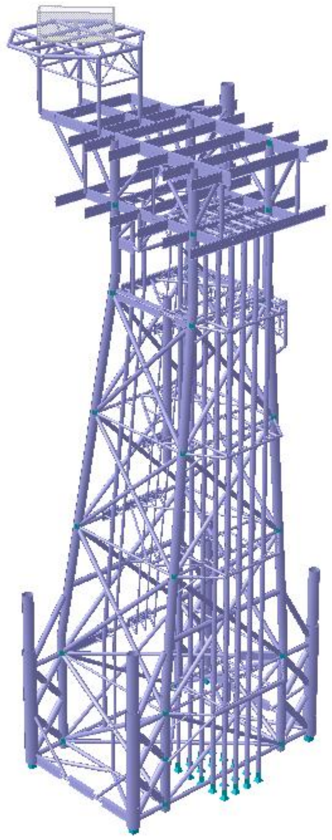
DNV

SESAM
USER MANUAL

GeniE Vol. 1



Concept design and analysis
of
offshore structures

DET NORSKE VERITAS

SESAM
User Manual
GeniE Vol. 1

Concept design and analysis
of offshore structures

29 August 2011

Valid from program version 6.0

Developed and Marketed by
DET NORSKE VERITAS

DNV Software Report No.: 00-000 / Revision 0, 29 August 2011

Published by:

Det Norske Veritas Software
Veritasveien 1
N-1322 HØVIK
Norway

Telephone: +47 67 57 99 00

Facsimile: +47 67 57 72 72

E-mail, sales: software.sesam@dnv.com

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

Website: www.dnvsoftware.com

GeniE User Manual
Version 6.0
Table of Contents

1. INTRODUCTION

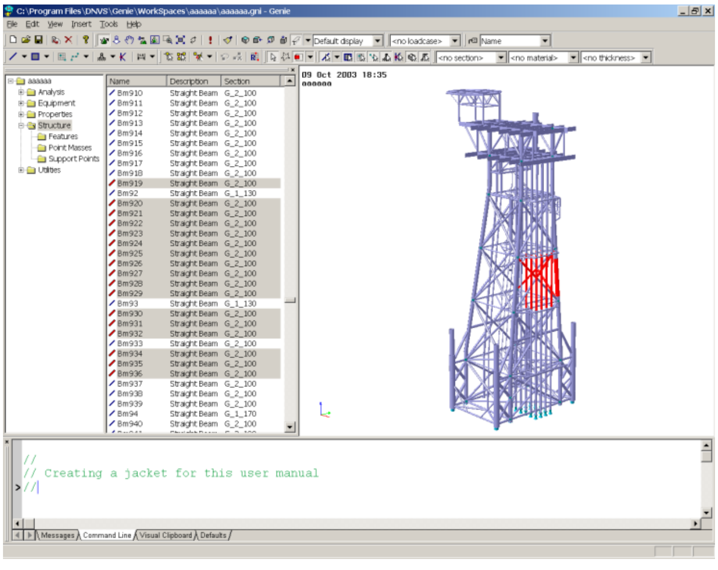
This User Manual aims at guiding new users of GeniE. Emphasis has also been put on advanced features for the more experienced users by separating commands from the textual & graphic description of features.

The User Manual is valid for GeniE version 3.2 – both for the full version of GeniE and the GeniE.lite version. The User Manual is split in two volumes, of which Vol. 2 documents hydrodynamic and pile-soil modelling and integrated analysis. 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

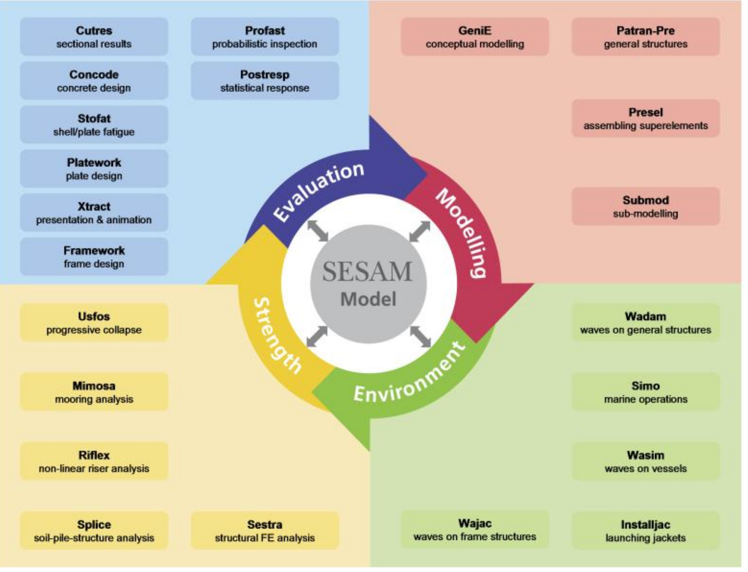
GeniE is also ideal for creating superelements (parts) in a superelement analysis. Typically, topsides and modules are created in GeniE and assembled with other parts of the structure like the hull of a FPSO, Semi-submersible, TLP, Spar or similar structure.

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 Software, an independent business unit of Det Norske Veritas 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

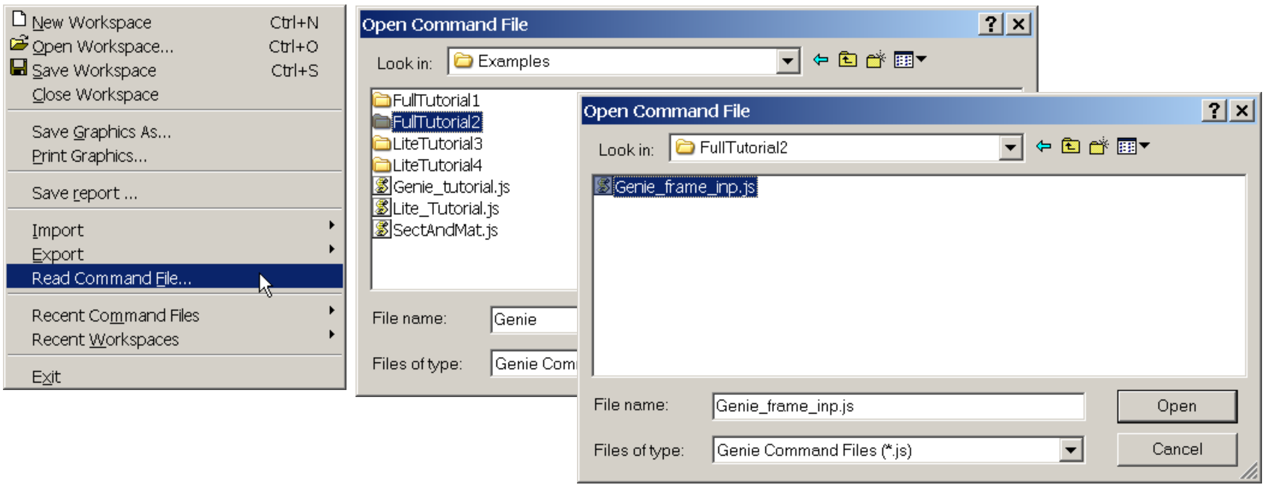
1.4 On-line help system and tutorials

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.

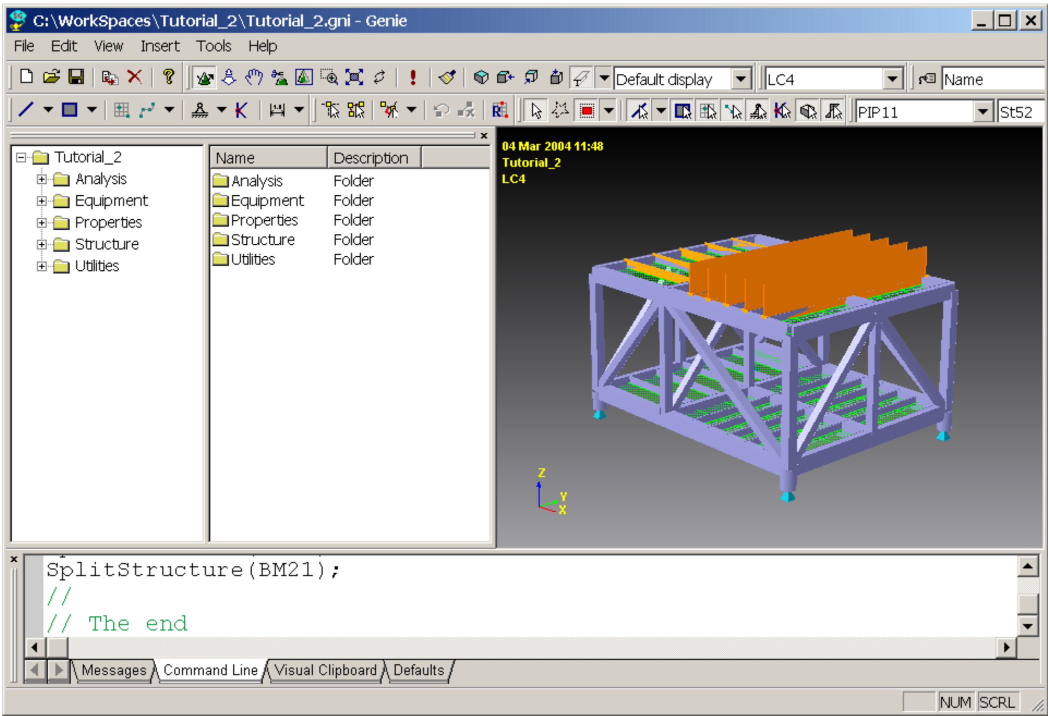
The most efficient way to work with the tutorials is to make a print-out of the tutorials, start GeniE, create a new workspace (command ***File|New workspace***) and follow the steps in the tutorial. Each tutorial comes with a pre-defined journal file (command file) – you find these from your catalogue C:\Program Files\DNVS\GeniE\Examples. 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 tutorials 1 – 4.

In the example below the <name> has been set to *Tutorial_2* and the imported file is for Tutorial no. 2 “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

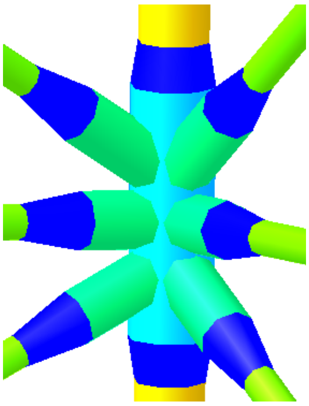
Acronym	Explanation
RMB	Right Mouse Button
LMB	Left Mouse Button
GUI	Graphical User Interface
DOF	Degree Of Freedom
CLI	Command Line Interface
LJF	Local Joint Flexibility
FEM file	SESAM Input Interface File

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

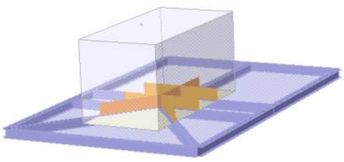
The visions of the program GeniE were established in the Joint Industry Project *SESAM 2000* (1994 -2000). The program itself has been developed by DNV Software following this project.



Built-in knowledge for joint design

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.



Built-in knowledge when deriving equipment loads

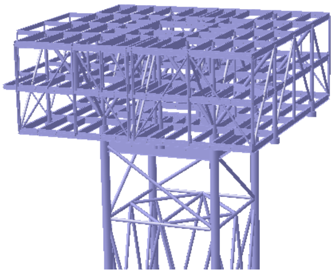
Focus has also been on continued usage of existing data or information. Typical examples are how data from a CAD system or other FE systems can be imported, automatic translated to a concept model and enriched with necessary data to do a design analysis. Equally important is the need for data transfer 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:

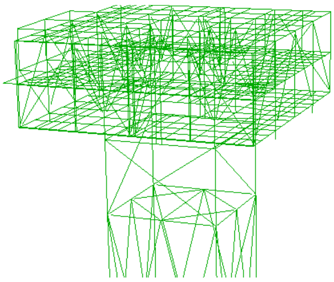
- Model a structure using libraries for sections, plates, material and boundary conditions. The modelling itself may be done directly, using guiding geometry or by importing an existing model.
- Make loadcases where you assemble equipments, weight lists or explicit point/line/surface/temperature loads.
- 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.

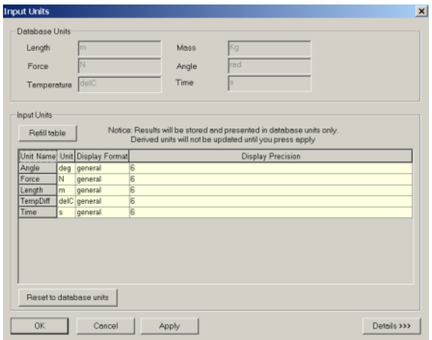
1. Decide the design premise – typically units, section libraries, mesh rules, gap calculations, tolerances and name settings.
2. Model the structural part of model – either from importing from other systems or other SESAM programs, using wizards or modelling from scratch. This part is normally the most 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

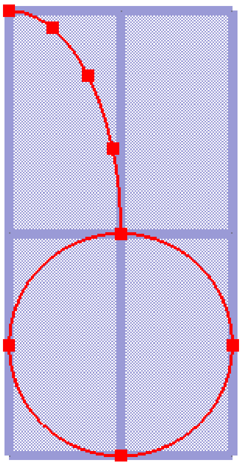
2.5 Use of guiding geometry

Guiding geometry has been introduced to allow for rapid modelling of structure. When modelling the structure snap-to-grid may be used. This helps you focus on modelling itself and less on calculating exact co-ordinates when e.g. modelling a beam or a plate. This modelling technique may be compared with snap-to-grid in e.g. MS PowerPoint.

Guiding geometry is simple to use. The main purpose is to guide you where to place a beam or a plate. One example of a guiding geometry is to a topside’s system lines – you may then easily insert the beams or plates.

When modelling curved plates you need to use guiding geometry to instruct the program how to determine the outer borders of a curved plate – that is, which boundaries to perform a skinning operation between.

There are several guiding geometries:

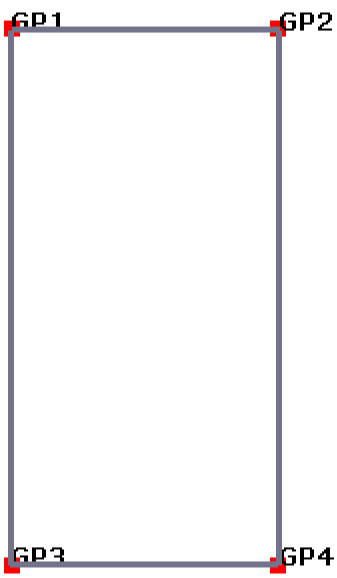


Elliptic and circle guidelines

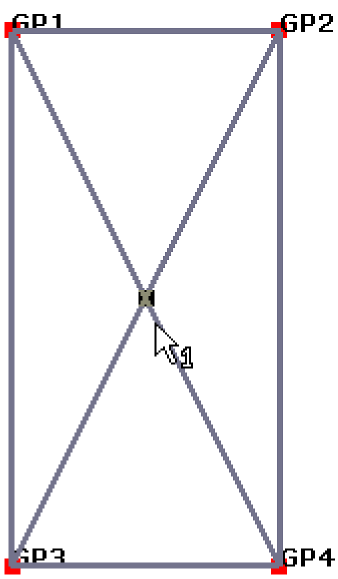
- Single guide point and guide line (along a straight line)
- Guide plane (in one plane).
- Guide spline (spline defined by many points).
- Guide arc elliptic (ellipse defined by origin and two end points on ellipse – the three points also define the plane).
- Guide circle (three points defining origin, radius and the plane)
- Polycurves (continuous line built up from various types of lines)

A topology point is automatically calculated when objects like beams or plates intersect each other. A topology point may be used when modelling new objects – you may also define the topology point as a geometry guide point if you later want to explicitly refer to the point. This is, however, not strictly necessary since the topology point may be found graphically. The difference between a topology point and a geometry guide point is thus how you may refer to it.

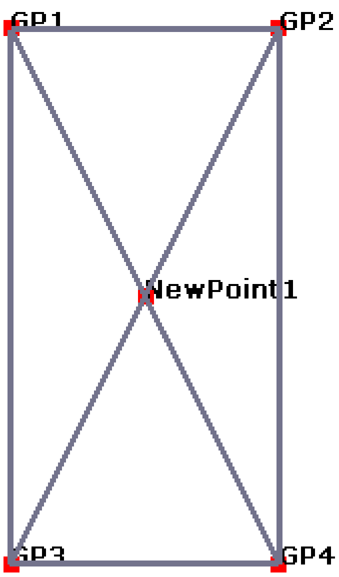
The figures below show how a topology point is created by intersecting two beams – the new topology point is defined as a guide point ready to be referred to when modelling a new object.



Beams created from 4 existing guide points



New topology point automatically created at intersection

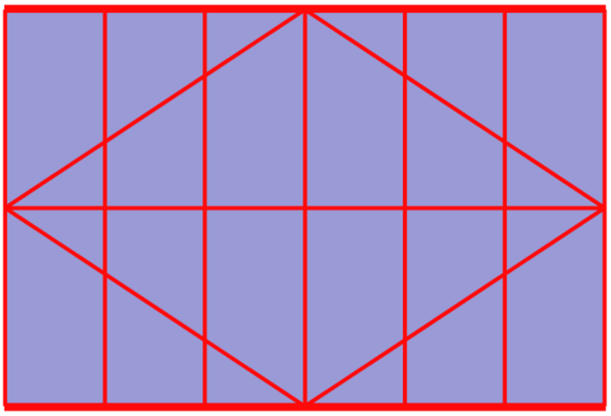


New guide point (NewPoint1) defined at topology point

One advantage of topology points is that you can start a modelling task with a minimum of guiding geometry and continue building using the new topology points automatically established.

2.6 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.

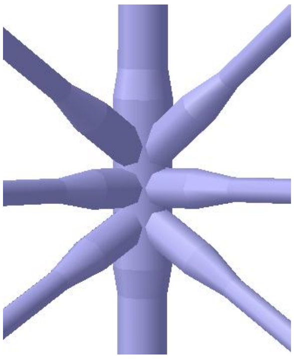
You may of course decide to split the beam into smaller beams. The selected beams to the left (highlighted in read colour) are examples of continuous beams intersected by several beams.

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.

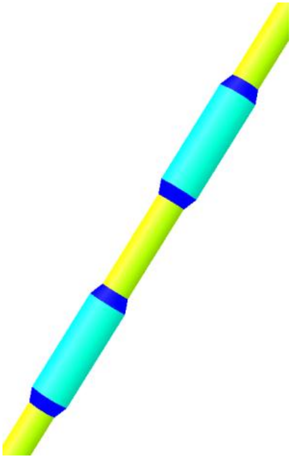
2.7 Advanced structure and panel modelling

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.

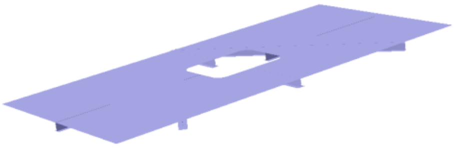
The figures below illustrate a number of examples where GeniE's features for advanced modelling have been used. GeniE is also capable of handling double-curved plates and connected beams.



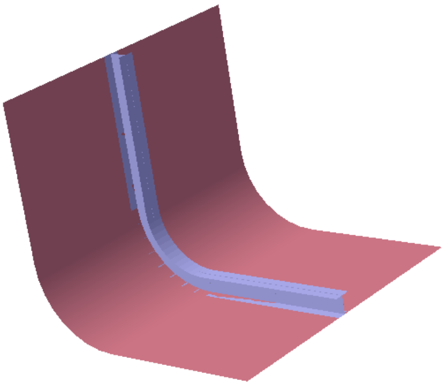
Tubular joint including cans, stubs, cones and planewise gaps



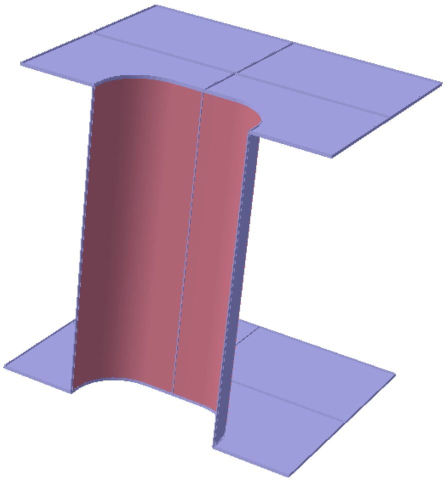
A segmented beam consisting of pipes and cones



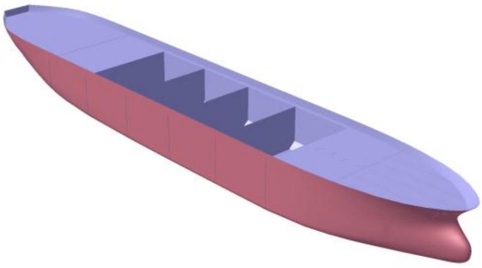
A stiffened panel punched by a 2D-profile



A single curved plat containing one flushed beam.



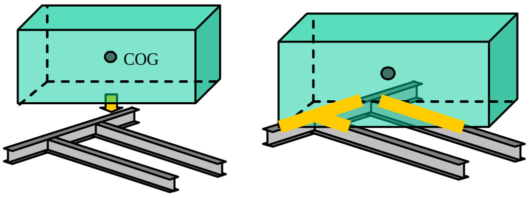
Two plates connected by a half cylinder.



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

2.8 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).

2.8.1 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*.

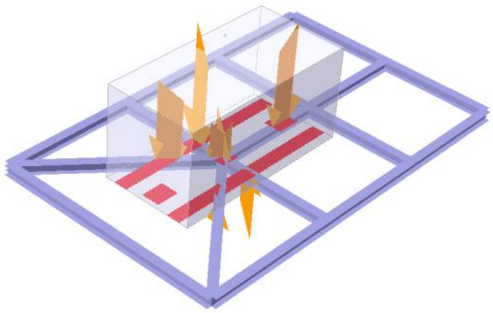
GeniE will always calculate a correct equipment mass and load accounting for

- 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

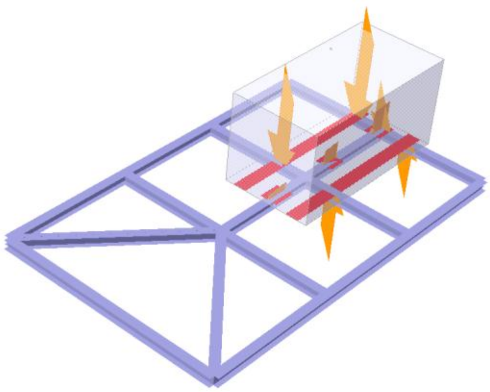
Once the load carrying interface is established the load generated by the equipment is automatically calculated. First, the total equipment force is calculated as $F_{equipment} = Mass_{equipment} * acceleration\ field$. This force is then distributed over the load carrying interface while ensuring force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



Line loads calculated at interface between supporting beams and equipment footprint



Equipment moved to new position and new line loads are automatically calculated and applied to model

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.

2.8.2 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.

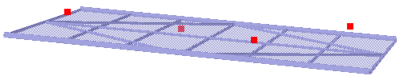
Equipments in a weight list may not always have information about its size, you can therefore decide a bounding box so that GeniE will know which structural members will receive load (where the box intersects with structure).

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).

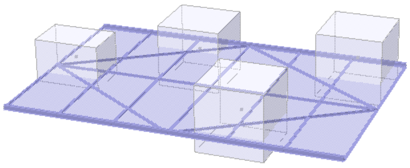
The loads or masses produced from a weight list may be used in many loadcases in the same way as other equipment. They can, however, not be used in different positions since their locations are specified in the weight list.

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.

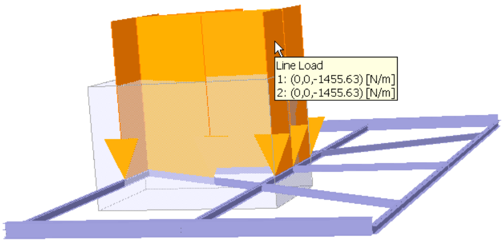
The figures below show how an imported weight list read and converted to line loads. The first step is to import the data, the next step (automatic) is loading a loadcase and the final step is automatic calculation of constant line loads.



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

2.8.3 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 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.

2.8.4 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.

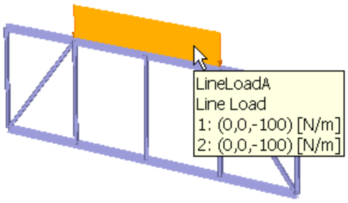
GeniE is flexible in how the loads are applied. For example, a point load does not have to be located at a previously defined point or at beam ends. Similarly, a line load may be applied to a part of the length of a beam. For surface loads the pressure may be applied to parts of a plate if desired. It 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.

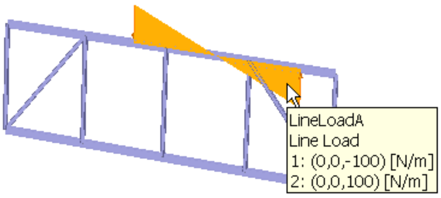
Point and line loads apply to beams while surface loads apply to plates. In other words, if you apply a line load to a plate no loads will be calculated for use in the analysis.

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

2.8.5 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.

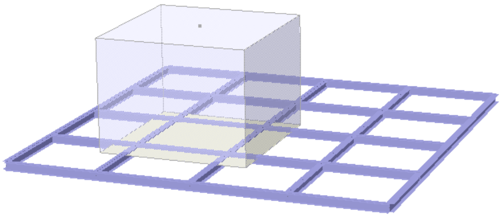
2.9 Prepare for structural analysis

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.

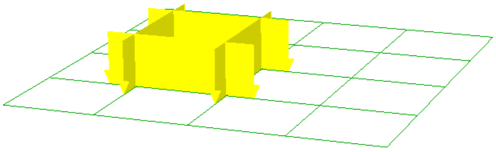
A model created in GeniE can be used for several purposes (see next page for illustrations):

- 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.

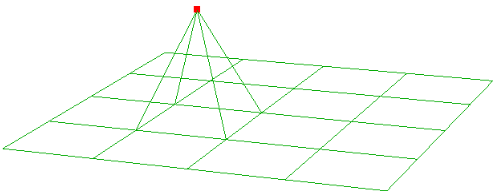
A common approach when making an analytical representation of a real structure is to neglect structural parts and coating that are not relevant for the stiffness of the structure. However, these have importance regarding the total mass of the structure. To overcome this limitation, it is possible to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



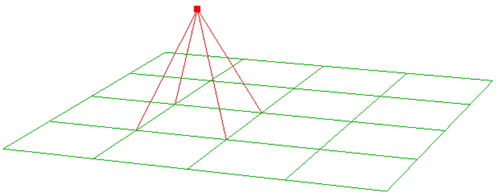
Equipment placed on top of frame.



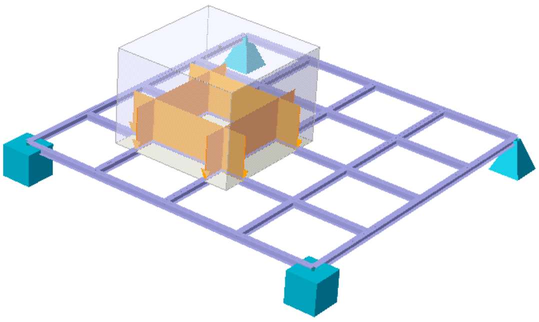
Linear static analysis: Equipment mass represented as line load in finite element model.



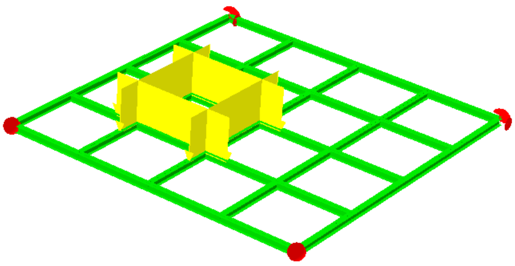
Mass model: Equipment mass represented as point mass plus beam connecting the structural part. Hinges are introduced between connecting beams and structure to avoid moment transfer when point mass is subjected to accelerations. Eccentricities may also be neglected.



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.

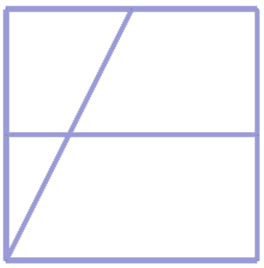
2.10 Run analysis and look at results

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.

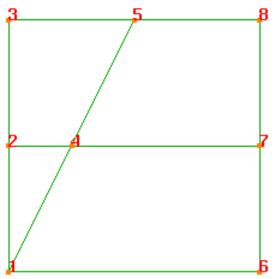
When you start the analysis GeniE will automatically recalculate all the loads, generate a new finite element model, run the analysis and create a result file that you can postprocess from the GeniE user interface.

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.

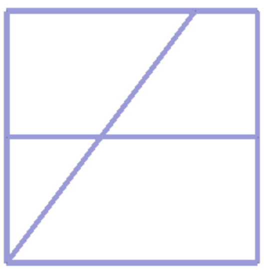
A topology point in the model will introduce a finite element node in the analysis model. In case you have done changes to your model you may want to remove unnecessary topology points to simplify the finite element model. GeniE comes with a special tool for this purpose – the simplify topology feature.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



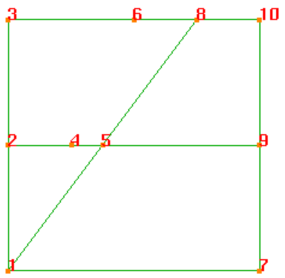
Small frame model.
Before moving member



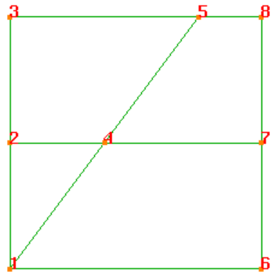
Small frame FE model.
Before moving member.
8 FE nodes created.



Model after moving upper end of inclined member.



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



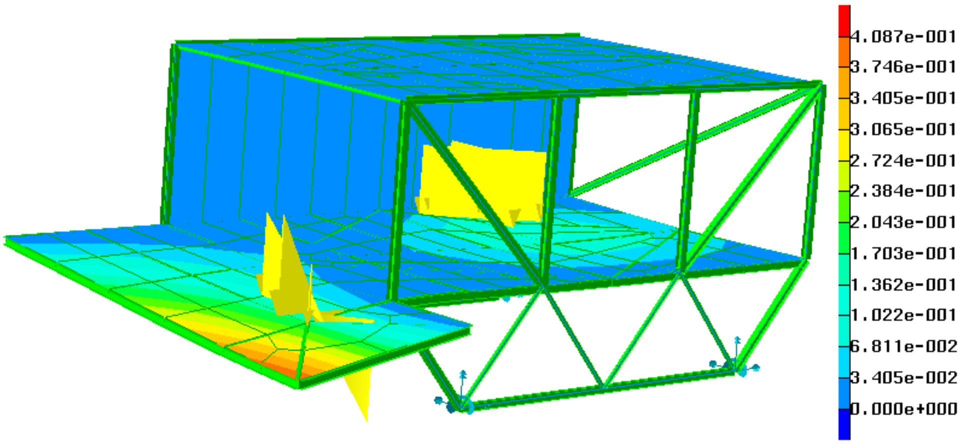
FE Model after simplify topology.
8 FE nodes created.

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.

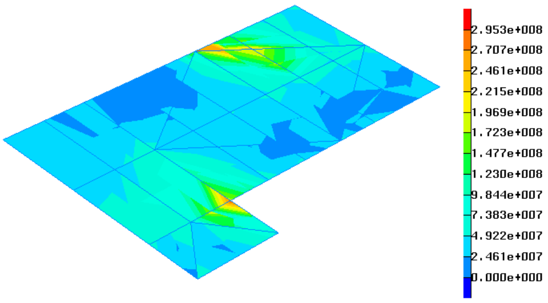
GeniE will let you look at certain results attributes like displacements, general plate stresses, principal stresses, and beam forces and diagrams. You may look at results for the whole model or parts of the model. Another advantage of the concept technology is that a concept holds information about 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).

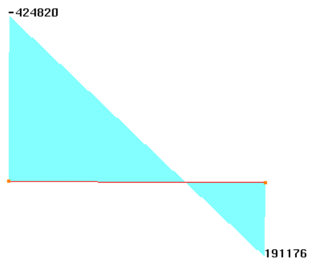
Observe that results are showed in the database units specified when you start a new modelling session in GeniE.



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.



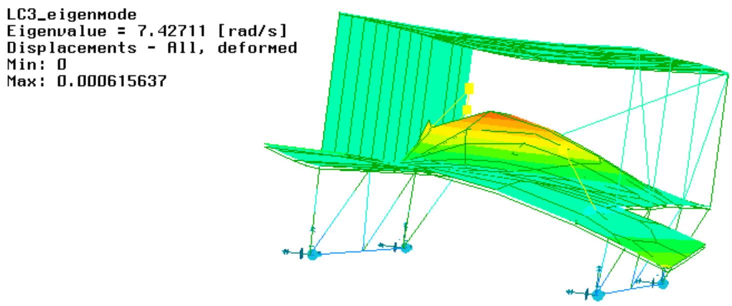
Moment diagram for one selected beam.

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.

2.11 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.

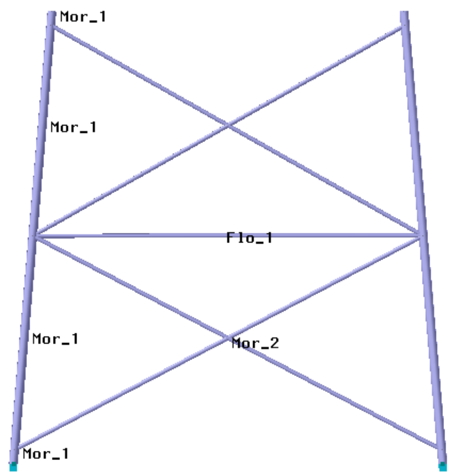


The example shows an eigenvalue analysis performed from the GeniE user interface. All equipment masses contribute to the mass matrix (structure mass plus equipment masses).

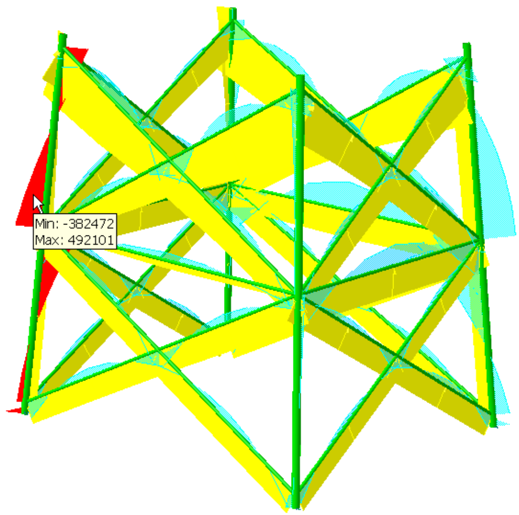
2.12 Run hydrodynamic analysis

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



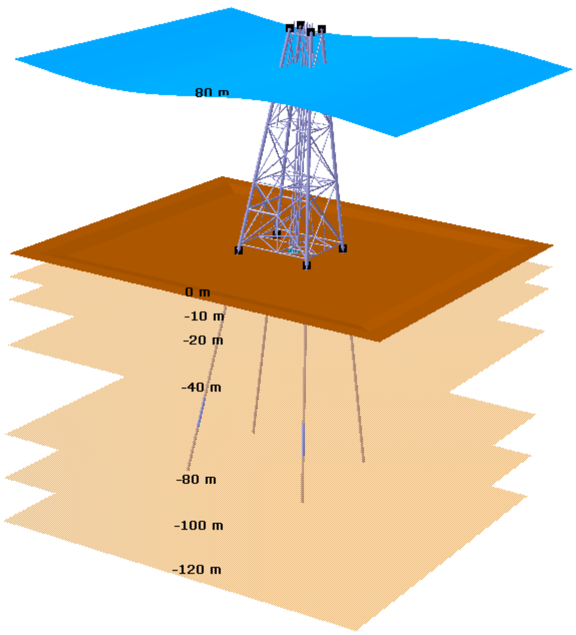
Beam moment diagrams (blue colour) on structure exposed to wave loads (yellow colour)

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.

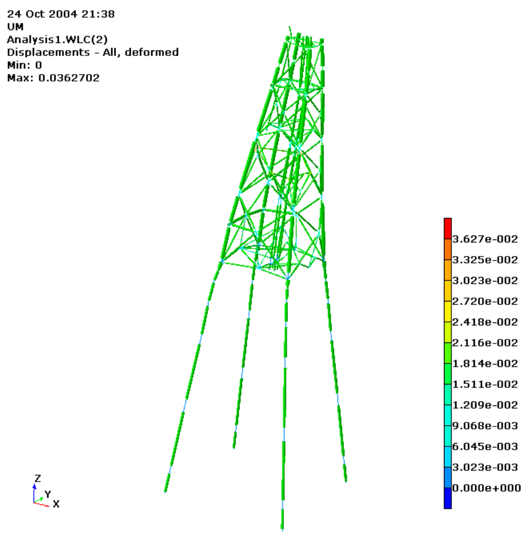
2.13 Run pile and soil analysis

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.

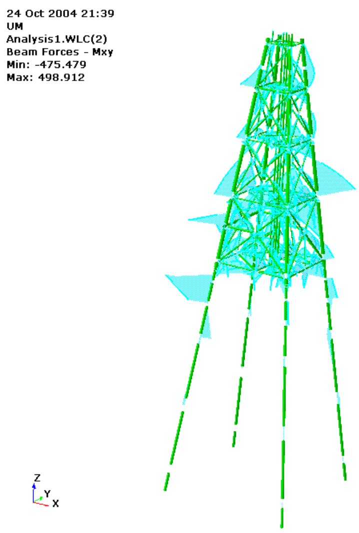
The picture below shows the various parts of a complete model and an example on how results may be investigated for a complete model.



A complete model consisting of both environment and structure.



Deformed shape

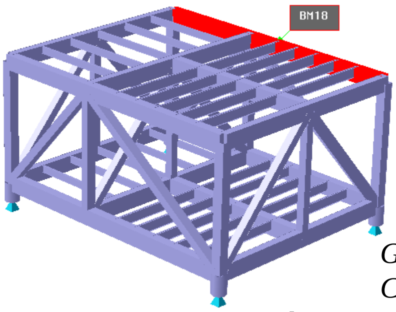


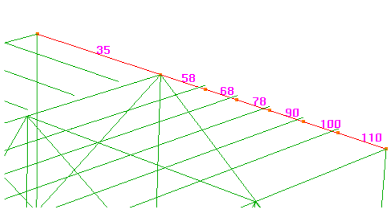
Bending moments

2.14 Code checking and advanced results processing

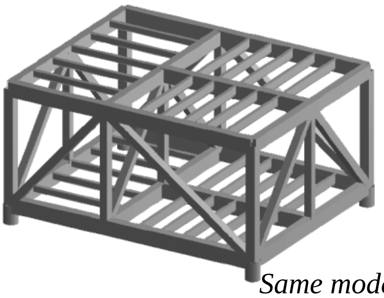
The code checking of frame structures is done in Framework while code checking of plate structures is done in Platework. More advanced post-processing can be done in Xtract. Platework and Xtract are based on finite element technology meaning that the concept information cannot be referred to in these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

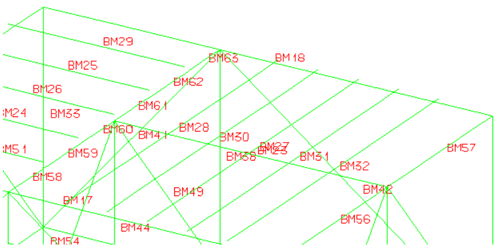
The concept information built into the model created in GeniE is understood and used by Framework meaning that all the naming conventions (e.g. beams, loads, materials, sets) are the same in the two programs. Note, however, that you should limit all names to 8 characters. The examples below show a small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



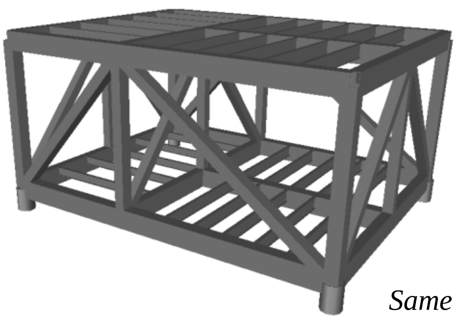


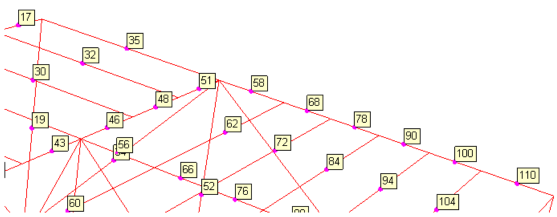
GeniE created model
Concept level: BM18
Finite element numbers: 35, 58, 68, 78, 90, 100, 110





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

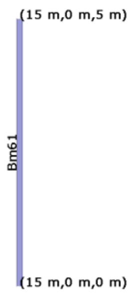
A re-analysis may easily be performed after changing some of the properties. Often new structural members have been inserted, section types edited and changed, mesh density refined and so on.

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.

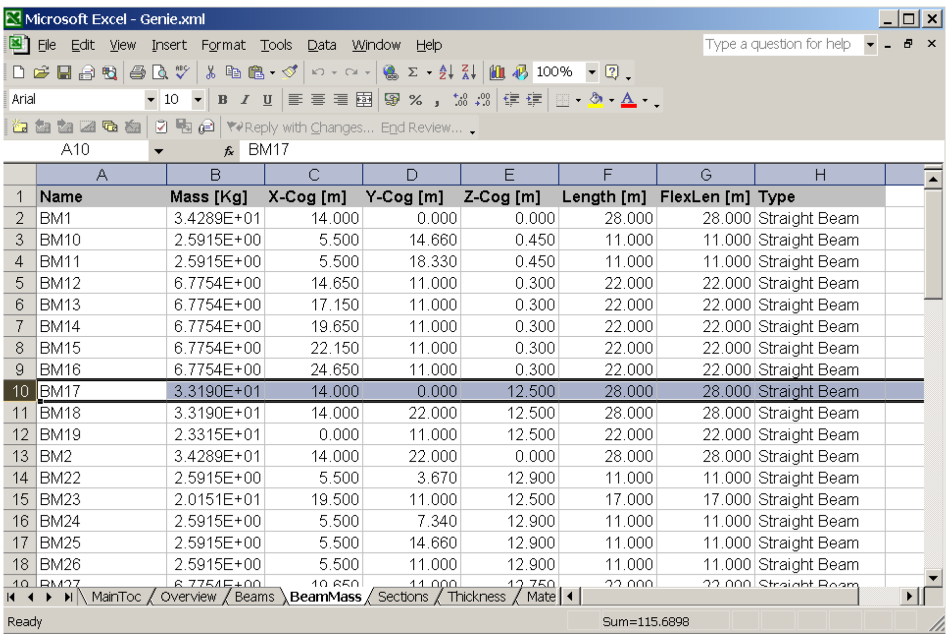
2.16 Documenting the model

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.



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.



Name	Mass [Kg]	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	Length [m]	FlexLen [m]	Type
BM1	3.4289E+01	14.000	0.000	0.000	28.000	28.000	Straight Beam
BM10	2.5915E+00	5.500	14.660	0.450	11.000	11.000	Straight Beam
BM11	2.5915E+00	5.500	18.330	0.450	11.000	11.000	Straight Beam
BM12	6.7754E+00	14.650	11.000	0.300	22.000	22.000	Straight Beam
BM13	6.7754E+00	17.150	11.000	0.300	22.000	22.000	Straight Beam
BM14	6.7754E+00	19.650	11.000	0.300	22.000	22.000	Straight Beam
BM15	6.7754E+00	22.150	11.000	0.300	22.000	22.000	Straight Beam
BM16	6.7754E+00	24.650	11.000	0.300	22.000	22.000	Straight Beam
BM17	3.3190E+01	14.000	0.000	12.500	28.000	28.000	Straight Beam
BM18	3.3190E+01	14.000	22.000	12.500	28.000	28.000	Straight Beam
BM19	2.3315E+01	0.000	11.000	12.500	22.000	22.000	Straight Beam
BM2	3.4289E+01	14.000	22.000	0.000	28.000	28.000	Straight Beam
BM22	2.5915E+00	5.500	3.670	12.900	11.000	11.000	Straight Beam
BM23	2.0151E+01	19.500	11.000	12.500	17.000	17.000	Straight Beam
BM24	2.5915E+00	5.500	7.340	12.900	11.000	11.000	Straight Beam
BM25	2.5915E+00	5.500	14.660	12.900	11.000	11.000	Straight Beam
BM26	2.5915E+00	5.500	11.000	12.900	11.000	11.000	Straight Beam
BM27	6.7754E+00	19.650	11.000	12.750	22.000	22.000	Straight Beam

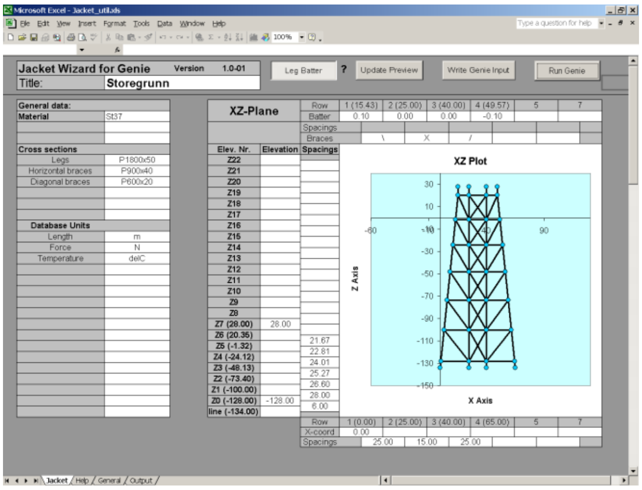
The model data documented using MS Excel.
In this case the mass of BM17 is in focus

2.17 Journalling and reading commands, wizards

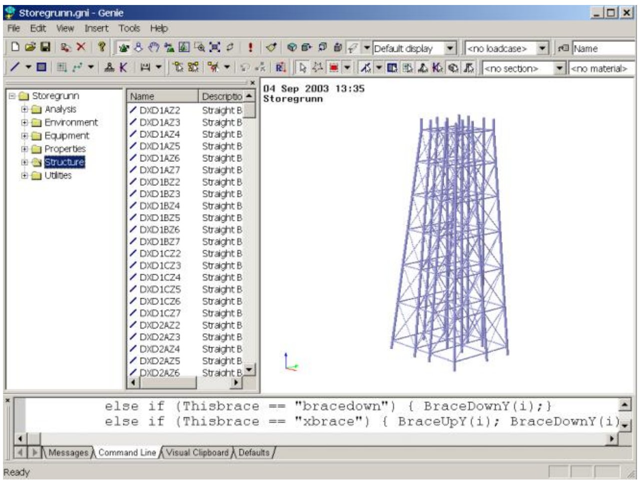
GeniE comes with full journal support of all modelling and analysis commands. This means that every change of the model is journalled to a file unique for each modelling session. The journal file (file extension *.js) can be read in to re-create the model and it can be modified in a standard text editor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

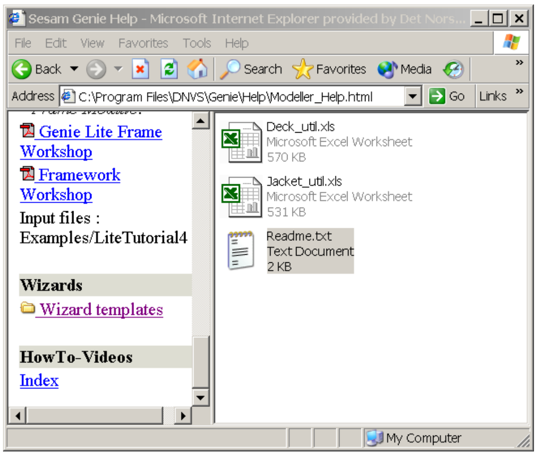


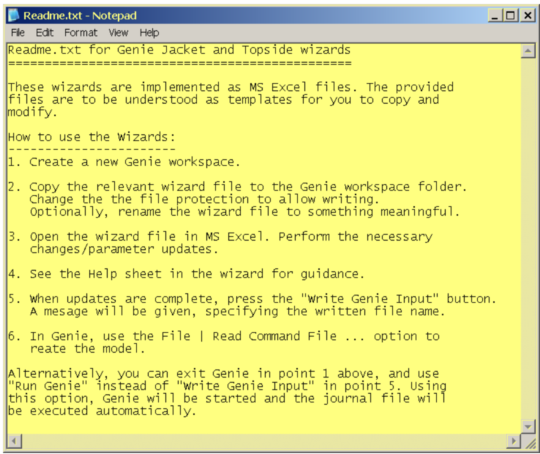
The wizard guides the user in how to specify a jacket (or topside) structure. The wizard will create a journal file and start GeniE that automatically creates the model. The wizard also contains information on how it is build up so that you can create your own wizards.



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.





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(10 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(10 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

//***** EQUIPMENTS AND LOAD MODELLING *****//
//Definition of LoadCases
LC1 = LoadCase();
LC1.setFemLoadcase(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 easy to understand how it is built up. The picture to the left shows an example of a clean journal file.

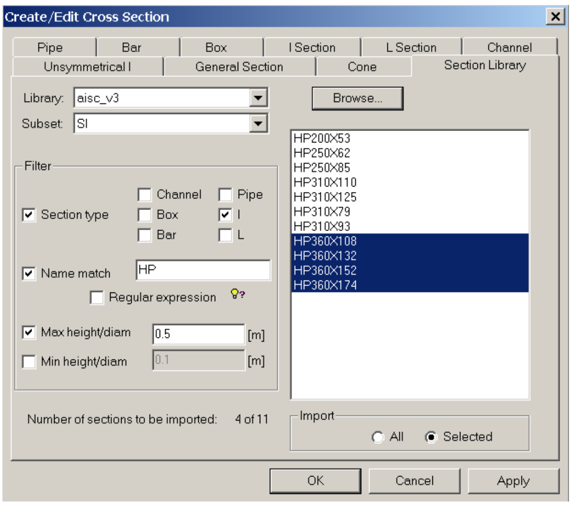
The journalling system is documented on the help pages (***Help|Help Topics*** -> Jscript commands). Further information may be found on http://msdn.microsoft.com/scripting/default.htm?/scripting/JScript/ and http://msdn.microsoft.com/scripting/jscript/download/jsdoc.exe.

2.18 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.

2.19 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.

2.20 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

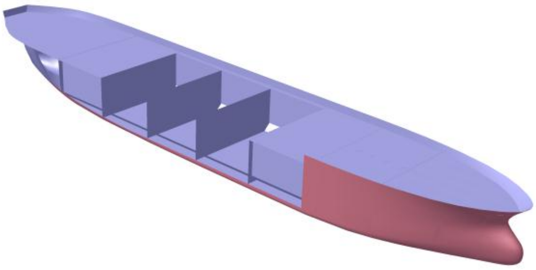
2.21 Panel modelling

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.

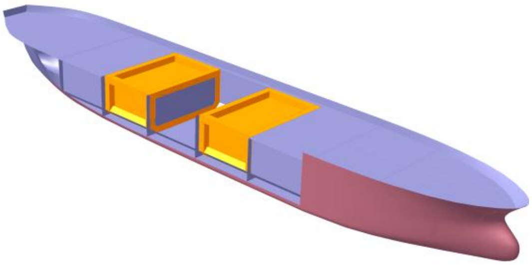
The hull form of a floating vessel is often built up from double curved surfaces. GeniE can handle such curves either from a direct import using existing structure as found on a SAT file or by a manual definition of all cross sections followed by a plate skinning operation.

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

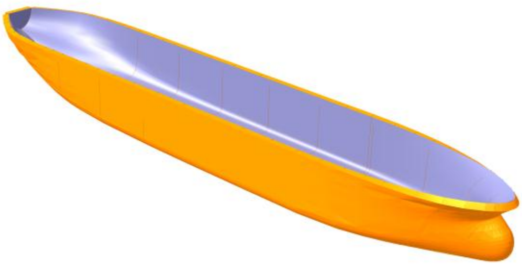
The pictures below show an example where the hull has been imported from a SAT format and tanks have been defined.



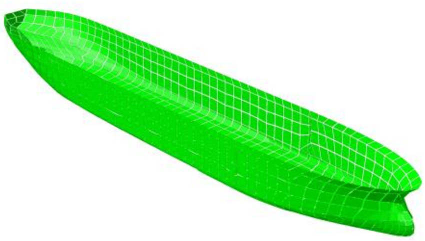
A ship hull form imported from SAT format. Parts of the structure removed for visualisation.



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.

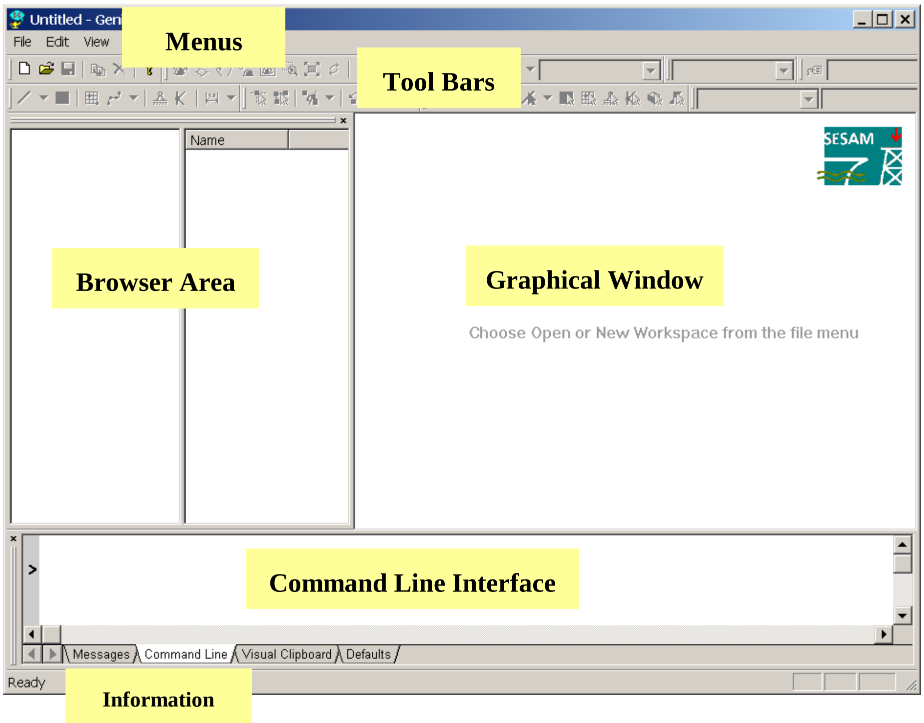
3. USER'S GUIDE TO GENIE

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:



The user interface is fully dockable, meaning you can configure the layout to your own preferences.

The pulldown menus are available from the area *Menus*, while the buttons (or short cut to a certain command) are available from the *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 journalling 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

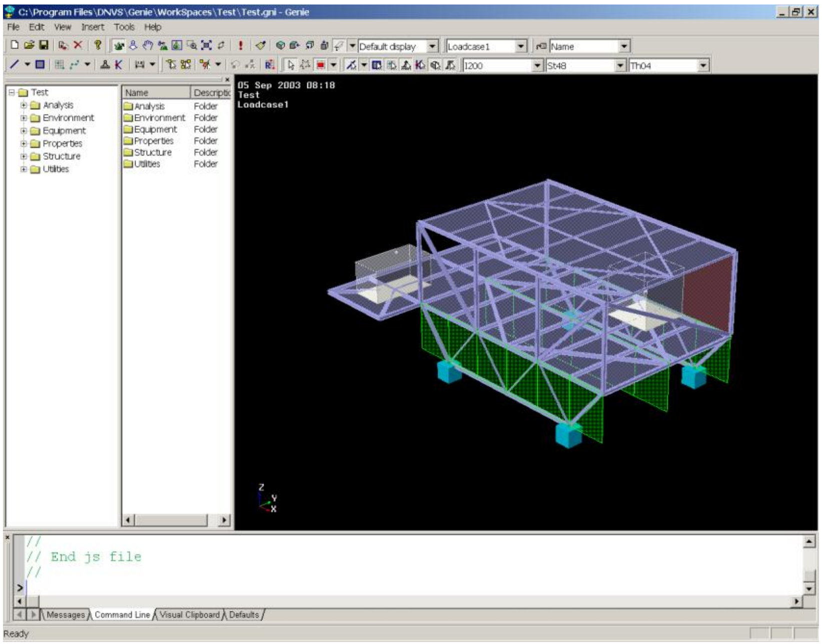
File|New Workspace <model name>

This will create a directory on your disk c:\program files\dnvs\GeniE\workspaces\<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:

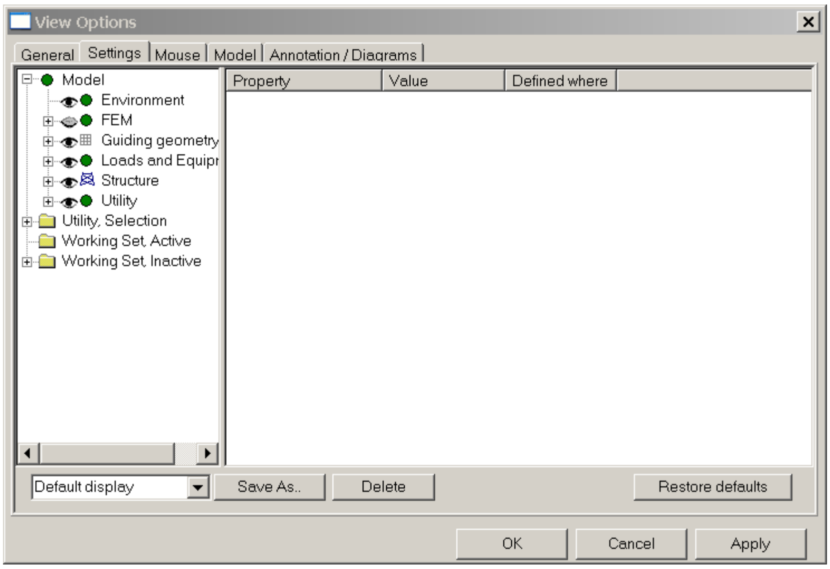


The picture shows GeniE with default setting enabled for graphic viewing. In the following several options on how to change these are mentioned.

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

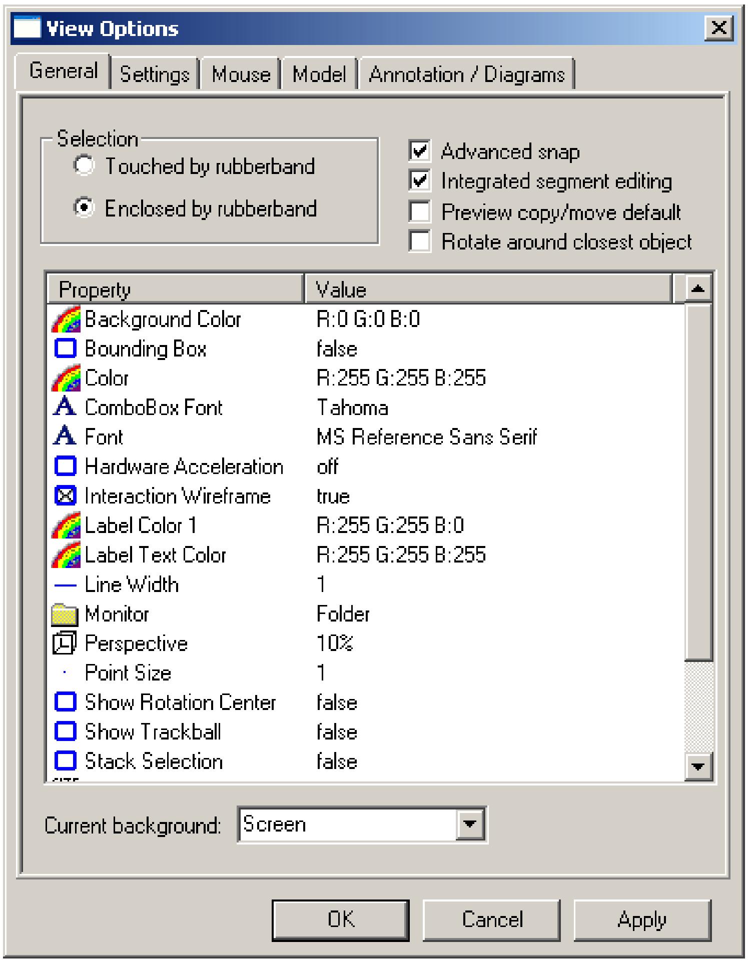
Settings – predefined model views

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.

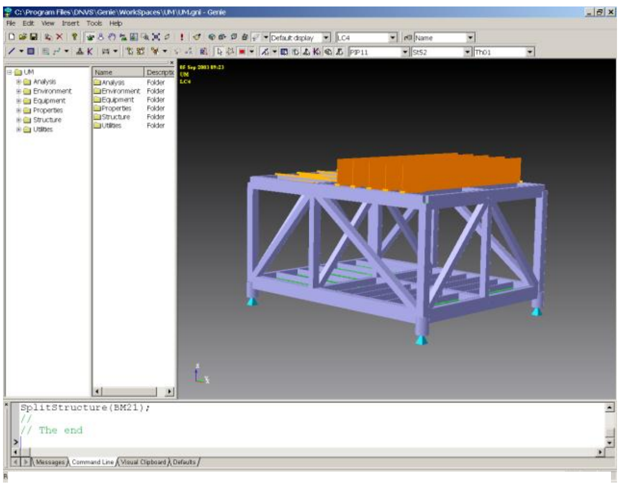
Also observe that if you want a preview of copy or move operations, you may set this to your default option by ticking the right box.

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

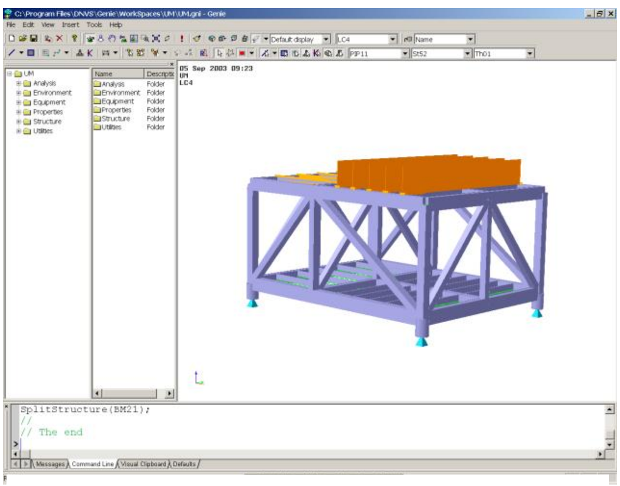
When producing a graphic file, it may be beneficial for you to change Current background from Screen to paper – this will in many cases make a better view when the picture has been imported to an office system like e.g. MS Word.

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



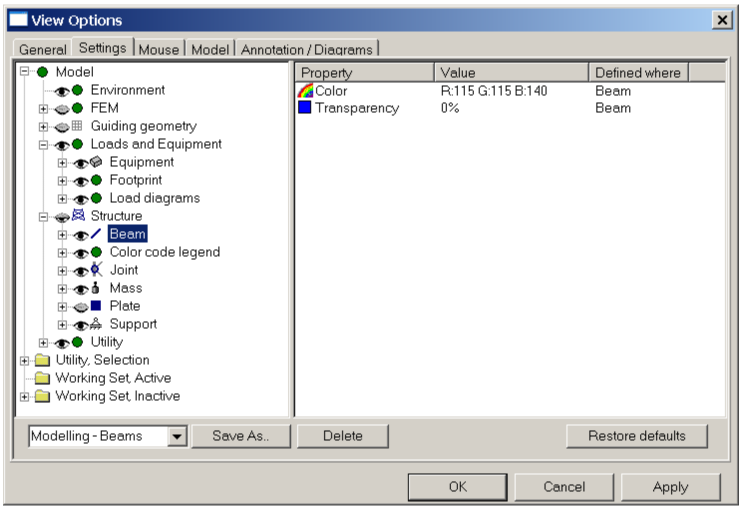
View model when background is set to Paper

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.

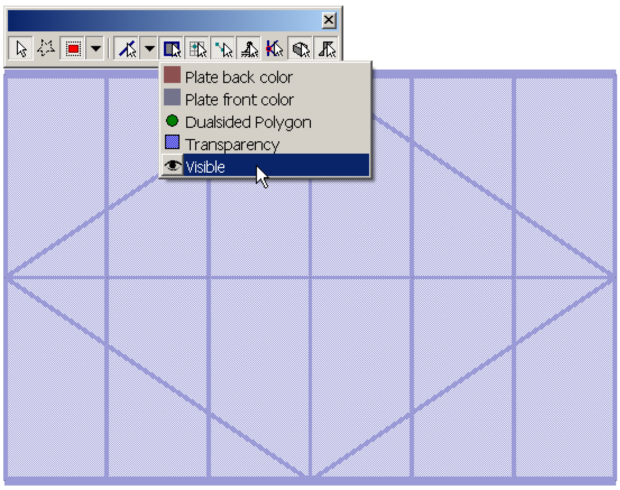


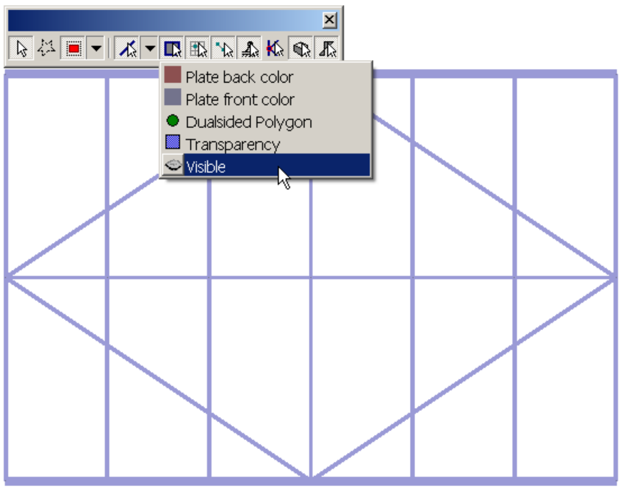
In the example to the left a new model view Modelling – Beams has been created from saving the Modelling – Transparent and removed visibility of plates. As can be seen, the eye symbol is closed. Furthermore, the colours of the beams are changed to Red:155, Green:115, Blue:140.

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

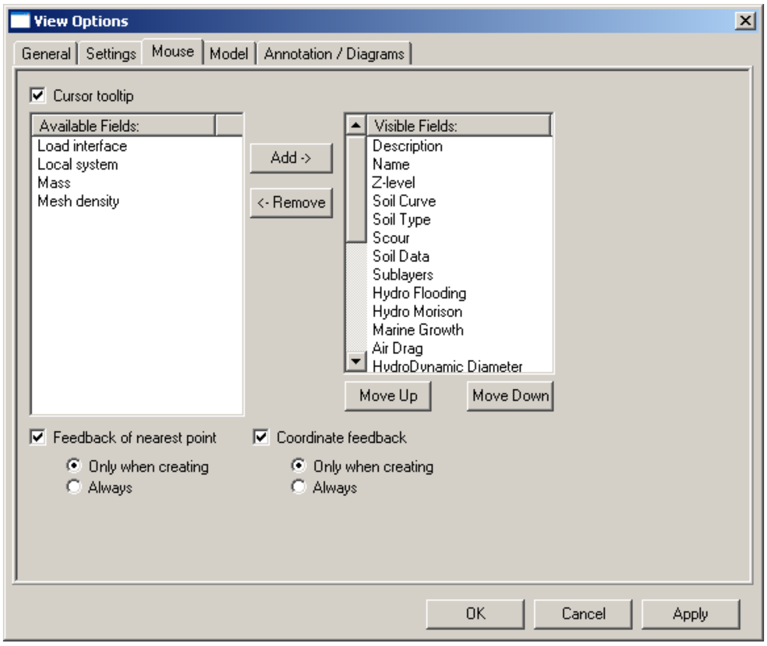
The new model view is permanent so that you can use it in other GeniE sessions as well.

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.





3.1.3.3 Change actions associated with the cursor



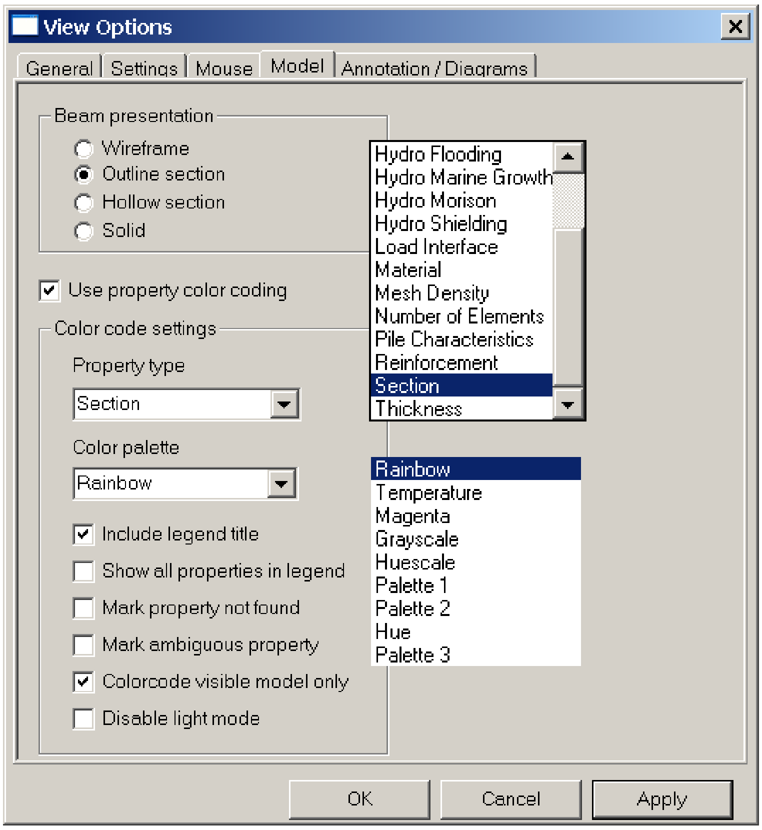
You may decide to see relevant information when moving the cursor over an object. For example, using the settings to the left will result in the following graphical feedback of a beam:

- 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*).

3.1.3.4 Change graphical views and property colour coding



There are a number of ways to view your model:

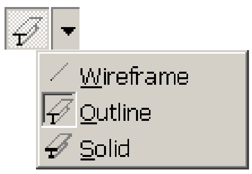
Wireframe will draw a beam as a line disregarding any section property. Plates are not showed as such except for the boundaries.

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.

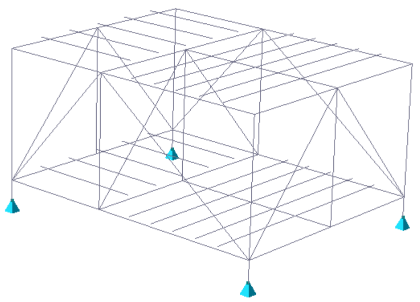
Solid will draw a section in full including thickness view. Plates are showed without thickness.

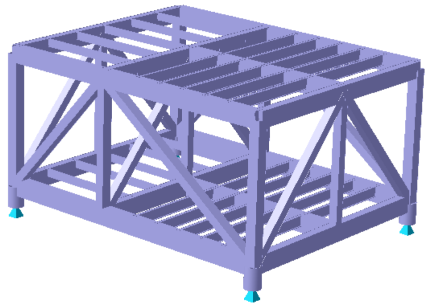
Observe there is a Toolbar pulldown button that gives you a quicker access to switching beam views.

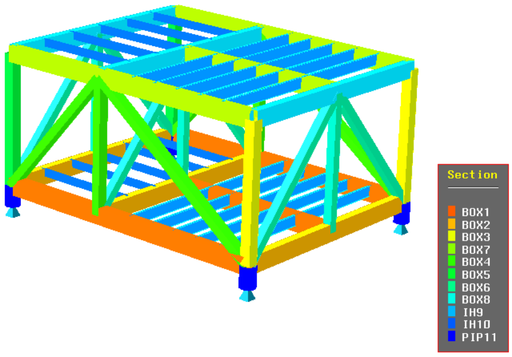


The property colour coding applies for the attributes shown above using a set of predefined colour palettes.

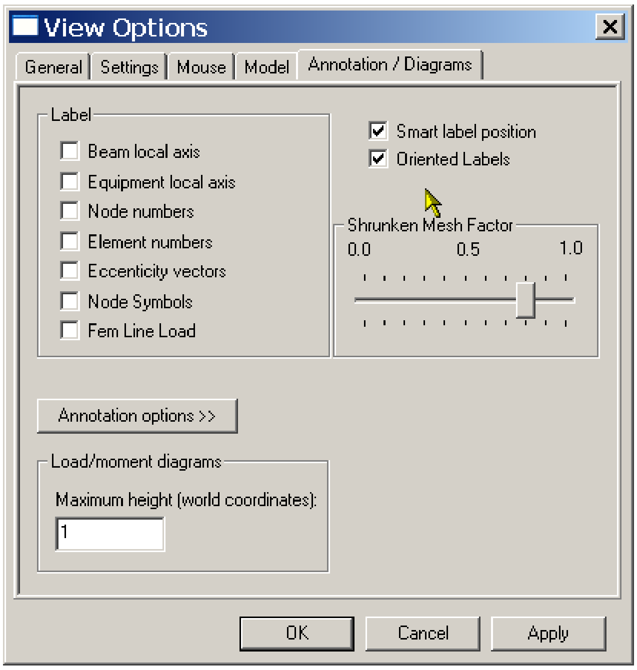
The examples below show a model viewed in wireframe, solid, and outline including colour coding of section types. All pictures have been produced using option *Paper background*.







3.1.3.5 Global labelling



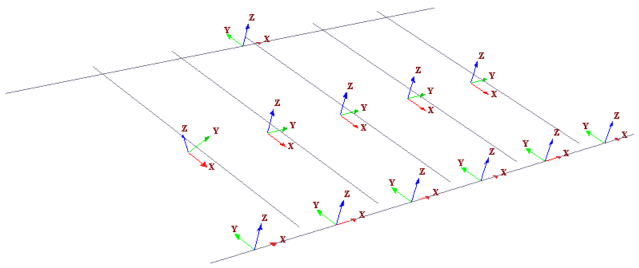
You may label the global model (or the selection as seen in the graphic window) and label certain attributes to it. You can add the following information to your model

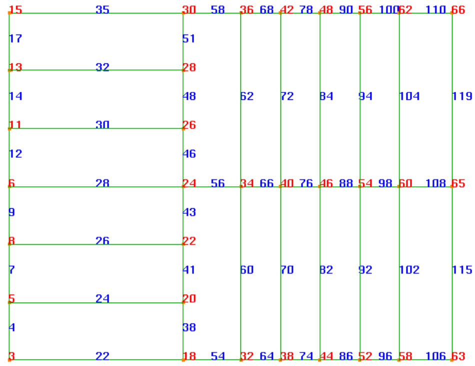
- Local axis of beams
- Local axis of equipments
- Node numbers
- Element numbers
- Eccentricity vectors
- Node symbols
- Shrunken mesh factor when displaying the FE mesh
- Ensure that labels are not stacked on top of each other (smart label positions)
- Make labels oriented along beam axis (oriented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

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

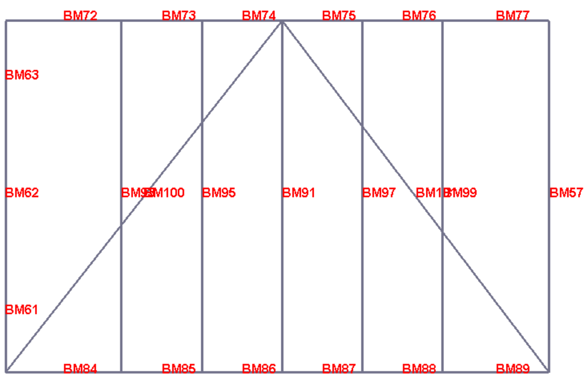
You may also do labelling per individual object(s); see later for a description on how to do it.

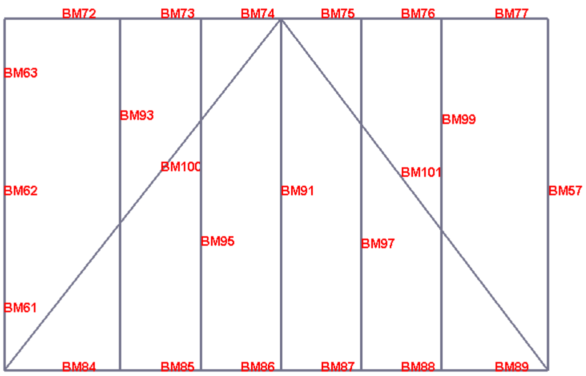
In the examples below are showed beam local axis (one of the beams have been rotated around local x-axis) and node and element numbers for a set of beams (note that the default colours for node and element numbers have been changed to red and blue in this case).





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.





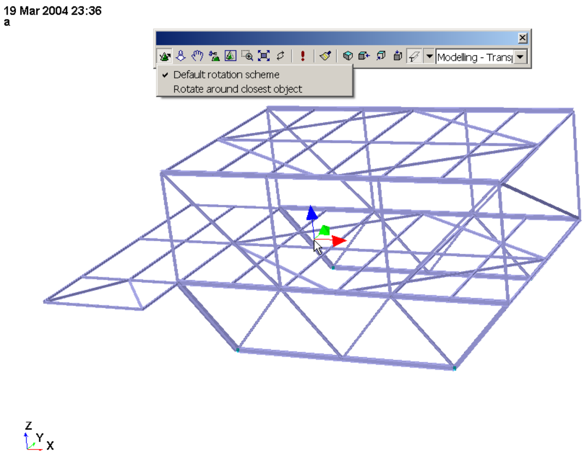
3.1.4 View manipulations (LMB & RMB)

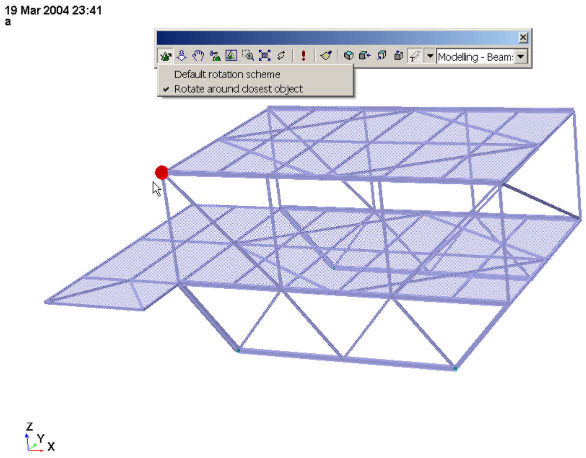
The left mouse button (LMB) is reserved for selections (including rubberband selection), input fields, and graphic modelling. The right mouse button (RMB) is used for view manipulations like rotate, zoom, and pan.

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

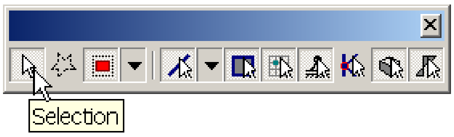
When selecting objects that are overlapping, the following prioritization order will be used, top priority first:

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

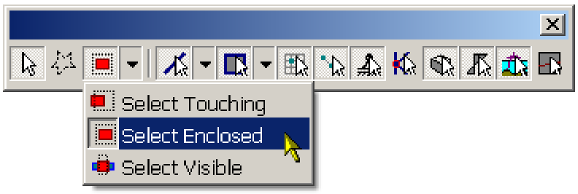
There are two ways of selecting objects (beam, plate, equipment), graphically or via the browser. A combination of these two is also possible. You may also select the finite element mesh while in graphic mode.

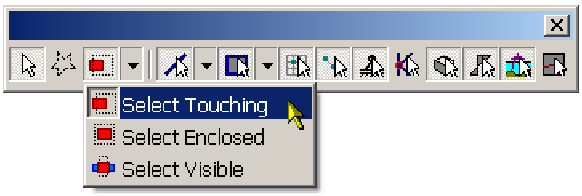
3.1.5.1 Graphic selection

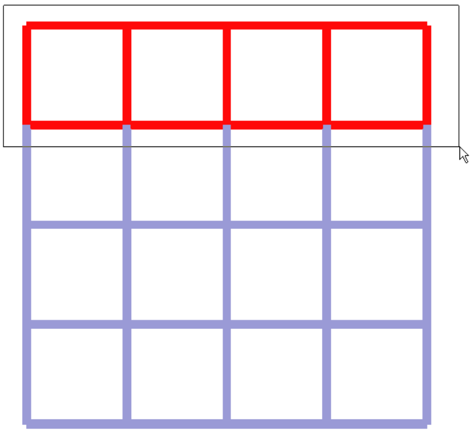
Single objects (e.g. beam, plate, equipment) are selected graphically by clicking the object. If you want to select more than one object you can click (LMB) the first one, then shift+click on the additional ones. Note that you need to set GeniE to selection mode by activating the selection button showed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

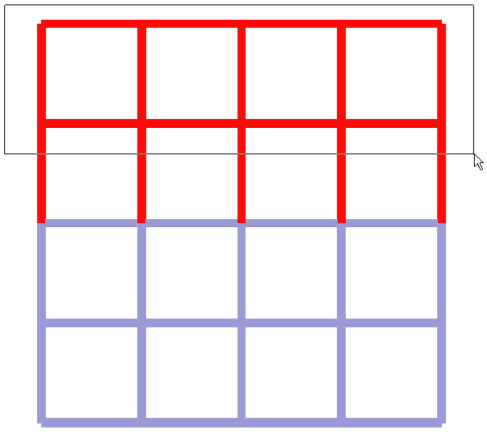


You may use rubberband selection, either all objects touched by or enclosed by the rubberband. You switch between these two options using the rubberband button on the selection toolbar. The examples below show the difference between enclosed, touched by or select only those visible. Note that all selected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

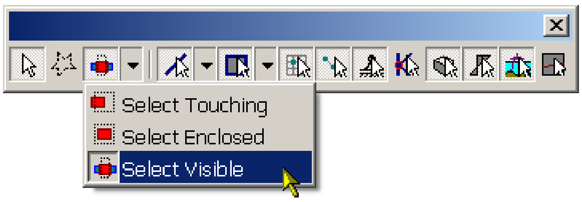


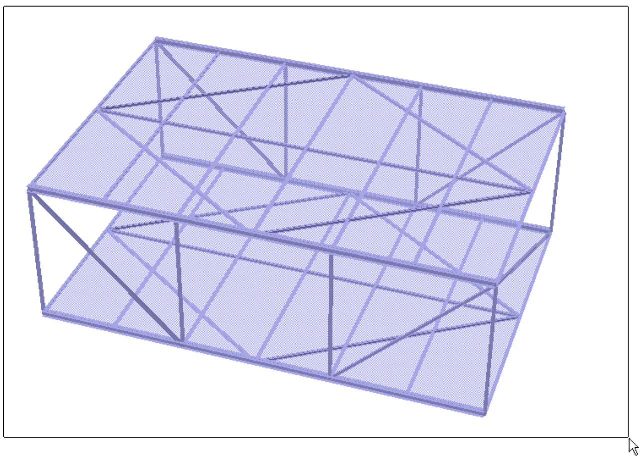




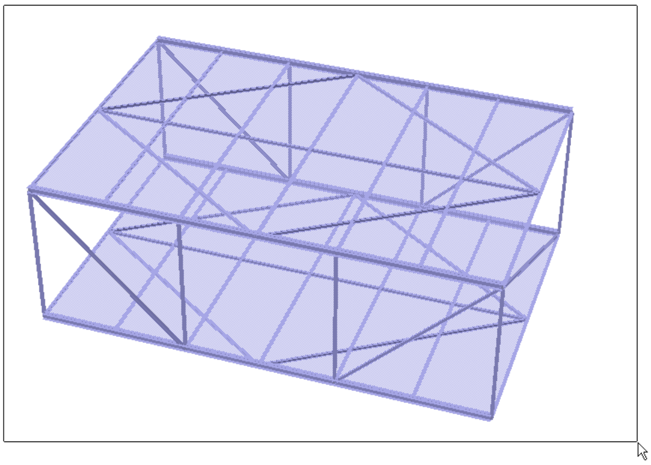


Observe that these selection methods will select all objects (visible or not) that are inside the rubberband. If you want to select only what you have selected, you should use the option *Select Visible*.

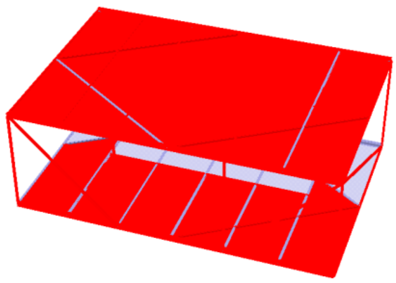


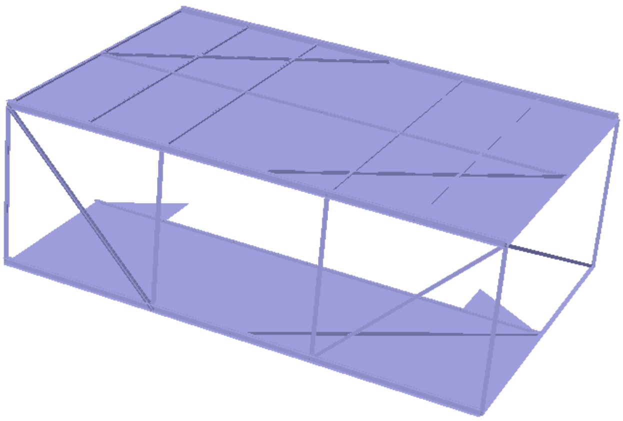


The model consists of plates and beams. When viewing from this eye-position, some of the beams and plates are not visible. In this example they are visible because transparency has been used for illustration effects.

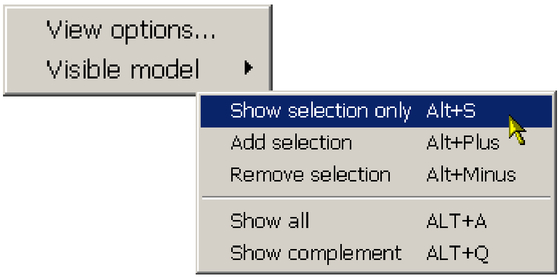


When making a rubberband select using the *Select Visible* option, the non-visible structural parts on the bottom level are not selected.

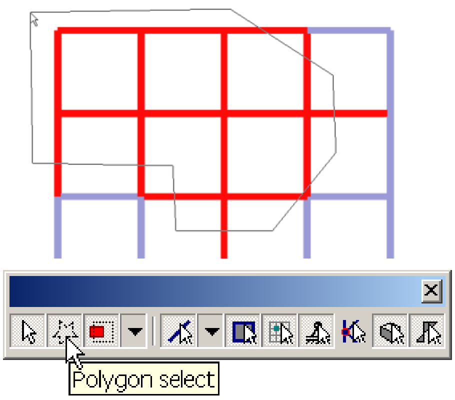




By using ALT+S or use the commands shown below, you visualise those selected. As can be seen, parts of the lower level are not selected.



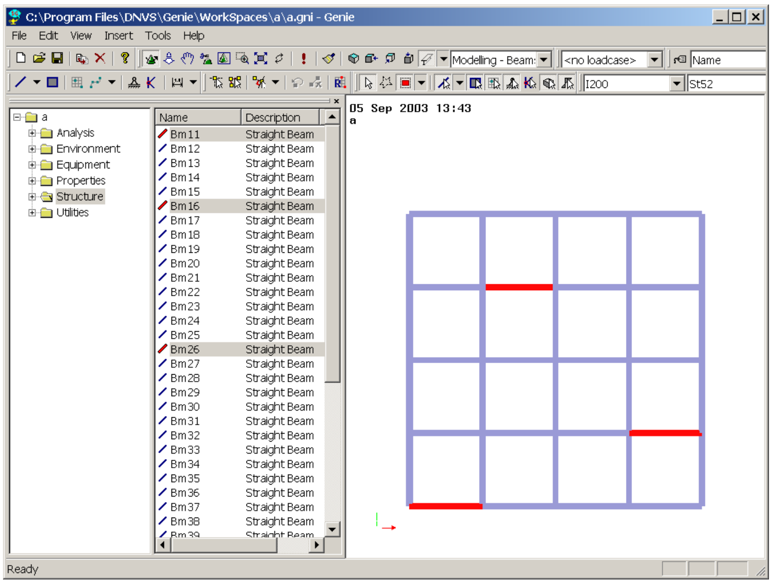
The example above assumed using rubberband selection. You may also use the polygon selection. In this example this method is used together with rubberband *Select Touching*.



