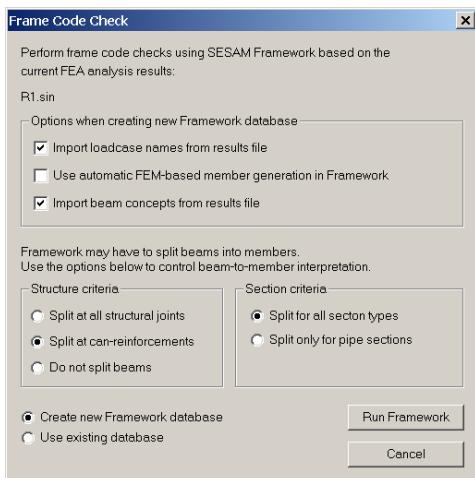
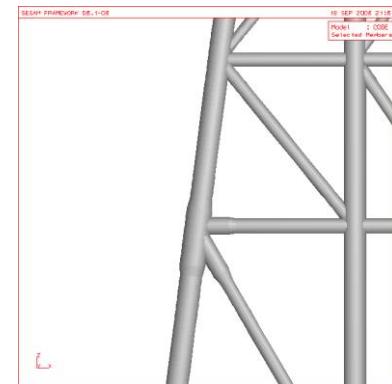


To show how a code checking model can be set up by GeniE, the jacket frame to the left is used.

This model consists of tubular members only, and there is one joint where cans, stubs, cones, and gaps are assigned. This structural joint is named Myjoint.

To visualise only the joint and incoming members, select the joint, RMB, *Select Connected Beams* and show these only.

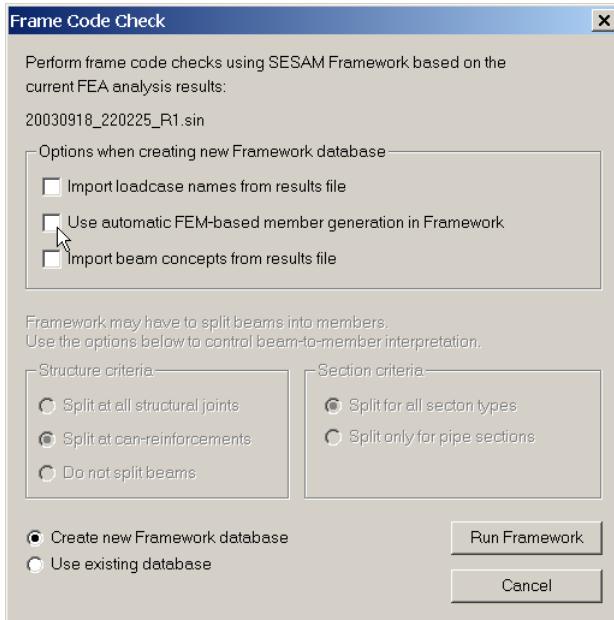
The picture at right shows the model imported to Framework



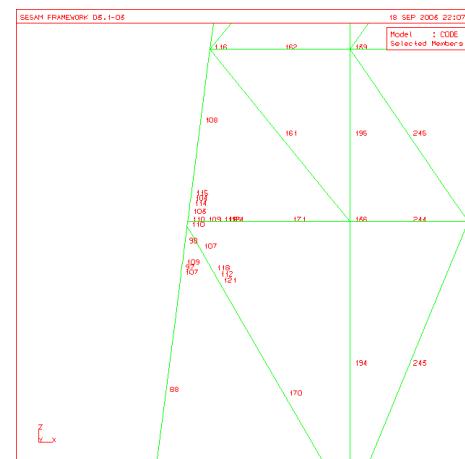
To set up the model for code checking, the dialog sheet at left is used (this is activated from **Tools/Analysis/Frame Code Check**)

In the following, the options on how to split up the structure to create wanted member lengths are discussed.

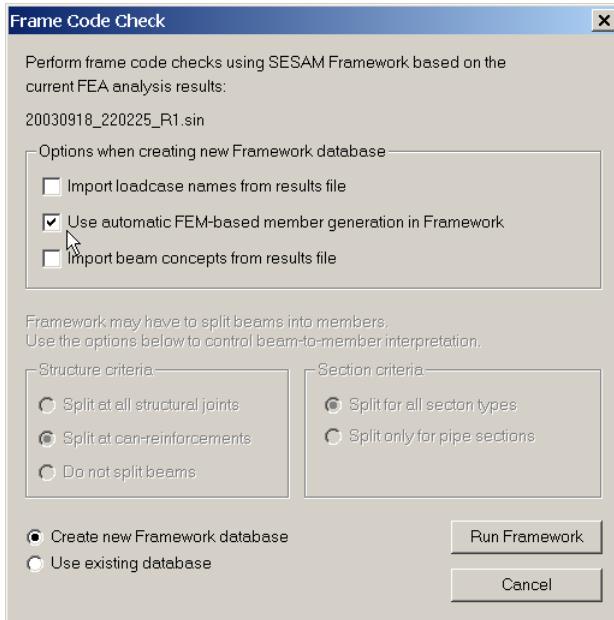
### 3.17.1.1 Import the model using finite element numbering



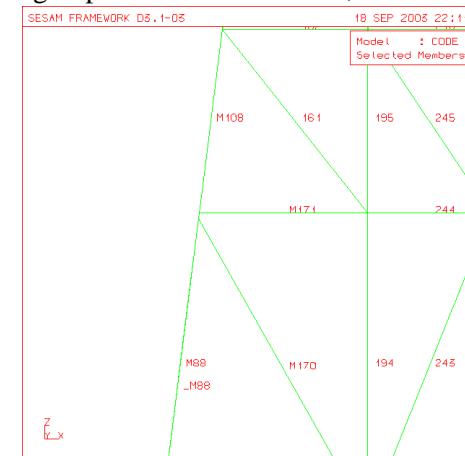
No concept information is exported to Framework. To do code checking all references must be made to finite element and node numbering. These numbers are identical in GeniE and Framework. Below is an example of the element numbers close to the structural joint Myjoint.



### 3.17.1.2 Import the model using FE numbering and generate members in Framework



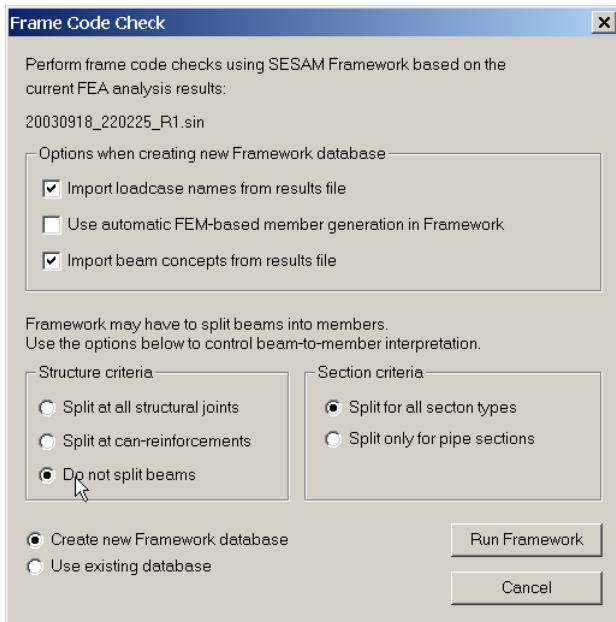
In Framework it is possible to automatically create members between two structural joints where each member may consist of many finite elements. By activating this feature, Framework will do this operation during import of the model data, see details close to Myjoint.



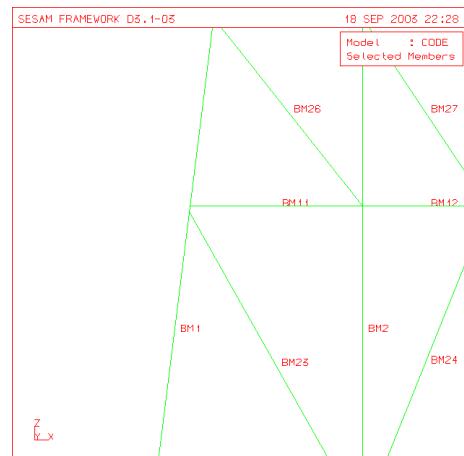
Both examples above imply that the user has knowledge about the finite element model, and as such not utilising the high level concept information already built into the model. By using the automatic member creation, the buckling length is always set equal to the member length unless manually specified or automatically computed by the program using an eigenvalue analysis approach (tubular members only).

Since members are modelled independently of finite element numbers in GeniE, the default buckling lengths are more inline with the real situation as compared to a finite element modelling approach.

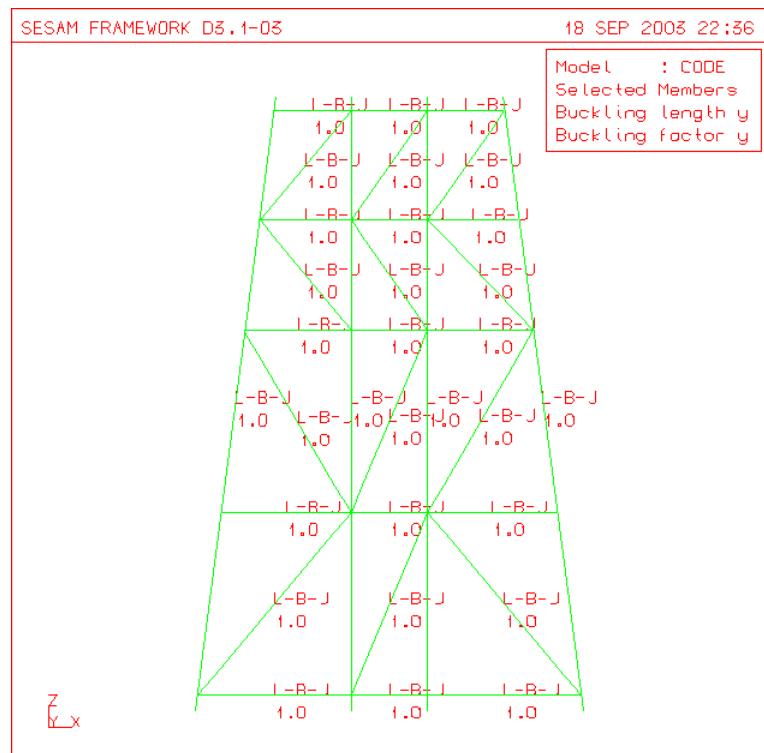
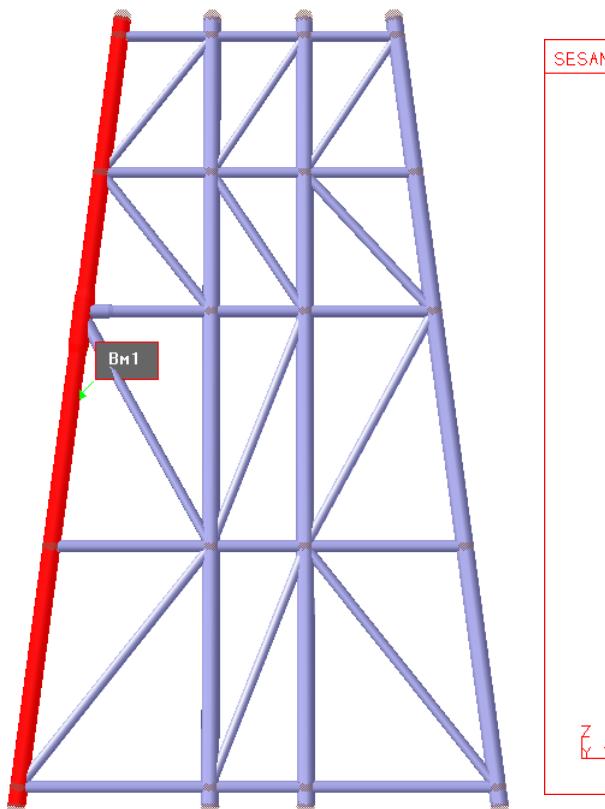
### 3.17.1.3 Import the model using concept names, no split of structure



The concept information is now read by Framework. The names of the beams close to Myjoint are shown below, and the names are identical to those in GeniE. This means that if the finite element numbering for some reason change, you can still use the same names in Framework since the beams (or concepts) are independent of mesh density.

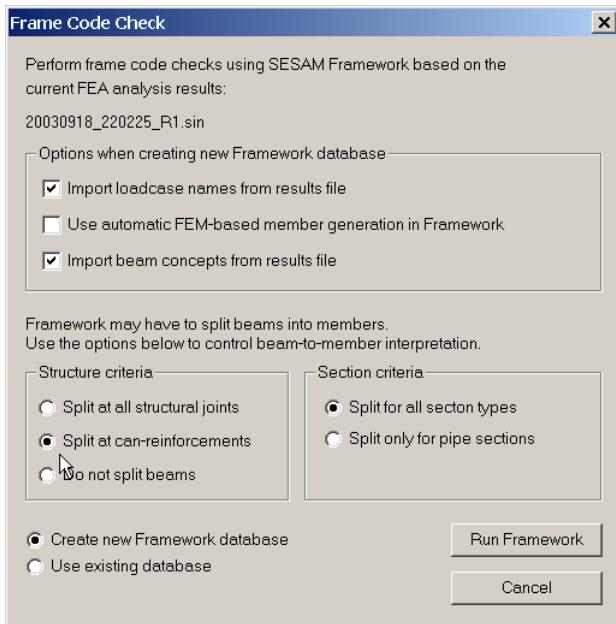


Since no split of structure is performed, the buckling length of Bm1 is identical to the member length. The pictures below show the extent of Bm1 and the buckling length of the same beam in Framework (buckling factor is 1.0, and buckling length is L-B-J (Length-Between-Joints)).



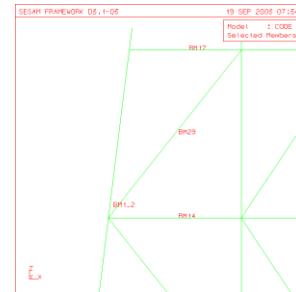
To override the buckling length factors, either specify manually or compute by using an eigenvalue approach (for tubular members only) in Framework.

### 3.17.1.4 Import the concept model, split structure at reinforced structural joints

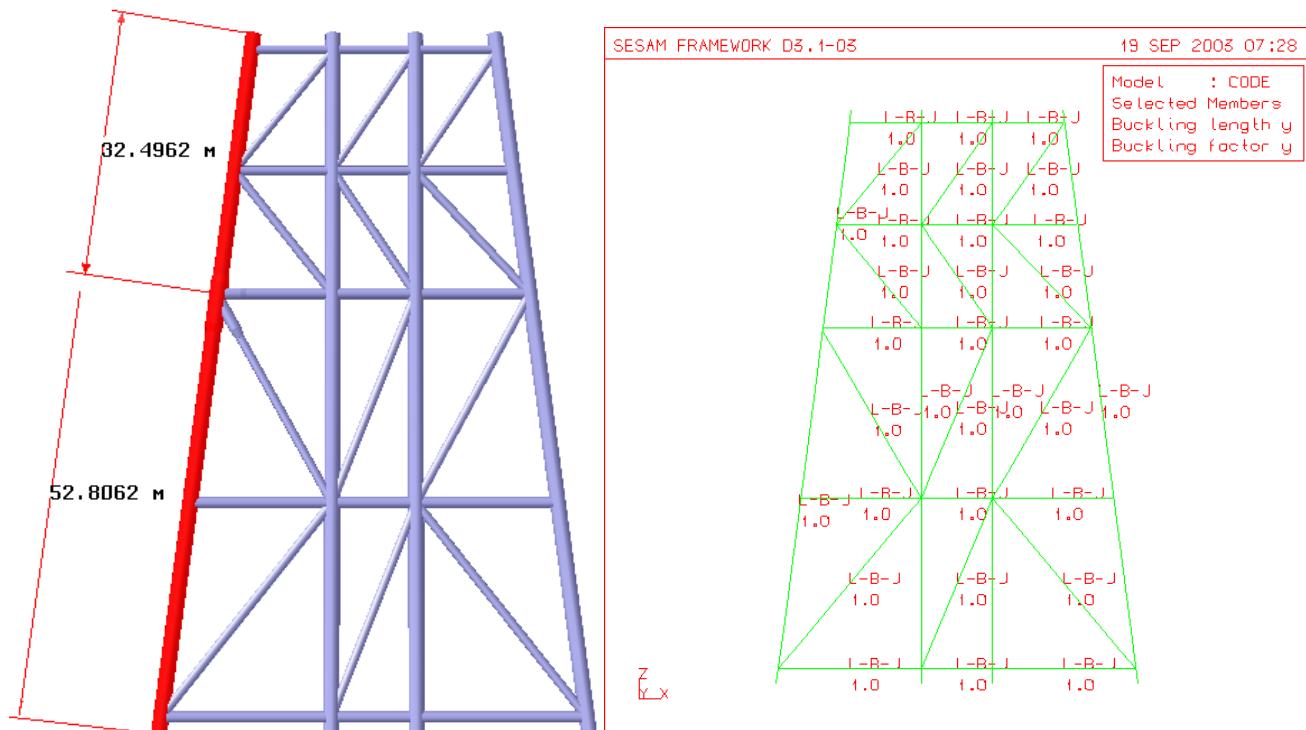


The split at can-reinforcements is the default option and will create new members in Framework. This means that you can model on a high level in GeniE and concentrate on the code checking model part of it afterwards.

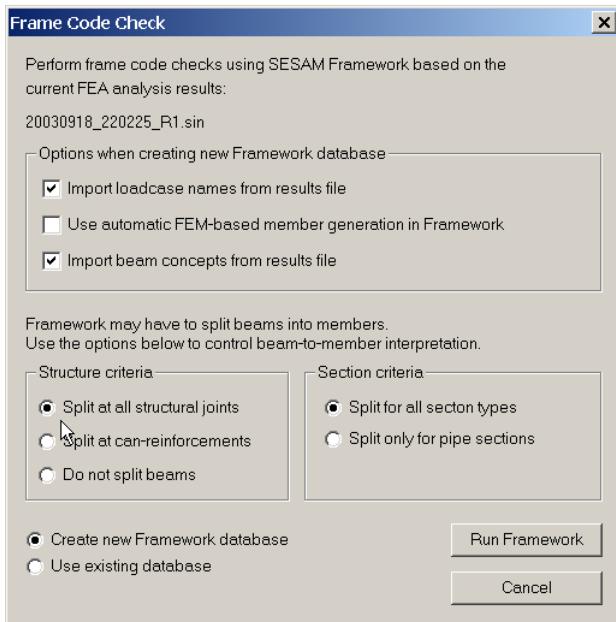
The new member names will always refer to the original beam name in GeniE, for example Bm1 split in two members in Framework receive the names Bm1\_1 and Bm1\_2. The upper part of the leg is shown below.



A split has been performed, and the default buckling length of the new Framework members Bm1\_1 and Bm1\_2 are equal to the length of the respective members. The pictures below show the extent of Bm1\_1 and Bm1\_2, in this case they are 52.8062 m and 32.4962 m.

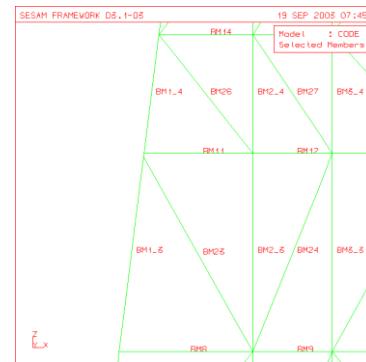


### 3.17.1.5 Import the concept model, split structure at all structural joints

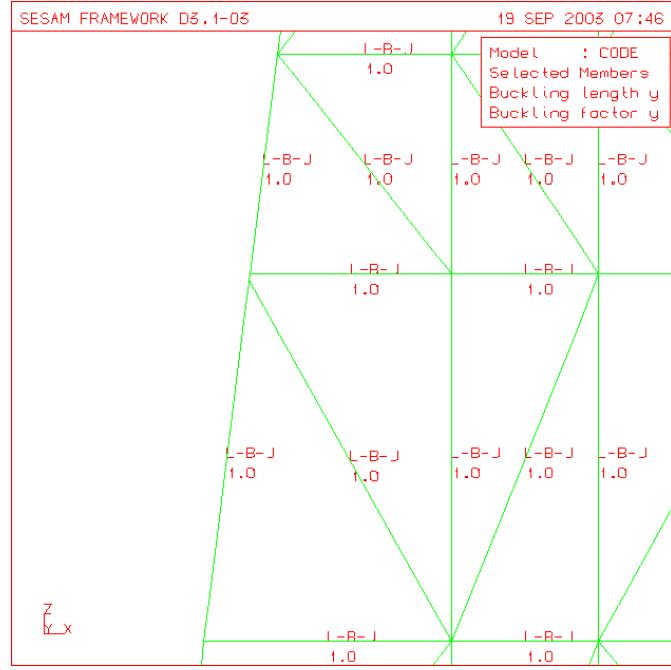
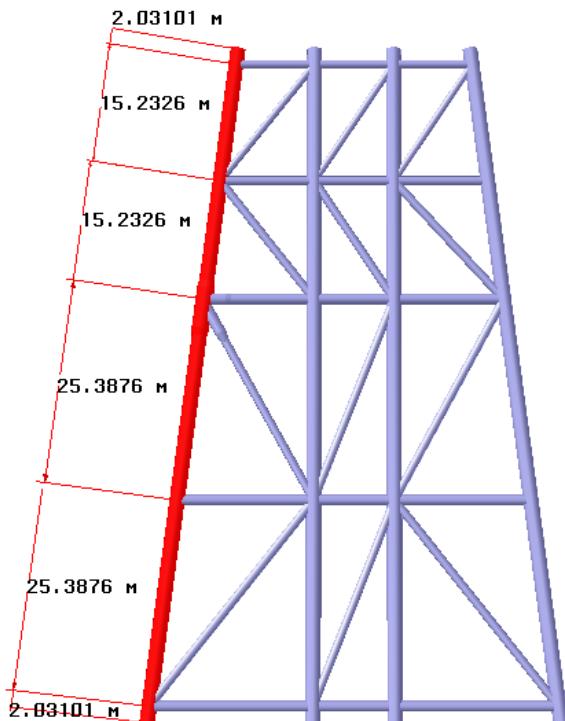


In this case the GeniE model is split at each structural joint. As such it is the same option as automatic FEM based member generation in Framework, the major difference is that concept information now is being used (beam names, sections, materials etc.).

