



SESAM WORKSHOP



Structural Reanalysis System Models
DNV Software

DET NORSKE VERITAS

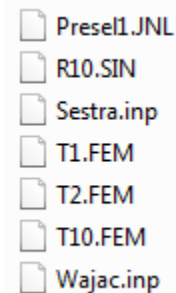
Sesam – GeniE Workshop Structural Reanalysis System Models

The purpose of this workshop is to show users how to make use of older FEM models. The old FEM files can be imported into GeniE for codechecking of superelements. Additionally models can be updated with section and material properties without complete remesh, thus keeping the node and element numbering. In this workshop we are not focusing on making an automated Sesam Explorer workflow. Most of the tasks will be carried out manually. After knowing the complete workflow, users can accomplish the tasks more efficiently by utilizing advanced functionalities of Sesam Explorer.

Preparations for the workshop:

Your computer should have the following Sesam software installed: GeniE V5.3-10 or later, Presel, Wajac, Sestra, Sesam Explorer and Xtract.

You should have access to the folder *Input*, as this folder contains the files we are using as input for the workshop. The files contained in the folder *Input* are listed to the right.

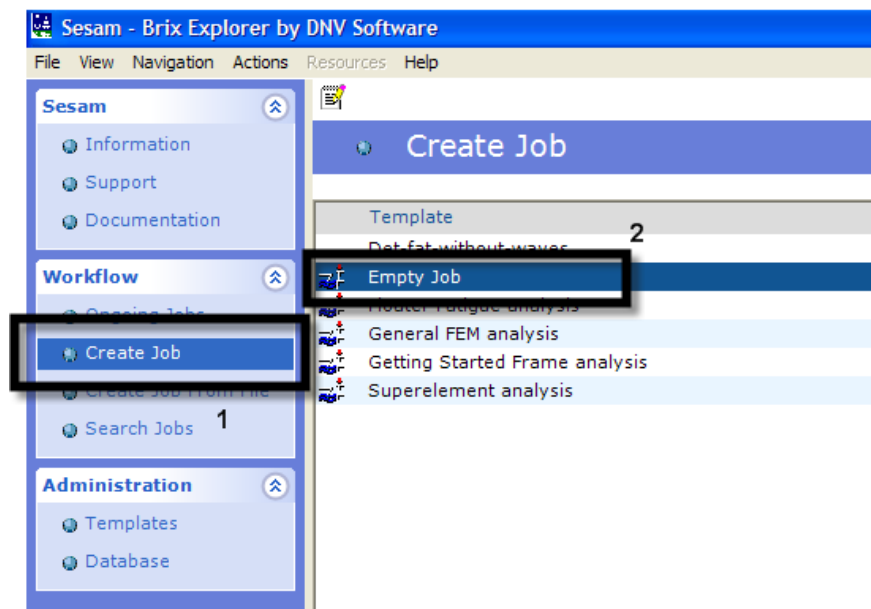


- Presel1.JNL
- R10.SIN
- Sestra.inp
- T1.FEM
- T2.FEM
- T10.FEM
- Wajac.inp

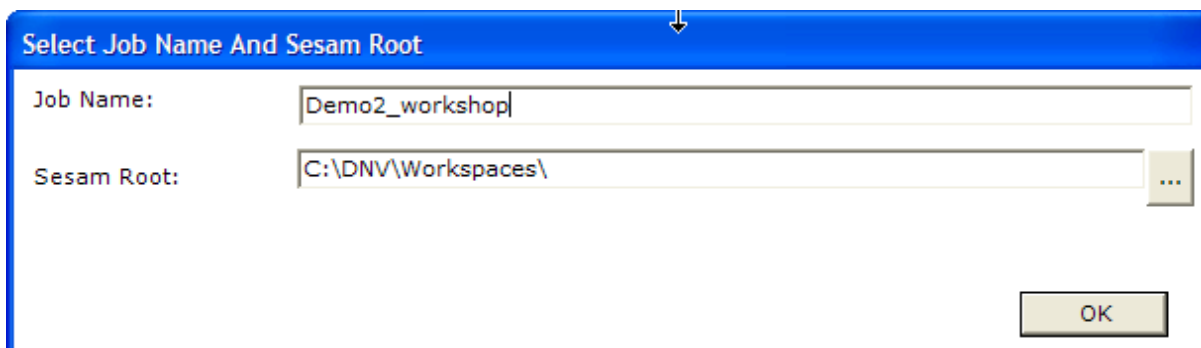
The workshop model consists of two super elements, *T1.FEM* is a jacket and *T2.FEM* is a topside. *R10.SIN* is the results file for the top level superelement.