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:

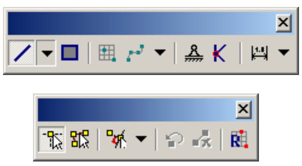
- Click (LMB) on individual object
- Click (LMB) and shift+click (LMB) to select all objects in-between a list.
- Click (LMB) and shift+arrow (up or down) to select all objects in-between a list.
- Click (LMB) and ctrl+click (LMB) to select individual objects



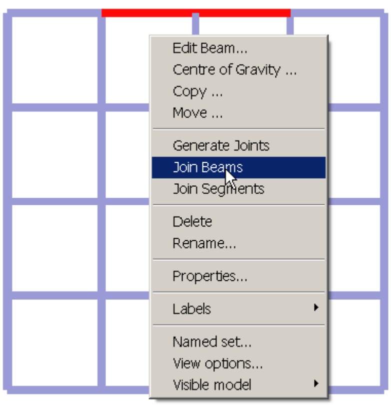
This example show that Bm11, Bm16, and Bm26 have been selected using the Ctrl+click option. Note that selected items are automatically highlighted in the graphic window (and vice versa if graphic selection is performed).

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.

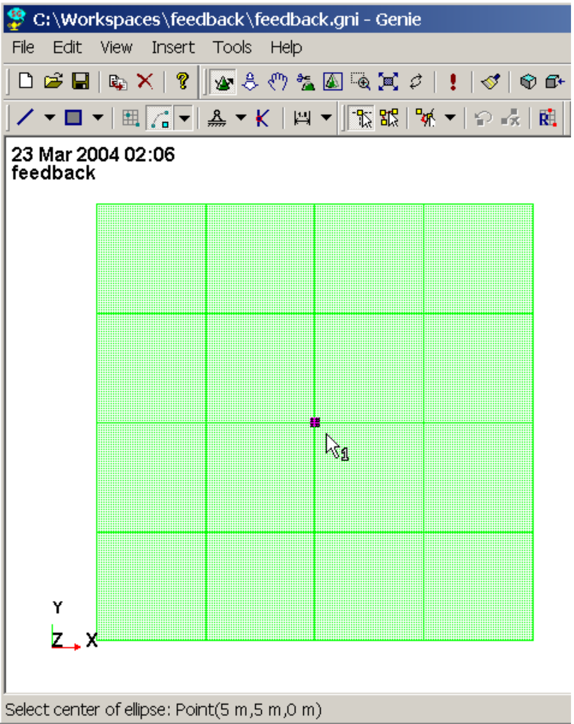


From the Toolbar you click Insert Straight Beam and make sure that GeniE is in modelling modus. Normally, when clicking *Insert Beam* (or other) this will automatically switch GeniE to modelling modus. You may also do the same from the pull-down menu by using command **Insert|Beam|Straight Beam Dialogue**.



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.

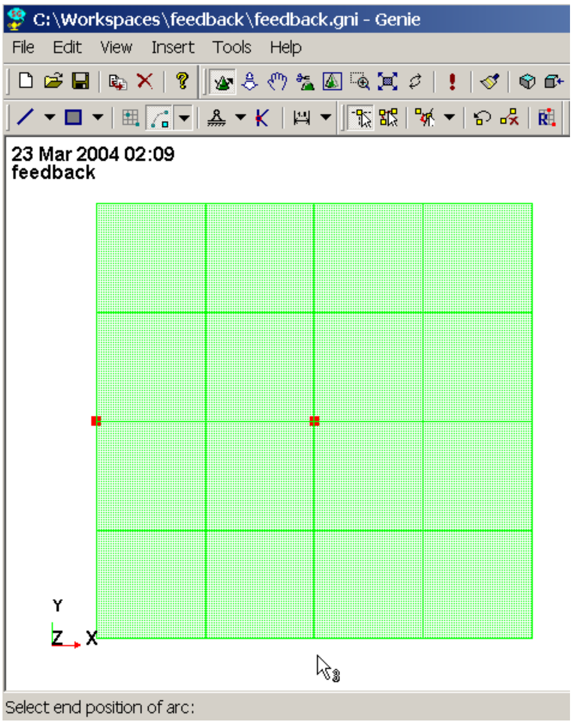
GeniE will query for expected input in the status bar (at the bottom left part of the GUI). The sequence of pictures below depicts how the program will ask for expected input. The example is based on inserting a guide spline requiring 3 input parameters – origin, start and end positions.



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.



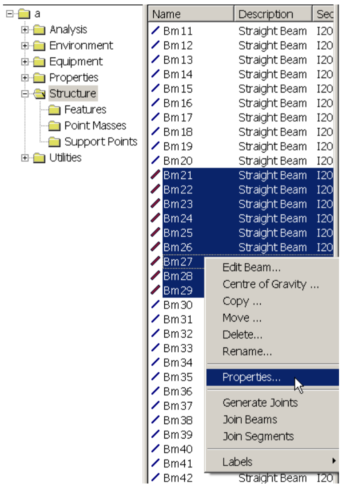
Step 3: Insert the end position

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.

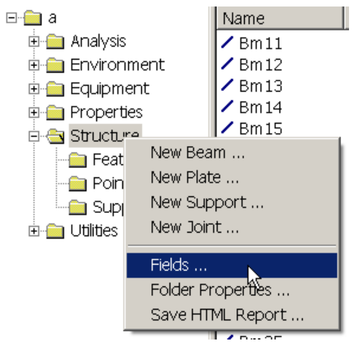


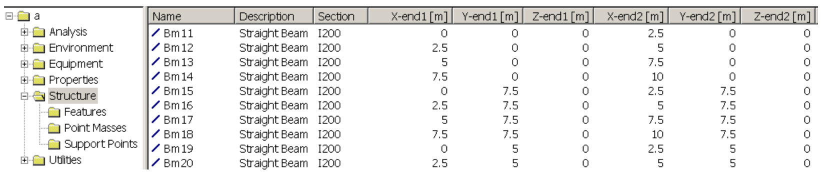
When you right click (RMB) in the browser area you activate the available commands. For example, when right clicking while in the Structure folder, you get access to commands for defining a new beam, plate, support or joint.

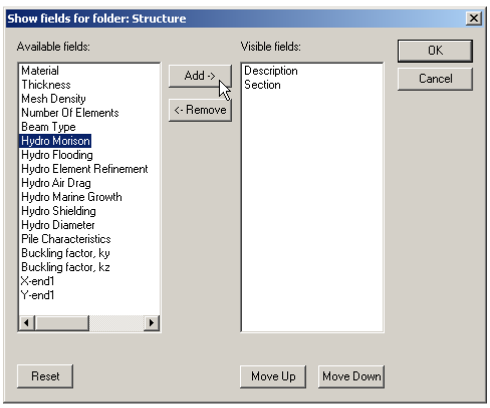
Selecting objects in the browser and the use of RMB will force the context sensitive menu. You can then make changes or operations to the objects selected. The example to the left shows that some beams have been selected, RMB to force the context sensitive menu, and Properties have been chosen

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.







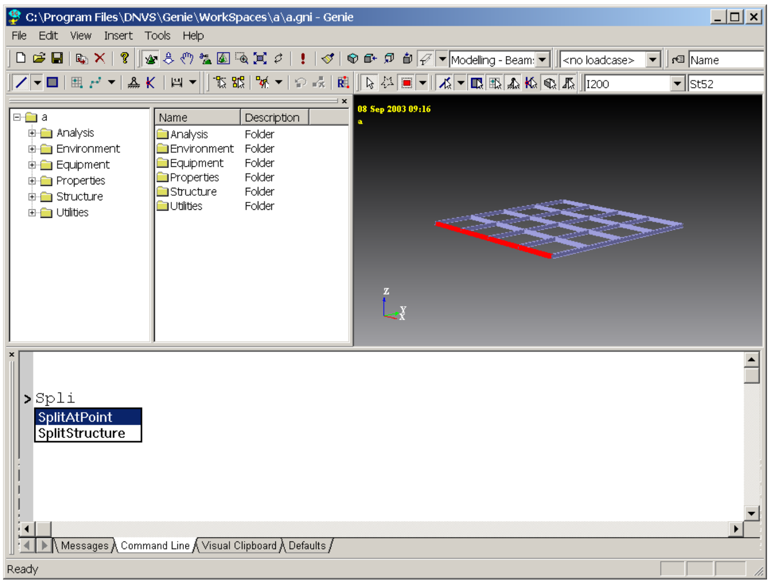
The picture to the left shows the fields you may add to the right tree of the browser.

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.

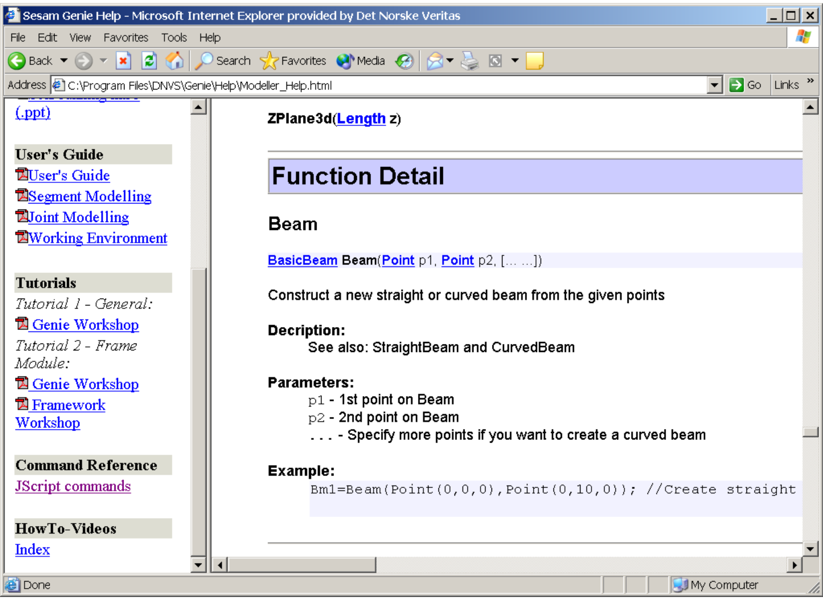


The journal file is created and read by the command line interface system (CLI) running GeniE, you see this window at the bottom of the GUI. You can type the commands directly from this window, if you don't remember the complete command you can type parts of it and push TAB to get further guidance. See 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.



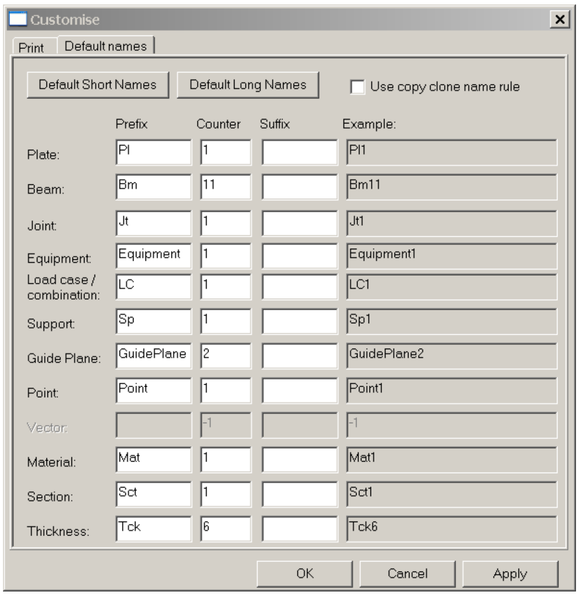
A journal file is read into GeniE using the following command ***File|Read Command File***. You may read in several journal files to the same model, but they will be logged as one current journal file.

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.

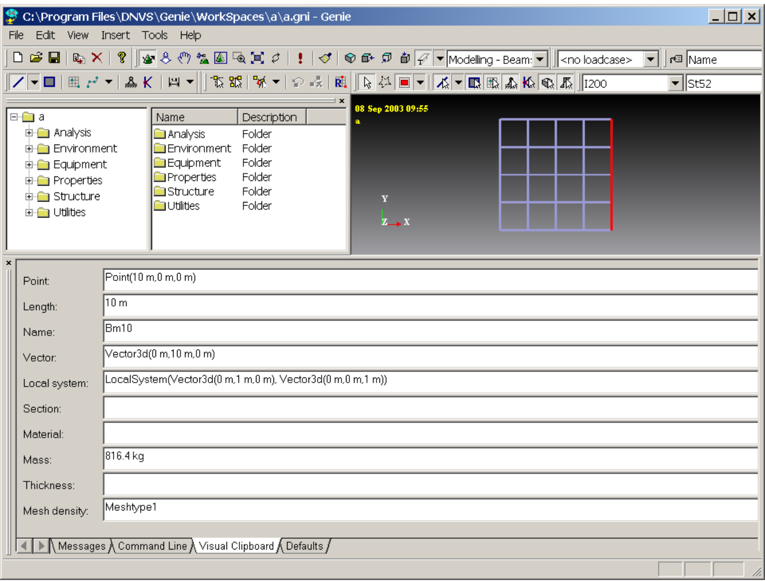
Default long names will give longer names, for example Prefix <Beam>.

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.

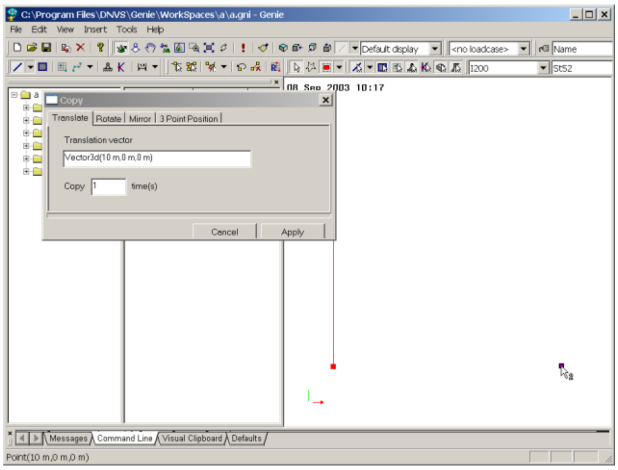
3.1.10 Using the clipboards

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.



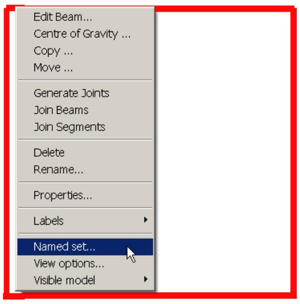
You may also want to find information directly when you are in a modelling modus. For example you want to copy a beam from a point to another point. You then start your copy operation, and find the copy vector graphically, see example below.

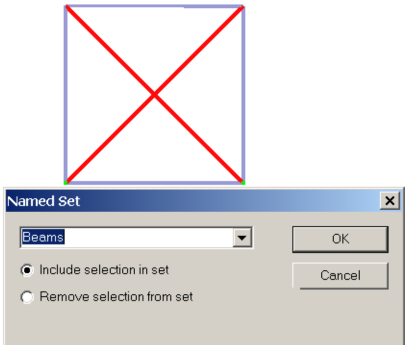


Select the beam you want to copy, then activate the copy command (either from RMB or pull-down menu), then click in the input field before clicking on the ‘from point’ before finally clicking on the ‘to point’. In this case a copy vector of 10 m is found and automatically inserted into the vector field.

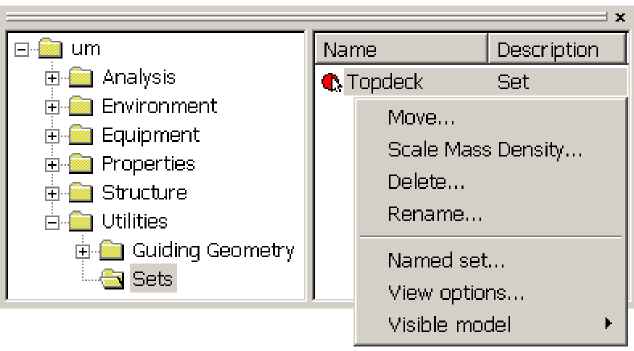
3.1.11 Making a named set

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.



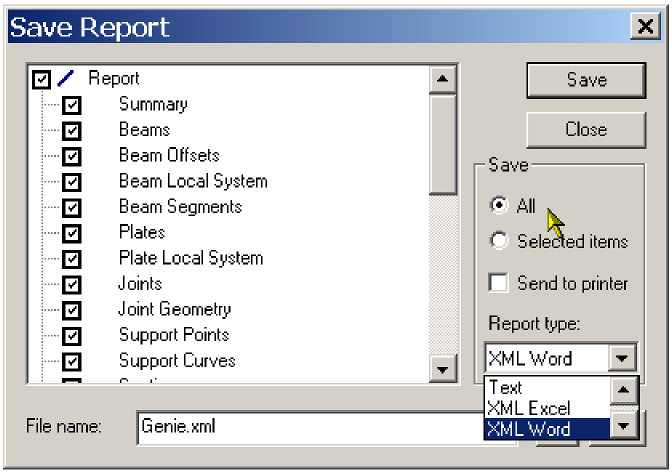
Note that when you use the option Move from this menu, the set name is referred to when scripting the move operation on the journal file, for example *autoMSet.moveTranslate(Vector3d(0 m,0 m,10 m),geUNCONNECTED);*. If you select the members of the set graphically and select Move from the context sensitive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

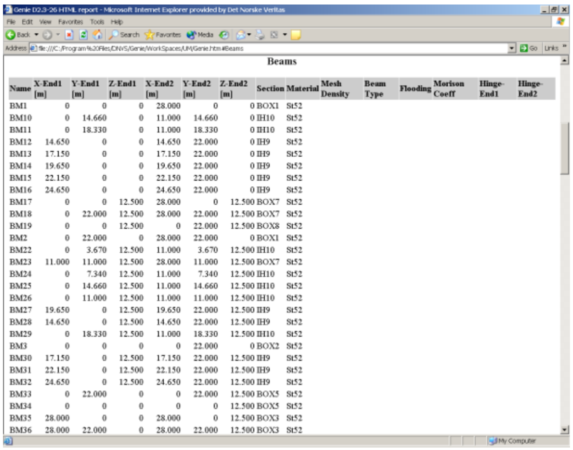
The command ***File|Save report*** will let you print out or save the items as selected by you.

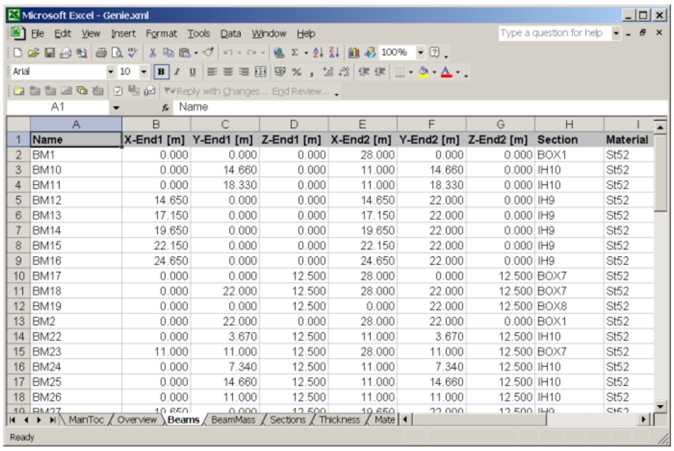


The file is saved either as:

- A text file. The View option uses Notepad as editor unless you have specified your preference when viewing a *.lis file.
- A set of html files. The View option starts Internet explorer
- A XML file that you can automatically view in MS Excel or MS Word. When creating

See below for examples.





3.1.12.1 Automatically include pictures in Word XML reports

When saving a report for XML Word, GeniE will create a report based on a predefined

- Page set up including headers and footers. You may change these when you open up the file in MS Word
- Table of content. The report contains a table of content made for on-line viewing. If you want to make the table of content intended for reading purposes, then simply use the following command from MS Word: *Insert|Reference|Index and Tables|Table and Contents*.
- It is possible to create and save pictures using an edited script file (or a journal file). The commands available for doing so are listed in Chapter 5.3.2 Useful script commands.
- It is advised that you use white background when producing the pictures to be added (use ***View|Options|General*** and set background to paper). From time to time it may, however be better with screen background.

The example below shows a model where four pictures have been automatically created by reading a journal file. In this journal file the viewing angle, the text and the resolution have been specified.

```
//
// Example on a journal file that will create 4 gif files
//
// Read in this file by File|Read Command File
//
Graphics.ViewISO();
Graphics.saveImage("001_Structure_ISO_view.gif",2000,1500);
//
Graphics.viewFromX();
Graphics.saveImage("002_Structure_From_X.gif",2000,1500);
//
Graphics.viewFromY();
Graphics.saveImage("003_Structure_From_Y.gif",2000,1500);
//
Graphics.viewFromZ();
Graphics.saveImage("004_Structure_From_Z.gif",2000,1500);
//
// End of file

```

An example of a journal file that contains

- A naming schema 00n_<text>
- Save plots as gif files
- Viewing from 4 different eye-positions
- Graphics resolution

GeniE V3.2-01	Report: Tutorial	Model Id: Tutorial	Sign: nek
		Description:	Date: 21-Aug-2005
		Model file name: C:\Workspaces\Tutorial\\Tutorial.gni	Last saved: 21-Aug-2005 06:08:15

1 Table of Contents

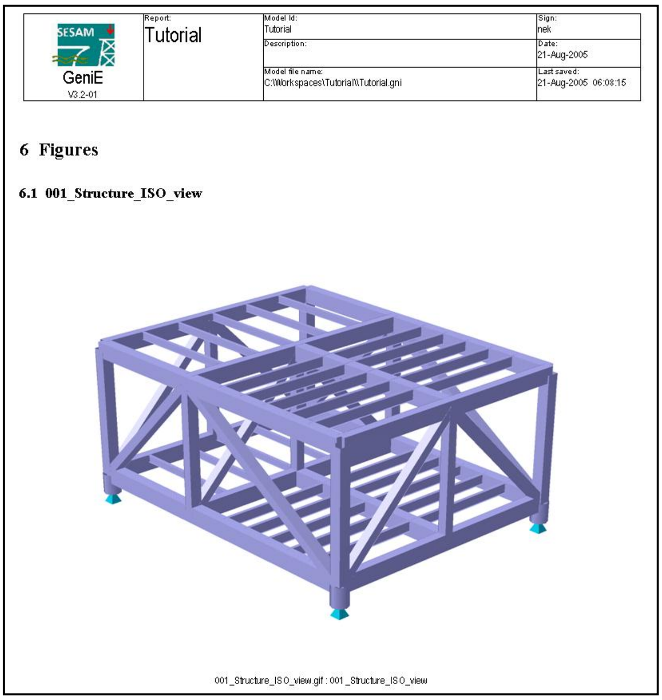
Example of a report created for viewing by MS Word.

The table of content has been changed from the automatic created.

The items on the table of content depend on which items you have selected to save.

Please note that Chapter 2 of this report lists some requirements and project data. You may want to add your own data here.

The table of content automatically includes all pictures of type *.gif that are stored on the project directory:

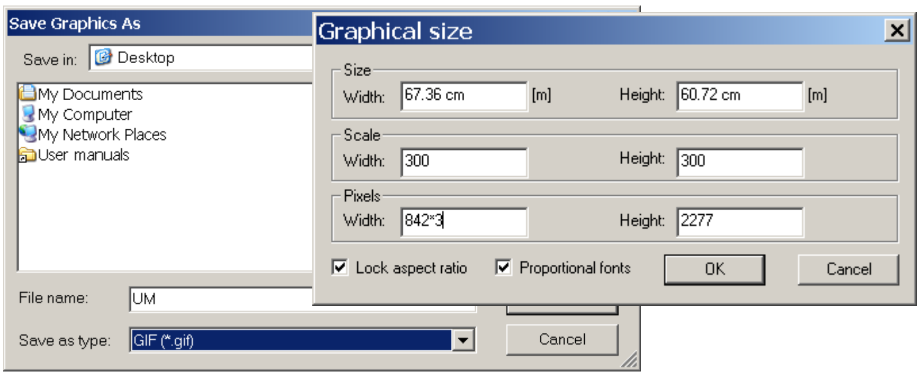


Page 20 of the report contains the chapter numbering, the picture and the picture text.

3.1.13 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.

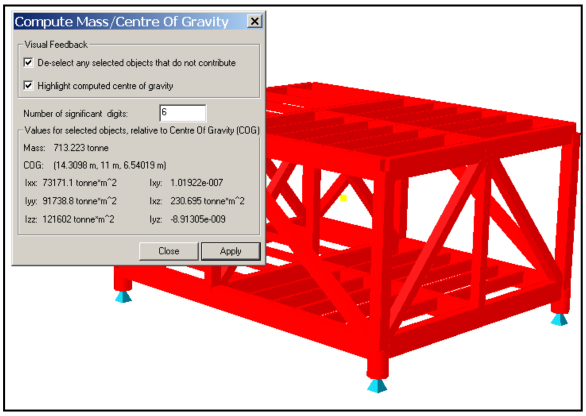
It is also possible to increase the graphics resolution of a plot. When saving a graphics file, you are prompted for such. The example below shows that the graphics resolution is increased with a factor of 3 (it is sufficient to multiply with 3 in this case, since the aspect ratio is locked).

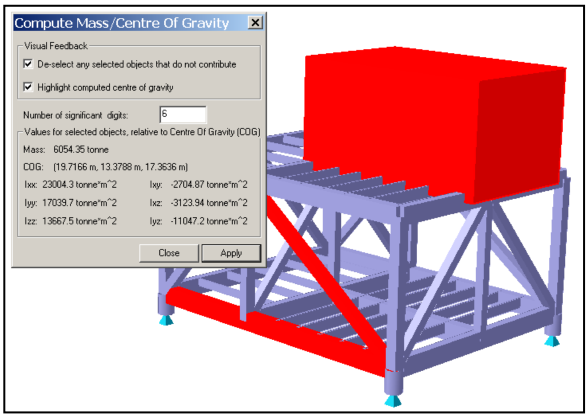


When activating the proportional fonts option, the fonts will be scaled accordingly to increase in graphics resolution.

3.1.14 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.

1	Group	Mass [tonne]	About	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	IXX [tonne*n	IYY [tonne*n	IZZ [tonne*m	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							

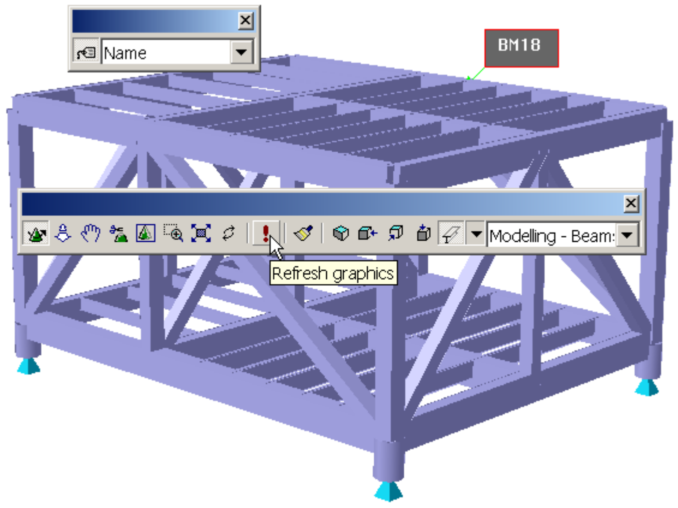
BeamOff / BeamLoc / Supports / Sections / Thickness / Materials / **Masses** / GlobMass / Bear

1	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	12.500	28.000	28.000	Straight Beam
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	BM24	3.1867E+00	5.500	7.340	12.900	11.000	11.000	Straight Beam

BeamLoc / Supports / Sections / Thickness / Materials / Masses / GlobMass / **BeamMass** / Se

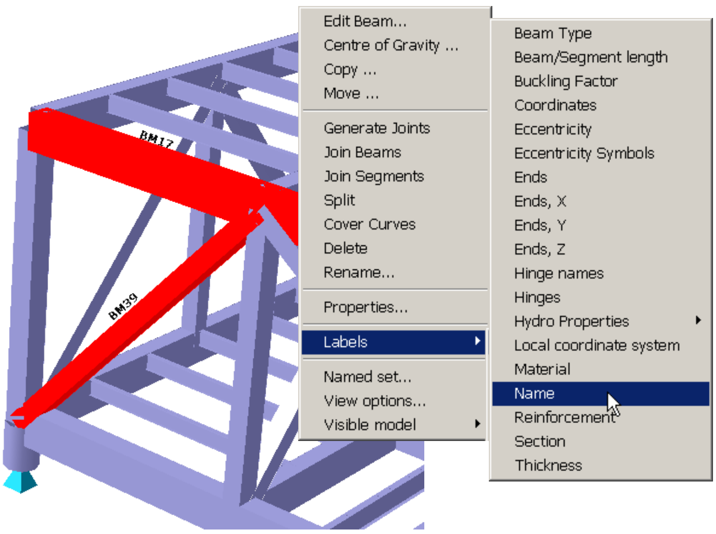
3.1.15 Labelling

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.

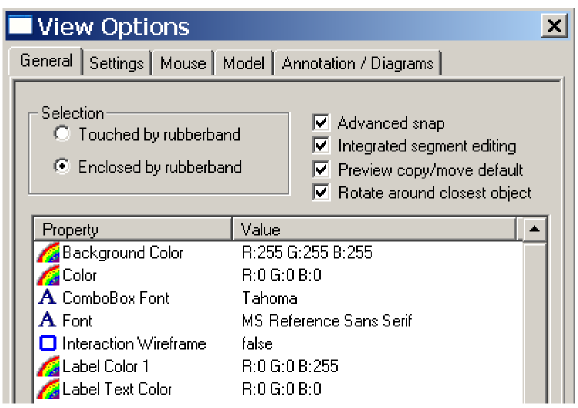


To the left, labelling is activated from the context sensitive menu by selecting an object and then RMB. The picture illustrates the available labels for a beam.

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***.

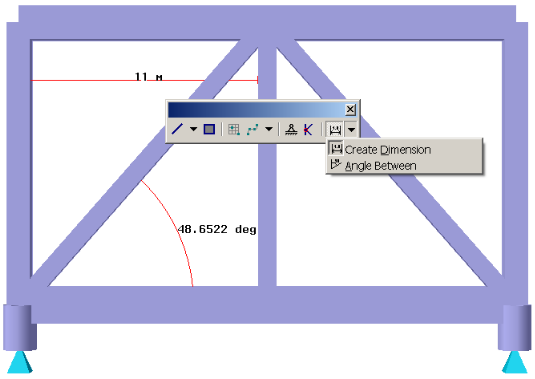


You control the appearance of labels from the ***View|Option|General*** menu.

- 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*.

3.1.16 Show length and angle

You may show length and angle graphically.

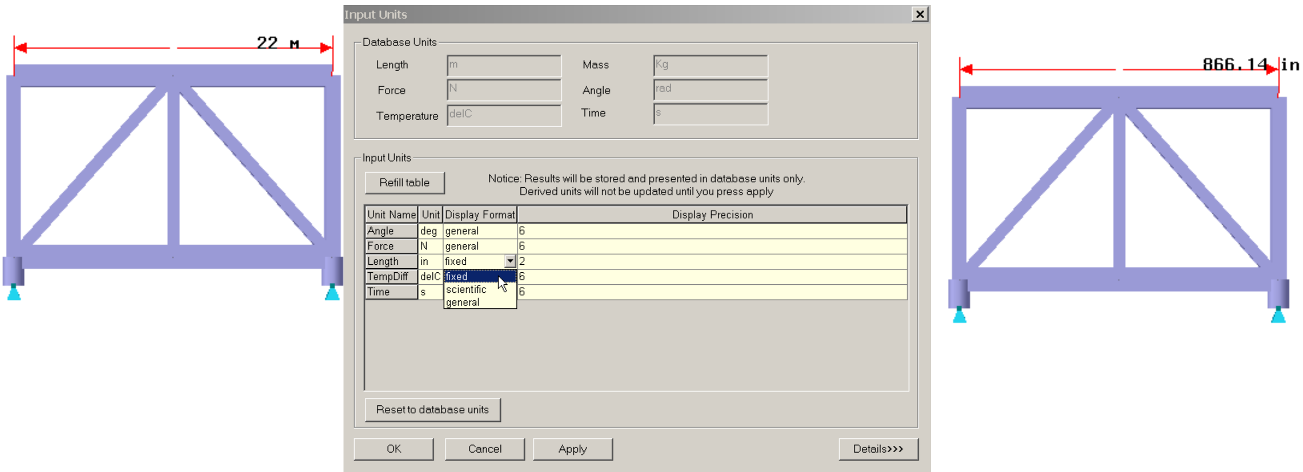


In this example length is found by activating *Create Dimension* and click on the two end points of the actual beam and drag the mouse to where you want the result to occur.

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

3.1.17 Working with units

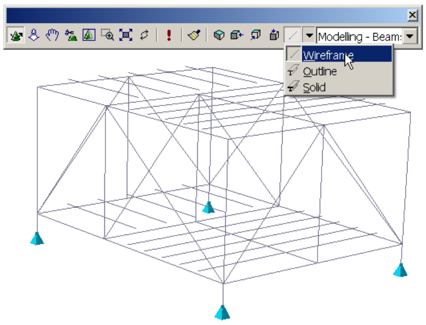
The database units are set when creating a new workspace. The interface files and results created are always given in the database units. The database units can not be changed later for a given workspace. You may, however, choose to work with different input units at different times. For example you 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.

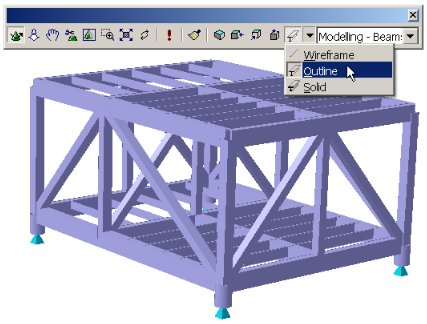


If you change your mind wrt. database units you can change the parameters in the command GeniERules.Units.SetDatabaseUnits and re-run the command input file. Note that when recreating a model based on an existing command input file you must skip setting the database units from the File|New workspace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

3.1.18 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***.

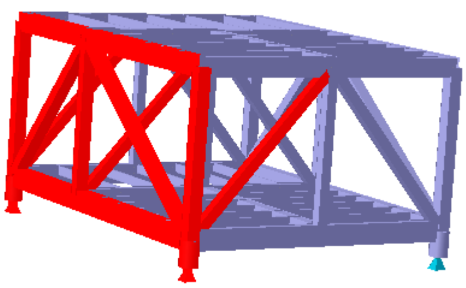
3.1.19 Customise the tool buttons

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.

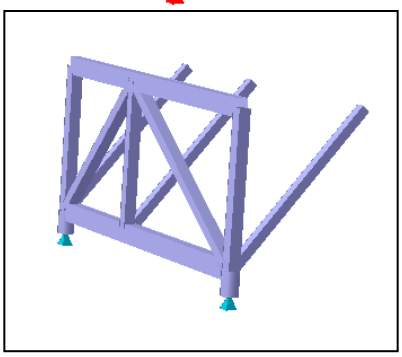
3.1.20 Work with visible model

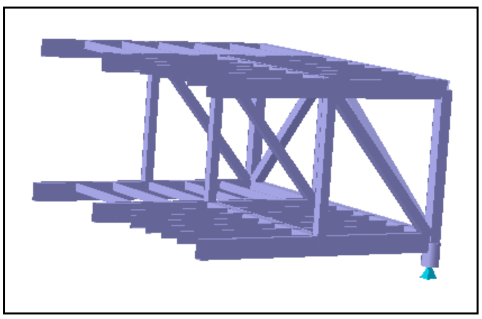
You may choose to view parts of the model when modelling or interrogating the model. You may choose to use Named Sets or make up your selection from time to time. The examples below show how you can manipulate your view, either by manual or global selection/deselection.

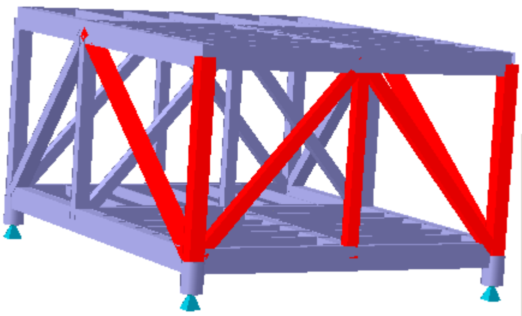


A number of beams have been selected to form the view to the left. By forcing the context sensitive menu, choose ***Visible Model|Show selection only*** to produce the picture down to the left.

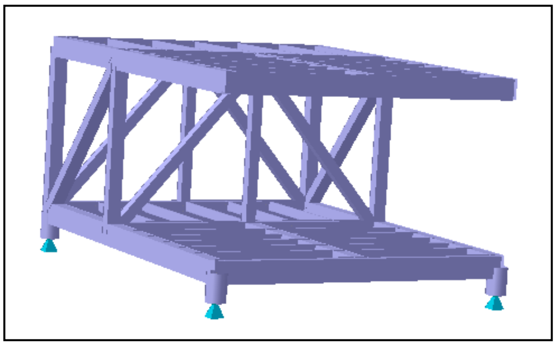
The picture down to the right has been generated by choosing ***Visible Model|Show complement***.You view the complete model by ***Visible Model|Show all***.

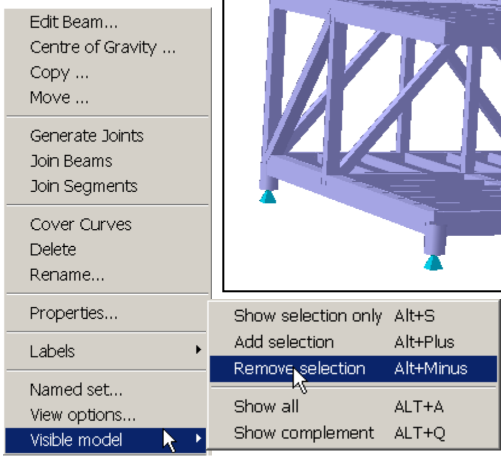






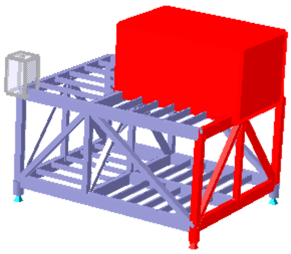
By using the commands showed below (**Visible model|Remove selection**) the picture down to the right is created.

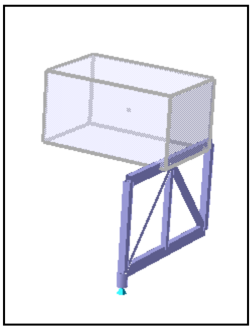




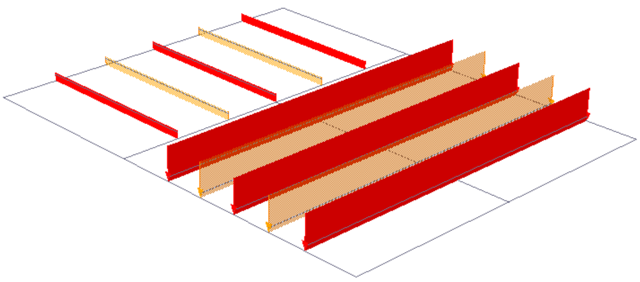
You should observe the short commands for these commands. Please note that when using a laptop you may need to change to mode *num lock* to be able to do ***Alt+Plus*** and ***Alt+Minus.***

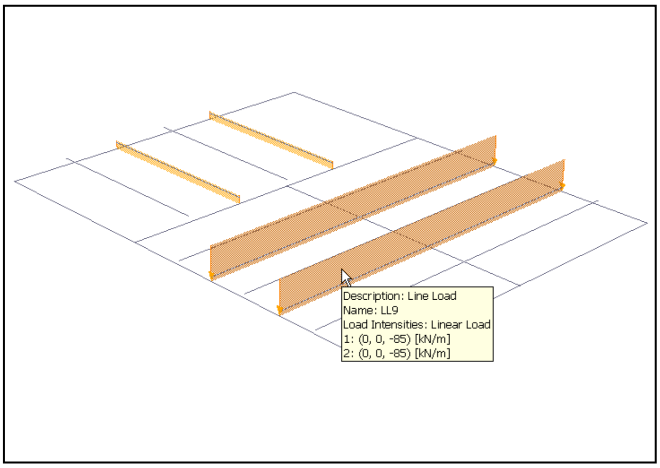
The features for visible model work for all types of concepts. The example below shows selection of structure, equipments and support points (***Visible Model|Show selection only***).



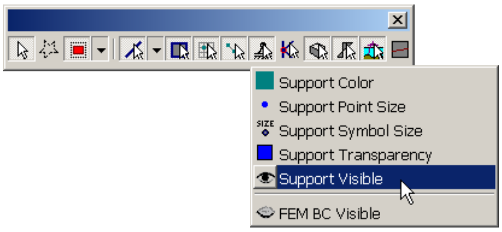


It is also possible to decide which explicit loads you want to visualise for a particular loadcase. In the example below two of the explicit line loads have been selected and by using ***Visible Model|Remove Selection*** they are not shown.





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.



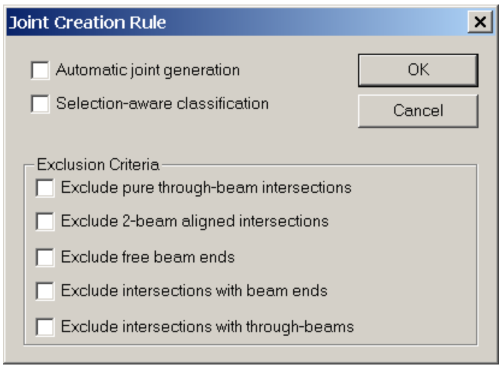
In this example boundary conditions will be shown except when displaying the finite element mesh. Observe that changing settings here is the same as changing from ***View|Option|Settings***.

3.2 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.

3.2.1 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).

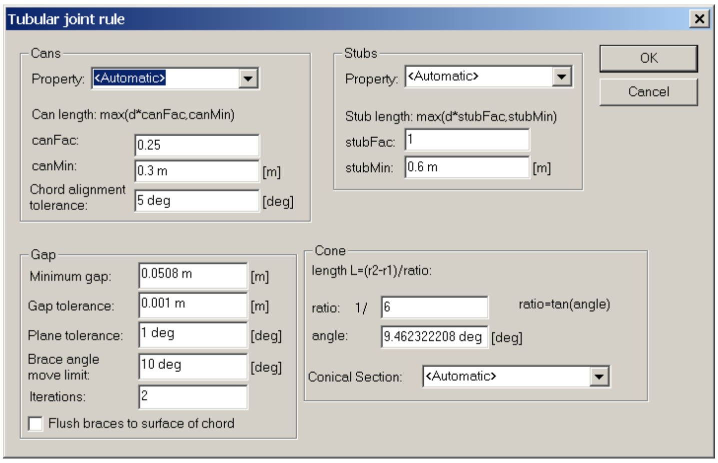
The fourth option excludes joints at all locations where beams start or ends.

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

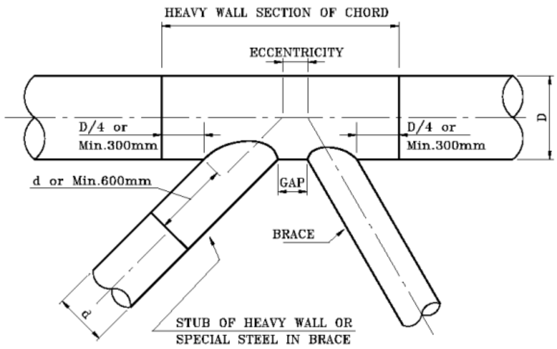
Joints are also used for referencing structural joints (tubular and non-tubular) to have a unique name that will not change when the model is changing. This means you can refer to unique names e.g. when doing code checking in Framework.

3.2.2 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 NORSOK. 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.

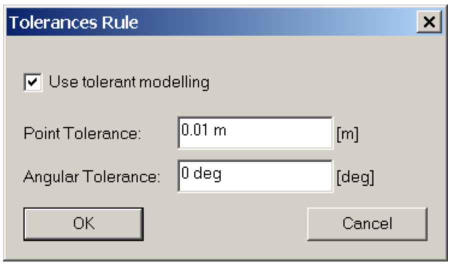


The parameters are – see figure above for references:

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.

3.2.3 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.***



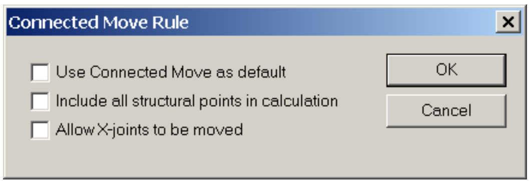
The first option ensures that one topology point is being used when input deviation is less than the tolerance. For example the end of a beam is located at (0m, 0m, 0m) and one end of a new beam is inserted at (0m, 0m, 0.005m) the first point is being used. In other words, the end point of the new beam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

3.2.4 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.

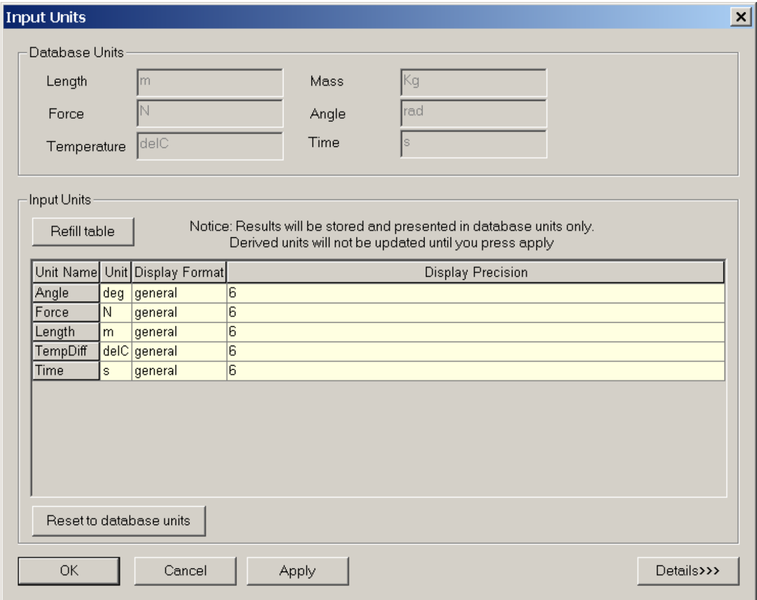
The second option will ensure that belonging parts are part of a move operation, see examples below.

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.

3.2.5 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***.



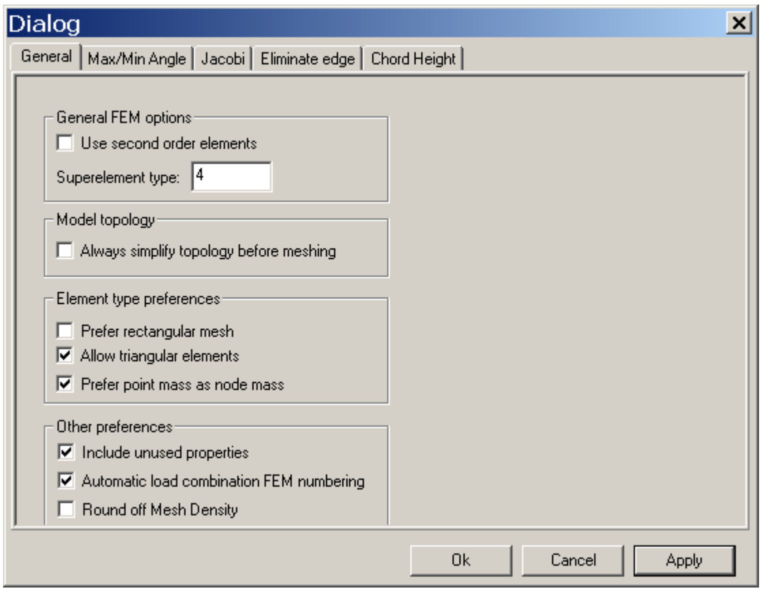
When changing any of these units, the new units will be used in e.g. input fields and on dimensions. They will appear according to the settings chosen.

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.

3.2.6 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).



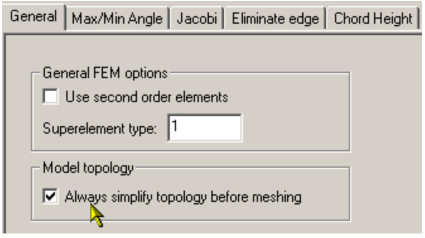
You may also specify that you want to use 2nd order finite elements when creating the finite element model. 1st order elements are the default.

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.

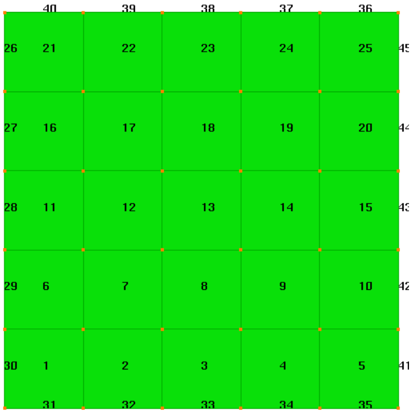
For more details on loadcase numbering see Chapter 3.8.5.

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.

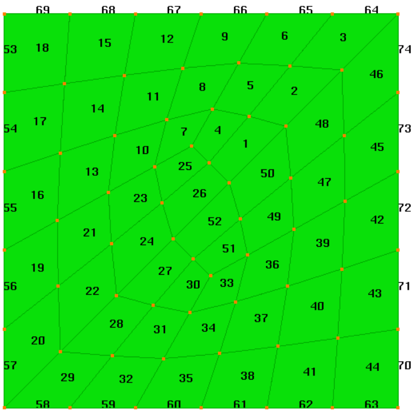


You may also instruct GeniE to always perform a simplify topology prior to making a finite element mesh, you may also do simplify topology manually from ***Tools|Structure|Geometry|Simplify Topology***.

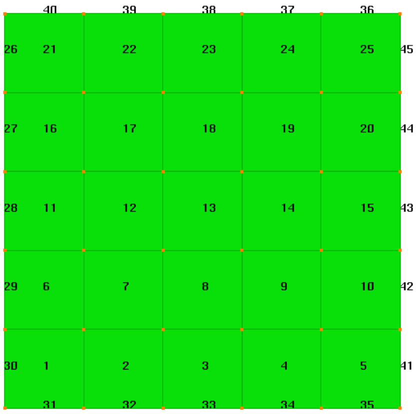
The operation will remove unnecessary topology points and lines in the model that will simplify the finite element model. It works for both straight beams and planar plates. In the example the finite element mesh is showed for a plate surrounded with beams, then intersected with a feature edge from lower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



Original finite element numbering



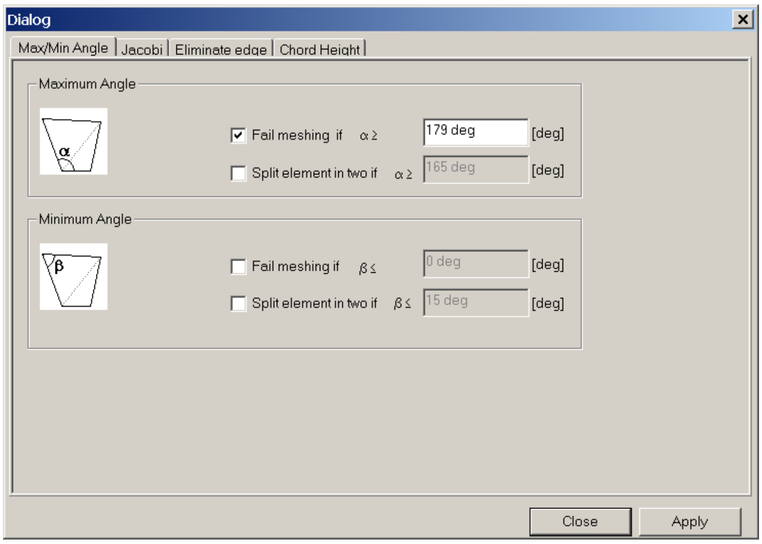
Finite element numbering after inserting feature edge



Finite element numbering after removing feature edge and simplify topology

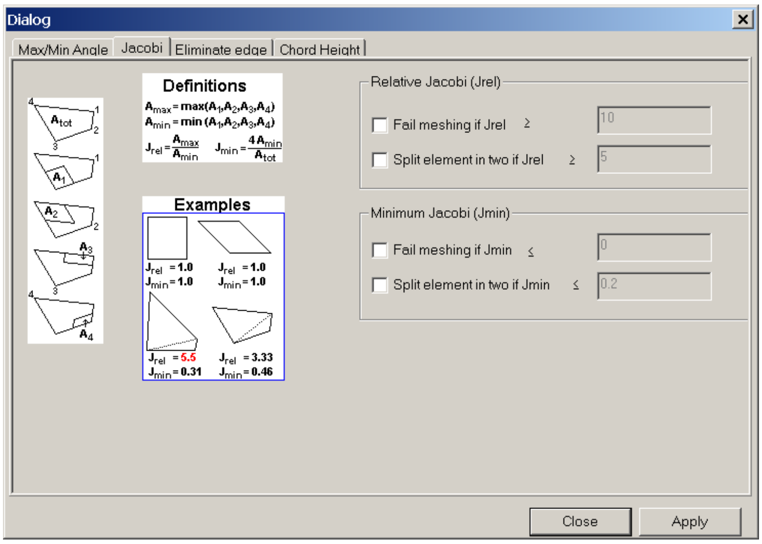
As can be seen, the simplify topology operation leads to a finite element mesh identical to the original mesh.

When you have specified a certain mesh density GeniE will ensure that the FE mesh created contains elements less 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.

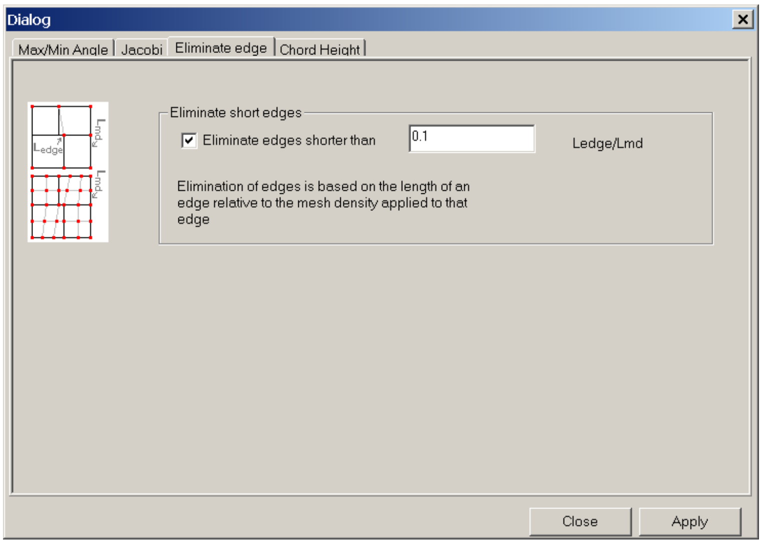


The settings here will instruct the program to fail meshing and inform you if Fail Meshing Angle is larger or smaller than given criteria.

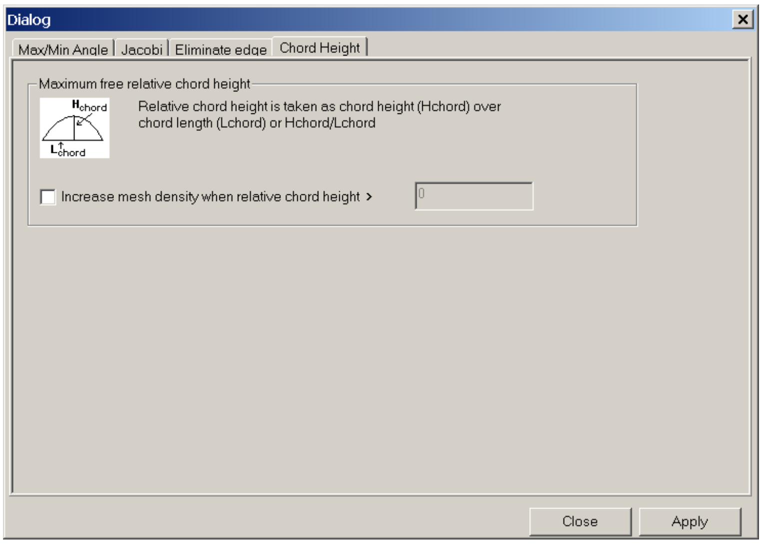
Similarly, you can instruct the program to split a quadrilateral element into 2 triangular elements if the element angle is larger or smaller than given criteria.



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.



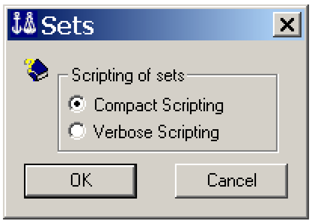
If there are curves in your model, e.g. when you have used a punch operation, the default mesh density on the face may not be fine enough to get a finite element mesh that properly represents your model.

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.

3.2.7 Rules for scripting of sets

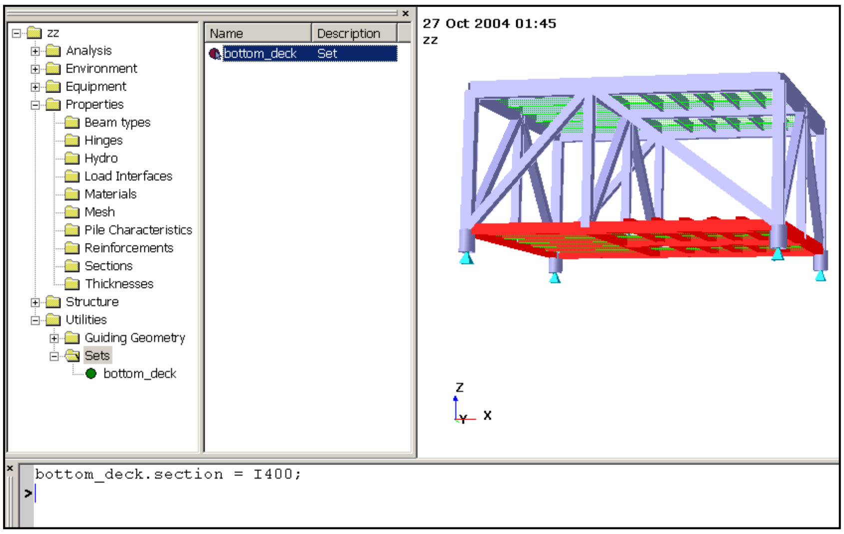
It is possible to instruct GeniE on how to script certain operations of sets. This is available from the ***Edit|Rules|Sets***.



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.

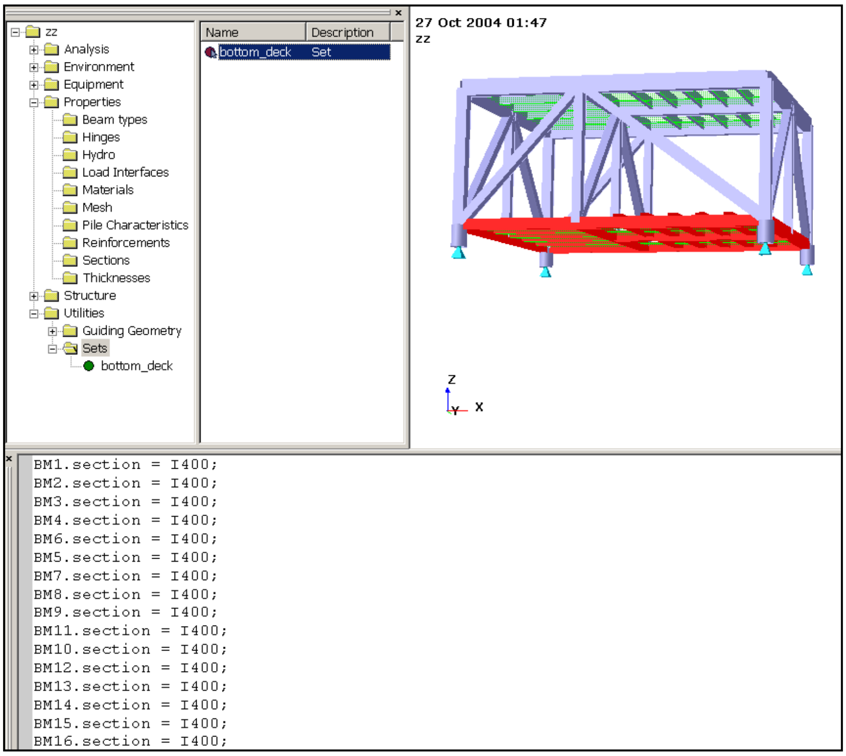
The *Verbose Scripting* enforces one scripting command per objects in the selection.

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



The *Compact Scripting* is chosen.

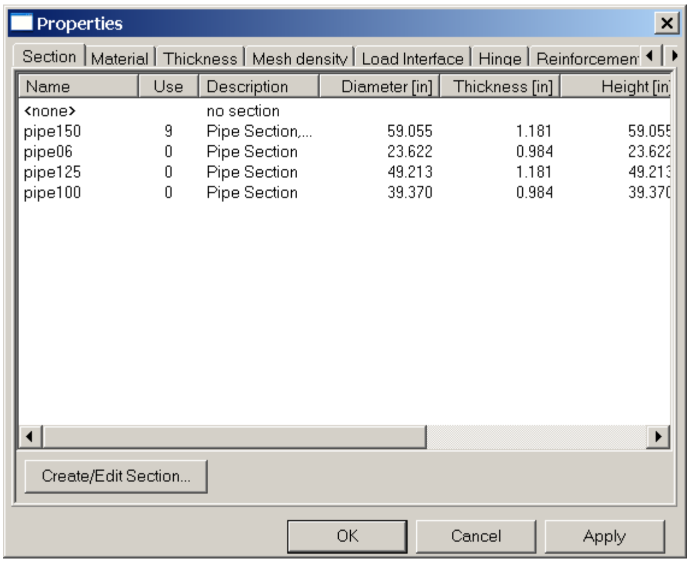
When selecting the set *bottom_deck* and applying the section type *I400* creates one scripting command in the journal file.



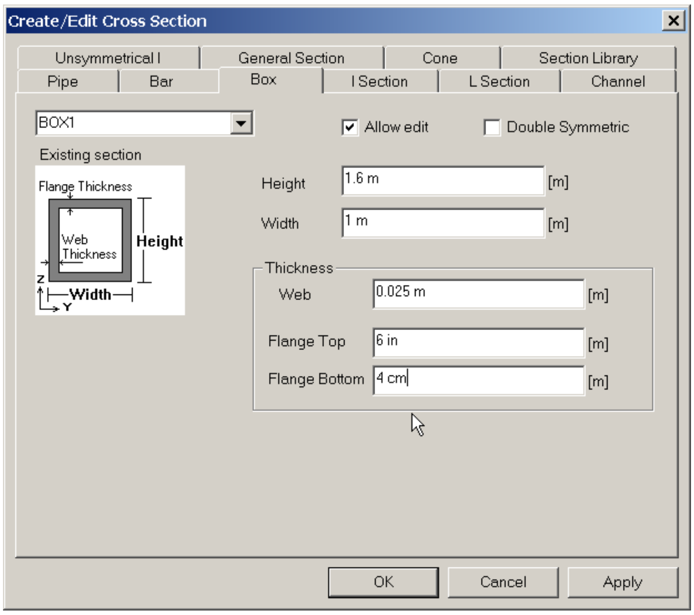
The *Verbose Scripting* is chosen. When performing the same operation as above, a scripting command is generated for all beams part of the set.

3.2.8 Property libraries

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



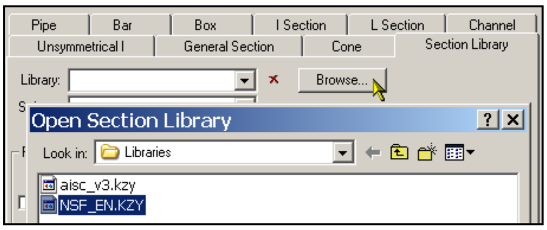
The example to the left shows that a total of 4 section types have been created. A common way of defining libraries is to start creating the properties fro the GUI and continue adding the remaining properties in a journal file. This journal file can be imported to new workspaces to create your library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

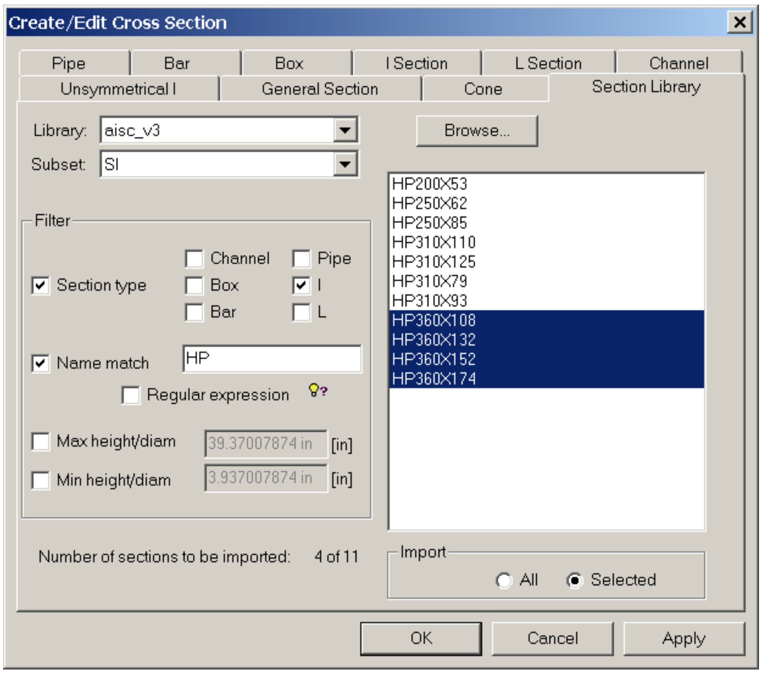
It is also possible to vary the flange thickness for I-section and Box, to the left is shown a Box with different flange thicknesses.

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*.

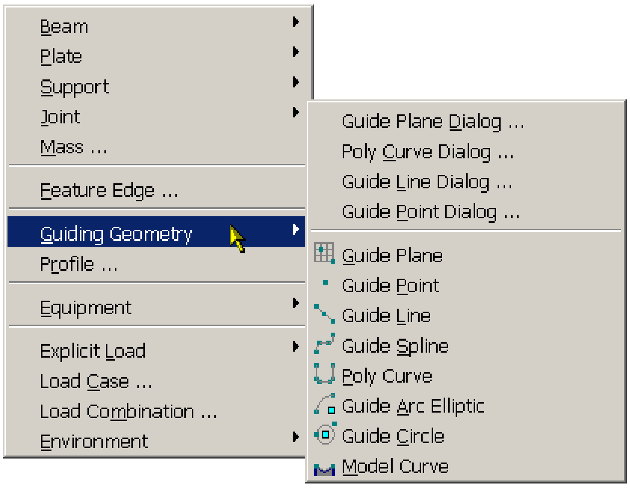
3.3 Guiding geometry

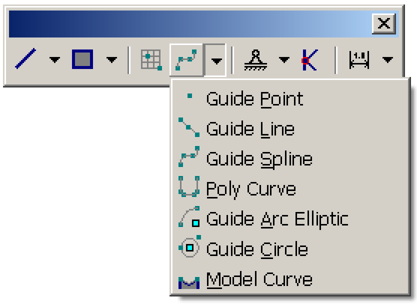
When modelling structure you refer to topology points, for example start and stop positions of a beam or the four corner positions for a plate. A topology point is automatically created where structural parts intersect or when you insert (snap to) structure referring to guiding geometry.

Guiding geometry is thus a mean to ease modelling whereby known coordinate positions are defined for later usage. The guiding geometry is not connected to any structure and the finite element mesh created is also independent of the guiding geometry.

This chapter lists the available guiding geometries. 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*.

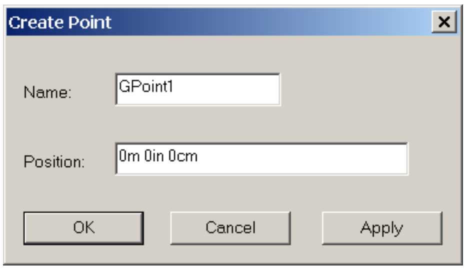
Inserting guiding geometry is available from the pulldown menu and the toolbar (you may also insert directly from the browser area), see below.

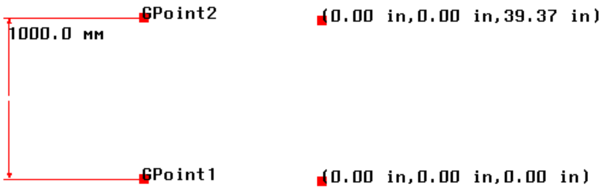




3.3.1 Guide point

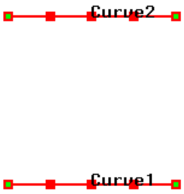
Two guide points are defined below and verified graphically. Observe the mix of units in the input field. The guide point is being used as reference point for modelling beams, plates and other guiding geometry.



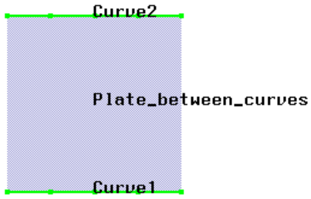


3.3.2 Guide line

Two guidelines are defined using guiding points as start and stop positions. The modelling is pure graphic, i.e. snap to guide points. A guide line is defined with four equal parts, i.e. the guideline has 5 snap points in total, see picture below for details.

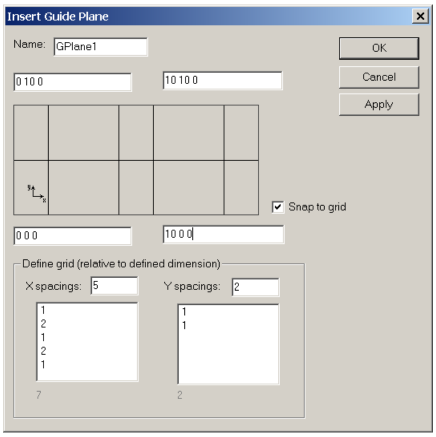


A guide line is being used to insert a beam, feature edge, support curve, or as a boundary for plates. In the example below a skin operation has been done between the guidelines Curve1 and Curve2.



3.3.3 Guide plane

Guideplanes are often used when modelling regular structures like a topside, a module frame, or a jacket. Very easily you can create a grid similar to the system lines of a structure.

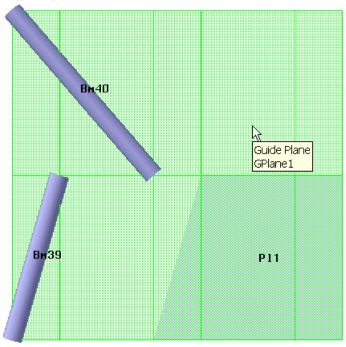


You specify the corner coordinate values (given according to the global axis system) and number of grid-lines in local x and y direction.

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

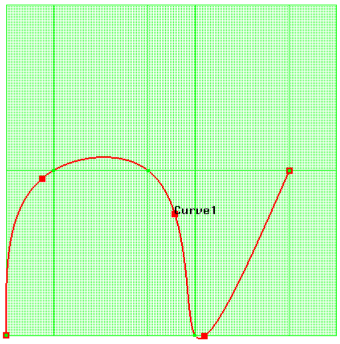
A guideplane may be oriented in any plane and it may very well have a trapezoidal form. It is required that it consists of four corner points.

Structure (beams and plates) may now be modelled simply by clicking on the snap points defined by the guide plane, see below.

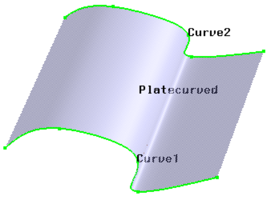


3.3.4 Guide spline

To insert a guide spline you need snap points available either from other existing guide geometries or structure. When specifying a guide spline you need to enter at least three points to establish the spline. You terminate the input sequence by double-clicking the end point. A guide spline is automatically divided in 4 equal parts, meaning that is has a total of 5 snap points. The spline below has been created by clicking on 5 snap points (double click on the last point).



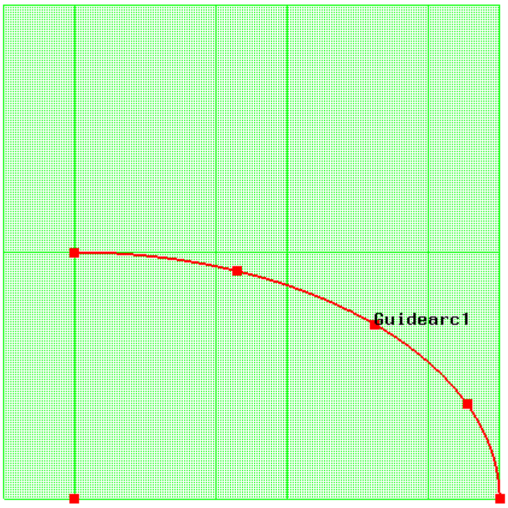
A guide spline is being used to insert a beam, feature edge, support curve, or as a boundary for plates. In the example below a skin operation has been done between the guidelines Curve1 and Curve2.

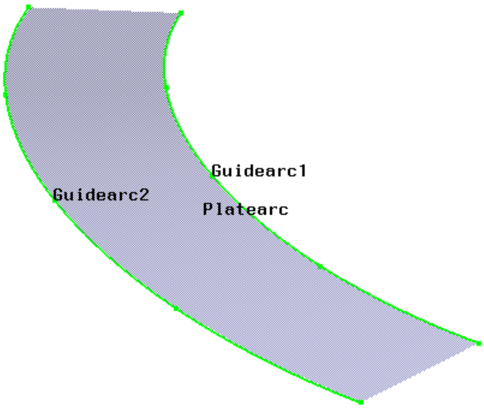


3.3.5 Guide arc elliptic

A guide arc elliptic is defined by three points, the first point being the origin, the second and third defining the start and end points of the arc. The plane is defined by the three points. The three points may be either from existing guide geometry or structure. A guide spline is automatically divided in 4 equal parts, meaning that is has a total of 5 snap points. In the example below a guide arc elliptic has been created by clicking three points specified by a guide plane.

A guide arc elliptic is being used to insert a curved beam, feature edge, support curve, or as boundary for plates. A skin operation is done to create curved plates between Guidearc1 and Guidearc2.

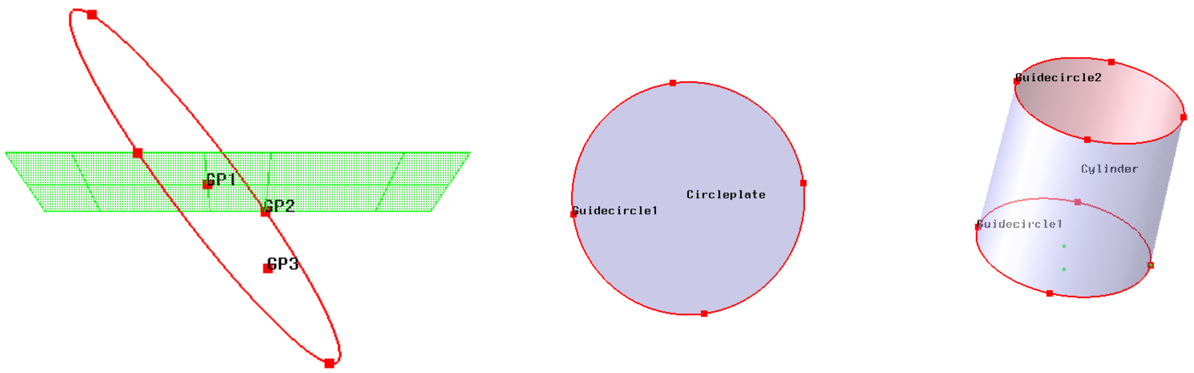




3.3.6 Guide circle

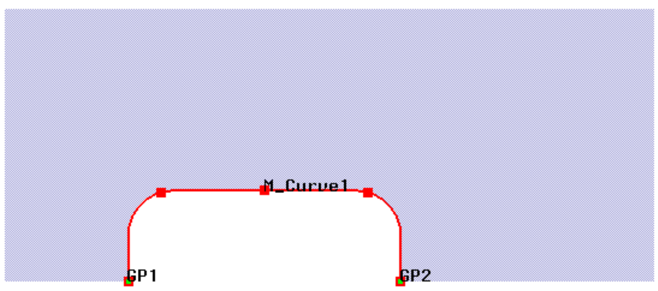
A guide circle is a variant of the guide arc elliptic. The difference is that the circle is defined by origin, a point defining the radius and a third point defining the plane. This command will always create a full circle divided into 4 equal parts and 4 snap points.

A guide circle may be used to insert a curved beam, feature edge, support curve, or as boundary for plates It can also be used to create a curved plate inside the guide circle (a cover operation). The example below shows a guide circle defined with GP1 (origin), GP2 (radius) and GP3 (together with GP1 and GP2 defining the plane). The example also shows a plate defined by a cover operation and a cylinder made from a skin operation between Guidecircle1 and Guidecircle2.

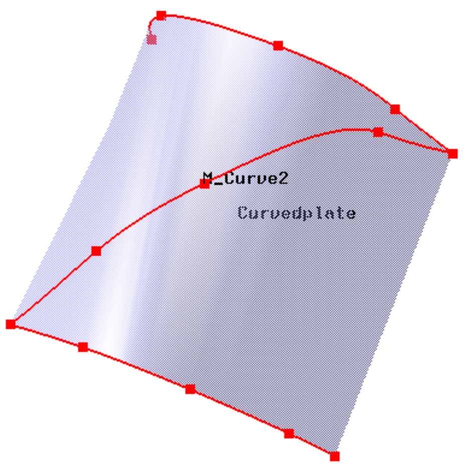


3.3.7 Model curve

A model curve is a curve that follows a topological edge path (can be several edges) between two topology points, see below for typical examples. A model curve can be used to define a curved beam or a feature edge. A model curve is per default split in 4 equal parts and has thus normally 5 snap points.



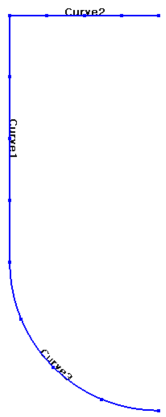
The plate at left has been punched by a profile. A model curve (M_Curve1) has been inserted by clicking GP1 and GP2. The command will automatically find the shortest way between the points following the topological edge.

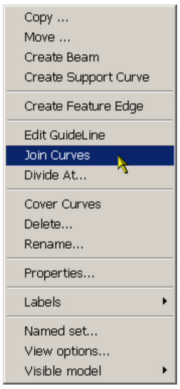


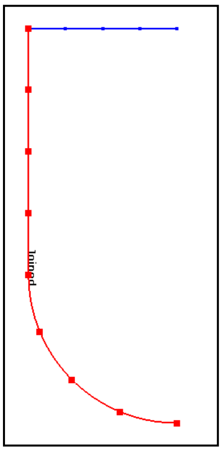
In the example at left a curved plate has been created from skinning two guide splines. A model curve (M_Curve2) has been inserted from bottom corner to upper opposite corner. The curve will follow the intersection line between the curved plate and a plane defined by start and end point based on the local plate normal at these points.

3.3.8 Join and divide guide lines

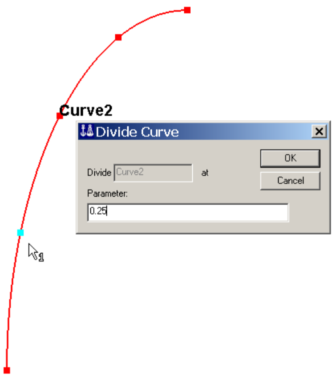
For complex structures it may be necessary to make up a guide line from several guide lines. This is done by selecting the guidelines, force the context sensitive menu, and choose ***Join Curves***. The curves must have a continuous transition to be joined. The example below shows continuous curves joined (Curve1 and Curve 3) to *Joined*. It is not possible to join Curve1 and Curve2.

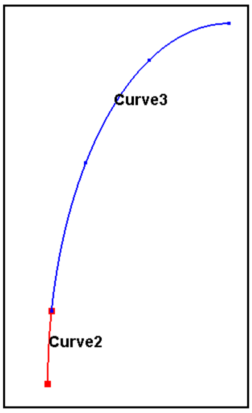






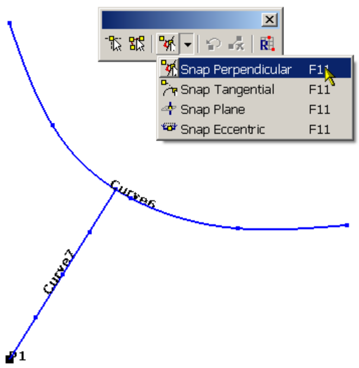
It is also possible to divide a guide line using a parameter split option. The parameter may be given explicitly or found by clicking on an existing point along the line.





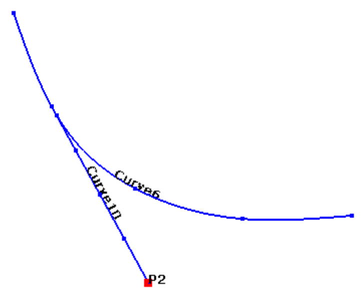
3.3.9 Guide lines using advanced snap functionality

It is also possible to define a guide line and make it perpendicular or tangential to existing guide lines (line or curve).



In the example to the left *Curve6* has been defined as a guide spline and P1 as a guide point. *Curve7* is perpendicular to *Curve6* and is modelled in the following sequence

1. Modelling modus *Guide Line*
2. Start of line at *P1* (click on *P1*)
3. Activate snap perpendicular (use short command F11 or do it graphically, see picture to the left)
4. Click anywhere on *Curve6* and *Curve7* is now defined



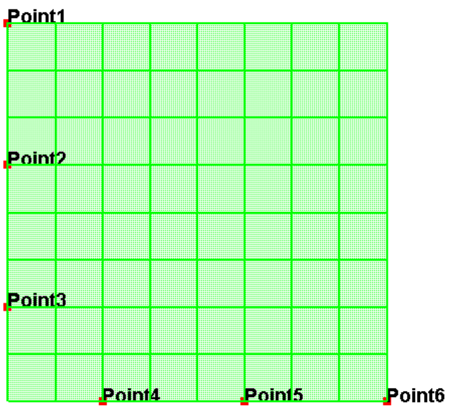
Similarly, Curve 10 is tangential to Curve6 starting at P2. The steps are the same as above, but using snap tangential in stead.

For the Snap perpendicular option, please refer to Chapter 3.4.11 for references.

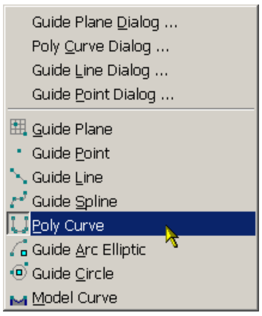
3.3.10 Polycurves

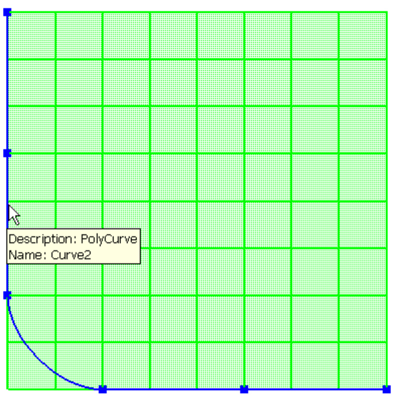
A polycurve is a continuous line whereby the individual parts of it are either straight lines or spline lines. The polycurves must be continuous and cannot have sharp angles between the individual parts. When modelling e.g. hull forms of floating vessels polycurves are efficient.

The following examples show how to build up a polycurve. The first example assumes usage of finding coordinate values from a guide plane and guide points while the second example shows how to do it by manual editing.

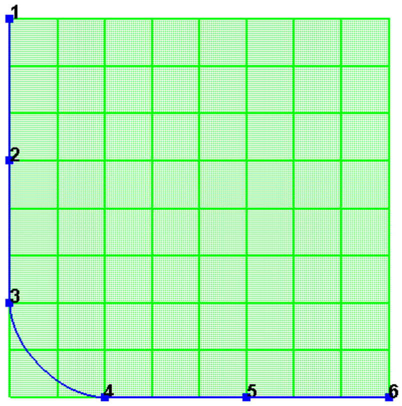


A guideplane has been used to define a total of 6 points to be referenced when making the polycurve. Use the command ***Insert|Guiding Geometry||Polycurve*** or a tool-button.

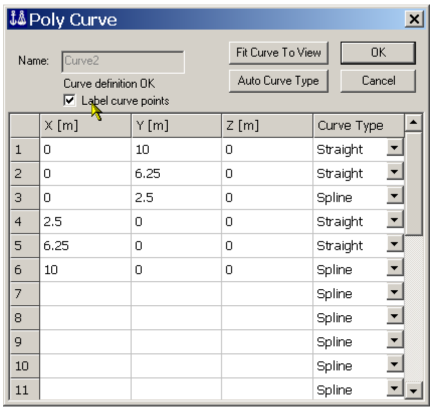


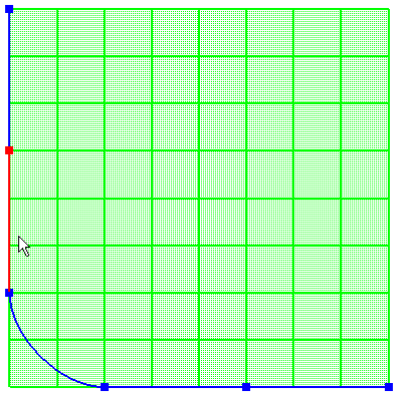


The polycurve is now defined by clicking sequentially on points from *Point1* to *Point6* (remember a double click on *Point6* to stop the input sequence).

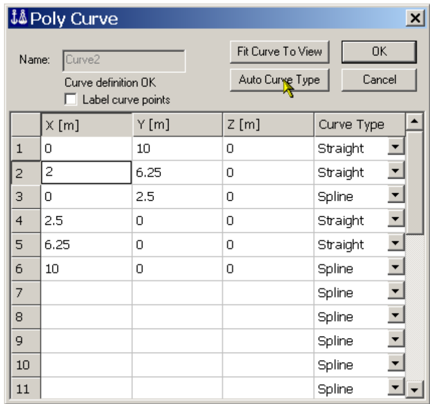


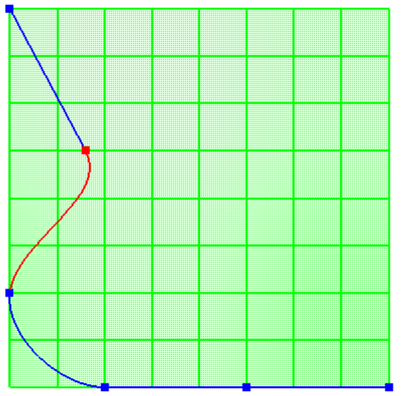
By selecting the polycurve, RMB and ***Edit Polycurve*** you may edit the curve. The polycurve's points are labelled from the edit menu.



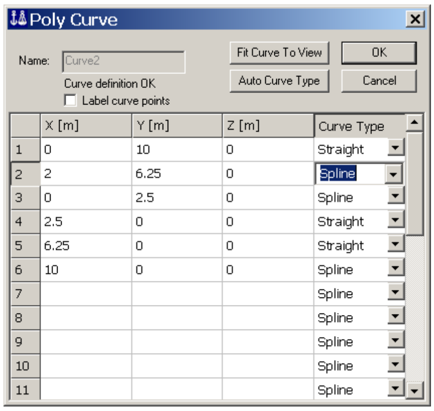


When clicking on one of the values in the table below, the actual segment of the polycurve is highlighted. It is thus easier for you to know which part you are editing.