Bm1 is now thus split in 6 members. The picture below shows the name details close to Myjoint,



The pictures below show the default buckling lengths of each member (picture focus around Myjoint).

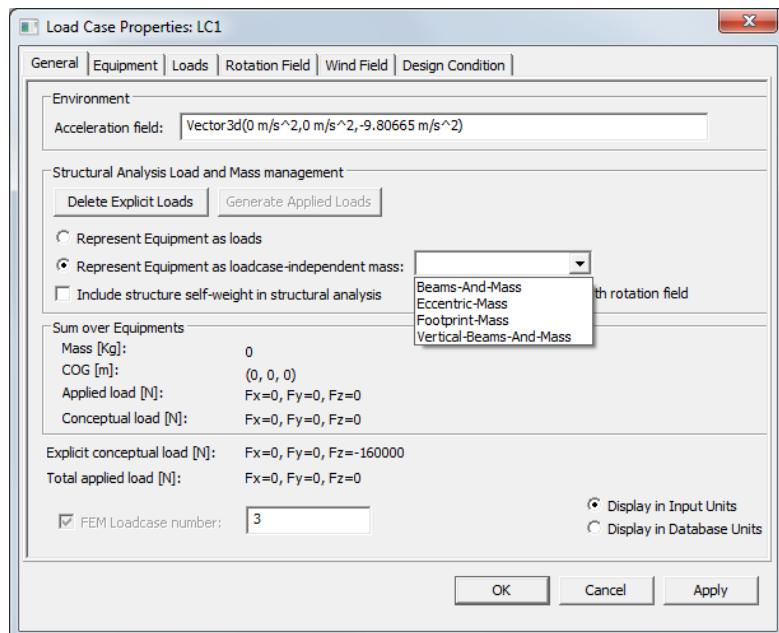


### 3.18 Create Mass Models

When creating a finite element model in GeniE (either from a mesh operation or when you run analysis) a mass model is automatically made. The mass model is built up from structural mass or any specific given point masses.

Of equal importance is the effect of equipments. When running a static structural analysis, the effect from equipments are treated as forces, while in a dynamic analysis (hydrodynamic or structural) these must be treated as masses to contribute to the mass model.

Note that each loadcase marked as Represent Equipment as loadcase-independent mass will add to the total mass model. Hence if same equipment is part of several loadcases you should set this switch in one loadcase only.



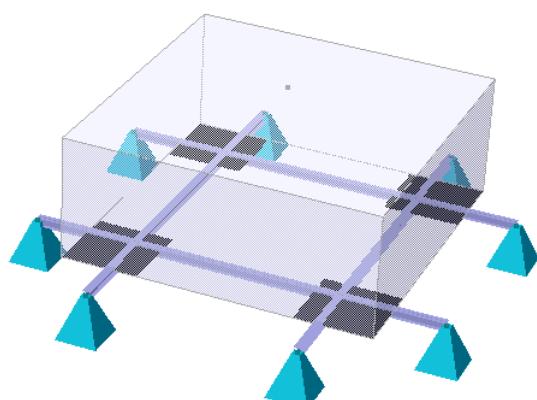
The same concept model may be used for both purposes (create forces or masses from equipments). You decide which option to use in the Analysis Property Sheet as show at left.

There are three options for creating mass models (from basic loadcases only):

- Eccentric mass – for hydrodynamic dynamic analysis only
- Beams and Mass – for hydrodynamic dynamic analysis followed by structural analysis, or dynamic structural analysis
- Footprint mass – eccentricities are not accounted for
- Vertical Beams and Mass –

Vertical beams are inserted to avoid horizontal tension and compression in the structure supporting the equipment. Vertical Beams and Mass is described in Vol 3 - Chapter 4.6.3.2.

The other options are described in the following.



To illustrate the differences between the two first alternatives, the following model is used for references.

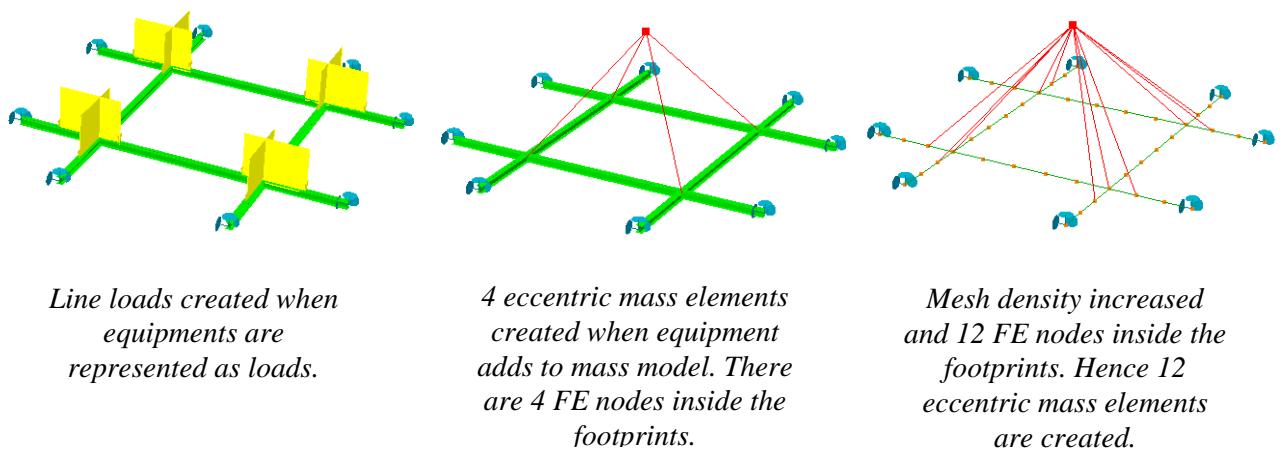
The model consists of four beams beams, one equipment with four footprints, and boundary conditions (free to rotate in all dof).

### 3.18.1 Mass model for hydrodynamics

A common scenario when making a mass model for hydrodynamics is when the GeniE model contributes to the overall mass model of a floater. Typically, the topside is modelled in GeniE, the hull is modelled in Patran-Pre and the complete model is assembled in Presel. The complete mass model is now (together with a panel model) analysed in Wadam to find e.g. rigid body motions and global sectional loads.

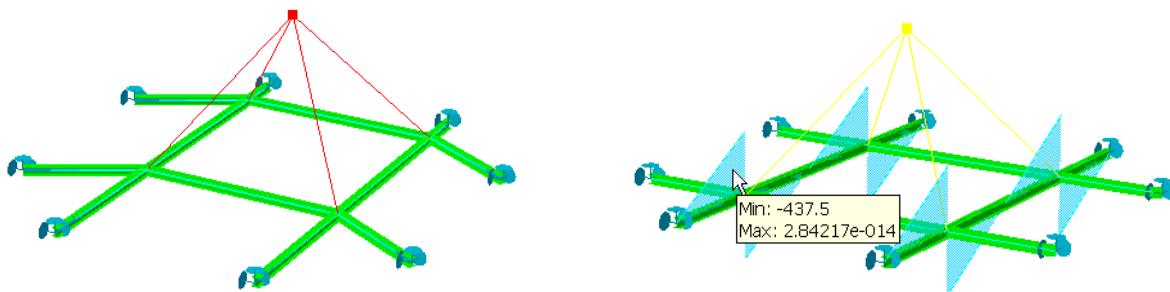
When sectional loads are to be computed by Wadam, a description of the mass *distribution* is required. The technique of modelling equipments rather than explicit loads will help significantly to establish such mass model.

In the example below a mass model using the eccentric mass option has been used. Finite element of type GMAS (one node mass element) are inserted. The eccentric mass elements have their mass centre always at the same position as the equipment local cog. Whenever there is an interface between finite element nodes and footprints, eccentric one node mass elements are created (sum of masses of mass elements = equipment mass).



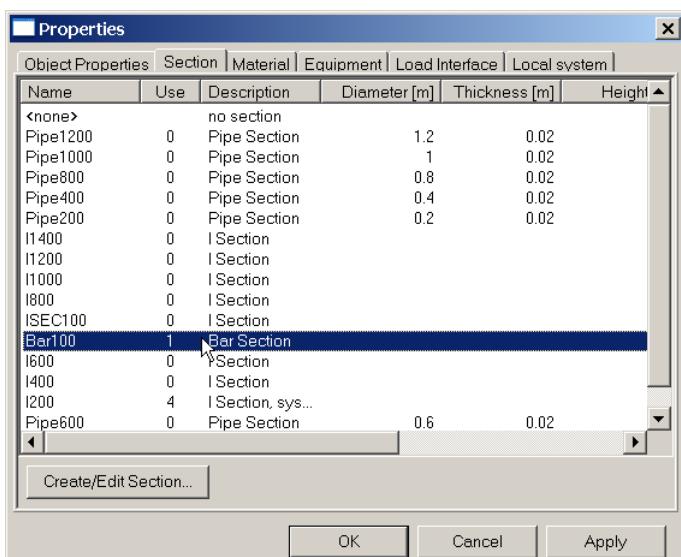
When applying a horizontal acceleration to a loadcase containing the equipment, correct displacements are computed, but the bending moment are not correct (the peaks are much higher) compared to a real case. The reason is that the connection between the one node eccentric mass elements and structure is fixed in all dof and hence moments are computed. In a realistic case, there are only vertical and lateral forces to be transferred from the equipment and no moment transfer (these forces will set up some moment effects, but not as large as shown below). This is the reason why this approach is applicable for pure hydrodynamic analysis only, if dynamic structural analysis shall be carried out the next alternative for mass representation should be used.

The pictures below show displacements and the undesired bending moments (to high peaks) due to a horizontal acceleration component.



### 3.18.2 Mass model for structural dynamics

When creating a mass model for structural dynamics it is important to avoid the undesired bending moments as discussed above. GeniE will do this automatically by inserting additional elements with hinges (i.e. no rotational connections) between the mass element and the structure. This technique is referred to as tripod, tent, or “chicken feet”. To be able to do so, it is required to enhance the equipment specification with sectional and material data. This information is added from the equipment property sheet.



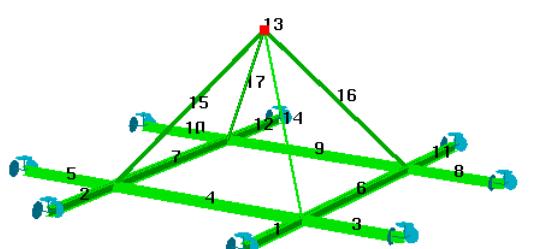
Section type Bar100 and a material type is assigned to the equipment from the equipment property sheet.

Normally, equivalent sections are used addressing no side effects when e.g. performing eigenvalue analysis.

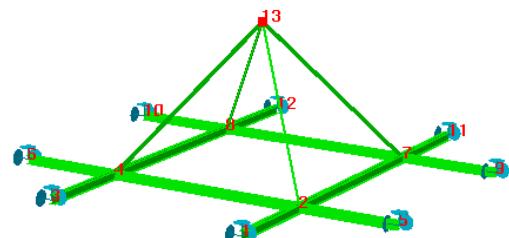
Also, it is common to use a material property with zero density (the equipment mass is represented from a point mass and not the structure associated with the equipment).

You may also verify the association between the equipment and the properties (section and material).

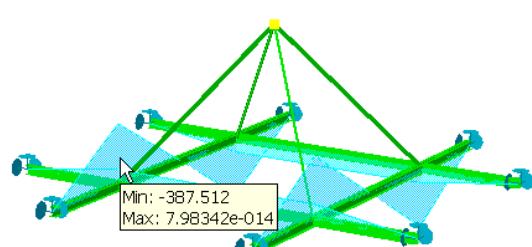
In the example below the feature Beams and Mass for representing equipments as loadcase independent mass has been used.



A finite element model has been created. The mass of the equipment is represented as one point mass (in this case finite element number 13) which is connected to the structure with beam finite elements 14, 15, 16, and 17.



Hinges are inserted where connected to the structure (free to rotate around local y and z axis) to avoid the undesired bending moments as in the case by representing equipments with one node eccentric mass elements.



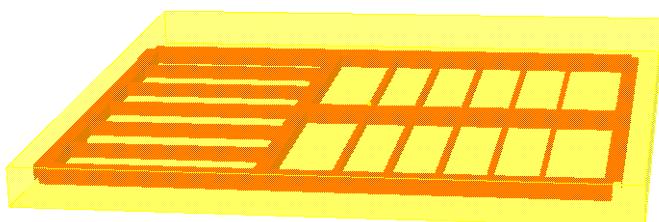
A better and more realistic moment distribution is now achieved. The peak of the bending moment has been reduced from 437.5 to 387.5 (or 12%) in this case. The differences in result depend highly on structure, equipment properties, and location of equipments.

### 3.18.3 Mass model when neglecting eccentricities

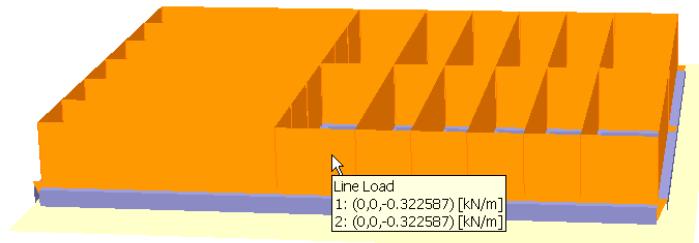
It is possible to create a mass model neglecting the eccentricities of the equipment's centre of gravity. This may be the desired mass model when working with large equipments to form uniform blanket loads (UDL) or blanket loads.

For this alternative GeniE will calculate the mass elements as for the first option (*Eccentric-Mass*), but neglect all eccentricities. Hence, it is not necessary to associate section and material properties to the equipment in this case.

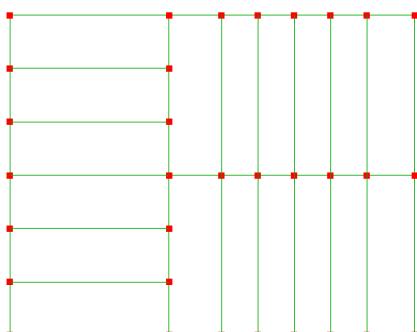
In the following is shown an example on the differences between UDL loadcase for a static linear and dynamic analysis.



The frame structure and the UDL (in this case the equipment is larger than the frame structure)

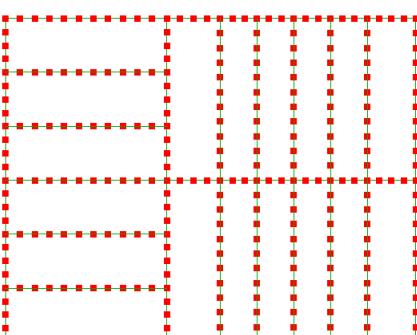


Analysis model for static analysis, constant line loads have been chosen.



A model for dynamic analysis where all eccentricities (from equipment masses) have been neglected. The mass elements are defined at the bottom level of the equipment (i.e. at the footprint level) and are thus inserted at the supporting structure level.

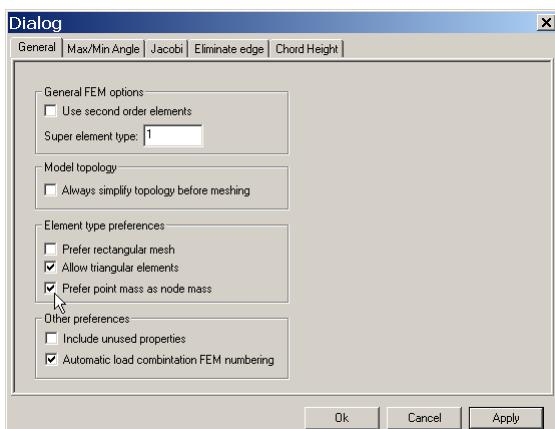
Default mesh density has been used, i.e. one finite element beam between each structural joint.



Mesh density has been refined; hence several more mass elements are automatically created.

### 3.18.4 Point mass versus node mass

Both *PointMass* and *Equipment* are concepts which may be represented in different ways in the FEM model. These concepts may currently be represented as one noded mass element(s) with optional eccentricity, or as node mass(es) when requested and possible. The user can indicate preferred representation via **Edit/Rules/Meshing**. Both mass representations will contribute to the systems mass matrix. See the Sestra user manual for more details on node and element masses.

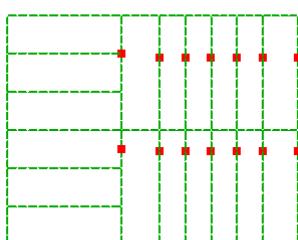
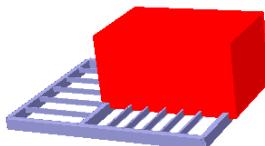


- The option ‘Prefer point mass as node mass’ is the default option, it indicates that the user prefers points masses and equipment masses to become node masses when possible.
- When deselecting ‘Prefer point mass as node mass’, points masses and equipment masses will be represented as one noded mass element(s). Such mass elements may have eccentricities, i.e. a mass may be eccentric connected to a structural node.

In most cases, ‘Prefer point mass as node mass’ will be the best choice, but there are situations where mass elements may be more suitable. This is for example when a pure mass model for hydrodynamic analysis in Wadam is created, because the Centre of Gravity will be accurate (when using node masses, the centre of gravity may be less exact due to elimination of eccentricities).

The example below shows how you can verify the differences graphically when looking at the finite element mesh.

One equipment supported by a frame grid.



The finite element when using default option ‘Prefer point mass as node mass’. The mass elements are displayed. The graphic display will be the same when unchecking ‘Point mass a one node mass element’.

### 3.18.5 Scale structural mass

When making a model it is quite common not to include all details. Examples may be structural members not adding to the load carrying structure or coating. Such details have a mass and these are not part of the mass model unless you have changed the material densities or added explicit point masses.

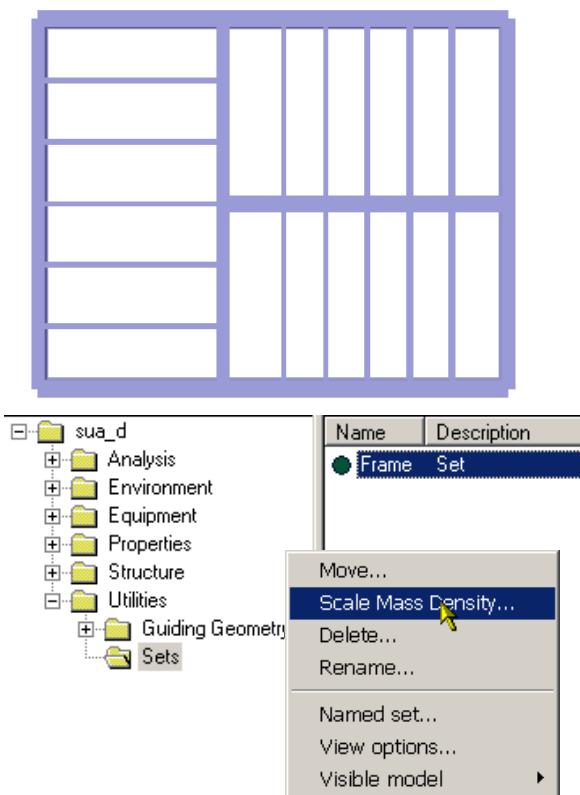
In order to overcome this limitation when making a model it is possible to scale structural masses to a given value. The principle behind such scaling is to:

- Make a named set
- Scale mass for the actual set

When scaling the masses, new material properties will be created. There are two options:

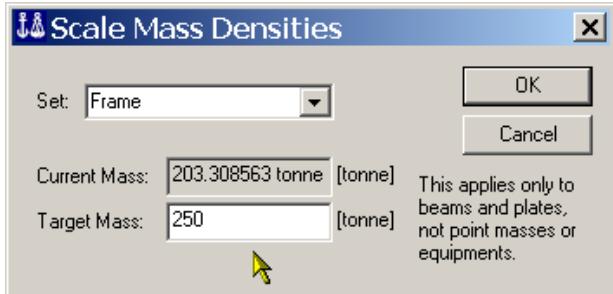
- Scale the mass, keep the materials unchanged in GeniE but update the materials as found on the FEM-file. This is the default option. Typically, the material MAT1 will be defined on the FEM-file as MAT1\_<scaling factor> and used by the relevant structural members. The scaling factor is automatically defined by GeniE and is the relation between the structural mass and the target mass.
- Scale the mass, and update the materials both in GeniE and on the FEM-file. By this you can also verify the new material names and connectivities in GeniE. You need to use the command **Tools/Properties/Create Scaled Materials** to do this.

In the example below, a named set has been created and the mass is changed from 203.308563 tonne to 250 tonne.



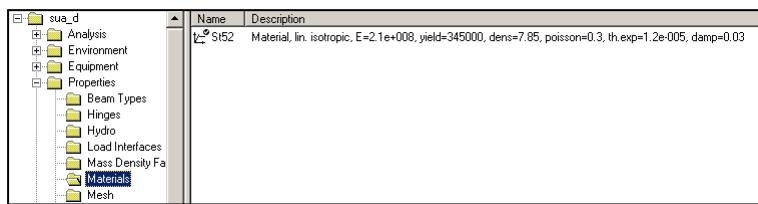
A named set (*Frame*) has been created. The structural mass is 203.308563.

Select the named set you want to scale the mass for, click RMB and choose Scale Mass Density.

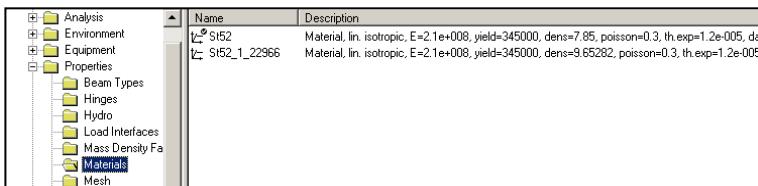


Specify the target mass and click OK. GeniE now ensures that the mass of named set *Frame* is now 250 tonne.