Start Sesam Explorer. You may have a shortcut icon on desktop or click on *Start> Programs> DNV Software> Sesam*

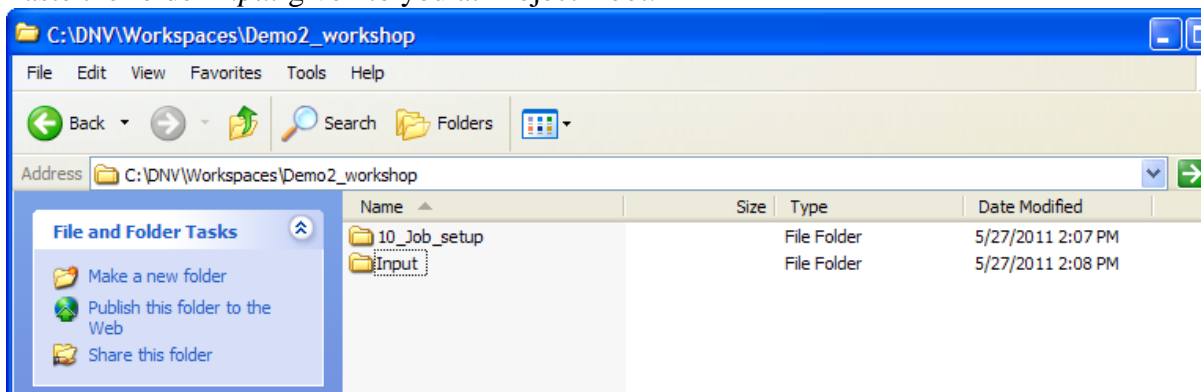
Click on *Create Job* and then *Empty Job* as shown in the image below.



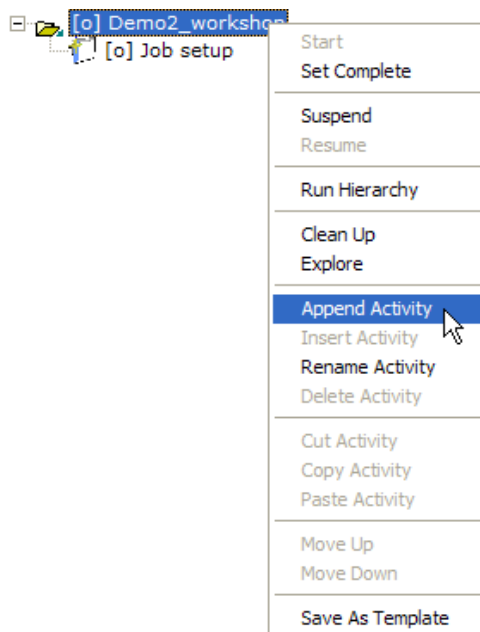
Select the workspace location and give a name to the job.



Paste the folder *Input* given to you at Project Root.



Click on the *Project name* and right click, click on *Append Activity*.