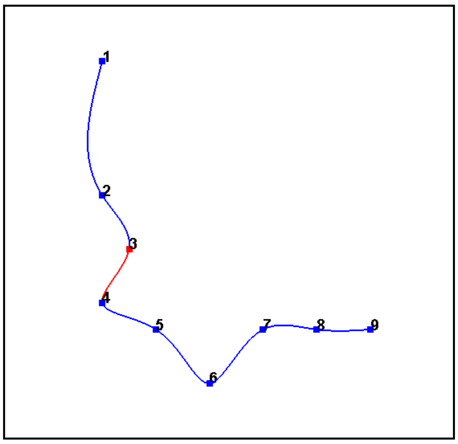




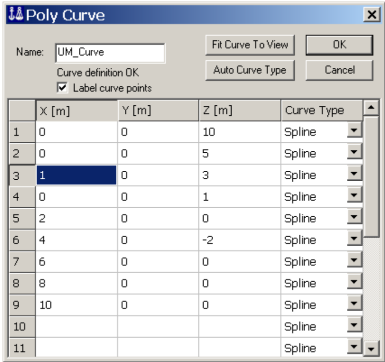
When clicking on ***Auto Curve Type*** as shown in the edit menu above, the actual segment is adjusted to its new end position as well as changed from Straight line to Spline.

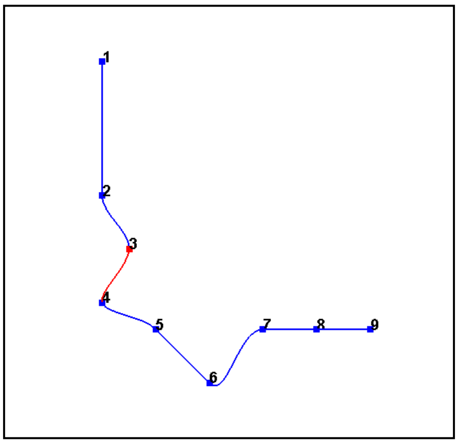


You may also create polycurves from manual editing. The command **Insert|Guiding Geometry|Poly Curve Dialog** you can define such curves without the need for referencing existing topology points. In case you define points that are outside the visible area, you may use the ***Fit Curve To View***.

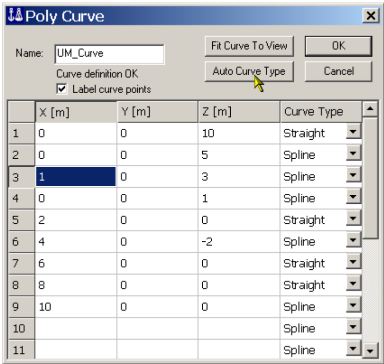


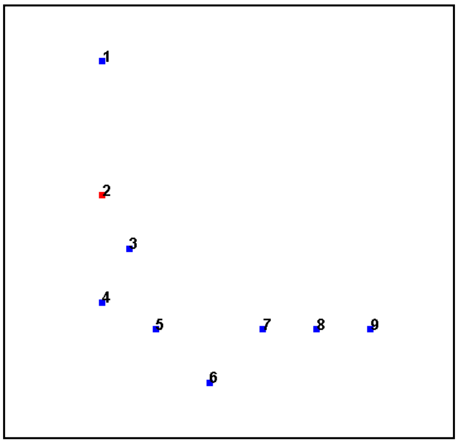
By editing the polycurve as shown below, the polycurve to the felt is automatically built up. Notice that in this case no line type (spline or straight) is specified – the default spline is used.



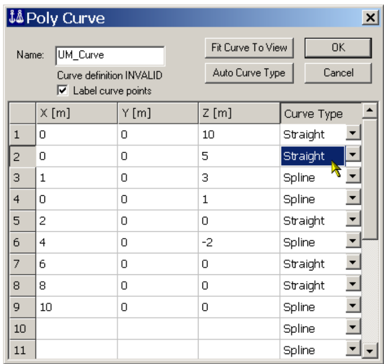


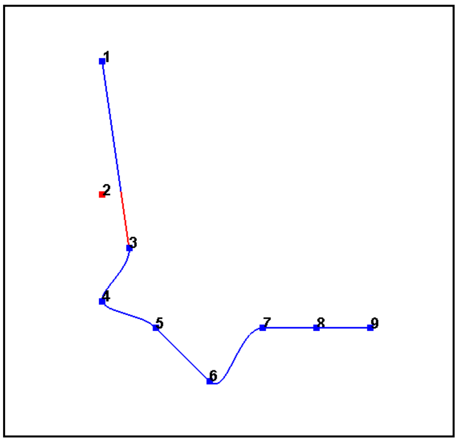
By using the feature for detecting the correct line types by using Auto Curve Type, some of the spline segments above are changed to straight line segments.



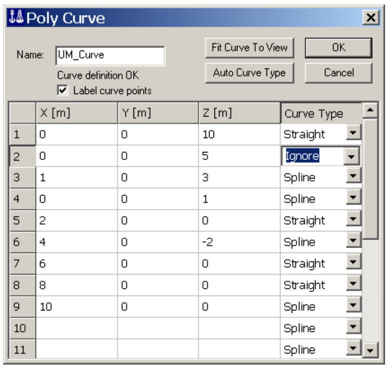


When changing the line type (in this case from spline to straight) and it is impossible to generate a continuous polycurve, GeniE will not make a polycurve, but highlight the incorrect line segment(s).





You may also wish to neglect certain positions when creating the curve. By selecting Ignore as shown below, point 2 is ignored when creating the polycurve and a straight line from point 1 to point 3 is created. If you click on Auto Curve Type it will restore the curve again.

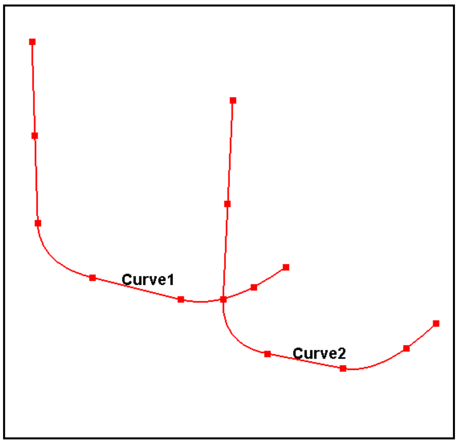


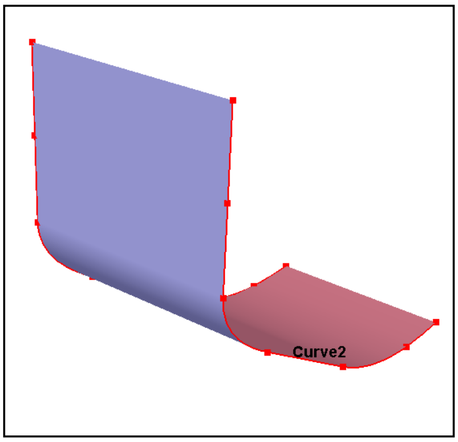
	X [m]	Y [m]	Z [m]	Curve Type
1	0	0	10	Straight
2	0	0	5	Ignore
3	1	0	3	Spline
4	0	0	1	Spline
5	2	0	0	Straight
6	4	0	-2	Spline
7	6	0	0	Straight
8	8	0	0	Straight
9	10	0	0	Spline
10				Spline
11				Spline

Also note that it s possible to cut and paste cell values from MS Excel – this may be beneficial if you have existing values for a hull form in form of an offset table. By using the scripting language you may create your own translator from an offset table to GeniE journal input file format.

```
//
// Example on a journal file to create 2 identical polycurves, but at different z-levels.
//
Curve1 = PolyCurve();
Curve1.clear();
Curve1.addPoint(Point(0, 10, 0), ggStraight);
Curve1.addPoint(Point(0, 8, 0), ggStraight);
Curve1.addPoint(Point(0, 6, 0), ggSpline);
Curve1.addPoint(Point(1.6667, 4, 0), ggStraight);
Curve1.addPoint(Point(5, 2, 0), ggSpline);
Curve1.addPoint(Point(8.3333, 1, 0), ggSpline);
Curve1.addPoint(Point(10, 1, 0), ggspline);
Curve1.rebuild();
//
//
Curve2 = PolyCurve();
Curve2.clear();
Curve2.addPoint(Point(0, 10, 5), ggStraight);
Curve2.addPoint(Point(0, 8, 5), ggStraight);
Curve2.addPoint(Point(0, 6, 5), ggSpline);
Curve2.addPoint(Point(1.6667, 4, 5), ggStraight);
Curve2.addPoint(Point(5, 2, 5), ggSpline);
Curve2.addPoint(Point(8.3333, 1, 5), ggSpline);
Curve2.addPoint(Point(10, 1, 5), ggspline);
Curve2.rebuild();
//
// End of file
//
```

This example below shows a journal file that has been created by editing the file. The file will produce two polycurves when read into GeniE.





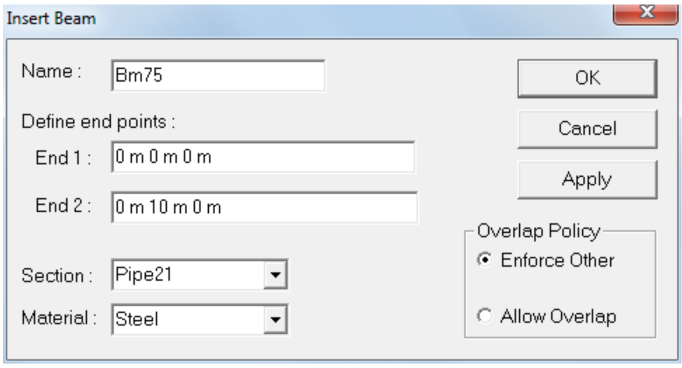
Two polycurves are created and it is now possible to create curved geometry by skinning operations, see Chapter 3.6.8.

3.4 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.

3.4.1 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.

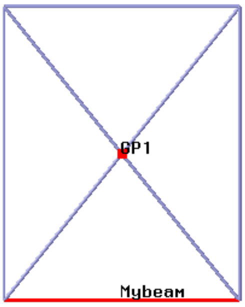


The pulldown menu ***Insert|Beam|Straight Beam Dialog*** opens up the dialog shown to the left. Here you may specify manually all required input, but you may also refer to existing snap points. You do this by clicking in e.g. input field for end 1 and then click graphically on the desired snap point. Relevant information is automatically filled in.

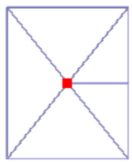
Overlap Policy

Enforce Other: If the new beam is overlapping an existing beam, the new beam will be cropped.

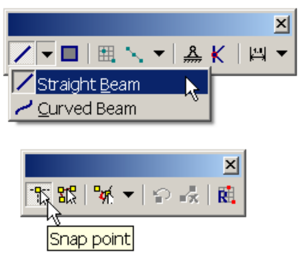
Allow Overlap: If the new beam is overlapping an existing beam, it will not be cropped. The existing beam and the new beam will overlap.



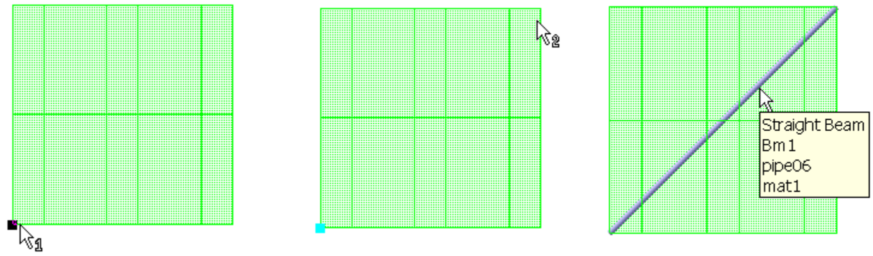
The figure at left shows Mybeam that was manually created together with 5 other beams. The outer beams and the X-beams have been inserted using existing snap points. Note that where the X-beams intersect, a new snap point is automatically created. In this case a guide point has been inserted to the same point. This point can now be used for connecting other members.



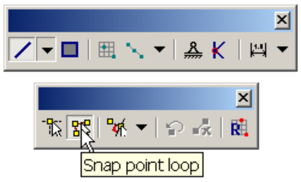
Provided you have snap points available, either from guiding geometry or existing structure, the most efficient way of defining beams is by graphic modelling. You enter graphic modelling of beams either from pulldown menu ***Insert|Beam|Straight Beam*** or from the toolbar *Object Types*.



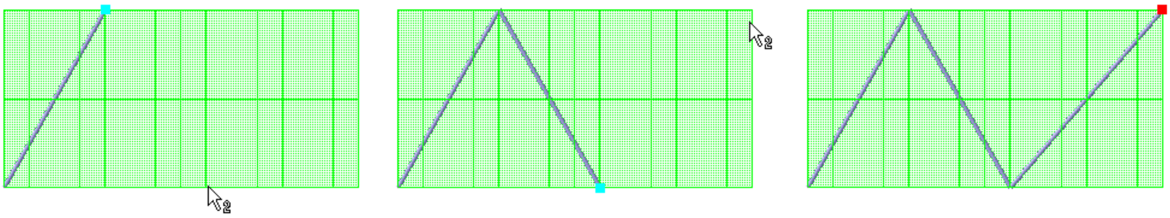
Make sure that the Straight Beam option is the current selection and that the Snap point button is activated. You may now start to model straight beams graphically.



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.

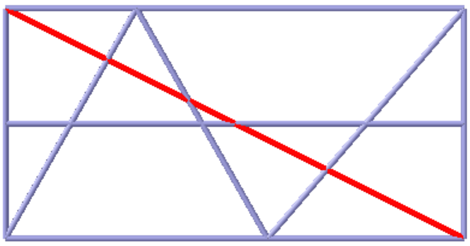


If you have regular modelling you may use a snap to point loop. In this case the end point of a beam is start point of next beam. This means you click end points for the second beam and onwards (or the program will prompt for end points only). The pictures below show an example of this.



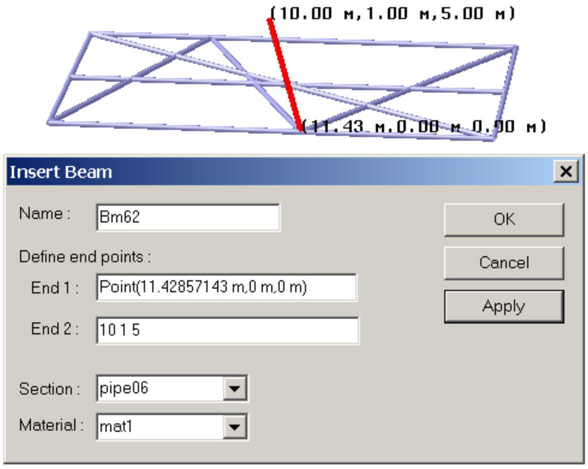
When working with intersecting beams, GeniE will calculate all intersection points and will manage the topology. You can therefore do top-down modelling, i.e. start with the rough and make details afterwards. If not, you may do bottom-up modelling or a combination of both approaches. It is possible to 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.



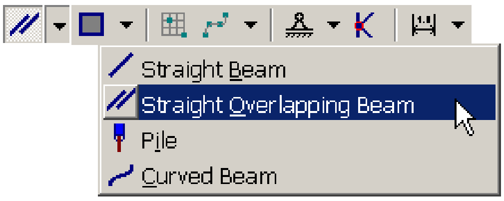
The beam to the left has been generated using the dialogue for inserting a straight beam by extracting coordinate values for End 1 from the existing structure and specifying co-ordinate values for End 2

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.

3.4.2 Overlapping beams

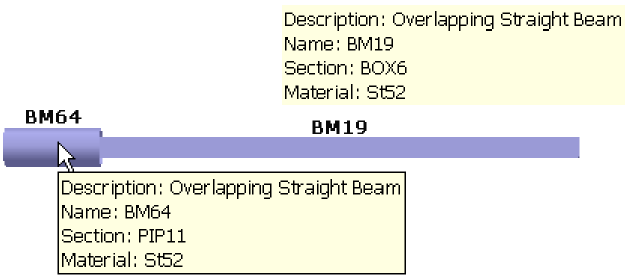
When inserting an overlapping beam you do the following steps:

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.

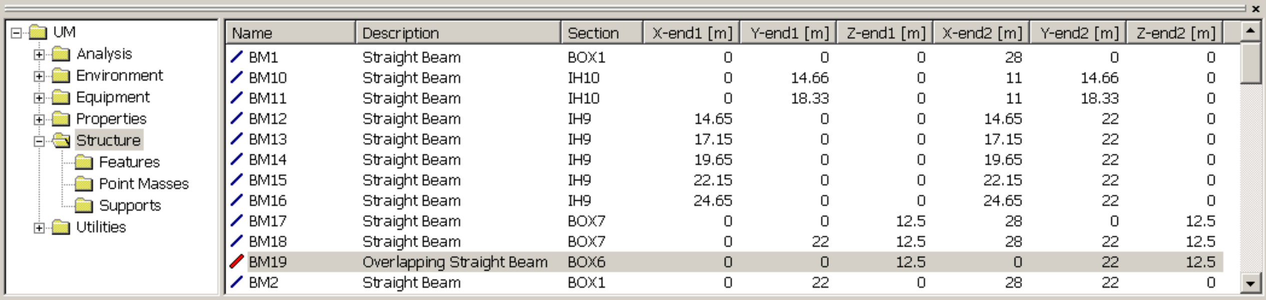
In the example below a beam with a box section is present prior to inserting a beam with pipe section. Furthermore, the mesh density is equal to both the inner and outer beam unless you specify otherwise. See Chapter 3.12.2 on how to make a finite element model.



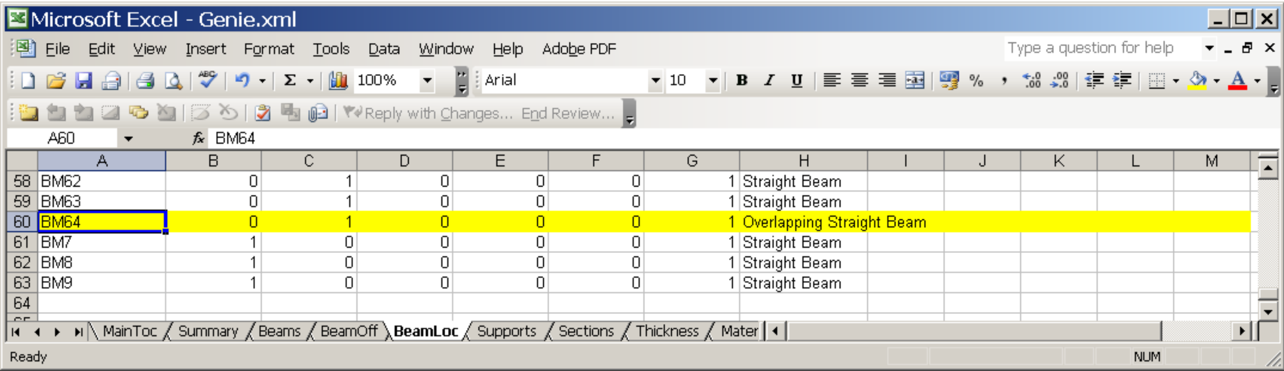
Notice that the original straight beam (BM19) also becomes an overlapping straight beam when the additional beam (BM64) has been added.

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 nodded depend on the mesh density).

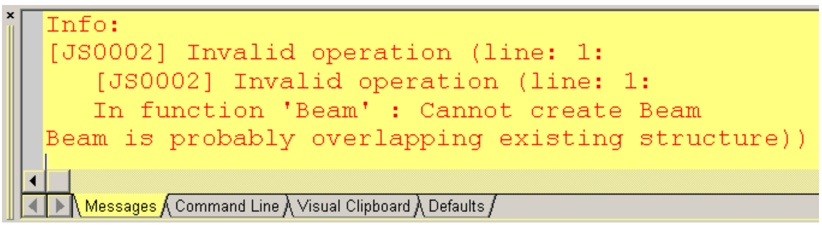
To verify the overlapping beams, you may either use the browser functionality or from the saved report. Both options are showed below.



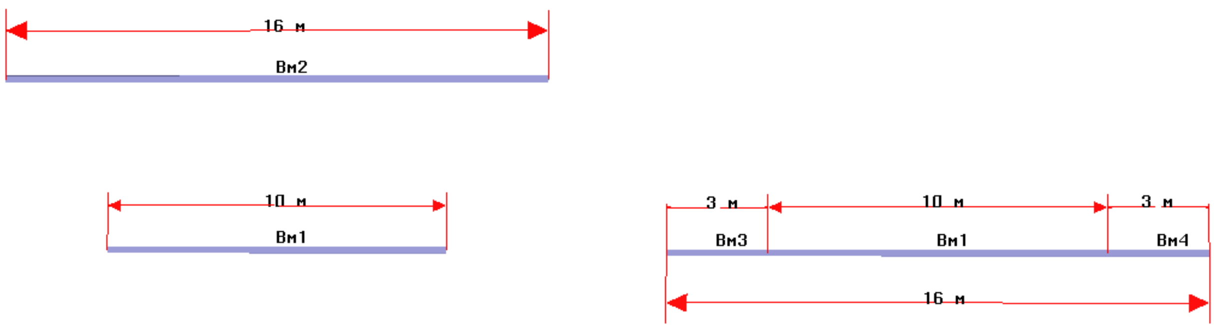
Notice to see the additional fields like coordinate values, you need to add the relevant fields to the view by selecting the folder *Structure*, push RMB and select *Fields*.



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:

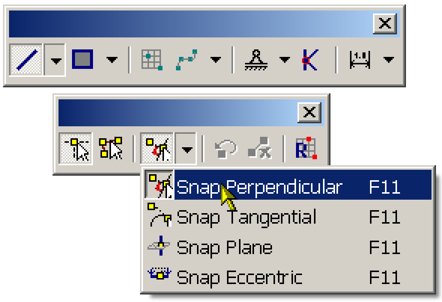


If you insert a beam that partially covers an existing beam, the delta part is inserted as new beams. The example below shows that Bm2 is inserted over Bm1 (Bm2 is 6 meters longer than Bm1). As can be seen, two new beams (Bm3 and Bm4) are created at each end of Bm1.



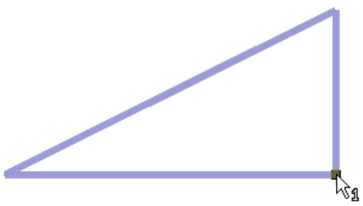
3.4.3 Modelling perpendicular beams

GeniE comes with a feature that allows for efficient modelling of beams perpendicular to another beam.

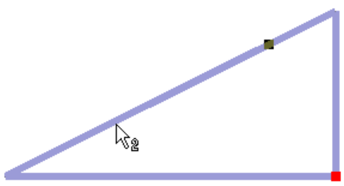


Make sure the Snap Perpendicular button is activated. You insert a perpendicular beam to another beam as follows:

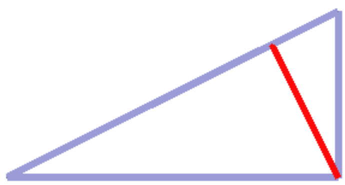
1. Click a snap point
2. Move the mouse over the beam you want to connect to and click
3. The beam is now inserted always perpendicular to the connected beam



Click end point 1

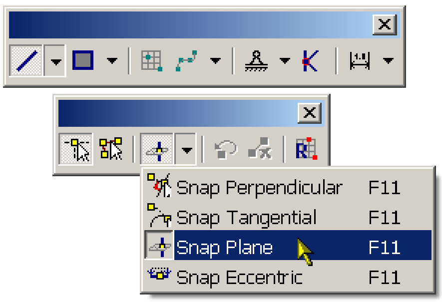


Move mouse over connected beam and click



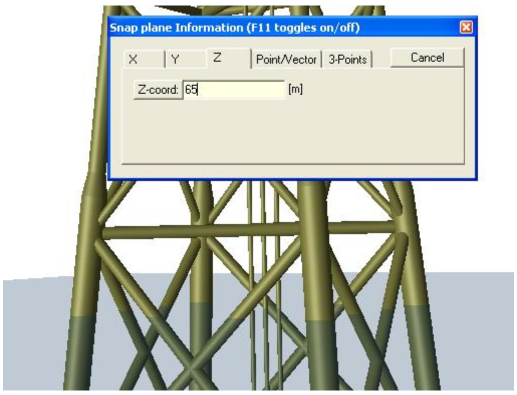
A perpendicular beam is inserted

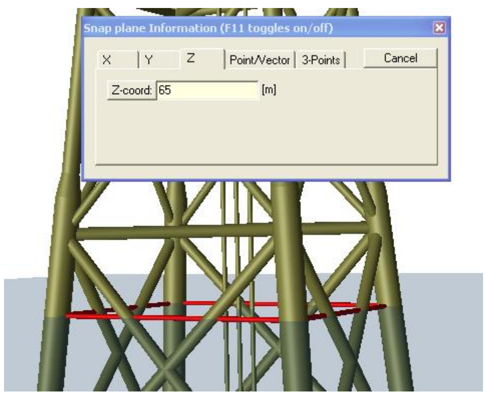
3.4.4 Snap planes



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.



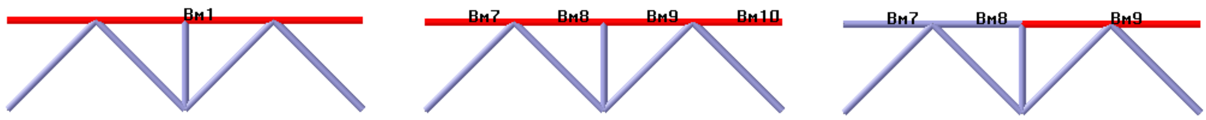


3.4.5 Split, join and divide beams

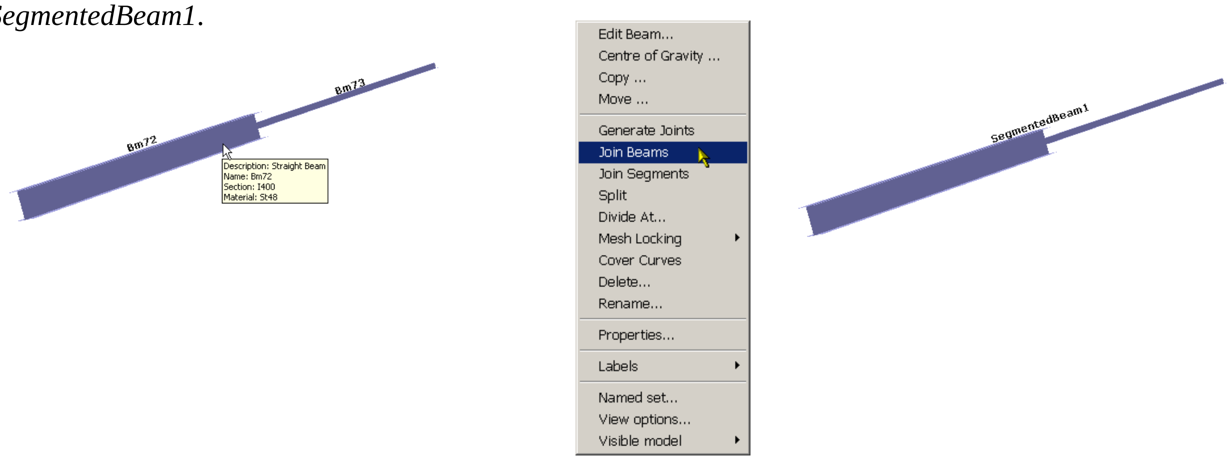
A continuous beam may be intersected by several beams. There may be reasons why you want the beams between intersections to be individual beams. To split such beam, you select the beam and use the command ***Tools|Structure Split*** from the pulldown menu. Alternatively you may do this from the context 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.

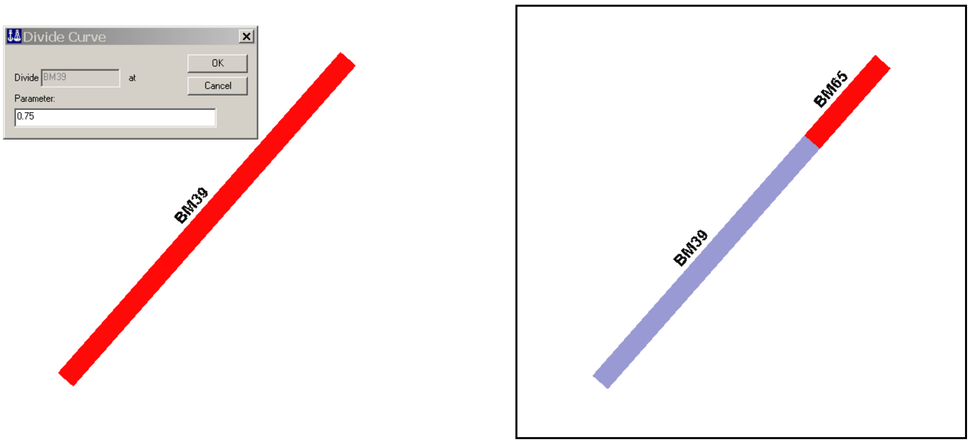
The example below shows the beam Bm1 split into minor beams (Bm7-Bm10), before Bm9 and Bm10 are joined.



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*.

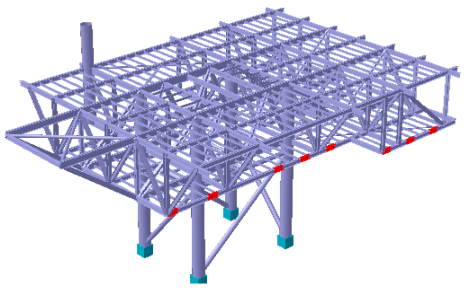


You may also split a beam at a known position by using the *Divide At* feature. The split position is given either as a length factor referring to the start point of the beam or by clicking an existing topology point. In the example below the beam is split ¾ of the length from end 1 and BM65 is created.

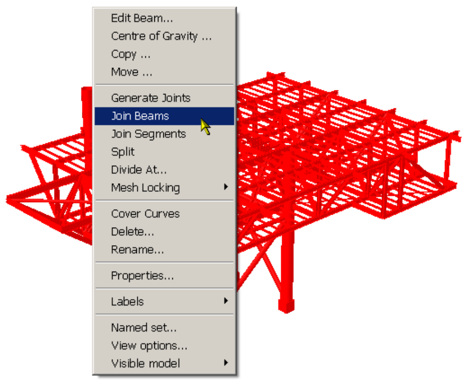


When joining beams GeniE will use the tolerant settings as specified under ***Edit|Rules|Tolerances***. Furthermore, it will look at those beams along the same line and create a beam (or segmented beam when differences in section or material properties) if tolerance criteria is met.

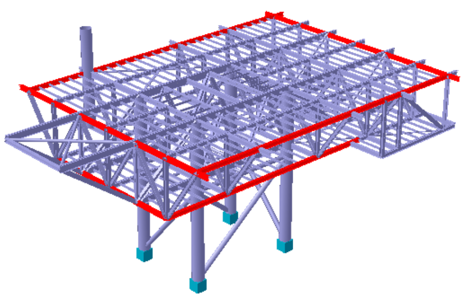
This means you can select many beams at the same time, and GeniE will automatically detect which beams that can be joined. This may be beneficial if you have used bottom up modelling or have imported a model from a FEM or CAD system – these are normally not based on concept modelling but geometry or 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.



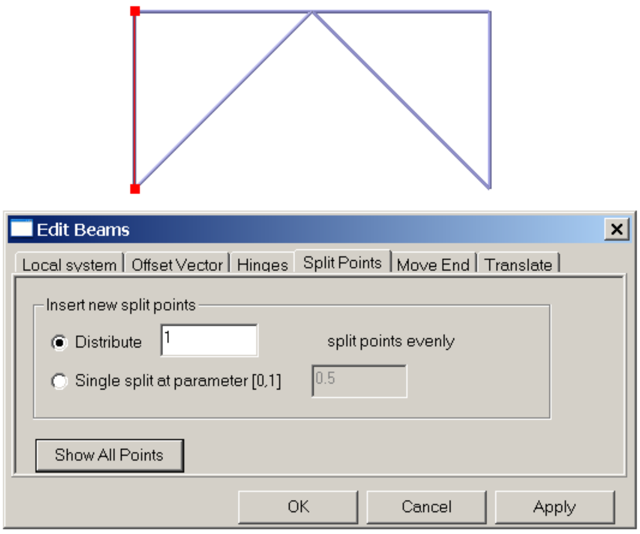
All beams along a line and satisfying the tolerance criteria are joined and a new concept model is established. As can be seen the small concepts along one of the deck edges are joined into one beam only.

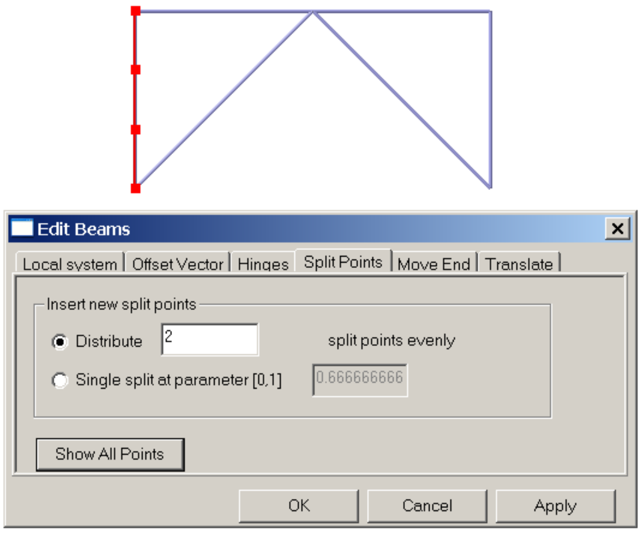
For this particular example, number of beams (or beam concepts) was reduced from 959 to 256.

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.

3.4.6 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.

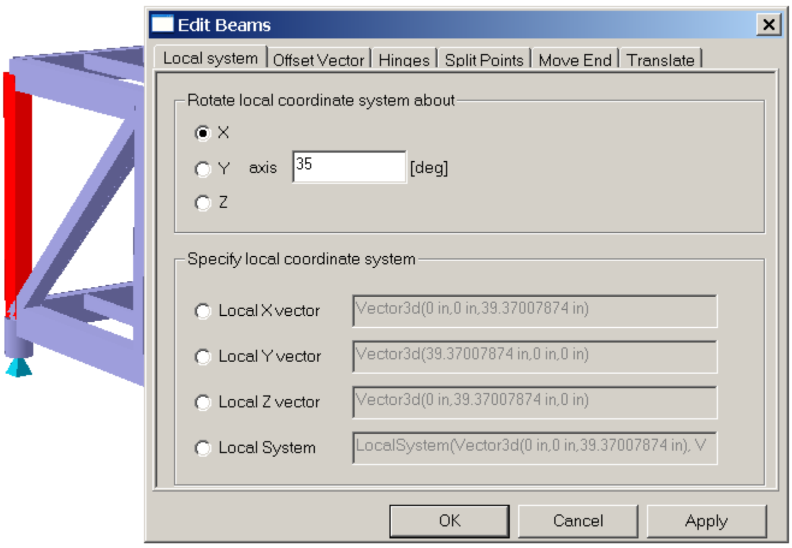




Snap points (or topology points) that are not used may be removed by using the command ***Tools|Structure|Geometry|Simplify Topology*** from the pulldown menu. A finite element node will be inserted at each topology point, hence a more clean finite element mesh can be achieved by using this feature.

3.4.7 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.

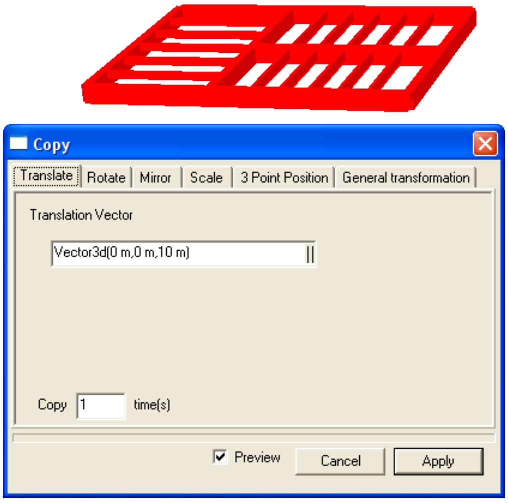


The example to the left shows how to rotate the local coordinate system 35 degrees around local x axis.

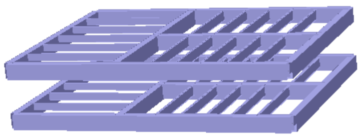
The command is available from the context sensitive menu, select Edit Beams and then Local System.

3.4.8 Delete, move and copy beams

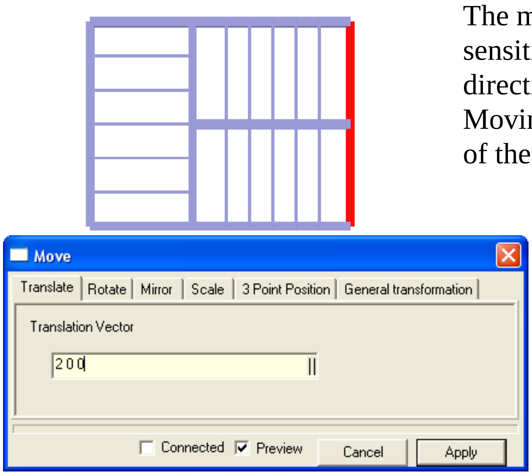
A beam may be deleted, moved or copied. The easiest way is by forcing the context sensitive menu and choose the relevant options. When a beam has been deleted you may re-use the name for a new beam.



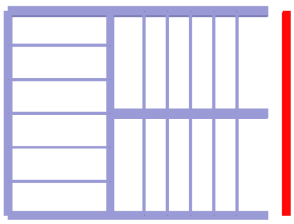
There are six options when performing a copy operation. They are translate (which is the most common), rotate, mirror, scale, 3 point positioning and general transformation. In this example a complete grid has been selected for a copy translate operation with vector 5 meters in vertical z-direction.

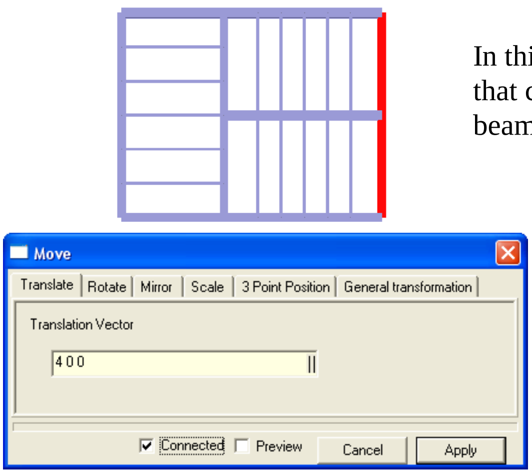


When using the feature for Move, you may move an entire beam or the end(s) of the beam. Both methods are explained below.

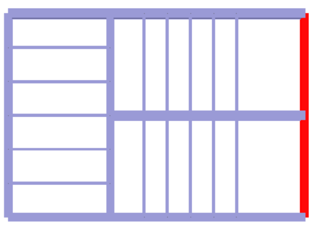


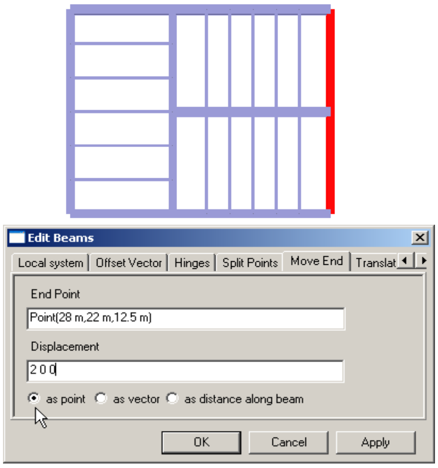
The move command has been activated by forcing the context sensitive menu and using *Move*. The translation vector 2 meters in x-direction will move the actual beam and de-connect it from the rest. Moving back again will automatically connect the beam to the rest of the model.



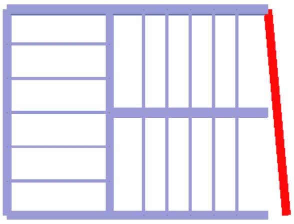


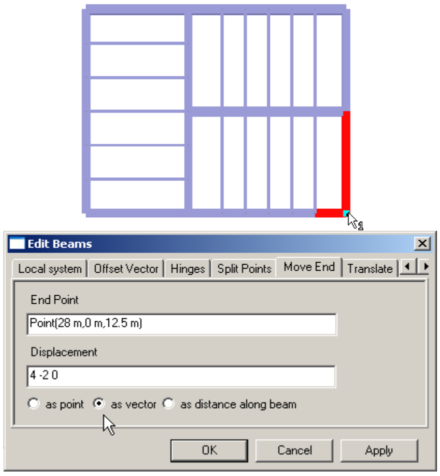
In this example the beam is moved 4 meters in x-direction. Observe that connected move option has been activated. All the connected beams will move accordingly (they are highlighted in red).



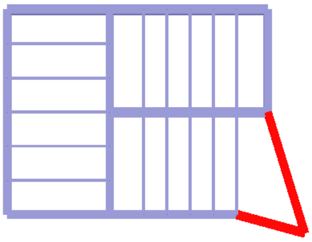


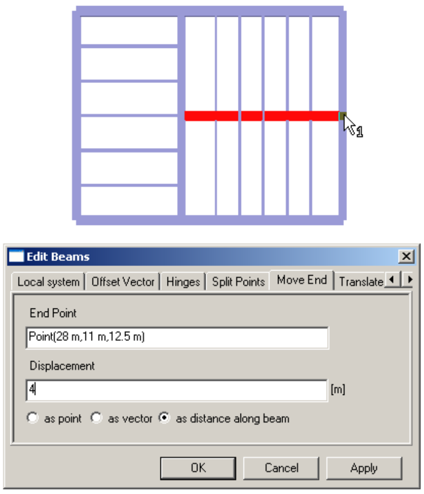
The context sensitive menu has been forced from where you select *Edit Beams|Move End*. In this example the end point at bottom has been selected to move 2 meters in x-direction (you may also move it to a given point). The highlighted beam is now connected only at top.



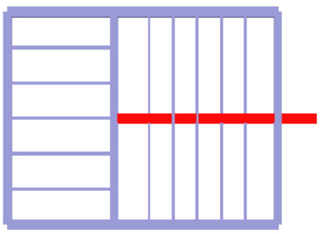


In this example common point of two beams is selected (note that the beams have been split compared to the previous examples). Moving this point a vector 4 meters in x-direction and -2 meters in y-direction will move the two connected beams as shown below.



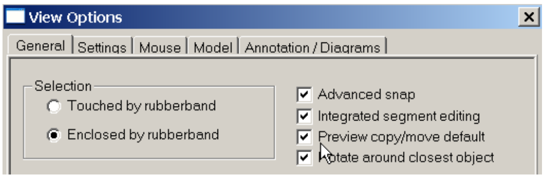


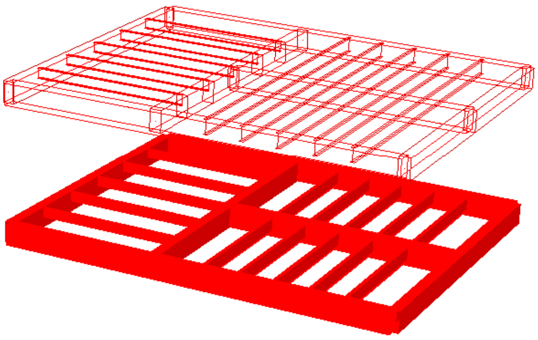
The example to the left shows how to extend (or shorten) a beam along the beam local x-axis. By giving a positive value the beam is extended; a negative value will shorten the length.

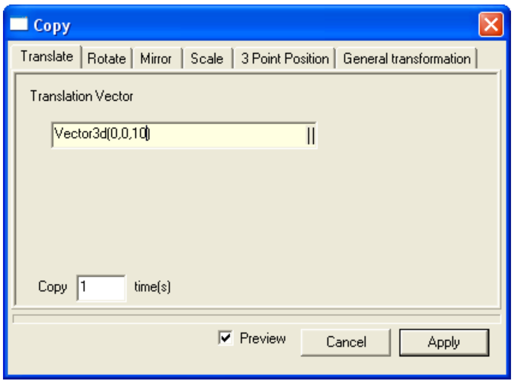


When activating the preview option it is possible to get a graphical preview of the actual move or copy before the operation is performed. It is thus easy to see if you have specified a wrong vector.

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.



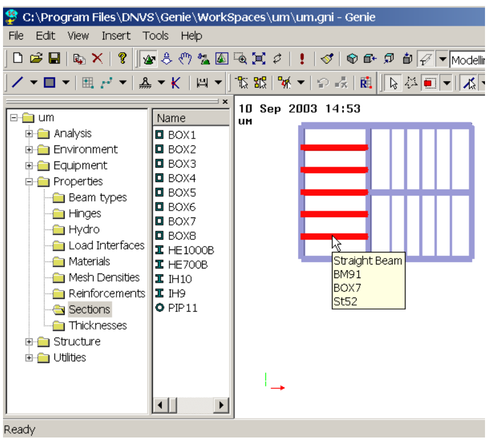


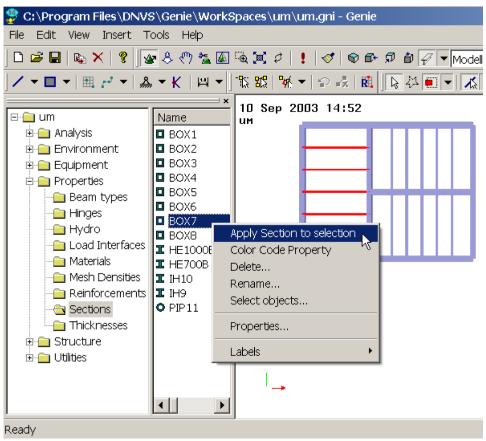


3.4.9 Change beam properties

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.





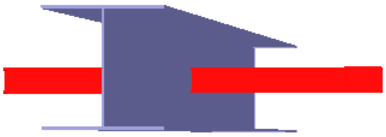
3.4.10 Beam eccentricities

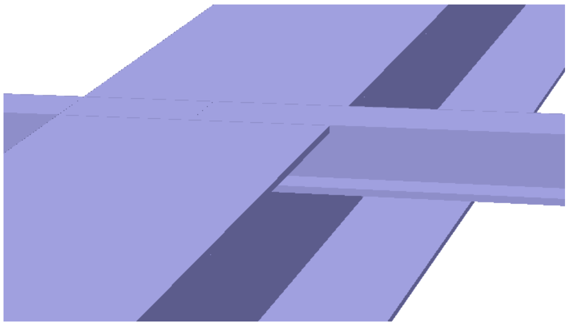
Eccentricities in the SESAM system are applied to the beams and not the plates. This means that for a stiffened panel the plate is modelled at top of steel level. Similarly, if you have beam with different section properties and want to align them at e.g. top of steel, it is necessary to apply eccentricities to beam with the least section height.

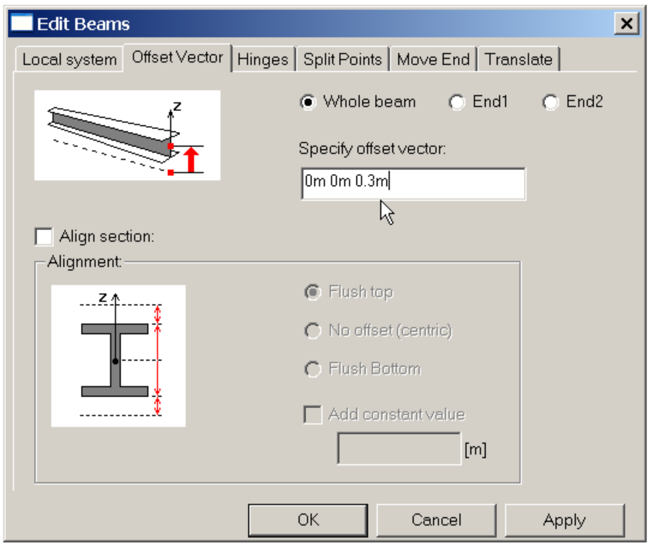
The example below shows two beams with different section profiles, I800 and I200. To align these at top of steel it is necessary to add an eccentricity to the I200 beam that equals (1/2 height of I800) – (1/2 height of I200). Using real values this gives an eccentricity (0.4 m) – (0.1 m) = 0.3 m.

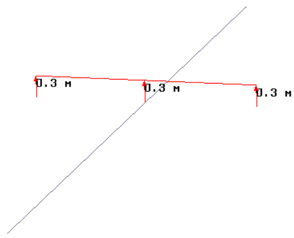
The features for inserting eccentricities are found from the context sensitive menu Edit Beams|Offset Vector. You may insert offsets to whole beam or to one of the beam ends.

To verify the eccentricities, select the beams, force context sensitive menu, then Labels|Eccentricities (best viewed in wireframe view).



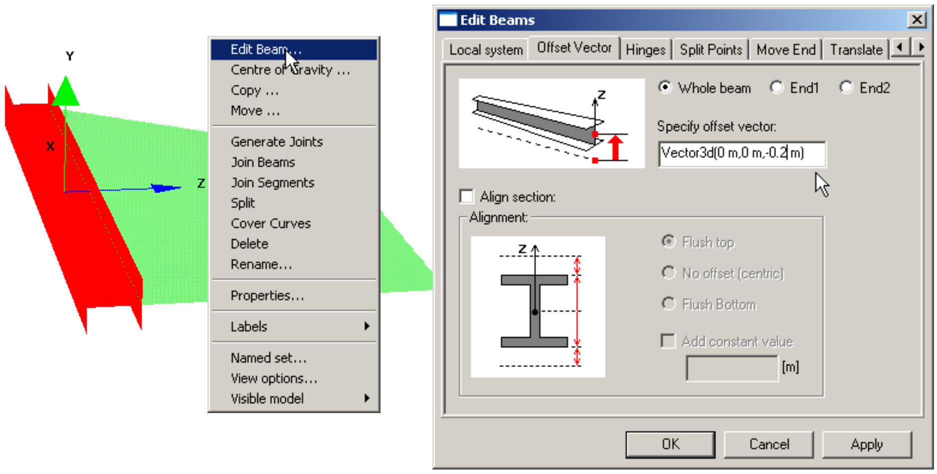


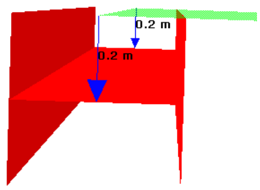




See also stiffened panels for further references on how to label and visualise eccentricities.

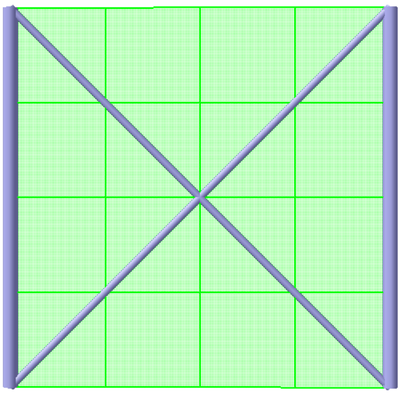
If you have rotated a beam and want to align it to a given level (for example a plate) you apply the eccentricity according to the global coordinate system. In the example below a beam has been rotated 90 degrees and is aligned below a plate. The eccentricity is 0.2 m downwards.



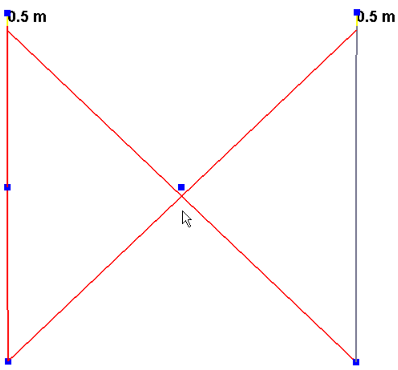


3.4.11 Connecting beams to an eccentric beam end

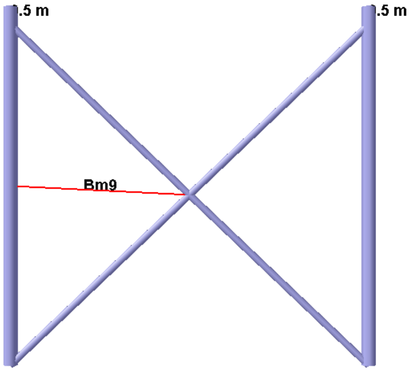
When using the feature for *Snap Eccentric* it is possible to connect a beam to another beam end using its eccentric position. This is typical for an X-brace in a jacket where the beam ends have eccentricities due to e.g. gap calculations and you want to connect a beam to the eccentric position of the X. The example below shows how to do this.



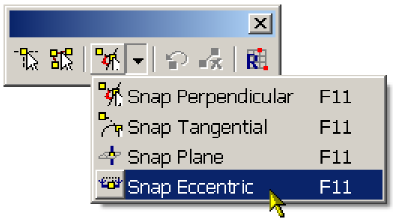
1. A regular X-brace is created. Please note that the vertical beam to the left is split in two



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.

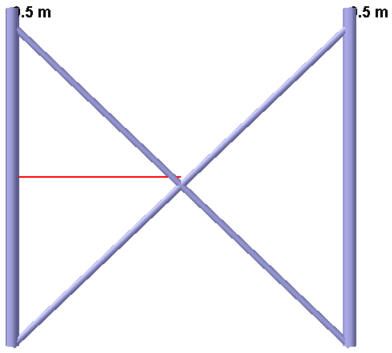


3. When activating the *Snap Eccentric* feature it is now possible create a beam that will snap to the eccentric position of the X-joint. Choose *Snap Eccentric* as shown and insert a beam by clicking fist at the vertical beam and then to the X-joint.



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 cammands are automatically created:
92 // First inserting the beam to the topology point of the X joint
93 // Then inserting the same eccentricties 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.

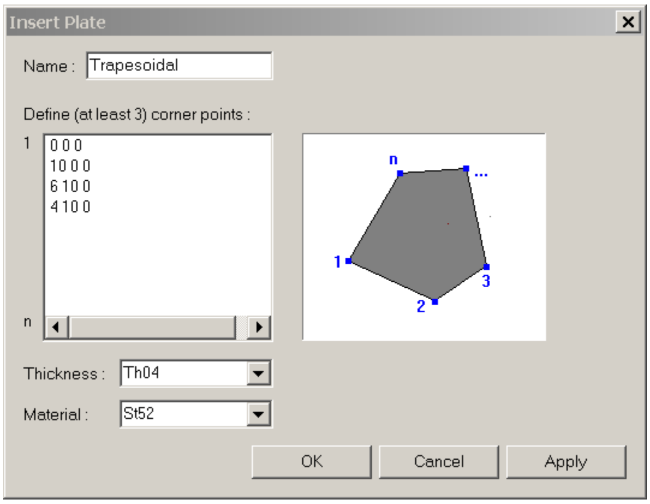
3.5 Basic plate modelling

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

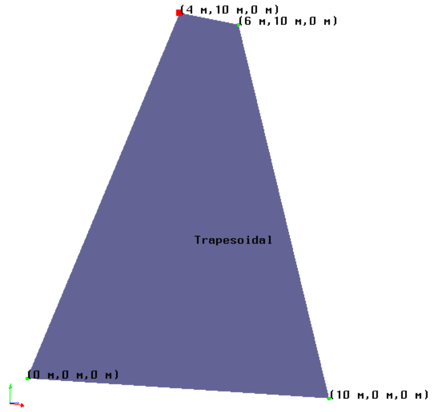
A plate is modelled using the same methodology as for a beam, in other words by explicit modelling of plate end co-ordinates or by clicking to existing snap point in a guide geometry or existing structure.

3.5.1 Regular plate in a plane

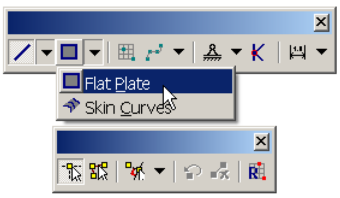
Explicit modelling of the plate is done from the pulldown menu Insert|Plate|Flat Plate Dialogue.



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.

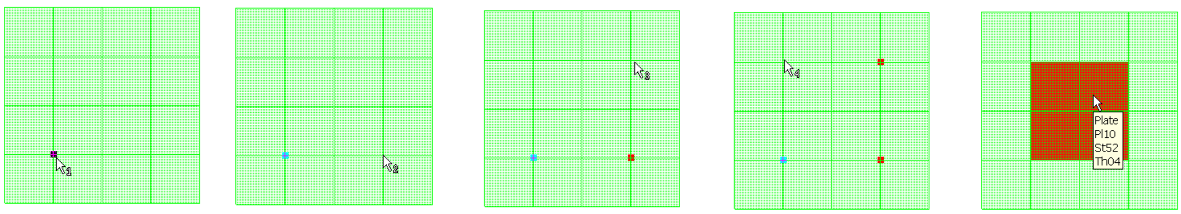


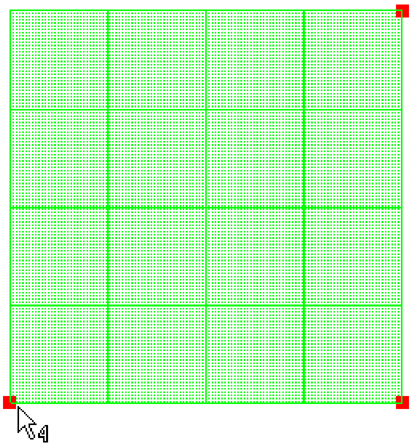
The other alternative of inserting a plate is by using the snap to point method.



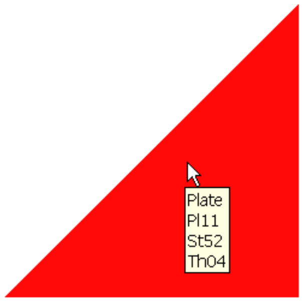
Make sure that the Flat Plate option is selected and that the insert mode button has been activated.

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.

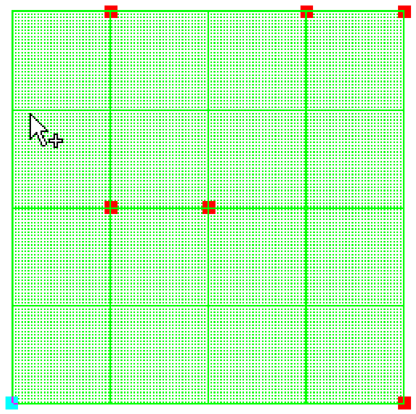


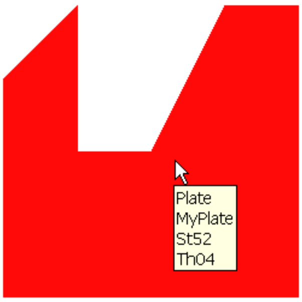


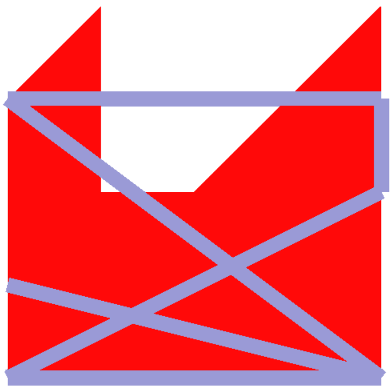
To insert a triangular plate the same technique is used. The difference is that the prompted 4th snap point is the same as the first snap point used when creating the plate.



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.





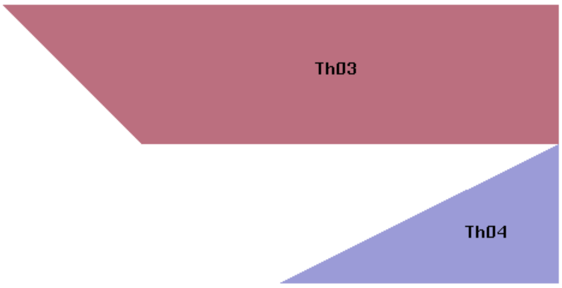


Beams and plates may be inserted at any time, GeniE calculates all intersections and manages a consistent topology. If you expect many changes to your model during structure modelling, it may be of advantage to do most of beam modelling prior to inserting plates. The reason for is that the topology model 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.

3.5.2 Overlapping plates

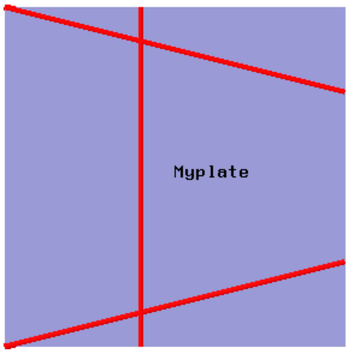
You can not insert a plate in the same position as an existing plate. The program will give a warning on this.



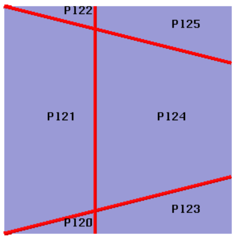
If you insert a plate larger then the original one, the program will insert the delta between these plates. See the example to the left where a larger plate is moved over a smaller plate.

3.5.3 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.



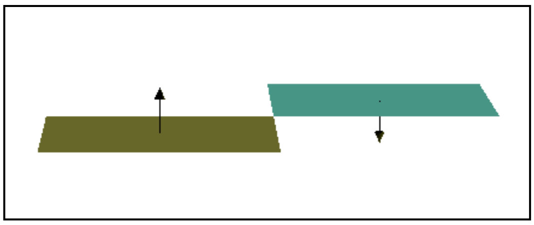
This example shows MyPlate intersected by 3 beams before and after a split operation. As can be seen, MyPlate is split into six smaller plates.



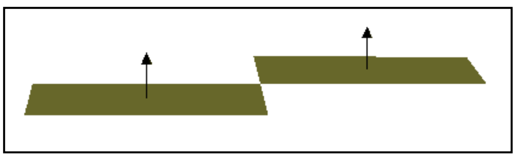
Plates can be joined from the command line area. Joining Pl124 with Pl121 has the command *Pl121.join(Pl124);*.

3.5.4 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.



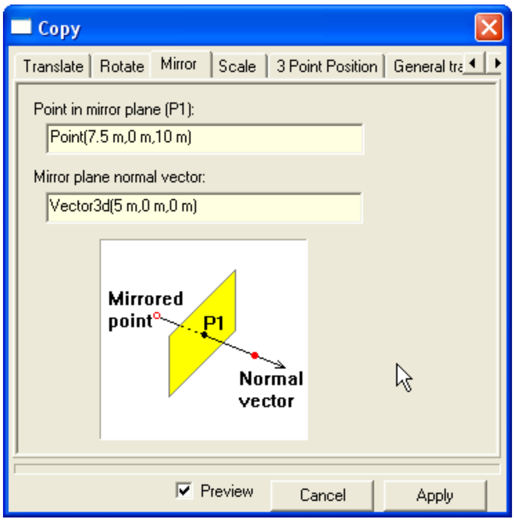
The example below show that two plates have been modelled in opposite directions, hence local z-axes in different directions (context sensitive menu and *Labels|Local coordinate system*).

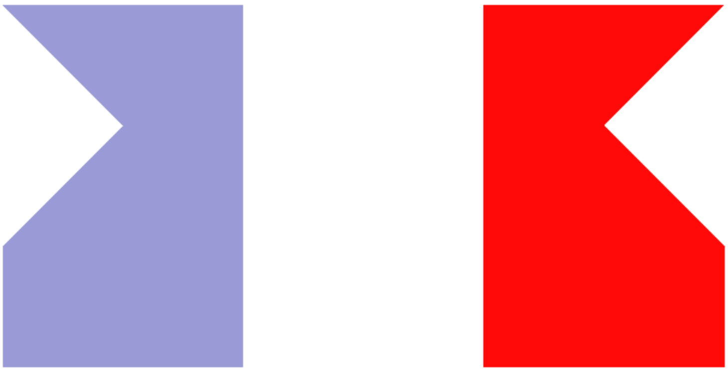


You change directions of plate normals by selecting the plate, RMB and *Flip Normal*).

3.5.5 Delete, move and copy plates

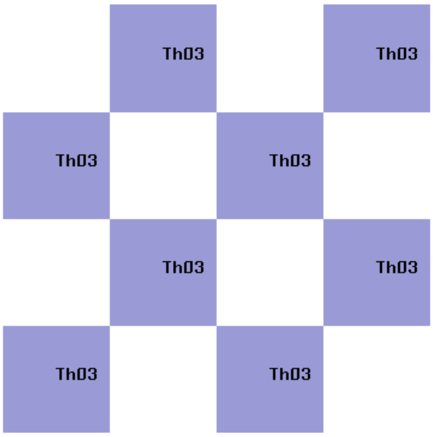
These operations work the same as for beams. The example here shows a plate (the left plate) that is copied using the mirror option.

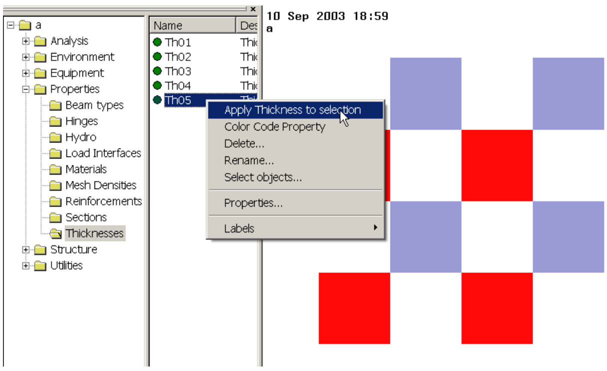


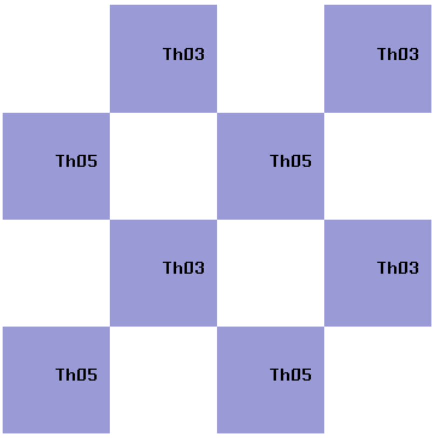


3.5.6 Change plate properties

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.

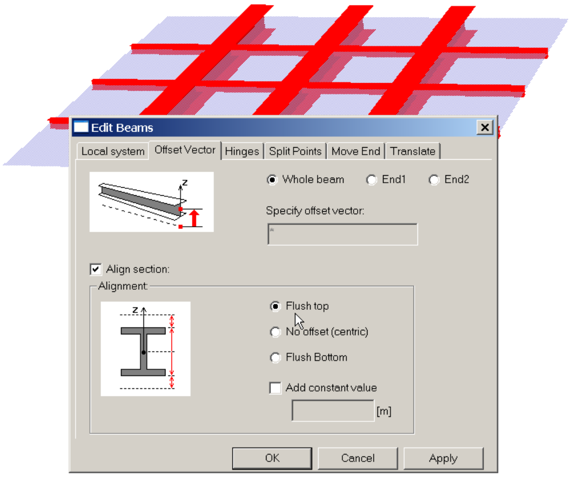


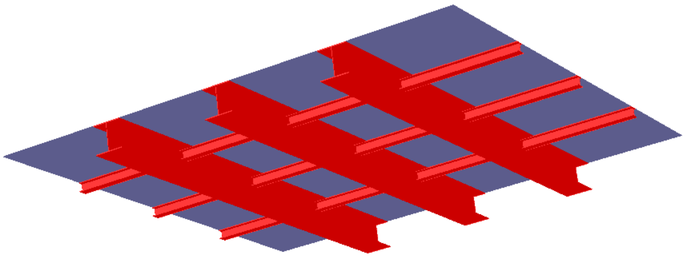




3.5.7 Stiffened panels

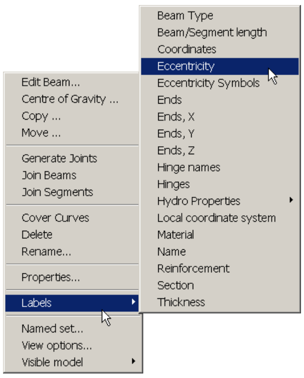
GeniE can do flushing (insert eccentricities to beams related to a plate) of multiple beams at the same time. The example below shows beams with two different section types. When modelling these they are inserted at the same level (system level or top of steel) as the plate. In most cases, the beams are flushed either above or below the plate. Select the beams, force the context sensitive menu and choose *Edit Beams|Offset Vector*.

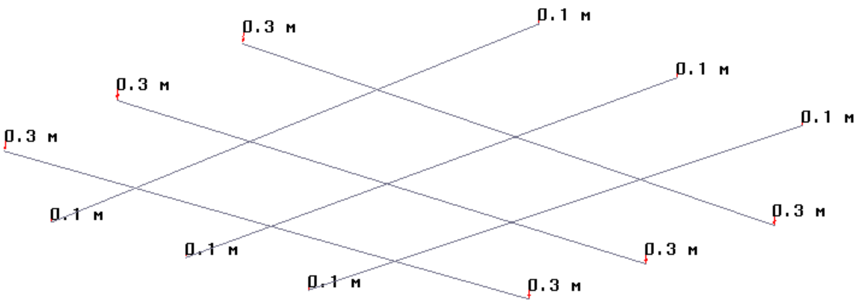


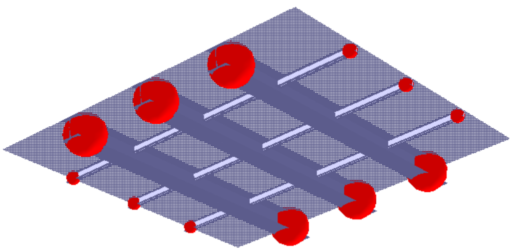


In the example above flush to top of beam was used (flush top). This means the beam is placed below the plate. Similarly flush bottom will place the beam on top of the plate. Observe that the flush operation will use eccentricities calculated along the beams local z-axis. This means that flushing also 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.







3.6 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).

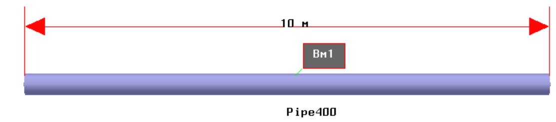
Emphasis has been put on describing the steps necessary to arrive at the desired model. There are several modelling approaches, this document describes some of them. Focus has also been put on the importance on how to document and verify your model.

3.6.1 Segmented beam modelling

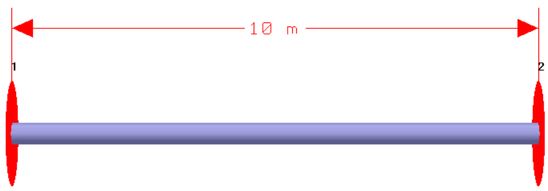
A segmented beam is a continuous beam consisting of several segments. A segment is normally inserted when there is a change of section or material property. It will act in the same way as several beams aligned, the main difference is that the segmented beam is a higher level concept compared to a set 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.

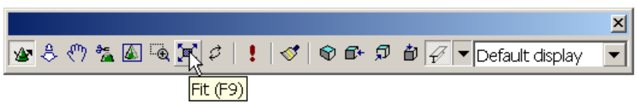
Create segments

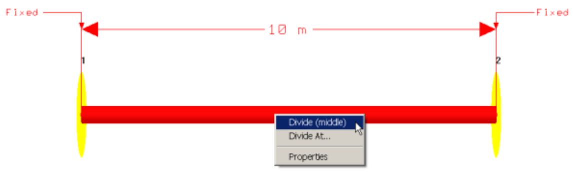


1. The original beam (Bm1) has section type Pipe400 and is 10 meters long

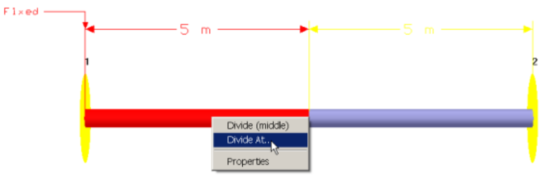


2. Double click the beam to enter segmented modelling modus. The picture shows that there is one segment to this beam only. You should also activate the *Fit to Screen button* (F9) for a best possible view.

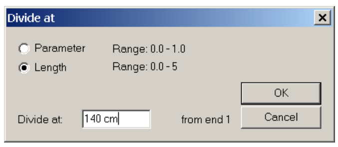


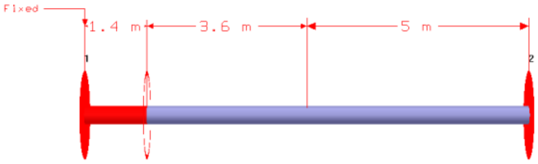


3. Select the beam, RMB and *Divide (middle)*

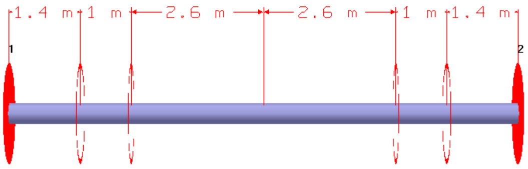


4. The beam is now split in two segments and you can continue to divide either at middle of a segment or at a given length. In this case, 140 cm from end1. Note that the length is always referring to end1.



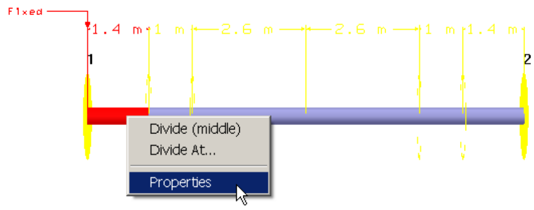


5. The beam is now split in 3 segments and the individual lengths are automatically calculated ensuring that the total beam length is maintained.

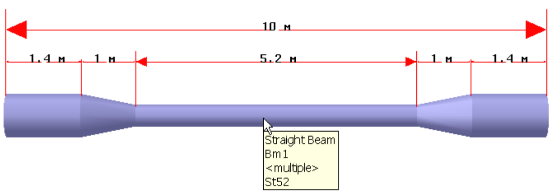


6. The final configuration of segments prior to assigning new section properties. Note that you can change individual lengths by clicking the relevant segment, type the new length from your keyboard, and complete the operation by pressing Enter. You may be prompted to fix one of the segment ends by using the TAB button.

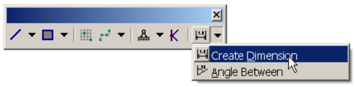
Assign new section properties

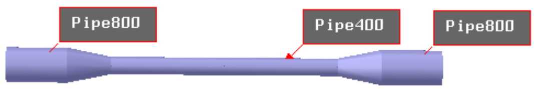


7. Select a segment(s), RMB and assign new section or other material. In this example a cone section has been used in the transition between larger and smaller section profiles.



8. The final configuration. Dimensions have been added using (remember to activate the *Filter Segment*) the dimension tool *Create Dimension*.





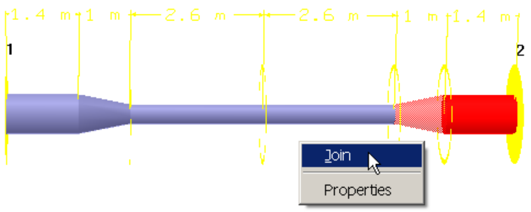
9. Verifying the model by applying labels, this is only possible when the *Filter Segment* is activated.
You may also verify from Tab BeamSeg on a saved report

Name	SegNo	Section	Material	Length [m]	FlexLen [m]
Bm1	1 of 6	Pipe800	St52	1.400	1.400
	2 of 6	Cone	St52	1.000	1.000
	3 of 6	Pipe400	St52	2.600	2.600
	4 of 6	Pipe400	St52	2.600	2.600
	5 of 6	Cone	St52	1.000	1.000
	6 of 6	Pipe800	St52	1.400	1.400

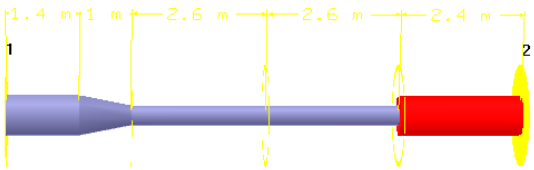
MainToc / Overview / Beams / BeamMass / BeamSeg / Sections / Thickness / Materials

Ready

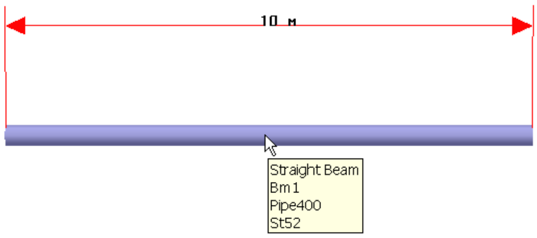
Join segments



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.

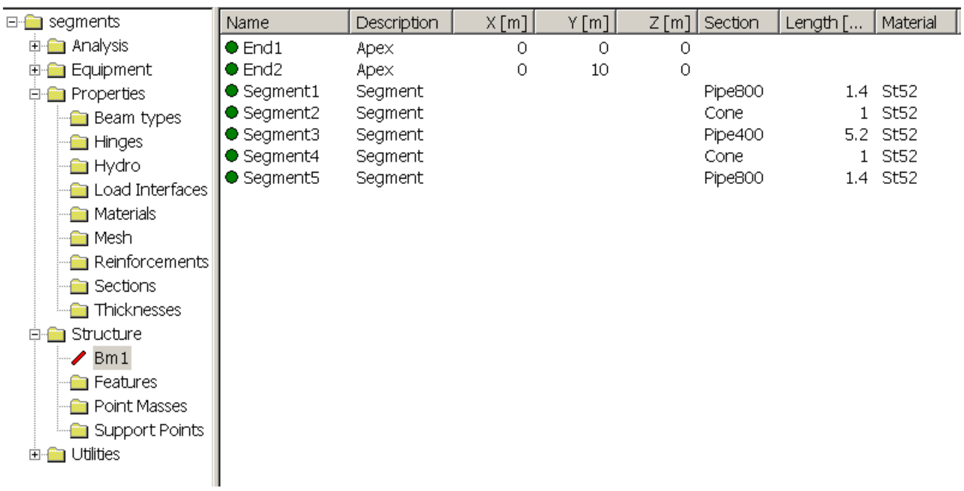


11. The two segments joined to one. You may join two segments or more at the same time.



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



When double clicking on Bm1, the details of the segmented beam appear in the browser.

You can change details of each segment by RMB.

3.6.2 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

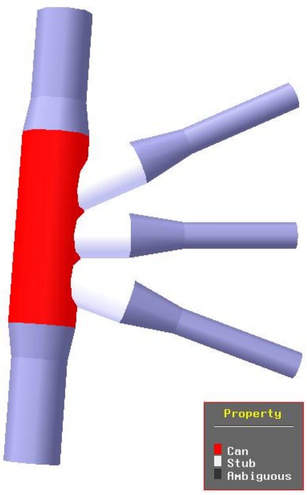
Step 2. Define rules for lengths of cans, stubs, cones and also gaps

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

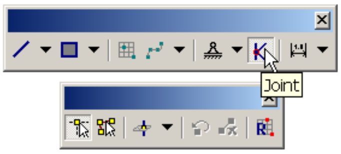
Step 2 – See Rules Chapter 3.2

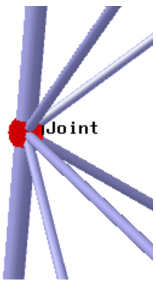
Step 3 – Assign joints

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.

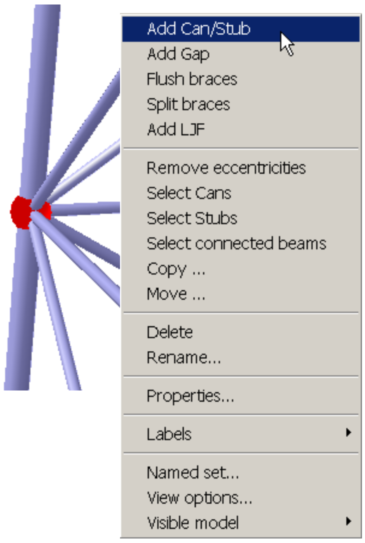


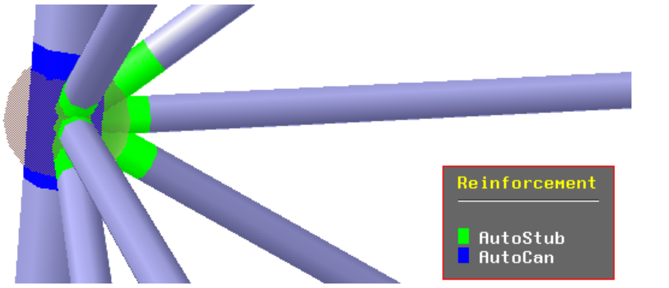


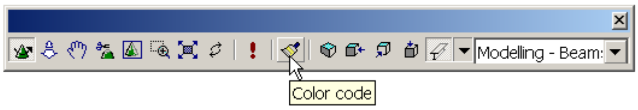
The example to the left shows that a joint has been inserted and renamed by selecting it, RMB and *Rename*. It is required to define a Joint at each tubular joint where you want to do automatic assignment of cans, stubs, cones, and gaps. It may be of advantage to specify 50% transparency of the joint symbol (***View Options|Settings|Structure|Joint|Transparency***). To be able to see the joints graphically, ensure that the Joint selection is activated.

Step 4 – Assign cans and stubs

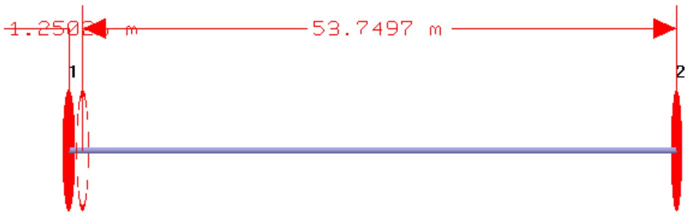
Cans and stubs may easily be assigned – again based on the rules specified - when a joint (or joints) have been selected. The example below shows assignment of can and stubs by the *Add Can Stub* option in the RMB menu for the selected joint. The cans and stubs have been colour coded by selecting them 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.





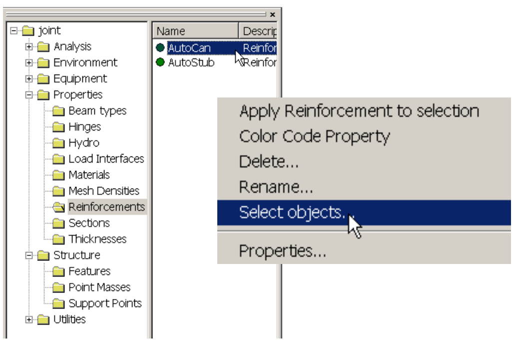


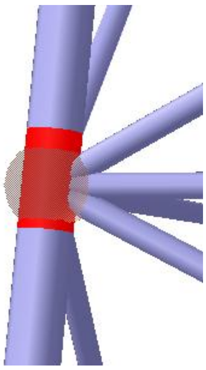
When assigning cans and stubs, the program makes the respective straight beams into segmented beams, where the segment lengths are according to the can and stub lengths specified in the rules for such. This command will also assign concept information (cans and stubs) that will be read by the code checking program Framework. The picture below shows the segmented members of the horizontal brace.

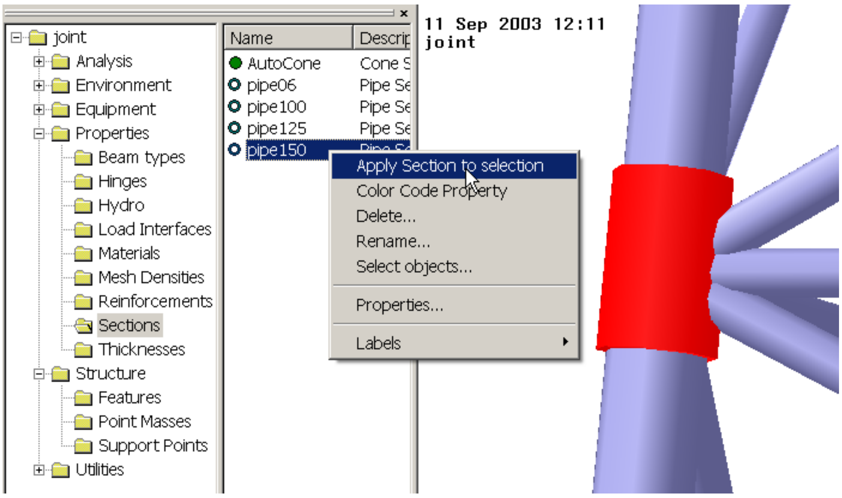


Step 5 – Assign new properties to cans and stubs

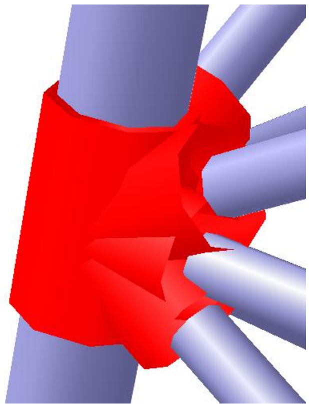
The final task to do with cans and stubs is to assign other section or material properties. The easiest way is to select these objects from the browser, choose properties from browser and apply. The sequence below shows how to select the cans and how to change section properties.







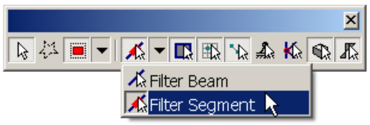
The cans are assigned new section property pipe150. Picture below shows both cans and stubs reinforced.

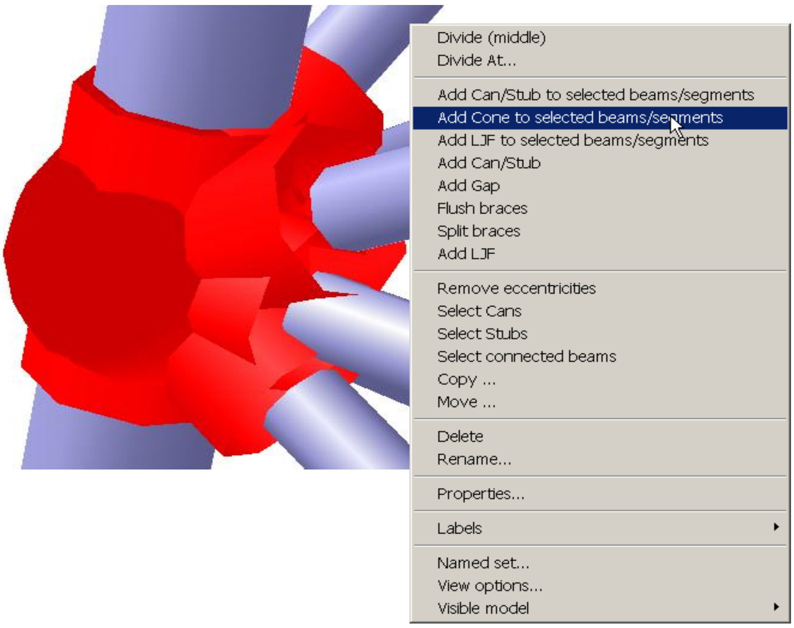


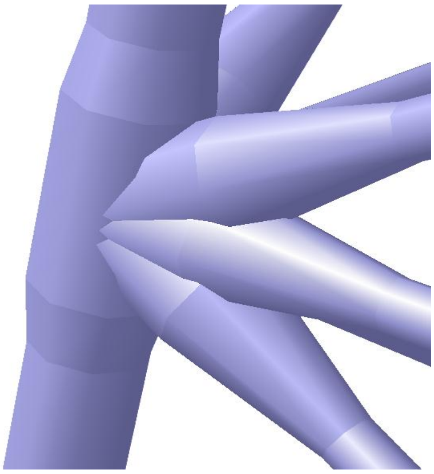
Note that if you do not want to add gaps you need to recalculate can and sub lengths (same as Step4) due to the updated geometry. The gap operation will adjust the can and stub lengths.

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*.



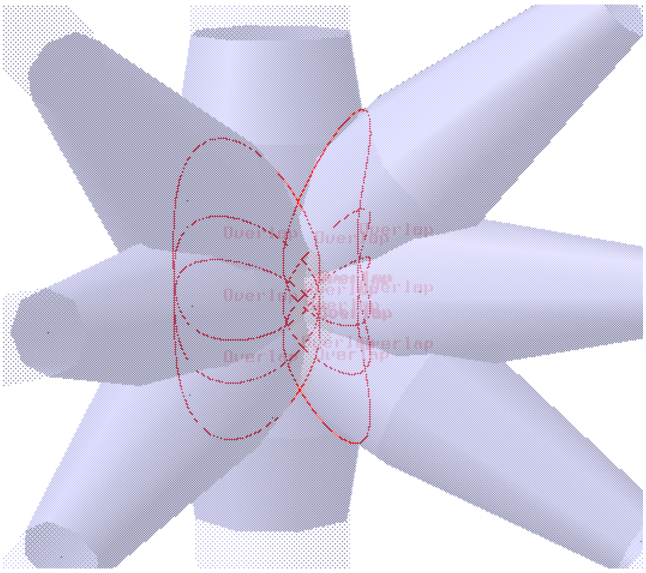




You may choose to not insert cones to all the cans or stubs, you then deselect the relevant cans or stubs and no cones will be calculated.