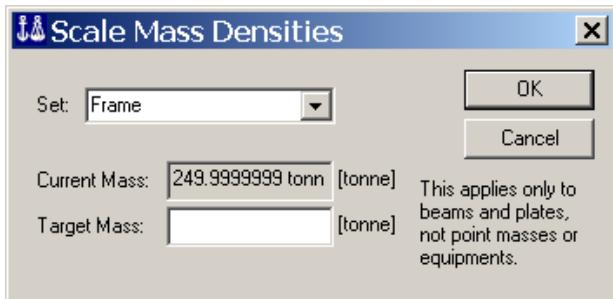
The scaling factor used by GeniE is  $250/203.308563 = 1.229657995$ . You can also find this factor from the command language window.



A new material has been created on the FEM file and connected to the relevant structural members, but not in GeniE.



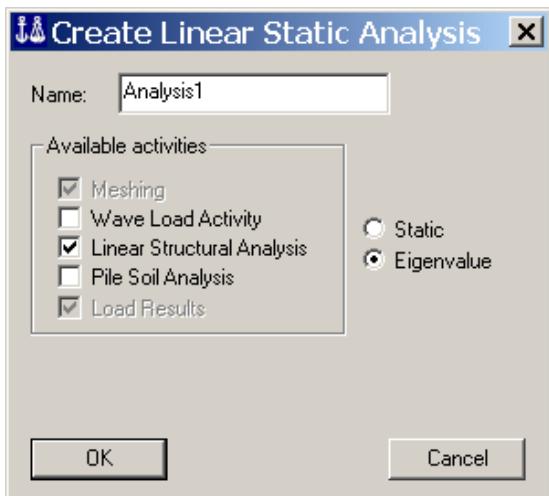
By using the command  
**Tools/Properties/Create Scaled Materials**  
a new material name is created and connected to the structural members in named set *Frame*.



The mass is now set to 250 tonne. You may change it again if you would like to do so.

### 3.19 Run Eigenvalue Analysis

Eigenvalue analysis started from the GeniE user interface is done from the menu **Tools/Analysis/Activity Monitor** (or Alt+D).

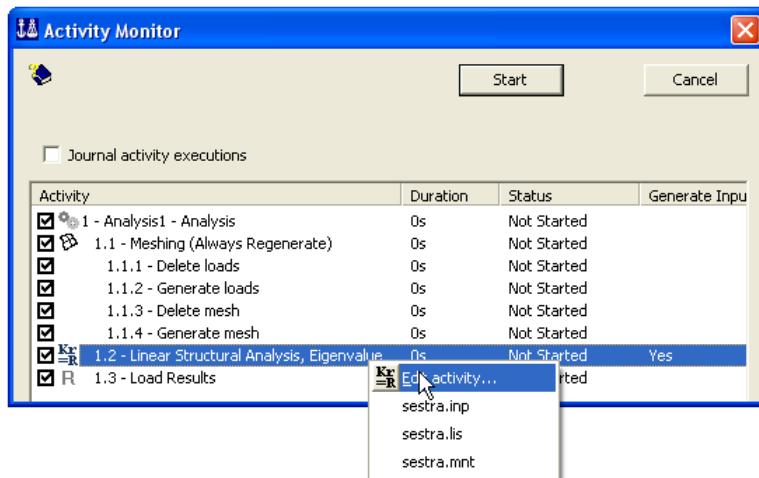


Activate the Eigenvalue option.

When analysis type is set to Eigenvalue, the 10 first eigenmodes using Lanczos method will be computed.

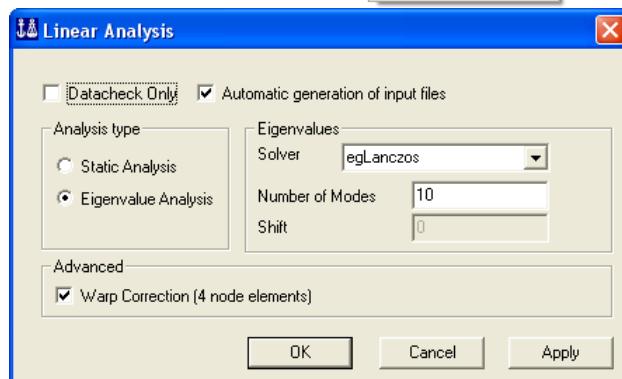
If you want masses from equipments included in an eigenvalue analysis, they must be placed in a static loadcase(s) and ‘the represent equipment as loadcase independent mass’ must be switched on.

In such cases, it is a requirement that the loadcase(s) must be numbered higher than the highest eigenvalue to avoid conflicts with result case numbering generated in the eigenvalue analysis.

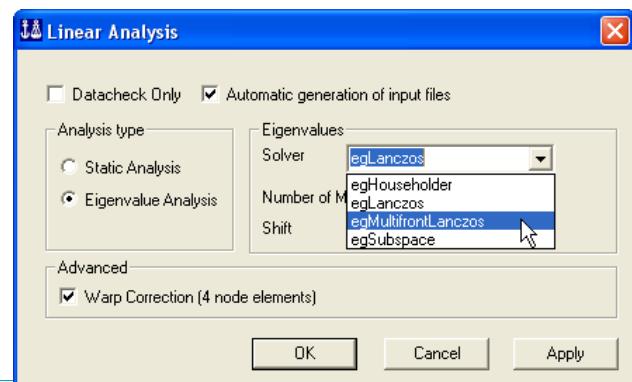


To change solver type and number of eigenmodes, select the *Linear Structural Analysis*, push **RMB** and choose *Edit activity...*.

If you are familiar with the Sestra input file from before, you may edit the input file manually as described in Chapter 3.13.1.2.



You may now change to e.g. another solver and run the eigenvalue analysis.

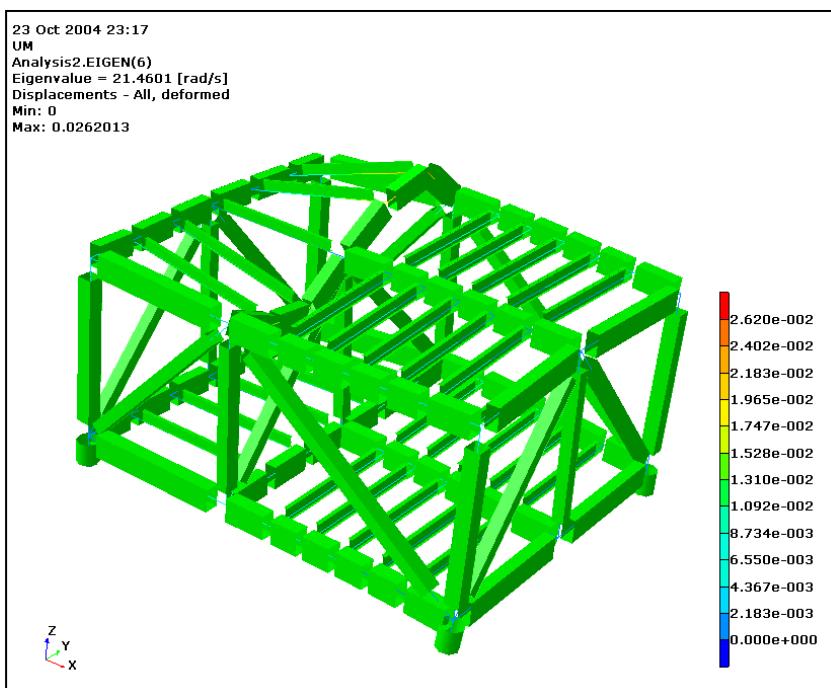


The solver MultifrontLanczos is selected.

When analysis is performed you can investigate the results from the results view (*Tools/Analysis/Presentations* or Alt+P).

	Name	Description	FEM Loadcase	FEM LC Rule
● Analysis2.EIGEN(1)	Eigenvalue	1	Variable	
● Analysis2.EIGEN(2)	Eigenvalue	2	Variable	
● Analysis2.EIGEN(3)	Eigenvalue	3	Variable	
● Analysis2.EIGEN(4)	Eigenvalue	4	Variable	
● Analysis2.EIGEN(5)	Eigenvalue	5	Variable	
● Analysis2.EIGEN(6)	Eigenvalue	6	Variable	
● Analysis2.EIGEN(7)	Eigenvalue	7	Variable	
● Analysis2.EIGEN(8)	Eigenvalue	8	Variable	
● Analysis2.EIGEN(9)	Eigenvalue	9	Variable	
● Analysis2.EIGEN(10)	Eigenvalue	10	Variable	

Eigenvalue analysis performed with 10 eigenmodes.



The 6<sup>th</sup> eigenmode shown.

```
*****
* E I G E N V A L U E S *
* F R E Q U E N C I E S *
*****
```

ALL EIGENVALUES BEING CALCULATED ARE PRINTED TOGETHER WITH THE CORRESPONDING FREQUENCIES AND PERIODS.  
FREQ = SQRT(EIGENV)/(2 \* PI)  
PERI = 1./FREQ

NO.	EIGENVALUE	UNIT: (SEC)-2	FREQUENCY	UNIT: HERTZ	PERIOD	UNIT: SEC
1	0.4061173E+02		1.014		0.98595	
2	0.4855398E+02		1.109		0.90171	
3	0.1710760E+03		2.082		0.48038	
4	0.2653113E+03		2.595		0.38575	
5	0.2945585E+03		2.732		0.36610	
6	0.321625E+03		3.495		0.28570	
7	0.4827904E+03		3.497		0.28596	
8	0.6836569E+03		4.161		0.24030	
9	0.8434090E+03		4.622		0.21635	
10	0.1166122E+04		5.435		0.18400	

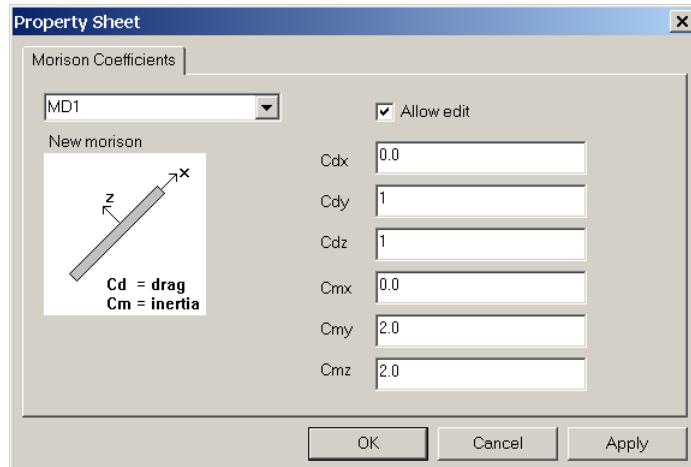
More information may also be found from the Sestra listing file.

## 3.20 Run Wave Load Analysis

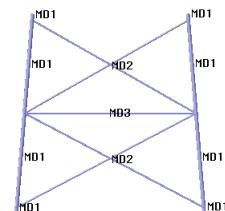
To run wave load analysis a number of wave data needs to be defined. Some of these belong to the concept structure model (Morison coefficients, non-structural elements, and flooding), while the remaining must be defined in the Wajac input file.

The hydrodynamic properties are defined as separate properties and connected to the structure model in the same way as for example a section profile or a material type. They are defined from the

### **Edit/Properties/Morison Coefficient or Flooding.**



A Morison property MD1 is defined with drag coefficients 1.0 and added mass coefficients 2.0. To verify where hydrodynamic properties have been assigned, either graphic or tabular verification can be done. The picture below shows that MD1, MD2, and MD3 have been assigned to different beams. You can do colour coding also.



	A	B	C	D	E	F	G	H	I	J	K
1	Name	X-End1 [m]	Y-End1 [m]	Z-End1 [m]	X-End2 [m]	Y-End2 [m]	Z-End2 [m]	Section	Material	Morison Coeff	Flooding
2	Beam1	-30.000	-25.000	-2.000	-25.000	-20.000	62.000	pipe125	mat1	MD1	FLO_0_9
3	Beam10	-29.845	24.845	-0.012	27.500	22.500	30.000	pipe06	mat1	MD2	
4	Beam11	-30.000	25.000	-2.000	-25.000	20.000	62.000	pipe125	mat1	MD1	FLO_0_9
5	Beam12	-27.500	22.500	30.000	25.155	20.155	60.012	pipe06	mat1	MD2	
6	Beam13	29.845	24.845	-0.012	-27.500	22.500	30.000	pipe06	mat1	MD2	
7	Beam14	-25.155	20.155	60.012	27.500	22.500	30.000	pipe06	mat1	MD2	
8	Beam15	29.845	-24.845	-0.012	27.500	22.500	30.000	pipe06	mat1	MD2	
9	Beam16	29.845	24.845	-0.012	27.500	-22.500	30.000	pipe06	mat1	MD2	
10	Beam17	27.500	-22.500	30.000	27.500	22.500	30.000	pipe06	mat1	MD3	

	A	B	C	D	E	F	G	H	I
1	Name	Type							
2	FLO_0_9	Flooding coefficient, flooding=0.9							
3	MD1	Morison coefficients, Cdx=0, Cdy=1, Cdz=1, Cmx=0, Cmy=2, Cmz=2							
4	MD2	Morison coefficients, Cdx=0, Cdy=0.5, Cdz=0.5, Cmx=0, Cmy=2, Cmz=2							
5	MD3	Morison coefficients, Cdx=0, Cdy=0.7, Cdz=0.7, Cmx=0, Cmy=2, Cmz=2							

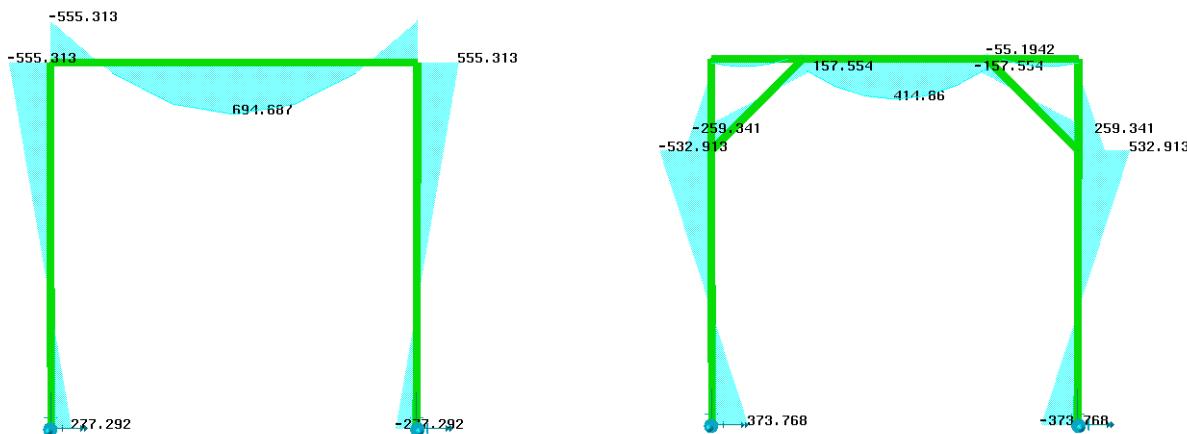
The hydro properties may be changed and edited from the property sheet. You may also find these properties from the browser area under Properties Hydro.

For further details on how to run hydrodynamic analysis, as well as pile soil analysis, please consult Vol. 2 of this manual.

### 3.21 Change and run analysis again

GeniE is designed for frequent changes of model and subsequent analysis (also known as re-analysis). Basically you change what is needed and start the analysis to get new results. It is always wise to keep copies of the journal file for the various changes so that you can easily backtrack to the various model revisions.

In the example below a frame is loaded with a uniform line load at the top beam and the bending moment diagram is shown (picture to left). The picture at right shows the same frame, but now with two supporting beams at the top corners (in total 4 clicks to insert the beams and 1 click to run the analysis).



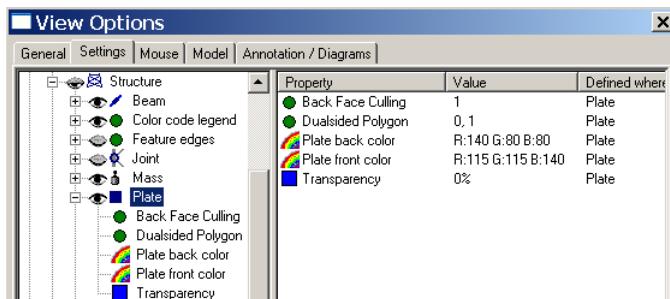
### 3.22 Panel modelling

This Chapter describes how to create a panel modelling for use by HydroD and DeepC. The panel model that describes the wetted surfaces together with the mass model (optional) may thus be used for hydrodynamic analysis in HydroD and coupled analysis in DeepC. Furthermore, you may include tank specifications in the panel model for use in hydrostatic and stability analysis in HydroD.

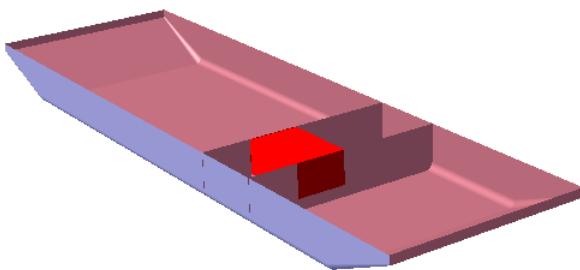
The principles behind panel modelling are as follows:

1. Create a hull form including tanks if relevant. The hull form may be modelled from scratch or by importing data from e.g. a SAT-file.
2. The plates created have a front and a back side. These can be given different wet surface properties. The front side is the one in direction of positive local z-axis for the plate.
3. A hydro pressure loadcase must be created and associated with one wet surface property only. Typically one loadcase for the wet surface on the outer shell and one loadcase per tank.

The following example shows how this can be done on a simple barge with one internal tank. Furthermore, emphasis has also been given on how to verify the panel model.

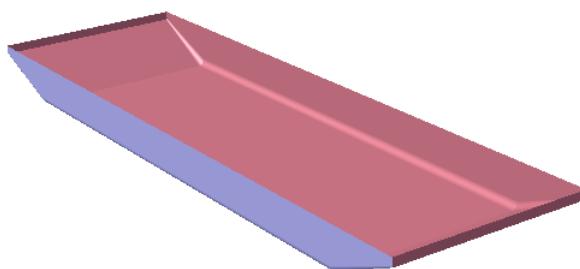


Please note that you can customize the colour appearance on front and back side of the plate from **View/Options/Settings**.



### Step 1

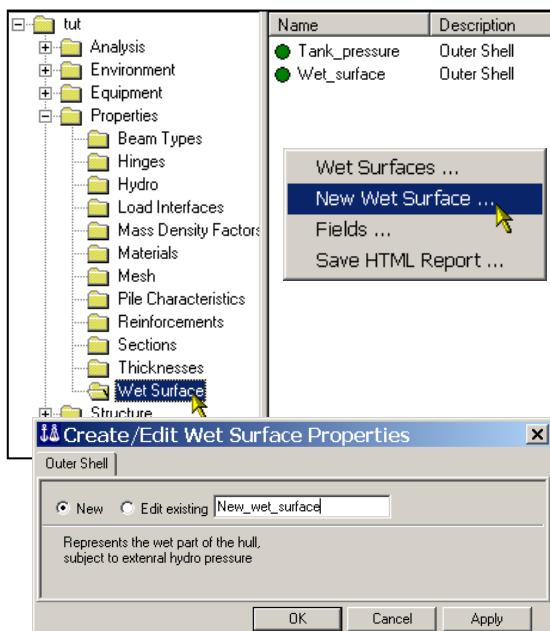
A small barge including one internal tank has been created.



### Step 2

Make a named set for the outer hull and the tank. Note that all surfaces needed to define the closed tank must be part of the set.

The figure to the left shows the named set called *Hull* while the named set *Tank* is shown above.

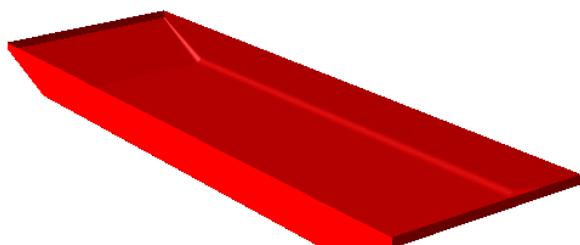


### Step 3

Make properties for the so-called wet surface. There must be one property per outer hull and one per internal tank. For example, a model with three internal tanks needs four properties.

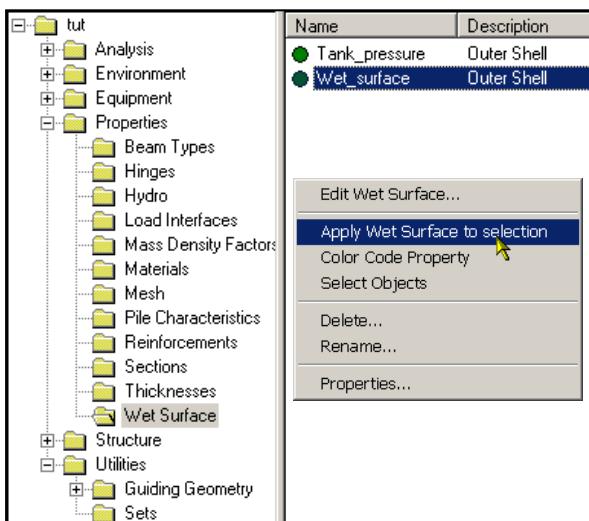
A wet surface property is defined as shown to the left or from **Edit/Properties/Wet Surface**.

In this case two properties have been defined, one for the hull and one for the internal tank.



### Step 4.

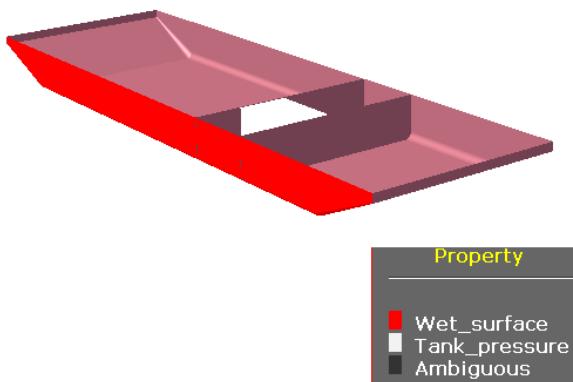
Assign the wet surface property to the outer hull. Select the outer hull from named set (in browser under Utilities, select and show visible model only). Open the folder Wet Surface, select the Wet\_surface and apply to selected members only.