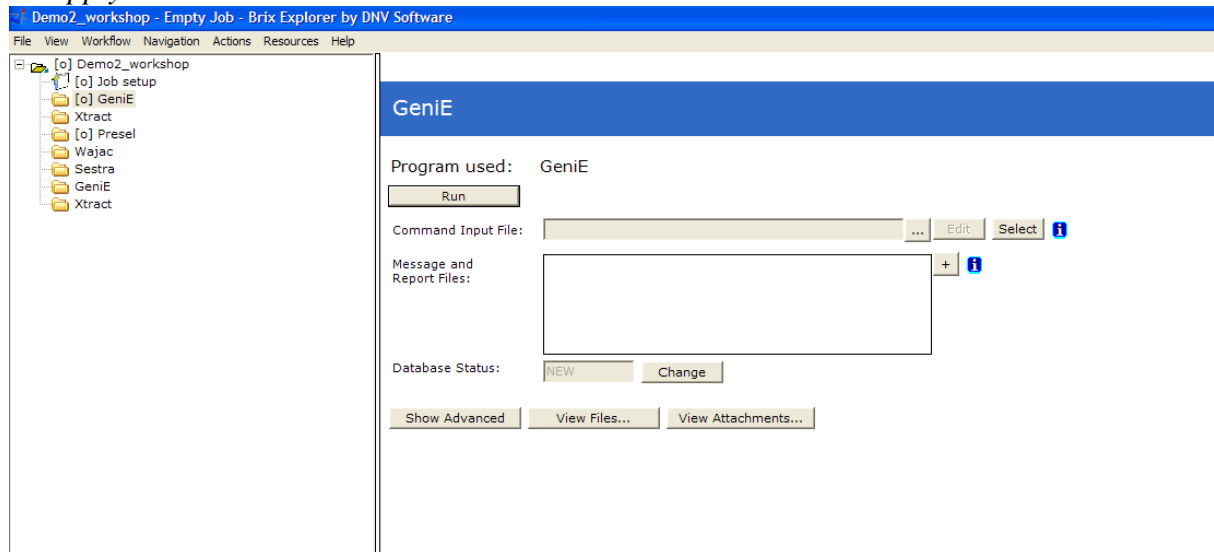


Select *GeniE* and click on *Apply*.

Repeat the procedure above and add the following programs to the workflow:

Xtract
Presel
Wajac
Sestra
GeniE
Xtract

Now you can see the Sesam programs in the list. Select the required programs and click on *Apply* after each selection. Workflow will look as follows:



The existing *T1.FEM* file is created with database units of *meters* and *kiloNewtons*, so click on the first *GeniE* folder shown in the image above, then click on *Show Advanced* and change the database units as shown in the next image.

Demo2_workshop - Empty Job - Brix Explorer by DNV Software

File View Workflow Navigation Actions Resources Help

[o] Demo2_workshop

- [o] Job setup
- [o] GeniE
- [o] Xtract
- [o] Presel
- [o] Wajac
- [o] Sestra
- [o] GeniE
- [o] Xtract

GeniE

Program used: GeniE

Run

Hide Advanced View Files...

Configuration | Input Files | Output Files | Attachments | Status

Model Name: GeniE Select model

Database Status: ☒ New ☐ Old

Input Mode: ☒ Interactive ☐ Background

Command Input File:

Output File Prefix: i

☐ Accumulate in database i

☒ Set Database Units i ☒ Enable tolerant modelling

Length: m

Force: kN i Read command file made using other GeniE version:

Temperature: delC Compatibility: Current i

☐ Reuse database from other GeniE activity i

Other activity folder: C:\DNV\Workspaces\Demo2_workshop\70_GeniE

☐ Copy Analysis folder(s) i

Hide Details Refresh

Command Line: C:\Program Files\DNVS\GeniE V5.3-10\Program\GeniE.exe "" /NEW /COMPATIBILITY-VERSION=current