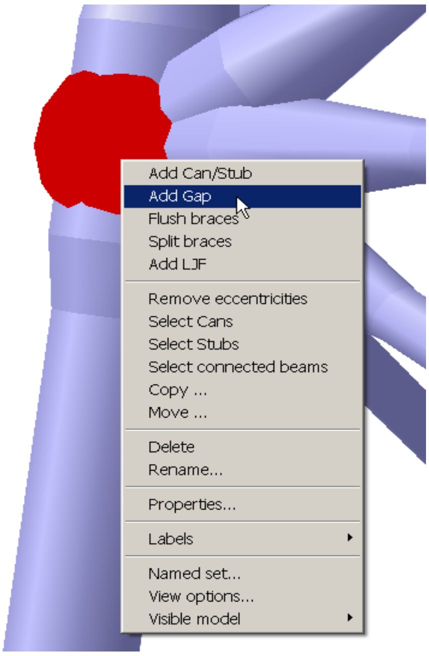
Step 7 – Perform planewise gap calculations

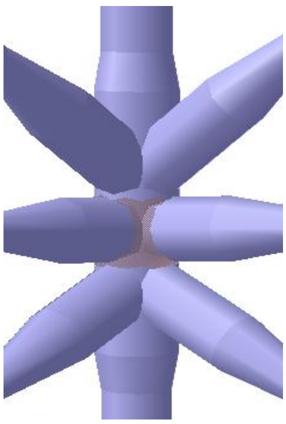
The gap calculations are performed by selecting the joint (graphically or form browser), RMB and *Add Gap*.

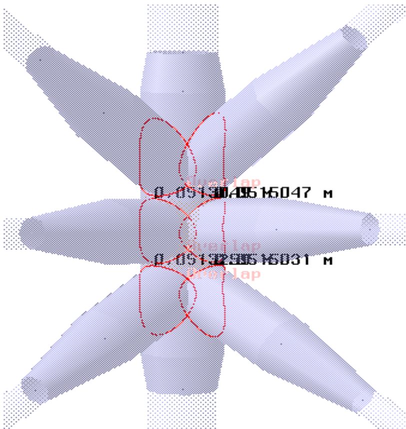


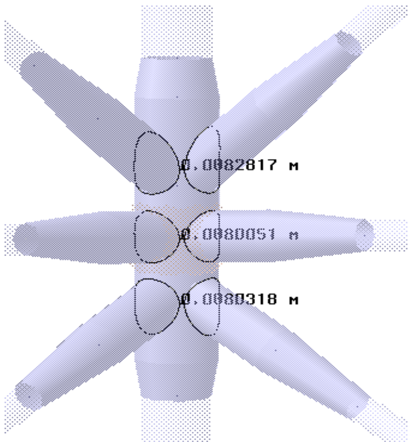
You may find the existing gap values before a gap calculation by double-clicking the joint. Gap values shown in red colour indicates that they are larger than the minimum gap value specified in the rules.

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.



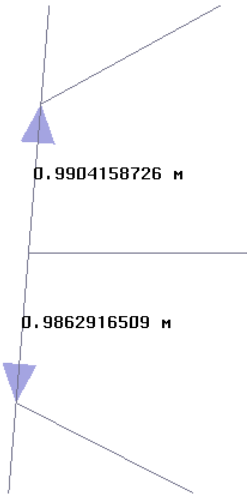






Reduced stub diameter

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.

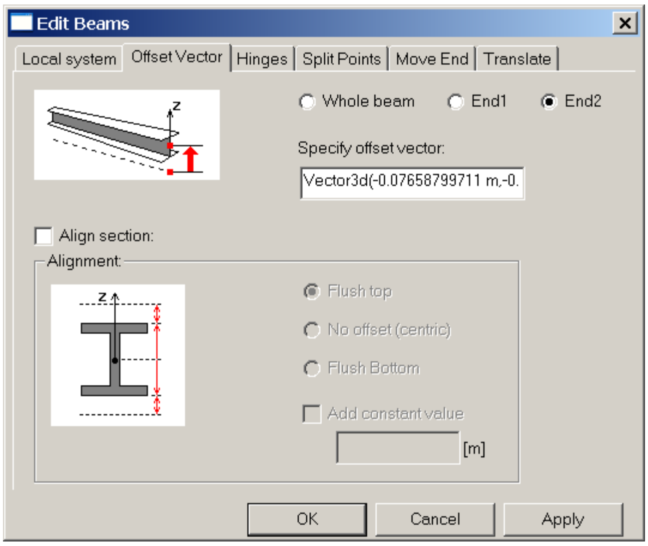
For the lower brace, an eccentricity vector δ = 0.98629 m has been inserted. For more details about δx, δy, δz this may be found by opening the beam editing function from the context sensitive menu.

In this case they are:

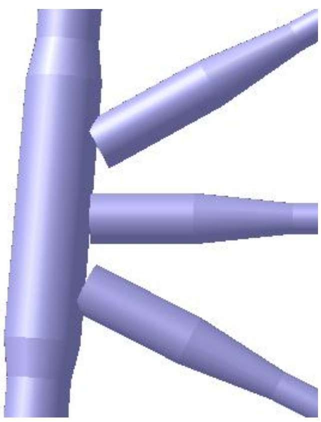
δx = -0.0765879971 m

δy = -0.0765879971 m

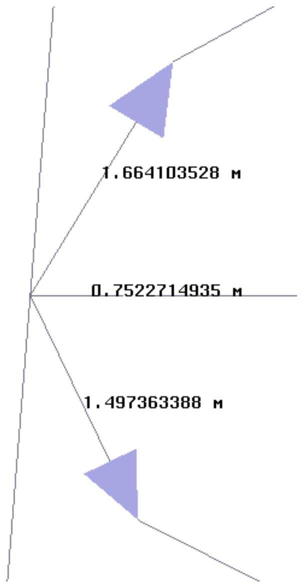
δz = -0.9803263631 m



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



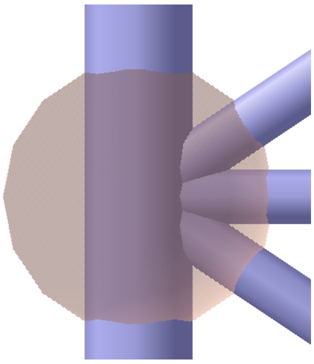
Eccentricities out to chord wall, horizontal brace also receives eccentricity

To remove eccentricities assigned, select a joint(s), then RMB and *Remove Eccentricities*.

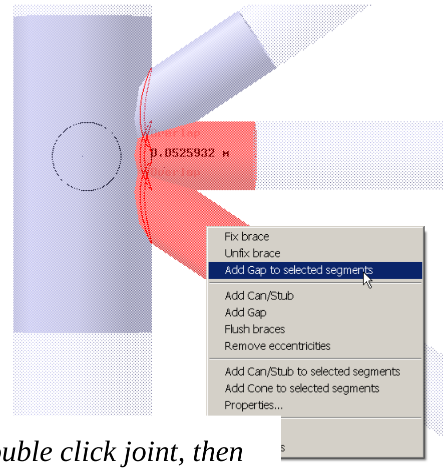
There may be cases where you want to perform other special planewise gap operations. Several options are available and in the following some are documented.

Option 1 – Add gap to one brace only

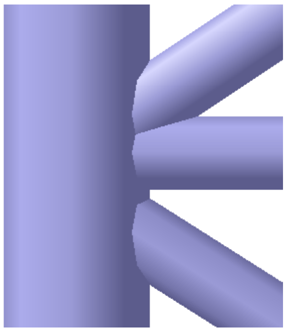
You need to create cans and stubs prior to calculating the actual gaps (double click the joint). Per default the modelling is switched to segmented modelling modus. You can now select the horizontal brace and the inclined brace to calculate new gaps, RMB and *Add gap to selected segments*. The figures below show the steps to add gap to the lower beam only.



Selecting the joint



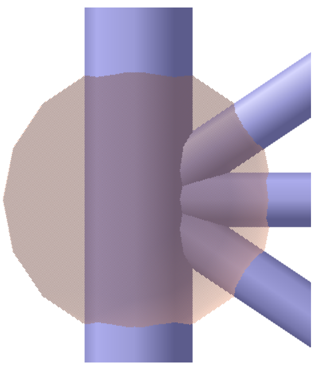
Double click joint, then select members, and add gap to the segmented members



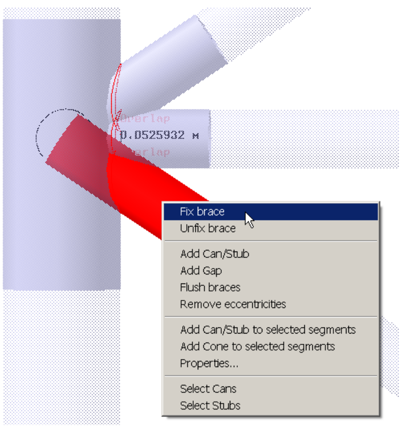
Final configuration where lower beam is assigned a new gap satisfying the minimum requirements

Option 2 – Move the T in a KT-joint up

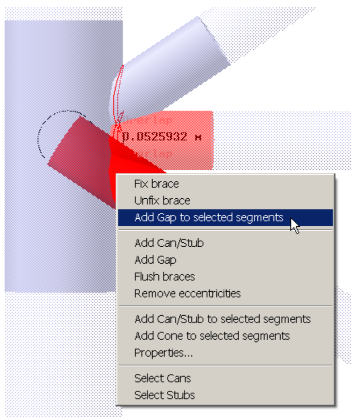
This example shows how you can move the T (or the horizontal brace) in a KT-joint up. It also shows how to ensure that the minimum gap requirements are met on both sides of the T.



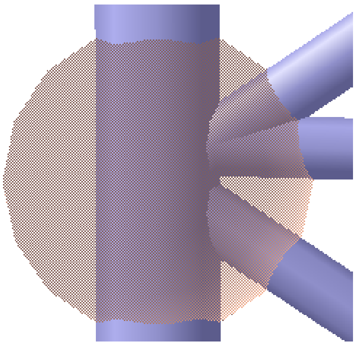
Select joint



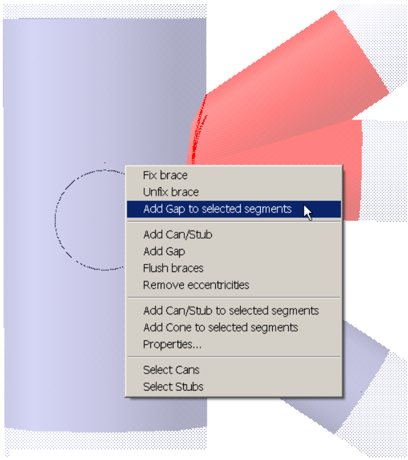
Double click joint, select lowest brace and fix it



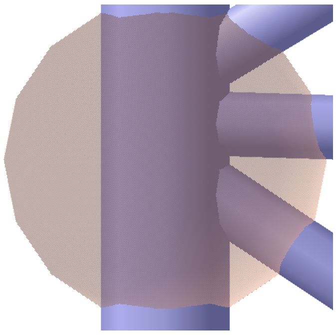
Select lowest and horizontal brace and add gap



Configuration before gap calculations on upper brace



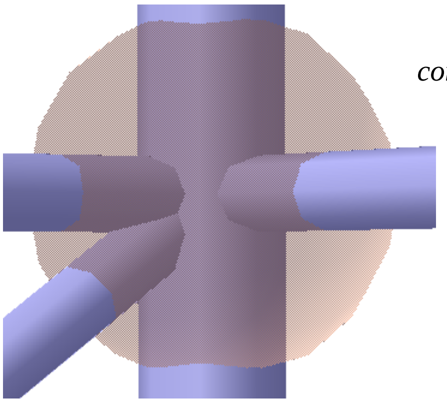
Double click joint, select upper braces and add gap



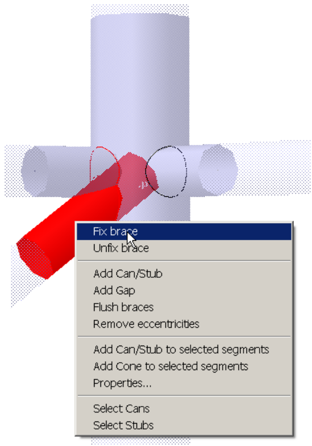
A KT-joint where T is moved upwards satisfying the minimum gap requirements

Option 3 – Move the T part of K-joint (Y+T)

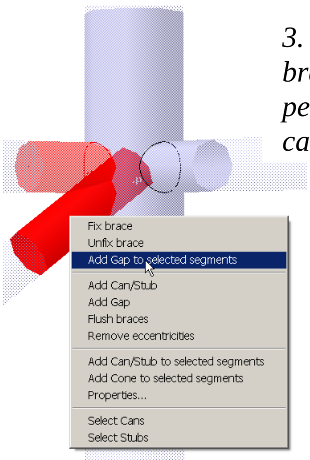
This example illustrates how to move a T (perpendicular to chord) in a Y+T joint. The steps are basically as showed before, the differences lie in which members to select.



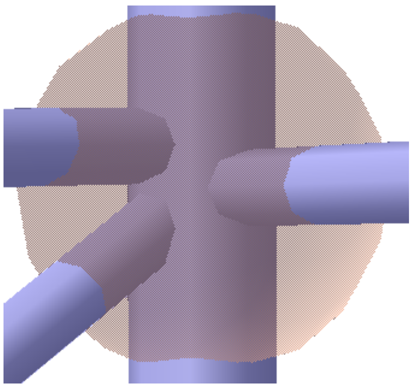
1. Initial configuration



2. Double click joint, select, RMB and fix brace



3. Select both braces, RMB, perform gap calculation

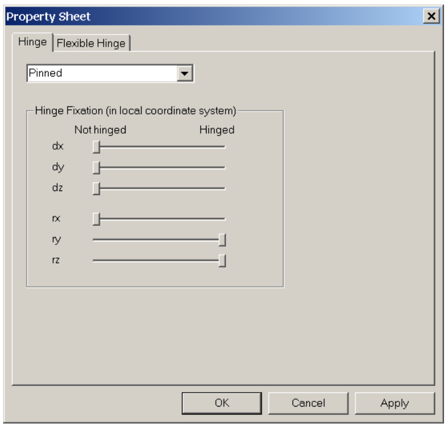


4. Final configuration where the upper horizontal beam has been moved upwards

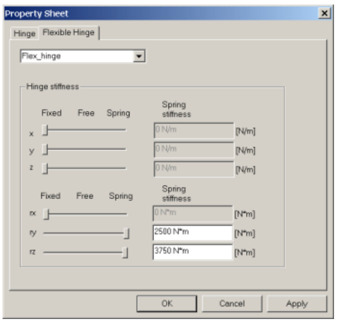
3.6.3 Hinges

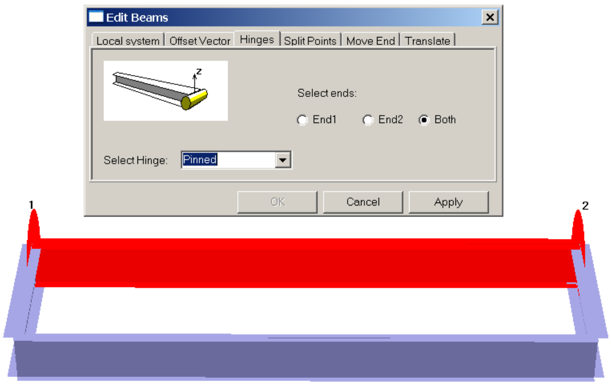
Hinges may be inserted at end of beams with the purpose to release local degrees of freedom. For example, a typical hinge is fixed from deflections in all translation DOF and torsion DOF. The two other rotation DOF (r_y and r_z) are free. The below example shows how to insert and verify hinges. You can 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.

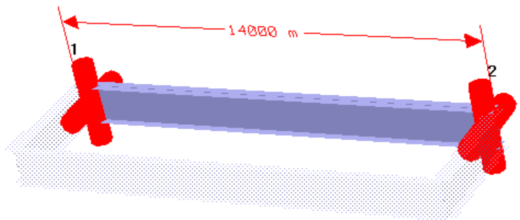




Select beam, RMB, Edit Beams|Hinges.

Apply hinge property to one or both beam ends.

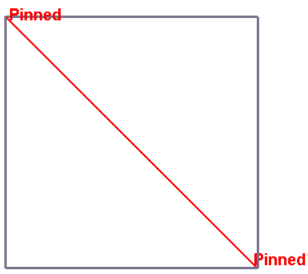
To verify, select same beam, RMB, *Edit Beams|Hinges* or double-click beam to enter segmented modelling modus. Both methods will show the hinges graphically. You may also save a report to do a tabular verification.



	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.



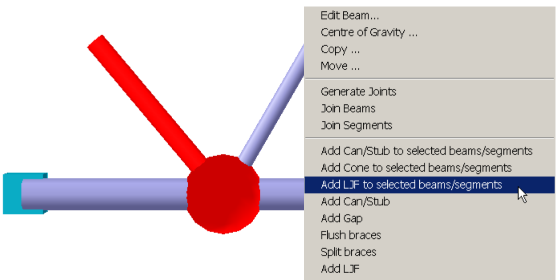
3.6.4 Local joint flexibility

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.

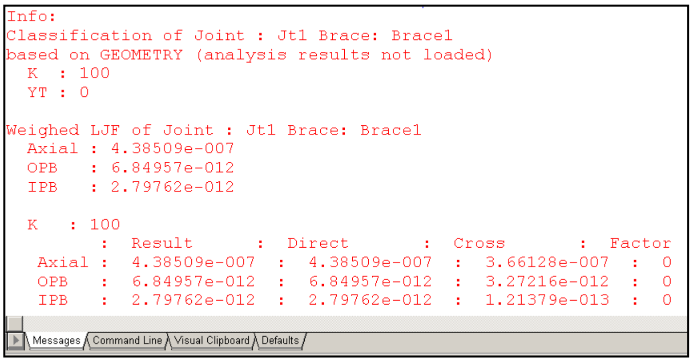
LJF may be assigned based on a pure geometry configuration or a load path.

- 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$_{OPB}$ and In Plane Bending$_{IPB}$) 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$_{OPB}$ and LJF$_{IPB}$ are independent of beam local co-ordinate system.

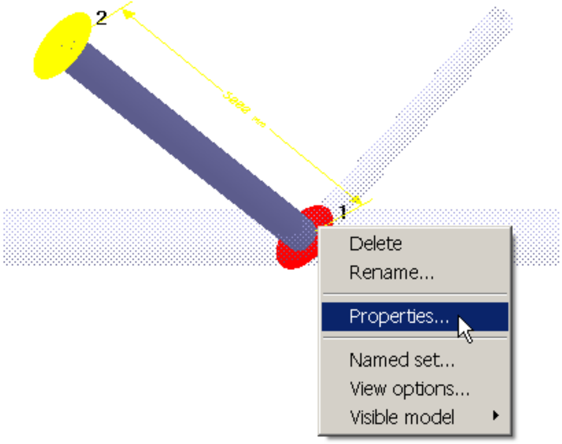
The example below shows how LJF is assigned to one beam end based on a pure geometry approach.



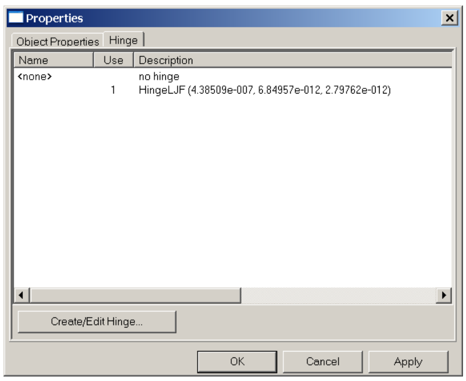
1. Select joint and beam, RMB, Add LJF to selected beam/segments



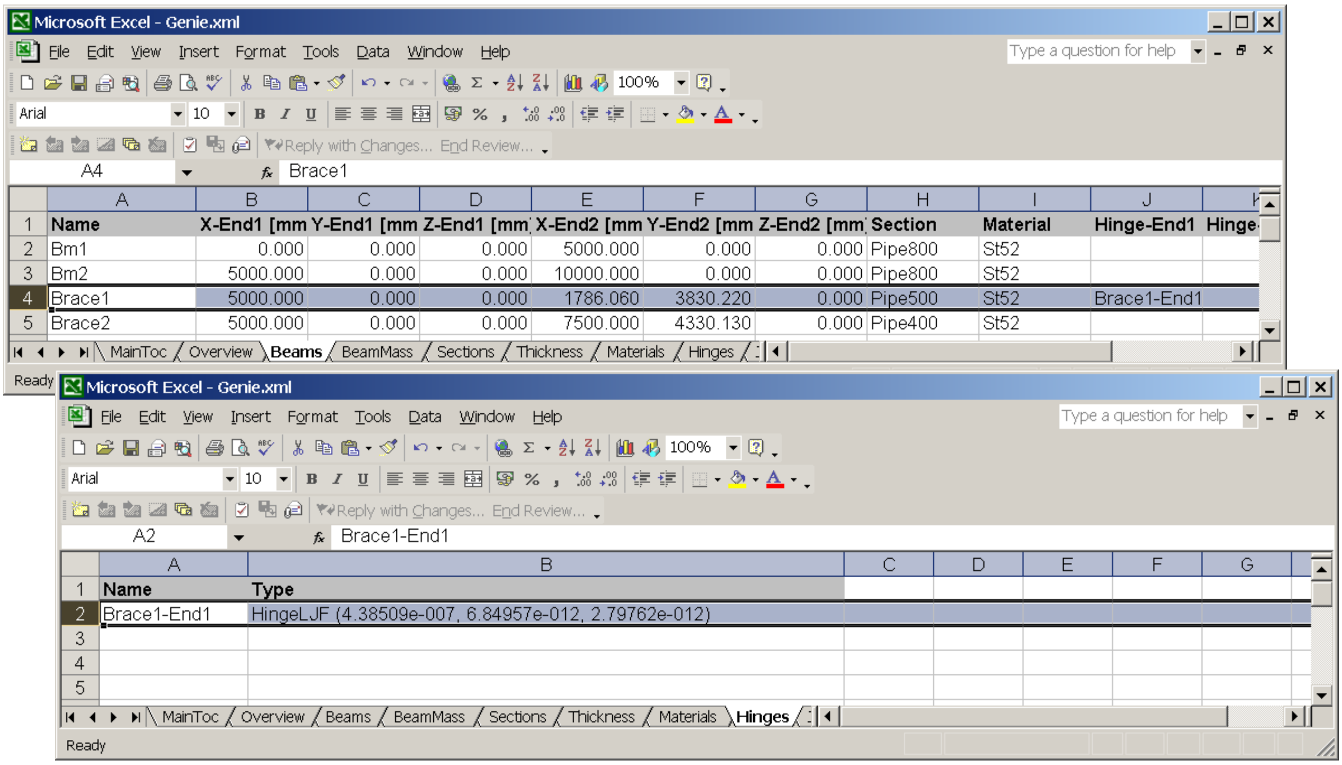
2. Verify details as found under the Message tab (see also the journal file)



3. Double click beam to enter segmented modelling modus, select joint, RMB and Properties

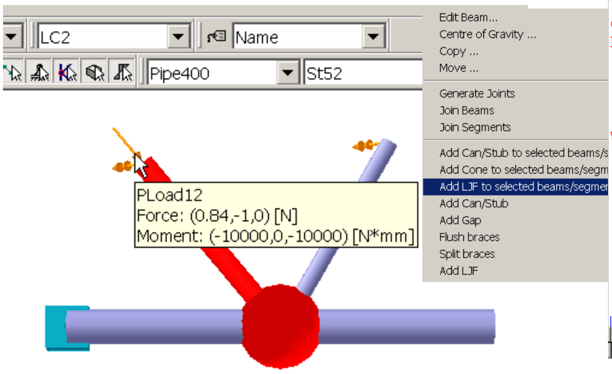


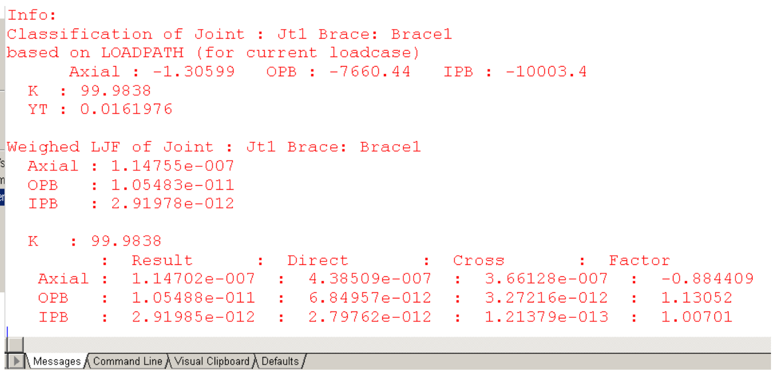
4. Verify from Property sheet the LJF details



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:

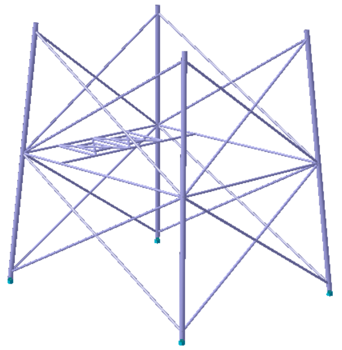
- When running an analysis after load path based LJFs has been created, the effect of these LJFs will be included in the analysis. Hence, if new LJFs are calculated based on an “updated” load path the LJF values will change accordingly.
- LJFs can only be assigned to start or end of a beam connected 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 γ, τ, ß, θ 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.

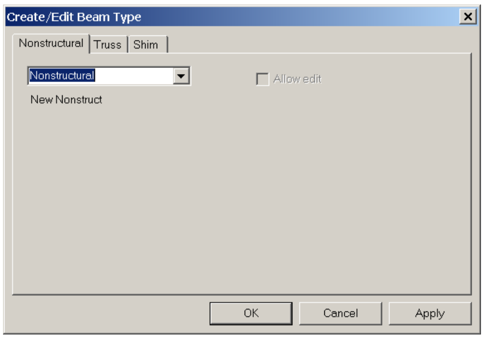
3.6.5 Non-structural beams, shims, and truss elements

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.

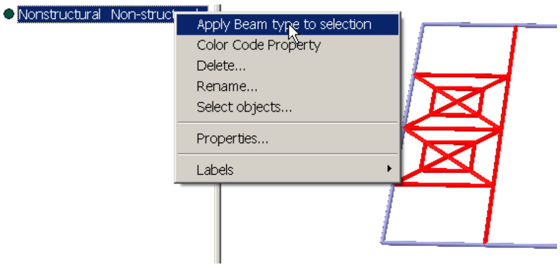
Non-structural beams

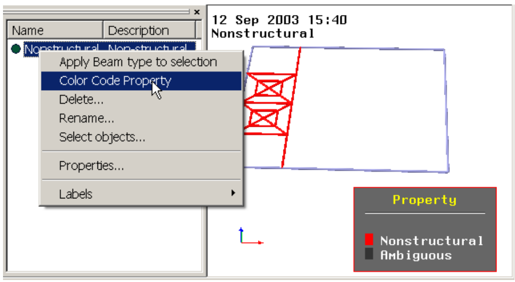


In this example the "conductor frame" is assigned as non-structural beams. This is achieved by a) making a beam type property called Nonstructural (***Edit|Properties|Beam Type)*** and b) apply this to the relevant beams.



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.

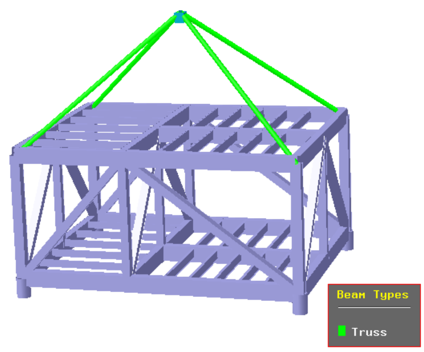




	A	B	C	D	E	F	G	H	I	J	K
1	Name	X-End1 [m]	Y-End1 [m]	Z-End1 [m]	X-End2 [m]	Y-End2 [m]	Z-End2 [m]	Section	Material	Beam Type	Floodi
34	Bm35	-24.063	8.438	30.000	-17.188	8.438	30.000	pipe06	mat1	Nonstructural	
35	Bm36	-24.063	2.813	30.000	-17.188	2.813	30.000	pipe06	mat1	Nonstructural	
36	Bm37	-24.063	8.438	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	

MainToc / Overview / Beams / BeamMass / BeamSeg / Sections / Materials / Supports

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.

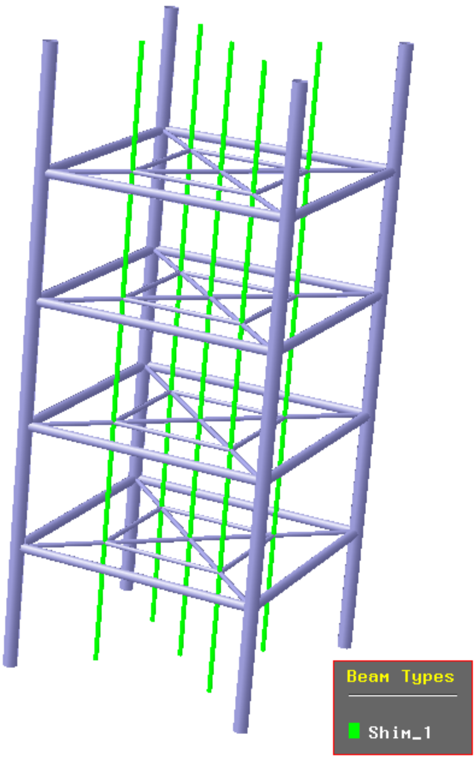
To verify you may use colour codcing from browser or from ***View|Options|Model|Use property colour coding*** or from the saved report under Tab Beams.

	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	

MainToc / Overview / Beams / BeamMass / Sections / Thickness / Materials / Supports

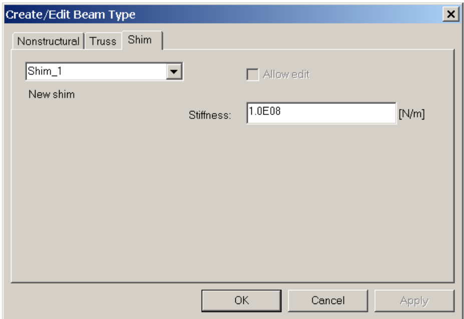
Shim elements

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.

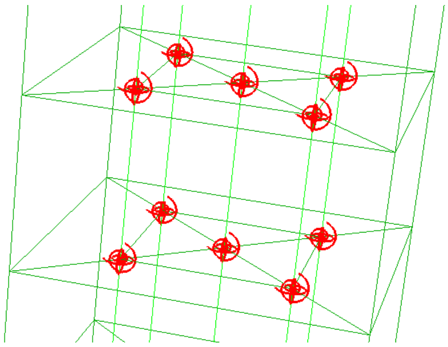


The conductors to the left are assigned a beam type Shim_1. When a finite element model is created spring elements (with stiffness perpendicular to the conductors) are automatically assigned at all connections between the conductors and the structure. The exception is when the end of a conductor intersects with structure.

See next page for more details.



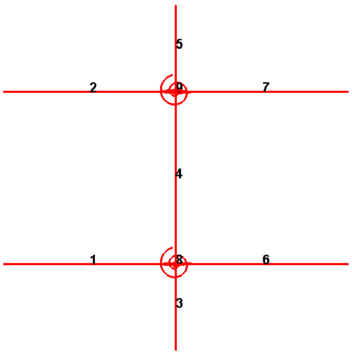
A shim property is manually created with stiffness 1.0E08 N/m.



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.

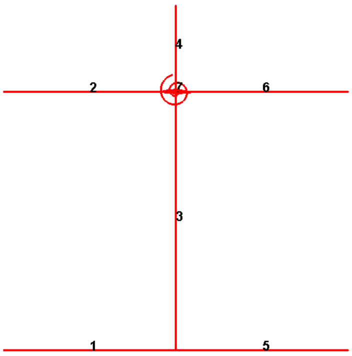
	A	B	C	D	E	F	G	H	I	J
1	Name	X-End1 [m]	Y-End1 [m]	Z-End1 [m]	X-End2 [m]	Y-End2 [m]	Z-End2 [m]	Section	Material	Beam Type
40	Bm46	7.500	7.500	20.000	2.500	7.500	20.000	Pipe200	St52	
41	Bm47	2.500	2.500	0.000	2.500	2.500	25.000	Pipe200	St52	Shim_1
42	Bm48	7.500	2.500	0.000	7.500	2.500	25.000	Pipe200	St52	Shim_1
43	Bm49	5.000	5.000	0.000	5.000	5.000	25.000	Pipe200	St52	Shim_1
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	Shim_1
46	Bm51	7.500	7.500	0.000	7.500	7.500	25.000	Pipe200	St52	Shim_1
47	Bm6	10.000	0.000	5.000	10.000	10.000	5.000	Pipe400	St52	

MainToc / Overview / Beams / BeamLoc / BeamMass / Sections / Thickness / Materials

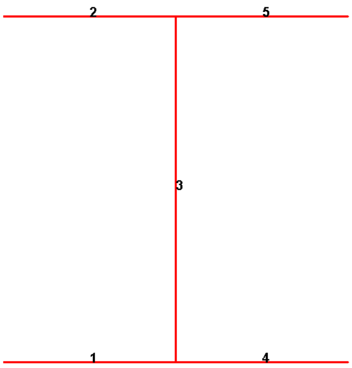


The vertical beam is assigned *Shim* and spring elements are assigned at each connection between structure and the shim structure. Hence two spring elements are created.

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.

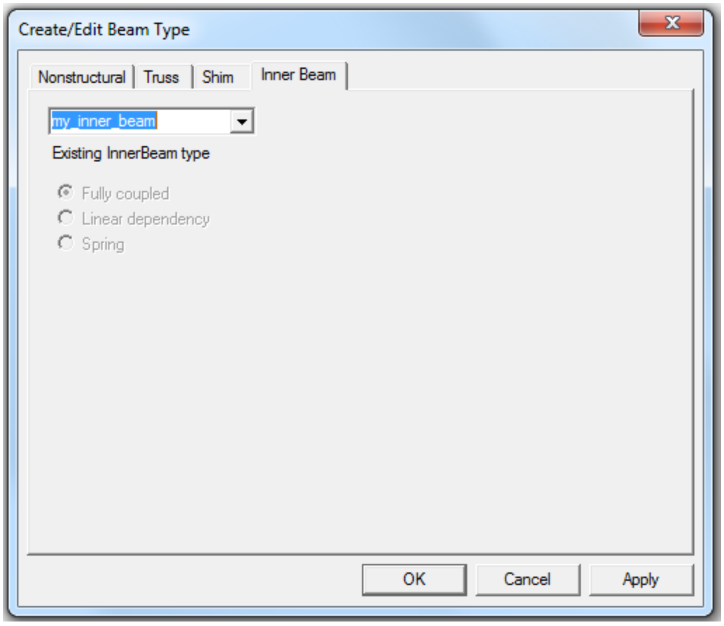


In this case, both ends of the Shim coincide with regular beams. Hence no spring elements are created.

Inner Beam

Fully coupled

All nodes are fully coupled in six degrees of freedom



Spring

All nodes in the inner beam are connected to corresponding nodes in the outer beam, using the specified spring stiffness per length unit.

Linear dependency

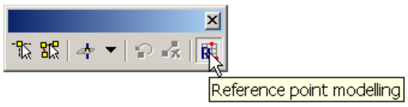
Feature is not available

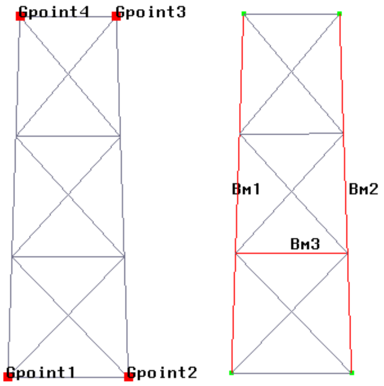
3.6.6 Reference point modelling of beam structures

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(GPpoint1, 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)

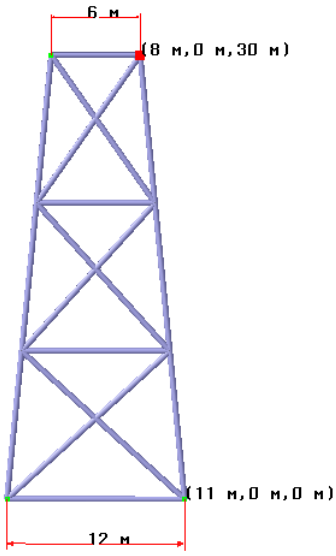
Gpoint4(1,0,30)

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.