The property *Wet\_surface* has been selected and may now be applied to the selected members, in this case the named set *Hull*.



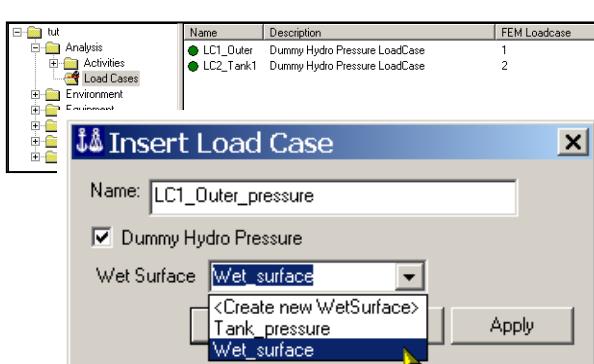
A dialogue will now ask you for which side to apply the wetted surface. In this case it has been applied to the Front side, i.e. the one in direction of positive local z-axis for the plate.



### Step 5

You can now verify that the properties wet surface have been applied to the right named sets. This can be done from the wet surface browser by selecting the properties, right clicking the mouse and select *Color Code Property*. The figure to the left shows that *Wet\_surface* and *Tank\_pressure* have been applied to the outer hull and the tank respectively.

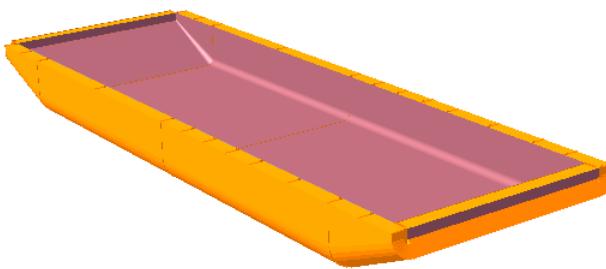
The tank pressure has been applied in the same manner as described in Step 4 above.



### Step 6

It is now necessary to create one loadcase per outer surface and one per internal tanks. You insert a loadcase as normal, in addition you need to tick off the Dummy Hydro Pressure box and specify which wet surface shall be part of the loadcase.

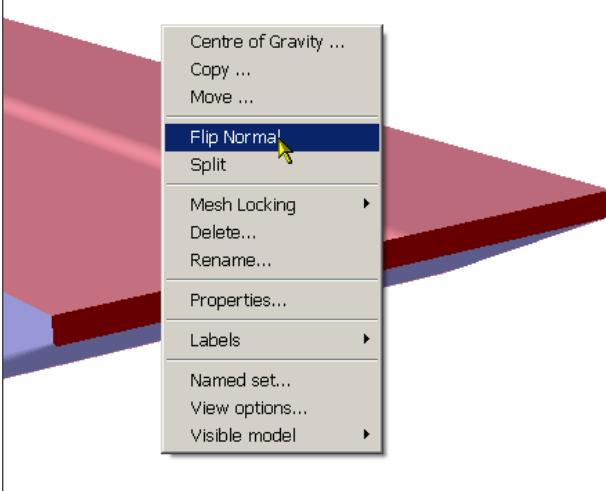
The figure to the left shows that the loadcase LC1\_Outer\_Pressure applies for the hull, while the LC2\_Tank1 applies for the internal tank.



### Step 7

You can now verify the applied hydro pressures by selecting the relevant loadcase. Before this it is necessary to generate the applied loads – either by a) selecting the loadcase, RMB and Generate Applied Loads or b) when making a finite element model (**Tools/Analysis/Create Mesh**).

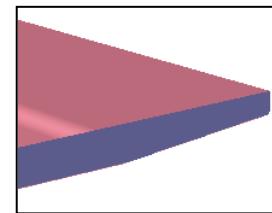
The figure to the left shows that the pressures are acting in the wrong direction at the bow and the aft of the barge.



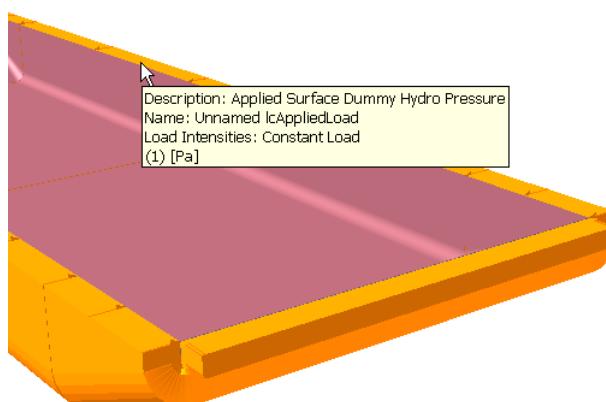
### Step 8

The plate normal may now easily be changed by selecting the plate, RMB and **Flip Normal**.

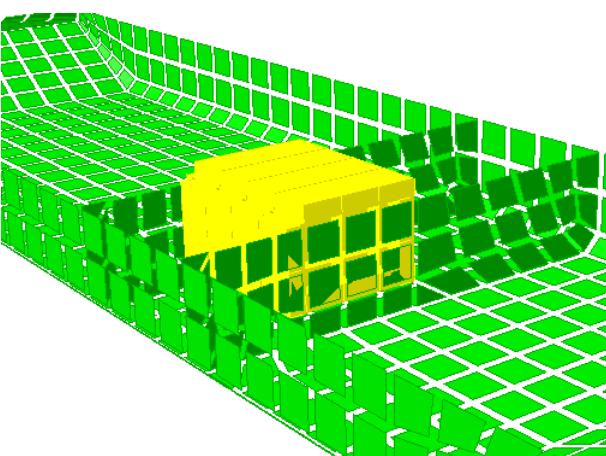
Control of a plate normal may be done at any time; when you change it that direction of the pressure is changed accordingly.



The end plate has now changed orientation as shown to the right.



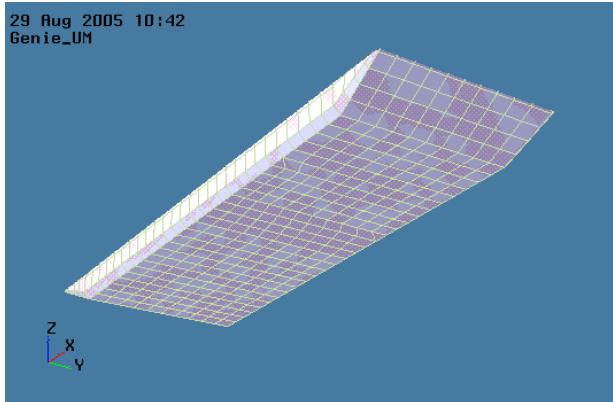
When displaying the loadcase, the pressures are now acting in the intended direction.



### Step 9

When you have ensured that the hydro pressures are acting as intended you can create the finite element model (also referred to as the panel model) by **Tools/Analysis/Create Mesh**. The finite element model may also be given an explicit name for easier import to HydroD and DeepC (**Tools/Analysis/Export FEM file**).

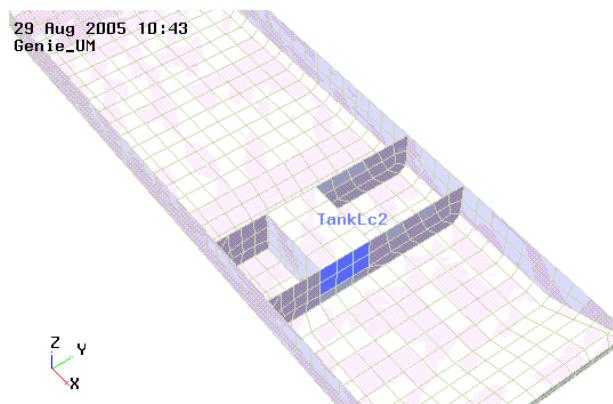
The figure to the left shows the tank pressure applied to the finite element model (loadcase *LC2\_Tank1*).



The model has now been imported to HydroD for subsequent hydrodynamic or stability analysis.

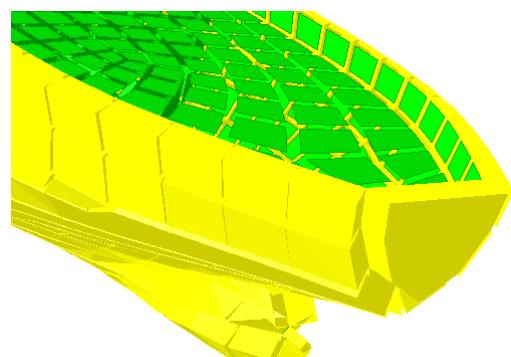
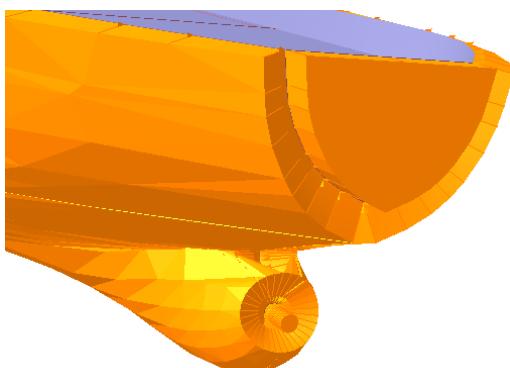
The figure to the left shows the panel model seen from below.

Note, this figure has been created by HydroD.

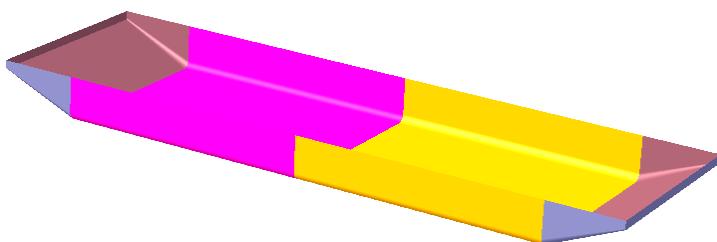


Similarly, the tanks may be visualised in HydroD as shown on this picture.

The pictures below show the hydro pressure on the concept model as well as on the finite element model to be imported by e.g. HydroD or DeepC.

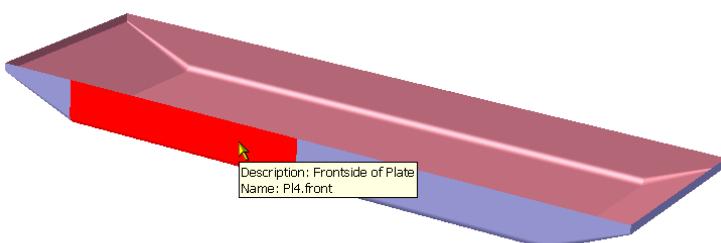
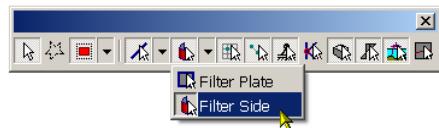


You may also apply wet surface by using a feature for selecting only one side of a plate. The example below illustrates how this feature works.

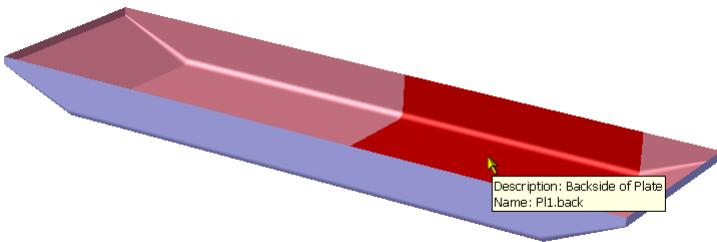


The barge has two large plate concepts in the middle part.

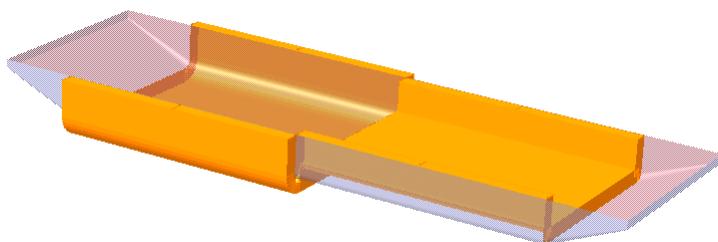
By using the filter side it is now possible to select one side only of a plate.



The outer part (front side) of the plate is selected and wetted surface can be applied.



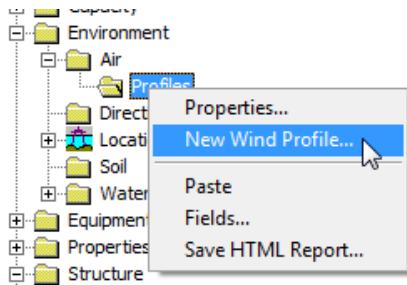
The inner part (back side) of the is selected and wetted surface can be applied.



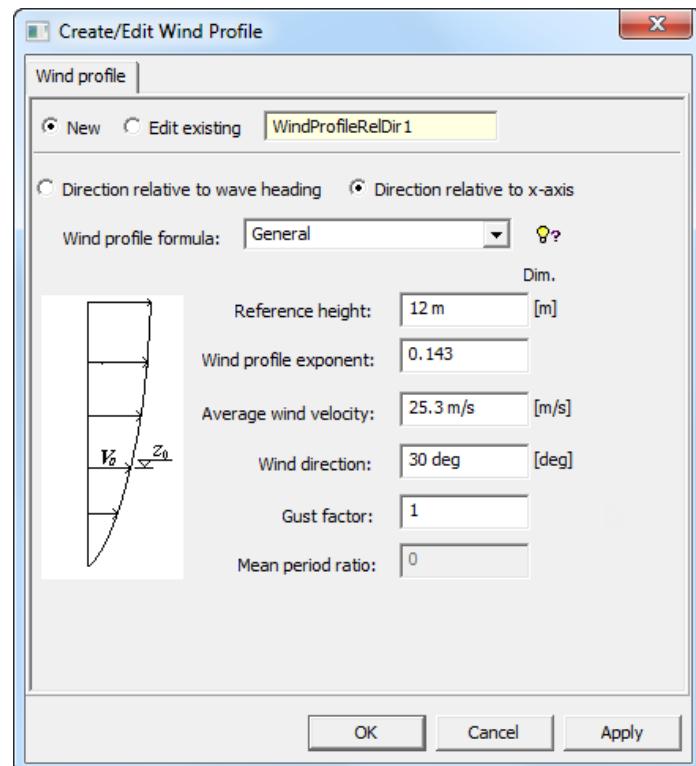
Hydrostatic pressure has been created – as can be seen the direction differs for the two plates in question.

## 3.23 Wind loads

### 3.23.1 Wind loads on equipment

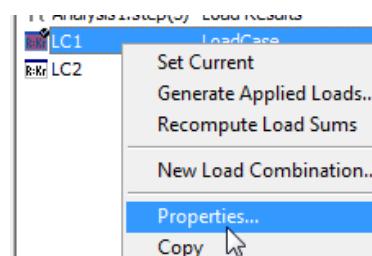


To include wind loads in a loadcase you first have to create a wind profile. This can be done from the browser. RMB “New Wind Profile” under *Environment/Air/Profiles*.



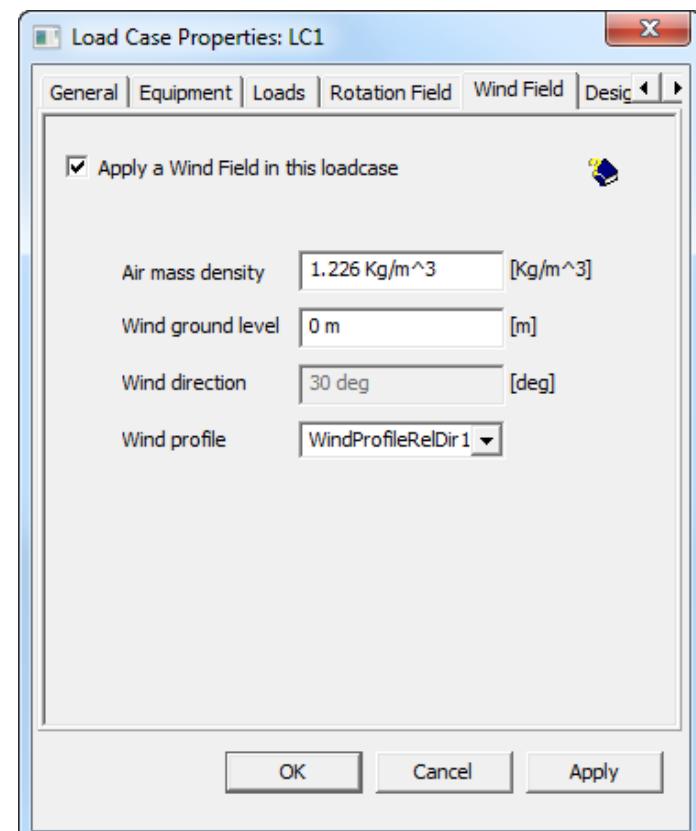
Enter the values corresponding with the wind profile you want to create.

This dialog is explained in Volume 2, chapter 5.2.3



Assign your wind field to a loadcase.

- In the “Wind Field” tab in the dialog you define the global wind field properties for a basic loadcase.
- Air density: The mass density of air (default value is 1.226 kg/m<sup>3</sup>).
- Wind ground level: The sea surface or ground level, in model coordinates, where the wind profile begins.
- Wind direction: The direction of the wind, used only when the “Relative to Wave direction” option was used when specifying the wind profile. The input field is disabled for editing when the “Relative to X” option was used when creating the wind profile. In this case the field shows the wind direction associated with the wind profile.
- Wind profile: Name of the wind profile used in this loadcase.
- The wind field is used to set up wind loads on plates and equipment with “Wind Pressure” loading assigned.



Note that wind load on equipments and plates cannot be included in ordinary wave/wind load cases (i.e. load cases for use in Wajac).

Hence, if the wind profile used has been defined by the "Direction relative to wave heading" option, the actual wind direction must be defined on the Wind Field assigned to the load case.

The wind direction is then defined relative to model x-axis, counter-clockwise and 0 degree is wind in positive X axis.

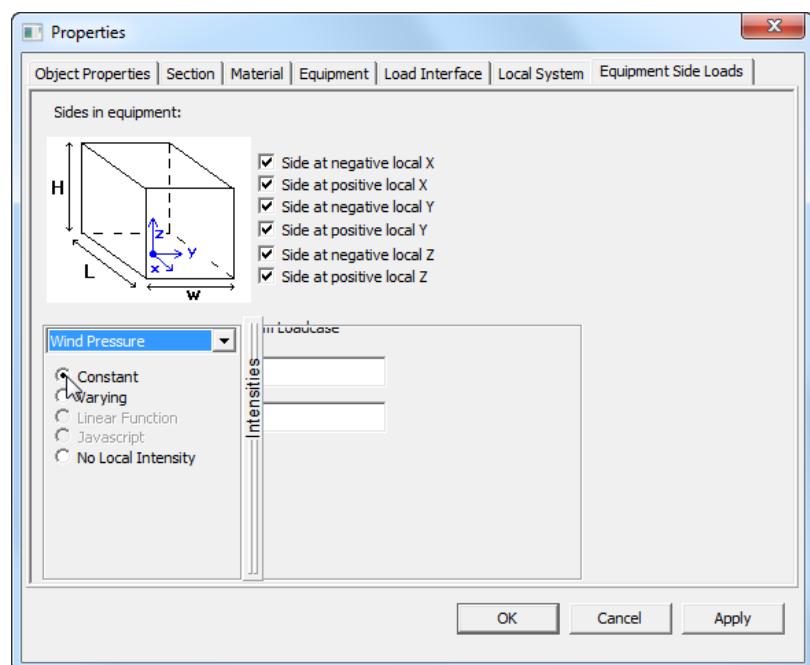
However, if wind direction is also given on the Wind Profile (i.e. different from zero), this direction is added to the direction given in the load case Wind Field.

To apply wind loads to an equipment, select the placed equipment in the loadcase, rightclick and select "Properties".

Select "Equipment Side Loads" and select "Wind Pressure" in the drop down menu and the "Constant" or the "Varying" radio button (see explanation below).

**Constant** will create a constant wind load so that the forces are the same for all z-values. The constant force will be the force calculated at the equipment's centre of gravity

**Varying** will create a wind load that varies along the z-axis according to the wind profile you specified earlier.



Insert a drag coefficient and a suction factor.

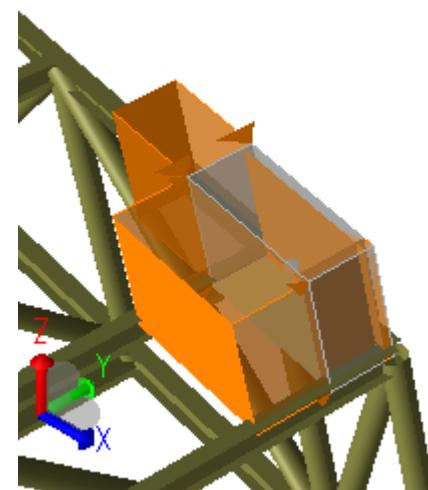
Constant Wind Pressure from Loadcase	
Drag coefficient	<input type="text" value="1"/>
Suction factor	<input type="text" value="0.1"/>

Drag coefficient is the factor the integrated pressure shall be scaled with to give total force on surface.

Suction factor; default value is 0.0. Note that when giving another value, ensure that the same value are given on the leeward side.

Run analysis or generate applied loads.