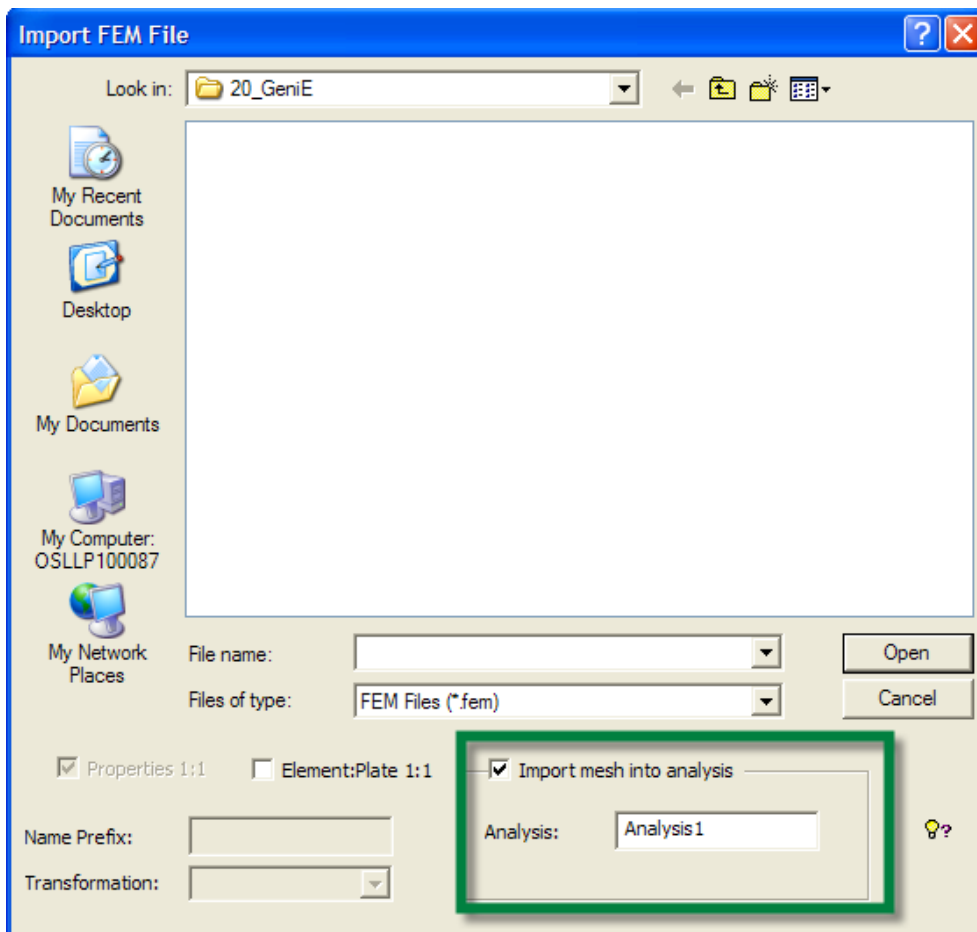
Job Root: C:\DNV\Workspaces\Demo2_workshop

Working Directory: C:\DNV\Workspaces\Demo2_workshop\70_GeniE

Now click on *Run*. In principle you may define all the steps shown below in the command input file but for workshop purpose we will do all the steps interactively.

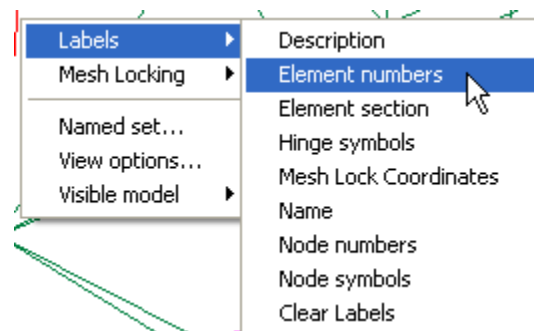
Now GeniE workspace opens.

Click on *Import> File> Fem File*. Make sure that you select the option *Import Mesh into Analysis*.