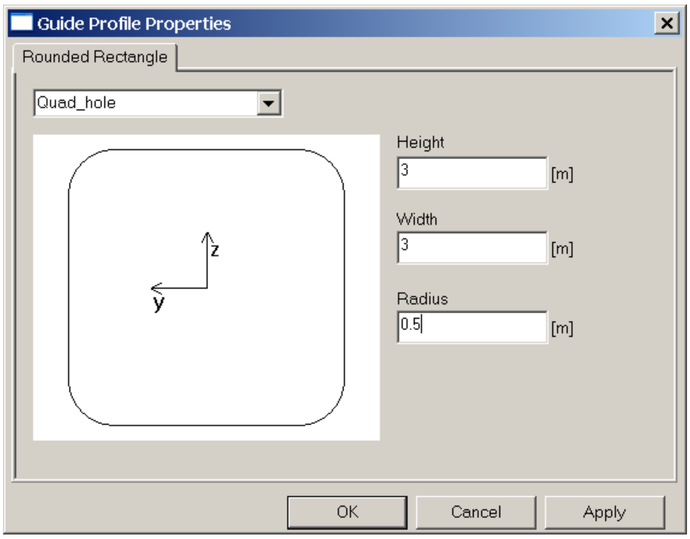
```
//
// Creating Guiding Points for use in parametric modelling
// Changing x value for all guiding points
//
Gpoint1 = Point(-1,0,0);
Gpoint2 = Point(11,0,0);
Gpoint3 = Point(8,0,30);
Gpoint4 = Point(2,0,30);
//
// Creating beams using reference point modelling
//
Bm1 = Beam(Gpoint1,Gpoint4);
Bm2 = 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.

3.6.7 Cut and punch

Cut and punch requires that a 2D profile has been created for use in the cut or punch operation. You may create several profiles for later usage. The profile is created from using the pulldown menu ***Insert|Profile***. 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).

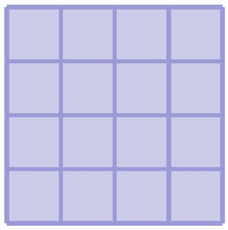
The example below shows the difference between cut and punch. When a cut line has been created a model curve or a feature edge may be inserted so that it may be referenced when applying boundary conditions or finite mesh details.

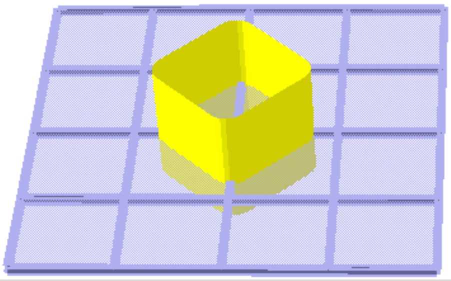


The profile Quad_hole has been created. Observe that the y and z-axis refer to a local co-ordinate system.

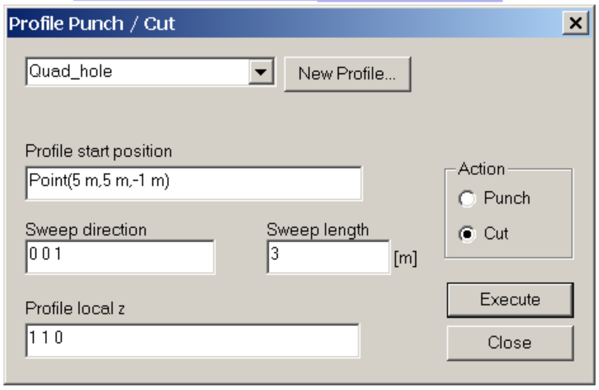
You may also create a circular profile by entering same values for Height and Width, the radius must be half the Height value.

This profile is now being used to cut and punch the below structure.



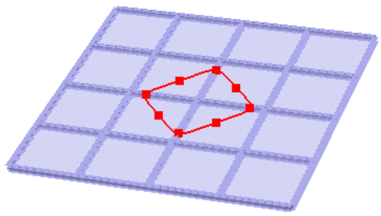


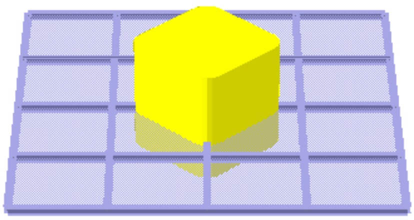
The cut operation is started from the pull down menu Tools|Structure|Punch.



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.

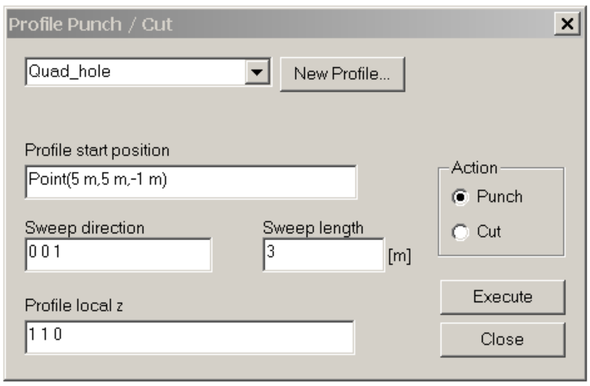
The input parameters to the left result in the following cut line on the model:

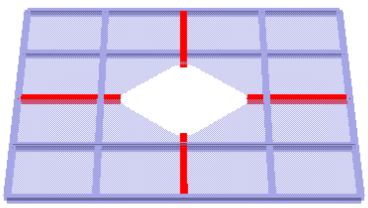


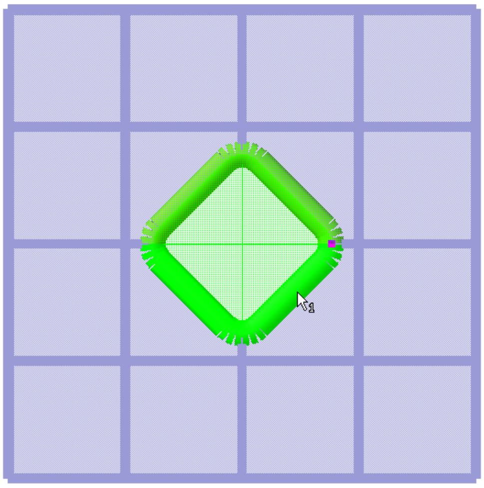


The punch operation is done in the same manner, but now *Punch* has been specified.

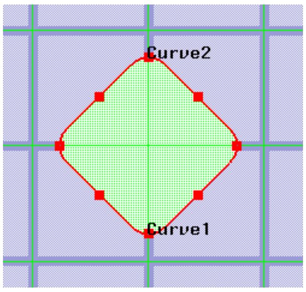
All material inside the volume defined by the profile and the height is removed. The plate now has a hole, and the beams are split in minor beams.

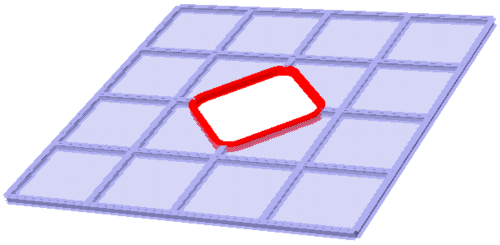




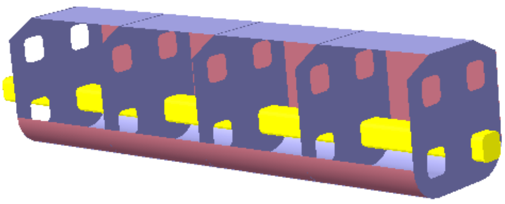


The cut line can now be used to e.g. insert beams. The first step is to create a model curve (**Insert|Guiding Geometry|Model Curve** and make sure the view specified shows guiding geometry. When clicking between two points the program prompts for which part to create a model curve – click on the part to perform operation (GeniE can not 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.

3.6.8 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.

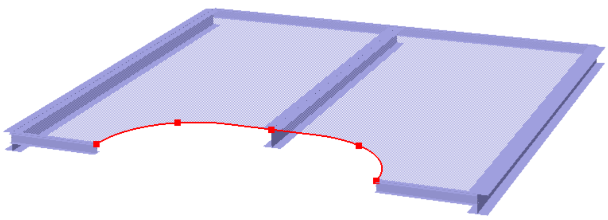
The *skin* operation assumes a series of curves that may be open or closed. A surface is adapted to stretch over the curves in the specified sequence; much like a skin can cover the ribs in a canoe. Notice that, in GeniE, the skin operation leaves a continuous surface, there are no sharp bends over the 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.

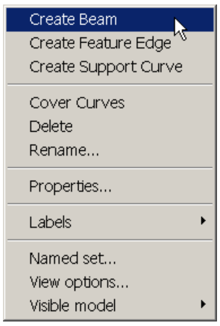
In the following a number of practical examples are given on how you can create curved geometry. Additional examples may also be found in Chapter 3.20.

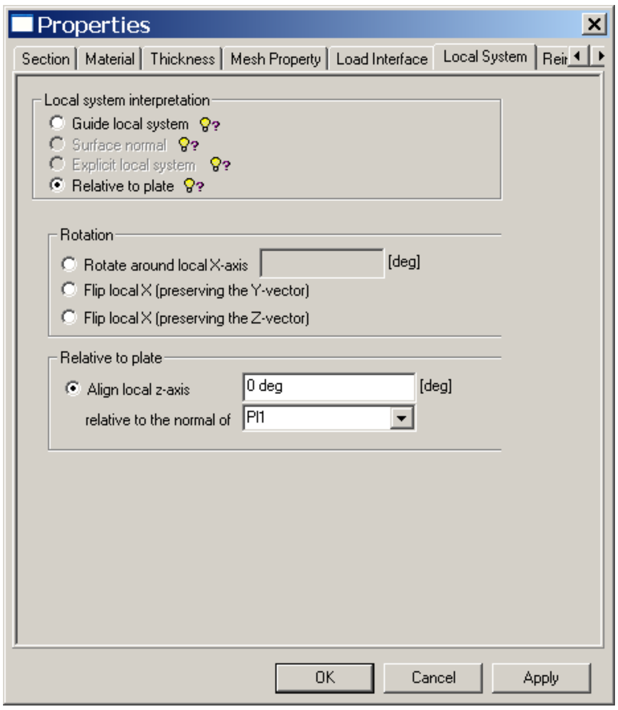
Observe that if you do not have access to the *Curved Geometry Extension* in GeniE you can not see any of the commands related to curved geometry.

3.6.8.1 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.





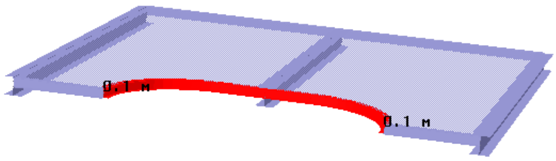
You may also control the orientation of the curved beam. By selecting the beam, RMB and then select *Properties|Local system* you have access to the dialogue showed to the left.

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 Pl1.

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));

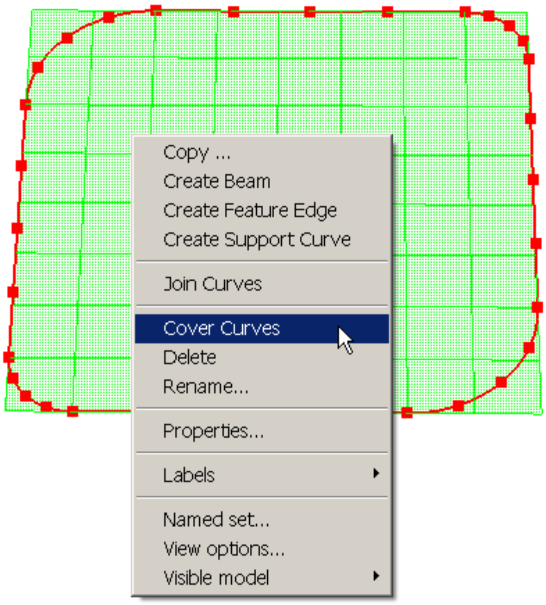
This command will insert an offset equal to the distance from local centre of gravity to the beam edge (or the half height of the beam). In this case an offset of 0.1 m is applied since the profile height is 0.2 m. If you want to move below the plate, then add half plate thickness to the offset vector.

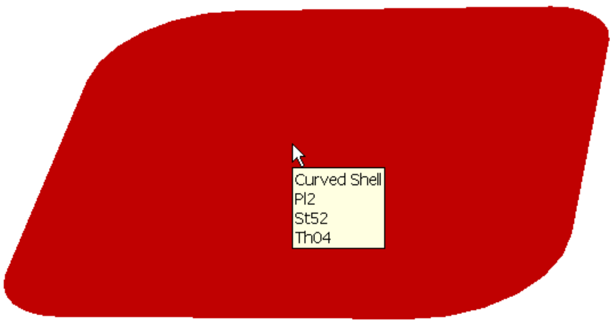


The curved beam Bm1 is now flushed to the plate.

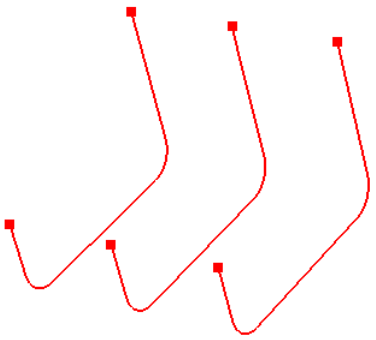
3.6.8.2 Fill a plate inside a closed guide curve circuit

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.



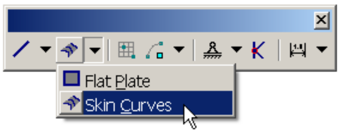


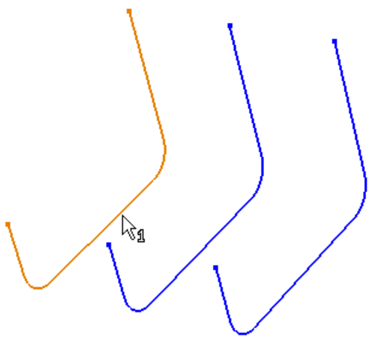
3.6.8.3 Insert a curved beam and plate between guide geometries



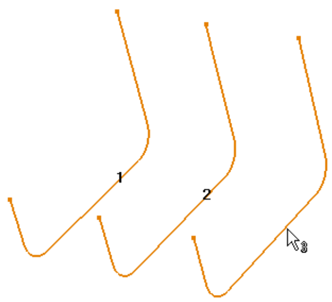
Three guide lines have been established from a) ***Insert Guiding Geometry|Guide Line*** and ***Guide Arc Elliptic*** and b) selecting lines, RMB and *Join Curves*.

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.

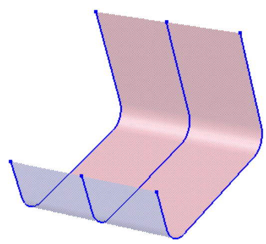


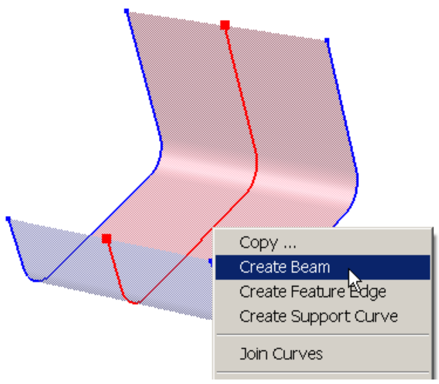


You are prompted for input when modelling in the graphic window. In the picture to the left, the first line is prompted for. Click the line and the mouse tooltip prompts for line number two. If you have other objects than guiding lines, it may be advantageous to use filters to help you pick lines only.

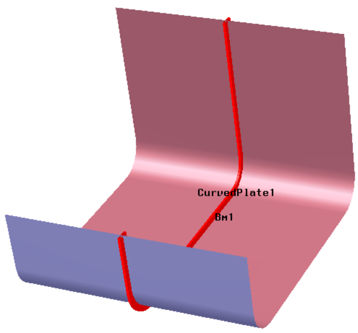


Guide lines one and two are already given, and GeniE waits for definition of line number 3. When this has been done, you need to click line number 3 again to complete the command input. A curved plate is then inserted between the lines.



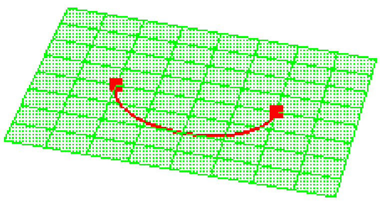


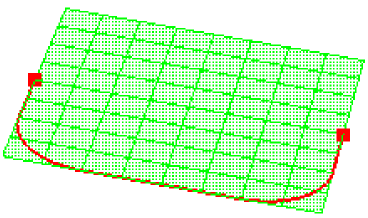
To insert a beam select the guide line, RMB and *Create Beam*.



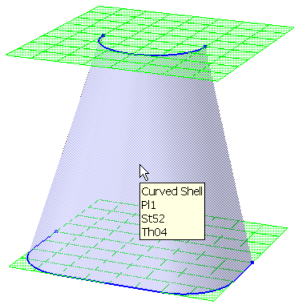
3.6.8.4 Transition between a rectangular and circular shape

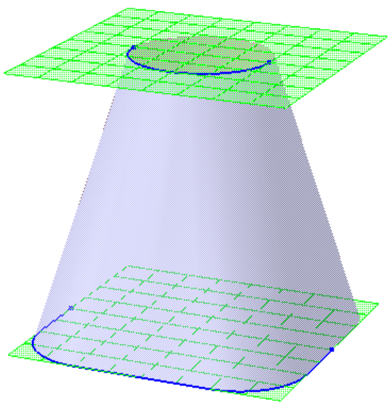
This example shows how to make a transition between a rectangular and a circular shape like in the transition from the pontoon to the column in a semi-submersible. Focus is on creating the hull, but inside plate structure has also been modelled. To insert curved beams, model curves need to be inserted so that they can be referred to when creating beams. Eccentricities may be applied from the Command Line Interface as described above.



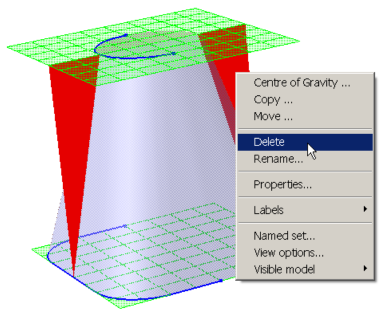


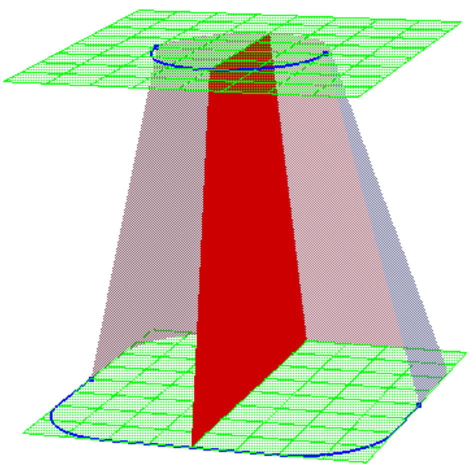
Two guide curves have been inserted to represent half the transition area. This example assumes that the structure is symmetrical in the vertical plane. A curved plate is inserted when using a skin operation and referring to the bottom and upper geometry curves.



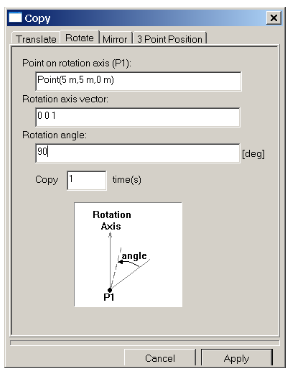


The outer hull plate is now created by copy and mirror operation. A vertical inside plate is inserted by referring to upper and lower guide plane. To remove superfluous material, the plate must be split (Tools|Structure|Split) and delete the parts outside the hull.

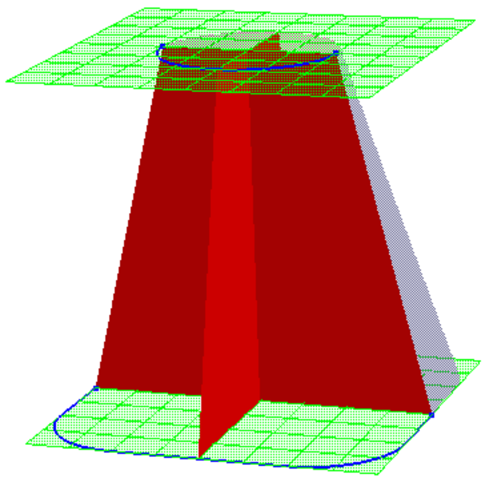




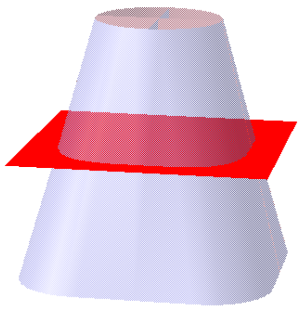
The vertical plate is now trimmed to the hull by a combination of split and delete.

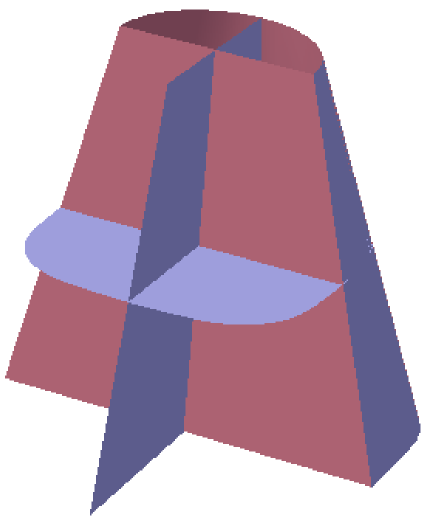


To insert a new vertical plate perpendicular to the first, select the first, copy and rotate 90 degrees.



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.

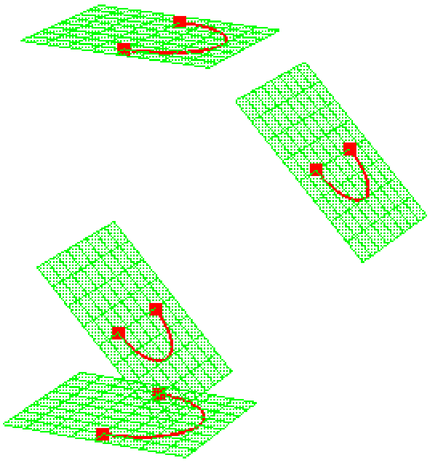




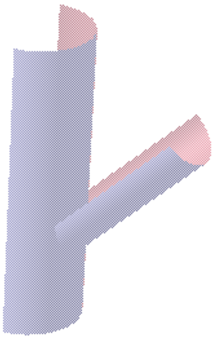
The transition part that includes trimmed vertical and horizontal plates.

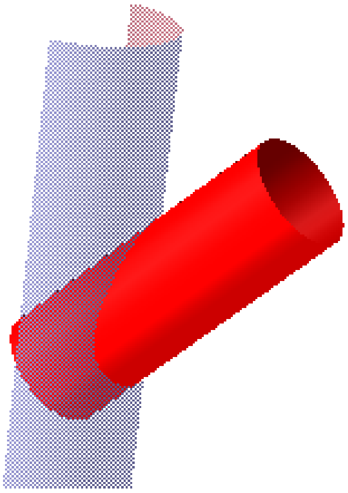
3.6.8.5 Two tubes intersecting each other

This example focuses two tubes that intersect each other. Trimming has also been done.

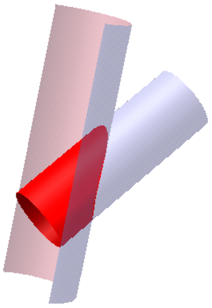


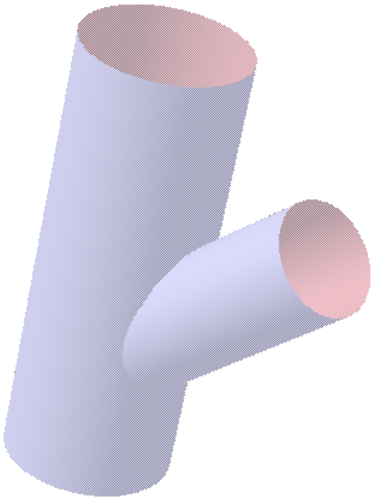
Guide curves have been inserted to define boundaries of each half cylinder. Curved plates are inserted by using skin curve operation.





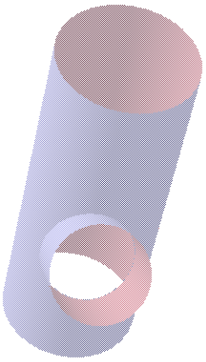
Copy and rotate (180 degrees) the lower part of the inclined tube gives a complete brace. To trim the brace it is necessary to split it and delete interior parts.





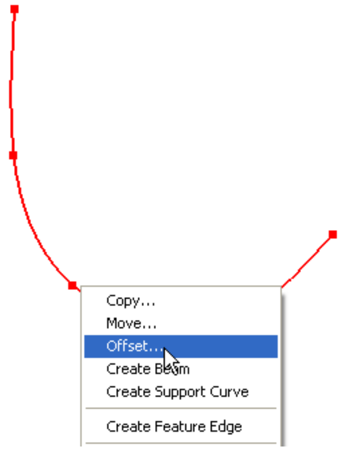
The complete vertical chord is now created by copy and rotate (180 degrees) the first chord part.

A trimming (split and delete) of the chord will result in a hole as shown on picture to the right.



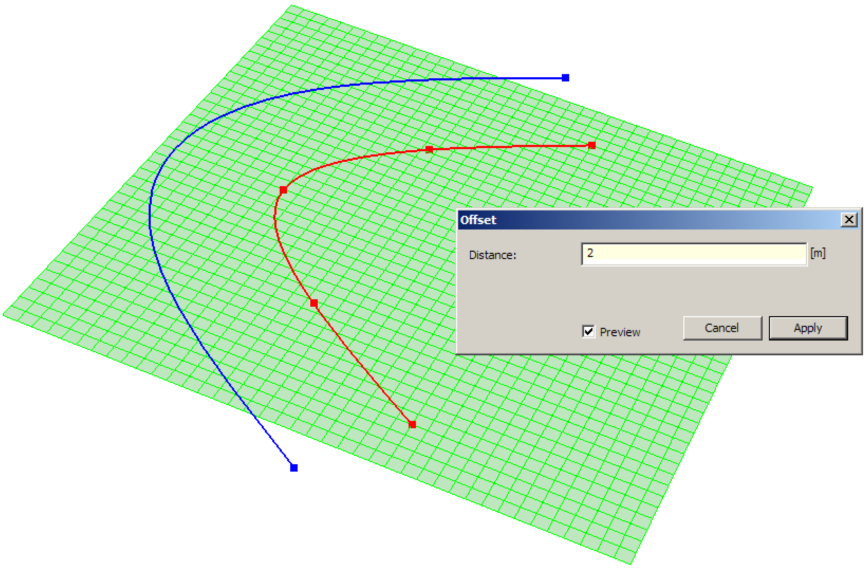
3.6.8.6 Offset

Offset lets you create an offset copy of a planar curve or a curved shell.

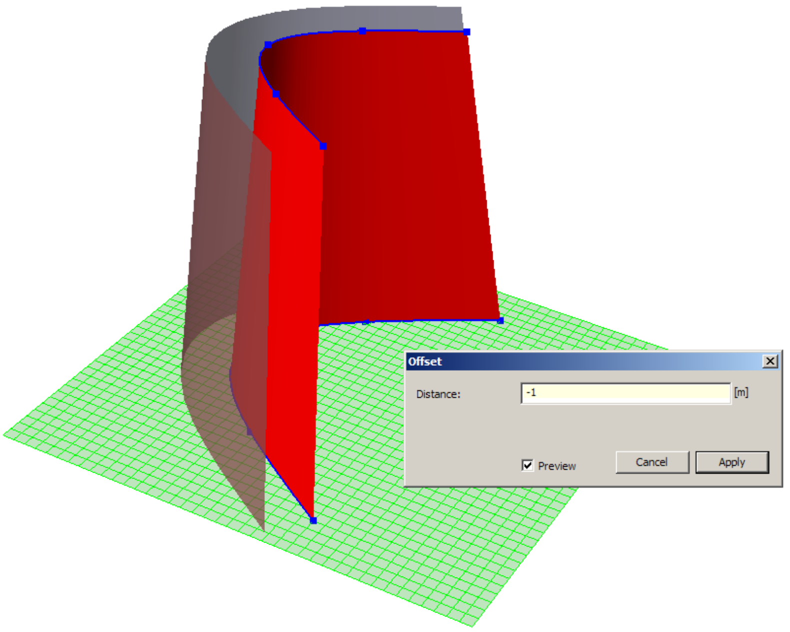


Open the offset dialog by rightclicking on your model and select *Offset*.

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.

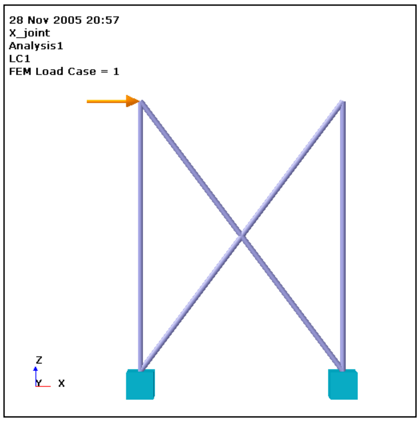
- Export a clean journal file (***File|Export|GeniE journal file (JS)***). After categorizing all the beams in the model, the most important beams (typically jacket legs) are exported first. Beams that should snap to the jacket legs are exported next, and so forth until all beams in the model have been 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*);*

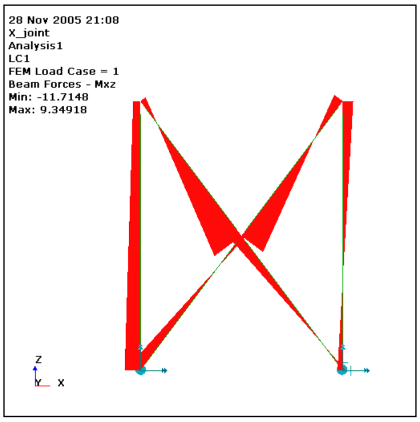
  The resulting journal file (the clean journal file) now contains new headings (Structure, level *n*) indicating the categorization of beams (i.e. in the order from more important to less important beams).

- 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.

3.6.9 Release X-joints

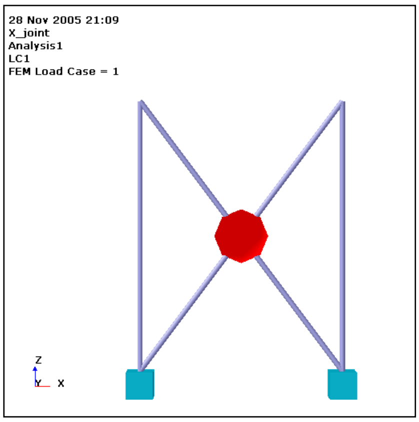
It is possible to release X-joints so that the beams are free to move independently. The example below shows a model prior to releasing the X-joint and how to release the X-joint.

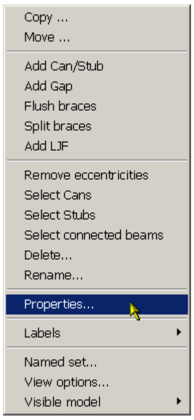


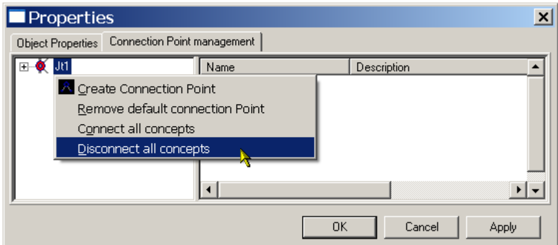


The beams in the X-joint are connected and both beams have displacements and bending moments because of the horizontal point load.

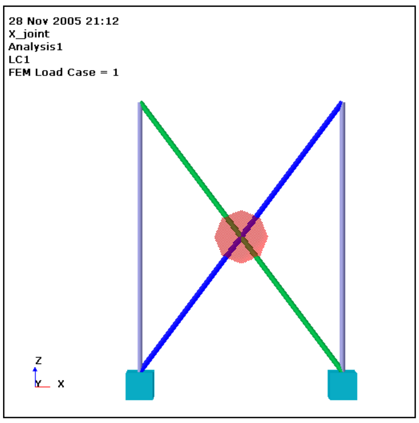
To release the beams in the X-joint the procedure is as follows:

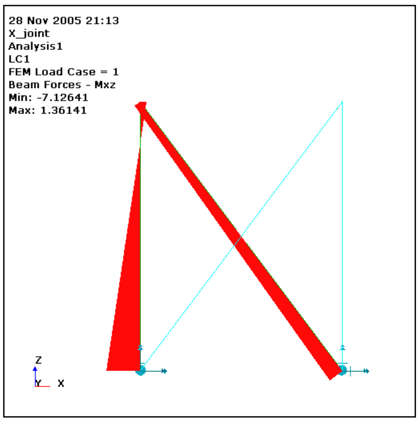






Insert a joint and select *Properties*. From *Connection Point Management* you release the beams in the X-joint by *Disconnect all concepts*. The beams may be connected again by selecting *Connect all concepts*. When you have disconnected the beams, you will see a colour coding of the released beams. If you later double click on the joint you will also see the same (you are then in segmented modelling view).

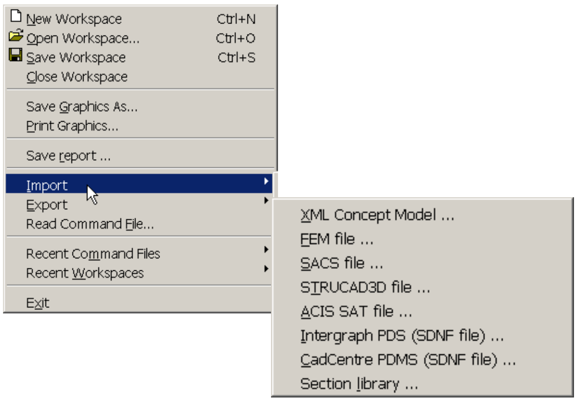


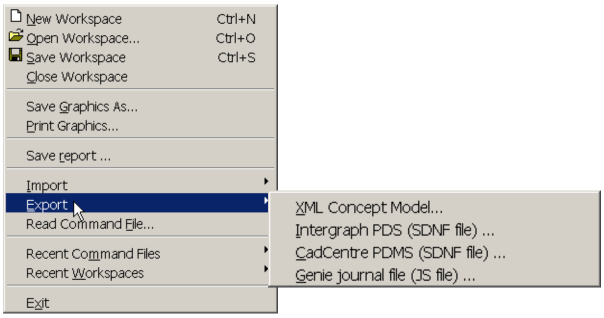


When the beams are disconnected in the X-joint one of the beams is deflected and has a bending moment due to the horizontal point load.

3.7 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***.

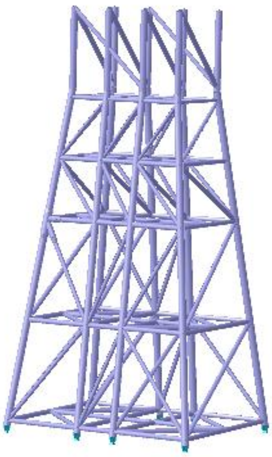


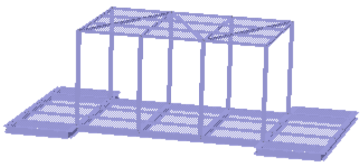


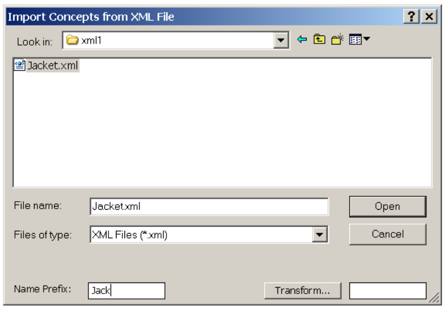
3.7.1 From/to XML

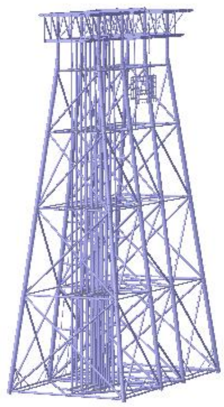
This option should be used when you want to share your model with others in stead of using the journal file. Your model is now stored without the history as is the case with the journal file. The XML format is also compatible over time, meaning the XML file can be imported to newer program versions of GeniE.

Another use of the XML format may be when you want to build a model from several minor subsets of the model created in GeniE. As such it is similar to the superelement technique used in SESAM, but the difference is that you don't need to specify supernodes at the boundaries. Note that no loads are supported on this format meaning all loads need to be specified after the complete structural model has been assembled. The example below shows two subsets created in different GeniE runs and assembled in a third run.









When importing the subsets the name prefix option has been used (Jack and Top). Observe that all names will receive the prefix, for example BM36 from the jacket subset becomes JackBm36 in the global model. The same apply for example for section and material names.

It is also possible to model subsets using different Cartesian co-ordinate systems. When importing the various XML models you may refer to transformations to ensure that the models intersect at the correct boundaries.

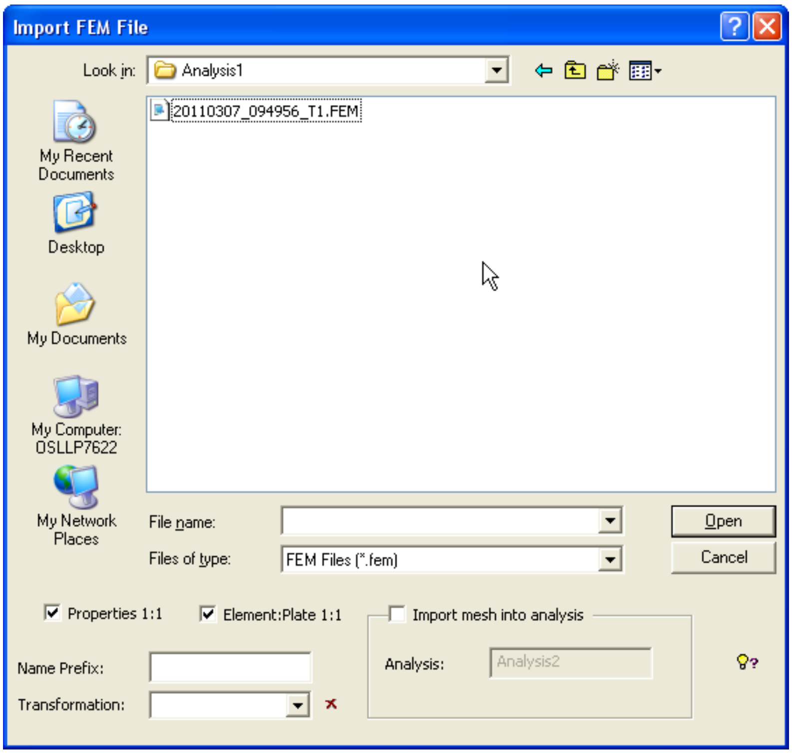
3.7.2 From/to FEM

When importing a FEM file (created by e.g. Preframe, Prefem or Patran-Pre) the finite element model is converted to a concept model. For beams there is a 1:1 relation between a beam finite element and a beam concept. There is a name recognition whereby a beam finite element number 25 becomes e.g. Bm25. If you want higher level concepts (for example a continuous leg) then select the beam concepts, RMB and *Join Beams*.

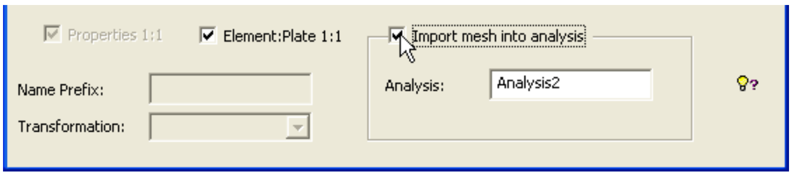
For plates GeniE will make the largest plate concept possible given that the plate finite elements are in the same plane and have the same properties (e.g. thickness and material). If you want a 1:1 relation between plate finite element and plate concepts you must tick off this option in the import dialogue box.

When “ Properties 1:1” is checked, GeniE imports all sections and materials according to the following priority: Any explicit name will be used (for example the FEM-file was created in another GeniE session, i.e. the FEM file contains concept information)

- Type prefix + section/material number (for example section number 23 on the FEM file becomes Sct23, in this case the FEM file has no concept information)
- Default name schema as found under ***Tools|Customize|Default Names***



When “Import mesh into analysis” is checked, GeniE imports the FEM file and create an analysis activity for the imported model. By default the created analysis activity only include the meshing part. User can add other analysis activities whenever it is needed.



In this mode, GeniE will allow user to change certain properties on the imported model and then export changes into FEM file without regenerating the imported finite element mesh. This implies that the imported element/node numbers are kept, and this saves much time when mesh regenerating large models.

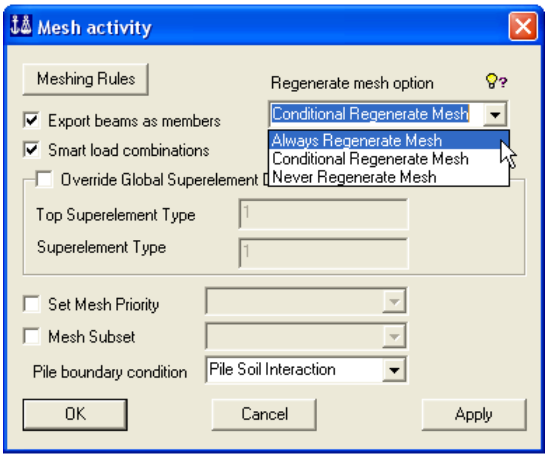
Properties that can be changed without changing the element/node numbers:

- cross sections for beam
- thickness for plate

- materials
- explicit loads (only point and line loads on beams).
- Local coordinate system for beam
- Eccentricities.

Note that other changes will make it necessary to regenerate the mesh causing the element/node numbers to change. An example of such a change is adding a beam or a plate.

By default when importing a FEM file, the “Regenerate mesh option” is set to “Never Regenerate Mesh”. Under such circumstances, if you have done changes that make regenerating mesh necessary and then run the mesh activity by clicking on the button “Start” on the activity monitor, the generated FEM file will not reflect the changes that are made. However user can bypass this default setting by either change the “Regenerate mesh option” or explicitly type in the JS command in the GeniE command window.



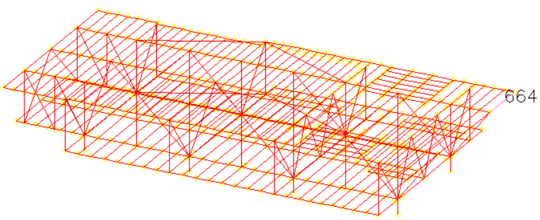
The FEM import retrieves all data found on the FEM file, piles and overlapping beams included. For overlapping beams, the inner and outer beams should have the same FE mesh density prior to import. However there are some data imported that you neither can see nor alter under the GeniE default view, for example data about line load on 3 noded beams. Those data need special care as they are not visible, sometimes wrong operation may cause them lost.

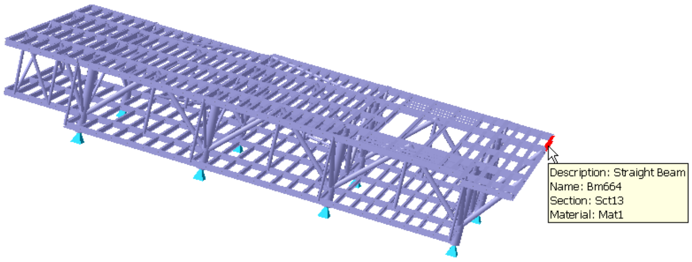
Observe that when importing hinges these are subject to name mangling, i.e. equal hinges will be reused. Hinges modelled in e.g. Preframe are finite elements and when translating these to concepts in GeniE hinge properties will be created and connected to relevant beam ends. When e.g. 14 hinge elements are equal on the FEM file, they will appear as one single hinge property (name mangling).

The main purpose of the FEM file import is to import data from other SESAM programs and systems (where data is converted to FEM format prior to import in GeniE) and not to act as a neutral storage format. The reason for this is that the FEM file format does not include all concept information. For the purpose of neutral storage the journal file should be used. You may choose to use the journal file generated during modelling activities (including the history) or a clean variant of the journal file (without history), see Chapter 3.7.6.

To export a FEM file use command ***Tools|Analysis|Export FEM***. This can be done after creating the mesh using the command ***Tools|Analysis|Create Mesh***.

The example below shows a model that was created in Preframe in 1989 and imported to GeniE. The beam finite element number has become Bm664 and section type has become Sct13 (it is no. 13 in the Preframe model).

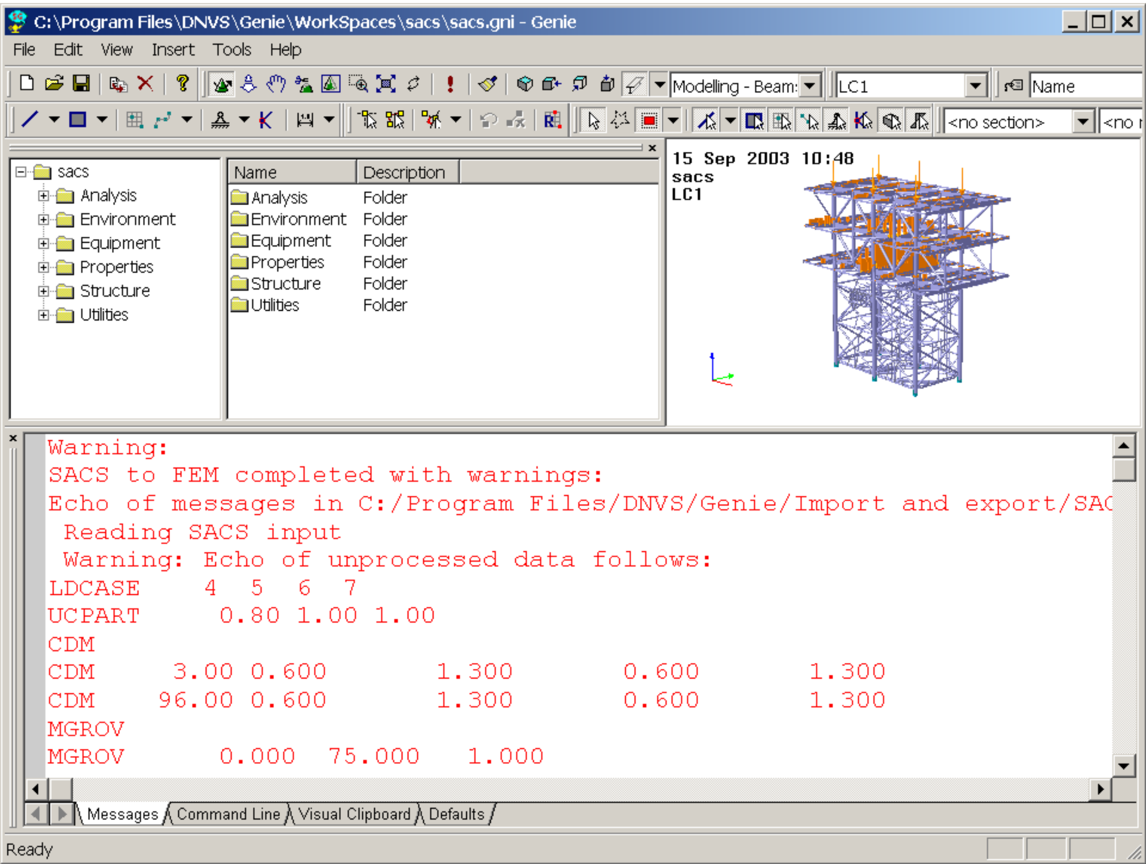




Note that the units given on the FEM file will be used when importing the model.

3.7.3 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).



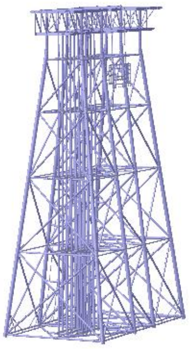
When importing a SACS or StruCad3D model you should specify which units is part of the model. The units may be specified when activating the ***File|Import|SACS File*** or ***File|Import|STRUCAD File***. The units are brought over during the FEM-file conversion prior to the actual import into GeniE

3.7.4 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.

3.7.5 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.



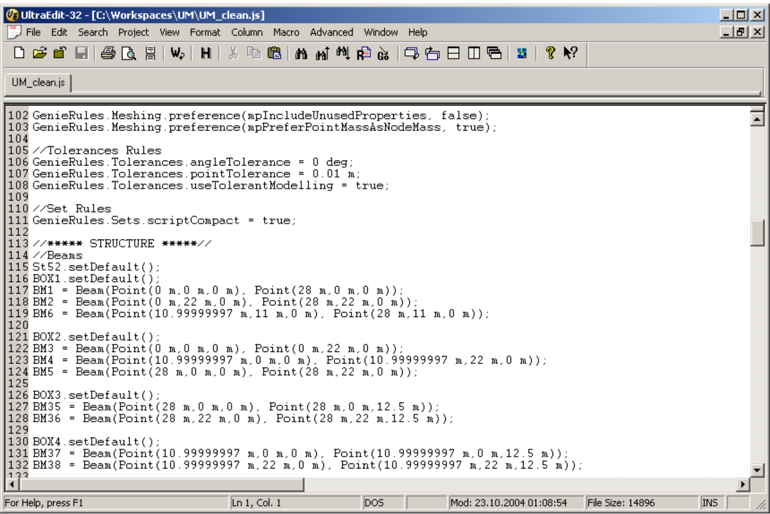
```
0 "SESAM Genie - Export to SDNF  (CadCentre PDMS flavour)"
"Name of Design Code"
0
Packet 10
"meters" 2996
1 10 0 1 "beam" "Bm1" 1
"Sct1" "Mat1" 0. 0 0
0. 0. 1. -32.6750030517578 -18.371001303196 -88.3920059204102 -32.6750030517578 -15.9260005950928
0. 0.
0. -0.755000054836273 0. 0. 0. 0.
0 0 0 0 0 0 0 0 0 0 0 0
2 10 0 1 "beam" "Bm2" 1
"Sct1" "Mat1" 0. 0 0
0. 0. 1. 7.62000036239624 -18.3704013228416 -88.3920059204102 7.62000036239624 -15.9260005950928
0. 0.
0. -0.755600035190582 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 0
```

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.

3.7.6 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.

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.

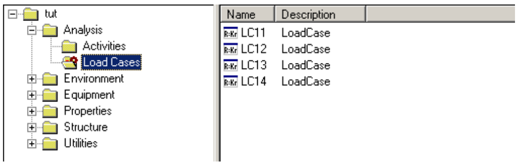
3.8 Loadcases and load combinations

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.

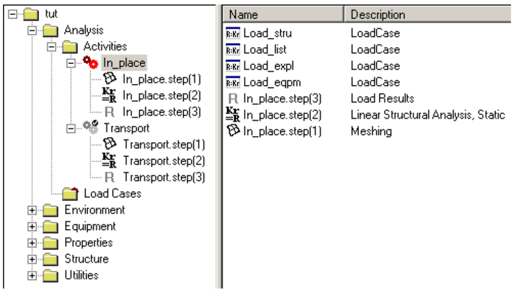
A loadcase may be used in a linear structural analysis where loads are being analysed. For a dynamic analysis where the mass model is of equal importance a loadcase may be treated as an addition to the structural mass. This is further explained in the Chapter describing how to make a finite element model.

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.

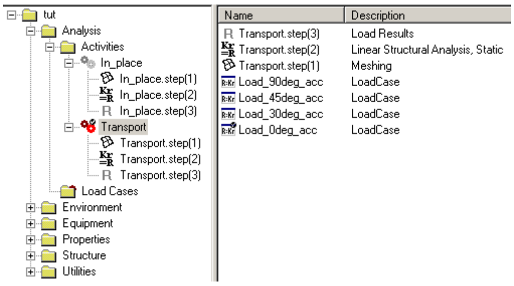
See Chapter 3.13 for more details on multiple analyses.



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>*.

3.8.1 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.

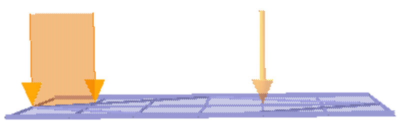
There are some differences between these load sources that you should be aware of:

- 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.

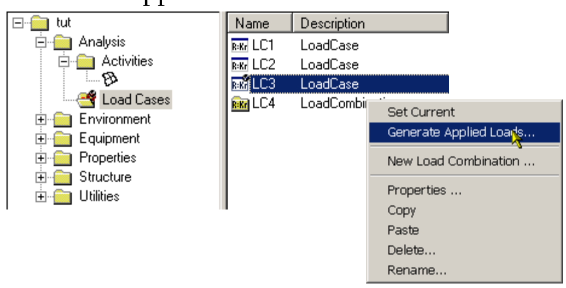
An explicit load or an equipment are considered a concept and loads are applied to the analysis model when there is a intersection between a structure concept and a load concept. Hence:

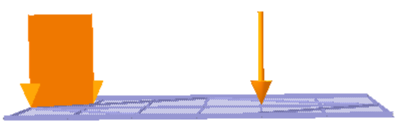
- *Applied Loads* result from applying concept loads to a structure.
- If the sum of concept and applied loads differ, some loads have not been properly applied.

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.

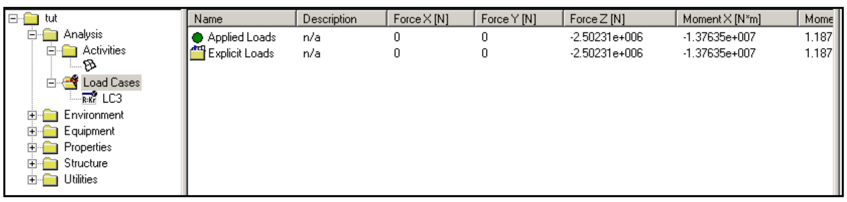


One line load and one point load applied to the model. Applied loads calculated when selecting Generate Applied Loads

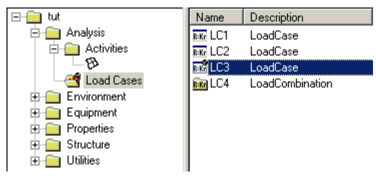




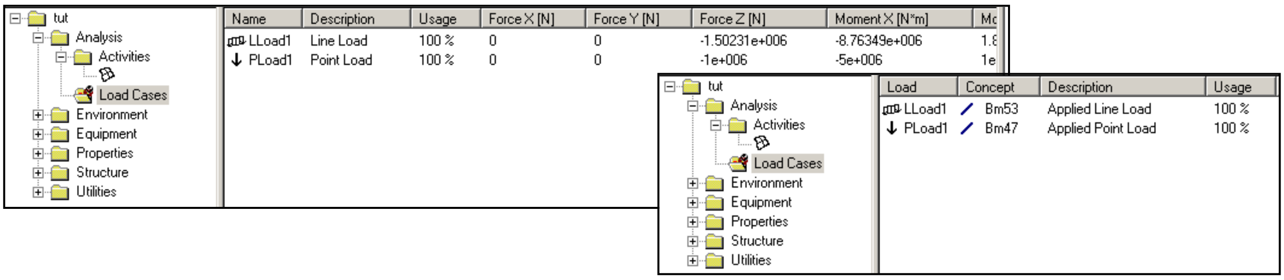
Applied loads have been calculated.



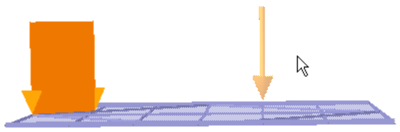
To verify whether the concept loads are fully loaded to the structure double click LC3 in the browser.



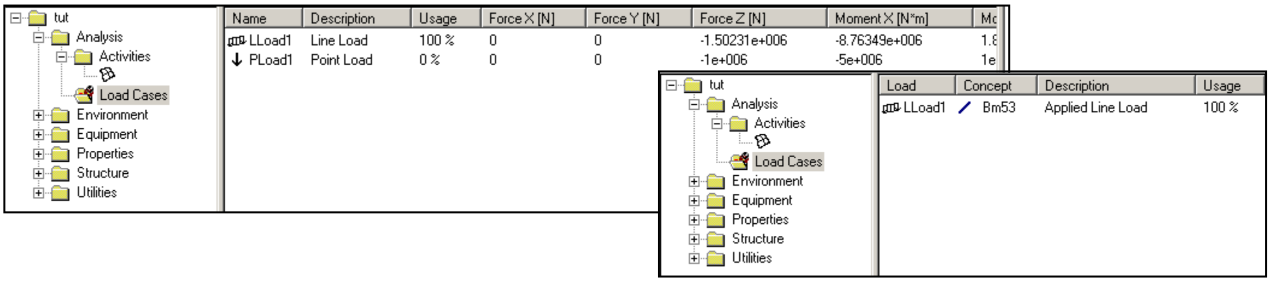
The reporting of concept and applied loads show they are 100% used:



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.

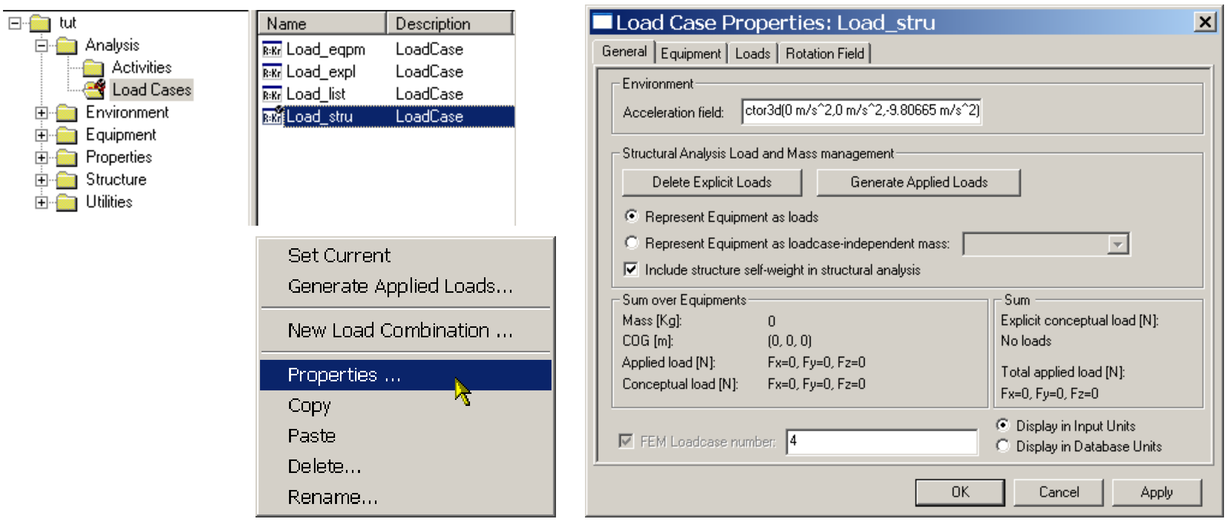


The point load will not give applied loads since it is not connected to structure and the reporting of concept and applied loads shows that the concept load is contributing with 0% to applied loads. If you are using the default visualisation options, you can also see from the graphics that no applied loads have been created (transparent colour).



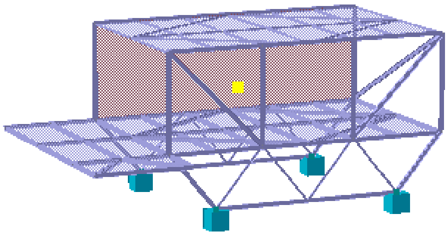
3.8.2 Making basic loadcases and account for selfweight

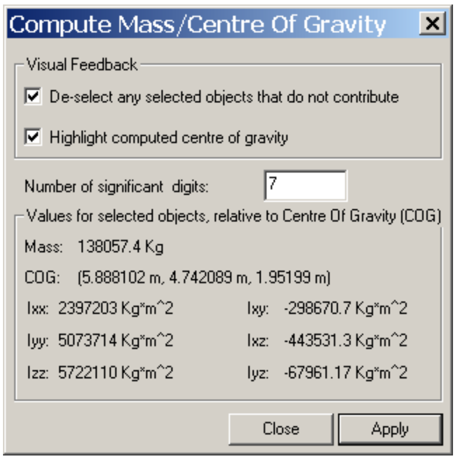
A loadcase is defined by the command ***Insert|Loadcase***. From the browser area you may select it, RMB and *Properties* to see the content of it. You can then re-assure that e.g. acceleration field is as expected and that the masses and loads correspond. From this dialog you also decide whether to 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.



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.

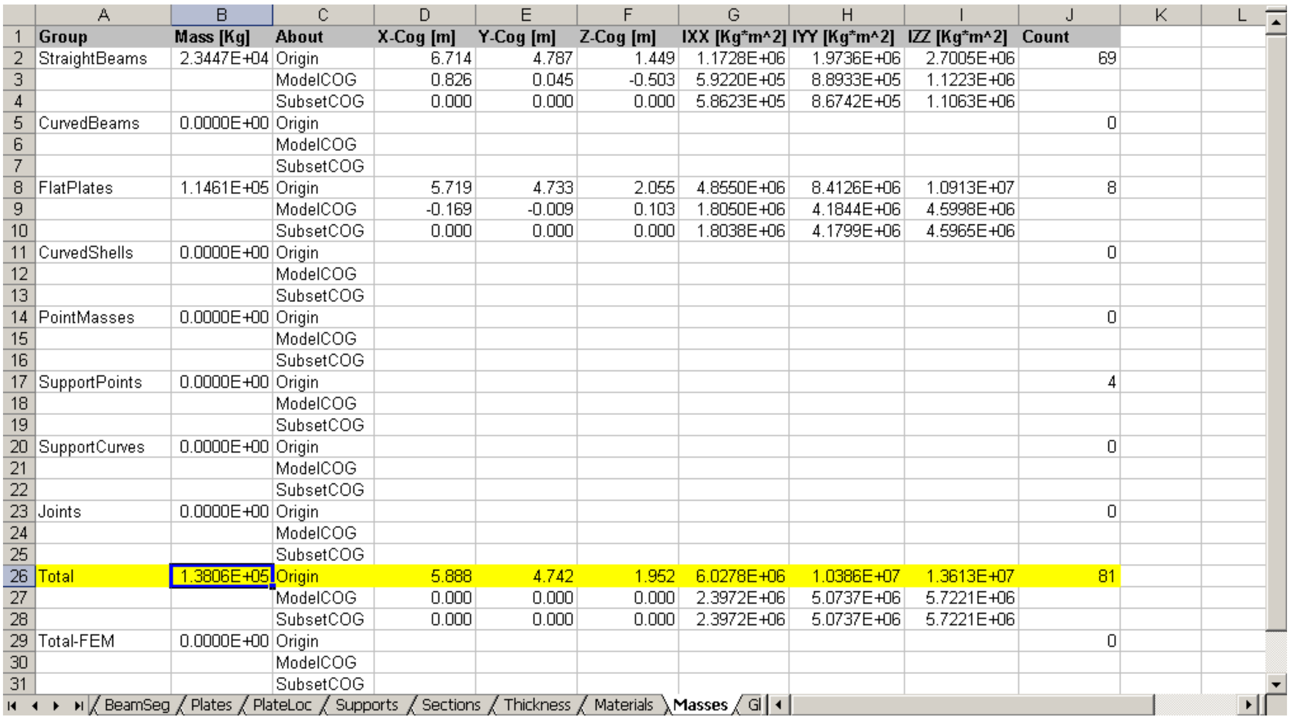
The Property dialogue box does not document the total structure mass. You do this either by selecting the whole structure (or parts of it), RMB and *Centre of Gravity* or investigate a saved report to verify the mass.





The numbers are shown with 7 significant digits.

Mass, centre of gravity and relevant moments are shown.



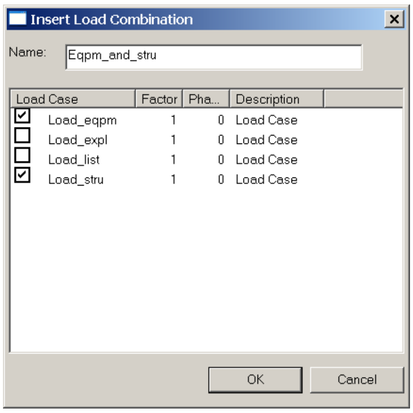
1	Group	Mass [Kg]	About	X-Cog [m]	Y-Cog [m]	Z-Cog [m]	IXX [Kg*m^2]	IYY [Kg*m^2]	IZZ [Kg*m^2]	Count
2	StraightBeams	2.3447E+04	Origin	6.714	4.787	1.449	1.1728E+06	1.9736E+06	2.7005E+06	69
3			ModelCOG	0.826	0.045	-0.503	5.9220E+05	8.8933E+05	1.1223E+06	
4			SubsetCOG	0.000	0.000	0.000	5.8623E+05	8.6742E+05	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							

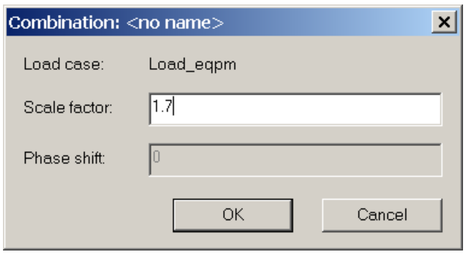
The above report created using ***File|Save Report|XML Excel*** shows the necessary details. The details are the same if you create a report in another supported format.

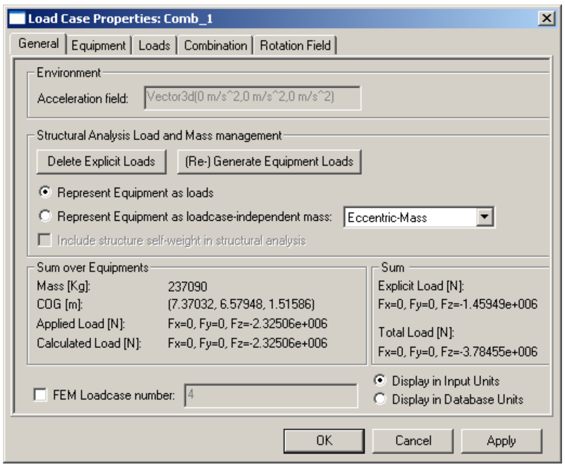
You may also select parts of structure and equipments in a loadcase to find the mass of the selection.

3.8.3 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*.

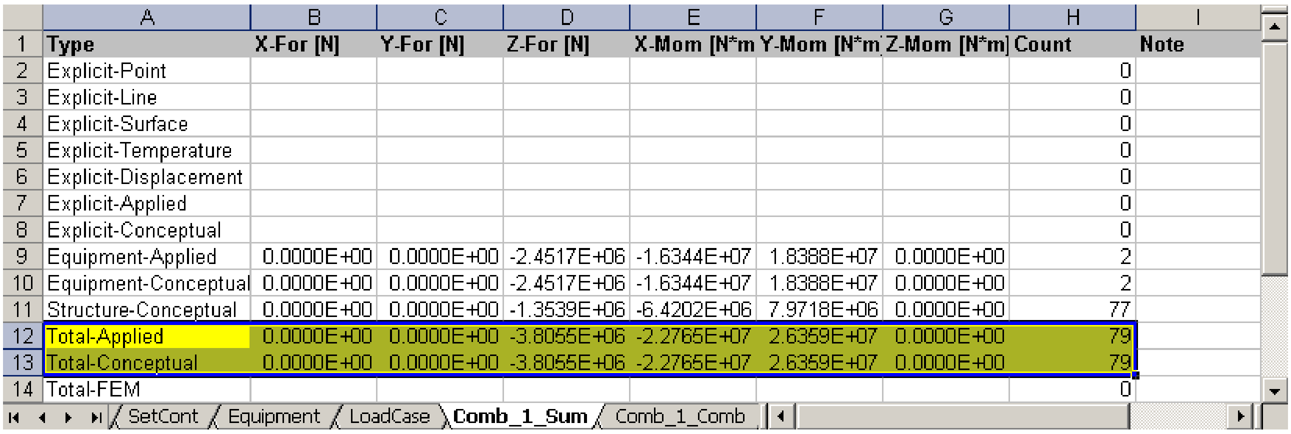






The total loads for a load combination may be found either from the load property sheet or from the saved report, both options are shown below.

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.

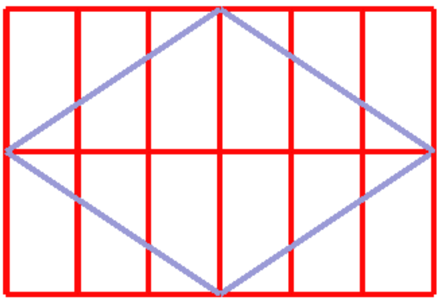
A load combination may include other load combinations, also known as nesting.

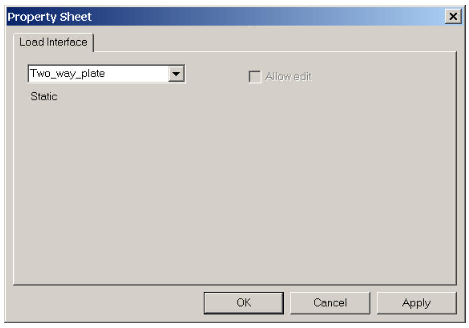
3.8.4 Making a load interface

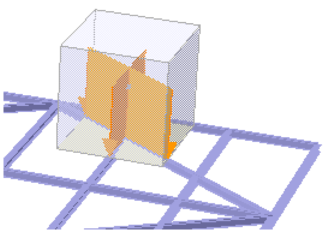
A load interface specifies which beams to receive line loads from equipments or weight list objects. If no load interface is specified all beams are loaded when there is an intersection between beams and equipment footprints.

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.

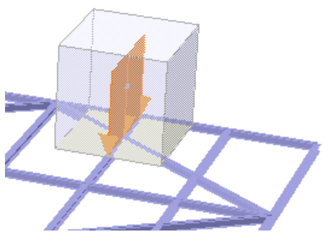
You create a load interface by selecting the beams to be part of the interface, RMB and *Properties|Load Interface*. The same load interface must be connected to relevant equipments in each loadcase where it shall apply. The example below shows that only beams in x and y direction shall carry load, the 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.



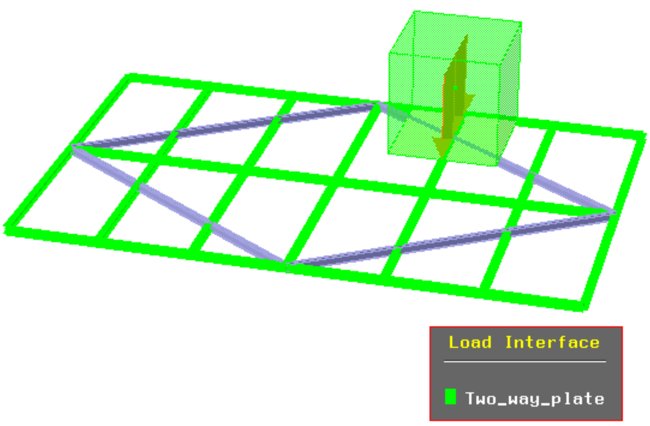




Line loads before load interface is connected



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*.

3.8.5 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.

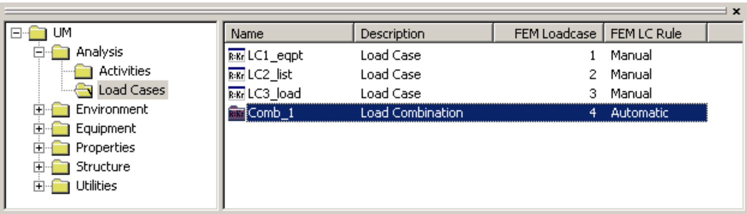
You may override the automatic numbering scheme by deselecting the option as found under ***Edit|Rules|Meshing***.

The logical loadcase sequence is:

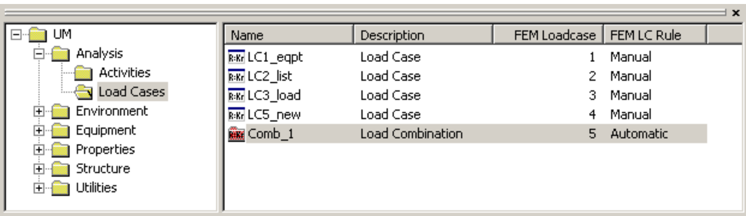
- Basic loadcases: Defaults numbers provided by GeniE, the user may override these
- Wave loadcases: Variable sequence numbering, the first wave loadcase may be defined in the wave load activity. See Vol. 2 for further details.
- Combined loadcase: Automatic numbering, but always after Manual and Variable. 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.

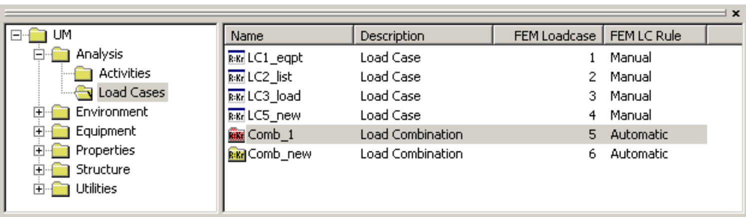
The examples below show how the automatic loadcase numbering scheme works and how the user may override it.



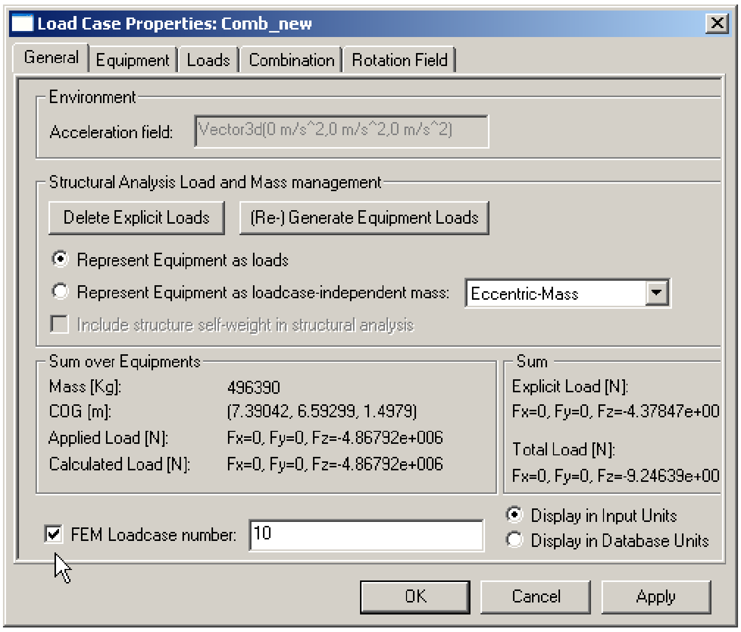
The model has three basic loadcases (manual FEM LC Rule) and one load combination (automatic FEM LC Rule).



A new basic loadcase has been inserted; hence the load combination Comb_1 receives FEM loadcase number 5.



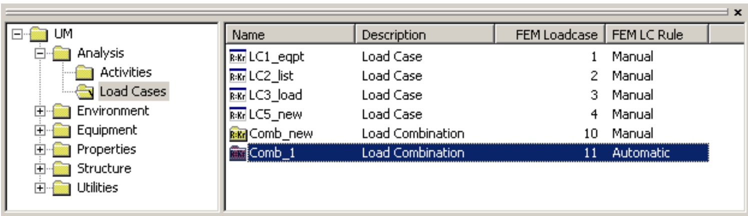
A new load combination Comb_new added, it receives FEM loadcase number 6 because it is using the automatic loadcase scheme



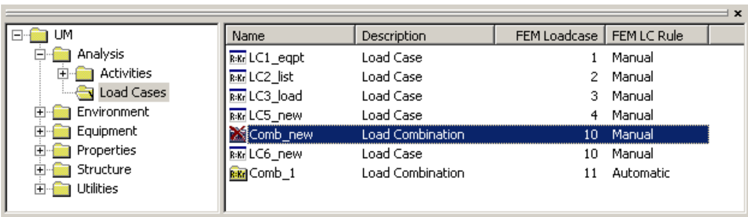
The load combination *Comb_new* has been selected from the browser and RMB to access the context sensitive command.

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.



The load combination *Comb_1* receives FE loadcase number 11 since *Comb_new* now follows the manual FEM LC Rule.

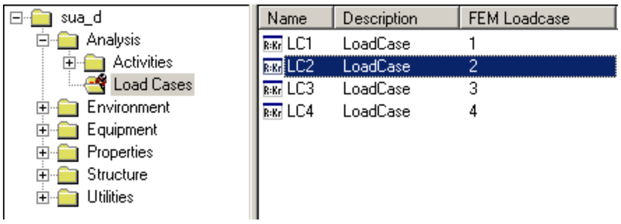


In this example a new loadcase has been added (*LC6_new*) and the FEM loadcase number has been set to 10. Since the basic loadcases have priority over the combined loadcases, the FEM loadcase number for *Comb_new* (which is using the manual FEM LC Rule) becomes invalid.

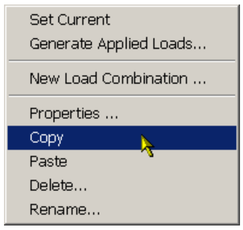
In addition to a graphic warning that the loadcase is invalid you will also be notified when running analysis. In this case GeniE will not perform the analysis, but stop after meshing has been performed. See Chapter 3.13 for more details.

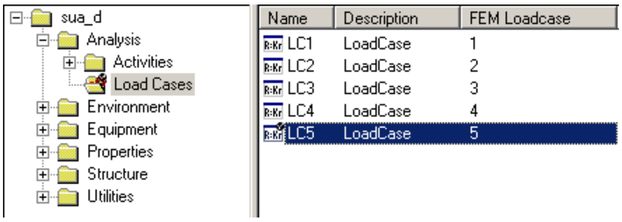
3.8.6 Copy a loadcase

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.

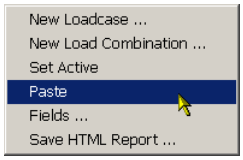


The model has four loadcases. Select loadcase LC2, RMB and choose Copy.





Place the mouse in browser Load Cases (or under an activity if multiple analysis) and choose Paste. Loadcase LC5 is now identical to LC2.



3.8.7 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
Analysis1.step(2)	Linear Structural Analysis, Static		
Analysis1.step(3)	Load Results		
Analysis1.step(1)	Meshing (Always Regenerate)		
LC1_eqpm	Reference to LoadCase	1	Manual
LC2_list	Reference to LoadCase	2	Manual
LC3_expl	Reference to LoadCase	3	Manual
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 Equipment modelling

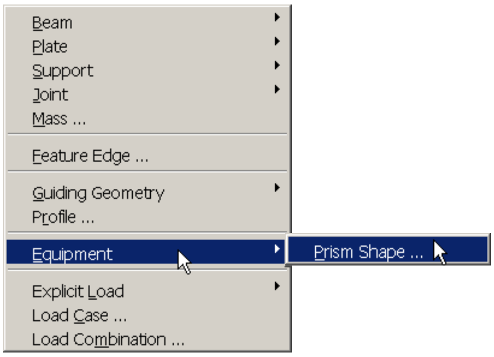
GeniE lets you model equipments and place these on the model. Based on the load calculation rule and load interface you have chosen, GeniE will calculate the reacting forces or masses to the model. In other words, when a finite element model is created, the equipment mass is converted to either a load or a mass.

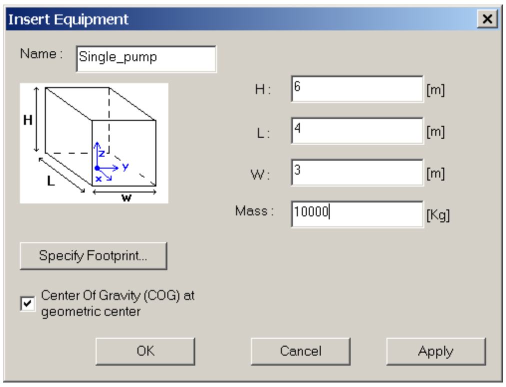
The normal steps are to create the equipments you would use in an analysis and then place them as needed in the various load cases. You may also build your library of equipments in a journal file. This will allow you to easily generate equipments you need for other projects.

When you have placed the equipments you should carefully check that the loads and masses are correctly calculated.

3.9.1 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.

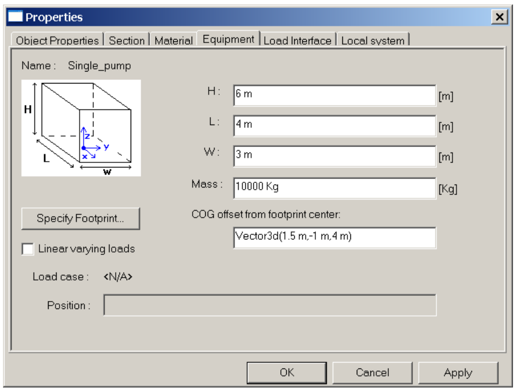




The equipments may be selected from the browser area (under Equipment) for editing and modifications. The following sections explain how to modify both c.o.g. and the foot-print.

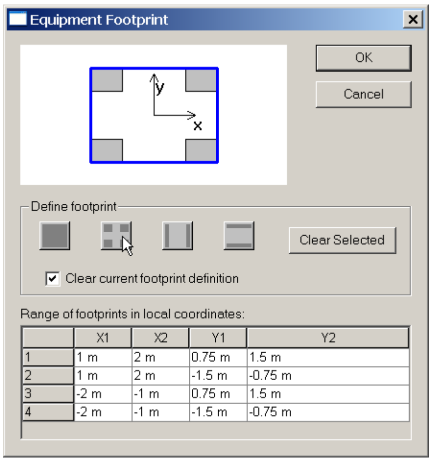
3.9.2 Editing the cog and footprint

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.

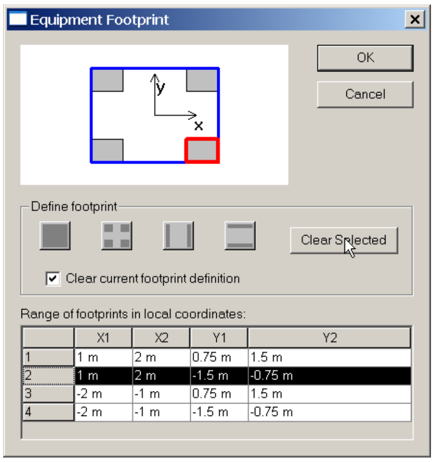


This example shows that the cog is located towards one of the corners and is 4 meters above bottom plane.

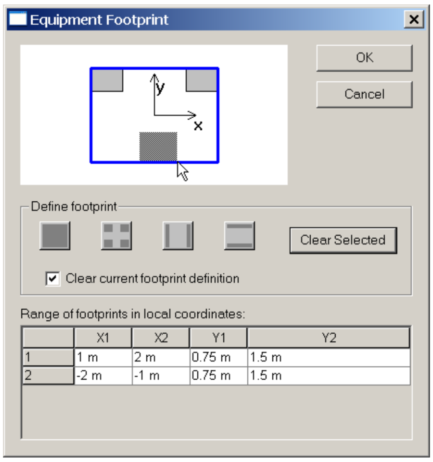
The footprint (or load distribution area) is per default the same as the area defined by the length and the width of the equipment. To change the footprint, select the equipment from the browser, RMB and click *Specify Footprint*. There are 4 predefined footprint layouts, all of these may be edited and 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.



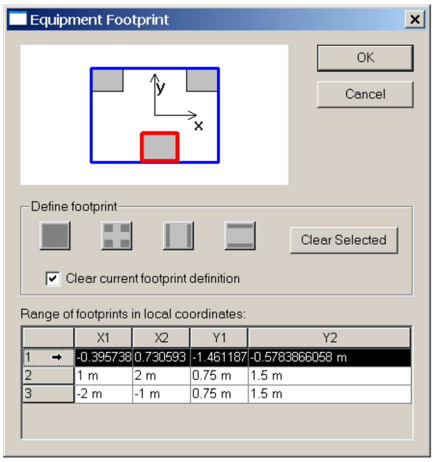
1. Select footprint template



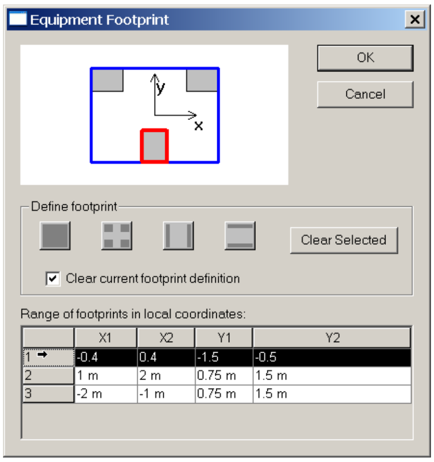
2. Click on 2nd and 3rd footprint and Clear Selected



3. With a "rubberband" select insert new footprint



4. Select the new footprint and change local co-ordinate values

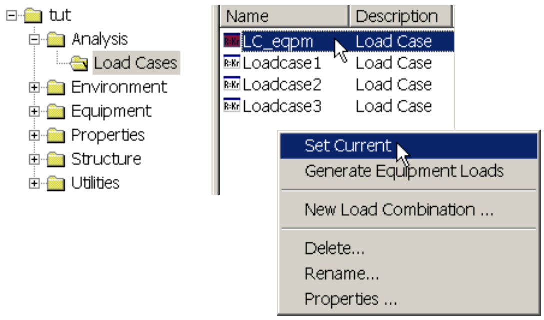


5. Final configuration with 3 support points

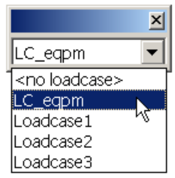
A footprint may also be larger and outside the bottom plane of the equipment. This technique may be used when defining blanket loads (or uniform distributed loads).

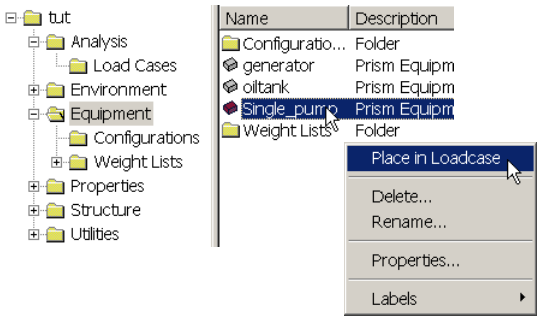
3.9.3 Placing the equipment

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.

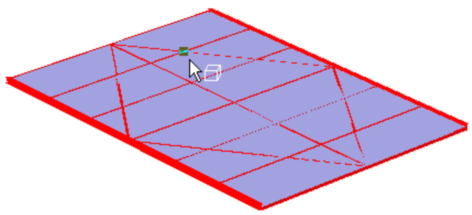


1. *Selecting the loadcase and set it to current. The graphic loadcase pulldown list also shows LC_eqpm.*

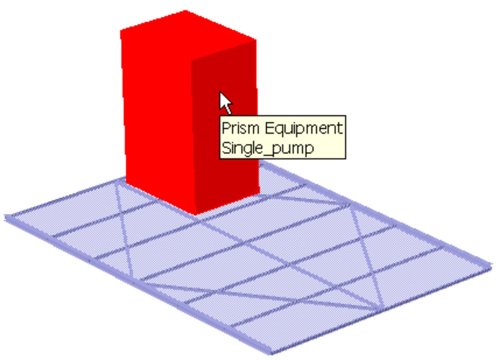




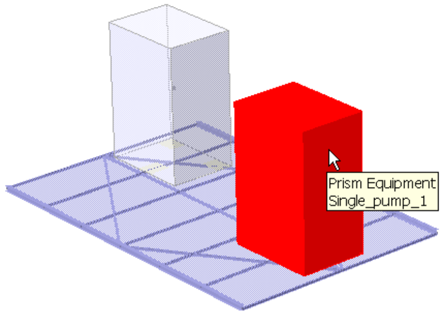
2. *Selecting the equipment to be placed.*



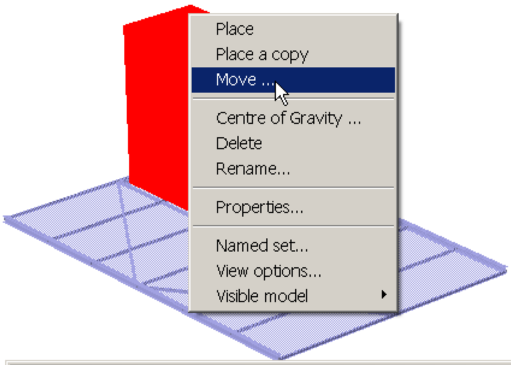
3. *When moving the mouse over the structure the mouse tooltip indicates a box symbol. The positioning of the equipment now takes place where the equipment snap point (origin of box at bottom) intersects with a beam structure snap point.*

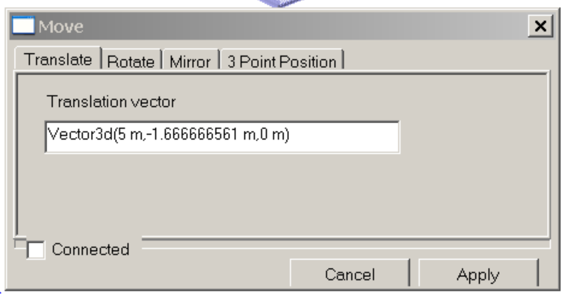


4. *The equipment is now placed onto the structure in the desired loadcase.*

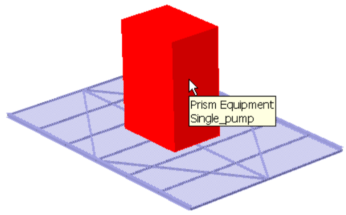


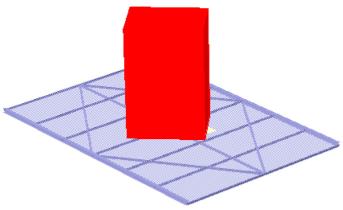
An equipment can be used only once in a loadcase. A copy of the equipment may be used in the same loadcase but at another position(s). Select the equipment, RMB and *Place a copy*. You place the copy in the same way as described above. This equipment can be renamed from Single_pump_1.

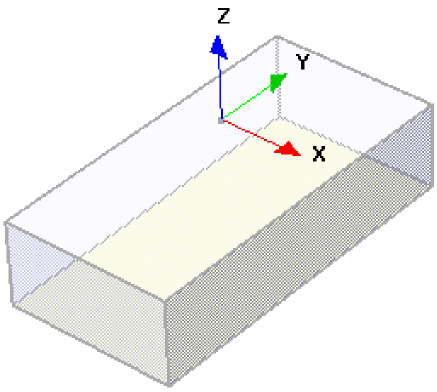




You may place the equipment more accurately by either moving it or by selecting the equipment, RMB and *Move*. The equipment may now be translated, mirrored, or rotated. In the first example it is translated, while the other shows a rotation (45 degrees).



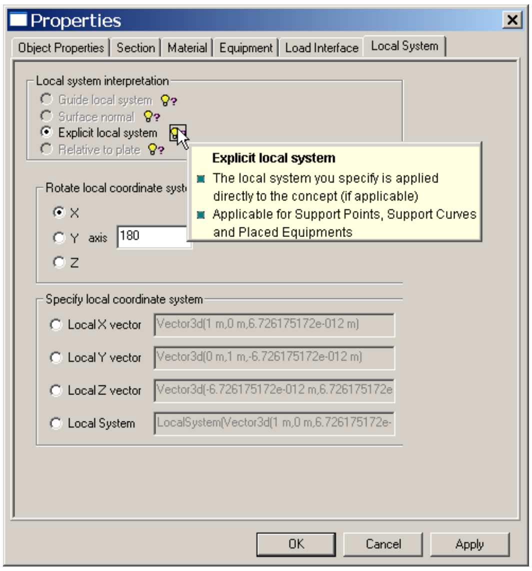


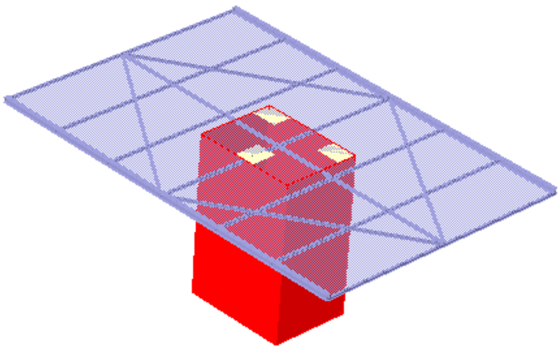


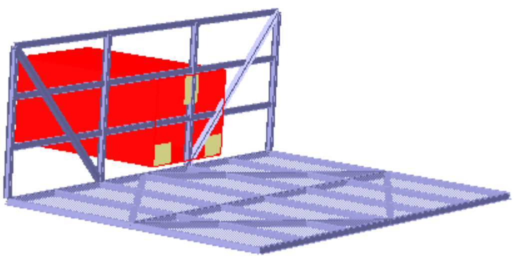
You may also verify the equipments local coordinate system by selecting the equipment, push **RMB** and choose *Labels|Local Coordinate System*.

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.







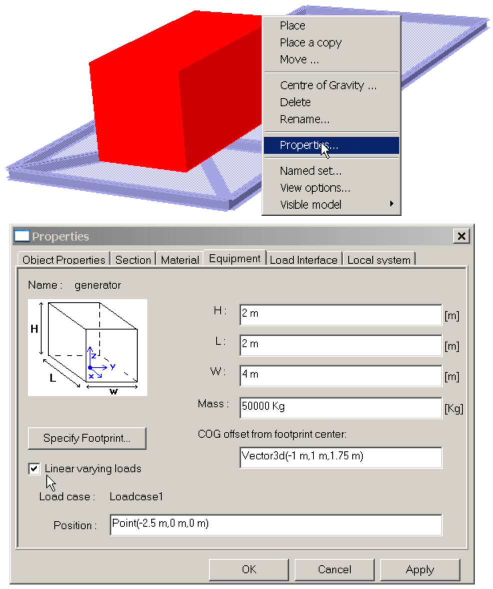
Same equipment placed along a vertical wall and rotated 90 degrees so that the footprints are aligned with the wall plane.

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.

3.9.4 Creating forces from placed equipments

GeniE allows you to operate with two different types of load calculation rules.

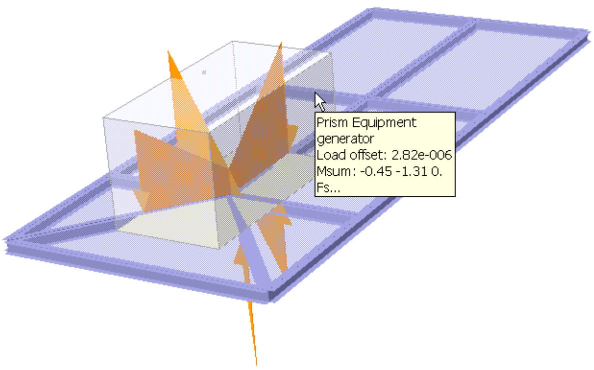
- Linearly varying loads. This method will always ensure equilibrium between the applied mass and calculated force-
- Constant loads. This method will not ensure equilibrium.

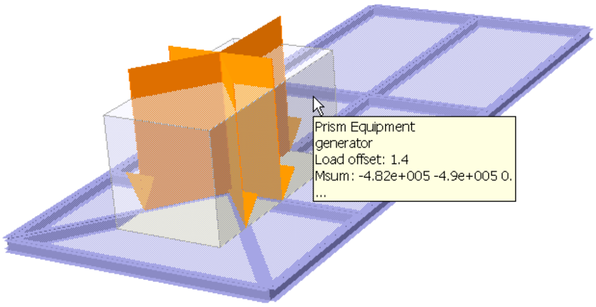


Default option is to calculate linearly varying loads. By de-activating the tick-off for linearly varying loads, the program calculates constant line loads. You may want to do this when e.g. working with uniform distributed loads (UDL).

The equipment loads are always calculated when creating a finite element mesh wither manually or as a result of running an analysis.

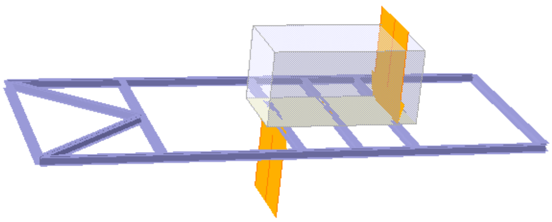
You may also force the program to calculate loads by selecting the loadcase from the browser, RMB and *Generate Equipment Loads*. You normally do this when you make up the loadcases and you want to verify the applied loads.



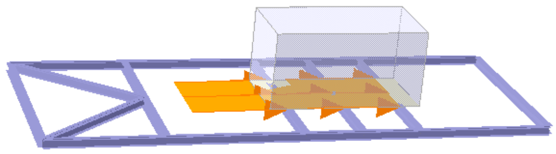


The two examples above show the difference between linear and constant – note that the mouse tooltip called *Load Offset* indicates the horizontal distance between the equipment's mass cog and the calculated force cog. I.e, it documents the error introduced by choosing constant loads. By using linearly 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.



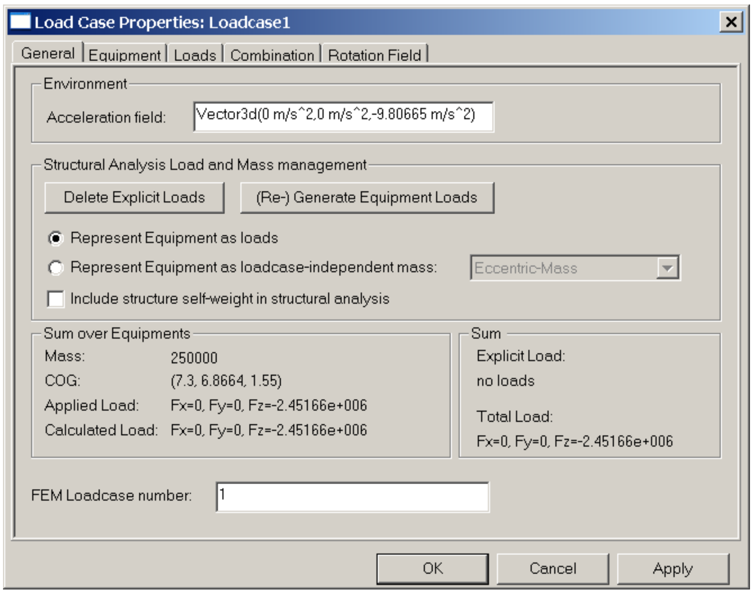
Force couple due to horizontal acceleration



Shear force couple due to horizontal acceleration

3.9.5 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.

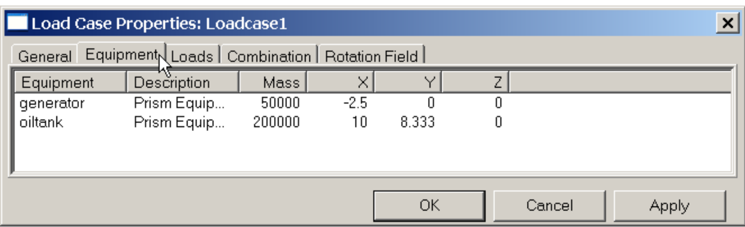


Another option is by selecting a loadcase in the browser, RMB and *Properties*.

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.

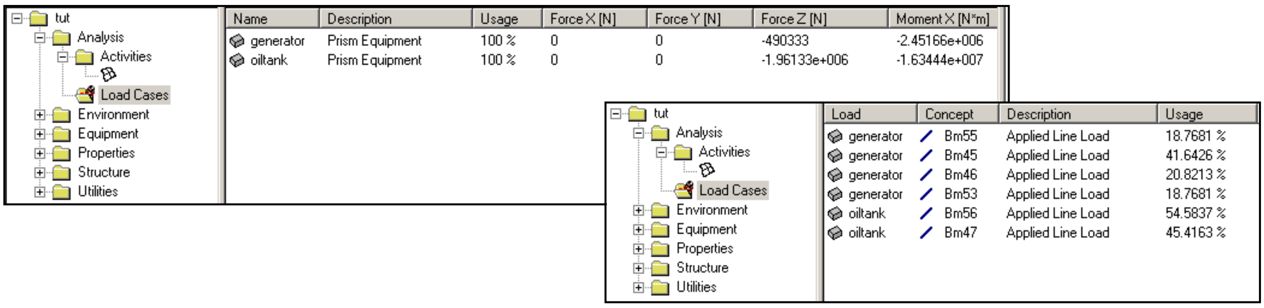
This example shows that the loadcase is loaded with a total equipment mass of 250000 and no explicit loads.

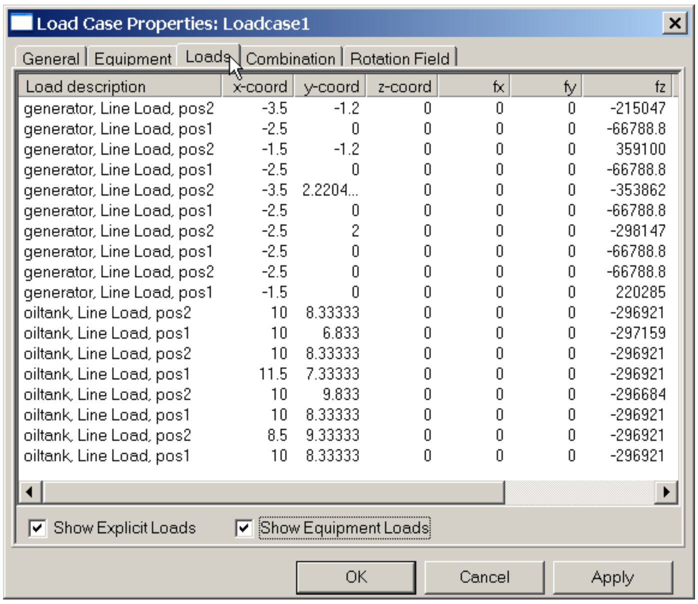


More details about the equipments and where they are placed may be found under the tab *Equipment*, or look in the equipment folder in the browser.

Two equipments (generator and oiltank) are placed in this loadcase (global co-ordinate 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.





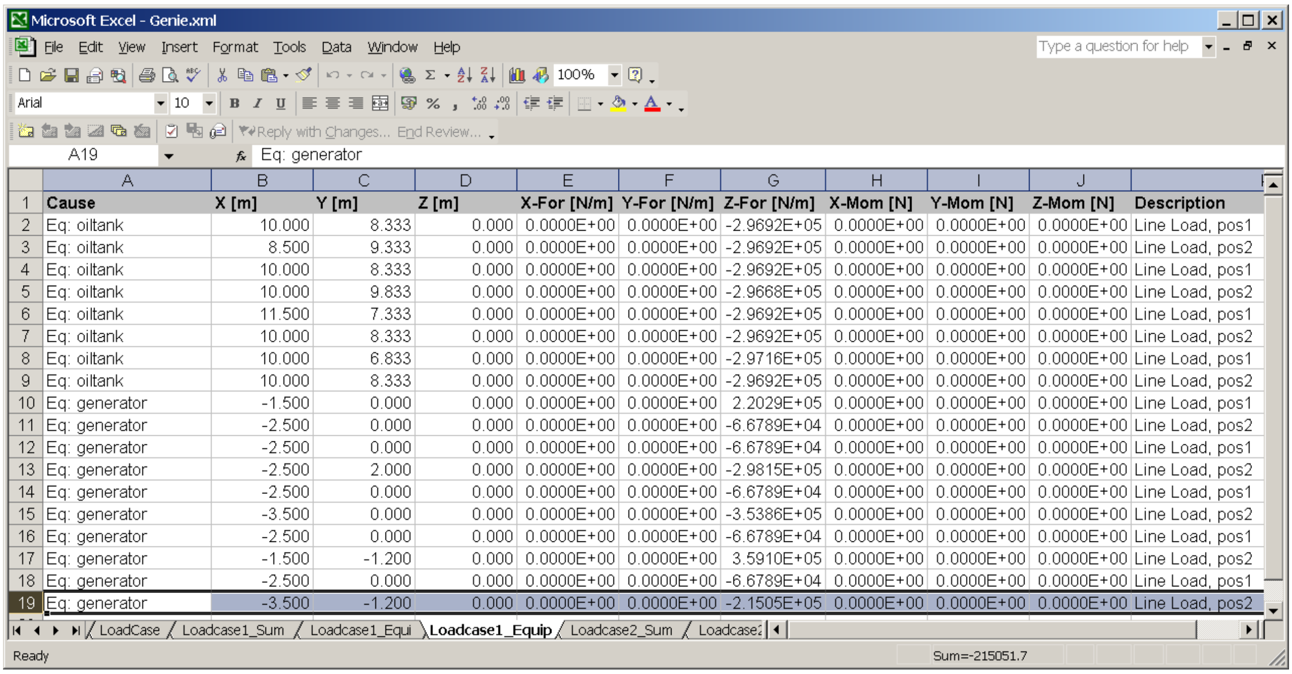
Load description	x-coord	y-coord	z-coord	fx	fy	fz
generator, Line Load, pos2	-3.5	-1.2	0	0	0	-215047
generator, Line Load, pos1	-2.5	0	0	0	0	-66788.8
generator, Line Load, pos2	-1.5	-1.2	0	0	0	359100
generator, Line Load, pos1	-2.5	0	0	0	0	-66788.8
generator, Line Load, pos2	-3.5	2.2204...	0	0	0	-353862
generator, Line Load, pos1	-2.5	0	0	0	0	-66788.8
generator, Line Load, pos2	-2.5	2	0	0	0	-298147
generator, Line Load, pos1	-2.5	0	0	0	0	-66788.8
generator, Line Load, pos2	-2.5	0	0	0	0	-66788.8
generator, Line Load, pos1	-1.5	0	0	0	0	220285
oiltank, Line Load, pos2	10	8.33333	0	0	0	-296921
oiltank, Line Load, pos1	10	6.833	0	0	0	-297159
oiltank, Line Load, pos2	10	8.33333	0	0	0	-296921
oiltank, Line Load, pos1	11.5	7.33333	0	0	0	-296921
oiltank, Line Load, pos2	10	9.833	0	0	0	-296684
oiltank, Line Load, pos1	10	8.33333	0	0	0	-296921
oiltank, Line Load, pos2	8.5	9.33333	0	0	0	-296921
oiltank, Line Load, pos1	10	8.33333	0	0	0	-296921

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



	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

3.10 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.

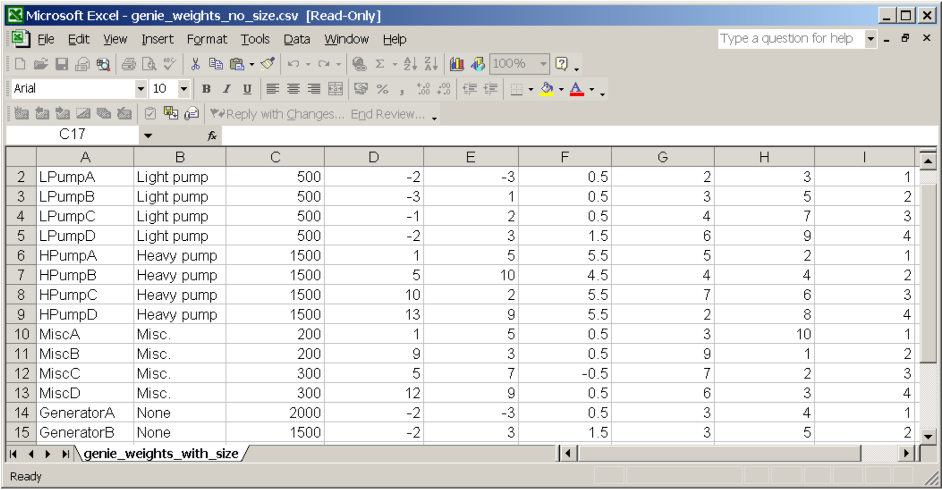
The process of importing weight list data is semi-automatic. The first step is to import the weight list data, then to create equipments, and finally to place them. The weight list data normally contain information about the size of each weight item – if so equipments may be automatically created. If not, you need to add dimension data to the weight items before equipments may be generated. Both options are described in the following.

3.10.1 Format of weight list data

GeniE can import data from a weight list using two different formats – either XML (E**X**tendable **M**arkup **L**anguage) or CSV (**C**omma **S**eparated **V**alues). 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.

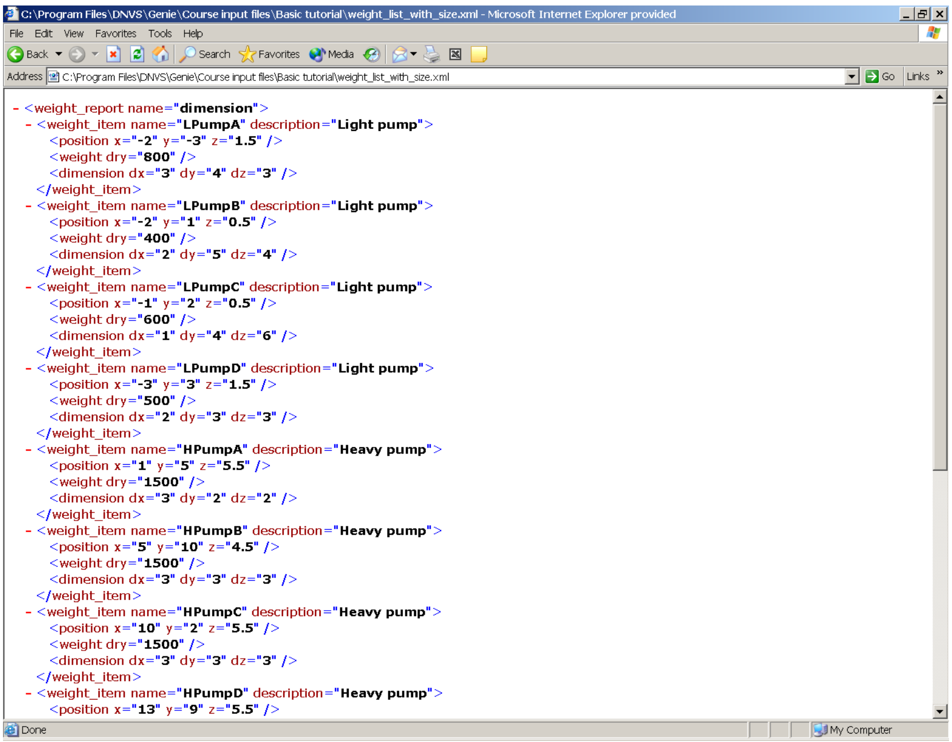
- weight_item.name : a unique name of the item (mandatory)
- weight_item description : additional description (optional)
- weight_item.weight.dry : the mass of the item (mandatory)
- weight_item.position.x/y/z : position of the item in global axis system (mandatory)
- weight_item.dimension.dx/dy/dz : 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).



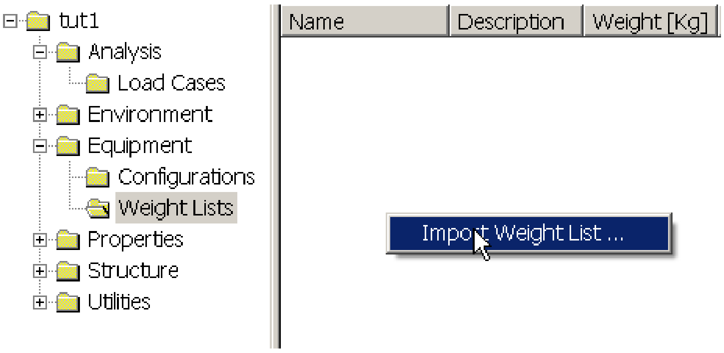