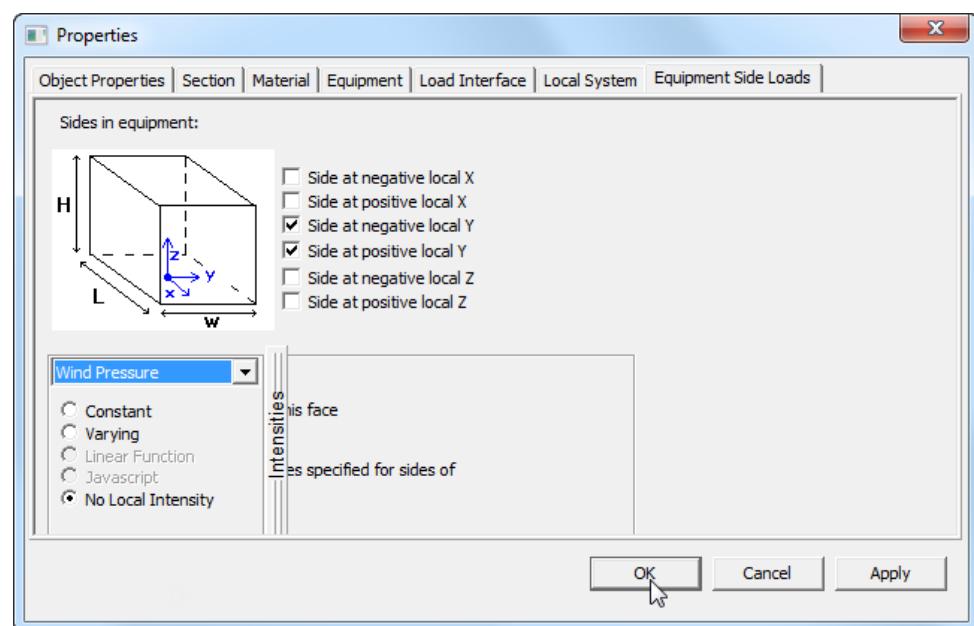
The placed equipment with the corresponding wind loads is visible in the graphical display.



If you want to remove the wind loads on some of the sides of the equipment, check the sides you want to remove in the dialog. In the example to the right sides at negative local Y and positive local Y has been selected.

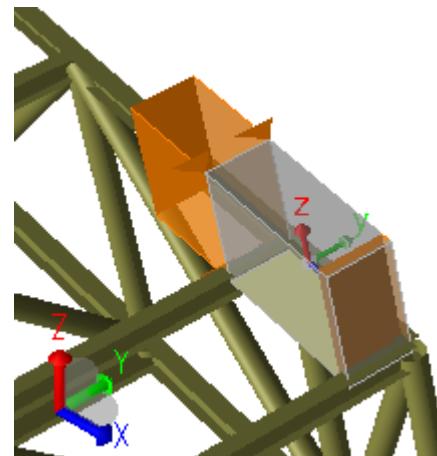
Open the Intensities sub dialog and select “No Local Intensity” as shown in the illustration.

Then click “OK”.

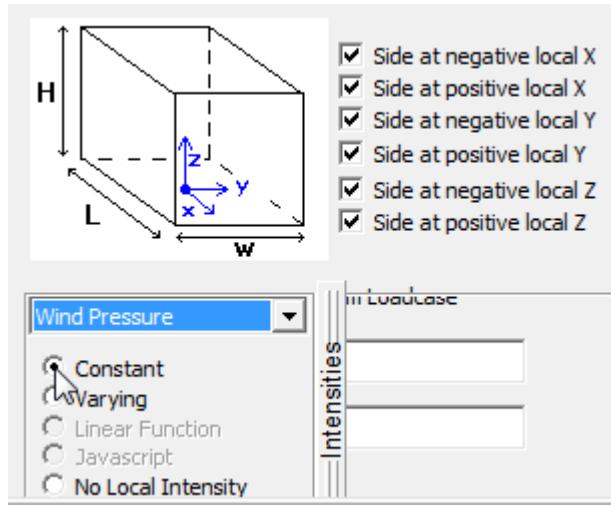
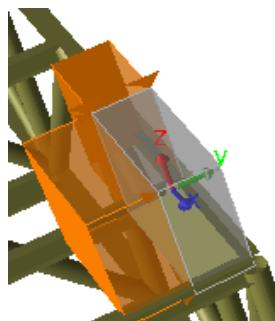


In the graphical view you can observe that loads on negative and positive local Y sides is removed.

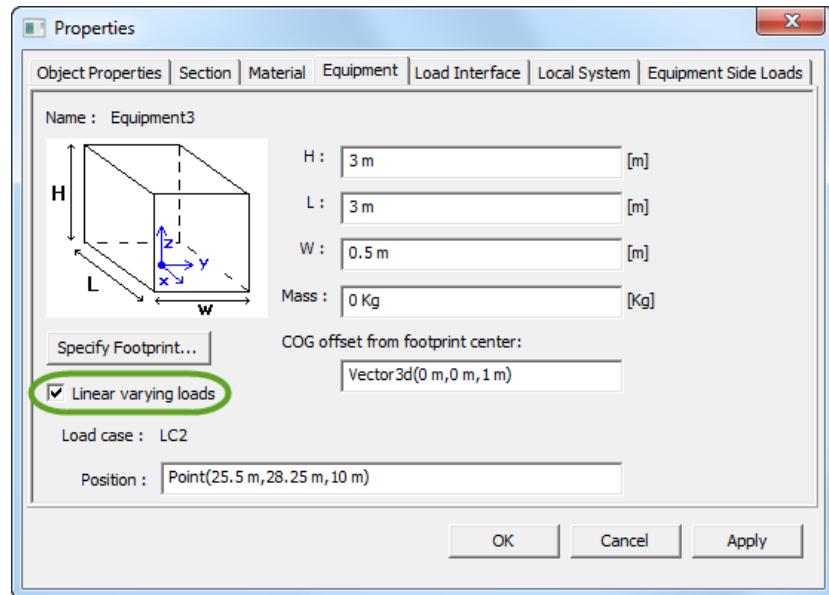
If you open the Properties dialog again you will see that the checkboxes for side at negative and positive local Y are unchecked.



If you want to include the loads on the Y-sides again, check all the sides you want to include, select Constant or Varying in the Intensities sub menu and click “OK”. In the graphical view you can observe that the loads on the Y-sides are once again present.



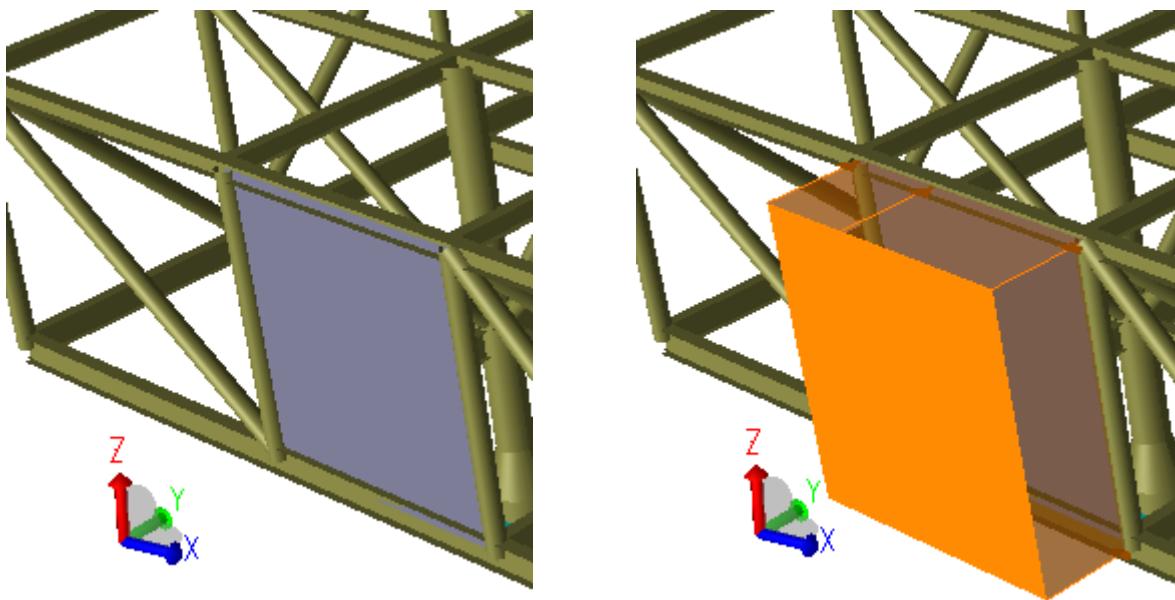
Note the effect of checking/unchecking “Linear varying loads” in the equipment properties dialog with respect to transferring wind loads to the structure.



<p>Linear varying loads checked. Overturning moment from the wind load is transferred to the structure.</p>	<p>Linear varying loads unchecked. Wind loads are transferred as shear forces only. This is usually the preferred method.</p>

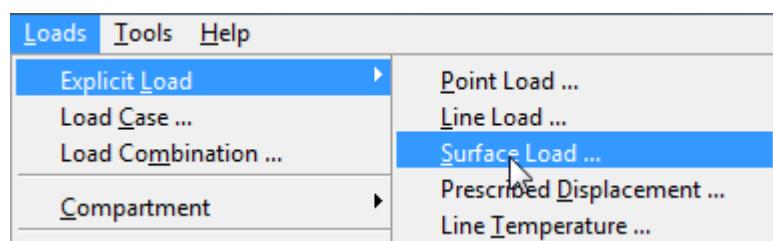
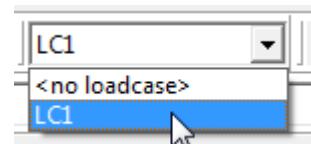
### 3.23.2 Wind loads on plates

You can assign wind loads to plates similar to how it's done for equipment.



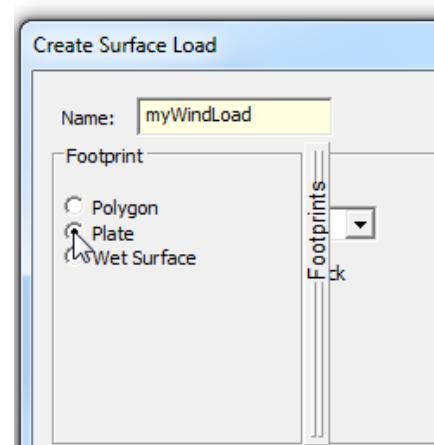
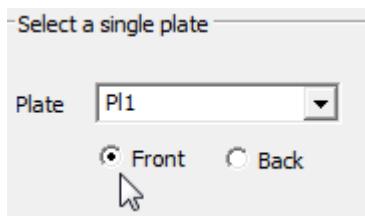
To assign a wind load to a plate first select your loadcase.

Then select Loads | Explicit Load | Surface Load.

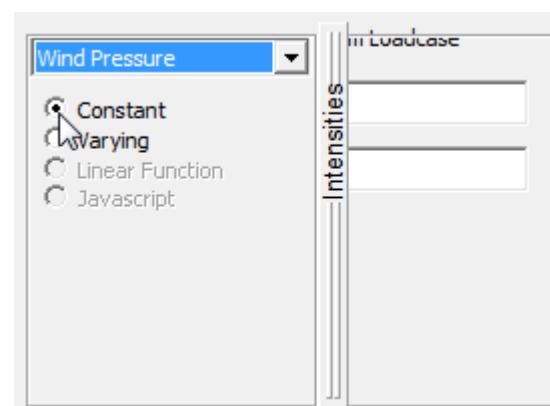


You then apply the surface load to a plate.

Select if you want the wind load to be applied to the front or back of the plate.

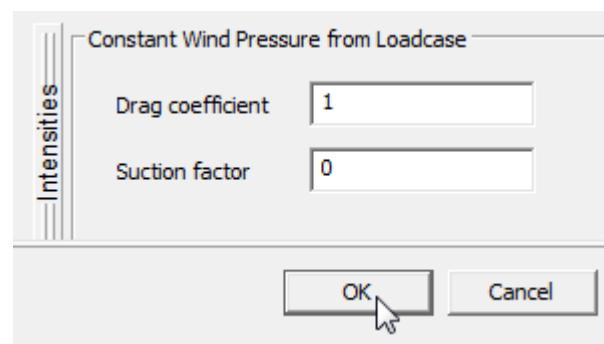


Select constant or varying wind pressure.



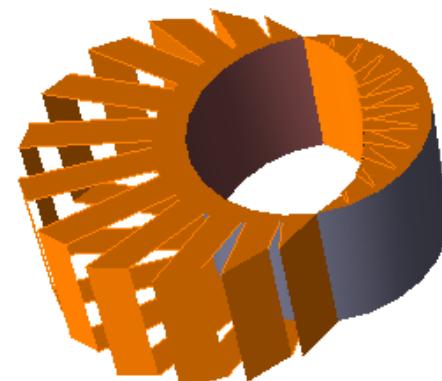
Select drag coefficient and suction factor and click "OK".

Note: For a flat plate you should let the suction factor remain at value 0.



For a curved plate, like the cylinder in the illustration to the right, you might want to set the suction factor larger than 0. In this case the suction factor is set to 0.3.

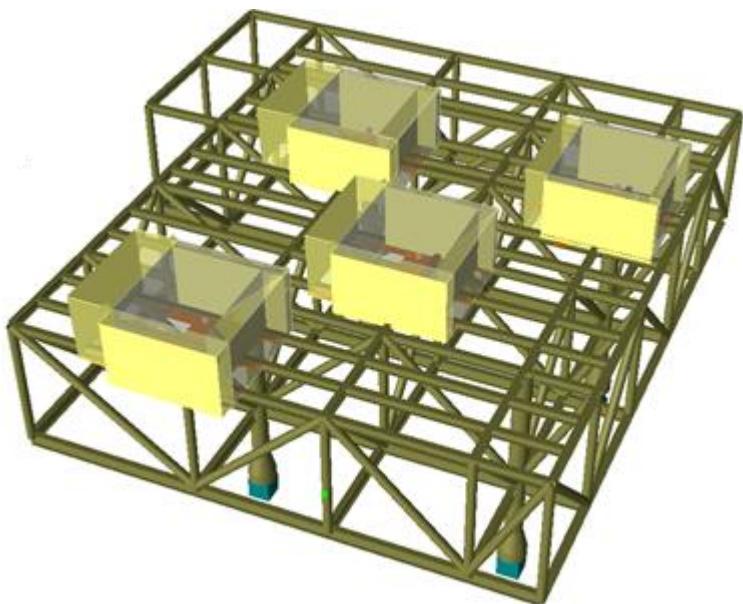
Note that for the cylinder case, modifying the suction factor will modify the pressure force and suction force while the total force remains constant.



## Wind loads – general

Note that you can use different drag coefficients to account for equipment that are partly shielded from the wind.

In the illustration to the right identical equipments are given different drag coefficients and hence the windfield gives different forces on them.





## 4. EXECUTION OF GENIE

This Chapter described how to start the program, which files are used by the program, and any high level limitations.

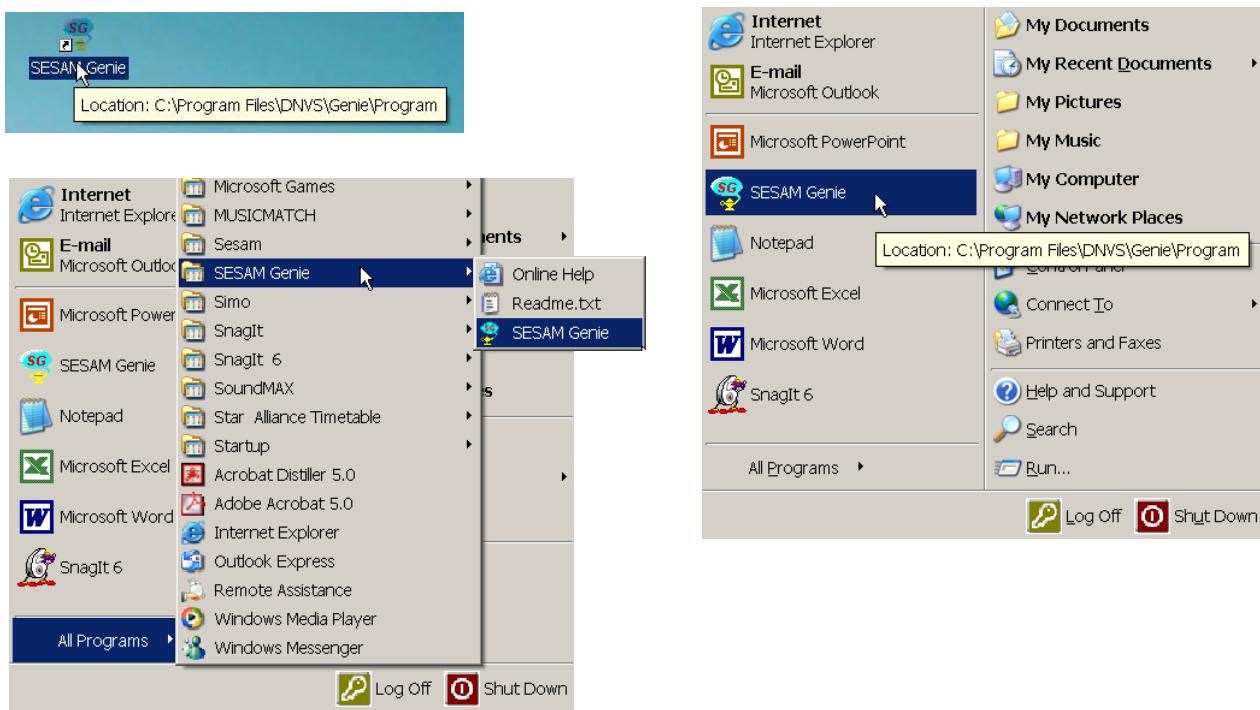
### 4.1 Program environment

GeniE runs in a Windows environment, be it Win 7 or Win 8.

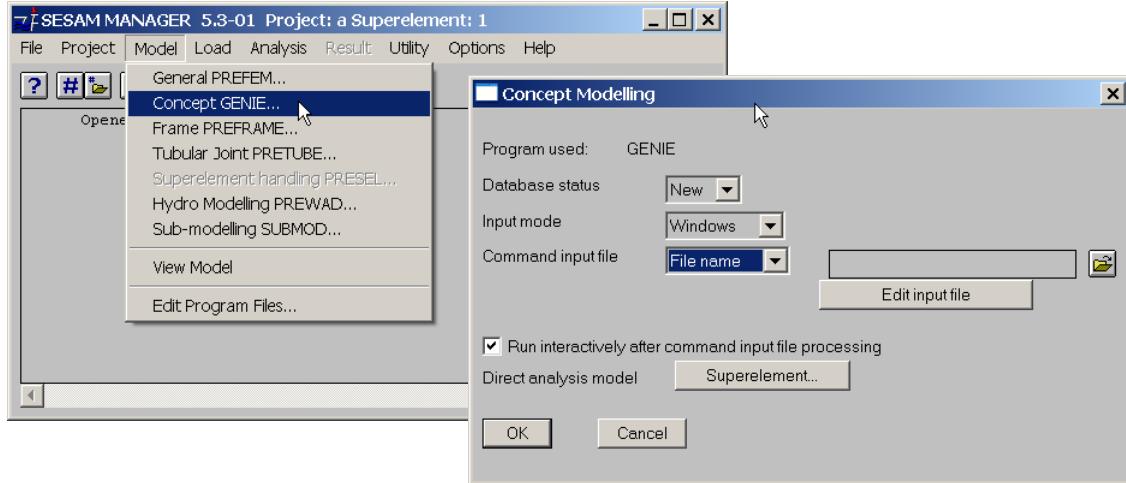
From V6.0 GeniE is available both as a 32 bit and a 64 bit application.

GeniE 64 bit can only be installed on a 64 bit operating system. It allows for a larger database, hence the models can be larger. The database can be up to 2 GB compared to 1 GB for GeniE 32 bit.

The installation program inserts an icon on your desktop as well as on your start menu. This means you can start the program in three different ways as shown below.



By changing the “start in” location in icon property dialog, you can select where workspaces will be created. SESAM Manager also supports GeniE, and you can run the program from this environment. This option is of importance when running GeniE in a superelement analysis together with other SESAM programs.

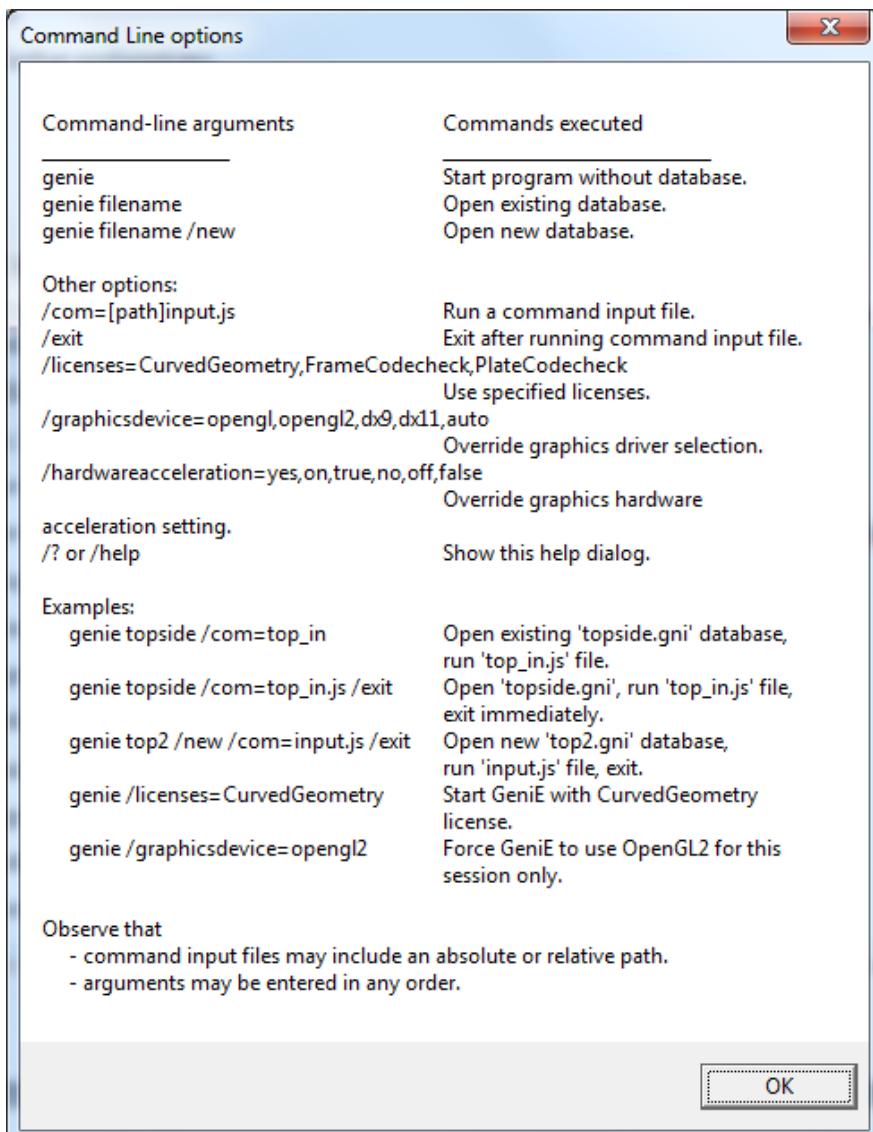


You may also start the program from a DOS window. With this option, you can add more execution information like for example “start the program, import a journal file, and exist while done”. You start the program from the installation folder (default: C:\Program Files\DNVS\GeniE\Program) as shown:

```
C:\>cd program files
C:\Program Files>cd dnus
C:\Program Files\DNUS>cd genie
C:\Program Files\DNUS\Genie>cd program
C:\Program Files\DNUS\Genie\Program>Genier
```

The program now starts, but no files are automatically read in.