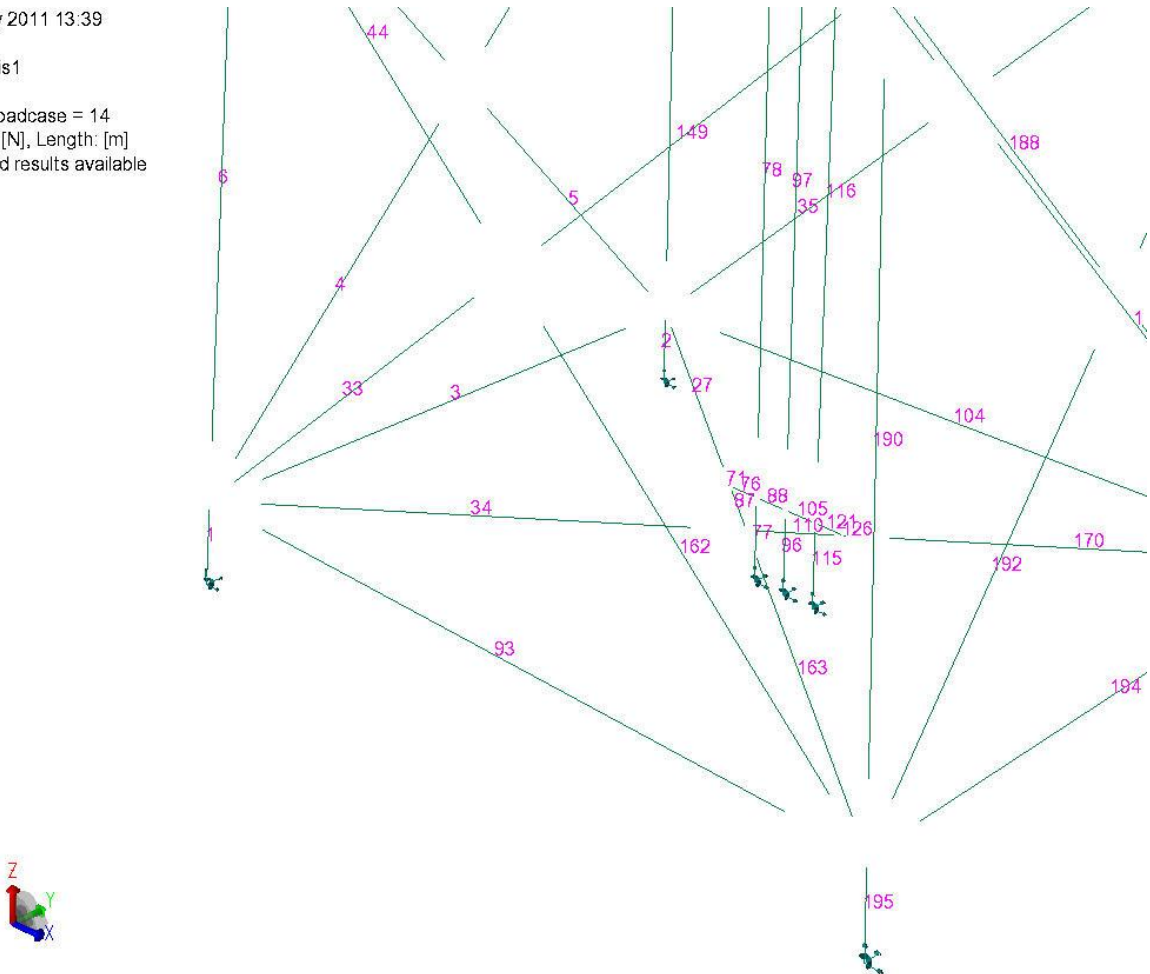


Browse for the *T1.FEM* file in the folder *Input* on the project root. After importing, you may verify the mesh of the model in GeniE by comparing it with Xtract.

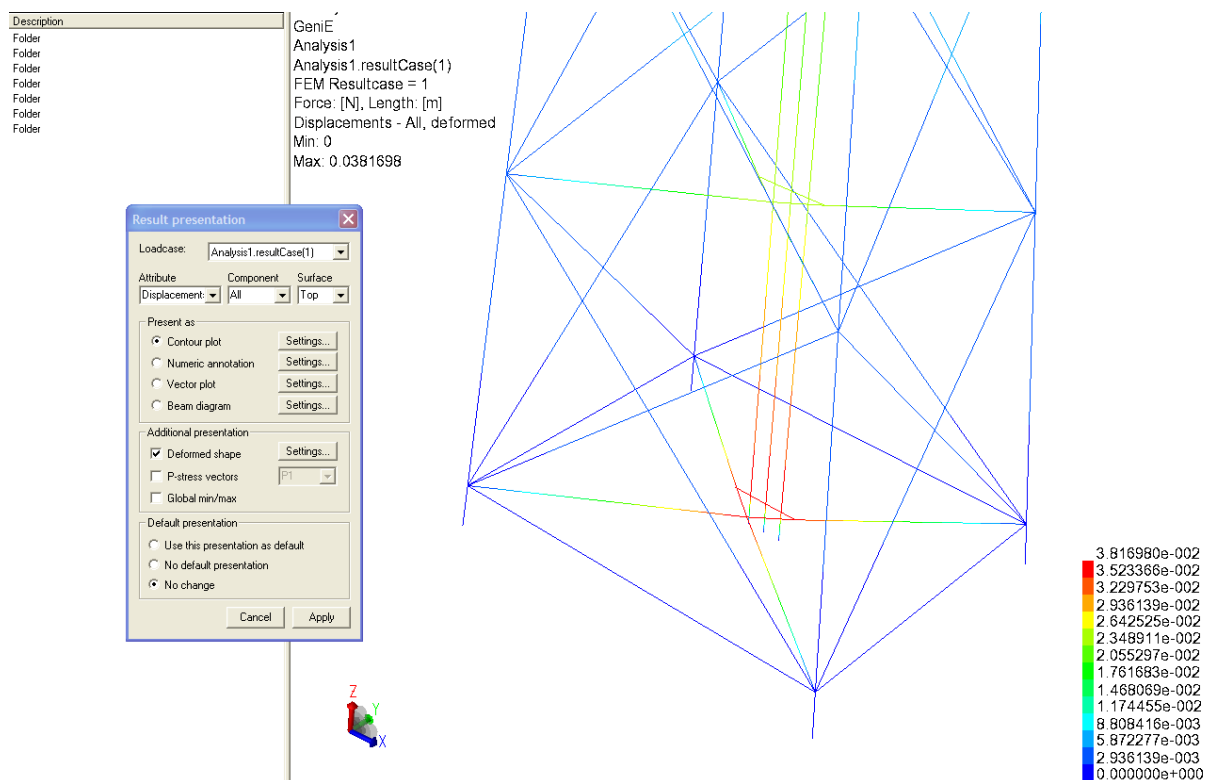
To achieve this, switch to *Mesh Transparent* view in GeniE, select some elements and right click. Select *Labels > Element Numbers*.