```
- <weight_report name="dimension">
  - <weight_item name="LPumpA" description="Light pump">
      <position x="-2" y="-3" z="1.5" />
      <weight dry="800" />
      <dimension dx="3" dy="4" dz="3" />
    </weight_item>
  - <weight_item name="LPumpB" description="Light pump">
      <position x="-2" 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" />
```

The same weight list data given on XML format.

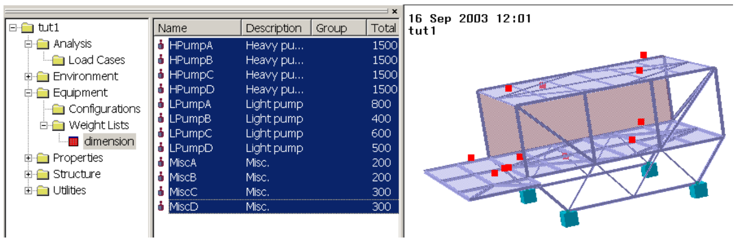
Visualised in Internet Explorer.

3.10.2 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.



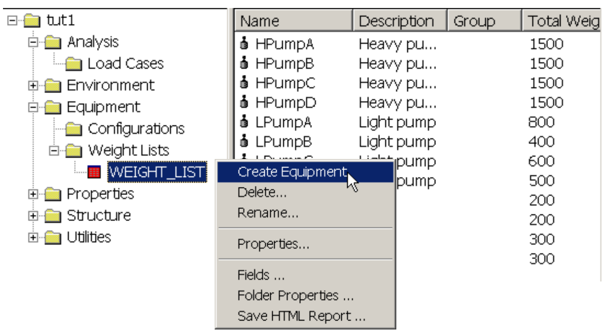
You may now import the format in either XML or CSV format



When expanding the browser and selecting the imported data, you may visualise where cog for each weight list item is located relative to the structural model. Note that none equipments have been generated so far.

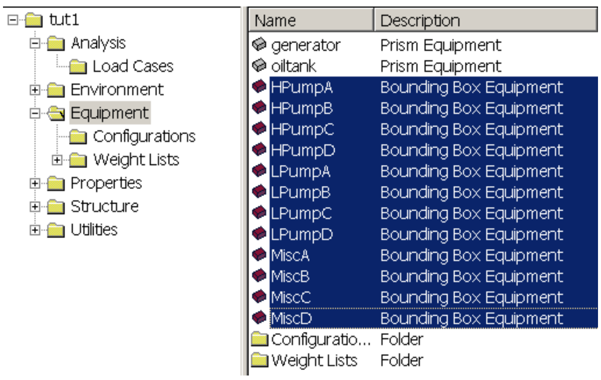
3.10.3 Creating and placing equipments from weight lists

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.



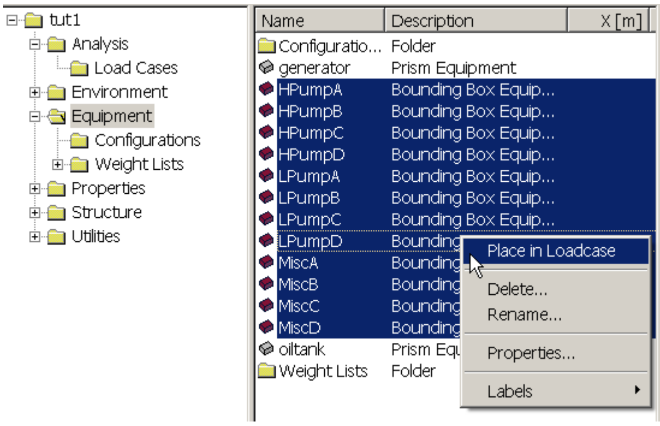
Select the WEIGHT_LIST, RMB and *Create Equipment*.

This command will create the Bounding Box Equipments and at the same time switch browser view to Equipments as shown below.

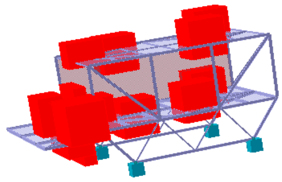


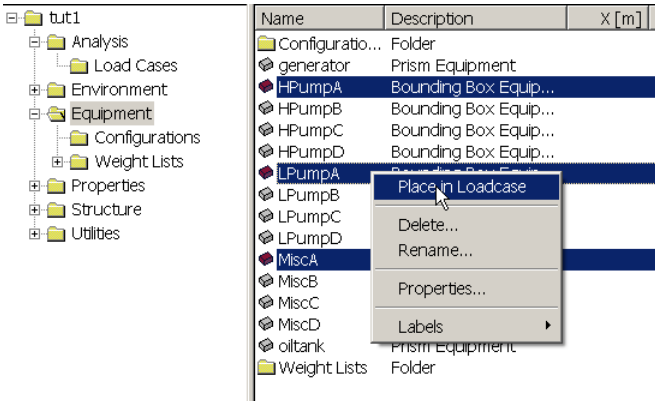
A total of 12 new equipment objects are created in addition to two existing prism equipments.

These equipments may now be placed (all or individually) in several different loadcases.

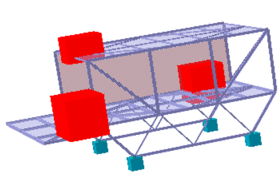


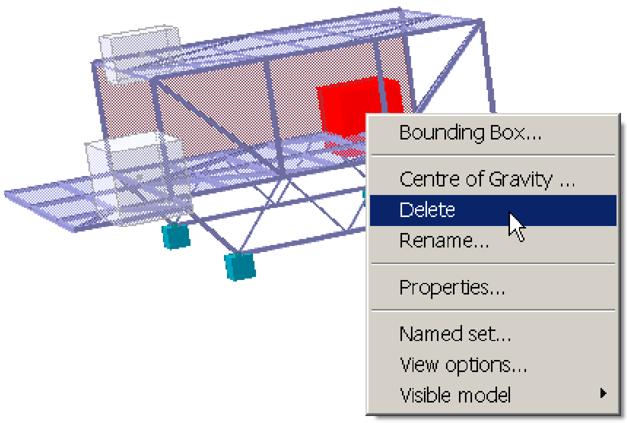
The equipments may now be placed by selecting them, RMB and *Place in Loadcase*. The position is known from before, this means you can not change position of a bounding box object without changing the weight list data file.





In this example three weight list objects are selected, RMB and *Place in Loadcase*.





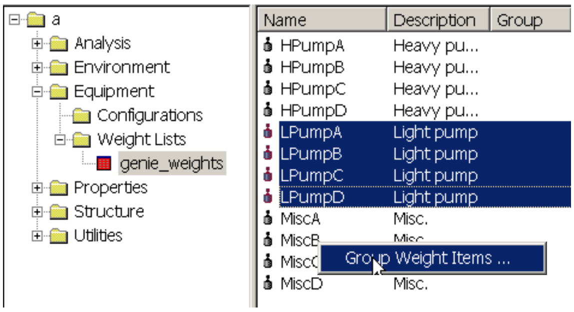
You may also remove a Bounding Box Equipment from the loadcase. By doing this, you do not delete the equipment as such. If the equipment is used in other loadcases it is still loaded for the loadcase in question.

This also applies to regular Prism Equipment.

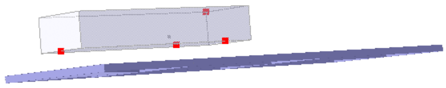
If you delete equipments from the Equipment browser you also remove the equipment from all loaded loadcases.

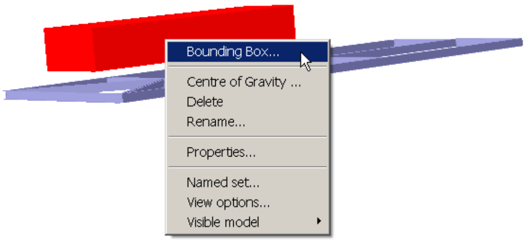
If the weight list does not contain dimensional information for the weight items it is necessary to create the Bounding Box Equipments manually. This is done by grouping weight items together and let the minimum box they fit into define the dimensions. The example below shows that 4 weight items without 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).

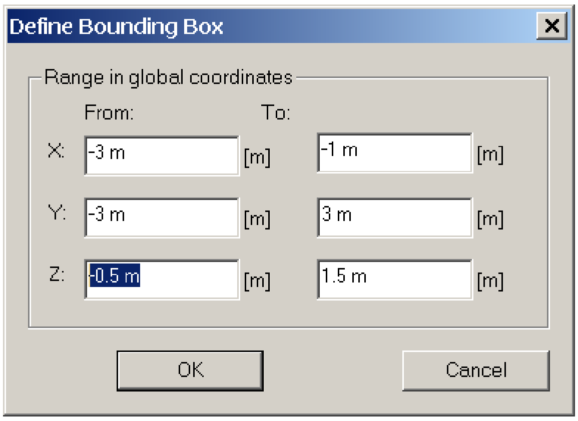


Selecting the weight list objects, RMB and *Group Weight Items*. The group is given the name LP. The Bounding Box Equipment is placed on structure below. As can be seen the equipment does not intersect the structure and hence no loads are calculated.





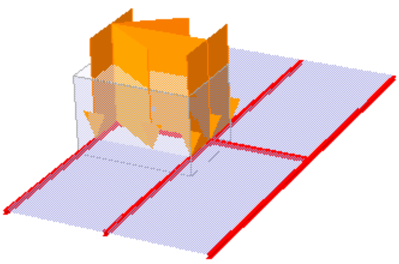
To ensure that the bounding box equipment intersects structure it is possible to change the size of the manually created bounding box object. Select the equipment, RMB and Bounding Box where the values can be adjusted. This will not change the cog which is important when creating a mass model or the 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

3.10.4 Calculating the loads

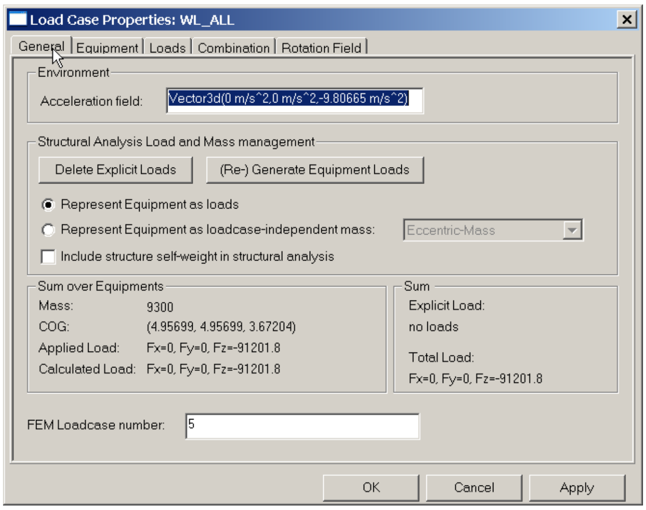
They are calculated as before by setting the actual loadcase as current and then *Generate Equipment Loads*. The major difference from Prism Equipments is that only constant loads are created, also if subjected to horizontal accelerations (i.e. no force couples are established). The constant line load is calculated from total equipment force divided by total beam length intersecting the bounding box equipment.

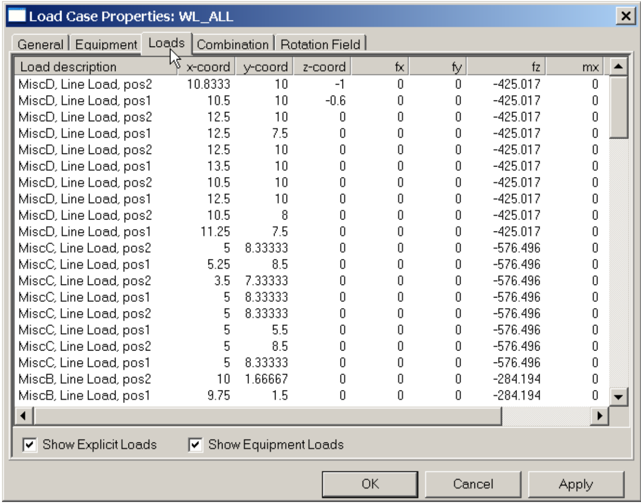


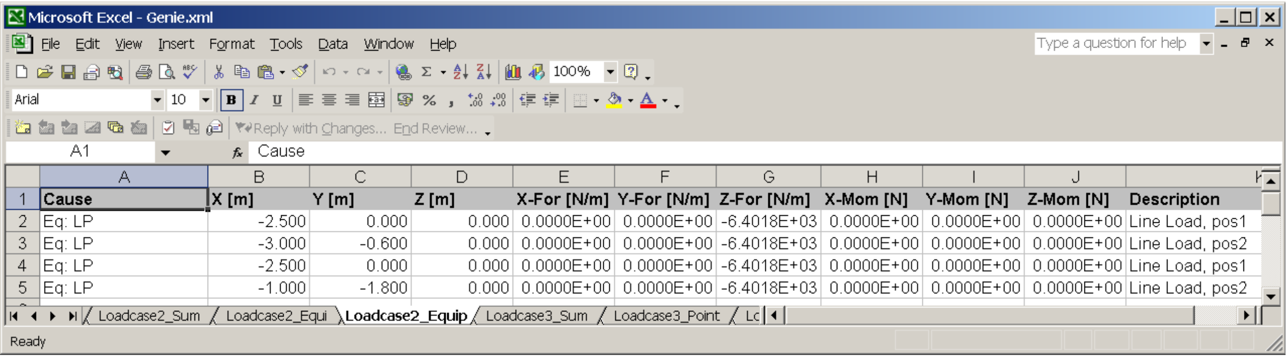
Constant loads applied to all beams intersecting the bounding box equipment. You may, however, direct the load transfer to beams by using Load Interface properties.

3.10.5 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).







3.11 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.

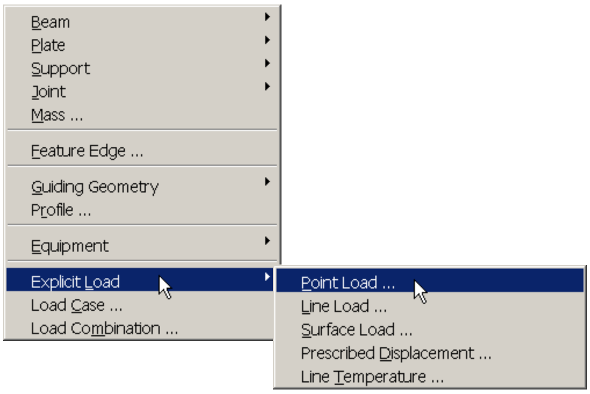
An explicit load does not need to be aligned with a snap point (or a structural joint) in your model. As long as there is intersection between the load and the structure GeniE will compute the necessary details to ensure a correct model.

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.

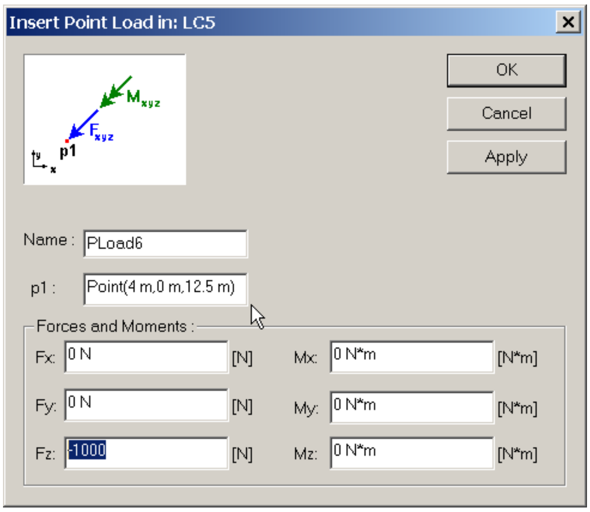
The applied explicit loads may be verified graphically, from the loadcase property sheet or from a saved report. In addition you can do it from the browser as explained in Chapter 3.8.1.

3.11.1 Point loads

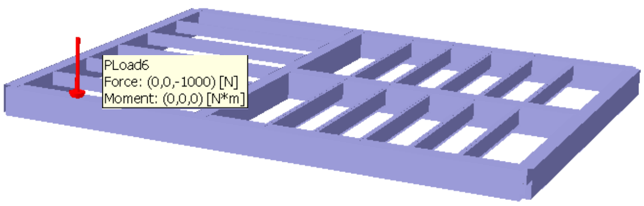
A point load on a beam member is applied by using the command ***Insert|Explicit Load|Point Load***.

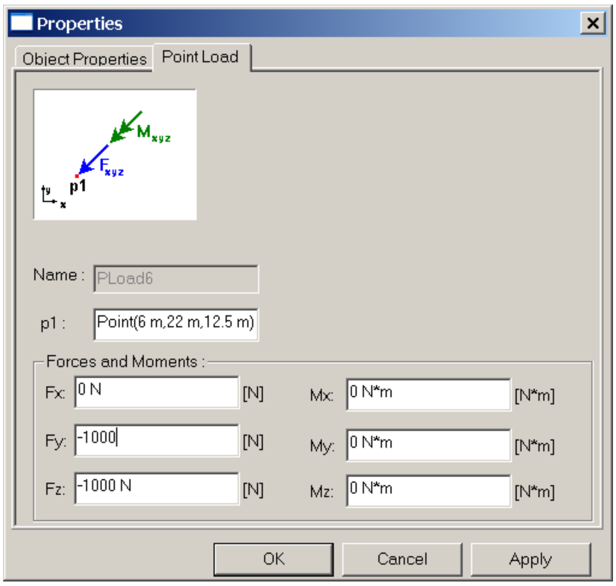


All explicit loads are defined from the pulldown menu ***Insert|Explicit Load***.

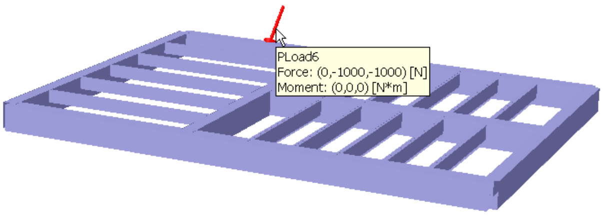


Inserting a point load in the middle of a beam where no snap point exists. Note that when moments are included in the point load it must be inserted at a topology point, e.g. at a beam end.

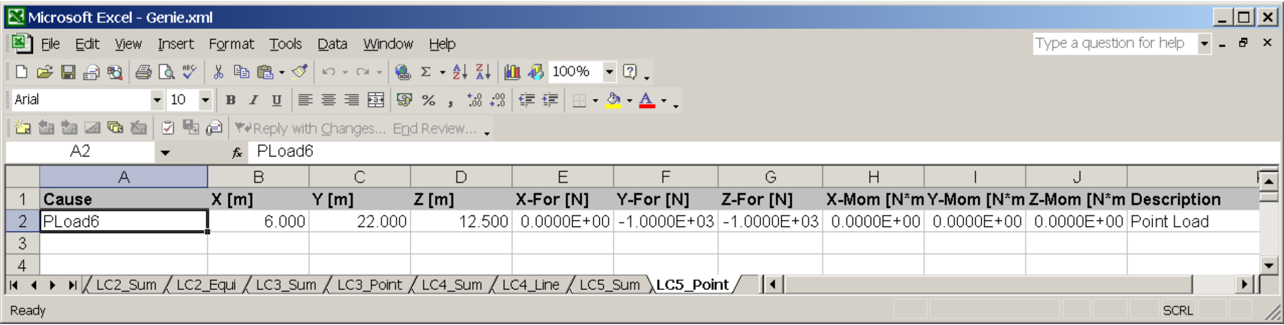




The point load may be changed by selecting the point load, RMB, and Properties. In this case both the location and intensity have been changed.

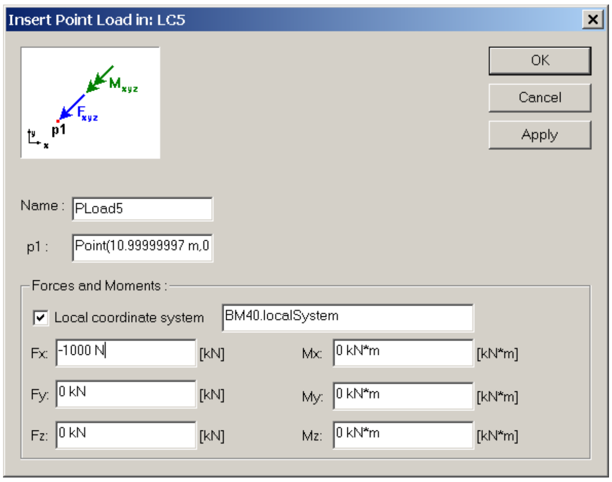


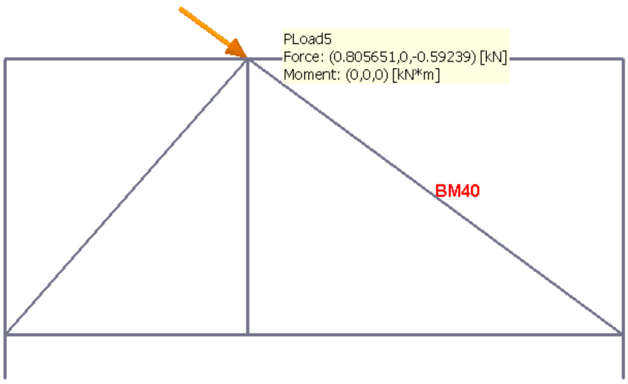
The load details of the saved report are shown below.



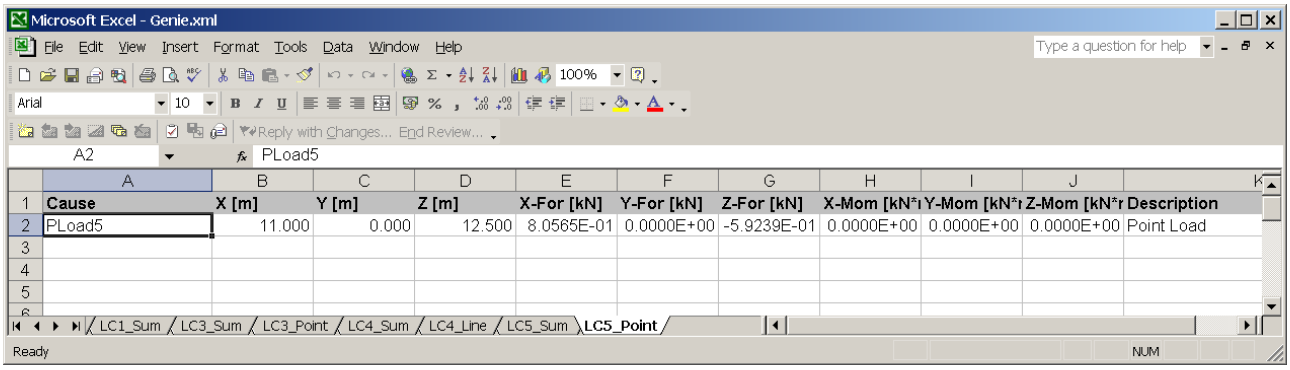
Cause	X [m]	Y [m]	Z [m]	X-For [N]	Y-For [N]	Z-For [N]	X-Mom [N*m	Y-Mom [N*m	Z-Mom [N*m	Description
PLoad6	6.000	22.000	12.500	0.0000E+00	-1.0000E+03	-1.0000E+03	0.0000E+00	0.0000E+00	0.0000E+00	Point Load

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.

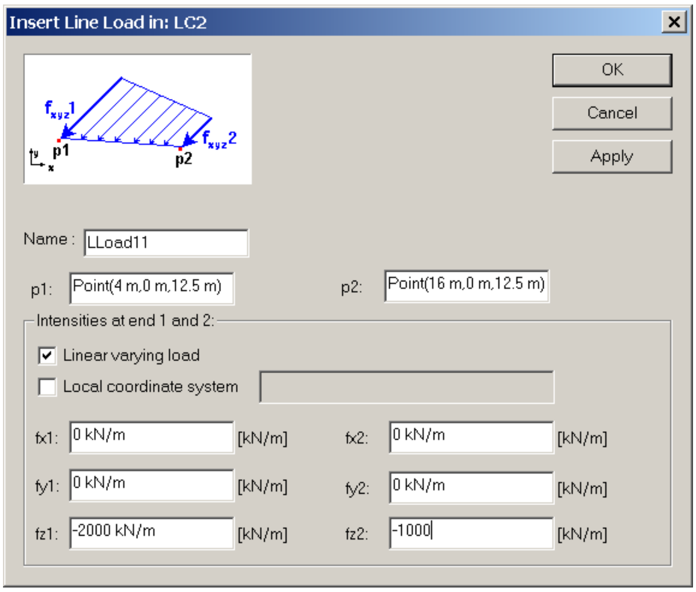


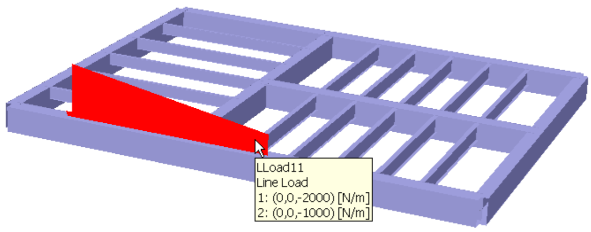
	A	B	C	D	E	F	G	H	I	J
1	Cause	X [m]	Y [m]	Z [m]	X-For [kN]	Y-For [kN]	Z-For [kN]	X-Mom [kN*ı	Y-Mom [kN*ı	Z-Mom [kN*r	Description
2	PLoad5	11.000	0.000	12.500	8.0565E-01	0.0000E+00	-5.9239E-01	0.0000E+00	0.0000E+00	0.0000E+00	Point Load

3.11.2 Line loads

The command ***Insert|Explicit Load|Line Load*** lets you insert a constant or varying line load along beams.

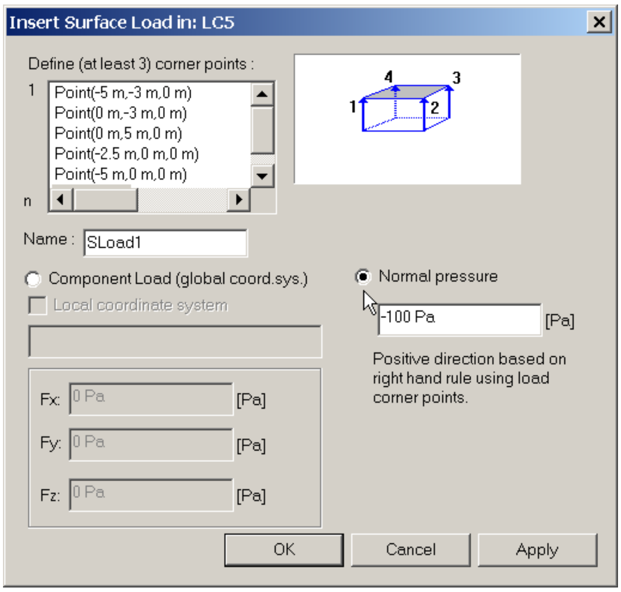
A linearly varying line load has been inserted with start and end point explicitly defined (no snap points exist at these locations, so the input field has been manually edited).





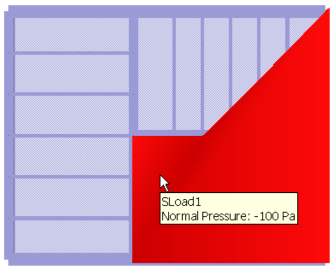
3.11.3 Pressure loads

Constant pressure loads are applied to flat plates using the command ***Insert|Explicit Load|Surface Load***.

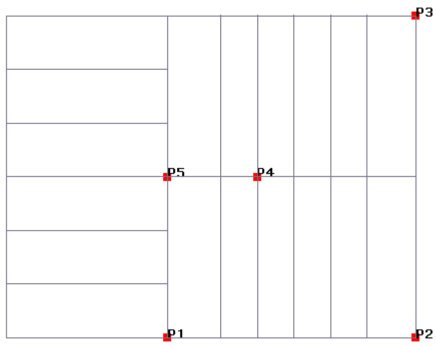


A normal pressure has been applied to parts of the plate. Make sure that the final point inserted forms a closed circuit with the first point.

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.

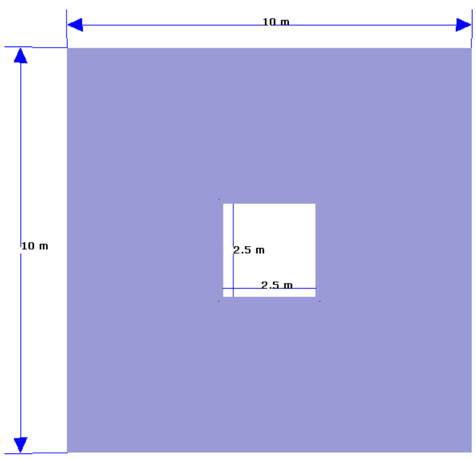


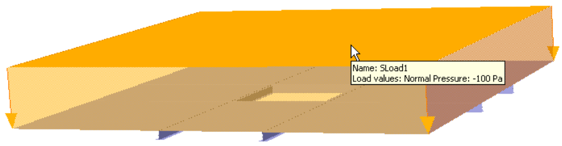
In the example above a total of five positions are used to specify the pressure area. You use the values (by clicking or manual input) for points P1-P5, i.e. you do not close the loop by entering data for P1 following P5.



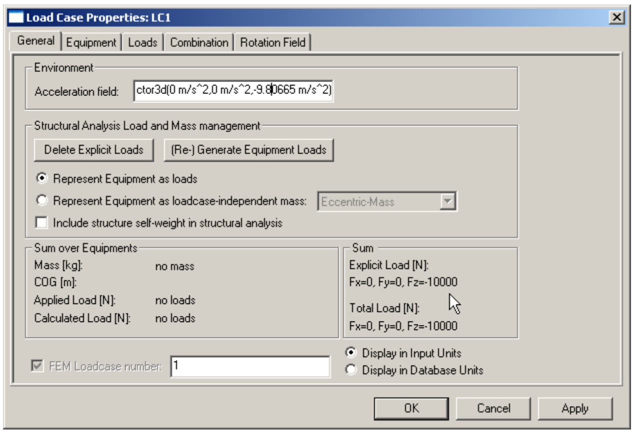
In case you have a plate with holes and you apply a surface load over the entire plate, GeniE will use the entire plate area when reporting the surface total surface load. You find the reported value either when selecting the loadcase and choose *Properties* from the context sensitive menu or from a 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 whole, the surface area is 93.75 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.

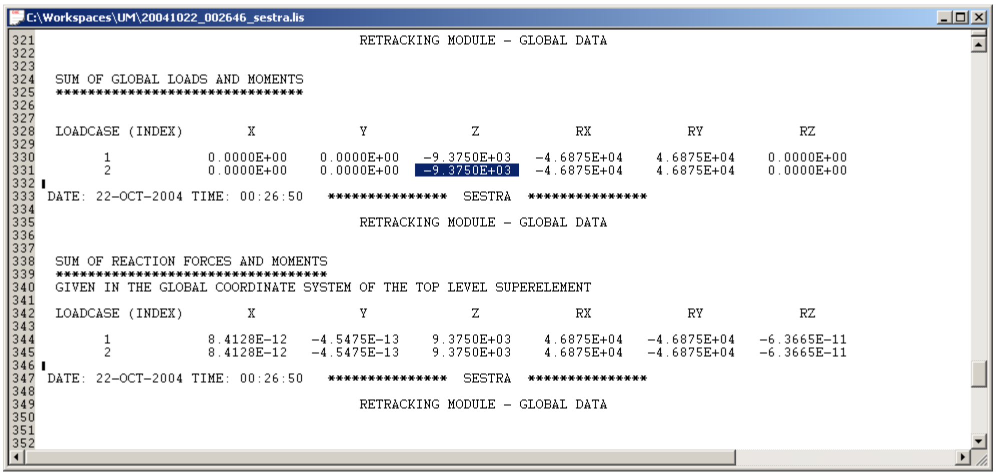




A uniform pressure of -100Pa applied over the whole area.



Genie reports -10000 N based on a simplified calculation.



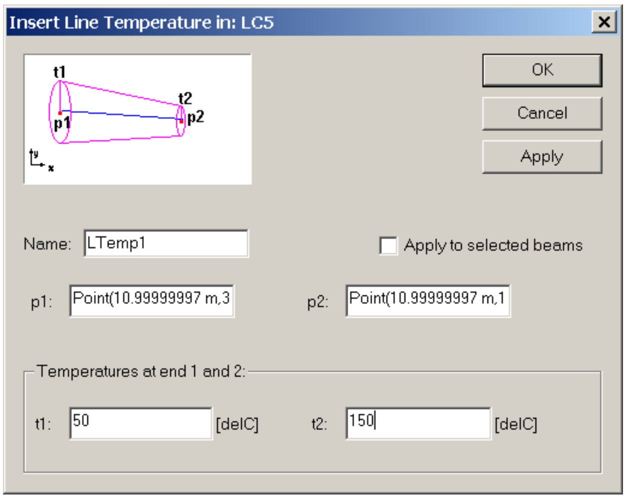
Sestra has computed the correct values, hence results like displacements and stresses are correct.

See chapter 3.13 for further details on how to run analysis.

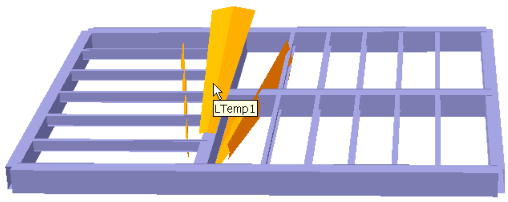
In case you have a pressure load defined by an area that differs from the topology of a plate, the correctness of the loads depend on the mesh density, type of mesh (first or second order elements). See the next Chapter on how to refine and control the finite element mesh.

3.11.4 Temperature load

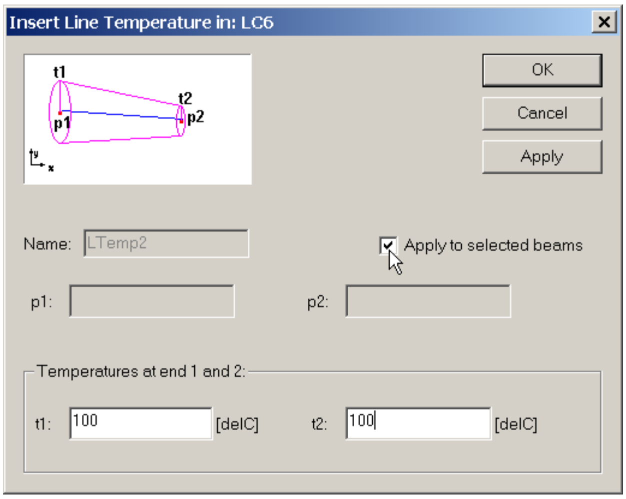
Temperature loads may be applied to beams only. The temperature loads are constant over the beam cross section, but they may vary in intensity along a beam. Temperature may be inserted to one or several beams at the same time. The command for doing so is ***Insert|Explicit Load|Line Temperature***.



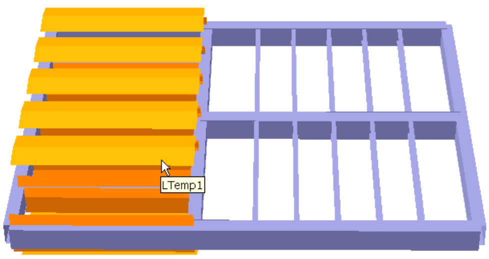
A varying temperature load is inserted along one beam. The temperature and positioning may be changed (or deleted) by selecting the load object, RMB and *Properties*.



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.



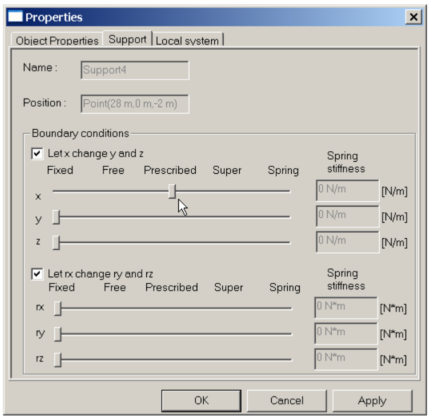
Constant temperature load applied to the beams selected prior to load definition.

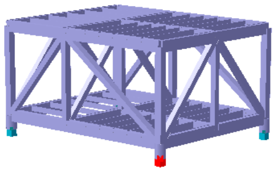


3.11.5 Prescribed displacements

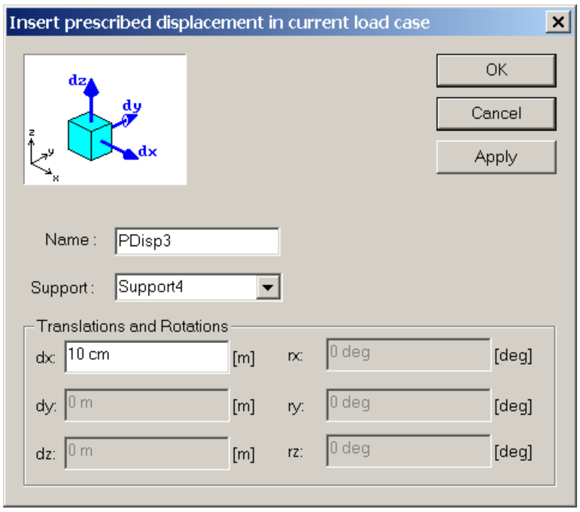
A prescribed displacement is a boundary condition combined with the actual displacement or rotation per loadcase. A support point needs to be inserted first before the actual displacement or rotation can be defined, see the next Chapter on how to define boundary conditions.

This means that a structure may have several prescribed displacements to the same support point, but in different loadcases.





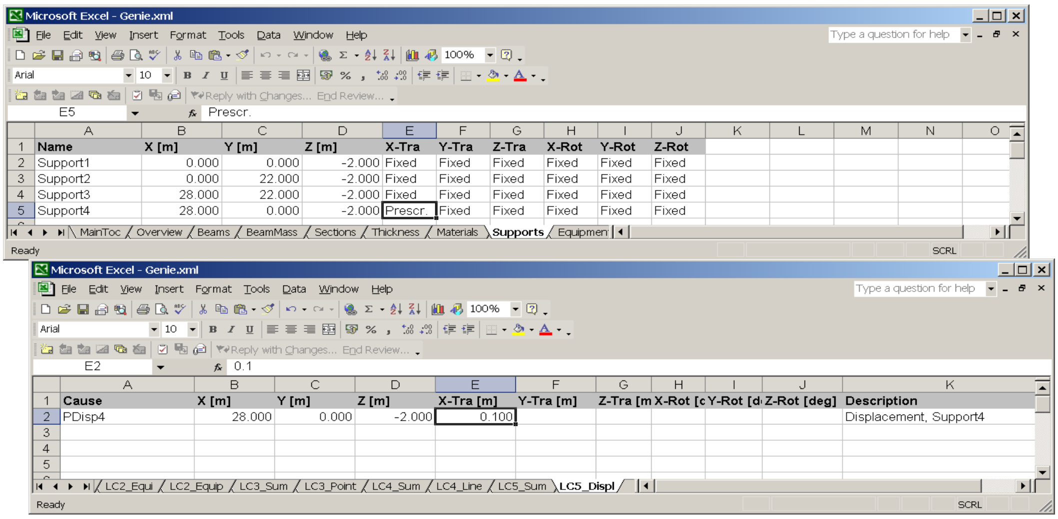
Prescribed displacement in x-direction applied to support point *Support4*.



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.



3.11.6 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)

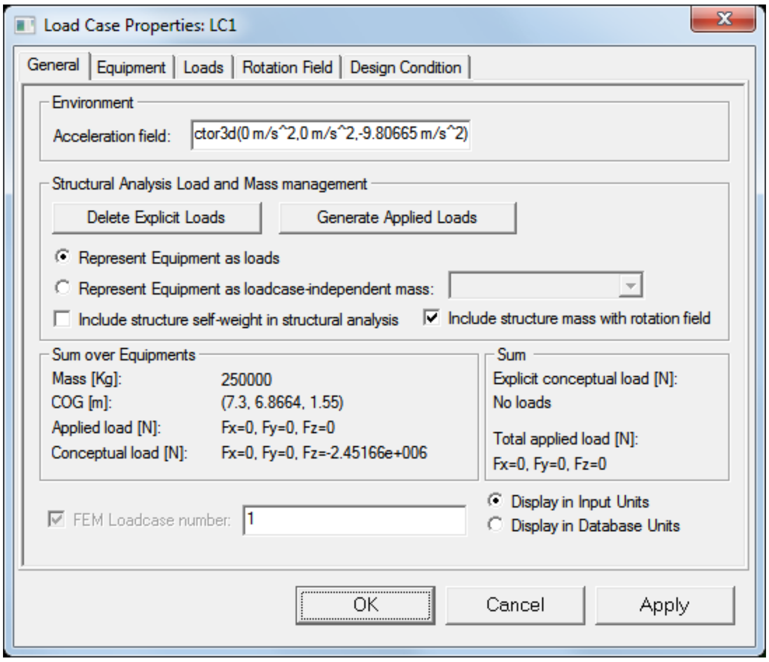
There are two ways of specifying acceleration loads, either by using a constant acceleration field or a rotational field, please note that the two methods are different (se below for explanations).

For both of them they are activated from the Loadcase Property Sheet and they may be different from loadcase to loadcase.

3.11.6.1 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.

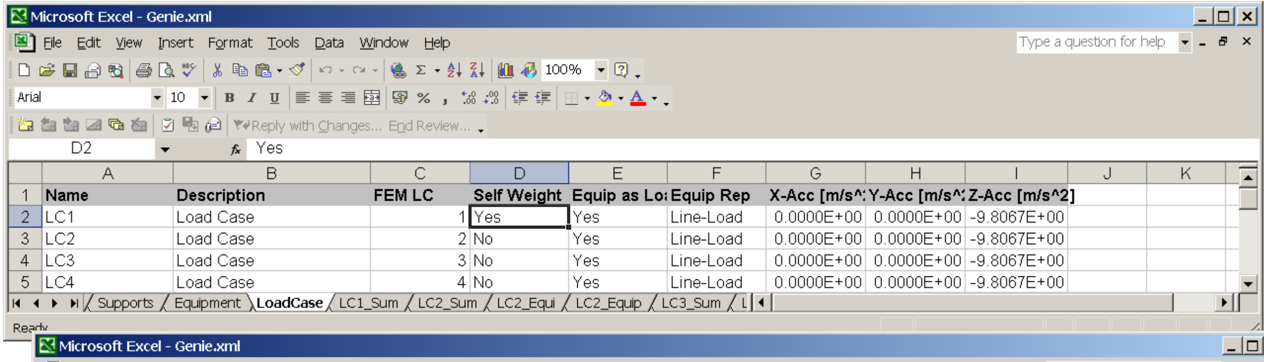
Include structure mass with rotation field

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.

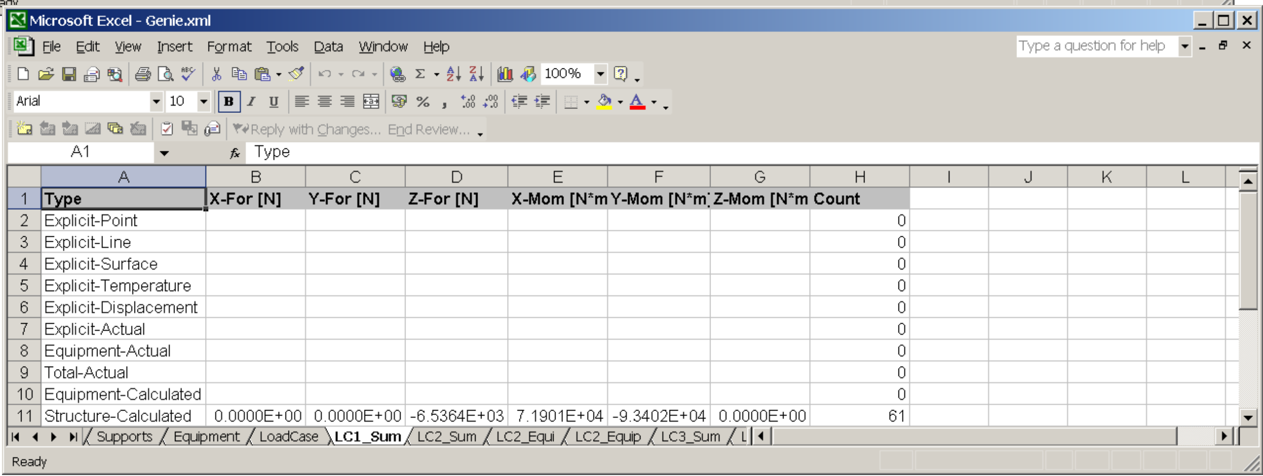
Gravity is a special case where the vertical acceleration is g = -9.81 m/s^2.

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.



Name	Description	FEM LC	Self Weight	Equip as Lo	Equip Rep	X-Acc [m/s^	Y-Acc [m/s^	Z-Acc [m/s^2]
LC1	Load Case	1	Yes	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00
LC2	Load Case	2	No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00
LC3	Load Case	3	No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00
LC4	Load Case	4	No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00



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

3.11.6.2 Rotation field

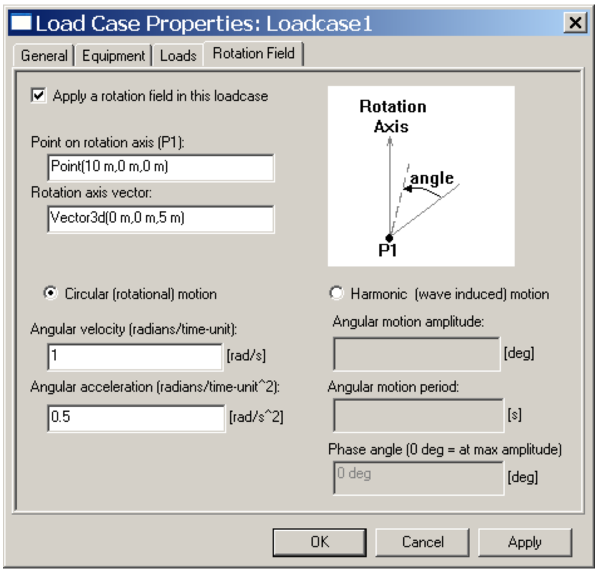
This is specified on the "Rotation Field" tab. A rotation field is transferred to the FEM model as applied nodal accellerations (BNACCLO). The sign of the applied accelerations is such that the resulting loads correspond to inertia forces.

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.

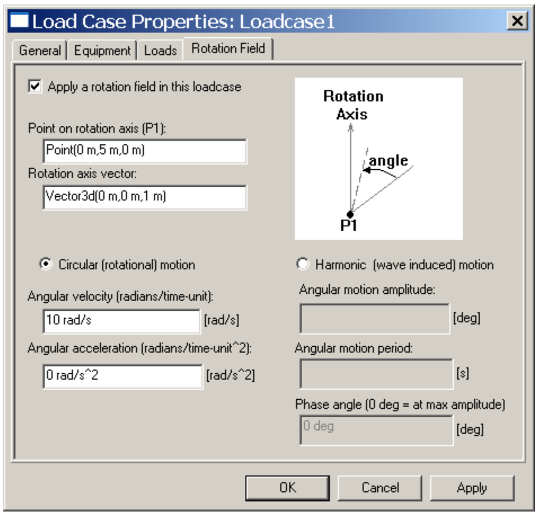
The direction of rotation is according to the right hand rule with respect to the direction of the rotation axis vector.

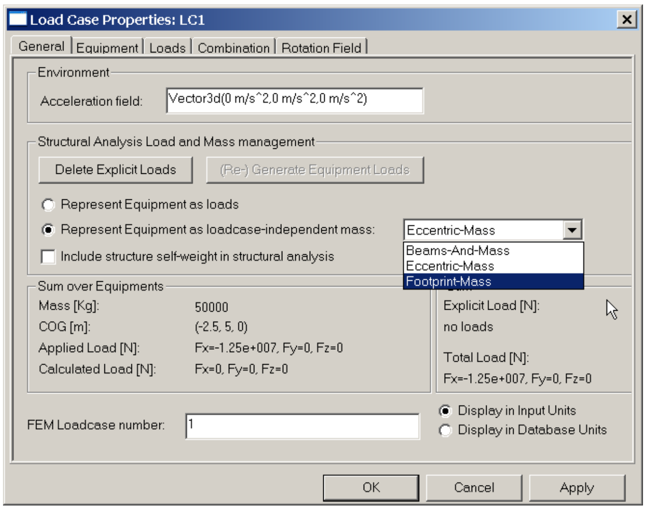
To verify and change you can use the loadcase property sheet.

When performing the analysis there are two alternatives:

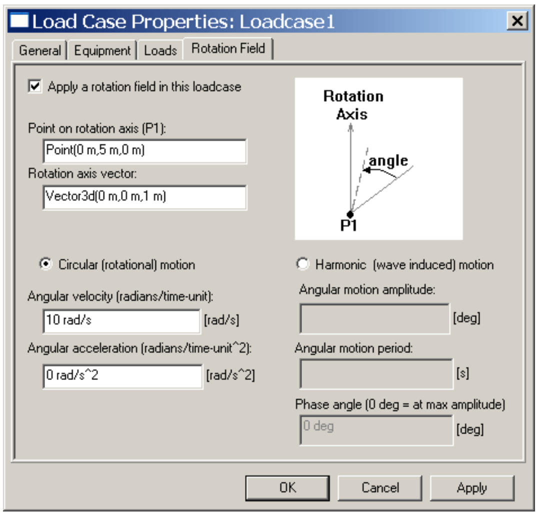
- The resulting inertia loads are computed by the analysis program Sestra based on the node accelerations calculated by GeniE when generating the finite element model.
- GeniE will compute the equipment loads based on an equivalent acceleration in the equipment’s centre of gravity and using the same 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.

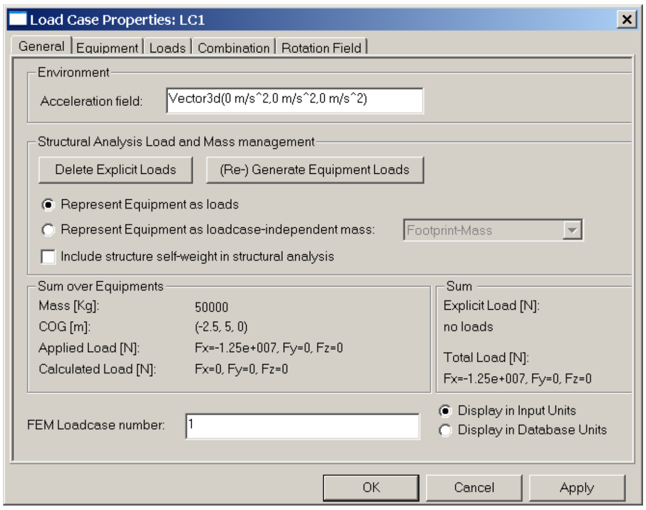
The pictures below illustrate the differences between these options.





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 3.13.2 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.

3.11.6.3 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
- Θ is the maximum roll (or pitch) angle in degrees

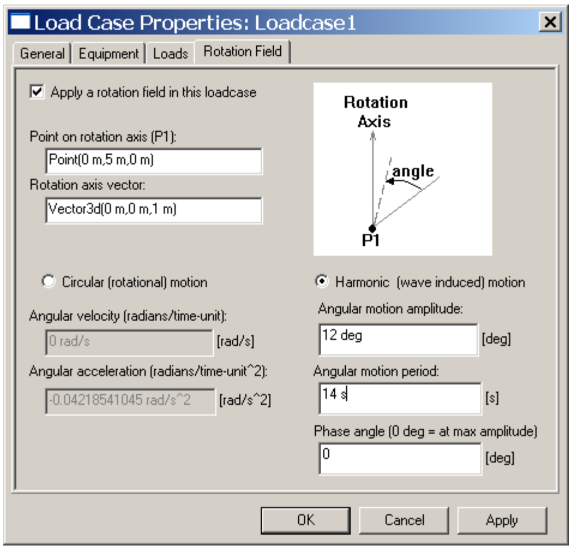
The variation of direction of the gravity vector is also accounted for, and will be:

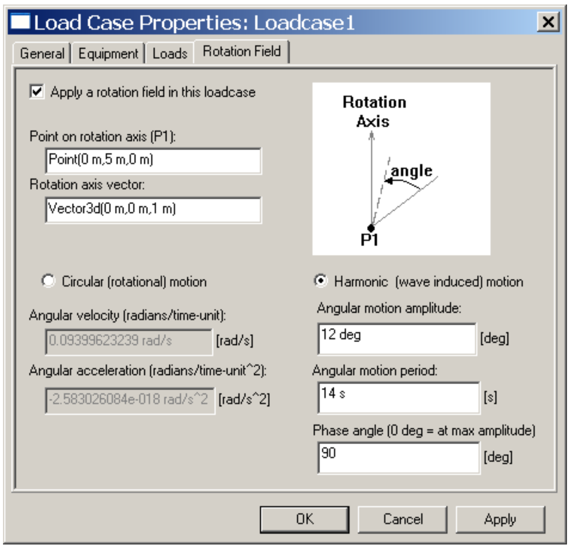
$G_y = - G \times \sin(\Theta)$ and $G_z = -G \times \cos(\Theta)$ for pure roll motion

$G_x = G \times \sin(\Theta)$ and $G_z = -G \times \cos(\Theta)$ for pure pitch motion

where

- G is the acceleration of gravity.
- Θ 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 it's 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.

3.12 Making a Finite Element Model

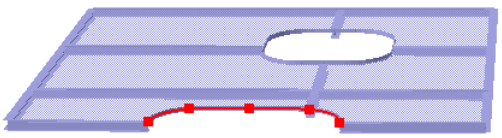
Three types of finite element models may be created in GeniE – either for use in a structural analysis, as a mass model for dynamic analysis (including hydrodynamic analysis followed by a structural analysis), or for the purpose of defining a mass model intended for a hydrodynamic analysis only.

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.

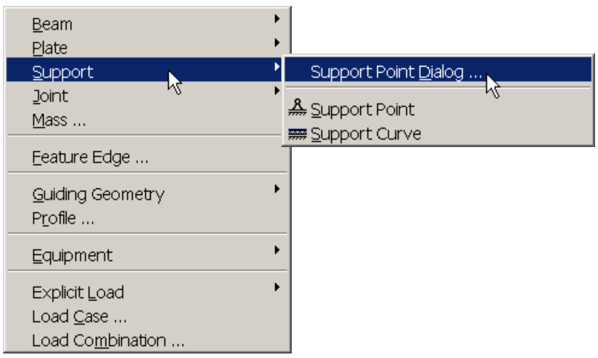
3.12.1 Boundary conditions

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).

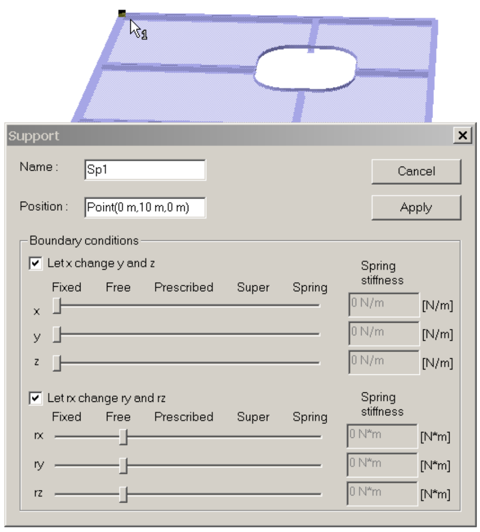


To illustrate the various boundary conditions and how they are applied, a common model (at left) will be used. It has one hole and a cut out along one of the plate edges. There is one model curve (highlighted in red).

To insert a boundary condition, the command ***Insert|Support*** may be used.

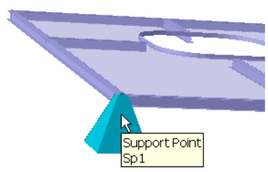


By using the support dialog option, you specify the boundary conditions before they are applied. Otherwise, the default settings are used (fixed in all dof) and you need to change afterwards.

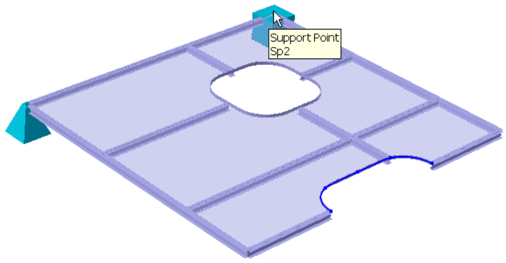


In this case a support point is inserted at the top left corner (all translation dof fixed and all rotational dof free).

When applied the support point looks like a pyramid to reflect that it is free to rotate.

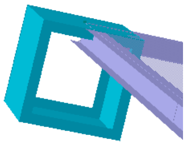


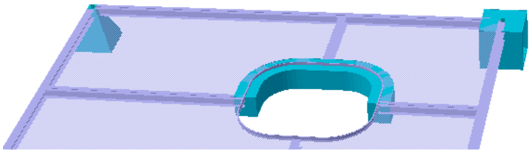
The boundary conditions are changed by selecting the support point, RMB and *Properties*.



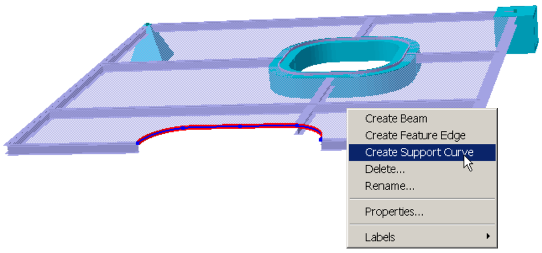
When using the command ***Insert|Support|Support Point*** (or from the toolbar) snap to point is being used and default boundary conditions are applied.

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.

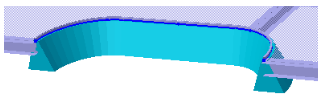




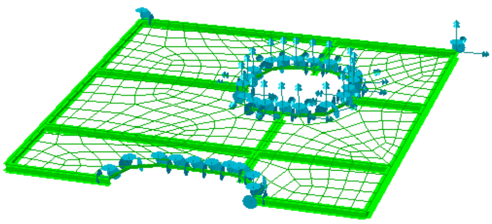
To insert boundary conditions along an edge the command ***Insert|Support|Support Curve*** (or from the toolbar) can be used. The command will prompt you for at least 3 snap points, in this case the first is at the left, second in middle, third and fourth at the right side (the fourth to complete the 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.

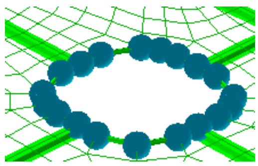


Note that when creating a finite element mesh, all relevant nodes will automatically receive boundary conditions. Typically, one finite element node for support points, and several nodes along a support curve.

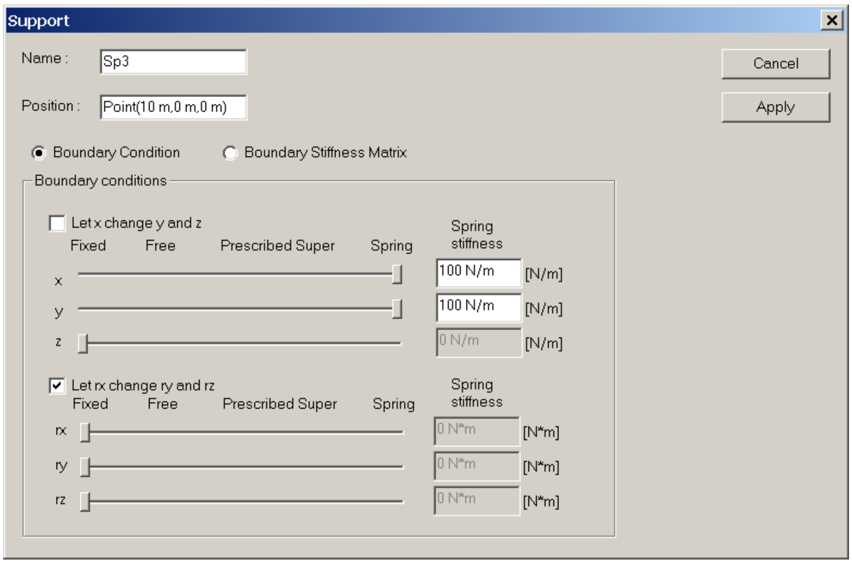


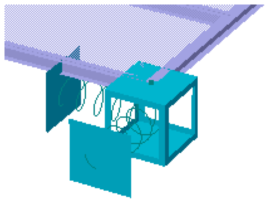
Boundary conditions automatically created in the finite element model based on the support points or curves.

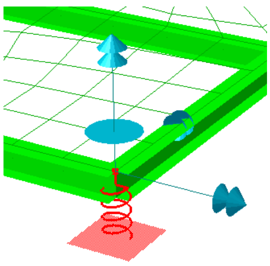
The boundary condition symbol changes depending on which dof is fixed, free, spring or supernode. A supernode is shown as a ball. Changing all dof to supernode along the inner hole leads to:



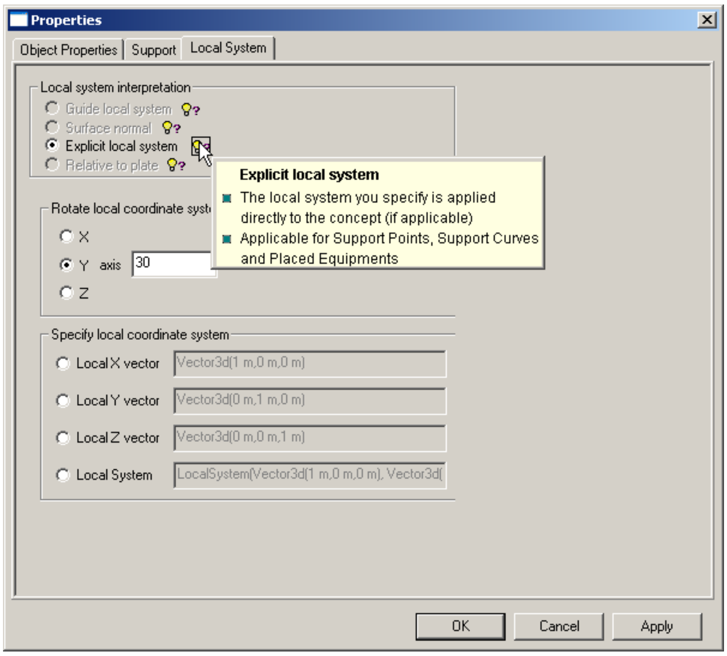
The example below shows a support point is inserted at one of the plate corners with a spring in global x and y directions. Graphical views of the support concept and corresponding finite element are shown.



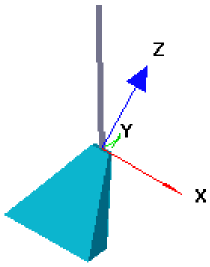




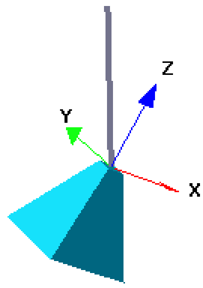
You may also apply a local coordinate system to the boundary conditions. Select a support point and force the context sensitive menu by RMB. When choosing *Properties* you have access to the Tab *Local System*, see below.



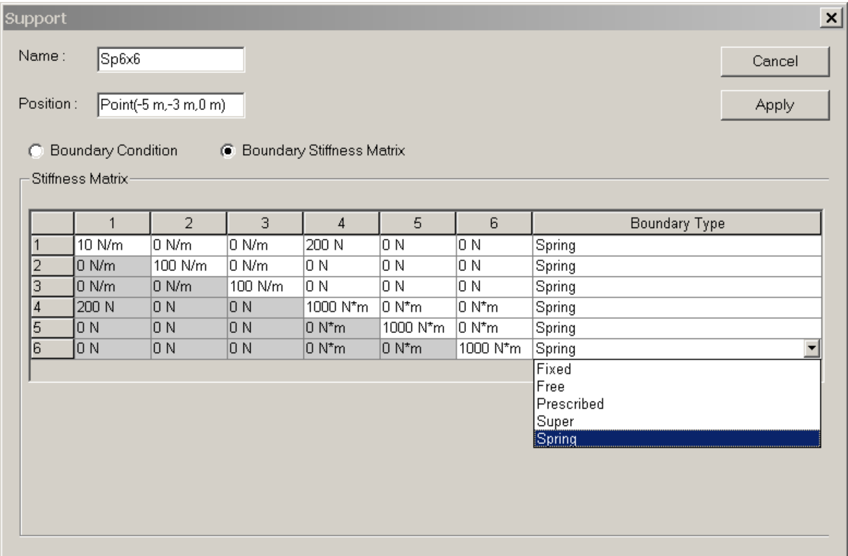
The support point is rotated 30 degrees around local y-axis, see figure below.



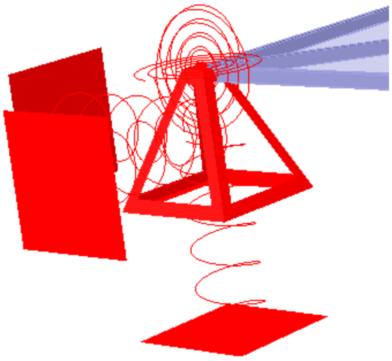
Please note that when rotating you do it relative to the local coordinate system of the support point. The picture below shows an additional rotation 45 degrees around local z-axis.



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.

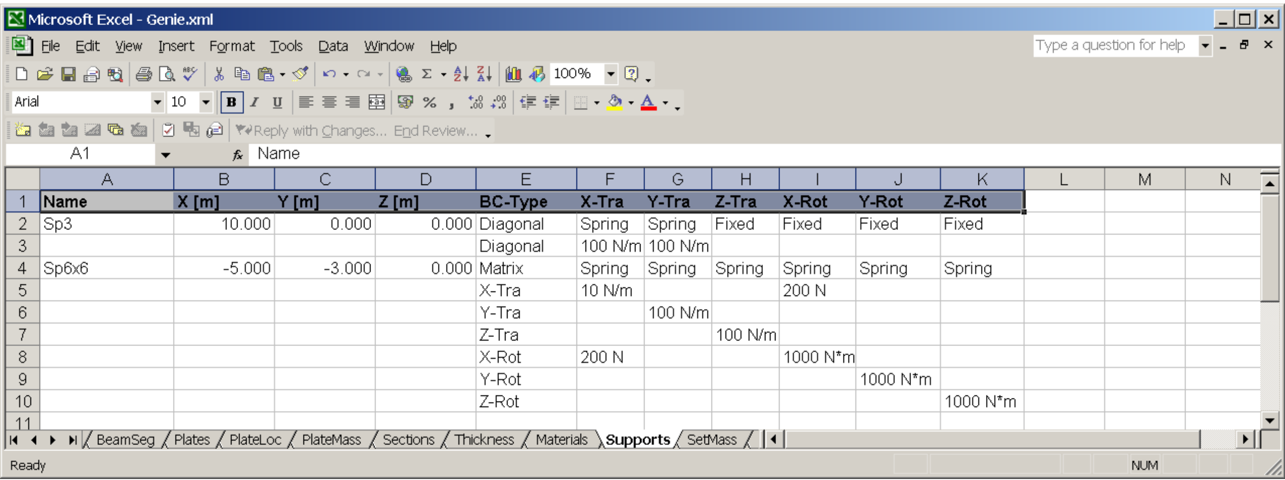


	1	2	3	4	5	6	Boundary Type
1	10 N/m	0 N/m	0 N/m	200 N	0 N	0 N	Spring
2	0 N/m	100 N/m	0 N/m	0 N	0 N	0 N	Spring
3	0 N/m	0 N/m	100 N/m	0 N	0 N	0 N	Spring
4	200 N	0 N	0 N	1000 N*m	0 N*m	0 N*m	Spring
5	0 N	0 N	0 N	0 N*m	1000 N*m	0 N*m	Spring
6	0 N	0 N	0 N	0 N*m	0 N*m	1000 N*m	Spring



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.



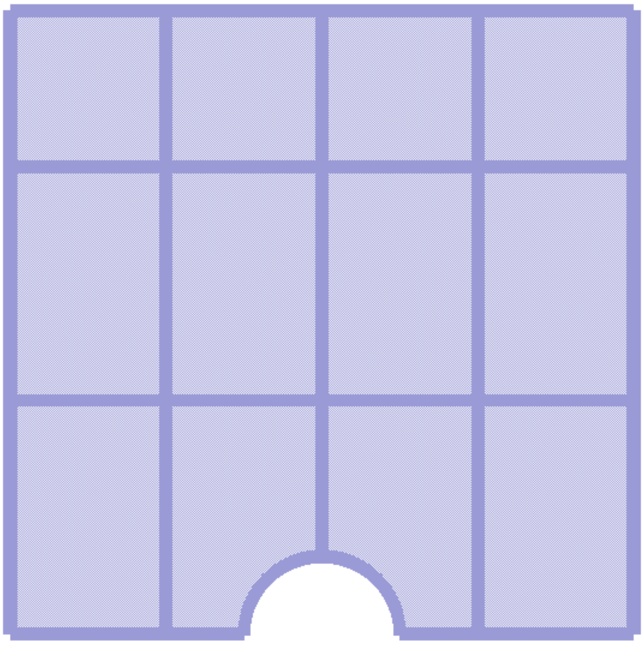
	A	B	C	D	E	F	G	H	I	J	K
1	Name	X [m]	Y [m]	Z [m]	BC-Type	X-Tra	Y-Tra	Z-Tra	X-Rot	Y-Rot	Z-Rot
2	Sp3	10.000	0.000	0.000	Diagonal	Spring	Spring	Fixed	Fixed	Fixed	Fixed
3					Diagonal	100 N/m	100 N/m				
4	Sp6x6	-5.000	-3.000	0.000	Matrix	Spring	Spring	Spring	Spring	Spring	Spring
5					X-Tra	10 N/m			200 N		
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

The final boundary condition – prescribed displacement - has been described in the previous Chapter about Loads.

3.12.2 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.

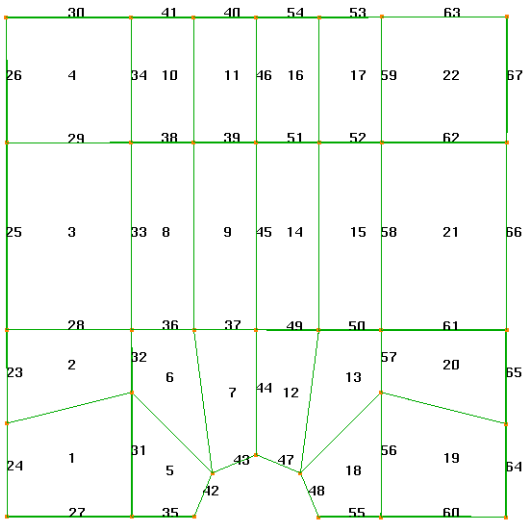


The same model is used to see the effect of the various changes in settings.

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.

3.12.2.1 Create a default finite element mesh.



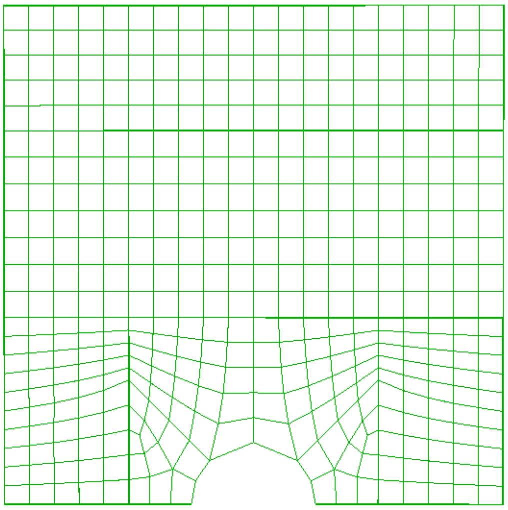
To create a finite element mesh you can use the command ***Tools|Analysis|Create Mesh***.

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*.

3.12.2.2 Adjust global mesh density

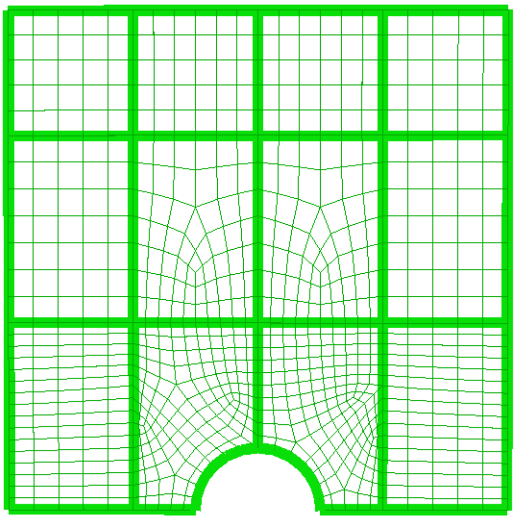


In this case the default global mesh density has been specified to 0.5 meters.

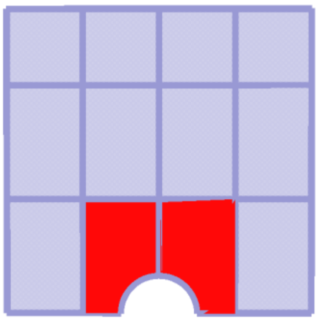
This is done by defining a mesh property from EditProperties|Mesh density. A property MD05 has been created with maximum element length 0.5 meters. This has been set to be the global default.

As can be seen, the mesh is much more dense than the example above.

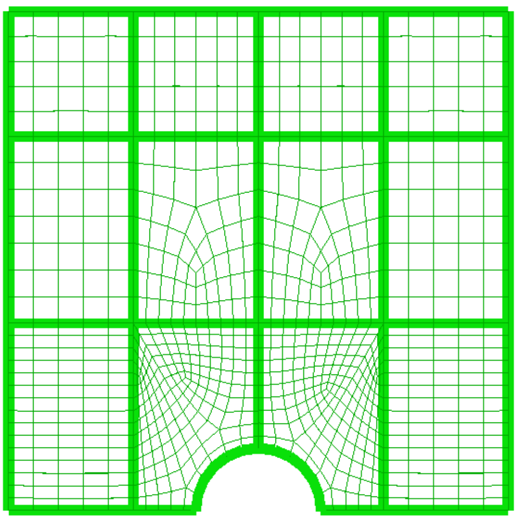
3.12.2.3 Adjust local mesh density



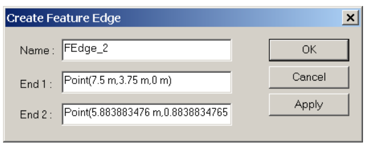
In this case local mesh density MD025 (max element length 0.25 m) has been applied to the selected plates as shown below. Note that the same settings apply to the beams that intersect the plates (i.e. not necessary to select all). The mesh is refined around the cut out and a transition zone towards the coarser mesh is automatically set up.



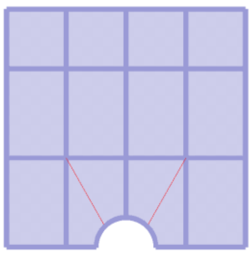
3.12.2.4 Adjust mesh with feature edges



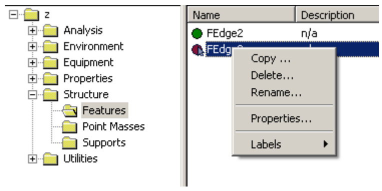
Two feature edges are inserted to improve the mesh close to the cut out. ***They are inserted from the menu Insert|Feature Edge.***

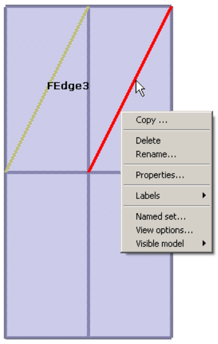


The picture to the right shows two feature edges inserted and the effect these have on the finite element mesh is shown to the left.

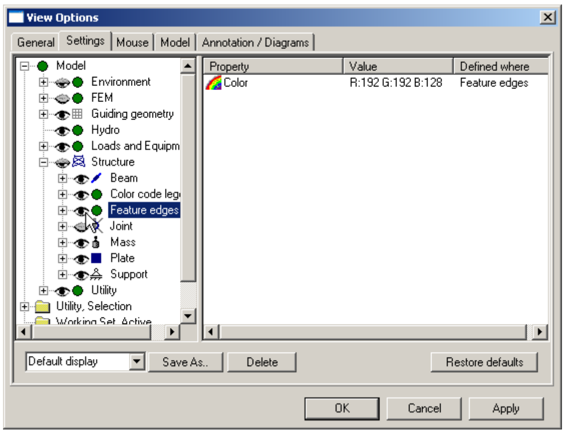


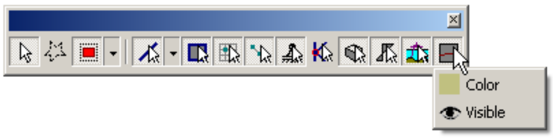
To verify, change or delete the feature edges you may select them from the browser or by graphic selection.



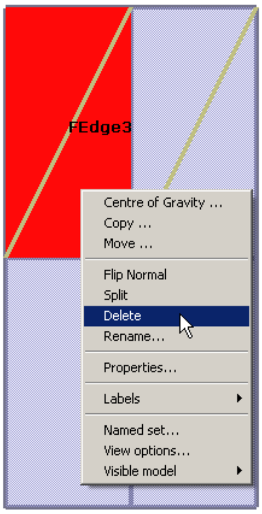


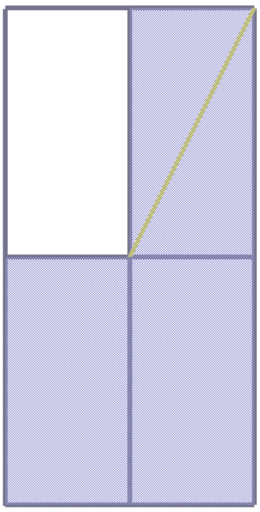
Before you can select them graphically make sure that the right options in the View|Option|Settings are ticked off. Alternatively, you may use the tool button for feature edge selection. See examples below.





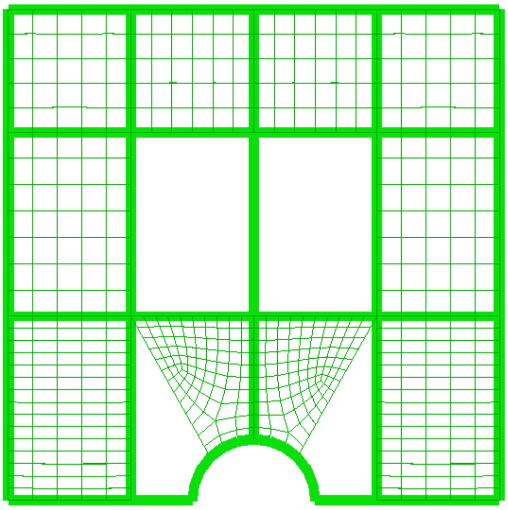
A feature edge will define a topology line (or curve) on a plate between two points. This means that the feature edge belongs to a plate and it is not possible to insert a feature edge when there are no plates. Furthermore, when deleting a plate including a feature edge the feature edge will also be deleted.



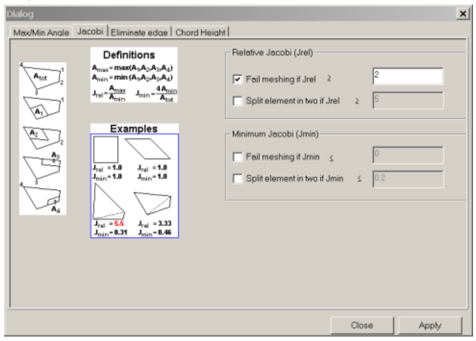


When deleting the plate, the feature edge *Fedge3* is automatically deleted,

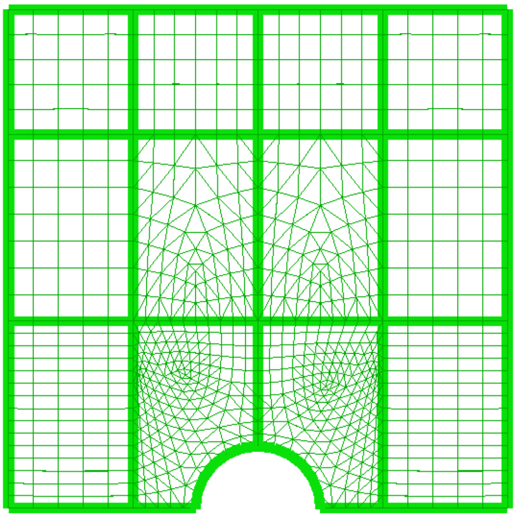
3.12.2.5 Check quality of mesh vs. Jacobi Matrix test



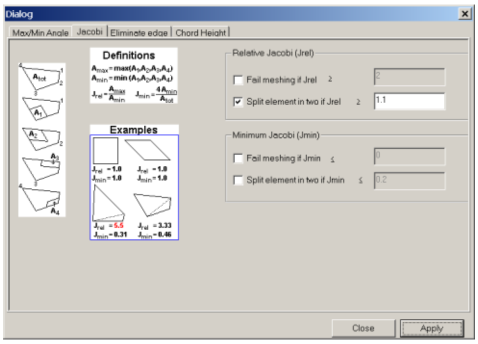
A Jacobi matrix test has been performed specifying to fail if the relative Jacobi matrix is larger than 2.0. As can be seen there are four plates where the mesh failed (GeniE will tell which plates that fail).



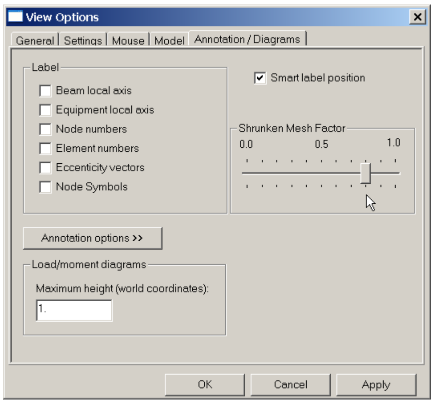
3.12.2.6 Improve quality of mesh vs. Jacobi Matrix test

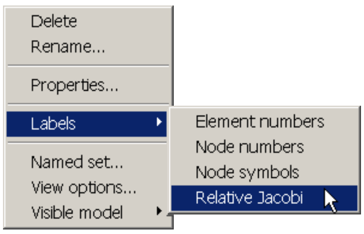


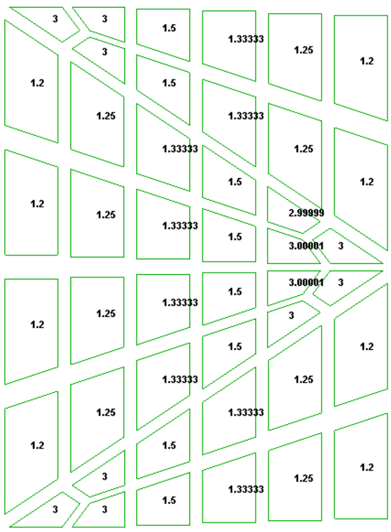
In this case it has been specified to insert triangular elements (split elements in two) when the relative Jacobi matrix is larger than 1.1.



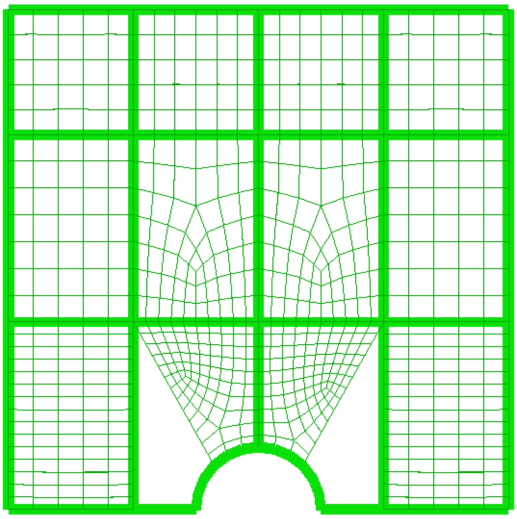
The results from a Jacobi determinant check is possible to visualise for 1st order quadrilateral elements. This example also shows how the shrunken mesh factor works.



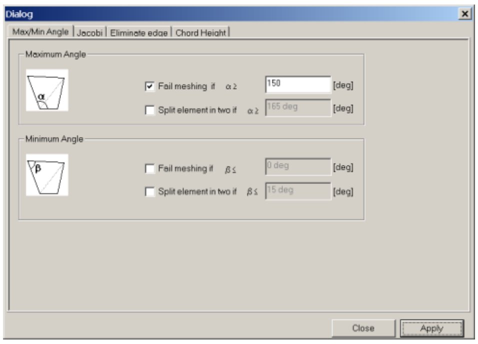




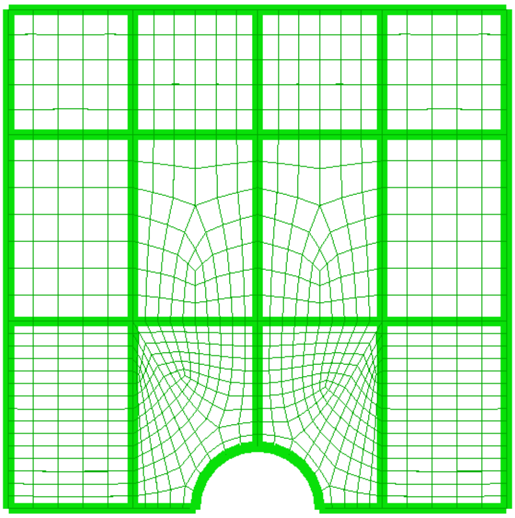
3.12.2.7 *Check quality of mesh vs finite element angles*



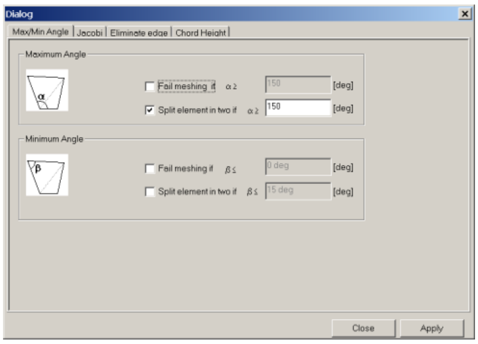
Two parts of the model was not possible to mesh because the element angles were above the failing angle. In this case GeniE is instructed to fail meshing if angles are exceeding 150 degrees.



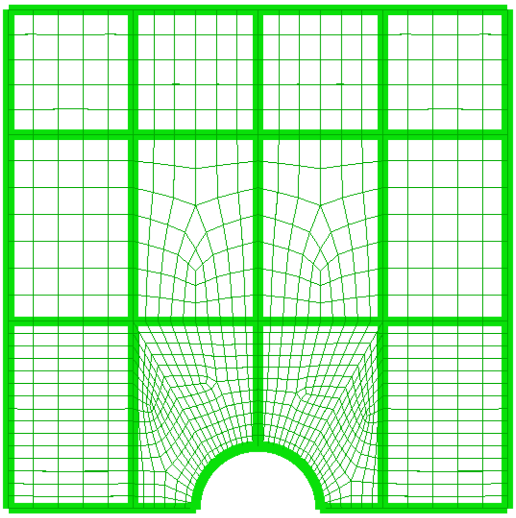
3.12.2.8 *Improve quality of mesh vs finite element angles*



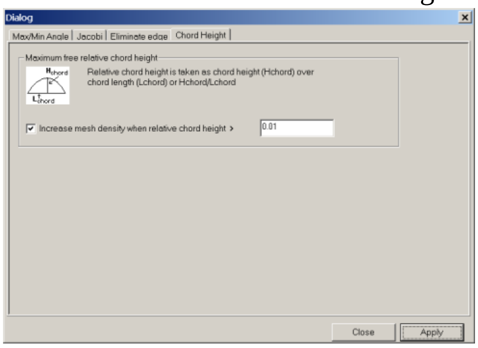
In this case the program is instructed to split elements in two if the mesh angle of the automatic created mesh is above 150 degrees.



3.12.2.9 *Adjust mesh along the curved edge*



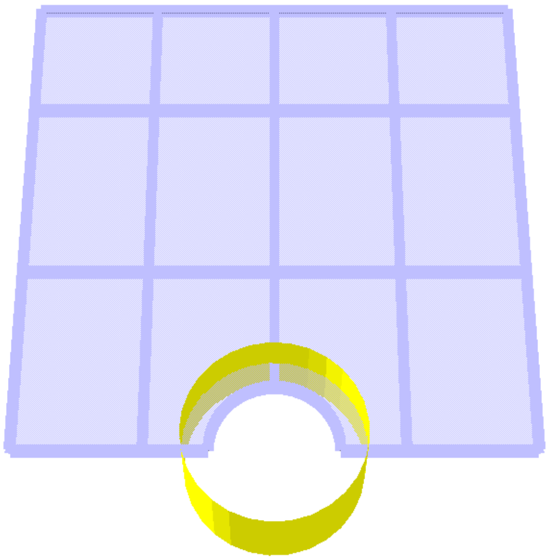
The option of controlling the relative chord height of mesh along a curved edge has been used. The relative chord height has been set to 1.1 and a dense mesh is established along the curve.



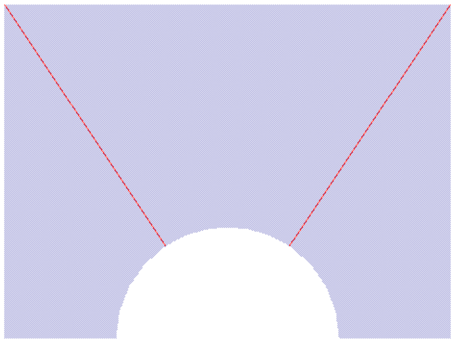
3.12.2.10 Controlling the mesh close to the cut out

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.

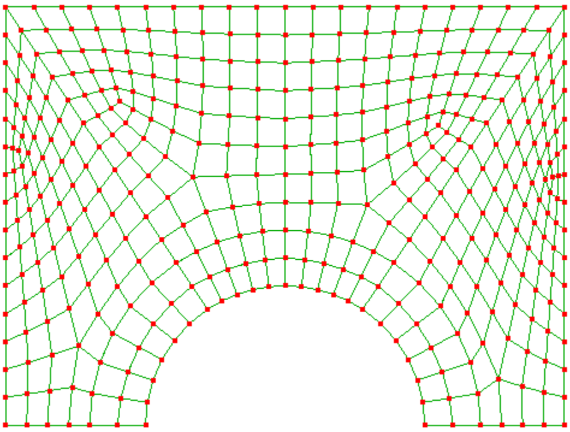
The pictures below explain how this can be achieved.



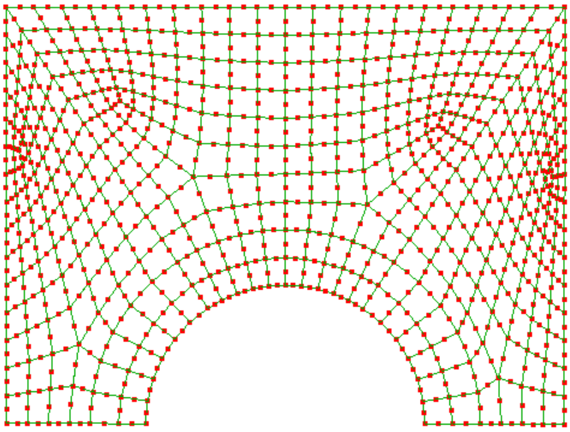
Make a cut with a profile larger than the cut out



Inserting two feature edges in addition to the feature edges inserted at the new curved cut lines



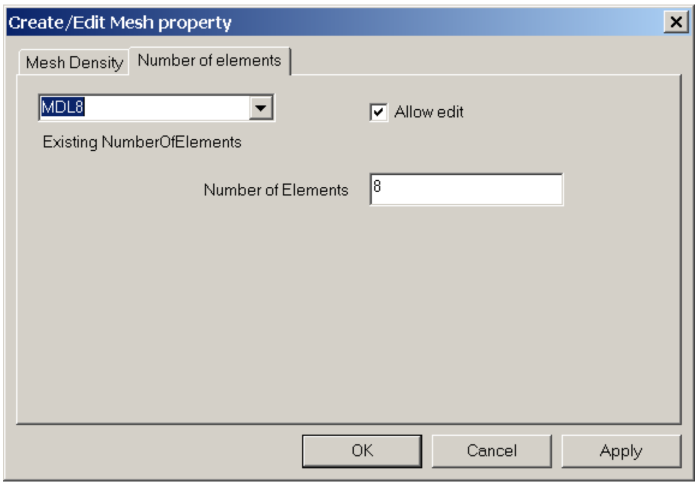
The finite element mesh using 1st order elements perpendicular to the curvature



The finite element mesh using 2ndt order elements perpendicular to the curvature

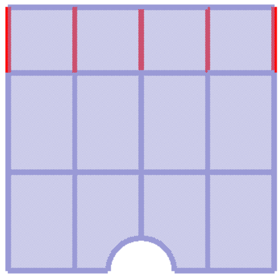
The feature ***Tools|Structure|Geometry|Simplify Topology*** should be used when you have done changes to the model that influences the topology. This feature is described under Rules setting and works only for planar plates and beams.

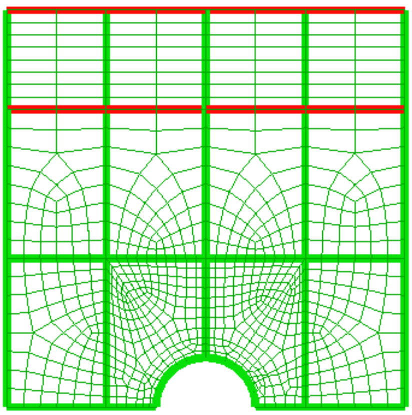
3.12.2.11 Controlling the mesh along straight lines



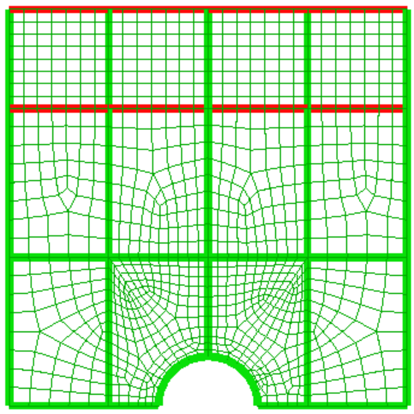
Using this feature it is possible to control number of elements (with equal length) along a straight line. These settings may be applied to beams and feature edges.

In this example the highlighted beams have been assigned 8 equal elements.



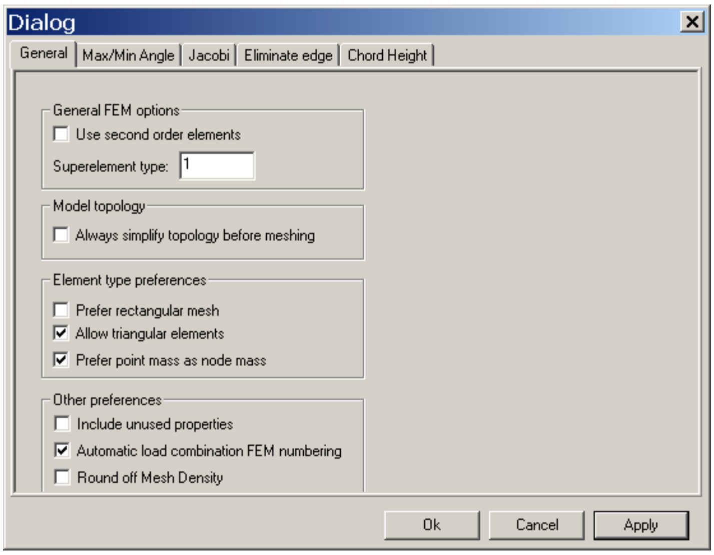


Number of elements along each segment set to 2



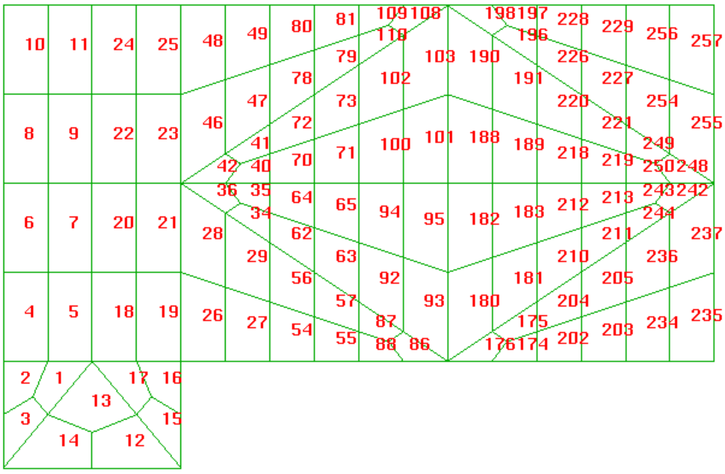
Number of elements along each segment set to 7

3.12.2.12 Specify type of elements to be used in FE mesh



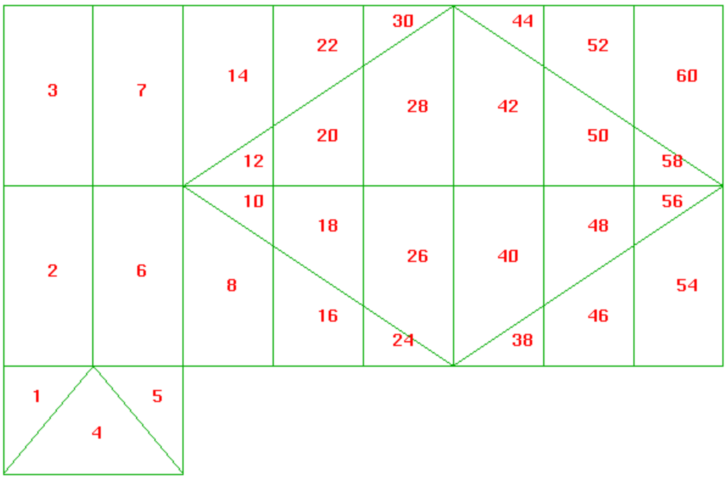
Per default GeniE will create quadrilateral and triangular finite elements. You may override this setting by using the ***Edit|Rule|Mesh settings***.

The example below shows the difference in using this rule.

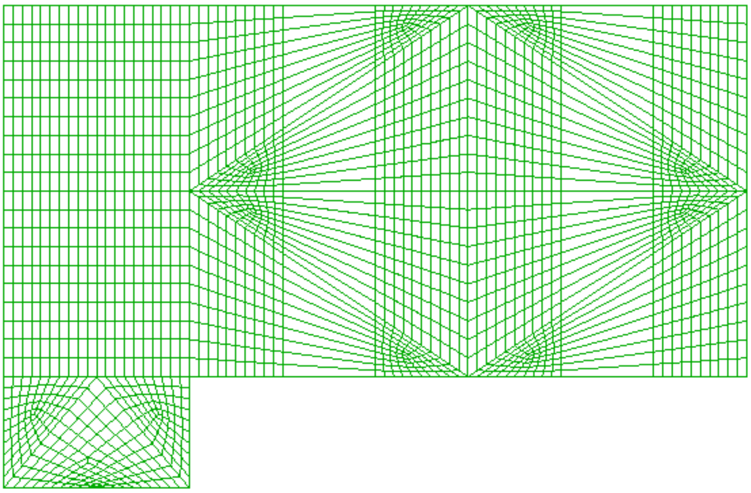


No triangular elements may be used. There are no specifications of mesh density.

The finite element mesh consists of quadrilateral elements only.



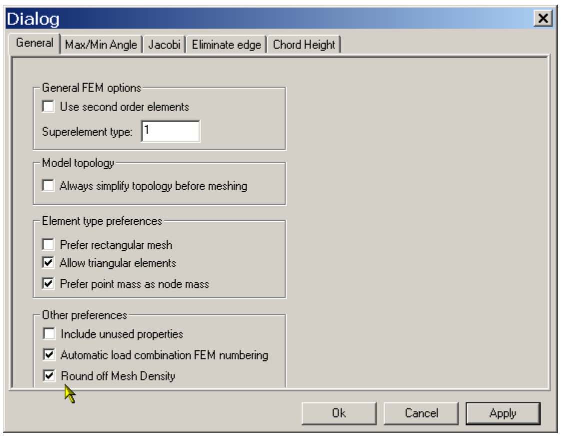
Default mesh rule to allow for triangular elements has been used. There are no specifications of mesh density.

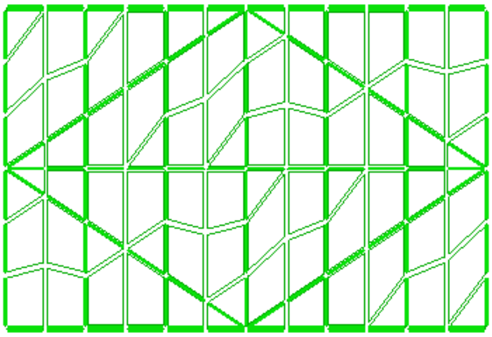


Preferred rectangular mesh and no triangular elements. There are no specifications of mesh density

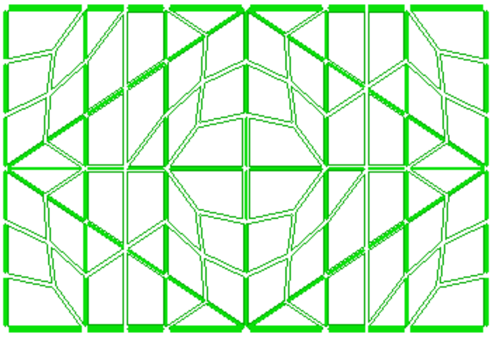
3.12.2.13 *Use feature for round off mesh density*

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.

3.12.2.14 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.

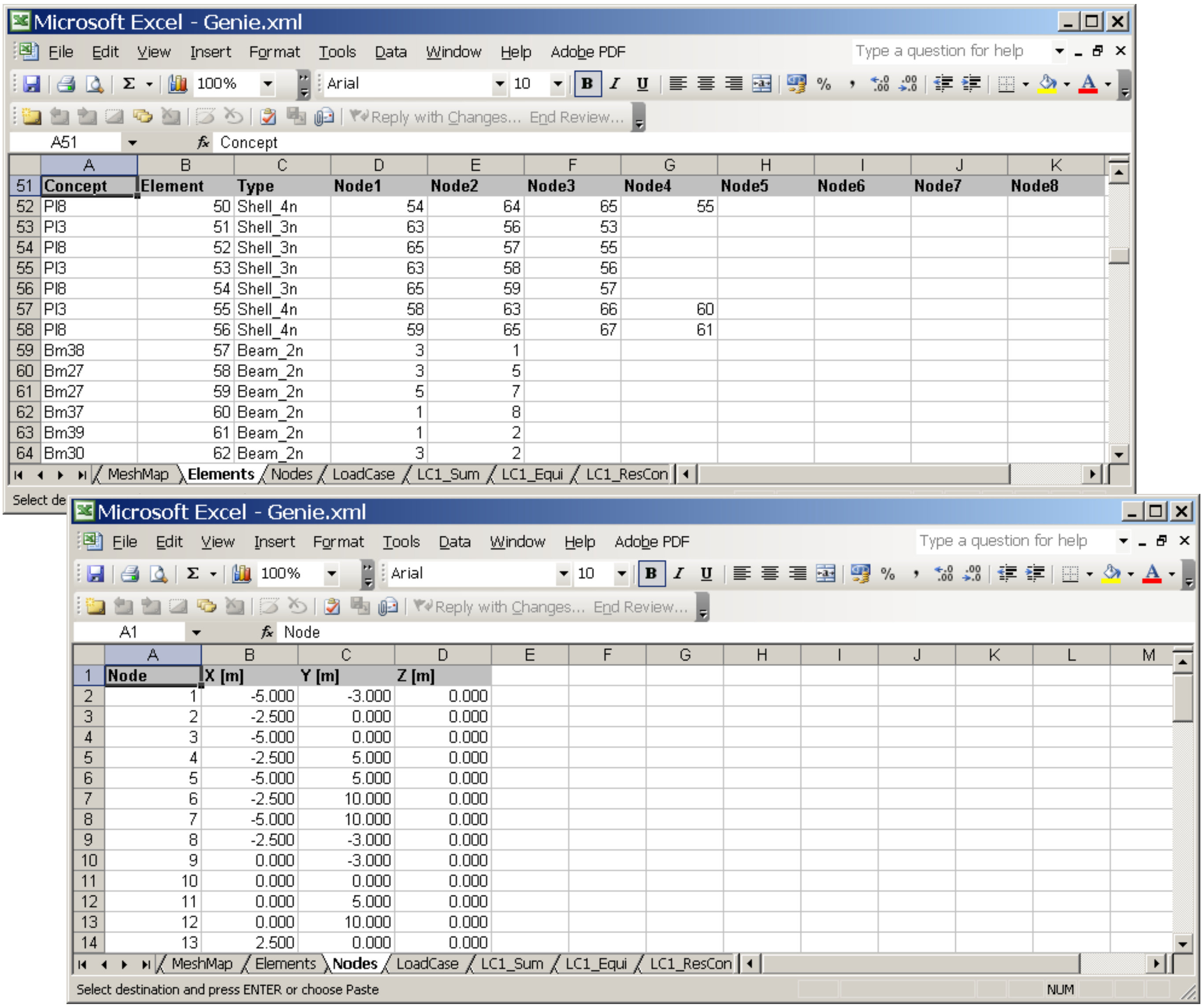
1. Number of elements along a line (beam or feature edge)
2. Mesh density applied to a line (beam or feature edge) and a plate
3. Global settings specified in the mesh rules
4. If there are no settings, the settings specified along edge are inherited

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.

3.12.2.15 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.



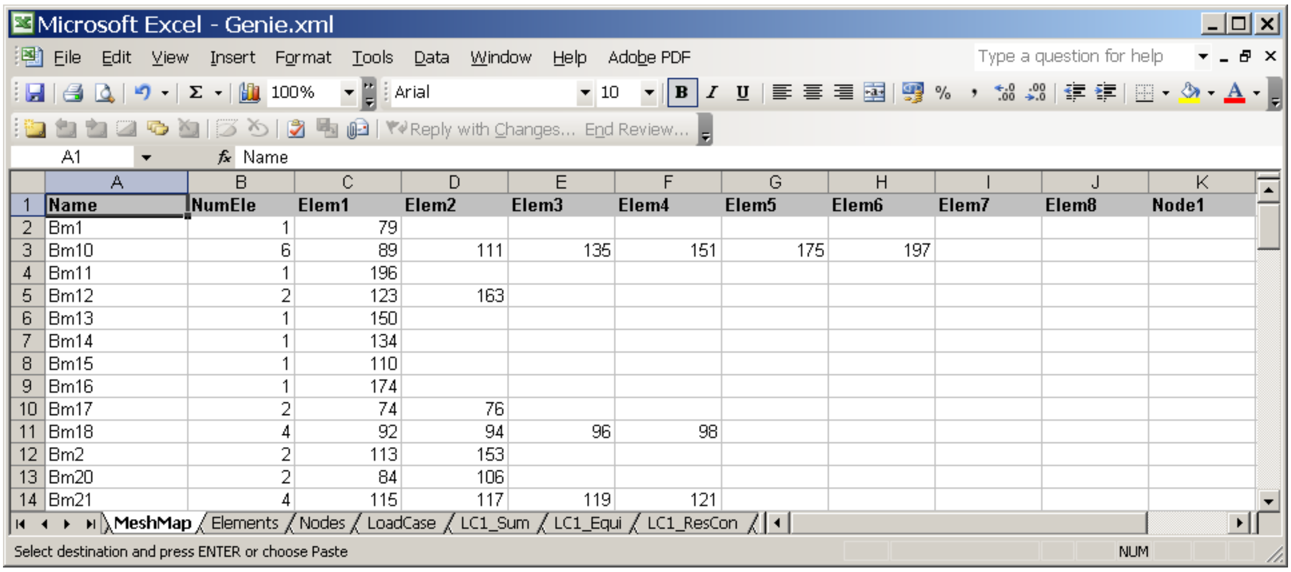
Microsoft Excel - Genie.xml

Concept	Element	Type	Node1	Node2	Node3	Node4	Node5	Node6	Node7	Node8
Pl8	50	Shell_4n	54	64	65	55				
Pl3	51	Shell_3n	63	56	53					
Pl8	52	Shell_3n	65	57	55					
Pl3	53	Shell_3n	63	58	56					
Pl8	54	Shell_3n	65	59	57					
Pl3	55	Shell_4n	58	63	66	60				
Pl8	56	Shell_4n	59	65	67	61				
Bm38	57	Beam_2n	3	1						
Bm27	58	Beam_2n	3	5						
Bm27	59	Beam_2n	5	7						
Bm37	60	Beam_2n	1	8						
Bm39	61	Beam_2n	1	2						
Bm30	62	Beam_2n	3	2						

Microsoft Excel - Genie.xml

Node	X [m]	Y [m]	Z [m]
1	-5.000	-3.000	0.000
2	-2.500	0.000	0.000
3	-5.000	0.000	0.000
4	-2.500	5.000	0.000
5	-5.000	5.000	0.000
6	-2.500	10.000	0.000
7	-5.000	10.000	0.000
8	-2.500	-3.000	0.000
9	0.000	-3.000	0.000
10	0.000	0.000	0.000
11	0.000	5.000	0.000
12	0.000	10.000	0.000
13	2.500	0.000	0.000

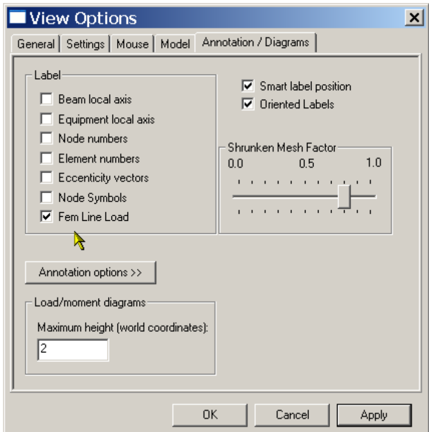
There is also a table showing the relation between concepts and finite elements, i.e. which finite elements belonging to a concept (beam or plate).



Name	NumEle	Elem1	Elem2	Elem3	Elem4	Elem5	Elem6	Elem7	Elem8	Node1
Bm1	1	79								
Bm10	6	89	111	135	151	175	197			
Bm11	1	196								
Bm12	2	123	163							
Bm13	1	150								
Bm14	1	134								
Bm15	1	110								
Bm16	1	174								
Bm17	2	74	76							
Bm18	4	92	94	96	98					
Bm2	2	113	153							
Bm20	2	84	106							
Bm21	4	115	117	119	121					

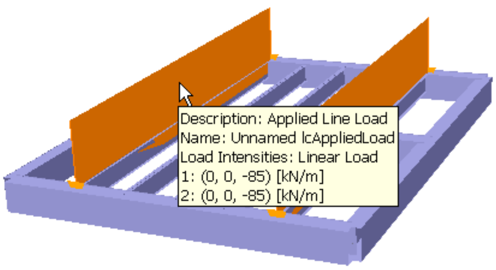
3.12.2.16 *Graphic verification of the finite element loads*

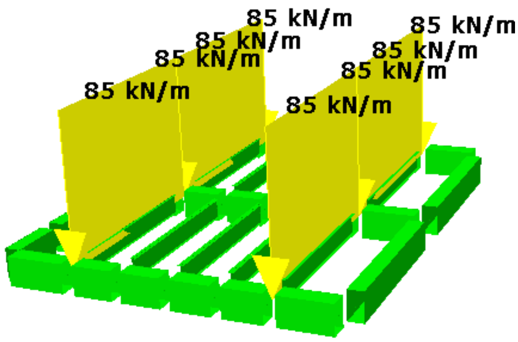
You can graphically verify the finite element loads by using the feature for such from ***View|Options|Annotation/Diagrams***.



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.

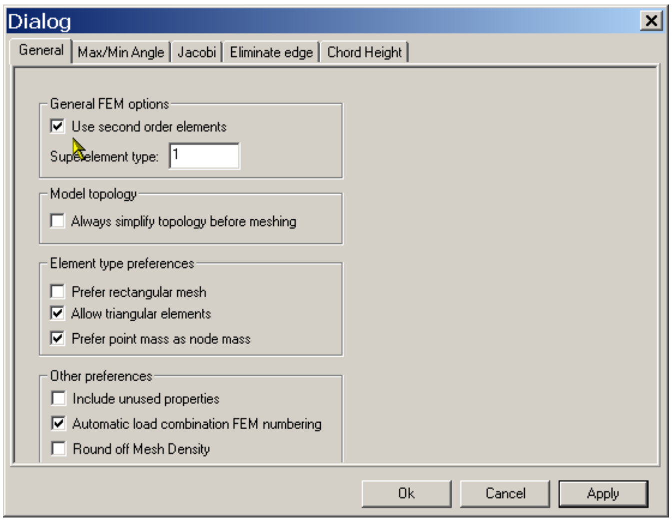
As can be seen, the values are in accordance with each other; hence all loads are safely transferred from the concept model to the finite element model.





3.12.3 Element types

GeniE can create both 1^{st} and 2^{nd} 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^{nd} 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.

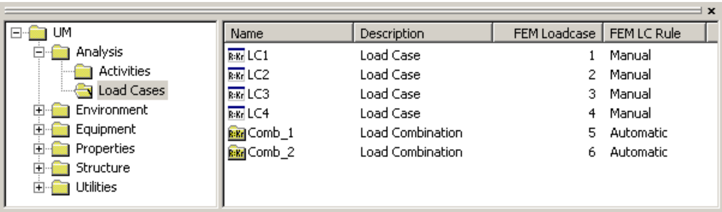
Name	**Type**	**Order**	**Comments**
2-node beam element	BEAS	1^{st}	
3-node beam element	BTSS	2^{nd}	Cannot be used in framecode checking
Quadrilateral flat thin shell element	FTRS	1^{st}	
Triangular flat thin shell element	FQUS	1^{st}	Inserted when adjusting mesh rules to split elements
Quadrilateral sub parametric curved thick shell element	SCQS	2^{nd}	
Triangular sub parametric curved thick shell element	SCTS	2^{nd}	Inserted when adjusting mesh rules to split elements
Quadrilateral flat thin shell with drilling dof	FQAS	1^{st}	Includes the rotational dof around the axis perpendicular to the membrane in the membrane formulation
Triangular flat thin shell with drilling dof	FTAS	1^{st}	- o -
Non-structural 2 node beam element	BEAS	1^{st}	Special variant of BEAS with no contribution of the structural stiffness
Truss element	TESS	1^{st}	Element type with no bending stiffness
Spring to ground	GSPR	1^{st} & 2^{nd}	Includes the 6x6 matrix
Shim element	GLSH	1^{st} & 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^{st} & 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^{st} and 2^{nd} order element types.

3.12.4 Controlling the FEM Loadcase numbers

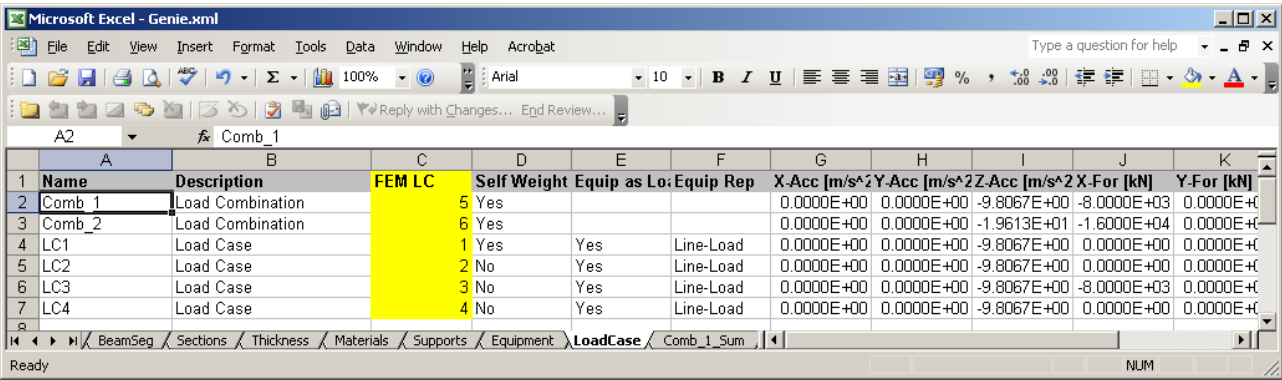
Chapter 3.8.5 explains how to control the loadcase names as well as the loadcase numbers to appear on the finite element model. When performing an integrated analysis completely from the GeniE domain, the sequence of FEM Loadcase numbers do not play an important role since you refer to the loadcase names 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*).



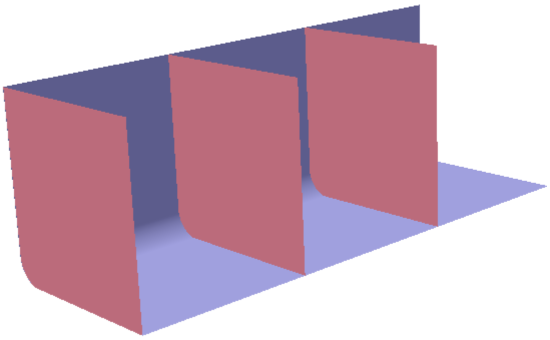
Name	Description	FEM LC	Self Weight	Equip as Lo	Equip Rep	X-Acc [m/s^2	Y-Acc [m/s^2	Z-Acc [m/s^2	X-For [kN]	Y-For [kN]
Comb_1	Load Combination	5	Yes			0.0000E+00	0.0000E+00	-9.8067E+00	-8.0000E+03	0.0000E+0
Comb_2	Load Combination	6	Yes			0.0000E+00	0.0000E+00	-1.9613E+01	-1.6000E+04	0.0000E+0
LC1	Load Case	1	Yes	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00	0.0000E+00	0.0000E+0
LC2	Load Case	2	No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00	0.0000E+00	0.0000E+0
LC3	Load Case	3	No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00	-8.0000E+03	0.0000E+0
LC4	Load Case	4	No	Yes	Line-Load	0.0000E+00	0.0000E+00	-9.8067E+00	0.0000E+00	0.0000E+0

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)

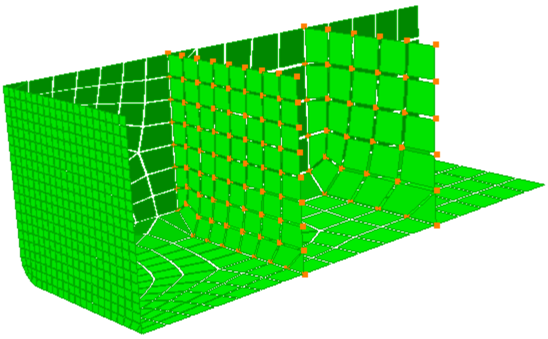
3.12.5 Mesh locking

When working with repetitive structure it may be desired to have the same mesh density for these parts. One example is transverse bulkheads along a ship or pontoon longitudinal direction. The example below shows the sequence of ensuring that the mesh is identical to specified parts.

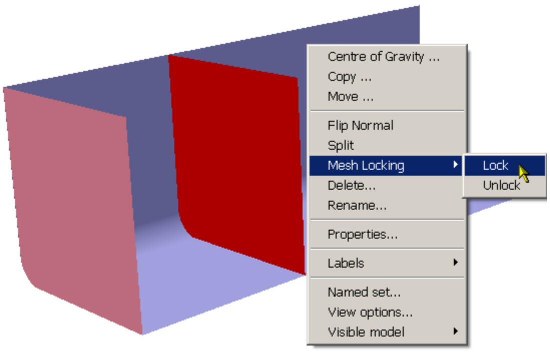


A small model has been created with the same maximum density for the whole structure except for the left bulkhead. This part has a finer mesh density.

For this model, you may want to have identical mesh for the middle and the right bulkheads.



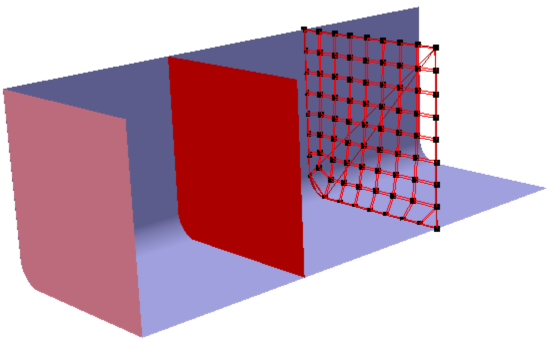
A finite element mesh is created (ALT+M or ***Tools|Analysis|Create Mesh***). The middle and right bulkheads have different mesh since the left bulkhead has a finer mesh density than the rest of the structure.

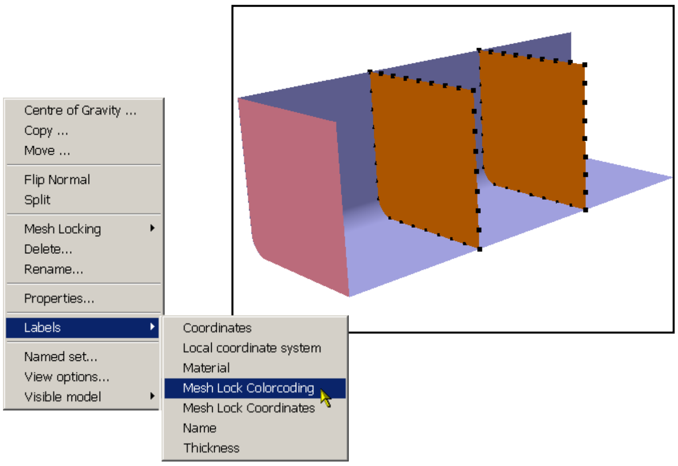


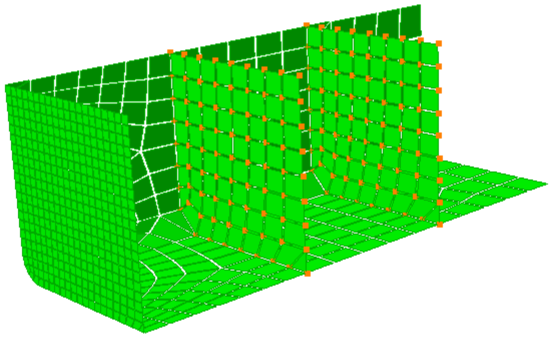
If you want to keep the mesh density of the middle bulkhead you select this part and use feature *Mesh Locking|Mesh* (select the plate and RMB).

When copying the plate you also copy the mesh settings.

You can also see where mesh locking has been applied by e.g. selecting the entire structure, RMB and choose label.

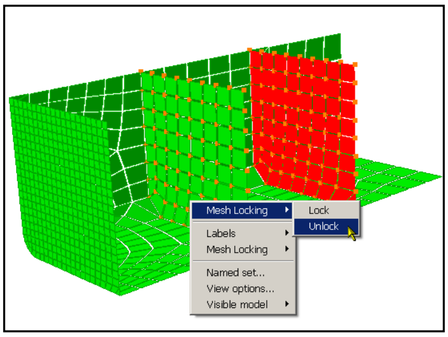


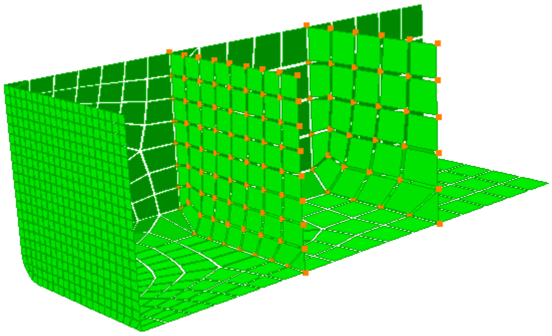




An identical mesh is now created for the middle and the right bulkheads.

If you want to remove the mesh settings for the right bulkhead you can select the plate, RMB and *Mesh Locking|Unlock*.

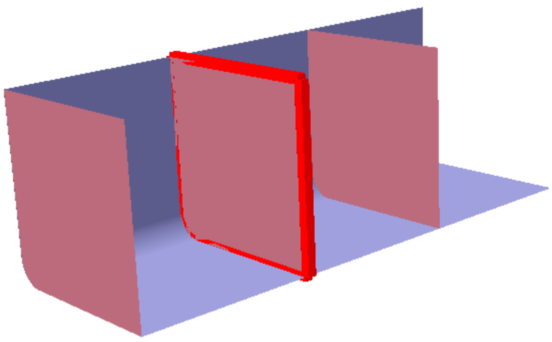




When creating a new mesh, the mesh is now identical to the original mesh without any mesh locking specified.

3.12.6 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.

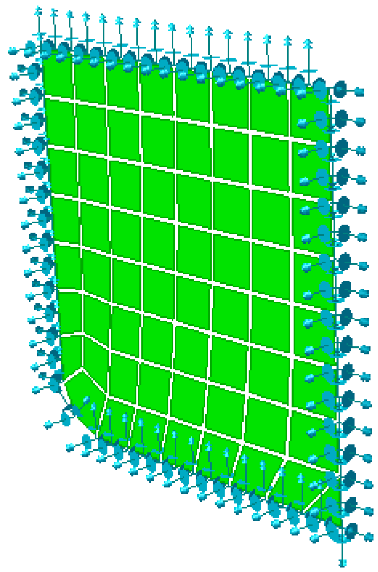
The analysis has one loadcase LC1_Weight.

When setting this activity current, a mesh operation will create a finite element model for the entire structure and the loadcases.

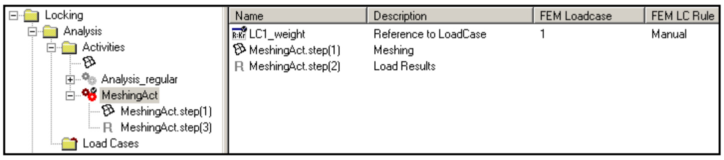
3.12.6.1 Mesh part of model using script language