By typing GeniEr /? the following information is provided on the screen:



This sheet provides information about the various command-line arguments that can be used when starting the program. For example:

C:\Program Files\DNVS\GeniE\Program\GeniEr top2 /new  
/com=input.js /exit

See sheet to left for further explanations.

## 4.2 Files used by GeniE

GeniE (and Sestra – Wajac – Splice (including Gensod) when started from the GeniE GUI) produces and uses the files listed below.

- The journal file (including the clean export functionality), format \*.js
- The database, format \*.gni
- The GeniE saved report, format \*.lis, \*.html, \*.xml
- The GeniE saved graphics file, format \*.gif, \*.jpg, \*.eps, \*.bmp, \*.tga, \*.tif, \*.ps, \*.vrml, \*.dfile
- The GeniE\_log.txt file in case of model errors and/or warnings
- The SESAM neutral file, format T\*.FEM
- The Sestra input file, format \*.inp
- The Sestra listing file, format \*.lis
- The Sestra maintenance file, format \*.mnt
- The Wajac input file, format \*.inp
- The Wajac listing file, format \*.lis
- The Wajac result file, format L\*.FEM
- The input data for soil (Gensod), format \*.INP
- The Gensod listing file, format \*.LIS
- The Splice input data, format \*.INP
- The Splice listing file, format \*.LIS
- The SESAM results file, format R\*.SIN
- Start up file for Framework, name fw\_new.jnl

These files are all stored on a directory (the user is prompted for installation route, in this case Workspaces):

C:\DNVGL\Workspaces\GeniE\Project name

All relevant files may be accessed from the GeniE user interface (from the *Activity Monitor*), meaning that you do not need to manually find them. You do not need to access these files to run GeniE, but if more details are needed it is convenient to look in these files.

The journal file and database are saved when you exit the program or save it. There are no automatic save in GeniE, you should thus remember to save from time to time. If you are in a “what-if-type” of modelling stage you should save before you do a change. If you are not satisfied, you can always close down the session without saving and go back to the previous revision.

While modelling GeniE will journal all commands onto a journal file named *<time stamp><workspace name>.js*. Each time pressing Exit or Save Workspace, the content of this file is copied to a journal file called *<workspace name>.js*. If for some reasons e.g. the program will crash or you encounter a power failure, you can re-run your model by using the time stamped journal file as command input file.

Alternatively, use **the File/Export/GeniE journal file (js)** to get a clean journal file without history.

## 4.3 Program limitations

The program limitations are relative to the content of this User Manual. Program errors and other limitations as such are listed on the GeniE status list.

For further details, please consult the status list system on

<https://projects.dnv.com/sesam/status/GeniE/GeniE.html>

The status lists are updated every week-day and comes with a free text search tool.

## 4.4 Program extensions

GeniE comes in three variants:

1. **GeniE** together with extensions. This is a standalone tool that contains all functionality covered by this user manual.  
To do curved shell modelling it is required to acquire the extension GeniE Curved Geometry (CGEO).  
To do beam capacity check, the extension CCBM is required.  
To do plate capacity checks, the extension CCPC is required.

In addition the user needs separate licenses for other program modules such as the analysis program Sestra, wave loading program Wajac, pile-soil interaction program Splice and code checking program Framework.

2. **GeniE.Lite** which is a subset of GeniE Basic bundled with Sestra. Curved geometry and beam capacity checks are supported by the GeniE Lite license, plate capacity checks are not.
3. **GeniE.RCLite** (light version to FE-import model and do code check according to ship rules)

### 4.4.1 The GeniE Lite licence

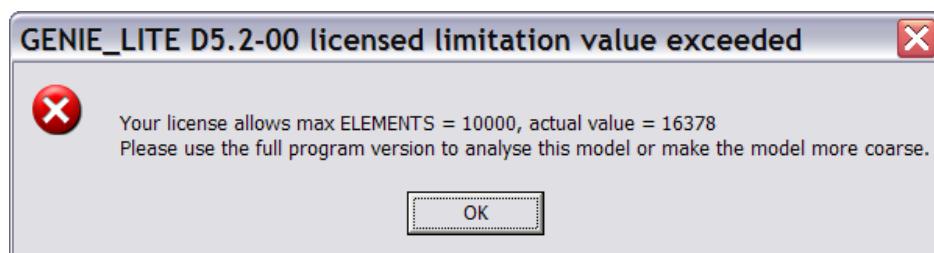
With a GeniE Lite license you can use GeniE in exactly the same way as for the full license with a few limitations:

1. Your mesh cannot contain more than 10.000 FEM elements
2. Your model cannot contain more than 500 beam concepts
3. Wave loads and Pile Soil analysis are not supported.

The first two limitations are enforced when creating a finite element mesh.

### Max 10.000 FEM elements

Under the GeniE Lite license you cannot run an analysis on a model with more than 10.000 elements. If you try to do this, the following message will appear:



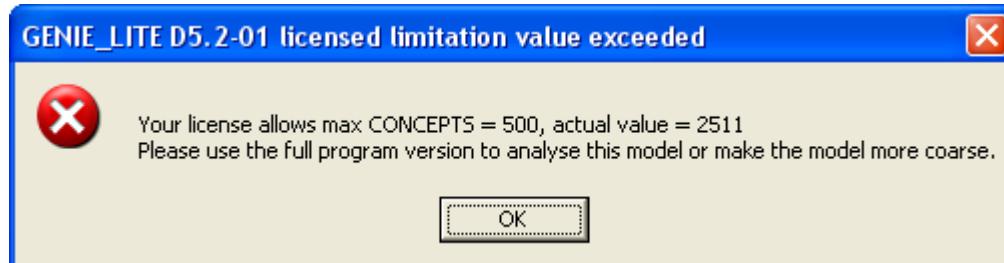
If you get this message you could try to make your mesh coarser in order to reduce the number of FEM elements.

If a coarse mesh does not give you a satisfying result, you should consider upgrading to a full GeniE license.

## Max 500 beam concepts

Under the GeniE Lite license there is one limitation to your concept model. Your model cannot contain more than 500 beam concepts when running the analysis. There are no limitations on other concepts, like plates, support points etc.

If you try to run the analysis with more than 500 beams, the following message will appear:



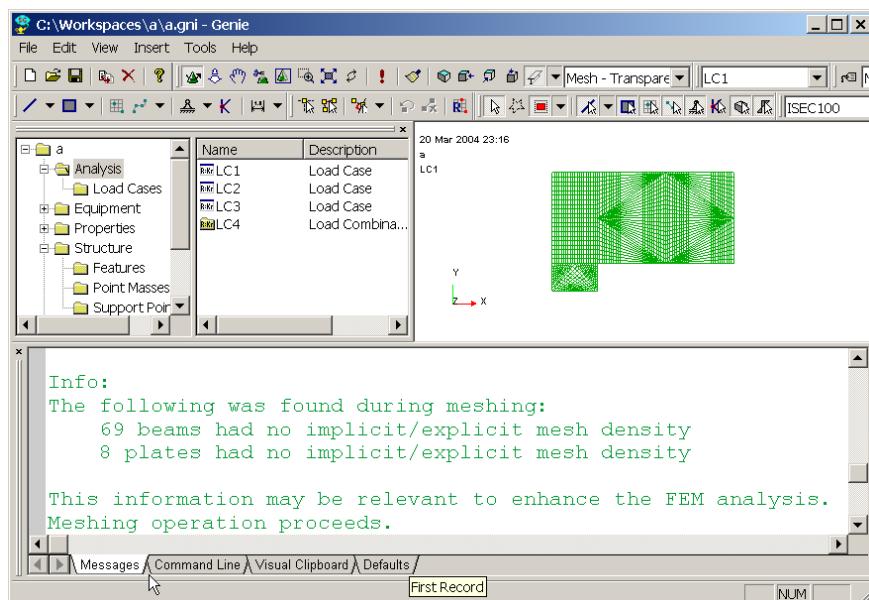
If you exceed the limit of 500 beam concepts with just a few beams, you should try simplifying your model.

If you exceed the limit of 500 beams with a significant amount, you should consider upgrading to a full GeniE license.

### 4.4.2 The GeniE RCLite licence

- No modeling features
- Only change and edit functionality :
  - Properties of structure/mesh (thickness, material, sections, corrosion additions)
  - Edit mesh (basic features )
- BC & Loads through NH import - ( modelling of support points & curves available, for repair of model)
- Compartment generation
- Plate code check (CSR and DNV GL rules)
- Reporting

## 4.5 Messages produced by GeniE



During an interactive session (or when importing data from other systems or a js-file), GeniE reports any warnings or error message on the Message Area on the Graphical user interface. The same messages are also copied to the file GeniE\_log.txt.

In addition other information is posted under the Messages Tab.

## 4.6 Software and hardware recommendations

Processor	At least a dual core processor is recommended as you can then work on your computer even when GeniE is running an analysis etc.  If you are planning on running PULS, you can speed up the process a lot if you have several processor cores as PULS is able to use all available cores in parallel.
Memory	At least 4 GB of RAM recommended.
Screen	24" screen with a resolution of 1920 x 1200 recommended.
Disk Storage	500 GB recommended.
Software	To be able to view the XML-reports created from the <b>File/Save report</b> you need Microsoft Office 2003 or newer – this also applies to running the wizards.  The user documentation is best viewed using an Acrobat Reader version 4.0 or later.

## 4.7 Database version checking

As the GeniE program evolves, the internal database schema sometimes changes (perhaps even significantly), in order to cater for new user demands in functionality or performance. This implies that the user must be aware that the database created with one version of GeniE cannot be guaranteed to be readable by another version of GeniE. As described in section 3.1.8, the compatibility over program versions is supported via the journal files. You can use the journal file created from an interactive session, one that has been edited or the clean journal file. Another compatibility alternative is XML (export XML file using old program version, import XML file using new program version).

Starting with GeniE version 2.5, the program performs an explicit database version check when opening an existing database. The requirement for success is that the database was originally created with the same version of GeniE as the one currently being used for accessing the database. If a difference is detected, the database will not be opened, and the user is instructed to instead use the journal file to recreate the database for the current version of GeniE.

For special cases, the database checking can be by-passed by means of a special "/nover" command line parameter. NOTICE: You are advised not to use this feature unless you are sure there is no other way to recreate the database for the current version (i.e. you are advised to use the journal file whenever possible). Overriding the database version check can cause unpredictable effects, including sudden program stop. Contact [software.support@dnvgl.com](mailto:software.support@dnvgl.com) on advice how to use the "nover" feature.

## 4.8 Backups

Each time you save your workspace, either by selecting File/Save Workspace or by clicking Ctrl + S, GeniE creates a backup of your workspace.

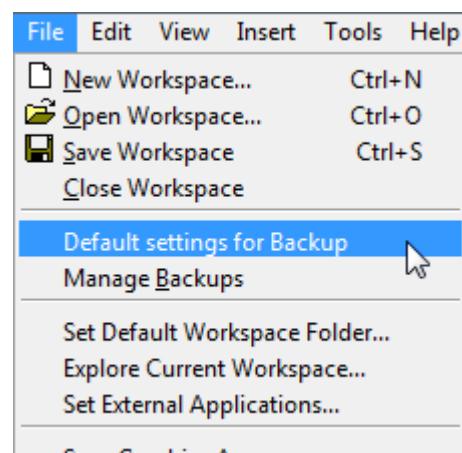
The backup system backs up *gni*, *sat* and *js* files for the current workspace.

Note that *fem*, *sin* and *hdf5* files are not handled by the backup system. These must be regenerated by running the analysis and code check after recovery.

### 4.8.1 Default settings for Backup (only for new workspaces)

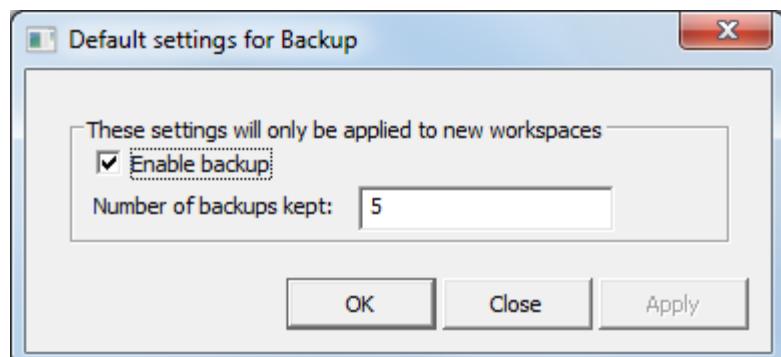
To set how many backups you want GeniE to keep, select File/Default settings for Backup.

In the “Default settings for Backup” you can change the number of backups that GeniE is keeping. The default is 5. This means that the 6. time you save your workspace the first backup will be discarded. The 5 latest saved workspaces are backed up.

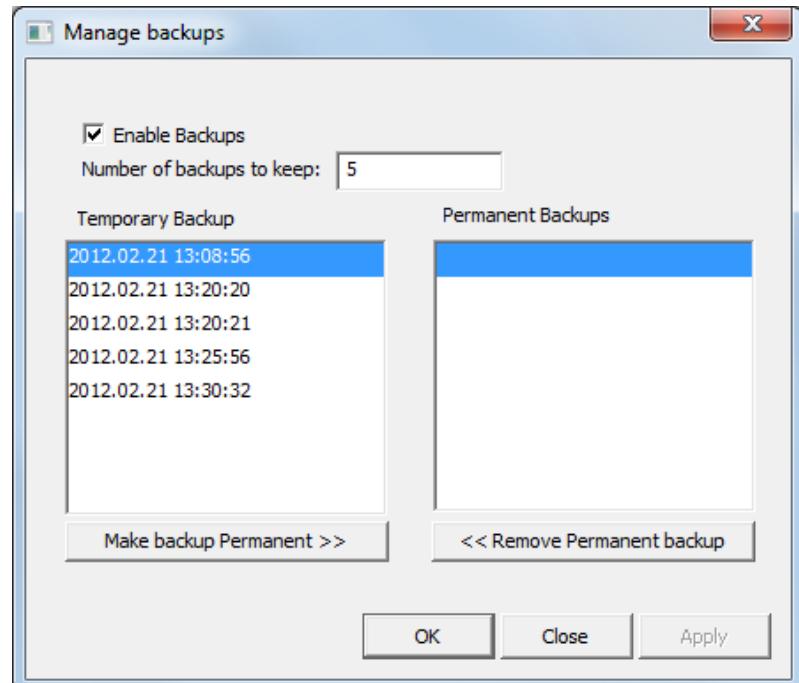
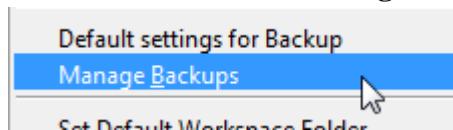


You can increase this number, but you are advised to keep it within a reasonable size, as a very large number of backups will fill up your disk. It will also be hard to keep track of the files if you have a list of hundreds of backups.

Note that changing this number will not have effect on the current workspace.



## 4.8.2 Manage Backups (for the current workspace)



### Enable backups

When this is checked backups are enabled. It's recommended to keep this checked.

### Number of backups to keep

This shows the number of backups to be kept for the current workspace. If you want to set the default number of backups to be kept, see how to do this in the previous paragraph.

### Make backup Permanent

You can make a backup permanent by selecting one of the temporary backups and clicking the "Make backup Permanent" button.

### Remove Permanent backup

Select one backup in the list and click "Remove Permanent backup" to remove a backup from the permanent backup list and move it back to the temporary backup list.

### 4.8.3 How to recover a backed up workspace

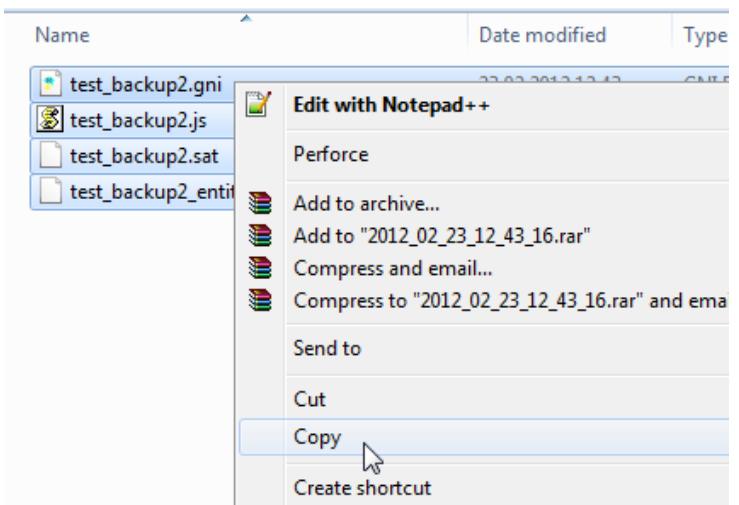
Open your workspace in the file explorer. Your workspace will look similar to the illustration to the right. The backups are located in the folder called "Backups". Open this folder.

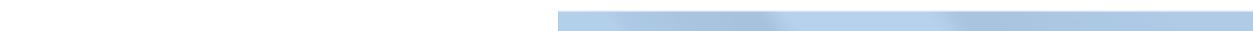
Name	Date modified	Type
Analysis1	23.02.2012 12:42	File folder
Backups	23.02.2012 12:43	File folder
acis_debug.txt	23.02.2012 12:43	Text Document
brsimport_log.TXT	23.02.2012 12:43	Text Document
brsimport_log.TXT.bak	23.02.2012 10:57	BAK File
brsSesamT1_log.TXT	23.02.2012 12:42	Text Document
MFR_Parameter.DAT	23.02.2012 12:42	DAT File
test_backup2.gni	23.02.2012 12:43	GNI File
test_backup2.js	23.02.2012 12:43	JScript Script

Inside the Backups folder, open the backup folder corresponding with the workspace you want to restore.

Name	Date modified	Type
2012_02_23_10_58_05	23.02.2012 10:58	File folder
2012_02_23_12_22_20	23.02.2012 12:22	File folder
2012_02_23_12_22_56	23.02.2012 12:22	File folder
2012_02_23_12_34_45	23.02.2012 12:34	File folder
2012_02_23_12_41_32	23.02.2012 12:41	File folder
2012_02_23_12_43_16	23.02.2012 12:43	File folder
BackupList.xml	23.02.2012 12:43	XML Document

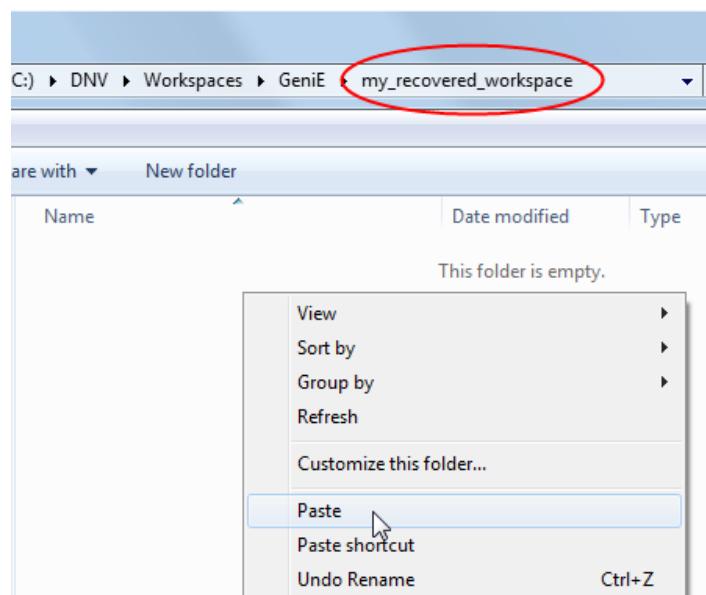
Copy all files inside the folder.



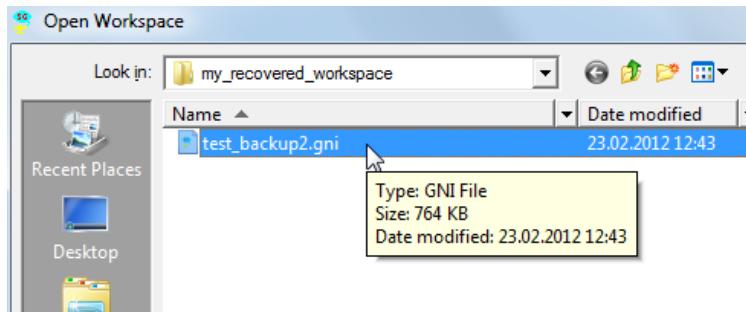


Create a new, empty folder and paste the files into it.

**Important:** It is important that you create a *new* and *empty* folder for recovering your backed up workspace.



Open a new instance of GeniE. If you have several versions of GeniE installed, you should choose the same version you used to create your backup. Select File/Open Workspace and open your backed up workspace.

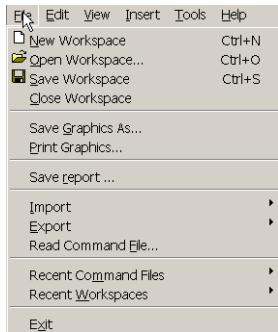


## 5. SHORT DESCRIPTION OF COMMANDS

### 5.1 Menus

#### 5.1.1 Pull-down menus

##### 5.1.1.1 The File pulldown menu



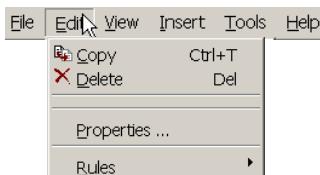
There are six main pull-down menu's, they are described in the following.

A pulldown menu shows the various commands available. The short commands are also listed here.

More information about the commands is available on-line by looking at the lower left corner of the GUI

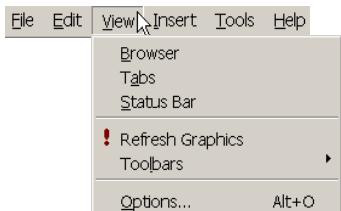
MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
File	New workspace		Make a new project
	Open workspace		Open up existing project
	Save workspace		Save model to database
	Close workspace		Close a workspace
	Save graphics as		Save a graphics file
	Print graphics		Print a graphics file directly on printer
	Save report		Save a print of your model (text, HTML, XML)
	Import	XML Concept Model	Import structural concept model
		FEM file	Import geometry and loads and create concept model
		SACS file	Import geometry and loads and create concept model
		STRUCAD 3D file	Import geometry and loads and create concept model
		ACIS SAT file	Import geometry from SAT and create concept model
		Intergraph PDS (SDNF file)	Import geometry from PDS and create concept model
		CadCentre PDMS (SDNF file)	Import geometry from PDMS and create concept model
		Section library	Import AISC, Euronorm/Norwegian Standard libraries
	Export	XML Concept Model	Export concept model to XML format
		Intergraph PDS (SDNF file)	Export geometry to PDS
		CadCentre PDMS (SDNF file)	Export geometry to PDMS
		GeniE journal file (JS)	Create and export a clean journal file
	Read command file		Read in and execute a journal file (.js file)
	Recent command file		Read in and execute one of 10 last used journal files
	Recent workspaces		Read in and open one of 10 last workspaces
	Exit		Exit program and save workspace

### 5.1.1.2 The Edit pulldown menu



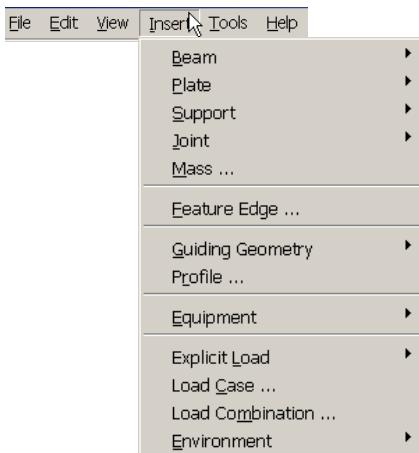
MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
Edit	Copy		Copy a selected object using translate, translate, rotate, or 3 point position
	Delete		Delete a selected object
	Properties		Edit and modify a property library
	Rules	Joint creation	Rules for automatic creation of joints
		Joint design	Rules for calculating can, stub, cone, planewise gap
		Tolerances	Set tolerances when working with tolerant modelling
		Connected move	Rules for connected move
		Units	Specify input units
		Meshing	Set global rules for creation of mesh.
		Sets	Set rules for compact or verbose scripting of sets

### 5.1.1.3 The View pulldown menu



MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
View	Browser		Tick off for browser window
	Tab		Tick off for viewing command line interface window
	Status bar		Show or hide the status bar
	Refresh graphics		Redraws the current graphic view
	Toolbars		Tick off options for toolbars & tooltips
	Options		Access to a number of options controlling the view (Display settings, Category settings, Cursor feedback, Load and result, Presentation)

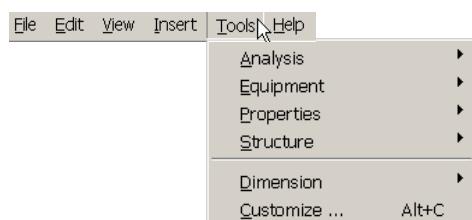
### 5.1.1.4 The Insert pulldown menu



MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
Insert	Beam	<b>Straight Beam Dialog</b>	Insert a beam and specify end points manually
		<b>Straight Beam</b>	Insert a beam graphically by snapping to 2 points
		<b>Curved Beam</b>	Insert a curved beam graphically by snapping to 3 points or more
	Plate	<b>Flat Plate Dialog</b>	Insert a plate and specify corner points manually
		<b>Flat Plate</b>	Insert a plate graphically by snapping to 4 points
		<b>Skin Curves</b>	Insert a curved plate graphically by clicking to 2 curved lines or more
	Support	<b>Support Point Dialog</b>	Insert a support and specify location manually
		<b>Support Point</b>	Insert a support graphically by snapping to 1 points
		<b>Support Curve</b>	Insert support conditions along a line (curved/straight)
	Joint	<b>Joint dialog</b>	Insert a joint by specifying it's coordinates manually
		<b>Joint</b>	Insert a joint by snapping to a point
		<b>Generate Joint</b>	Insert joints according to the joint creating rules
	Mass	<b>Uniform</b>	Insert a point mass
		<b>Generic point mass</b>	Define a directional mass or diagonal mass matrix
	Feature edge		Insert an edge for controlling quality of mesh
	Guiding Geometry	<b>Guide Plane Dialog</b>	Insert a guide plane and specify corner points manually
		<b>Guide-Curve-DIALOG</b>	Insert a guide curve and specify coordinates manually
		<b>Guide-Line-DIALOG</b>	Insert a guide line and specify end points manually
		<b>Guide Point Dialog</b>	Insert a guide point and specify coordinates manually
		<b>Guide Plane</b>	Insert a guide plane graphically by snapping to 4 points
		<b>Guide-Point</b>	Insert a guide point graphically by snapping to 1 point
		<b>Guide Line</b>	Insert a curved guide line graphically by snapping to 2 points or more
		<b>Polycurve</b>	Insert a polycurve graphically by snapping to 3 points or more

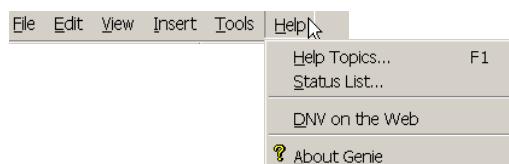
MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
		<b>Guide Spline</b>	Insert a guide spline graphically by snapping to 3 points or more
		<b>Guide Arc Elliptic</b>	Insert a guide arc elliptic graphically by snapping to 3 points (origin, start and end of ellipse)
		<b>Guide Circle</b>	Insert a guide circle graphically by snapping to 3 points (origin, radius, plane)
		<b>Model Curve</b>	Insert a model curve by snapping between two points. Curve will follow topology between
	<b>Profile</b>		Insert a profile to be used for punching or cut operation
	<b>Equipment</b>	<b>Prism shape</b>	Define a prismatic equipment
	<b>Explicit load</b>	<b>Point load</b>	Define a point load
		<b>Line load</b>	Define a constant or linearly varying line load between two points along a beam
		<b>Surface load</b>	Define a surface load (pressure)
		<b>Prescribed Displacement</b>	Specify a prescribed displacement at a support point. Must be used in connection with boundary condition prescribed
		<b>Line Temperature</b>	Define constant or linearly varying temperature intensity between two points along a beam. Temperature constant over cross section.
	<b>Load case</b>		Define a loadcase where equipments, weight lists, and explicit loads are assembled
	<b>Load combination</b>		Define a loadcase build up of other loadcases
	<b>Environment</b>	<b>Location</b>	Please see Vol.2 of this User Manual for details
		<b>Deterministic Time Condition</b>	Please see Vol.2 of this User Manual for details

### 5.1.1.5 The Tools pulldown menu



MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
Tools	Analysis	Create Mesh	Create a finite element mesh and produce a FEM file
		Activity Monitor	A graphic driven menu on running integrated analysis (structural, eigenvalue, hydrodynamic, pile-soil).
		Export FEM	Feature for creating a FEM file with user specified name
		Frame Code Check	Starting the code check program Framework and data transfer of FEM and concept data
		Locate FE	Feature allowing you to locate finite element and node numbers
		Presentation	A graphic driven menu on presenting various types of results, setting attributes, and changing loadcases
		Show Analysis and Results	Feature to import existing analysis results
	Equipment	Import weight list	Read in an existing weight list (xml, csv format)
	Properties	Create scaled materials	Make new materials and connect to beam or plate following a mass scaling operation
	Structure	Geometry Simplify topology	Remove internal plate splits introduced by previous plate/plate plate/beam or plate/feature edge intersections
		Geometry Heal structure	Feature for reconnection of bad models
		Punch	Create holes in the structure using a user-defined profile
		Split	Split selected beams and/or plates to minor parts
		Verify	Tool to verify details of the model
	Dimension	Create Dimension	Find distance between two points and present value
		Angle Between	Find angle between two beams and present value
	Customise		Customise print (graphics) and default name settings

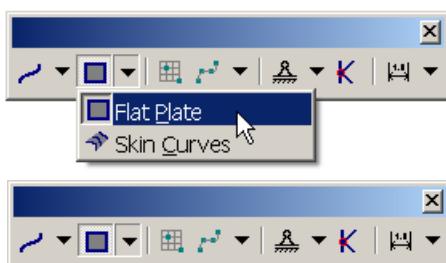
### 5.1.1.6 The Help pulldown menu



MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
Help	Help topics		On-line help available from GeniE
	Status List		Starts the Status List program and looks up GeniE items
	DNV GL on the web		Start-up of DNV Software address on web
	About GeniE		Lists copyrights and 3 <sup>rd</sup> party software used

## 5.1.2 Tool buttons

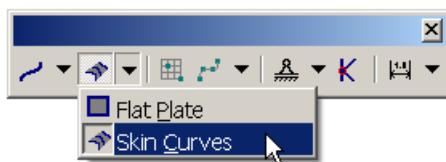
There are a number of tool buttons that give access to the most commonly used commands. They may be dynamic, meaning that the active tool button may give access to different commands. A dynamic tool button is recognised by having a pulldown arrow embedded on the GUI. One example is the tool button for Insert Plate. It can give access to two commands, either Insert a Flat Plate or Insert a Curved Plate using a skin curve operation.



*Flat Plate* option selected



*Flat Plate* is the active operation of the tool button



*Skin Curves* option selected



*Skin Curves* is the active operation of the tool button

The tool buttons are grouped in eight Toolbars described below. All tool buttons have tool tips, these are listed in **bold**.

### 5.1.2.1 The Main toolbar



Tool button	Tool tip	Description
	<b>New Workspace</b>	Create a new workspace
	<b>Open Workspace</b>	Open existing workspace
	<b>Save Workspace</b>	Save current workspace
	<b>Copy with transform</b>	Copy selection using transformation
	<b>Delete</b>	Delete selection
	<b>About</b>	Explains GeniE program version and subcontractors

### 5.1.2.2 The View Manipulation toolbar



Tool button	Tool tip	Description
	<b>Rotate</b>	Rotation in all 3 degrees of freedom (use right mouse button RMB)
	<b>Zoom</b>	Zoom in or zoom out (RMB). Place pointer to decide where to do zoom. Moving mouse upwards or to the right mean zoom-in, while down or to the left means zoom-out Move model to desired position on display (RMB)
	<b>Pan</b>	Move model to desired position on display (RMB)
	<b>Zoom rubberband</b>	Create a rubber band to zoom in (RMB)
	<b>Turtable rotate</b>	The model can be turned around one axis only. The most vertical axis in the view will be selected.
	<b>Fit</b>	Automatic scale view so that whole model is shown on display
	<b>Camera Undo/Redo</b>	Remembers the camera position for each time you stop moving/rotating the model. Use the arrows to step back/forward between the camera positions.
	<b>Spin</b>	Remembers the last rotation and speed of it and makes this a continuous spin
	<b>Refresh graphics</b>	Cleans up graphics and remove all dimensions
	<b>Color code</b>	Toggle on/off your selection of labelling
	<b>Iso view</b>	View from isometric point
	<b>View from X</b>	View in negative X-direction
	<b>View from Y</b>	View in positive Y-direction
	<b>View from Z</b>	View in negative Z-direction
	<b>Outline view</b>	Display beams in outline view (3D view, but no thickness)
	<b>Wireframe view</b>	Display beams in wireframe view
	<b>Solid view</b>	Display beams in solid view (3D view including thickness)
	<b>Default display configuration</b>	Select the current display configuration

### 5.1.2.3 The Loadcase toolbar



Tool button	Tool tip	Description
	<b>Default loadcase selection</b>	Set loadcase to current and displays current loadcase

#### 5.1.2.4 The Label toolbar



Tool button	Tool tip	Description
Name	Label	Labels selected object (Coordinates, Diagram value, Material, Name, Section, Thickness)

#### 5.1.2.5 The Object Types toolbar



Tool button	Tool tip	Description
	Straight beam	Insert a straight beam between two snap points
	Straight overlapping beam	Insert overlapping beams between two snap points
	Pile	Insert a pile between an elevation and snap point along a beam axis
	Curved Beam	Insert a curved beam between three or more snap points
	Flat plate	Insert a plan plate between four or more snap points
	Skin curves	Insert a curved plate between two or more curved lines
	Guide plane	Insert a guide plane between four snap points. The guideplane will be created having 5 equal relative spacings
	Guide point	Insert a guide point by clicking one snap point
	Guide line	Insert a guide line between two snap points
	Guide spline	Insert a guide curve (spline) between three or more snap points
	Poly Curve	Insert a poly curve with three or more snap points
	Guide elliptic arc	Insert a guide ellipse with three snap points (origin, start, end)
	Guide circle	Insert a guide circle with three snap points( origin radius, plane)
	Model curve	Insert a model curve along a topological edge with start and stop points
	Support	Insert a support point
	Support curve	Insert a support curve along beam, model curve, or guide lines
	Joint	Insert a single joint
	Dimension	Find and display the length between two points (1 <sup>st</sup> and 2 <sup>nd</sup> click on points, 3 <sup>rd</sup> click to position the length on the display)
	Angle between	Find and display angle between two beams (1 <sup>st</sup> click on 1 <sup>st</sup> beam, 2 <sup>nd</sup> click on 2 <sup>nd</sup> beam)

### 5.1.2.6 The Create Methods Toolbar



Tool button	Tool tip	Description
	<b>Snap point</b>	Positioning beam, plate, guideplane, support points etc. one by one
	<b>Snap point loop</b>	Positioning beam, plate, guideplane, support points etc. sequentially, e.g. end of beam is start of next beam
	<b>Snap perpendicular</b>	Insert a beam perpendicular to another, 1 <sup>st</sup> click is from start point, 2 <sup>nd</sup> click on perpendicular beam
	<b>Snap tangential</b>	Insert a guideline tangential to another curve, 1 <sup>st</sup> click is from start point, 2 <sup>nd</sup> click on curve
	<b>Snap plane</b>	Temporary snap points are defined at the intersection between beams and a snap plane
	<b>Snap eccentric</b>	Connect a beam to another beam end using its eccentric position.
	<b>Undo snap point</b>	Undo selection of previous snap point
	<b>Clear snap points</b>	Undo selection of all snap points
	<b>Reference point modelling</b>	Specifies journalling of reference point modelling on the journal file

### 5.1.2.7 The Selection Toolbar



Tool button	Tool tip	Description
	<b>Selection</b>	Method for selecting one by one or rubberband (LMB). Together with Shift more advanced selections can be made
	<b>Polygon select</b>	Method for advanced selection where you can make an arbitrary select area (LMB). Make sure that you make a closed envelope.
	<b>Enclosed by rubberband</b>	Objects need to be fully enclosed by rubberband to be part of selection
	<b>Touched by rubberband</b>	Objects need to be touched by rubberband to be part of selection
	<b>Select visible</b>	Visible objects only selected
	<b>Select automatic</b>	Automatic Enclosed or Touched based on how you move the mouse during selection.
	<b>Filter beam</b>	Toggle on/off for selection of beams
	<b>Filter segment</b>	Toggle on/off for selection of segmented members
	<b>Plate selection on/off</b>	Toggle on/off for selection of plates
	<b>Filter side</b>	Toggle on/off for selection of one side of a plate. Used when applying wet surface for panel modelling
	<b>Guide selection on/off</b>	Toggle on/off for selection of guide planes
	<b>Guide curve selection on/off</b>	Toggle on/off for selection of guide curves
	<b>Support selection on/off</b>	Toggle on/off for selection of support (points and curves)
	<b>Joint selection on/off</b>	Toggle on/off for selection of joints
	<b>Equipment selection on/off</b>	Toggle on/off for selection of equipment
	<b>Diagram selection on/off</b>	Toggle on/off for selection of load diagrams
	<b>Environment selection on/off</b>	Toggle on/off for selection of environment
	<b>Feature edge selection on/off</b>	Toggle on/off for selection of feature edge

Note also that these tool buttons may be used to change the view settings, see also Chapter 3.1.3.2.

### 5.1.2.8 The Default Properties toolbar



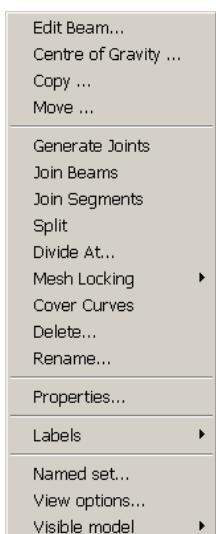
Tool button	Tool tip	Description
<no section> ▾	<b>Default section</b>	Set and display the default section type
<no material> ▾	<b>Default material</b>	Set and display the default material type
<no thickness> ▾	<b>Default thickness</b>	Set and display the default thickness

### 5.1.3 Context sensitive menus

When an object(s) or finite element mesh has been selected (from the graphical window or the browser), it is possible to activate the context sensitive menus by clicking RMB.

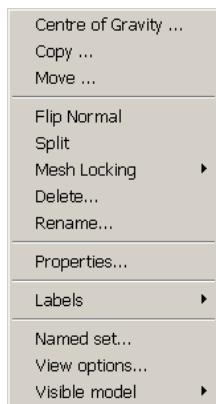
A context sensitive menu means that only relevant command for the selection will appear. The menus contain many of the commands available from the pulldown and toolbar menus, but there are some additional. Among these are the Cover Curves, the Join Beams, the Join Segments, the Select Connected beams (to a Joint), the Joint design features, and the Insert Local Joint Flexibility commands.

#### 5.1.3.1 Object type Beam



Note that Centre of Gravity, Move, Join Beams, Join Segments, Cover Curves, Labels, Named Set, Visible Model are not available from pulldown or from toolbar menus.

### **5.1.3.2 Object type Plate**



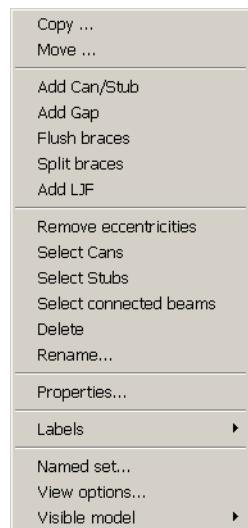
Note that Centre of Gravity, Move, Flip Normal, Labels, Named Set, Visible Model are not available from pulldown or from toolbar menus.

### **5.1.3.3 Object type Equipment**



Note that Place, Place a Copy, Named Set, Visible Model are not available from pulldown or from toolbar menus.

### **5.1.3.4 Object type Joint**



Note that most of these are not available from pulldown or from toolbar

### **5.1.3.5 Object type Support Point**



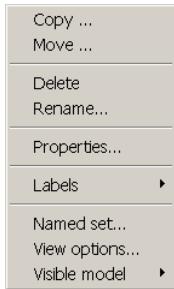
Note that Move, Labels, Named Set, Visible Model are not available from pulldown or toolbar menus.

### **5.1.3.6 Object type Explicit Load**



Note that Move, Named Set, Visible Model are not available from pulldown or toolbar menus

### **5.1.3.7 Object type Guide Plane**



Note that Move, Named Set, Visible Model are not available from pulldown or toolbar menus.

### **5.1.3.8 Object type Guide Line**



Note that Move, Create beam, Create Feature Edge, Create Support Curve, Join Curves, Cover Curves, Labels, Named Set, Visible Model are not available from pulldown or toolbar menus

### **5.1.3.9 Object type Water Surface**



### **5.1.3.10 Object type Water Layer**



### **5.1.3.11 Object type Air Layer**



### **5.1.3.12 Object type Seabed**



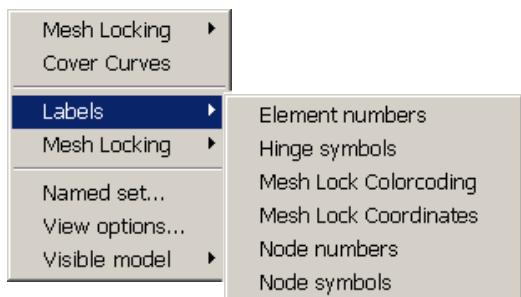
### **5.1.3.13 Object type Soil Layer**



### **5.1.3.14 Object type Soil Border**



### 5.1.3.15 *Selecting mesh*



When selecting a mesh, the most common used operations would be to label details.

## 5.2 Short commands and Windows compliance

GeniE comes with a number of short commands utilising e.g. ALT, CTRL, and keys F1-F10 like in other Windows applications. Below is a table listing the available ones:

SHORT COMMAND	2nd LEVEL PULLDOWN	3rd LEVEL PULLDOWN
CTRL+N	File New Workspace	
CTRL+O	File Open Workspace	
CTRL+S	File Save Workspace	
CTRL+T	Edit Copy	
ALT+O	View Options	
ALT+C	Tools Customize	
ALT+M		Tools Analysis Create mesh
ALT+D		Tools Analysis Activity Monitor
ALT+P		Tools Analysis Presentation
ALT+S		Visible Model>Show selection only
ALT+Plus		Visible Model>Add selection
ALT-Minus		Visible Model Remove selection
ALT+A		Visible Model>Show all
ALT+Q		Visible Model>Show complement
Del	Edit Delete	

KEY	TOGGLE COMMAND
F1	Help
F2	Rotate - rotation in all 3 degrees of freedom
F3	Zoom - zoom in or zoom out
F4	Pan - move model to desired position on display
F5	View Iso - view from isometric point
F6	View from X - view in negative X-direction
F7	View from Y - view in negative Y-direction
F8	View from Z - view in negative Z-direction
F9	Fit screen
F10	Spin - remembers the last rotation and speed of it and makes this a continuous spin
F11	Toggles the snap perpendicular, tangential, plane mode
ALT+B	Toggle background colour between black and white

KEY COMBINATION	EFFECT
S+RMB	Align model with major axis.