```
MeshingAct = Analysis();
MeshingAct.add(MeshActivity());
MeshingAct.add(LoadResultsActivity());
MeshingAct.step(1).subset = MySet;
```

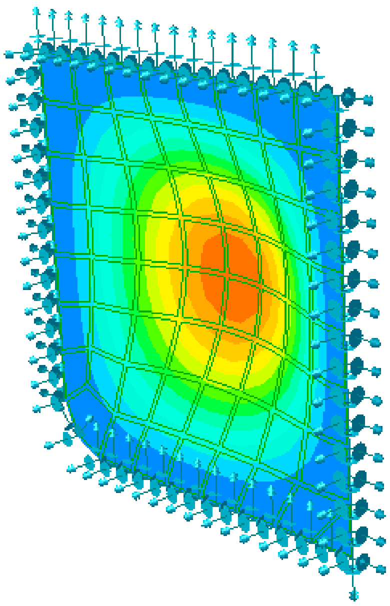
To create an activity that will mesh only the selected part it is necessary to define the script commands as shown to the left. In this case, it is the set called *MySet* that will be meshed in the activity *MeshingAct*.



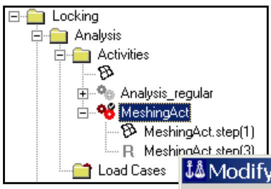
When making a mesh (Alt+M or ***Tools|Analysis|Create mesh***) only the selected parts and the loadcases will be part of the model. You need to ensure that *MeshingAct* is the active activity.

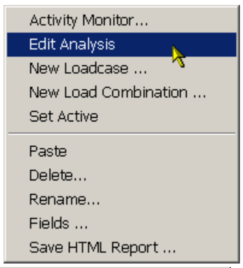


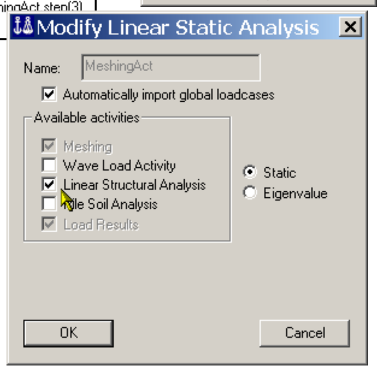
You can now export the FEM file from ***Tools|Analysis|FEM file***.



In case you want to run analysis of the selected part you can add linear analysis to the activity *MeshingAct* and run the analysis.

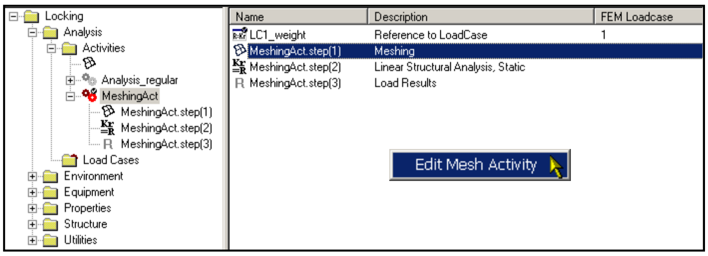




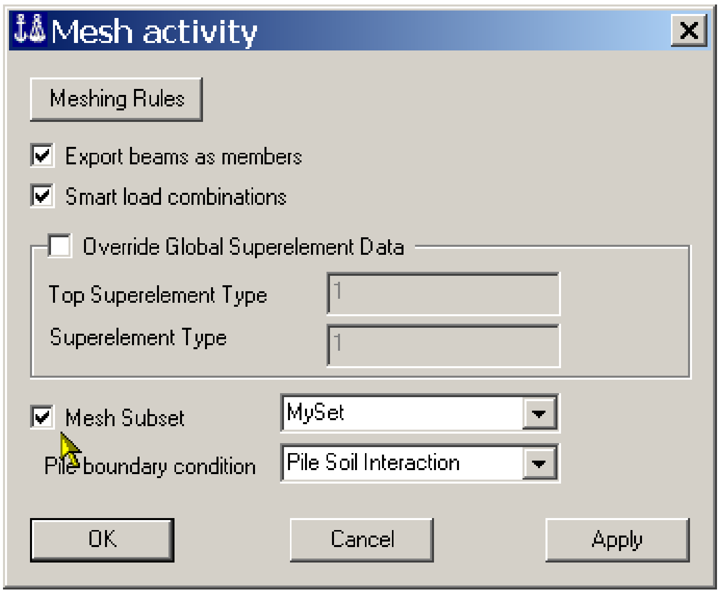


If you want to analyze the whole structure you set the other activity *Analysis_rgular* to active and run analysis.

3.12.6.2 *Mesh part of model using GUI*



Select the mesh operation in the activity *MeshingAct*, RMB and choose *Edit Mesh Acivity*.



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.

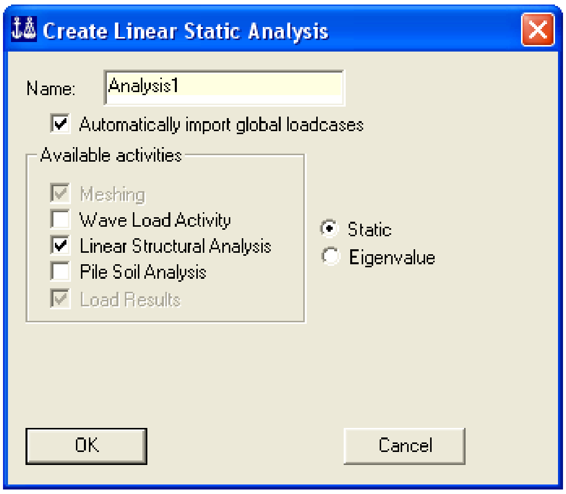
Also observe that you can access meshing rules as well as change the superelement numbering. This means that you can create several superelements and export these for a subsequent analysis using Presel to build an entire structure, please consult the Presel User Manual for more details.

3.13 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

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.

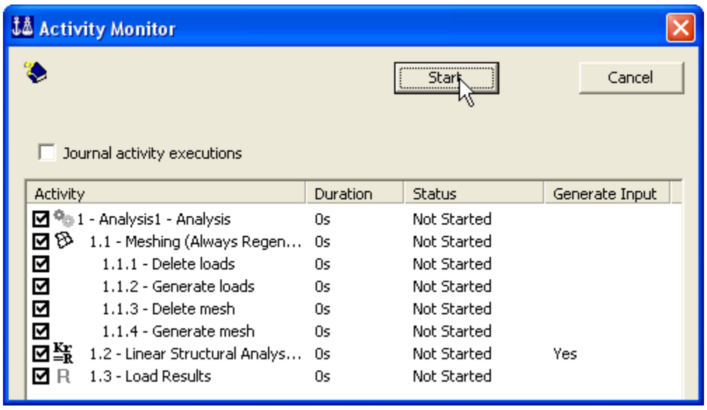
When starting a structural analysis in GeniE (***Tools|Analysis|Activity Monitor)*** this implies

- 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.

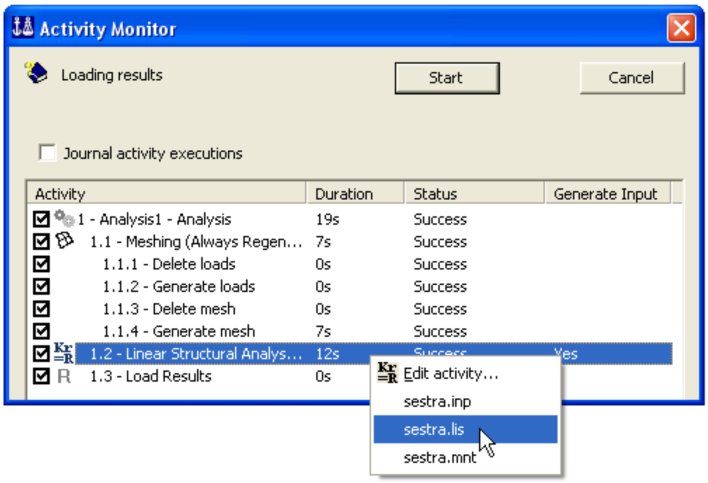
You start a linear static structural analysis by

- ***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

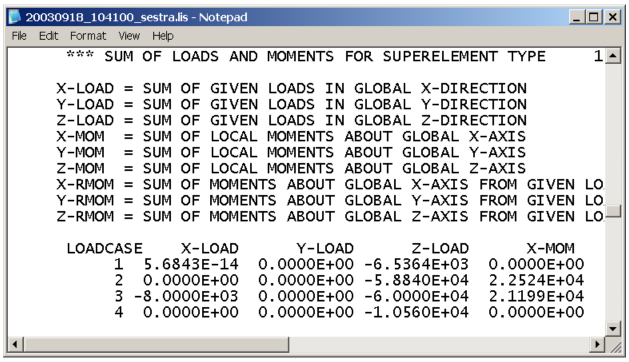


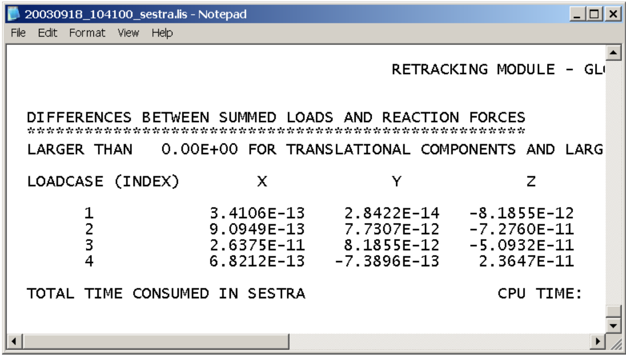
When the analysis is done, the *Activity Monitor* specifies if the analysis is successful or whether there are warnings or errors in the respective analysis steps. You have instant access to the details from analysis runs. By pushing RMB on an activity you can open listing files (or others) to investigate the details.

The files will be opened in your default editor for file types *.LIS, *.MNT and *.INP.

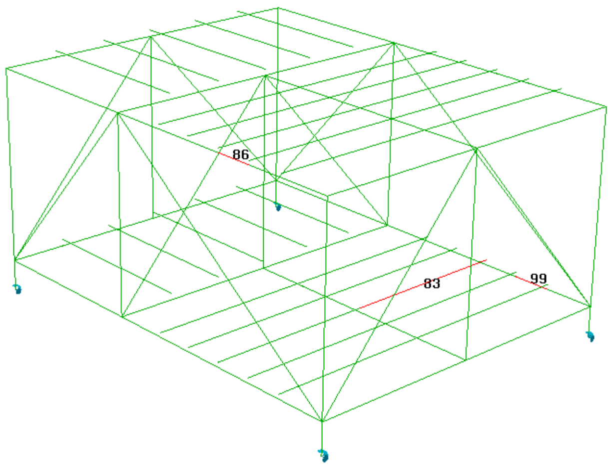


Below are examples of a Sestra listing file viewed in MS Notepad (load sums and difference between load sums and reaction forces).

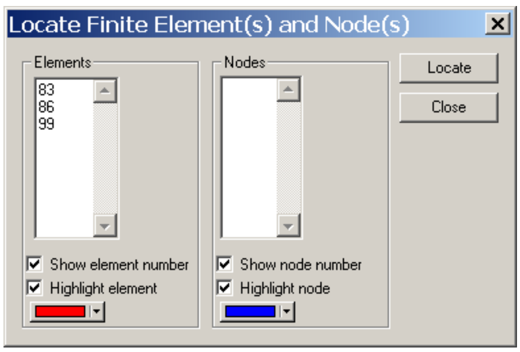




If the analysis for some reasons fails, Sestra will report that e.g. element number 99 has failed. To find the exact location of this element you may use the feature ***Tools|Analysis|Locate FE***. The example below shows how to find finite element numbers 83, 86, and 99 (FE nodes not shown).



Tools|Analysis|Locate FE gives this input dialog

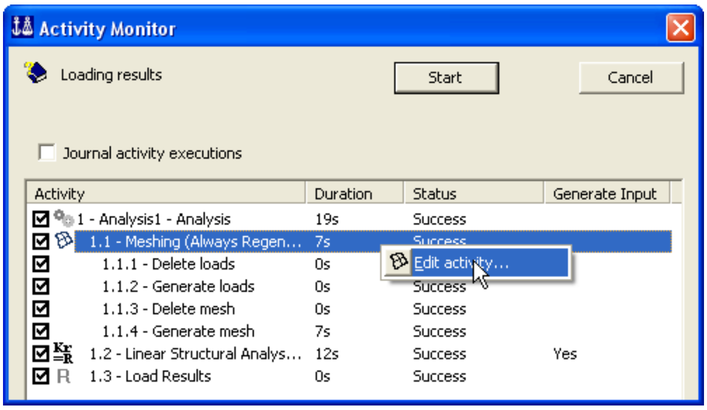


There are several ways to improve the mesh if the analysis fails, for example by use of split element if Jacobi matrix test fails or maximum angle is exceeded, inserting feature edges or by increasing the mesh density. See Chapter 3.12.2 for further details.

3.13.1 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

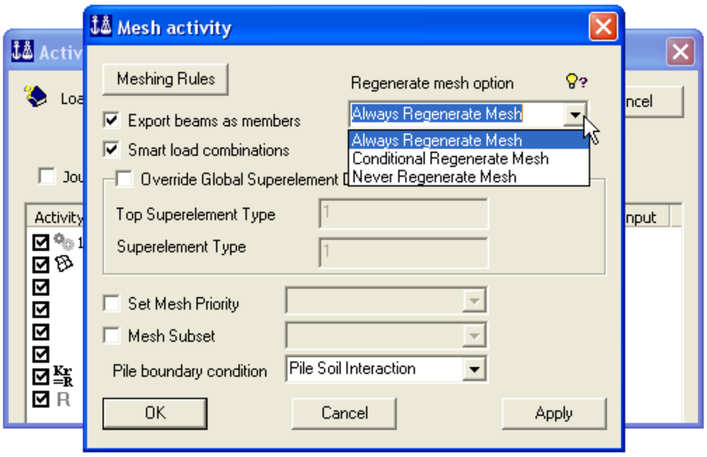
When the Meshing activity option is set to *Always Regenerate Mesh* – then a complete mesh generation is performed and the user may not turn off any of the individual steps, i.e. the steps 1.1.1, 1.1.2, 1.1.3 and 1.1.4 are all executed.

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.

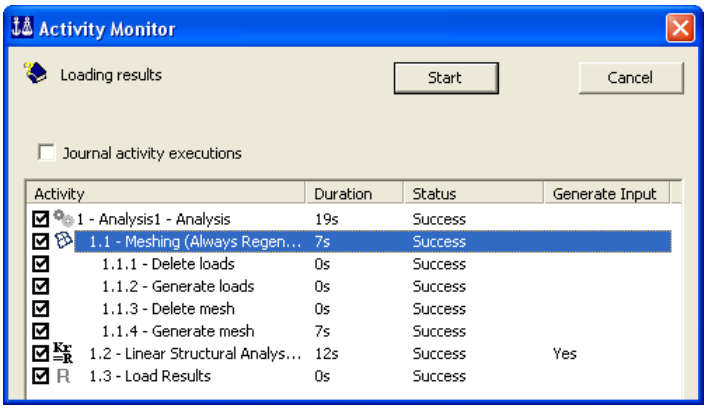
Never Regenerate Mesh

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.



3.13.2 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.

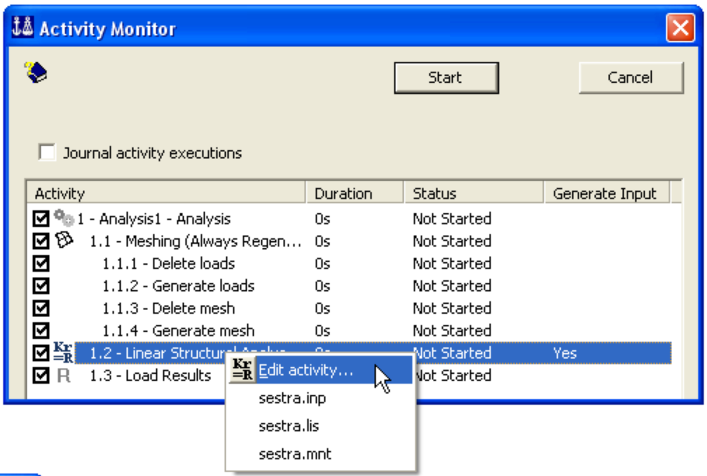
If “Delete Mesh” is also checked, a new mesh will be generated.

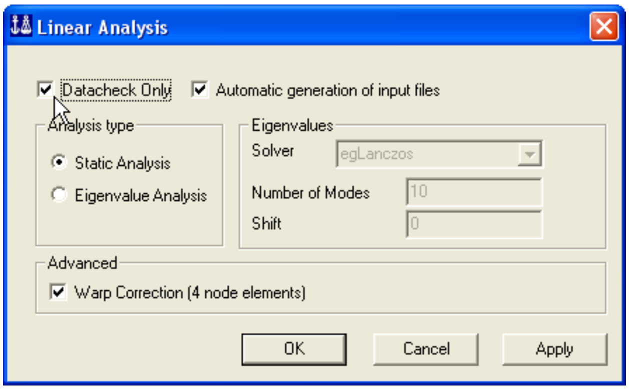
3.13.3 Modify the analysis parameters

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*.

3.13.3.1 Datacheck only

This example shows how to run a datacheck. Select the *Linear Structural Analysis*, push **RMB** and select *Edit Activity*.



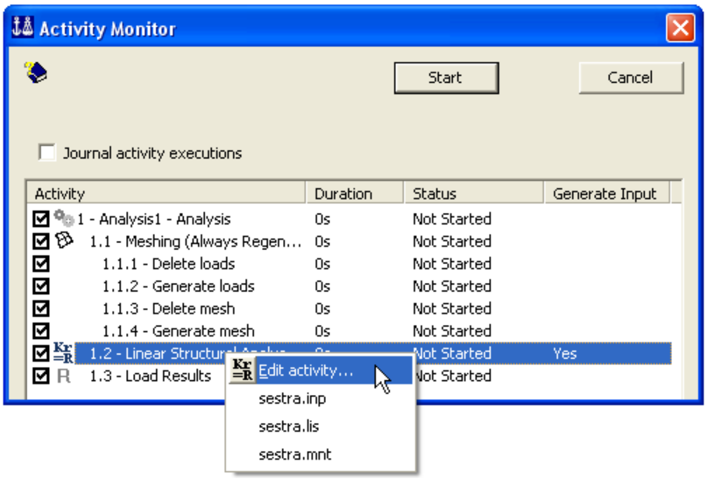


Then tick off the *Datacheck Only* and push OK.

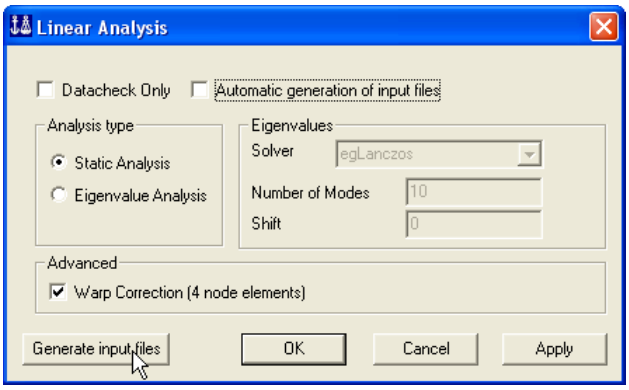
When the datacheck is done, no results are calculated

3.13.3.2 Editing the analysis parameters

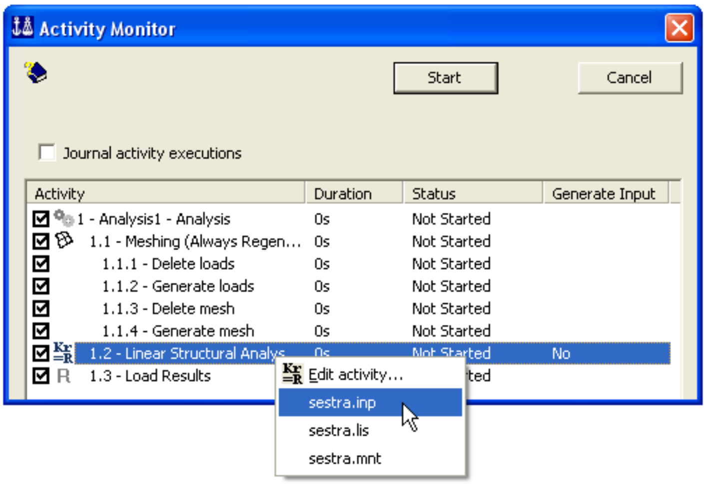
You may also edit the input analysis parameters to Sestra. In such case you should have experience in running Sestra from before or access to the Sestra user manual to know the consequences of changing such parameters. In the following an example is given on how to change the input file so that the block solver in Sestra is used in stead of the multifront solver (this is done by removing the SOLM card).



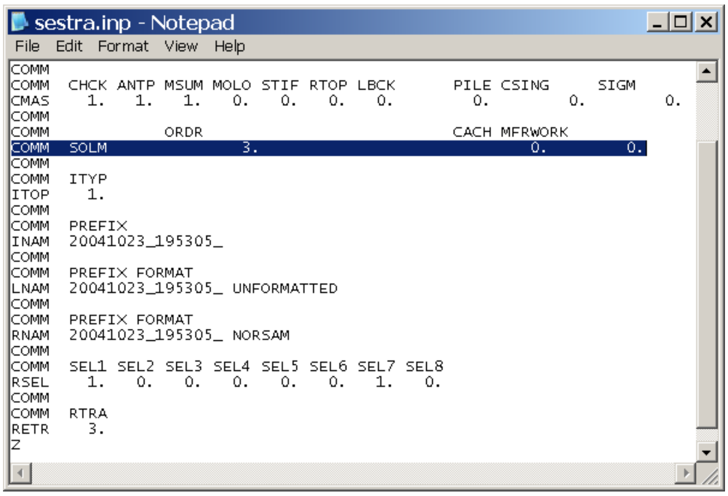
Select the *Linear Structural Analysis*, push **RMB** and select *Edit activity*.



Deactive *Automatic generation of input files* and push *Generate input files*. You have now forced the program to create a Sestra input file which you may edit.



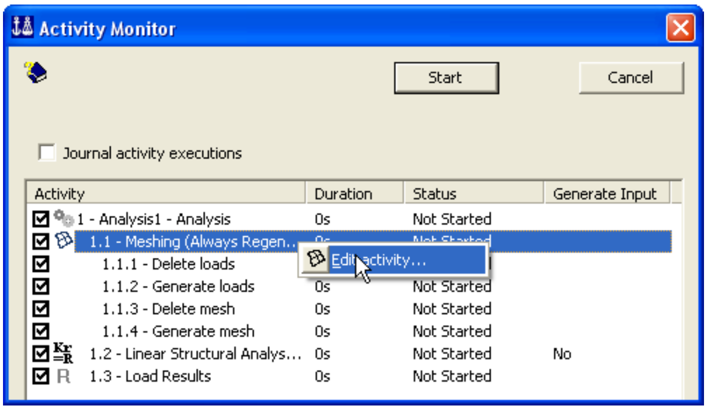
Edit the Sestra input file by selecting *sestra.inp*, the editor being used depends on your default editor for files with extension **.inp*.



The multifront solver is used when the SOLM card is present. In this case it has been commented out, hence the block solver (or the supermatrix solver) will be used when performing the analysis).

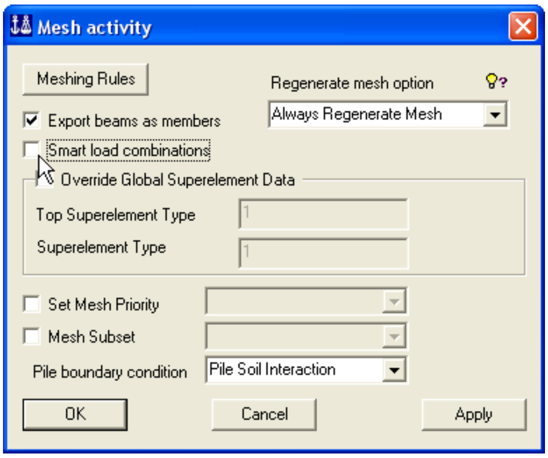
The manually specified input file for Sestra will be used for all analysis of this model until you either change it manually (edit) or activate *Automatic generation of input files* on the *Activity Monitor*.

3.13.3.3 De-activate smart load combinations



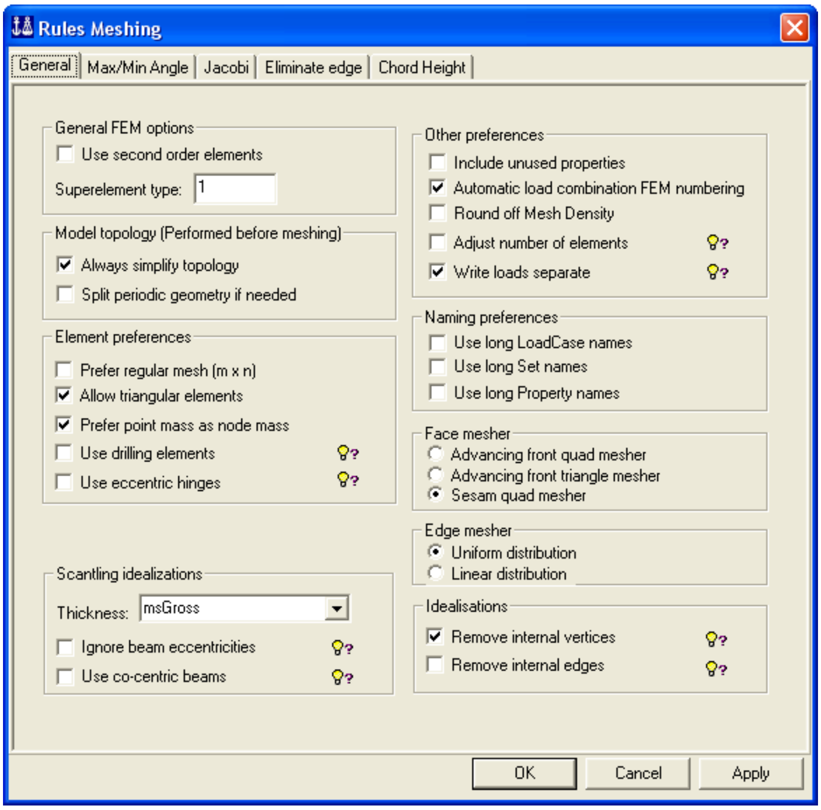
Select the *Meshing*, push **RMB** and select *Edit activity*.

Remember that smart load combinations is the default setting.



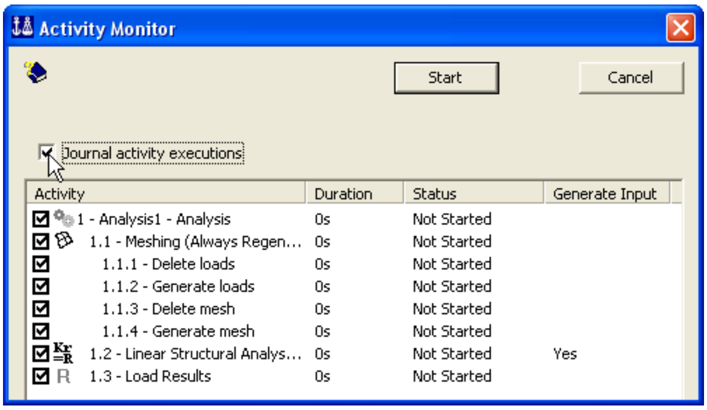
De-activate the *Smart load combinations*. When running the analysis the load computations are computed by Sestra Sestra, see Chapter 3.13.2 for more details.

3.13.3.4 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).

3.13.3.5 Journal analysis runs

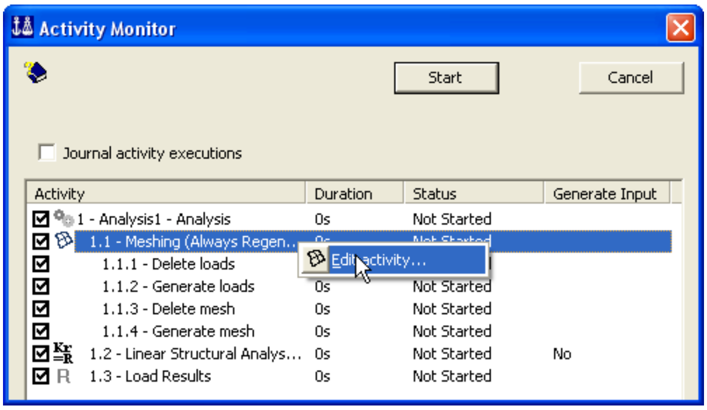


Activate the Journal activity executions. This operation will add the outlined details on the journal file in addition to those generated when defining an activity:

```
Analysis1 = Analysis();
Analysis1.add(MeshActivity());
Analysis1.add(LinearAnalysis());
Analysis1.add(LoadResultsActivity());
Analysis1.execute();
```

3.13.4 Smart load combinations

When this feature is activated this option the load combinations are computed when needed inside GeniE when looking at results.

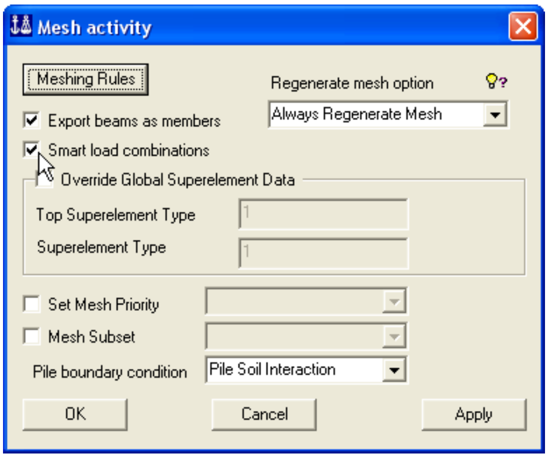


When deactivating the smart load combinations GeniE calculates the complete sets of the combined/scaled loads and writes this load information onto the *.FEM file, and Sestra will calculate results for all basic load cases and load combinations.

Hence, use of smart load combinations reduces the number of result cases to be solved and then reduce the time spent by Sestra.

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.



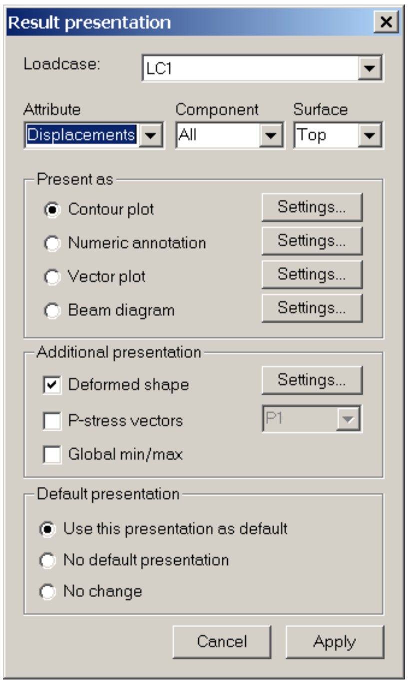
Note that Framework, Platework and Xtract, are also capable of handling load combination defined as smart load combinations. I.e. the computation of results for the combined loadcases takes place in Framework, Platework or Xtract based on results from basic loadcases computed in Sestra.

3.14 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.

3.14.1 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.



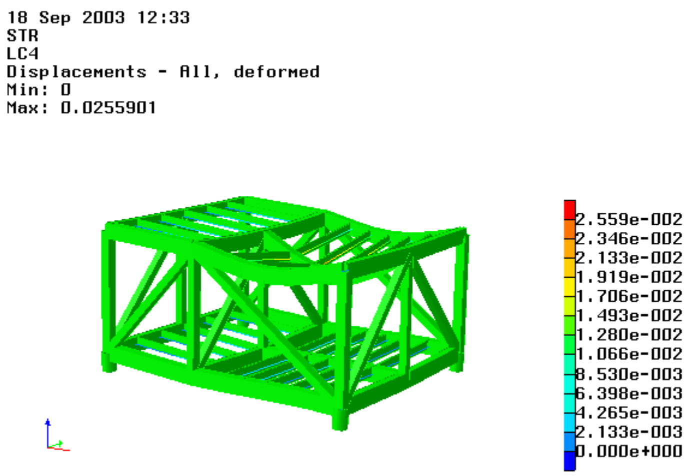
From this dialog you switch between result attributes, loadcase, and how to present. The presentation is according to the selected structure (typically individual beams or plates, set names), default is the complete model.

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.

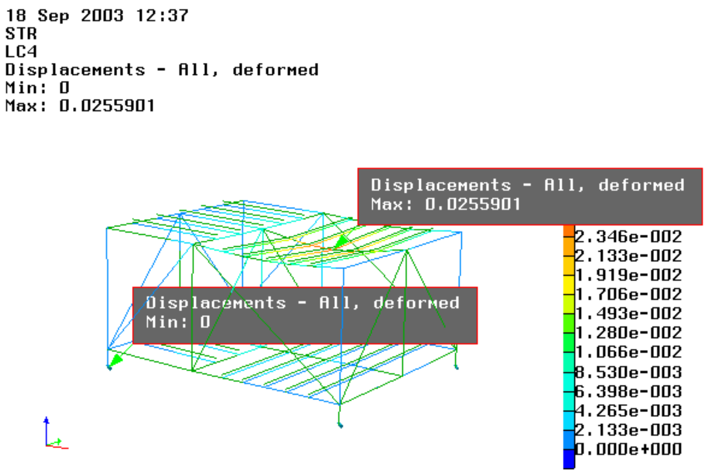
Common for them all is that a structural analysis has been run without errors (to create results) and that you have switched (in the combo box for predefined views) to a view that shows results (e.g. *Results – All* or *Results – with Mesh*).

3.14.2 Displacements



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.

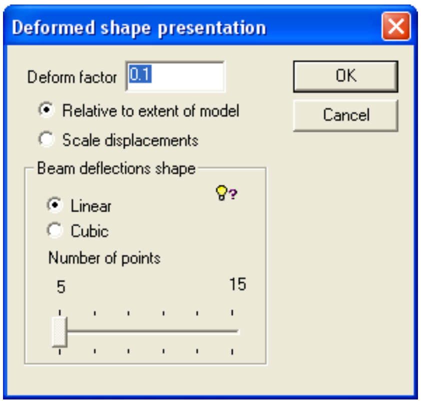


In this case, view has been switched to wireframe.

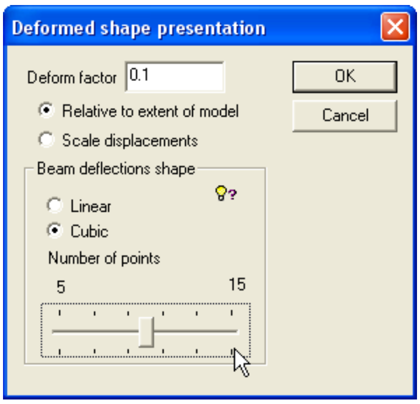
The displacements have been scaled with a factor of 50 and maximum and minimum values are presented.

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.



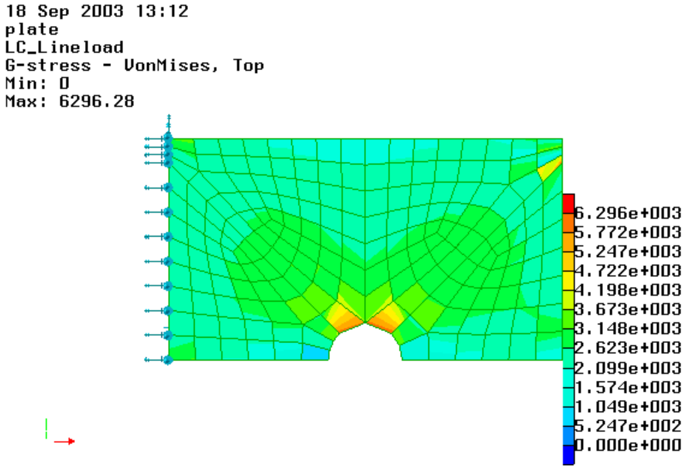
We have switched to the “Mesh – All” view. Under “Beam deflection shape” Linear is selected. The mesh consists of two elements along the beam. The member elements remain straight.



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.

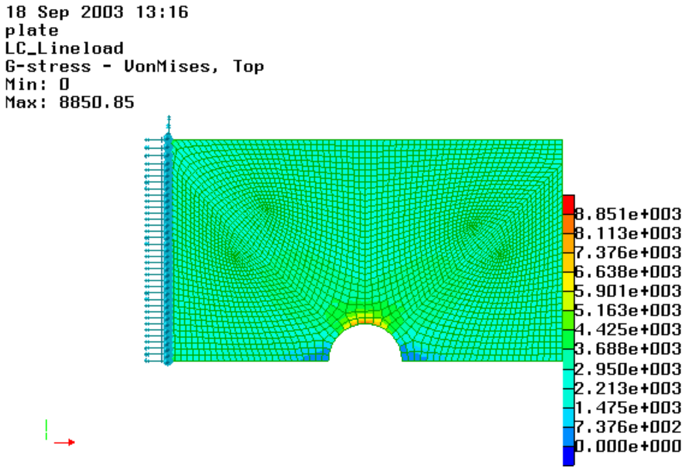
3.14.3 Element General Stresses

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 adjoing 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).

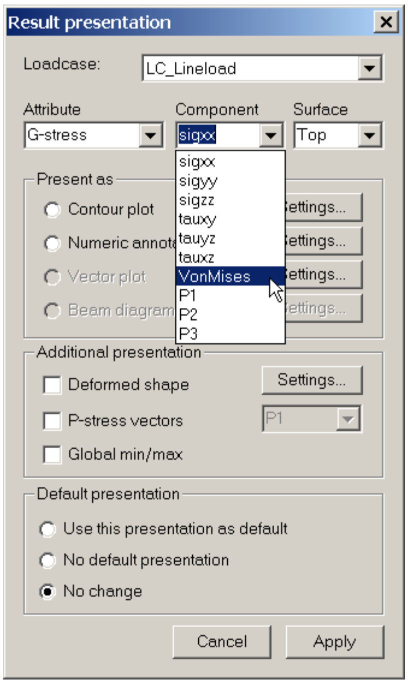
This picture shows that the finite element created is not adequate for representing the real stress pattern due to the forces applied.



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 1st order shell elements)

TAUXZ – shear stress in the direction of local x/z axes (irrelevant for 1st order shell elements)

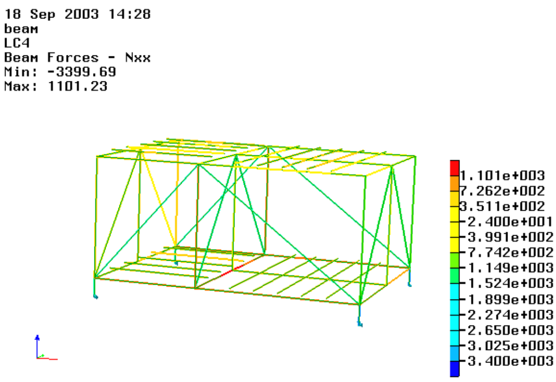
VONMISES – von Mises stress

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.

3.14.4 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.



Contour plot of axial beam forces are shown for the whole model.

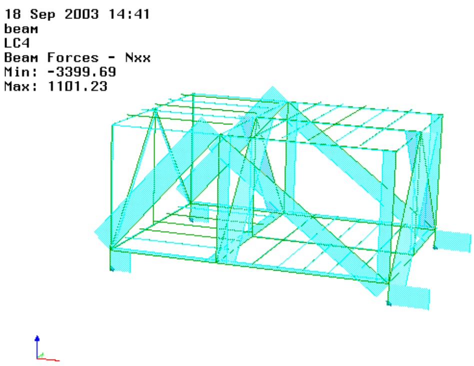
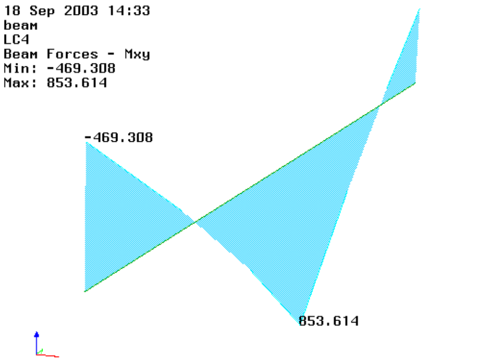
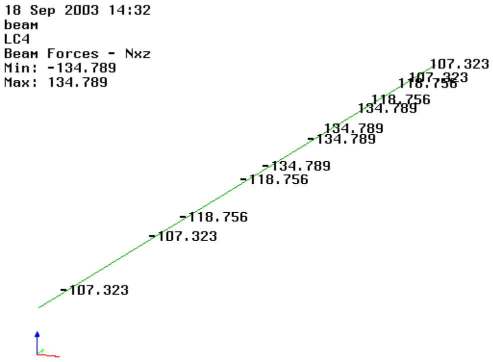


Diagram of axial forces are shown for the whole model

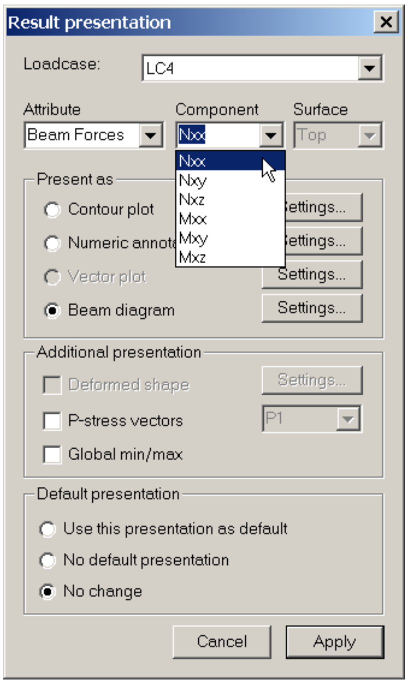


Bending moment diagram presented for one beam only.

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

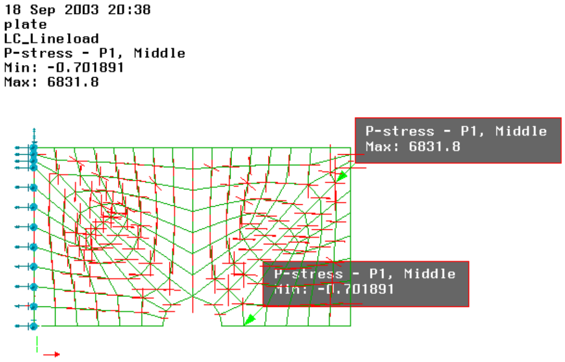
NXX - Axial force
NXY - shear force in the direction of the local y-axis
NXZ – shear force in the direction of the local z-axis

MXX – torsional moment
MXY - 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

3.14.5 Principal stresses

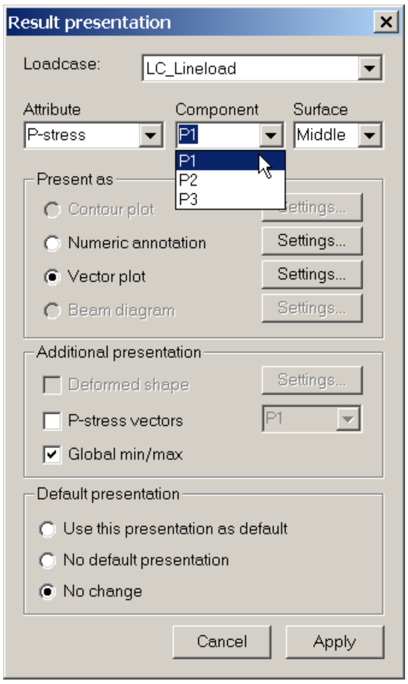
Principal stresses are shown as vector plots, and it also possible to include principal stresses together with presentation of general stresses.



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

3.14.6 Other results presentation

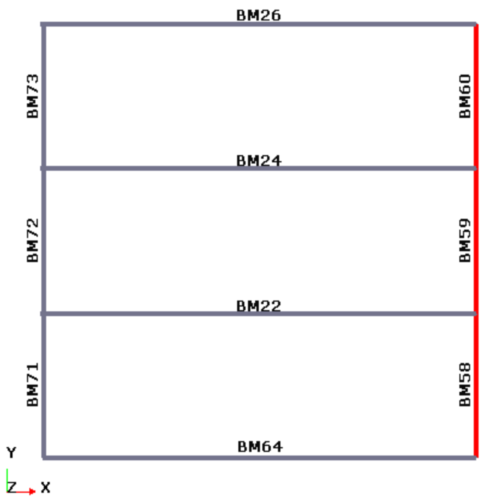
The model and results may be visualised in Xtract which is a general finite element postprocessor. Xtract has many features for presenting other result attributes and has a user interface that is similar to GeniE. When importing a model from GeniE into Xtract, please note that concept information is 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.

3.15 Run Code Checking using Framework

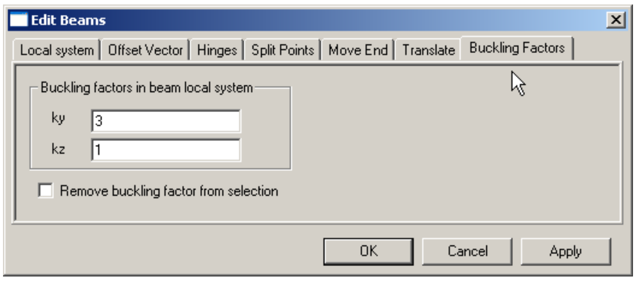
Code checking in GeniE is now normally done inside GeniE. However it is still possible to do code checking in the program Framework. The program may be started from GeniE. When model data is imported in Framework, all concept information is read in. Examples of concept data may be names of objects (beams, sections, materials, load case names, joint names, buckling length factors) and relationship between beam members and the finite elements.

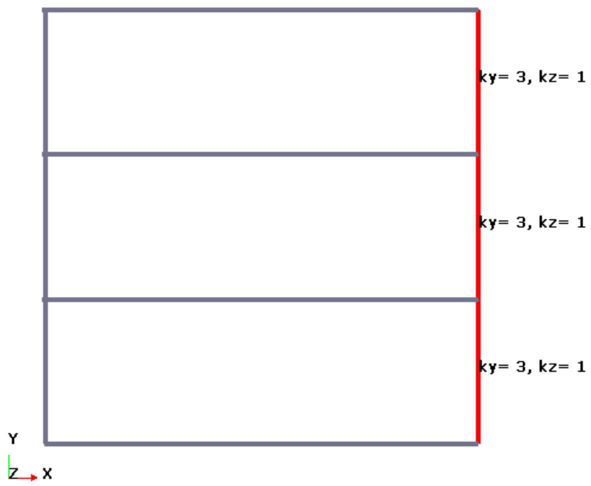
This means that when referring to an object the name in GeniE is used, or you may use the finite element numbering system if this is the preferred solution. Note that when creating a model in GeniE the number of characters in a name should not exceed eight (8) to avoid truncation of names.

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



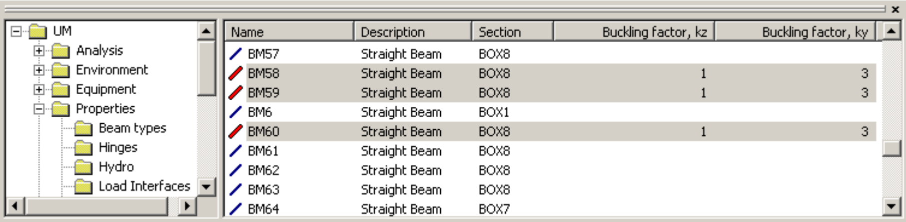


The buckling factors are defined and applied to the selected beams.

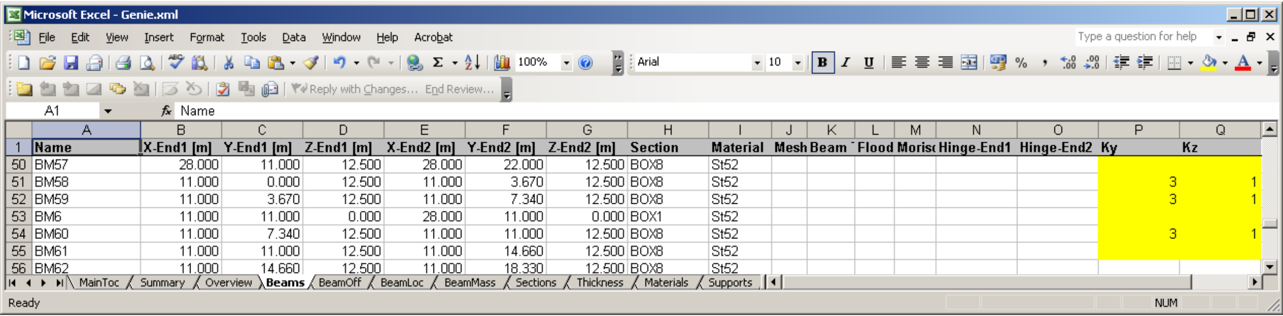
Note that if you split a beam or join beams, the assigned buckling factors will be set to default (k_y=-1 and K_z=-1) which means default buckling factors will be used by Framework).

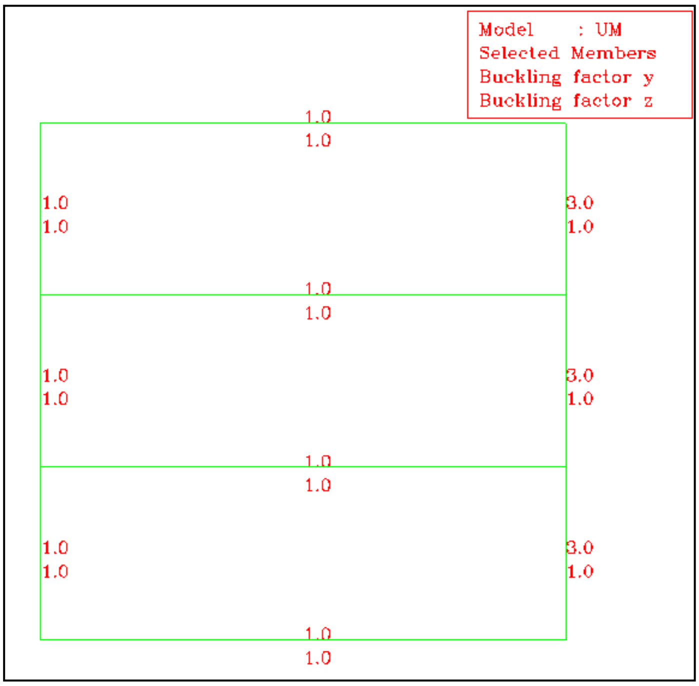
The default values are not shown when you label the buckling factors.

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.



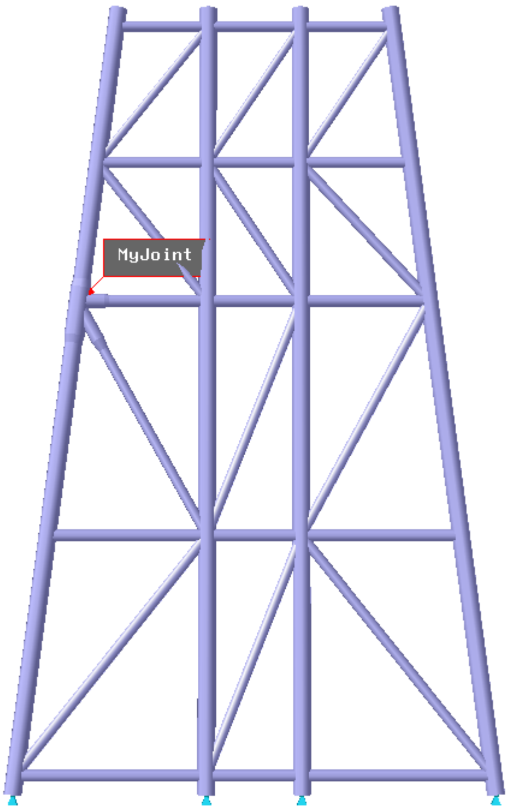
The fields kz, and ky are added to the browser view.





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.

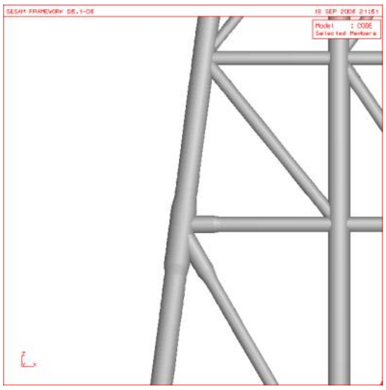


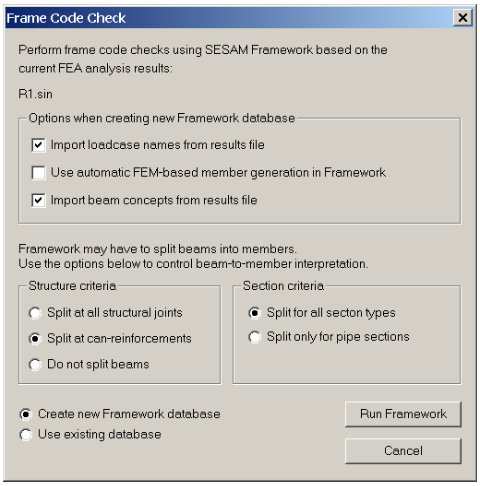
To show how a code checking model can be set up by GeniE, the jacket frame to the left is used.

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

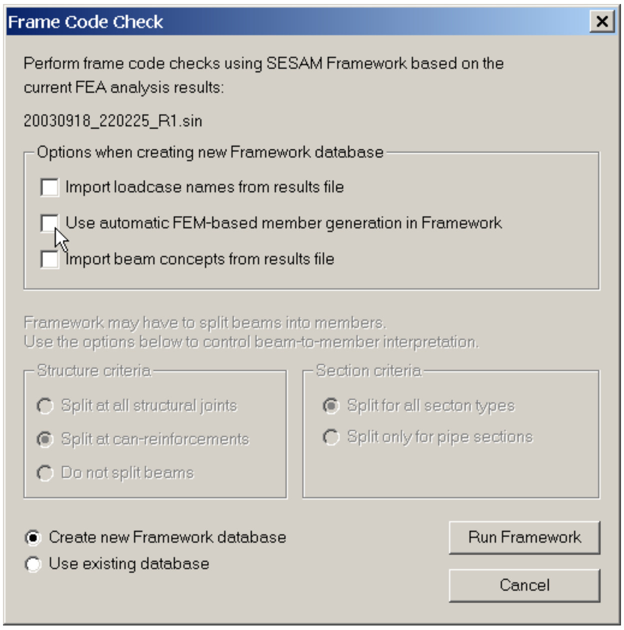




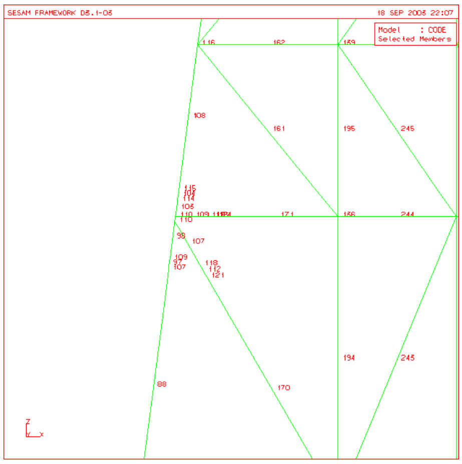
To set up the model for code checking, the dialog sheet at left is used (this is activated from ***Tools|Analysis|Frame Code Check***)

In the following, the options on how to split up the structure to create wanted member lengths are discussed.

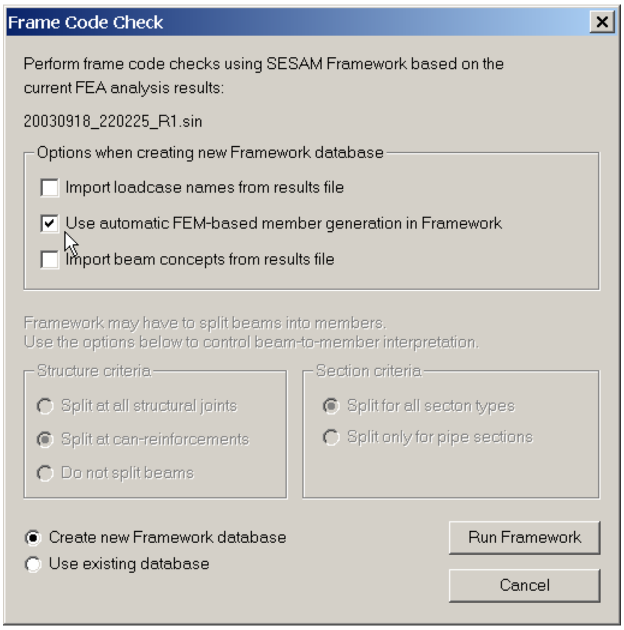
3.15.1.1 *Import the model using finite element numbering*



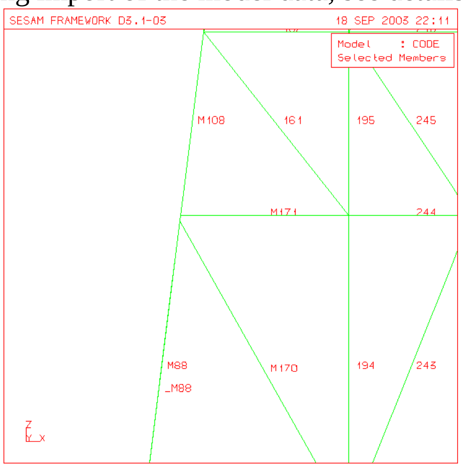
No concept information is exported to Framework. To do code checking all references must be made to finite element and node numbering. These numbers are identical in GeniE and Framework. Below is an example of the element numbers close to the structural joint Myjoint.



3.15.1.2 *Import the model using FE numbering and generate members in Framework*



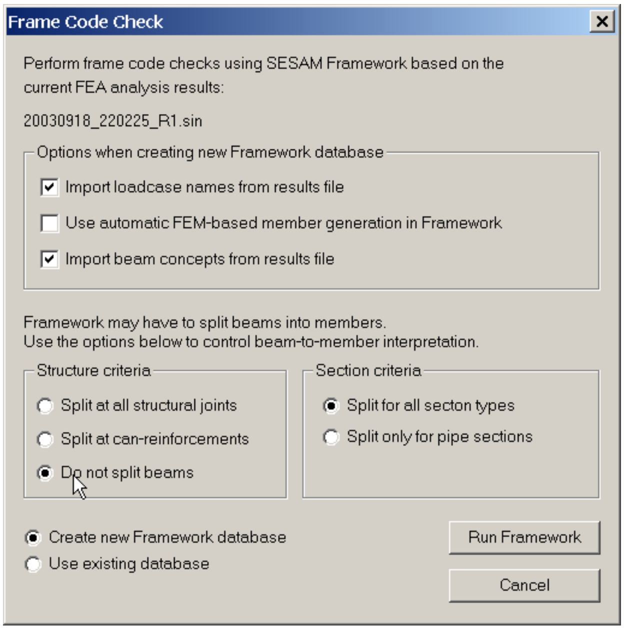
In Framework it is possible to automatically create members between two structural joints where each member may consist of many finite elements. By activating this feature, Framework will do this operation during import of the model data, see details close to Myjoint.



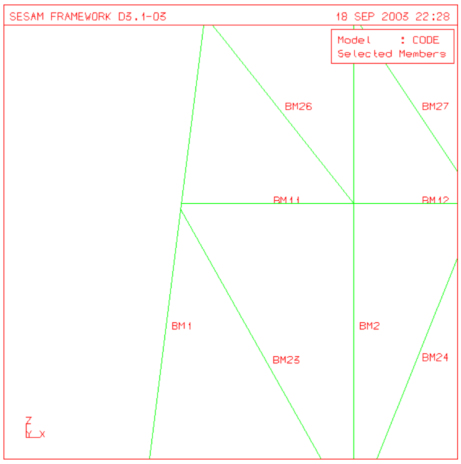
Both examples above imply that the user has knowledge about the finite element model, and as such not utilising the high level concept information already built into the model. By using the automatic member creation, the buckling length is always set equal to the member length unless manually specified 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.

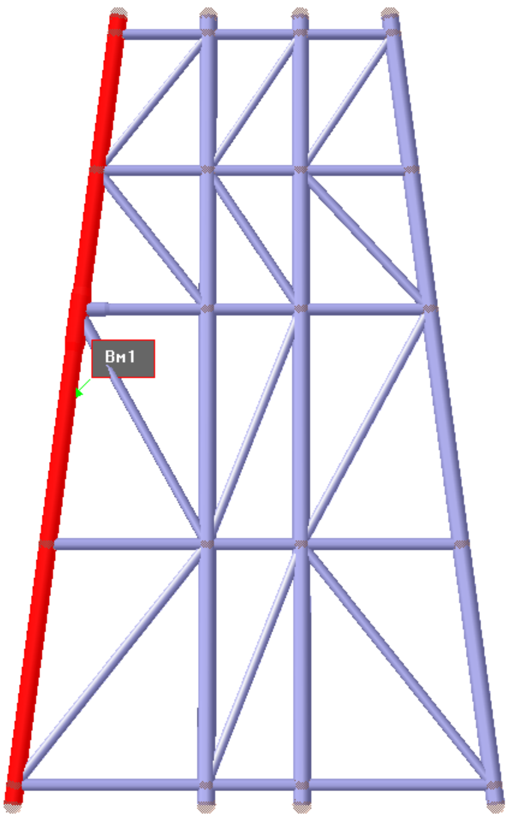
3.15.1.3 *Import the model using concept names, no split of structure*

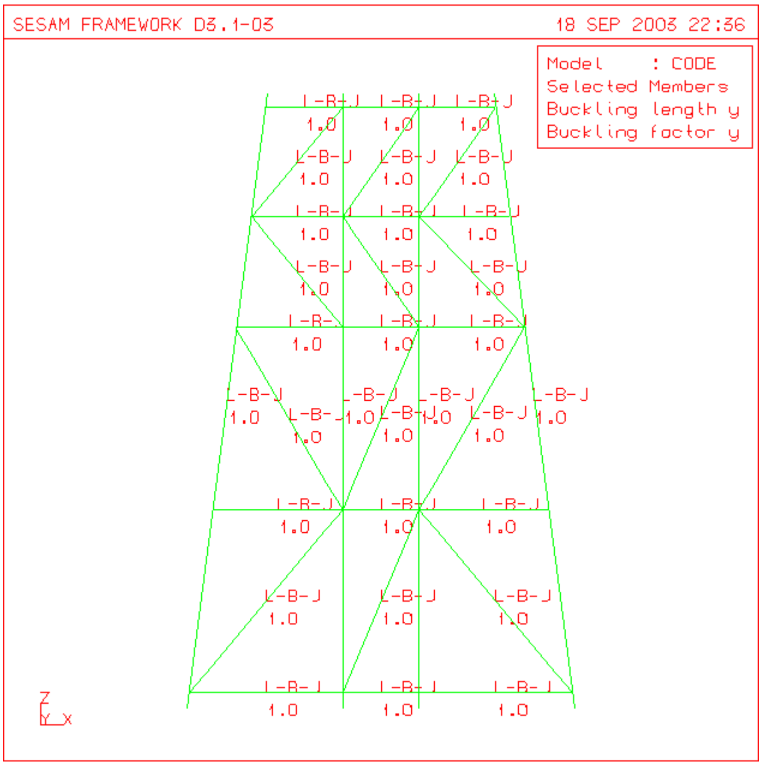


The concept information is now read by Framework. The names of the beams close to Myjoint are shown below, and the names are identical to those in GeniE. This means that if the finite element numbering for some reason change, you can still use the same names in Framework since the beams (or concepts) are independent of mesh density.



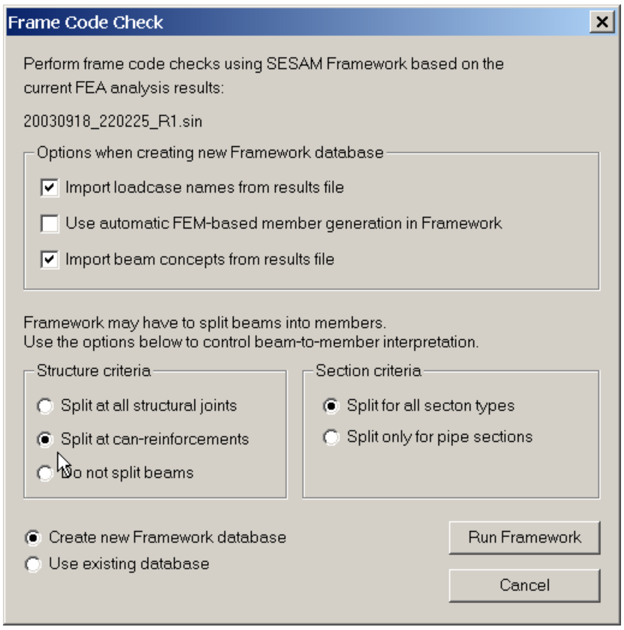
Since no split of structure is performed, the buckling length of Bm1 is identical to the member length. The pictures below show the extent of Bm1 and the buckling length of the same beam in Framework (buckling factor is 1.0, and buckling length is L-B-J (Length-Between-Joints)).





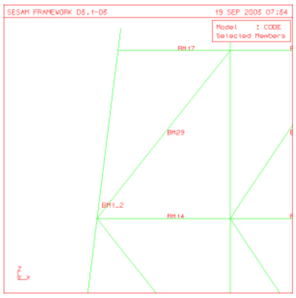
To override the buckling length factors, either specify manually or compute by using an eigenvalue approach (for tubular members only) in Framework.

3.15.1.4 *Import the concept model, split structure at reinforced structural joints*

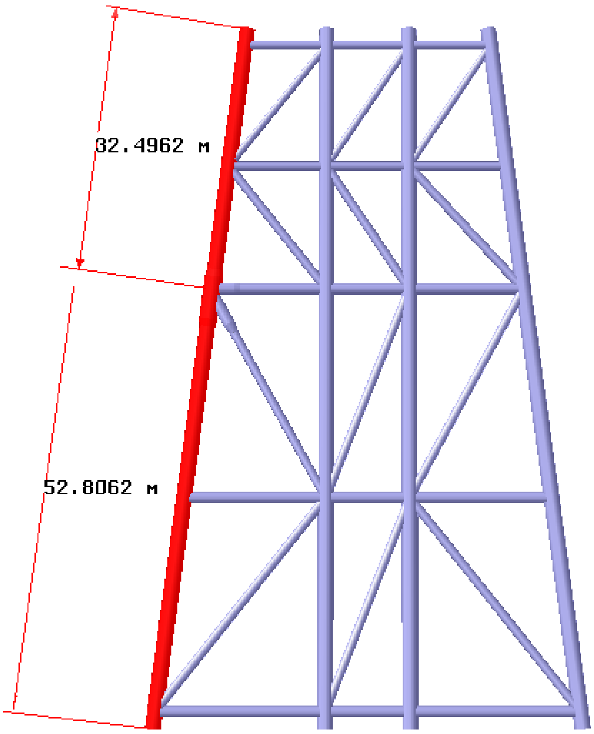


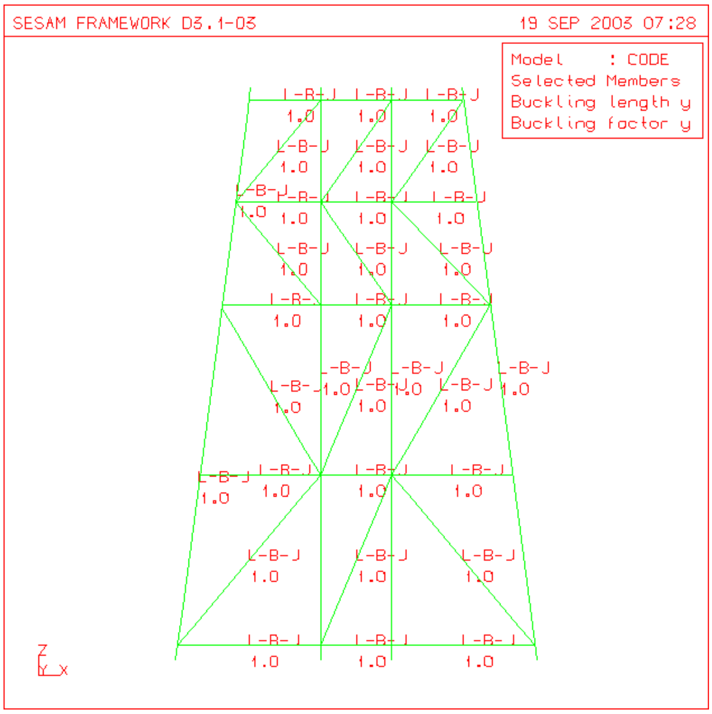
The split at can-reinforcements is the default option and will create new members in Framework. This means that you can model on a high level in GeniE and concentrate on the code checking model part of it afterwards.

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.

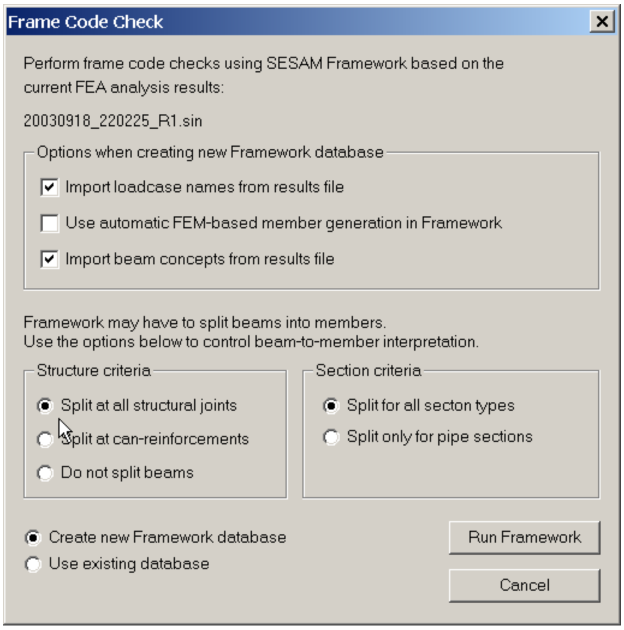


A split has been performed, and the default buckling length of the new Framework members Bm1_1 and Bm1_2 are equal to the length of the respective members. The pictures below show the extent of Bm1_1 and Bm1_2, in this case they are 52.8062 m and 32.4962 m.



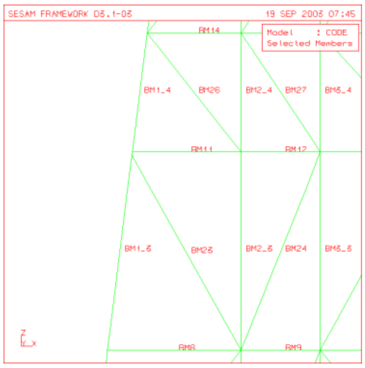


3.15.1.5 Import the concept model, split structure at all structural joints

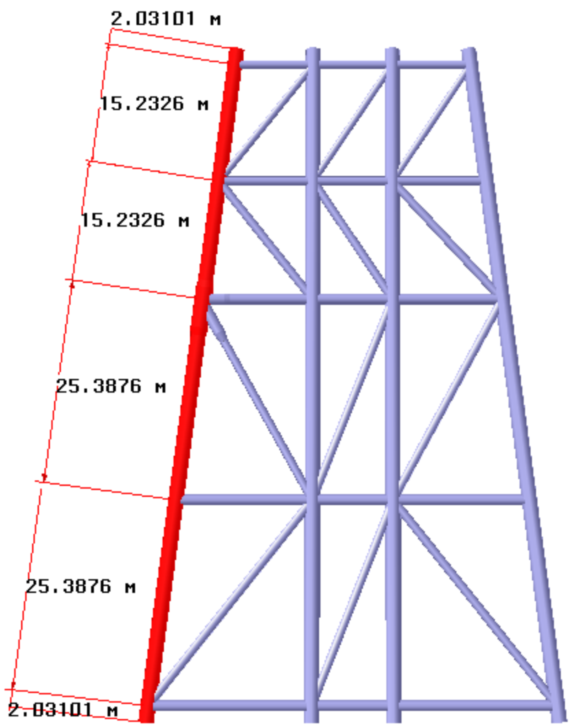


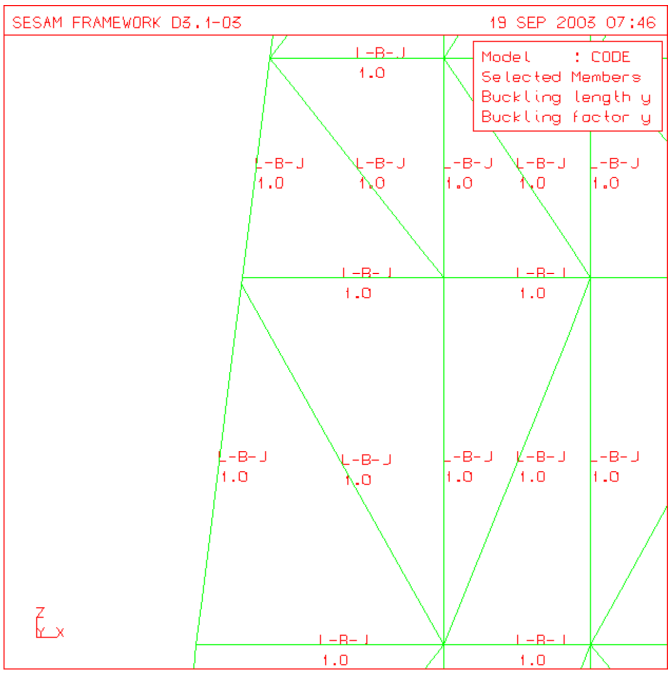
In this case the GeniE model is split at each structural joint. As such it is the same option as automatic FEM based member generation in Framework, the major difference is that concept information now is being used (beam names, sections, materials etc.).

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).



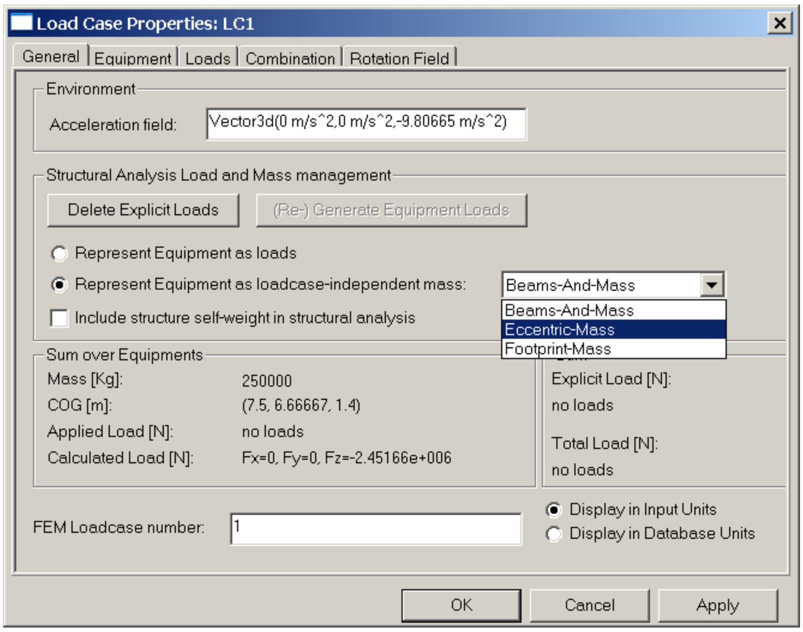


3.16 Create Mass Models

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.

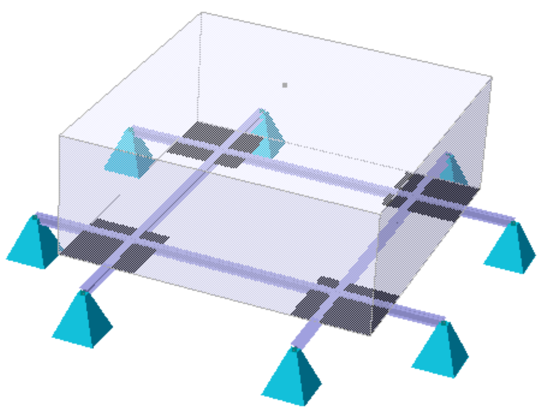


The same concept model may be used for both purposes (create forces or masses from equipments). You decide which option to use in the Analysis Property Sheet as show at left.

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

All three options are described in the following.



To illustrate the differences between the two first alternatives, the following model is used for references.

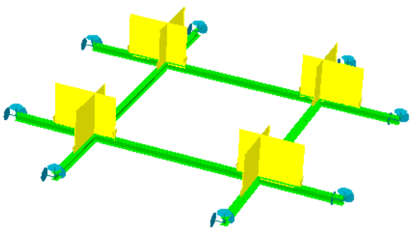
The model consists of four beams beams, one equipment with four footprints, and boundary conditions (free to rotate in all dof).

3.16.1 Mass model for hydrodynamics

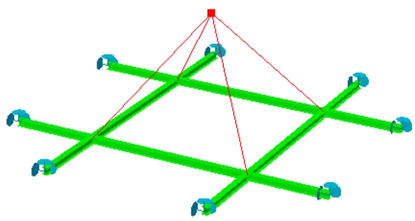
A common scenario when making a mass model for hydrodynamics is when the GeniE model contributes to the overall mass model of a floater. Typically, the topside is modelled in GeniE, the hull is modelled in Patran-Pre and the complete model is assembled in Presel. The complete mass model is now (together 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.

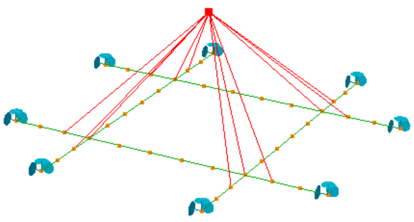
In the example below a mass model using the eccentric mass option has been used. Finite element of type GMAS (one node mass element) are inserted. The eccentric mass elements have their mass centre always at the same position as the equipment local cog. Whenever there is an interface between finite element nodes and footprints, eccentric one node mass elements are created (sum of masses of mass elements = equipment mass).



Line loads created when equipments are represented as loads.



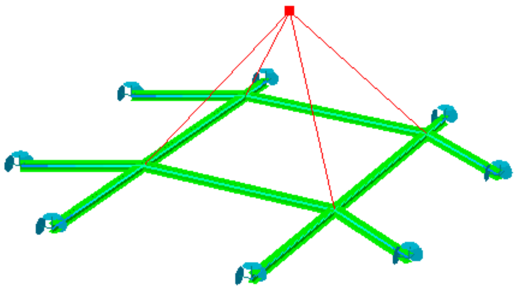
4 eccentric mass elements created when equipment adds to mass model. There are 4 FE nodes inside the footprints.

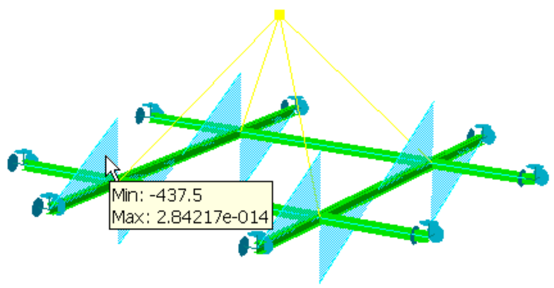


Mesh density increased and 12 FE nodes inside the footprints. Hence 12 eccentric mass elements are created.

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.

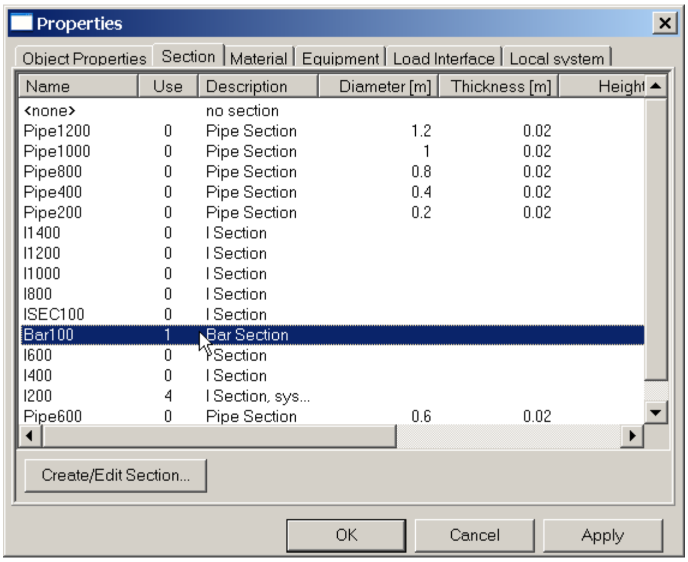
The pictures below show displacements and the undesired bending moments (to high peaks) due to a horizontal acceleration component.





3.16.2 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.



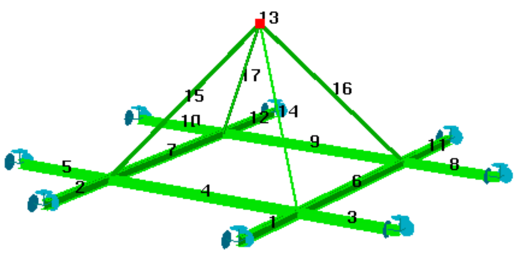
Section type Bar100 and a material type is assigned to the equipment from the equipment property sheet.

Normally, equivalent sections are used addressing no side effects when e.g. performing eigenvalue analysis.

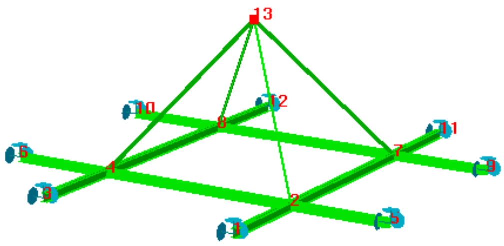
Also, it is common to use a material property with zero density (the equipment mass is represented from a point mass and not the structure associated with the equipment.

You may also verify the association between the equipment and the properties (section and material).

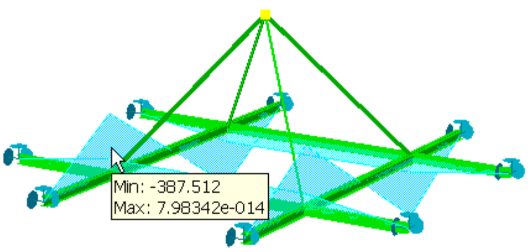
In the example below the feature Beams and Mass for representing equipments as loadcase independent mass has been used.



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.



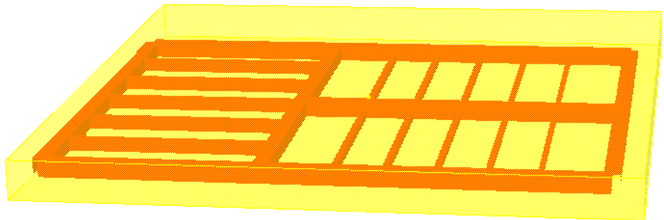
A better and more realistic moment distribution is now achieved. The peak of the bending moment has been reduced from 437.5 to 387.5 (or 12%) in this case. The differences in result depend highly on structure, equipment properties, and location of equipments.

3.16.3 Mass model when neglecting eccentricities

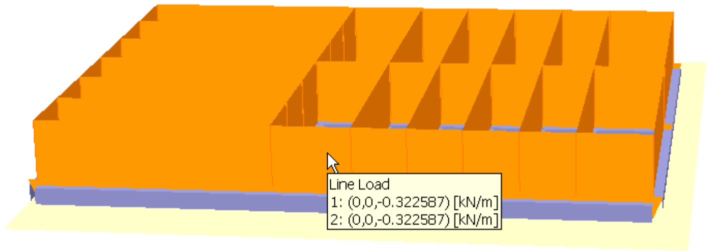
It is possible to create a mass model neglecting the eccentricities of the equipments centre of gravity. This may be the desired mass model when working with large equipments to form uniform blanket loads (UDL) or blanket loads.

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.

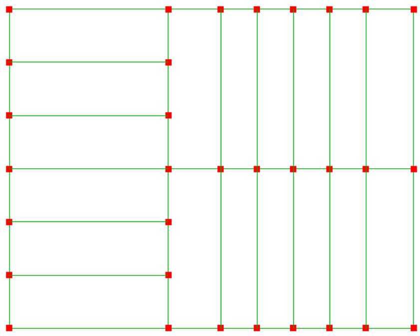
In then following is shown an example on the differences between UDL loadcase for a static linear and dynamic analysis.



The frame structure and the UDL (in this case the equipment is larger than the frame structure)

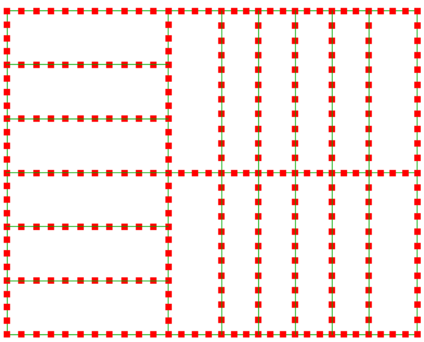


Analysis model for static analysis, constant line loads have been chosen.



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 equipments (i.e. at the footprint level) and are thus inserted at the supporting structure level.

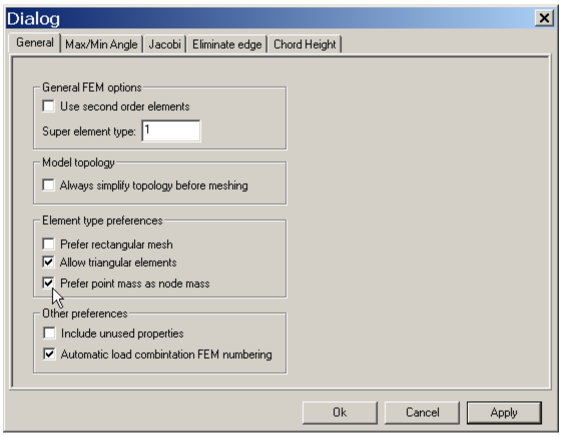
Default mesh density has been used, i.e. one finite element beam between each structural joint.



Mesh density has been refined; hence several more mass elements are automatically created.

3.16.4 Point mass versus node mass

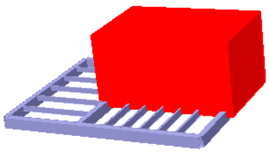
Both *PointMass* and *Equipment* are concepts which may be represented in different ways in the FEM model. These concepts may currently be represented as one noded mass element(s) with optional eccentricity, or as node mass(es) when requested and possible. The user can indicate preferred representation 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.



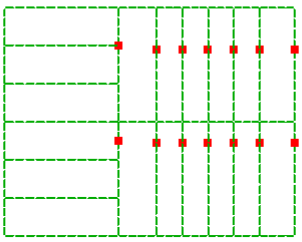
- The option ‘Prefer point mass as node mass’ is the default option, it indicates that the user prefers points masses and equipment masses to become node masses when possible.
- When deselecting ‘Prefer point mass as node mass’, points masses and equipment masses will be represented as one noded mass 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’.

3.16.5 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.

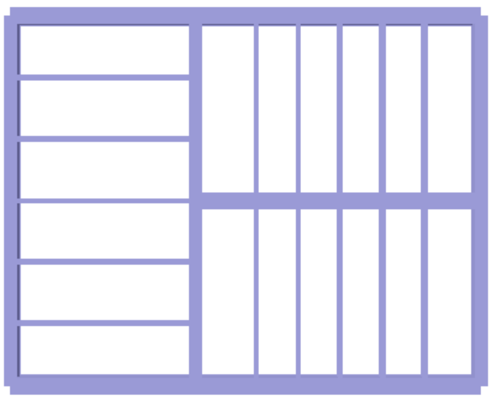
In order to overcome this limitation when making a model it is possible to scale structural masses to a given value. The principle behind such scaling is to:

- Make a named set
- Scale mass for the actual set

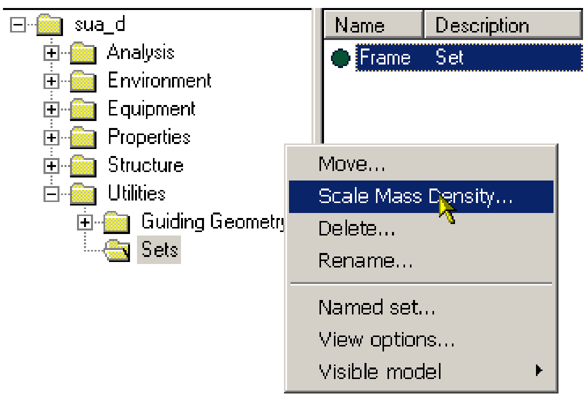
When scaling the masses, new material properties will be created. There are two options:

- 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.

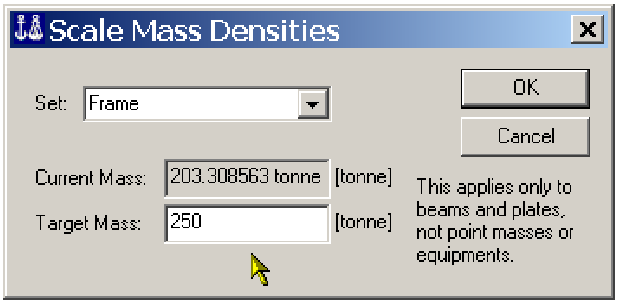
In the example below, a named set has been created and the mass is changed from 203.308563 tonne to 250 tonne.



A named set (*Frame*) has been created. The structural mass is 203.308563.

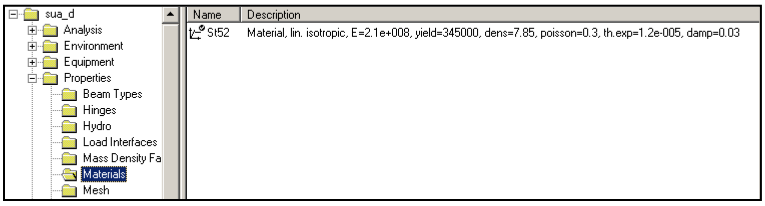


Select the named set you want to scale the mass for, click RMB and choose Scale Mass Density.

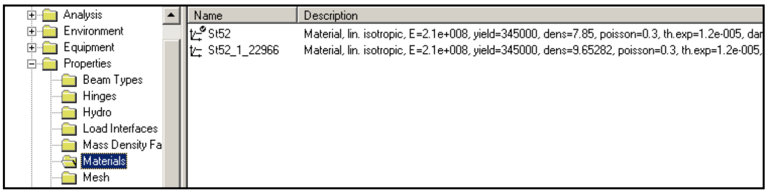


Specify the target mass and click OK. GeniE now ensures that the mass of named set *Frame* is now 250 tonne.

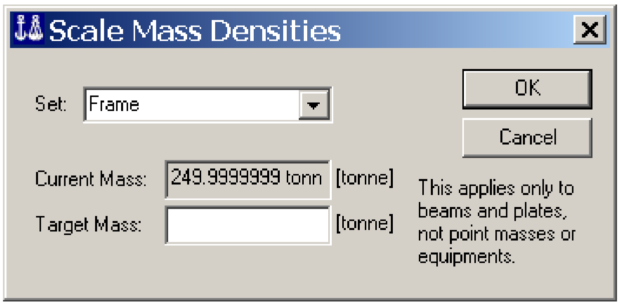
The scaling factor used by GeniE is 250/203.308563 = 1.229657995. You can also find this factor from the command language window.



A new material has been created on the FEM file and connected to the relevant structural members, but not in GeniE.



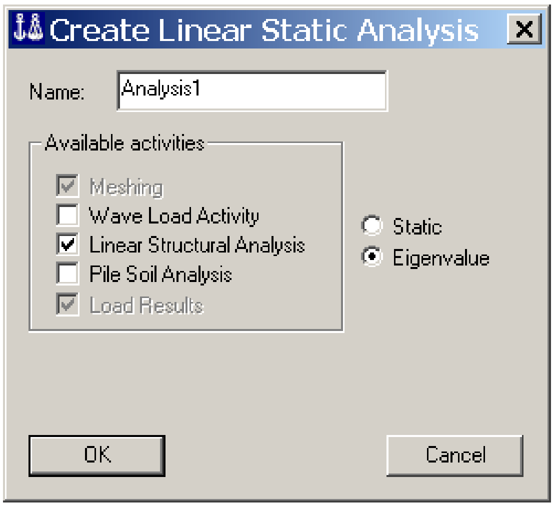
By using the command ***Tools|Properties|Create Scaled Materials*** a new material name is created and connected to the structural members in named set *Frame*.



The mass is now set to 250 tonne. You may change it again if you would like to do so.

3.17 Run Eigenvalue Analysis

Eigenvalue analysis started from the GeniE user interface is done from the menu ***Tools|Analysis|Activity Monitor*** (or Alt+D).

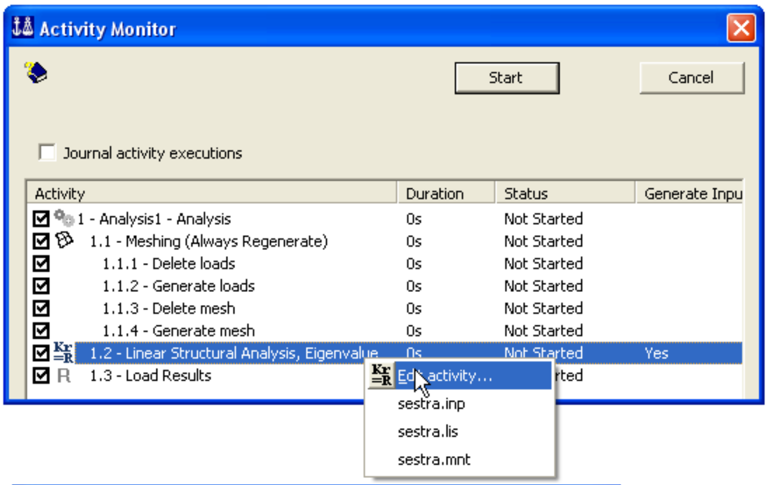


Activate the Eigenvalue option.

When analysis type is set to Eigenvalue, the 10 first eigenmodes using Lanczos method will be computed.

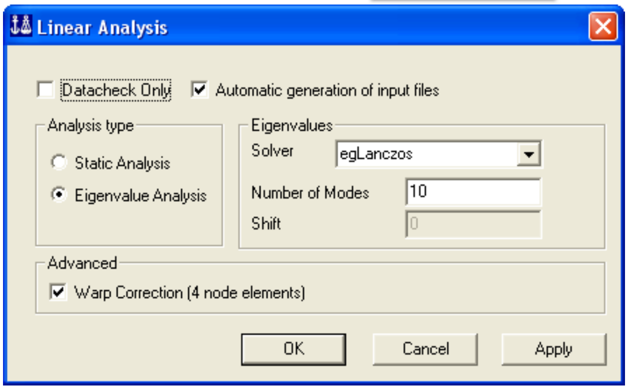
If you want masses from equipments included in an eigenvalue analysis, they must be placed in a static loadcase(s) and 'the represent equipment as loadcase independent mass' must be switched on.

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.



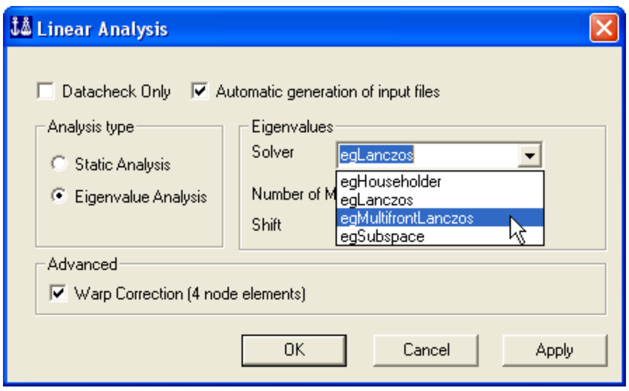
To change solver type and number of eigenmodes, select the *Linear Structural Analysis*, push **RMB** and choose *Edit activity*.

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.

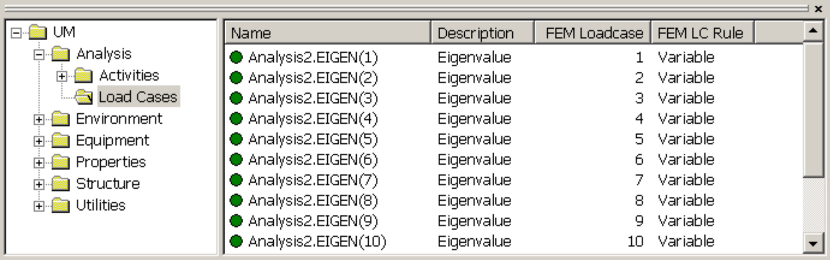


You may now change to e.g. another solver and run the eigenvalue analysis.

The solver MultifrontLanczos is selected.

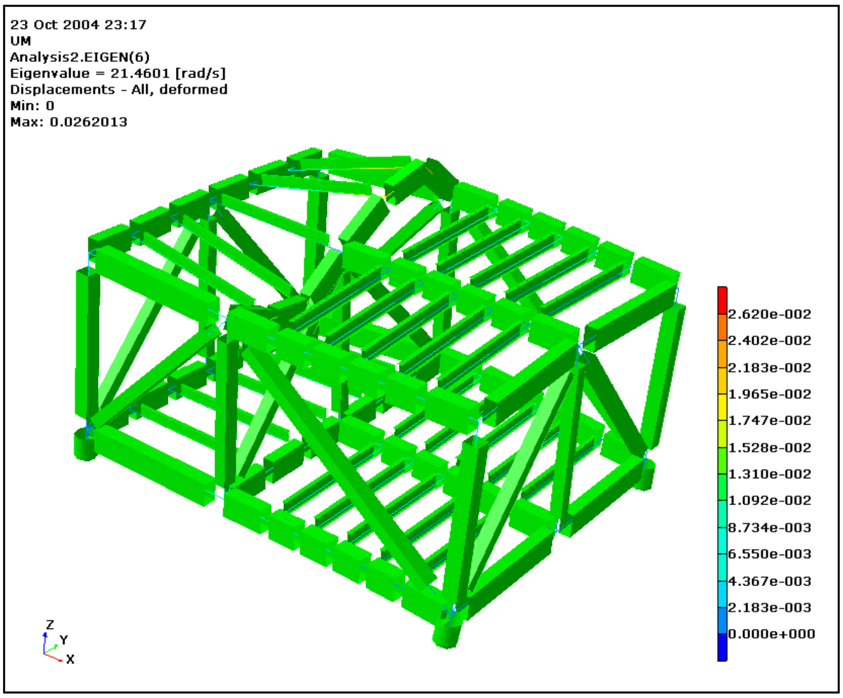


When analysis is performed you can investigate the results from the results view (***Tools|Analysis|Presentations*** or Alt+P).

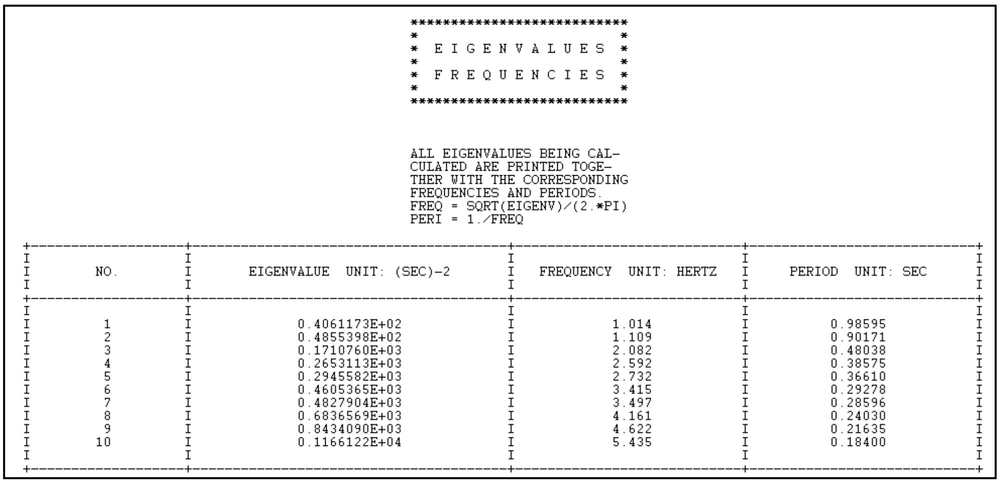


Name	Description	FEM Loadcase	FEM LC Rule
Analysis2.EIGEN(1)	Eigenvalue	1	Variable
Analysis2.EIGEN(2)	Eigenvalue	2	Variable
Analysis2.EIGEN(3)	Eigenvalue	3	Variable
Analysis2.EIGEN(4)	Eigenvalue	4	Variable
Analysis2.EIGEN(5)	Eigenvalue	5	Variable
Analysis2.EIGEN(6)	Eigenvalue	6	Variable
Analysis2.EIGEN(7)	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 6th 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 CAL-
CULATED ARE PRINTED TOGE-
THER WITH THE CORRESPONDING
FREQUENCIES AND PERIODS.
FREQ = SQRT(EIGENV)/(2.*PI)
PERI = 1./FREQ

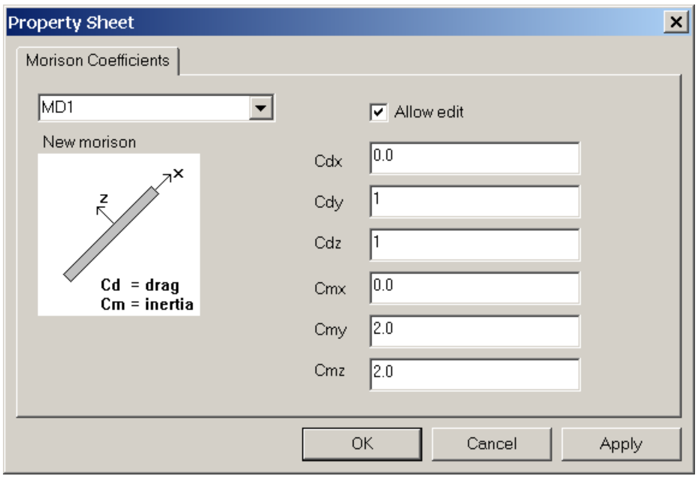
NO.	EIGENVALUE UNIT: (SEC)-2	FREQUENCY UNIT: HERTZ	PERIOD UNIT: SEC
1	0.4061173E+02	1.014	0.98595
2	0.4855398E+02	1.109	0.90171
3	0.1710760E+03	2.082	0.48038
4	0.2653113E+03	2.592	0.38575
5	0.2945582E+03	2.732	0.36610
6	0.4605365E+03	3.415	0.29278
7	0.4827904E+03	3.497	0.28596
8	0.6836569E+03	4.161	0.24030
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.

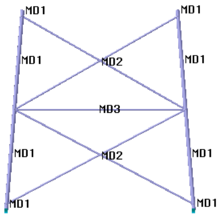
3.18 Run Wave Load Analysis

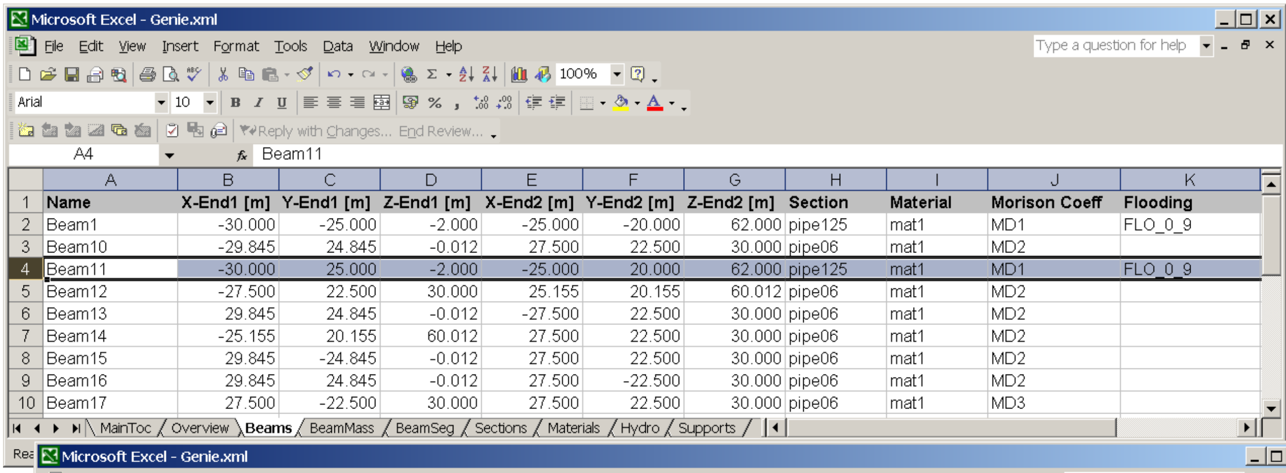
To run wave load analysis a number of wave data needs to be defined. Some of these belong to the concept structure model (Morison coefficients, non-structural elements, and flooding), while the remaining must be defined in the Wajac input file.

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***.

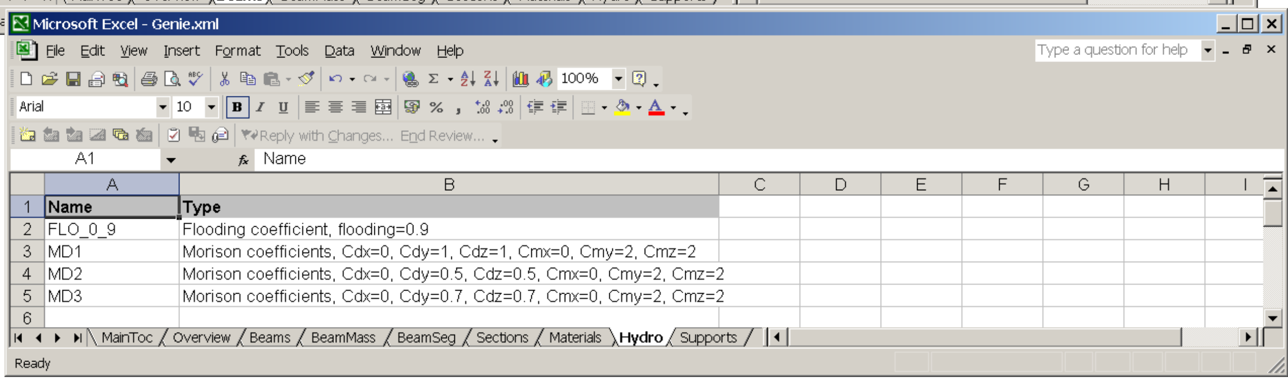


A Morison property MD1 is defined with drag coefficients 1.0 and added mass coefficients 2.0. To verify where hydrodynamic properties have been assigned, either graphic or tabular verification can be done. The picture below shows that MD1, MD2, and MD3 have been assigned to different beams, You can do colour coding also.





	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	



	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

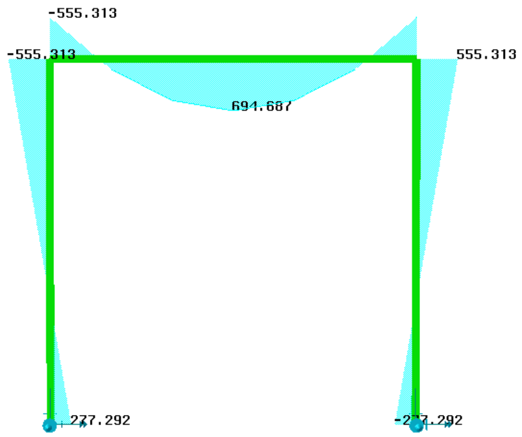
The hydro properties may be changed and edited from the property sheet. You may also find these properties from the browser area under Properties Hydro.

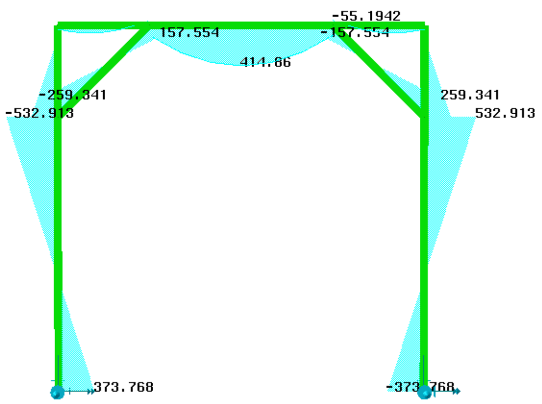
For further details on how to run hydrodynamic analysis, as well as pile soil analysis, please consult Vol. 2 of this manual.

3.19 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).





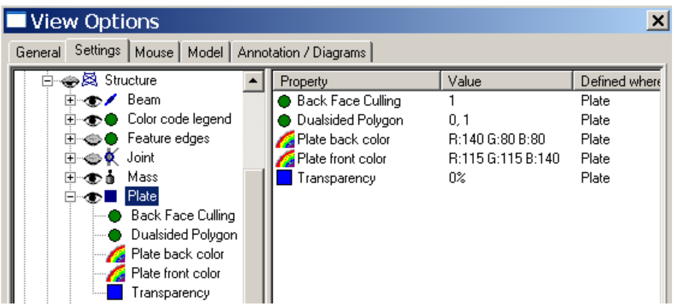
3.20 Panel modelling

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.

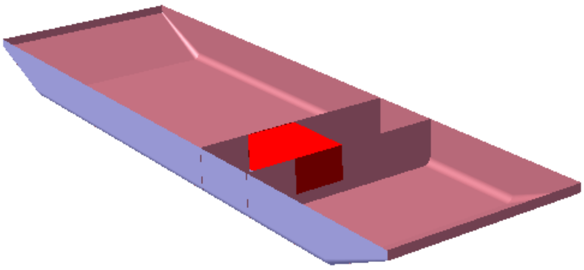
The principles behind panel modelling are as follows:

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.

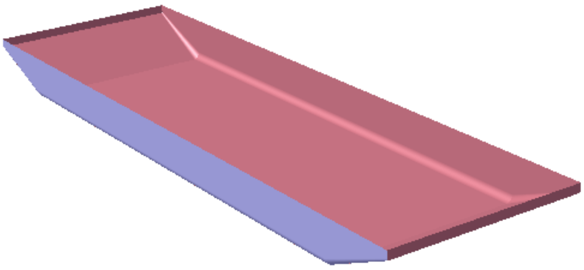


Please note that you can customize the colour appearance on front and back side of the plate from ***View|Options|Settings***.



Step 1

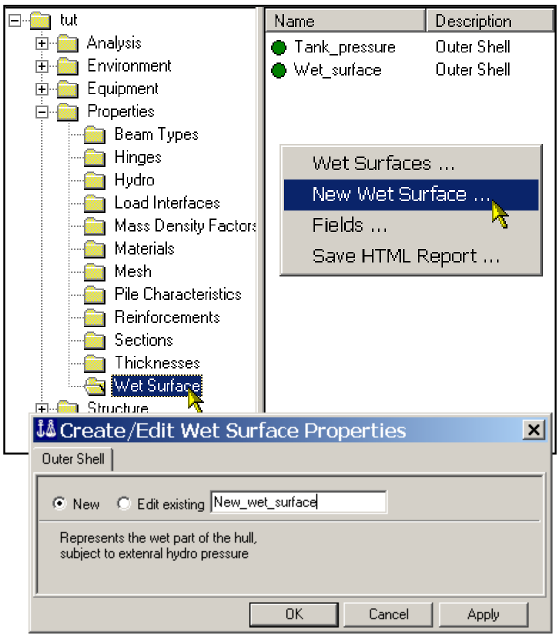
A small barge including one internal tank has been created.



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.

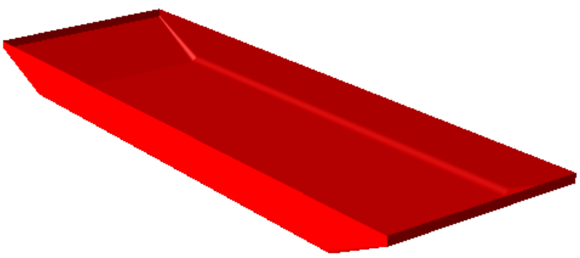


Step 3

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.

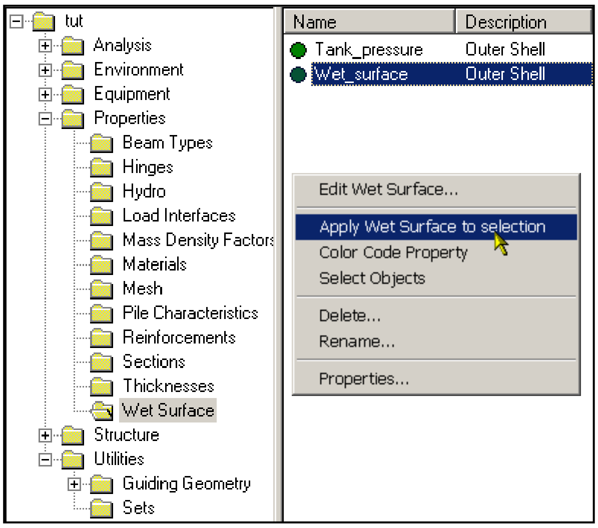
A wet surface property is defined as shown to the left or from ***Edit|Properties|Wet Surface***.

In this case two properties have been defined, one for the hull and one for the internal tank.

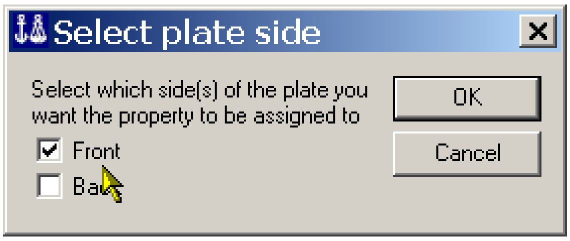


Step 4.

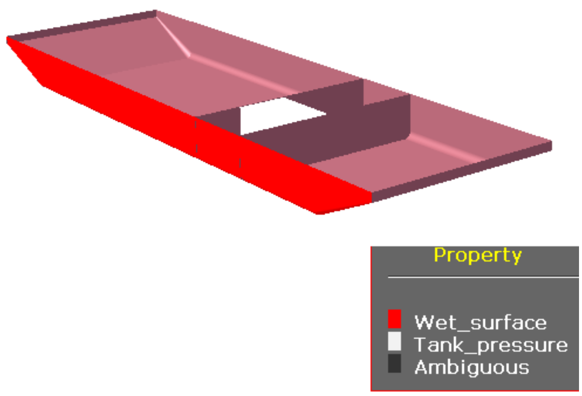
Assign the wet surface property to the outer hull. Select the outer hull from named set (in browser under Utilities, select and show visible model only). Open the folder Wet Surface, select the Wet_surface and apply to selected members only.



The property Wet_surface has been selected and may now be applied to the selected members, in this case the named set *Hull*.



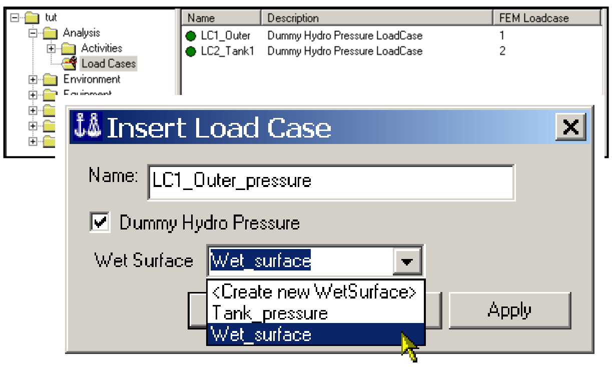
A dialogue will now ask you for which side to apply the wetted surface. In this case it has been applied to the Front side, i.e. the one in direction of positive local z-axis for the plate.



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.

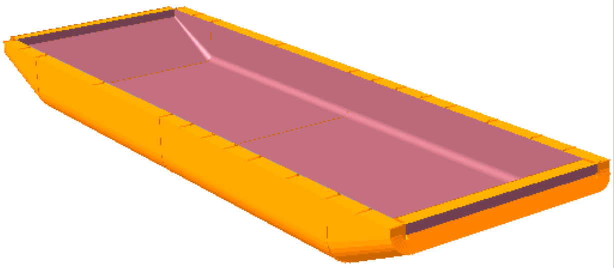
The tank pressure has been applied in the same manner as described in *Step 4* above.



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.

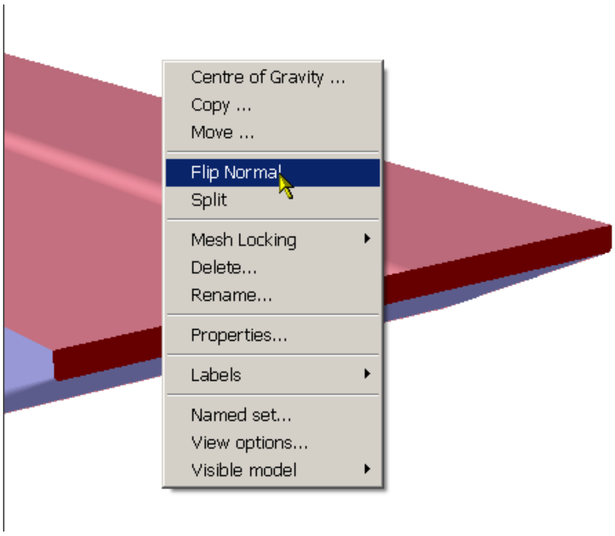
The figure to the left shows that the loadcase LC1_Outer_Pressure applies for the hull, while the LC2_Tank1 applies for the internal tank.



Step 7

Yoy 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.

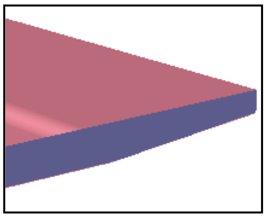


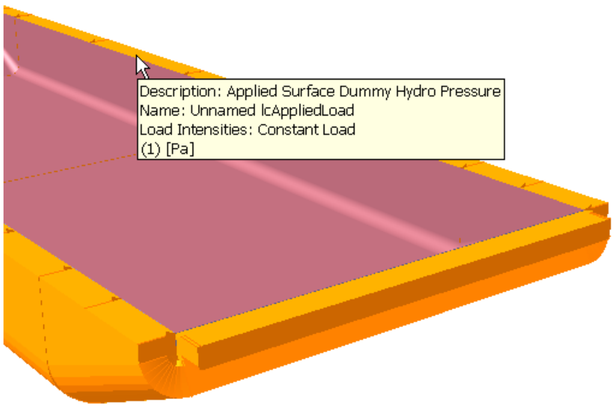
Step 8

The plate normal may now easily be changed by selecting the plate, RMB and Flip Normal.

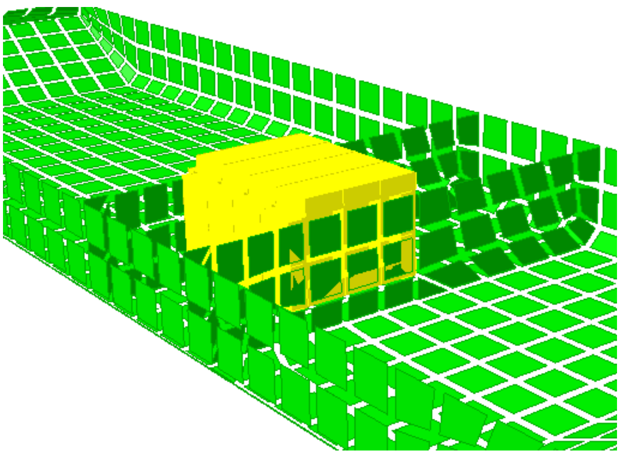
Control of a plate normal may be done at any time; when you change it that direction of the pressure is changed accordingly.

The end plate has now changed orientation as shown to the right.





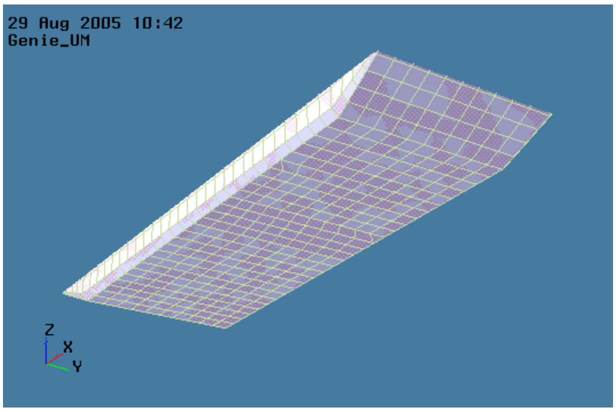
When displaying the loadcase, the pressures are now acting in the intended direction.



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***).

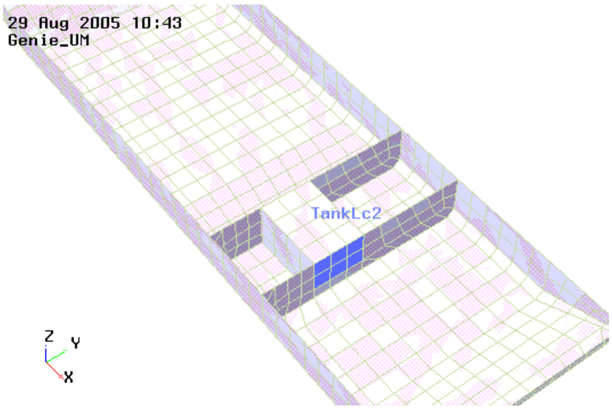
The figure to the left shows the tank pressure applied to the finite element model (loadcase *LC2_Tank1*).



The model has now been imported to HydroD for subsequent hydrodynamic or stability analysis.

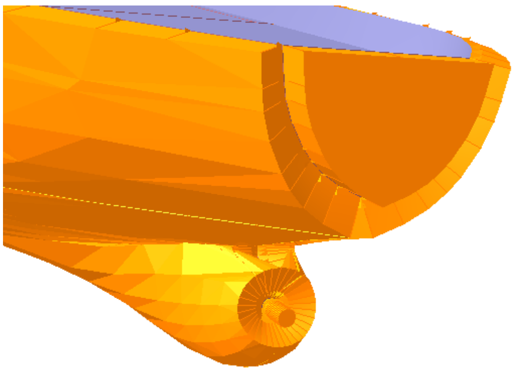
The figure to the left shows the panel model seen from below.

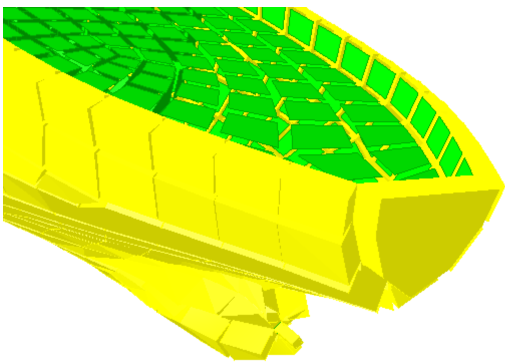
Note, this figure has been created by HydroD.



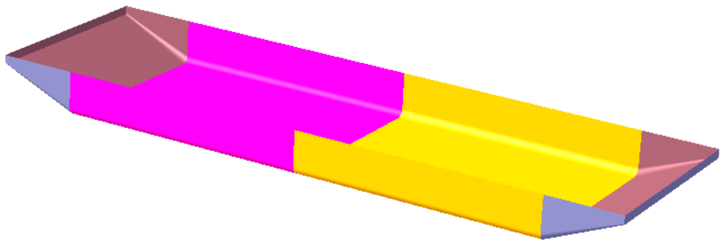
Similarly, the tanks may be visualised in HydroD as shown on this picture.

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.



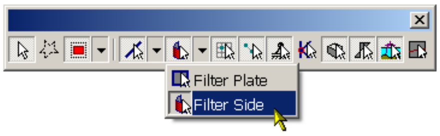


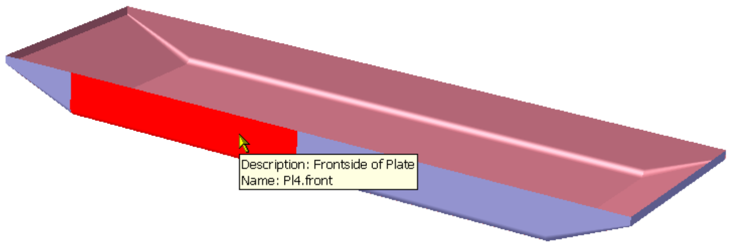
You may also apply wet surface by using a feature for selecting only one side of a plate. The example below illustrates how this feature works.



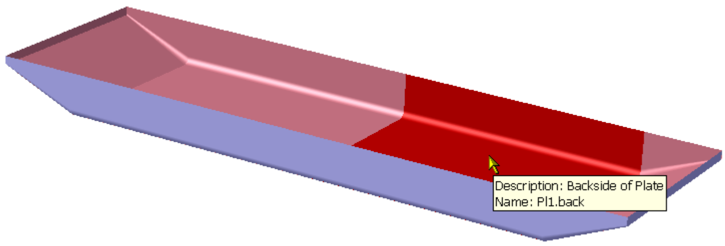
The barge has two large plate concepts in the middle part.

By using the filter side it is now possible to select one side only of a plate.

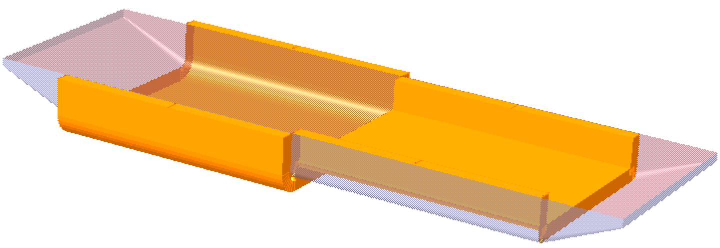




The outer part (front side) of the plate is selected and wetted surface can be applied.



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.

4. EXECUTION OF GENIE

This Chapter described how to start the program, which files are used by the program, and any high level limitations.

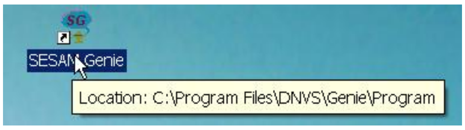
4.1 Program environment

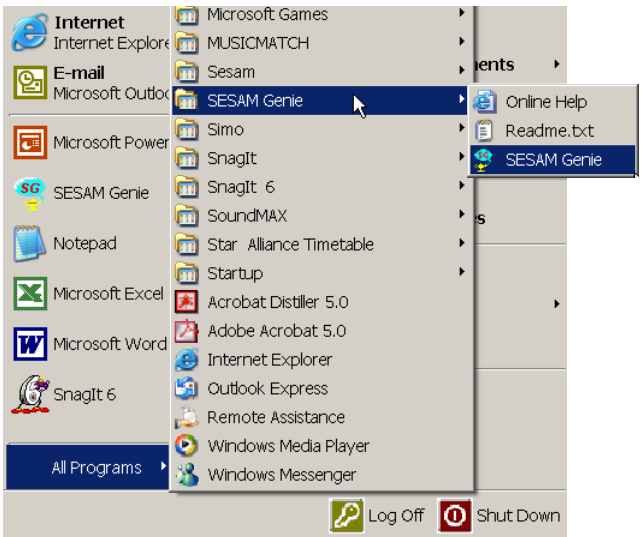
GeniE runs in a Windows environment, be it Win XP, Vista or Win 7.

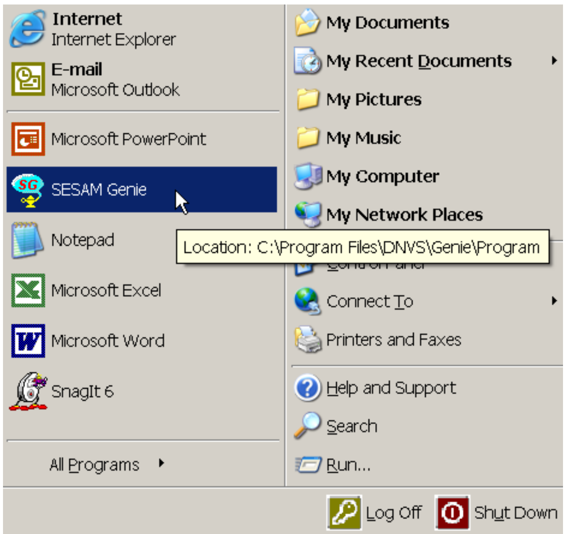
From V6.0 GeniE is available both as a 32 bit and a 64 bit application.

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.

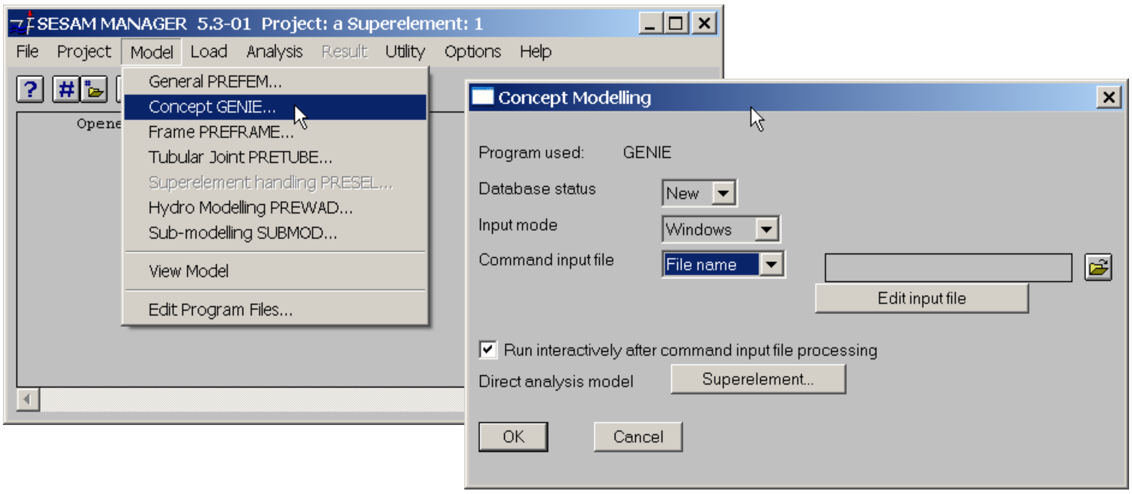






By changing the "start in" location in icon property dialog, you can select where workspaces will be created.

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:

```
Command Prompt
C:\>cd program files

C:\Program Files>cd dnvs

C:\Program Files\DNVS>cd genie

C:\Program Files\DNVS\Genie>cd program

C:\Program Files\DNVS\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:

```
Command Line options

Command-line arguments            Commands executed

genie                             Start program without database.
genie filename                    Open existing database.
genie filename /new               Open new database.
          /com=[path]input.js     Run a command input file.
          /exit                   Exit after running command input file.
          /? or /help             Show this help dialog.

Examples:
  genie topside /com=top_in               Open existing 'topside.gni' database, run 'top_in.js' file.
  genie topside /com=top_in.js /exit      Open 'topside.gni', run 'top_in.js' file, exit immediately.
  genie top2 /new /com=input.js /exit     Open new 'top2.gni' database, run 'input.js' file, exit.

Observe that
  - command input files may include an absolute or relative path.
  - arguments may be entered in any order.

OK
```

This sheet provides in formation 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:\Workspaces*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 www.dnvsoftware.com -> select support and then Status List. The status lists are updated every week-day and comes with a free text search tool.

4.4 Program extensions

GeniE comes in two variants:

1. **GeniE** together with extensions. This is a stand alone 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.

4.4.1 More about the GeniE Lite licence

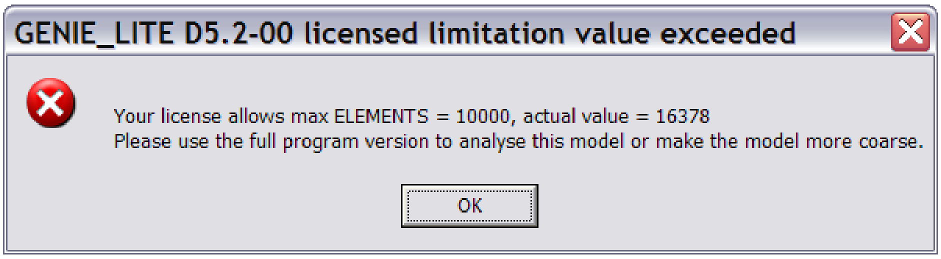
With a GeniE Lite license you can use GeniE in exactly the same way as for the full license with a few limitations:

1. Your mesh cannot contain more than 10.000 FEM elements
2. Your model cannot contain more than 500 beam concepts
3. Plate capacity checks are not supported.
4. 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:



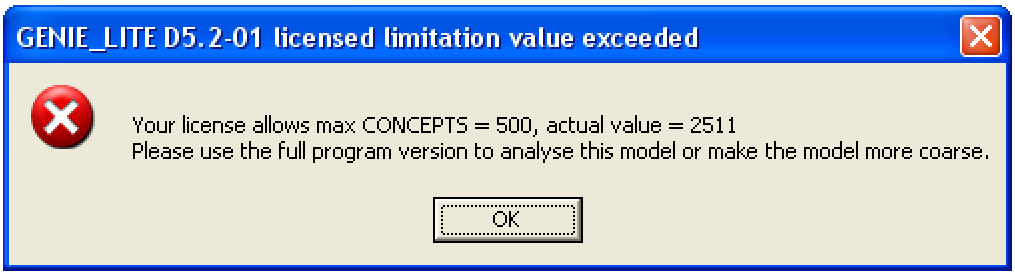
If you get this message you could try to make your mesh coarser in order to reduce the number of FEM elements.

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.

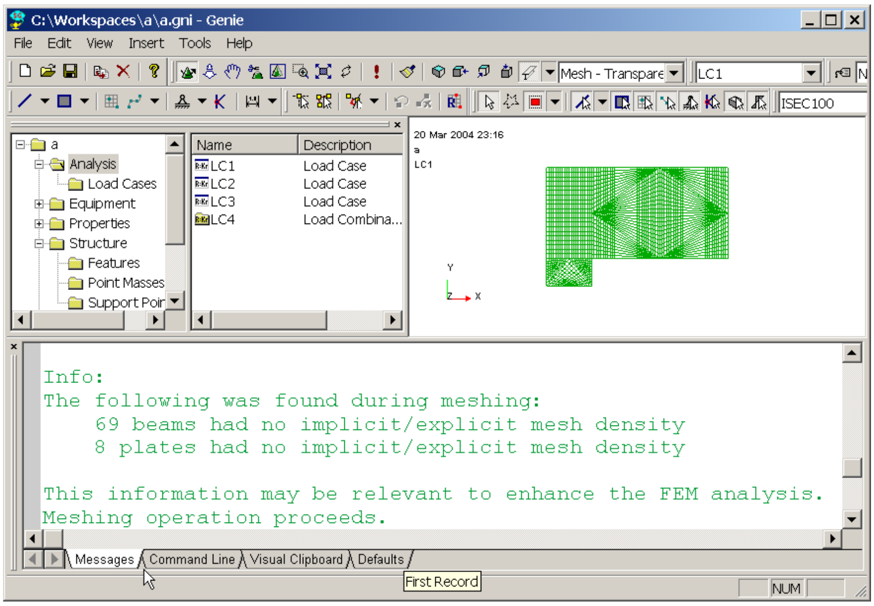
If you try to run the analysis with more than 500 beams, the following message will appear:



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.

4.5 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 ***File|Save report*** 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.

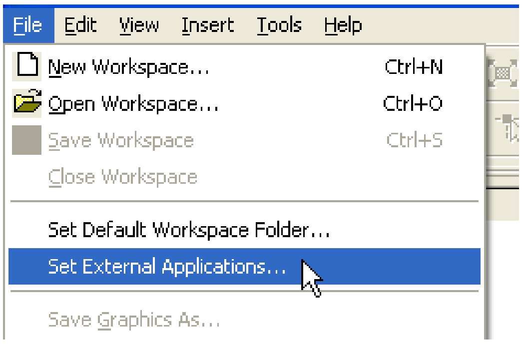
4.7 Database version checking

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@dnv.com on advice how to use the “nover” feature.

4.8 Setting the path to external applications



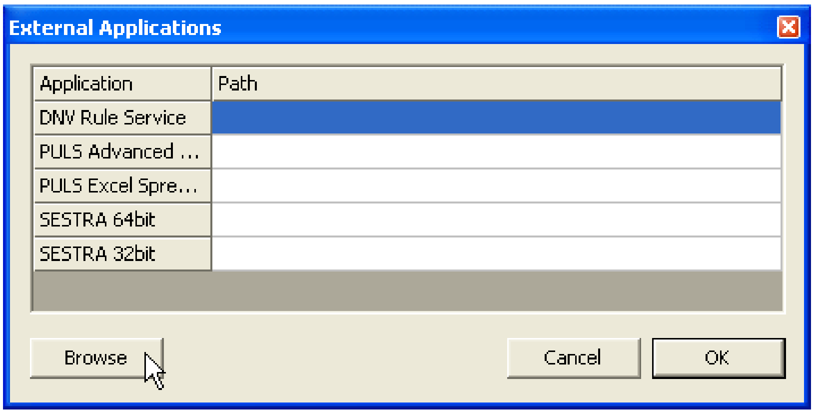
GeniE uses several external applications. The path to these applications is normally set automatically when you install GeniE.

In some cases you may want to manually change the path to an external application. It could be that GeniE did not succeed in finding the correct path while installing or that you want to try another version of the external applications etc.

To change the path to an external application, you select ***File|Set External Applications***.

The External Application dialog appears and you can change the path to the following external applications:

- DNV Rule Service
- PULS Advanced Viewer
- PULS Excel Spreadsheet
- SESTRA 64bit
- SESTRA 32bit



Select a column, press the Browse button – go to the folder where the appropriate executable exists – select it and press Ok.

DNV Rule service

- when DNV Rule service was **not** installed as part of the Genie Installation.

PULS Advanced viewer and PULS Excel

- when Nauticus Hull or the Buckling Package is installed to default location, but these tools fail to start.

SESTRA 64 bit

- when Sestra 64 bit is installed on your computer (you are running on a 64 bit OS), you **must** set the program location here.

SESTRA 32 bit

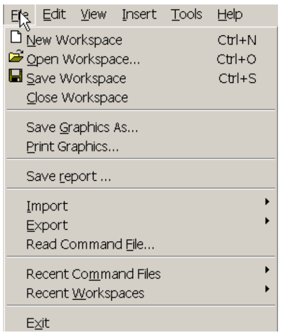
- when Sestra 32 bit is installed (you are running on a 32 or 64 bit OS), but you do not have only Sestra installed. If you are running GeniE.lite, you will normally not need the full Sesam Products installation, and you **must** set the Sestra location here. Further, if C:\Windows\Sesam.ini does not exist, or points to a wrong Sestra installation, you may overide the Sestra location here.

5. SHORT DESCRIPTION OF COMMANDS

5.1 Menus

5.1.1 Pull-down menus

5.1.1.1 The File pulldown menu



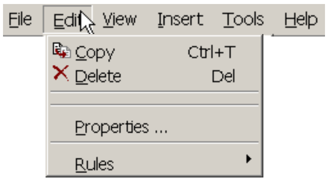
There are six main pull-down menu's, they are described in the following.

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

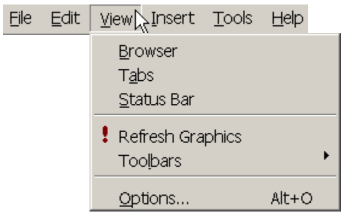
MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
File	**New workspace**		Make a new project
	Open workspace		Open up existing project
	Save workspace		Save model to database
	Close workspace		Close a workspace
	Save graphics as		Save a graphics file
	Print graphics		Print a graphics file directly on printer
	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



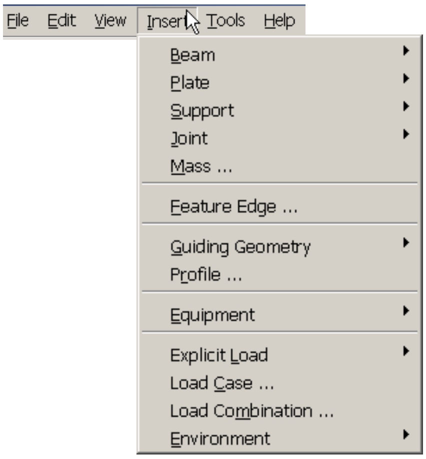
MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
Edit	**Copy**		Copy a selected object using translate, translate, rotate, or 3 point position
	Delete		Delete a selected object
	Properties		Edit and modify a property library
	Rules	**Joint creation**	Rules for automatic creation of joints
		Joint design	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)

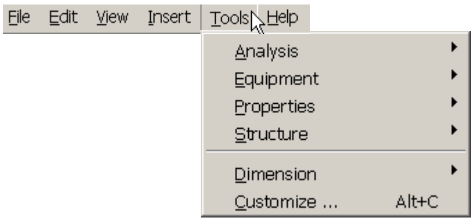
5.1.1.4 The Insert pulldown menu



MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
Insert	**Beam**	**Straight Beam Dialog**	Insert a beam and specify end points manually
		Straight Beam	Insert a beam graphically by snapping to 2 points
		Curved Beam	Insert a curved beam graphically by snapping to 3 points or more
	Plate	**Flat Plate Dialog**	Insert a plate and specify corner points manually
		Flat Plate	Insert a plate graphically by snapping to 4 points
		Skin Curves	Insert a curved plate graphically by clicking to 2 curved lines or more
	Support	**Support Point Dialog**	Insert a support and specify location manually
		Support Point	Insert a support graphically by snapping to 1 points
		Support Curve	Insert support conditions along a line (curved/straight)
	Joint	**Joint dialog**	Insert a joint by specifying it’s coordinates manually
		Joint	Insert a joint by snapping to a point
		Generate Joint	Insert joints according to the joint creating rules
	Mass	**Uniform**	Insert a point mass
		Generic point mass	Define a directional mass or diagonal mass matrix
	Feature edge		Insert an edge for controlling quality of mesh
	Guiding Geometry	**Guide Plane Dialog**	Insert a guide plane and specify corner points manually
		Guide-Curve-Dialog	Insert a guide curve and specify coordinates manually
		Guide-Line-Dialog	Insert a guide line and specify end points manually
		Guide Point Dialog	Insert a guide point and specify coordinates manually
		Guide Plane	Insert a guide plane graphically by snapping to 4 points
		Guide-Point	Insert a guide point graphically by snapping to 1 point
		Guide Line	Insert a curved guide line graphically by snapping to 2 points or more
		Polycurve	Insert a polycurve graphically by snapping to 3 points or more

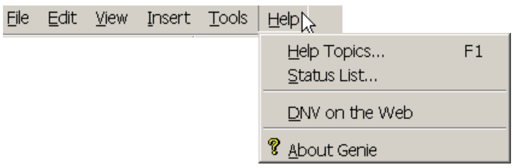
MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
		Guide Spline	Insert a guide spline graphically by snapping to 3 points or more
		Guide Arc Elliptic	Insert a guide arc elliptic graphically by snapping to 3 points (origin, start and end of ellipse)
		Guide Circle	Insert a guide circle graphically by snapping to 3 points (origin, radius, plane)
		Model Curve	Insert a model curve by snapping between two points. Curve will follow topology between
	Profile		Insert a profile to be used for punching or cut operation
	Equipment	**Prism shape**	Define a prismatic equipment
	Explicit load	**Point load**	Define a point load
		Line load	Define a constant or linearly varying line load between two points along a beam
		Surface load	Define a surface load (pressure)
		Prescribed Displacement	Specify a prescribed displacement at a support point. Must be used in connection with boundary condition prescribed
		Line Temperature	Define constant or linearly varying temperature intensity between two points along a beam. Temperature constant over cross section.
	Load case		Define a loadcase where equipments, weight lists, and explicit loads are assembled
	Load combination		Define a loadcase build up of other loadcases
	Environment	**Location**	Please see Vol.2 of this User Manual for details
		Deterministic Time Condition	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

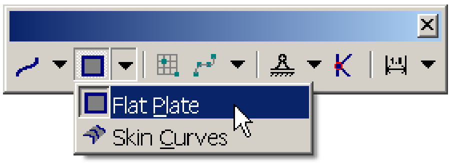
5.1.1.6 The Help pulldown menu



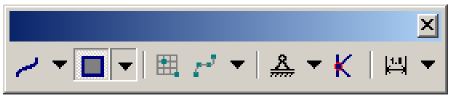
MAIN LEVEL	SECOND LEVEL	THIRD LEVEL	DESCRIPTION
Help	**Help topics**		On-line help available from GeniE
	Status List		Starts the Status List program and looks up GeniE items
	DNV on the web		Start-up of DNV Software address on web
	About GeniE		Lists copyrights and 3rd party software used

5.1.2 Tool buttons

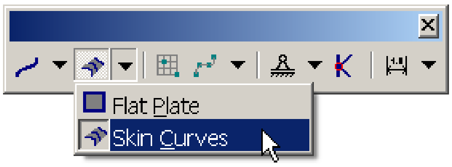
There are a number of tool buttons that give access to the most commonly used commands. They may be dynamic, meaning that the active tool button may give access to different commands. A dynamic tool button is recognised by having a pulldown arrow embedded on the GUI. One example is the tool button for Insert Plate. It can give access to two commands, either Insert a Flat Plate or Insert a Curved Plate using a skin curve operation.



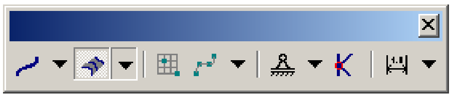
Flat Plate option selected



Flat Plate is the active operation of the tool button



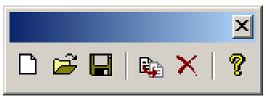
Skin Curves option selected



Skin Curves is the active operation of the tool button

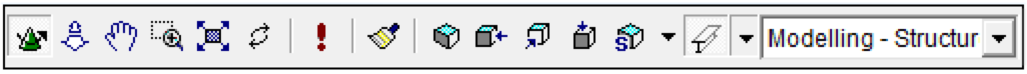
The tool buttons are grouped in eight Toolbars described below. All tool buttons have tool tips, these are listed in **bold**.

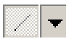
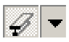
5.1.2.1 The Main toolbar



Tool button	*Tool tip*	*Description*
	**New Workspace**	Create a new workspace
	**Open Workspace**	Open existing workspace
	**Save Workspace**	Save current workspace
	**Copy with transform**	Copy selection using transformation
	**Delete**	Delete selection
	**About**	Explains GeniE program version and subcontractors

5.1.2.2 The View Manipulation toolbar



Tool button	***Tool tip***	***Description***
	**Rotate**	Rotation in all 3 degrees of freedom (use right mouse button RMB)
	**Zoom**	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)
	**Pan**	Move model to desired position on display (RMB)
	**Zoom rubberband**	Create a rubber band to zoom in (RMB)
	**Fit**	Automatic scale view so that whole model is shown on display
	**Spin**	Remembers the last rotation and speed of it and makes this a continuous spin
	Refresh graphics	Cleans up graphics and remove all dimensions
	**Color code**	Toggle on/off your selection of labelling
	**Iso view**	View from isometric point
	**View from X**	View in negative X-direction
	**View from Y**	View in positive Y-direction
	**View from Z**	View in negative Z-direction
	**Outline view**	Display beams in outline view (3D view, but no thickness)
	**Wireframe view**	Display beams in wireframe view
	**Solid view**	Display beams in solid view (3D view including thickness)
Default display	**Default display configuration**	Select the current display configuration

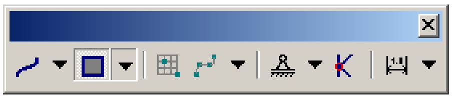
5.1.2.3 The Loadcase toolbar

Tool button	***Tool tip***	***Description***
<no loadcase>	**Default loadcase selection**	Set loadcase to current and displays current loadcase

5.1.2.4 The Label toolbar

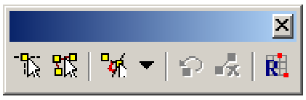
Tool button	***Tool tip***	***Description***
Name	**Label**	Labels selected object (Coordinates, Diagram value, Material, Name, Section, Thickness)

5.1.2.5 The Object Types toolbar



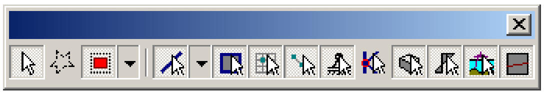
Tool button	***Tool tip***	***Description***
	Straight beam	Insert a straight beam between two snap points
	Straight overlapping beam	Insert overlapping beams between two snap points
	Pile	Insert a pile between an elevation and snap point along a beam axis
	Curved Beam	Insert a curved beam between three 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 (1st and 2nd click on points, 3rd click to position the length on the display)
	Angle between	Find and display angle between two beams (1st click on 1st beam, 2nd click on 2nd beam)

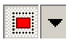
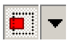
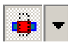
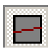
5.1.2.6 The Create Methods Toolbar



Tool button	***Tool tip***	***Description***
	**Snap point**	Positioning beam, plate, guideplane, support points etc. one by one
	**Snap point loop**	Positioning beam, plate, guideplane, support points etc. sequentially, e.g. end of beam is start of next beam
	**Snap perpendicular**	Insert a beam perpendicular to another, 1st click is from start point, 2nd click on perpendicular beam
	**Snap tangential**	Insert a guideline tangential to another curve, 1st click is from start point, 2nd click on curve
	**Snap plane**	Temporary snap points are defined at the intersection between beams and a snap plane
	**Snap eccentric**	Connect a beam to another beam end using its eccentric position.
	**Undo snap point**	Undo selection of previous snap point
	**Clear snap points**	Undo selection of all snap points
	**Reference point modelling**	Specifies journalling of reference point modelling on the journal file

5.1.2.7 The Selection Toolbar



Tool button	*Tool tip*	*Description*
	**Selection**	Method for selecting one by one or rubberband (LMB). Together with Shift more advanced selections can be made
	**Polygon select**	Method for advanced selection where you can make an arbitrary select area (LMB). Make sure that you make a closed envelope.
	**Enclosed by rubberband**	Objects need to be fully enclosed by rubberband to be part of selection
	**Touched by rubberband**	Objects need to be touched by rubberband to be part of selection
	**Select visible**	Visible objects only selected
	**Filter beam**	Toggle on/off for selection of beams
	**Filter segment**	Toggle on/off for selection of segmented members
	**Plate selection on/off**	Toggle on/off for selection of plates
	**Filter side**	Toggle on/off for selection of one side of a plate. Used when applying wet surface for panel modelling
	**Guide selection on/off**	Toggle on/off for selection of guide planes
	**Guide curve selection on/off**	Toggle on/off for selection of guide curves
	**Support selection on/off**	Toggle on/off for selection of support (points and curves)
	**Joint selection on/off**	Toggle on/off for selection of joints
	**Equipment selection on/off**	Toggle on/off for selection of equipment
	**Diagram selection on/off**	Toggle on/off for selection of load diagrams
	**Environment selection on/off**	Toggle on/off for selection of environment
	**Feature edge selection on/off**	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

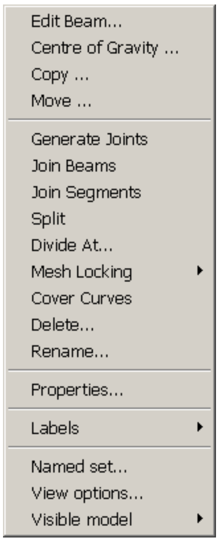
Tool button	***Tool tip***	***Description***
<no section>	**Default section**	Set and display the default section type
<no material>	**Default material**	Set and display the default material type
<no thickness>	**Default thickness**	Set and display the default thickness

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.

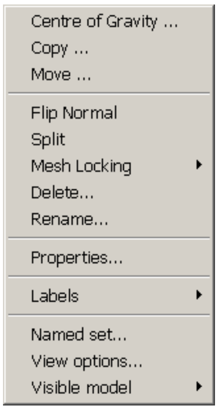
A context sensitive menu means that only relevant command for the selection will appear. The menus contain many of the commands available from the pulldown and toolbar menus, but there are some additional. Among these are the Cover Curves, the Join Beams, the Join Segments, the Select Connected beams (to a Joint), the Joint design features, and the Insert Local Joint Flexibility commands.

5.1.3.1 Object type Beam



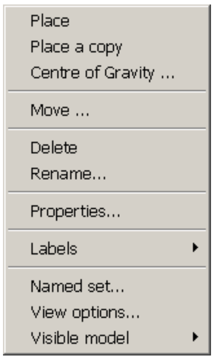
Note that Centre of Gravity, Move, Join Beams, Join Segments, Cover Curves, Labels, Named Set, Visible Model are not available from pulldown or from toolbar menus.

5.1.3.2 Object type Plate



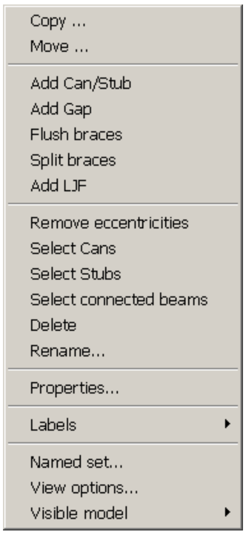
Note that Centre of Gravity, Move, Flip Normal, Labels, Named Set, Visible Model are not available from pulldown or from toolbar menus.

5.1.3.3 Object type Equipment



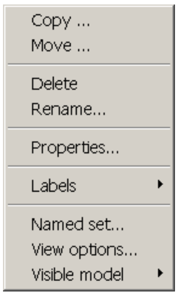
Note that Place, Place a Copy, Named Set, Visible Model are not available from pulldown or from toolbar menus.

5.1.3.4 Object type Joint



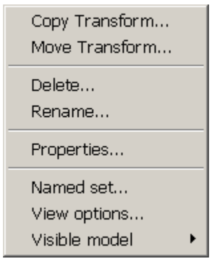
Note that most of these are not available from pulldown or from toolbar

5.1.3.5 Object type Support Point



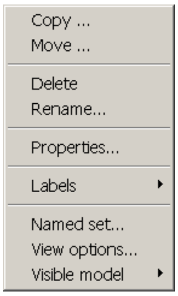
Note that Move, Labels, Named Set, Visible Model are not available from pulldown or toolbar menus.

5.1.3.6 Object type Explicit Load



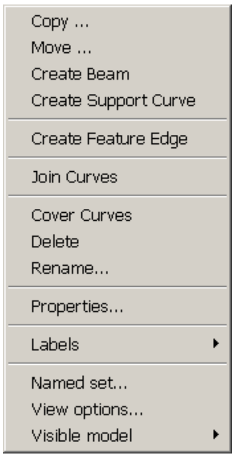
Note that Move, Named Set, Visible Model are not available from pulldown or toolbar menus

5.1.3.7 Object type Guide Plane



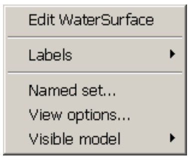
Note that Move, Named Set, Visible Model are not available from pulldown or toolbar menus.

5.1.3.8 Object type Guide Line

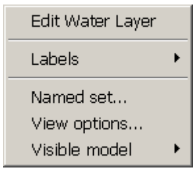


Note that Move, Create beam, Create Feature Edge, Create Support Curve, Join Curves, Cover Curves, Labels, Named Set, Visible Model are not available from pulldown or toolbar menus

5.1.3.9 Object type Water Surface



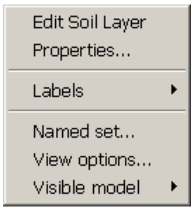
5.1.3.10 Object type Water Layer



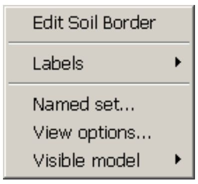
5.1.3.11 Object type Air Layer

5.1.3.12 Object type Seabed

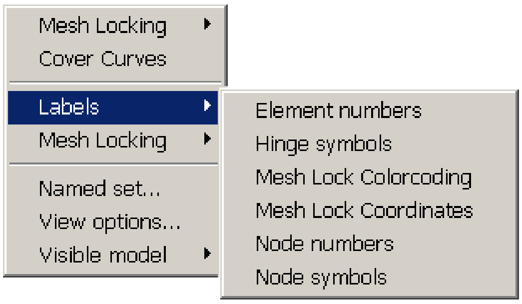
5.1.3.13 Object type Soil Layer



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 compement
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

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\DNVS\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

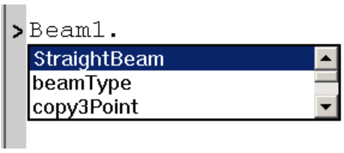
- Specific GeniE commands
- General JScript commands

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:

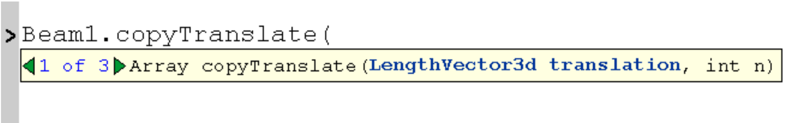
```
// Create 2 points
Point1 = Point(0,0,0);
Point2 = Point(0,0,10);
// A double slash indicates a comment line
// Copy the two points 10 m in x-direction
Point3 = Point1.copyTranslate(Vector3d(10 m,0 m,0 m));
Point4 = Point2.copyTranslate(Vector3d(10 m,0 m,0 m));
// Note that all command must end with ();
// 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
```

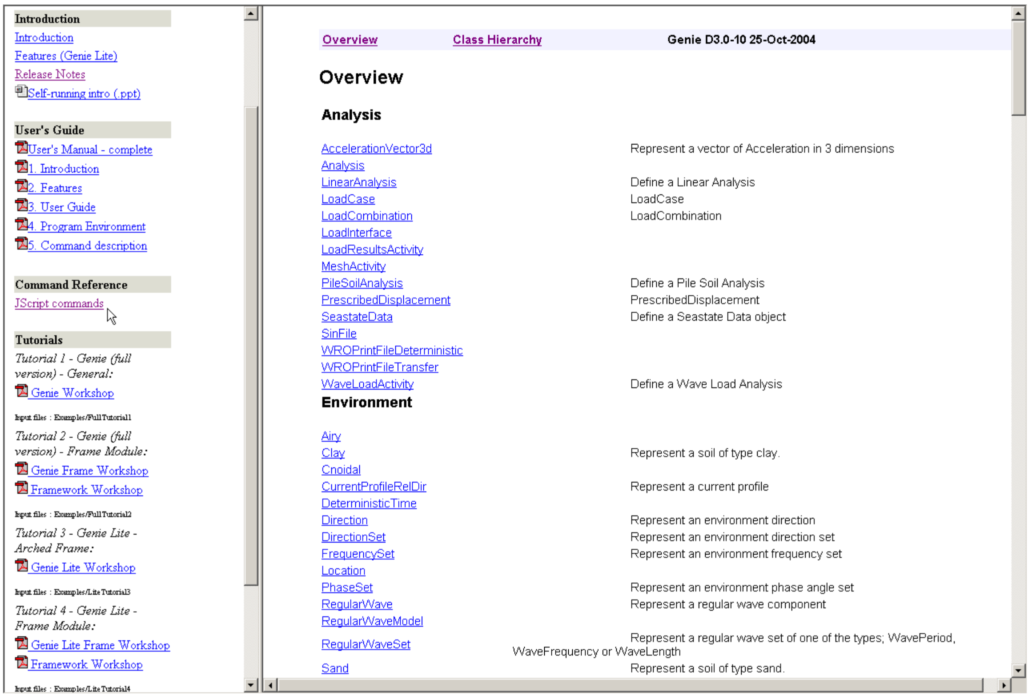
For more sophisticated modelling you may use For loops and If testes to program the creation of your model. You may also invoke other applications that support Automation (e.g. Excel) to exchange data with your GeniE model. Automation is a technology that allows software packages to expose their features 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 JSript 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.



The documentation is organised in classes:

- Analysis
- Environment
- Exchange
- Guiding geometry
- Loads
- Properties
- Sets
- Structure
- Units
- Other

Structure

Beam	Construct a new straight or curved beam from the given guide curve
Beam	Construct a new straight or curved beam from the given points
Beam	Create a straight Beam
ConstantLocalSystem	Define a constant local system for the curve
CriterionInPlane	
CurvedBeam	Curved Beam
FeatureEdge	Feature Edge
GuideLocalSystem	Let the curves local X follow the direction of the curve
Joint	Represent a tubular joint
NormalToCurvePlane	If a curve lies in a plane, use the normal as the curve local Z
Pile	Pile
Plate	Flat Plate
PointMass	PointMass
RelativeToPlate	The local Z for the curve is locked to the normal of a shell
Shell	Curved Shell
SimplifyTopology	Simplifies the structure by removing unnecessay topology.
SplitAtPoint	Split structure at the given point
SplitStructure	Explode selected structure at all structural connection points
StraightBeam	Straight Beam
SupportCurve	Represent a Support Curve
SupportPoint	Represent a Support Point

Looking up straight beam found under the class Structure.

Function Detail

Beam

BasicBeam **Beam**(Point p1, Point p2, OverlapPolicy overlapPolicy)

Create a straight Beam

Parameters:
`p1` - End 1 of the Beam
`p2` - End 2 of the Beam
`overlapPolicy` - Specifies how this beam is to be inserted into the model

Example:

```
//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 reomve 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.

Overview Class Hierarchy **Genie D3.0-10 25-Oct-2004**

BasicBeam

Class hierarchy:

```
ModelObject
  |
  +--NamedObject
        |
        +--Transformable
              |
              +--BasicConcept
                    |
                    +--BasicBeam
```

Direct Known Subclasses:
CurvedBeam, Pile, StraightBeam

Function Summary

BasicBeam	**divideAt**(double parameter) *Divide beam at parameter value and return the second half of the beam*
	divideSegmentAt(int iseg, double parameter) *Divide beam segment at parameter value*
	explode(NameMask nameMask) *Split structure into smaller parts*
bool	**extendEnd**(long iend, Length extension) *Extend or shorten beam at end 1 or 2 along direction of beam*
Material	**getSegmentMaterial**(int)
Section	**getSegmentSection**(int)
Point	**intersect**(Plane3d plane) *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	
ModelObject	**copy3Point**(Point sp1, Point sp2, Point sp3, Point dp1, Point dp2, Point dp3) *Copy the object without scaling from one location to another.*
ModelObject	**copyMirror**(Point p1, LengthVector3d v1) *Mirror a copy of the object around `p1` and `v1`*
ModelObject	**copyRotate**(Point p1, LengthVector3d p2, Angle angle) *Rotate a copy of the object around `p1` and `v1` the angle `angle`*
Array	**copyRotate**(Point p1, LengthVector3d p2, Angle angle, int n) *Make `n` copies of the object,incrementing the `angle` for each copy*
	copyRotate(Point p1, LengthVector3d p2, Angle angle, int n, NameMask 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`

Decription:

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 **Alt+D** or ***Tools|Analysis|Create Mesh***)
ExportMeshFem().DoExport(name);	Export a finite element model to the default working directory (same as ***Tools|Analysis|Export FEM File***), 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**
Graphics.saveImage(name,width,height);	Save the image in a given format, scaled to the size in pixels given by width and height, e.g. Graphics.saveImage(“Picture.jpg”,2000,2000);
Graphics.zoomArea(left,bottom,right,top);	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 *Graphics.zoom Area(150,150,450,450);*. If you want to zoom out, you may use *Graphics.zoomArea(-150,-150,750,750);*.
Graphics.setEye(eyePos);	The command gives a view of the model focused on its origin,e.g. Graphics.setEyePosition(Point (-30,30,5),Vector3d(1,-1,0));. To be used to recreate a specific view.
Graphics.viewISO();	Same as the graphical operation 'View ISO (F5)' Note, you need to refresh graphics to yield immediate screen update
Graphics.viewFromX();	Same as the graphical operation 'View from X (F6)' Note, you need to refresh graphics to yield immediate screen update
Graphics.viewFromY();	Same as the graphical operation 'View from Y (F7)' Note, you need to refresh graphics to yield immediate screen update
Graphics.viewFromZ();	Same as the graphical operation 'View from Z (F8)' Note, you need to refresh graphics to yield immediate screen update
Graphics.fitModel();	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>);	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);
Math.timer(long t0);	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): *t0=Math.timer(0);* *t1=Math.timer(t0);* *print(t1);* -> *returns time since t0 in seconds* *t2=Math.timer(t0);* *print(t2);* ->*returns time since t0 in seconds*

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 *L*	Mass Unit *M*	Force Unit ML/T^2	Typical program input values: Yield stress for steel $M/(LT^2)$	Density of steel (Mass/Volume) M/L^3	Young's modulus for steel (Force/Area) $M/(LT^2)$
m	kg	1 N	$4.2 \cdot 10^8$	$7.85 \cdot 10^3$	$2.10 \cdot 10^{11}$
m	10^3 kg = 1 t	10^3 N = 1 KN	$4.2 \cdot 10^5$	7.85	$2.10 \cdot 10^8$
cm	kg	10^{-2} N	$4.2 \cdot 10^6$	$7.85 \cdot 10^{-3}$	$2.10 \cdot 10^9$
cm	10^3 kg = 1 t	1 kgf ≈ 10 N	$4.2 \cdot 10^3$	$7.85 \cdot 10^{-6}$	$2.10 \cdot 10^6$
mm	kg	10^{-3} N	$4.2 \cdot 10^5$	$7.85 \cdot 10^{-6}$	$2.10 \cdot 10^8$
mm	10^3 kg = 1 t	1 N	$4.2 \cdot 10^2$	$7.85 \cdot 10^{-9}$	$2.10 \cdot 10^5$
cm	10^2 kg	1 N	$4.2 \cdot 10^4$	$7.85 \cdot 10^{-5}$	$2.10 \cdot 10^7$
m	10^4 kg	1 tonnef ≈ 10000 N	$4.2 \cdot 10^4$	$7.85 \cdot 10^{-1}$	$2.10 \cdot 10^7$
cm	10^6 kg	1 tonnef ≈ 10000 N	4.2	$7.85 \cdot 10^{-9}$	$2.10 \cdot 10^3$
mm	10^7 kg	1 tonnef ≈ 10000 N	$4.2 \cdot 10^{-3}$	$7.85 \cdot 10^{-13}$	2.10
m	10 kg	1 kgf ≈ 10 N	$4.2 \cdot 10^7$	$7.85 \cdot 10^2$	$2.10 \cdot 10^{10}$
cm	10^3 kg	1 kgf ≈ 10 N	$4.2 \cdot 10^3$	$7.85 \cdot 10^{-6}$	$2.10 \cdot 10^6$
mm	10^4 kg	1 kgf ≈ 10 N	$4.2 \cdot 10^1$	$7.85 \cdot 10^{-10}$	$2.10 \cdot 10^4$

7.2 Consistent Empirical units

Length Unit *L*	Mass Unit *M*	Force Unit ML/T^2	Typical program input values: Yield stress for steel $M/(LT^2)$	Density of steel (Mass/Volume) M/L^3	Young's modulus for steel (Force/Area) $M/(LT^2)$
Foot	1 lb	Poundal		491	$1.39 \cdot 10^{11}$
Inch	12 lbs	Poundal		$2.37 \cdot 10^{-2}$	$9.66 \cdot 10^8$
Foot	32.2 lbs (1 slug)	Pound(f)		15.2	$4.32 \cdot 10^9$
Inch	386 lbs	Pound(f)		$7.35 \cdot 10^{-4}$	$3.0 \cdot 10^7$
Foot	$3.22 \cdot 10^4$ lbs	Kip		$1.52 \cdot 10^{-2}$	$4.32 \cdot 10^6$
Inch	$3.86 \cdot 10^5$ lbs	Kip		$7.35 \cdot 10^{-7}$	$3.0 \cdot 10^4$
Foot	$7.21 \cdot 10^4$ lbs	Ton(f)		$6.81 \cdot 10^{-3}$	$1.93 \cdot 10^6$
Inch	$8.66 \cdot 10^5$ lbs	Ton(f)		$3.28 \cdot 10^{-7}$	$1.34 \cdot 10^4$

---------------------------- o ----------------------------