## 5.3 The command line interface system

GeniE is primarily intended to be operated by the graphical user interface. All program features may, however, also be accessed by GeniE commands.

The GeniE commands are basically used to create journals during interactive sessions. The journal file can now be used to re-create the model (you may also edit and change the journal file). The GeniE commands may also be used to write a model input file directly or to invoke features that have no graphical interface.

There are 3 ways of entering commands into GeniE,

1. by typing or pasting commands into the Command Line tab in GeniE
2. by the “Read command file” option in the File menu of GeniE
3. by starting GeniE with an input command file from the command prompt (DOS window)  
E.g. “C:\Program Files\DNVGL\GeniE\Program\GeniEr” MyProject /NEW  
/COM=MY\_JOURNAL.JS /EXIT

Please note that if you are using another editor than e.g. MS Notepad, you need to specify that the output format is for PC format and not Unix format.

GeniE supports two kinds of commands

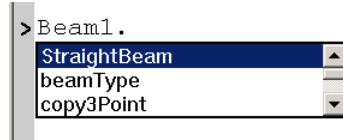
- Specific GeniE commands
- General JScript commands

Specific GeniE commands are typically for creating and editing GeniE model entities such as plates and beams. A simple session of GeniE commands may be:

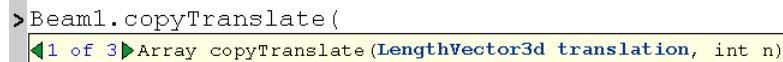
```
// Create 2 points
Point1 = Point(0,0,0);
Point2 = Point(0,0,10);
// A double slash indicates a comment line
// Copy the two points 10 m in x-direction
Point3 = Point1.copyTranslate(Vector3d(10 m,0 m,0 m));
Point4 = Point2.copyTranslate(Vector3d(10 m,0 m,0 m));
// Note that all command must end with ();
// Create a beam between point 3 and 4
BeamA1 = Beam(Point3, Point4);
BeamA1.material = Material1;
BeamA1.section = Section1;
//
// Create a support at Point3, rotate the local Z-axis 30 deg. and
define the boundary conditions
Sp1 = SupportPoint(Point(10 m,0 m,0 m));
Sp1.rotateLocalZ(30);
Sp1.boundary = BoundaryCondition(Free, Fixed, Fixed, Free, Fixed,
Free);
```

Note that the objects in the GeniE model have properties and functions that are addressed by a dot like in “BeamA1.material” or “Point1.copyTranslate”.

When using the GeniE Command Line tab you will get a list of relevant properties and functions by pressing the Tab key while typing the command;



The command window will also provide a tool tip to assist you with the command syntax;



GeniE also supports the general programming language JScript. By combining GeniE commands with JScript you may use programming features in your model input file.

Typically you may start by defining all basic model data as variables in the beginning of the file and then refer to these variables when creating the model.

```
// Coordinate arrays
var X1 = new Array();
var Y1 = new Array();
var Z1 = new Array();
// X1-values
X1[0] = -23;
X1[1] = 12;
X1[2] = 34;
// Y1-values
Y1[0] = -45;
Y1[1] = -20;
Y1[2] = -5;
// Z1 elevations
Z1[0] = -128.987;
Z1[1] = -124.987;
//
Point_A = Point(X1[0], Y1[0], Z1[0]);
```

Further you may use arithmetic expressions within the GeniE commands or to do separate calculations;

```
TopElevation = Z1[1] + 2.45 ;
Point_B = Point( 2.54 , 0.0, (TopElevation + 10)) ;
```

Please note that most mathematical functions are addressed through the Math object in JScript.

```
Radius = 5.0;
MyArea = Math.PI() *Math.pow(Radius, 2);
Print(MyArea);
78.539816
```

For more sophisticated modelling you may use For loops and If testes to program the creation of your model. You may also invoke other applications that support Automation (e.g. Excel) to exchange data with your GeniE model. Automation is a technology that allows software packages to expose their features to scripting tools and other applications.

Example of creating beams in a loop:

```
//Adding beams to a leg
for (k = 0; k < 5 ; k++)
{
    var Bms1 = new Array();
    Bms1[k] = Beam(LegPoint[k], LegPoint[k+1]);
    Bms1[k].name = "LegBeam" + k ;
    Bms1[k].section = LegSection;
    Bms1[k].material = LegMaterial;
}
```

### 5.3.1 The GeniE JScript Command Reference

The GeniE commands are described in the JScript Command Reference found in the GeniE help menu. The most feasible approach to writing a command file is rather to create a sample model interactively and then use the generated journal as a template. When you have found the kind of object you are working with you may use the Command Reference to get a full list of relevant features and the command syntax for accessing these. The following pictures show how you find information about the script command for generating a basic beam and which other commands are associated this operation.

Section	Commands	Descriptions
Analysis	AccelerationVector3d	Represent a vector of Acceleration in 3 dimensions
	Analysis	
	LinearAnalysis	Define a Linear Analysis
	LoadCase	LoadCase
	LoadCombination	LoadCombination
	LoadInterface	
	LoadResultsActivity	
	MeshActivity	
	PileSoilAnalysis	Define a Pile Soil Analysis
	PrescribedDisplacement	PrescribedDisplacement
Environment	SeastateData	Define a Seastate Data object
	SinFile	
	WROPrintFileDeterministic	
	WROPrintFileTransfer	
	WaveLoadActivity	Define a Wave Load Analysis
	Airy	
	Clay	Represent a soil of type clay.
	Cnidial	
	CurrentProfileRelDir	Represent a current profile
	DeterministicTime	
Direction	Represent an environment direction	
DirectionSet	Represent an environment direction set	
FrequencySet	Represent an environment frequency set	
Location		
PhaseSet	Represent an environment phase angle set	
RegularWave		
RegularWaveModel	Represent a regular wave component	
RegularWaveSet	Represent a regular wave set of one of the types; WavePeriod, WaveFrequency or WaveLength	
Sand	Represent a soil of type sand.	

The documentation is organised in classes:

- Analysis
- Environment
- Exchange
- Guiding geometry
- Loads
- Properties
- Sets
- Structure
- Units
- Other

<b>Structure</b>	
<a href="#">Beam</a>	Construct a new straight or curved beam from the given guide curve
<a href="#">Beam</a>	Construct a new straight or curved beam from the given points
<a href="#">Beam</a>	Create a straight Beam
<a href="#">ConstantLocalSystem</a>	Define a constant local system for the curve
<a href="#">CriterionInPlane</a>	
<a href="#">CurvedBeam</a>	Curved Beam
<a href="#">FeatureEdge</a>	Feature Edge
<a href="#">GuideLocalSystem</a>	Let the curves local X follow the direction of the curve
<a href="#">Joint</a>	Represent a tubular joint
<a href="#">NormalToCurvePlane</a>	If a curve lies in a plane, use the normal as the curve local Z
<a href="#">Pile</a>	Pile
<a href="#">Plate</a>	Flat Plate
<a href="#">PointMass</a>	PointMass
<a href="#">RelativeToPlate</a>	The local Z for the curve is locked to the normal of a shell
<a href="#">Shell</a>	Curved Shell
<a href="#">SimplifyTopology</a>	Simplifies the structure by removing unnecessary topology.
<a href="#">SplitAtPoint</a>	Split structure at the given point
<a href="#">SplitStructure</a>	Explode selected structure at all structural connection points
<a href="#">StraightBeam</a>	Straight Beam
<a href="#">SupportCurve</a>	Represent a Support Curve
<a href="#">SupportPoint</a>	Represent a Support Point

Looking up straight beam found under the class Structure.

<b>Function Detail</b>	
<b>Beam</b>	
<a href="#">BasicBeam Beam(<a href="#">Point</a> p1, <a href="#">Point</a> p2, <a href="#">OverlapPolicy</a> overlapPolicy)</a>	
Create a straight Beam	
<b>Parameters:</b>	
p1 - End 1 of the Beam	
p2 - End 2 of the Beam	
overlapPolicy - Specifies how this beam is to be inserted into the model	
<b>Example:</b>	
	//Create a beam that is allowed to overlap existing beams: b1 = Beam(Point(0,0,0),Point(1,0,0),geAllowOverlap); //Create a beam that will remove portions of existing beams in order to make room for itself: b2 = Beam(b1.end1),Point(2,2,0),geEnforceThis);

The *Function Detail* now documents the command itself and practical examples.

<a href="#">Overview</a>	<a href="#">Class Hierarchy</a>	Genie D3.0-10 25-Oct-2004
<h2>BasicBeam</h2>		
Class hierarchy:		
<pre> ModelObject       +--NamedObject           +--Transformable               +--BasicConcept                   +--BasicBeam   </pre>		
Direct Known Subclasses:		
<a href="#">CurvedBeam</a> , <a href="#">Pile</a> , <a href="#">StraightBeam</a>		
<b>Function Summary</b>		
<a href="#">BasicBeam</a>	<a href="#">divideAt(double parameter)</a>	Divide beam at parameter value and return the second half of the beam
	<a href="#">divideSegmentAt(int iseg, double parameter)</a>	Divide beam segment at parameter value
	<a href="#">explode(<a href="#">NameMask</a> nameMask)</a>	Split structure into smaller parts
<a href="#">bool</a>	<a href="#">extendEnd(<a href="#">Long</a> iend, <a href="#">Length</a> extension)</a>	Extend or shorten beam at end 1 or 2 along direction of beam
<a href="#">Material</a>	<a href="#">getSegmentMaterial(int)</a>	
<a href="#">Section</a>	<a href="#">getSegmentSection(int)</a>	
<a href="#">Point</a>	<a href="#">intersect(<a href="#">Plane3d</a> plane)</a>	Find the intersection point, if between the beam and a plane

You may look at further details belonging to the BasicBeam. When scrolling down the Function Summary the additional commands (ore features) are listed.

Function Summary	
<a href="#">ModelObject</a>	<a href="#">copy3Point</a> ( <a href="#">Point</a> sp1, <a href="#">Point</a> sp2, <a href="#">Point</a> sp3, <a href="#">Point</a> dp1, <a href="#">Point</a> dp2, <a href="#">Point</a> dp3) Copy the object without scaling from one location to another.
<a href="#">ModelObject</a>	<a href="#">copyMirror</a> ( <a href="#">Point</a> p1, <a href="#">LengthVector3d</a> v1) Mirror a copy of the object around p1 and v1
<a href="#">ModelObject</a>	<a href="#">copyRotate</a> ( <a href="#">Point</a> p1, <a href="#">LengthVector3d</a> p2, <a href="#">Angle</a> angle) Rotate a copy of the object around p1 and v1 the angle angle
<a href="#">Array</a>	<a href="#">copyRotate</a> ( <a href="#">Point</a> p1, <a href="#">LengthVector3d</a> p2, <a href="#">Angle</a> angle, <a href="#">int</a> n) Make n copies of the object,incrementing the angle for each copy
	<a href="#">copyRotate</a> ( <a href="#">Point</a> p1, <a href="#">LengthVector3d</a> p2, <a href="#">Angle</a> angle, <a href="#">int</a> n, <a href="#">NameMask</a> nameMask) Make n copies of the object,incrementing the angle for each copy

Scroll down to  
[copyRotate](#) to find out  
how this command works

By clicking the link “copyRotate” you will get a detailed description of this function as follows:

[ModelObject](#) [copyRotate](#)([Point](#) p1, [LengthVector3d](#) p2, [Angle](#) angle)

Rotate a copy of the object around p1 and v1 the angle angle

Description:

Make one copy of the object. The transformation is defined as counterclockwise rotation around the given axis vector at the anchor point.

Parameters:

p1 - Point on rotation axis

p2 - Rotation axis vector

angle - Rotation angle

Returns:

the copied object

Example:

```
//Rotate Bm1 45 degrees around Bm1.end1 and the axis (0,0,1):
Bm2=Bm1.copyRotate(Bm1.end1,Vector3d(0,0,1),45deg);
```

The description shows the syntax of the command with the type and name of each parameter. By clicking the link for the type like [LengthVector3d](#) you will get all valid forms for entering a vector in GeniE.

More about general JScript commands may be found e.g. at <http://msdn.microsoft.com/scripting>.

### 5.3.2 Useful script commands

GeniE creates a journal file including all the operations you perform except a few – typically related to making a finite element model (when not part of analysis activity), export the FE model or graphical interactions. These commands may be executed directly from the Command Line Interface or when importing a journal file containing the commands. You may find some of the following script commands useful.

Command	Description
CreateMesh();	Force the creation of a mesh (same as <b>Alt+D</b> or <b>Tools/Analysis/Create Mesh</b> )
ExportMeshFem().DoExport(name);	Export a finite element model to the default working directory (same as <b>Tools/Analysis/Export FEM File</b> ), e.g. ExportMeshFem().DoExport("T1.FEM");
GenieRules.Meshing.preference(mpUseDrillingElements,true);	Activation of 3 and 4 noded drilling elements (FTAS and FQAS)
Graphics.move(vector);	Move the view of the model along the given vector in the global system, e.g. Graphics.move(Vector3D(1,1,0));
Graphics.pan(x,y);	Pan a model. Same as the graphical operation 'Pan (F4)', e.g. Graphics.pan(1,10); will move the model 1 pixel to the right and 10 pixels up. Note that x and y are measured in pixels.
Graphics.rotate(rotationAxis,angle);	Rotate the view of the model a given angle around the centre of the model with a rotation axis given in the global system, e.g. Graphics.rotate(Vector3d(0,0,1),45); The model is now rotated 45degrees around global z-axis.
Graphics.rotate(rotationCenter,rotationAxis,angle);	Rotate the view of the model a given angle around rotationCenter with a rotation axis given in the global system, e.g. Graphics.rotate(Point(2,5,3),Vector3d(0,0,1),45); The model is now rotated 45degrees around global z-axis around the point with coordinate values (2,5,3)
Graphics.rotationCenter;	Return the models centre of rotation, i.e. when you rotate the model graphically (using the default rotation scheme, the model is rotated around the centre of the model). Graphics.rotationCenter);: will return this point.
Graphics.saveImage(name);	Save the image in a given format (given by the filename), e.g. Graphics.saveImage("Picture.jpg");

Command	Description
<code>Graphics.saveImage(name,width,height);</code>	Save the image in a given format, scaled to the size in pixels given by width and height, e.g. <code>Graphics.saveImage("Picture.jpg",2000,2000);</code>
<code>Graphics.zoomArea(left,bottom,right,top);</code>	Rubberband zoom. Same as the graphical operation 'Zoom Rubberband'. The input (left,bottom,right,top) are given in pixels. The bottom left pixelcoordinate of the 3d-view is 0,0. The top right pixelcoordinate of the 3d-view is dependent on your screen size. (You can find this value on your computer by File Save Graphics As... <Save> Under Pixels Width Pixels Height Pixels Width is the rightmost pixel coordinate of the screen. Pixel Height is the top coordinate of the screen. If the width and height of the screen are 600,600, the size of our 3d viewport is (0,0,600,600). If you want to zoom in on the middle portion of the screen, you may use <i>Graphics.zoomArea(150,150,450,450);</i> . If you want to zoom out, you may use <i>Graphics.zoomArea(-150,-150,750,750);</i> .
<code>Graphics.setEye(eyePos);</code>	The command gives a view of the model focused on its origin,e.g. <code>Graphics.setEyePosition(Point(-30,30,5),Vector3d(1,1,0));</code> . To be used to recreate a specific view.
<code>Graphics.viewISO();</code>	Same as the graphical operation 'View ISO (F5)' Note, you need to refresh graphics to yield immediate screen update
<code>Graphics.viewFromX();</code>	Same as the graphical operation 'View from X (F6)' Note, you need to refresh graphics to yield immediate screen update
<code>Graphics.viewFromY();</code>	Same as the graphical operation 'View from Y (F7)' Note, you need to refresh graphics to yield immediate screen update
<code>Graphics.viewFromZ();</code>	Same as the graphical operation 'View from Z (F8)' Note, you need to refresh graphics to yield immediate screen update
<code>Graphics.fitModel();</code>	Same as the graphical operation 'Fit (F9)' Note, you need to refresh graphics to yield immediate screen update

Command	Description
<platename_A>.join(<platename_B>);	<p>Command for joining plate B into plate A. The new plate keeps name plate A. Example: Joining Pl124 with Pl121 has the command Pl121.join(Pl124);</p>
Math.timer(long t0);	<p>Useful feature that will return elapsed seconds from t0. You need to specify start time (t0), when to measure elapsed time and print to the journal file window. Example returning elapsed time at t1 and t2 (to be edited in the journal file):</p> <pre>t0=Math.timer(0); t1=Math.timer(t0); print(t1);     -&gt; returns time since t0 in seconds t2=Math.timer(t0); print(t2);     -&gt; returns time since t0 in seconds</pre>

By using these commands together with commands automatically created during GeniE sessions it is possible to run GeniE from batch mode to do among others

- Create structure
- Apply loads
- Model environment (wave, current, air, soil)
- Perform analysis – structural, wave and pile-soil (or export FEM model for later usage in e.g. a superelement analysis)
- Specify view settings, create and save graphics images

## **6. APPENDIX A: REFERENCES**

1. Sestra User Manual
2. Framework User Manual
3. Wajac User Manual
4. Splice User Manual
5. Buitrago et. al. (1993)
6. HydroD User Manual
7. DeepC User Manual
8. SESAM Manager User Manual
9. Presel User Manual

## 7. APPENDIX B: CONSISTENT UNITS

This Appendix shows some typical consistent units.

### 7.1 Consistent SI units

Length Unit <i>L</i>	Mass Unit <i>M</i>	Force Unit <i>ML/T<sup>2</sup></i>	Typical program input values		
			Yield stress for steel <i>M/(LT<sup>2</sup>)</i>	Density of steel (Mass/Volume) <i>M/L<sup>3</sup></i>	Young's modulus for steel (Force/Area) <i>M/(LT<sup>2</sup>)</i>
m	kg	1 N	4.2·10 <sup>8</sup>	7.85·10 <sup>3</sup>	2.10·10 <sup>11</sup>
m	10 <sup>3</sup> kg = 1 t	10 <sup>3</sup> N = 1 KN	4.2·10 <sup>5</sup>	7.85	2.10·10 <sup>8</sup>
cm	kg	10 <sup>-2</sup> N	4.2·10 <sup>6</sup>	7.85·10 <sup>-3</sup>	2.10·10 <sup>9</sup>
cm	10 <sup>3</sup> kg = 1 t	1 kgf ≈ 10 N	4.2·10 <sup>3</sup>	7.85·10 <sup>-6</sup>	2.10·10 <sup>6</sup>
mm	kg	10 <sup>-3</sup> N	4.2·10 <sup>5</sup>	7.85·10 <sup>-6</sup>	2.10·10 <sup>8</sup>
mm	10 <sup>3</sup> kg = 1 t	1 N	4.2·10 <sup>2</sup>	7.85·10 <sup>-9</sup>	2.10·10 <sup>5</sup>
cm	10 <sup>2</sup> kg	1 N	4.2·10 <sup>4</sup>	7.85·10 <sup>-5</sup>	2.10·10 <sup>7</sup>
m	10 <sup>4</sup> kg	1 tonnef ≈ 10000 N	4.2·10 <sup>4</sup>	7.85·10 <sup>-1</sup>	2.10·10 <sup>7</sup>
cm	10 <sup>6</sup> kg	1 tonnef ≈ 10000 N	4.2	7.85·10 <sup>-9</sup>	2.10·10 <sup>3</sup>
mm	10 <sup>7</sup> kg	1 tonnef ≈ 10000 N	4.2·10 <sup>-3</sup>	7.85·10 <sup>-13</sup>	2.10
m	10 kg	1 kgf ≈ 10 N	4.2·10 <sup>7</sup>	7.85·10 <sup>2</sup>	2.10·10 <sup>10</sup>
cm	10 <sup>3</sup> kg	1 kgf ≈ 10 N	4.2·10 <sup>3</sup>	7.85·10 <sup>-6</sup>	2.10·10 <sup>6</sup>
mm	10 <sup>4</sup> kg	1 kgf ≈ 10 N	4.2·10 <sup>1</sup>	7.85·10 <sup>-10</sup>	2.10·10 <sup>4</sup>

### 7.2 Consistent imperial units

Length Unit <i>L</i>	Mass Unit <i>M</i>	Force Unit <i>ML/T<sup>2</sup></i>	Typical program input values		
			Yield stress for steel <i>M/(LT<sup>2</sup>)</i>	Density of steel (Mass/Volume) <i>M/L<sup>3</sup></i>	Young's modulus for steel (Force/Area) <i>M/(LT<sup>2</sup>)</i>
Foot	1 lb	Poundal		491	1.39·10 <sup>11</sup>
Inch	12 lbs	Poundal		2.37·10 <sup>-2</sup>	9.66·10 <sup>8</sup>
Foot	32.2 lbs (1 slug)	Pound(f)		15.2	4.32·10 <sup>9</sup>
Inch	386 lbs	Pound(f)		7.35·10 <sup>-4</sup>	3.0·10 <sup>7</sup>
Foot	3.22·10 <sup>4</sup> lbs	Kip		1.52·10 <sup>-2</sup>	4.32·10 <sup>6</sup>
Inch	3.86·10 <sup>5</sup> lbs	Kip		7.35·10 <sup>-7</sup>	3.0·10 <sup>4</sup>
Foot	7.21·10 <sup>4</sup> lbs	Ton(f)		6.81·10 <sup>-3</sup>	1.93·10 <sup>6</sup>
Inch	8.66·10 <sup>5</sup> lbs	Ton(f)		3.28·10 <sup>-7</sup>	1.34·10 <sup>4</sup>

----- O -----