27 May 2011 13:39
GeniE
Analysis1
WLC9
FEM Loadcase = 14
Force: [N], Length: [m]
No valid results available

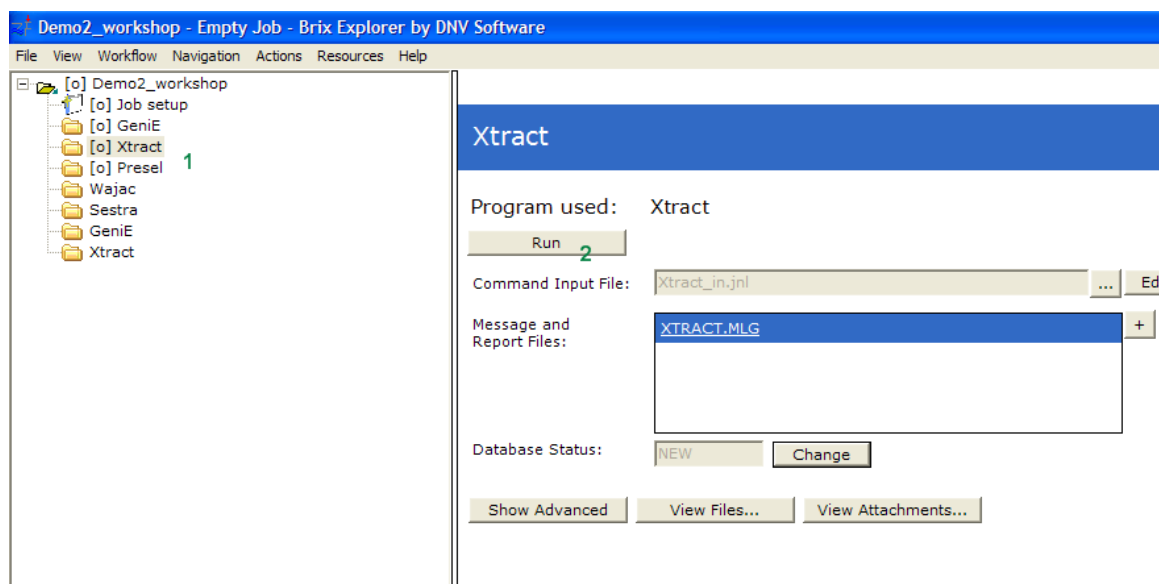


Now we should import the results file into GeniE. Select *File> Import> External Results SIN file*. You have to browse for the results file *R10.SIN* in the *Input* folder. After importing the results file, we can display some results e.g. for resultcase *Analysis1.resultCase(1)*. In the Result Presentation window (*Tools> Analysis> Presentation* or *Alt+D* on keyboard). Display the results e.g. for Displacements 'All'.

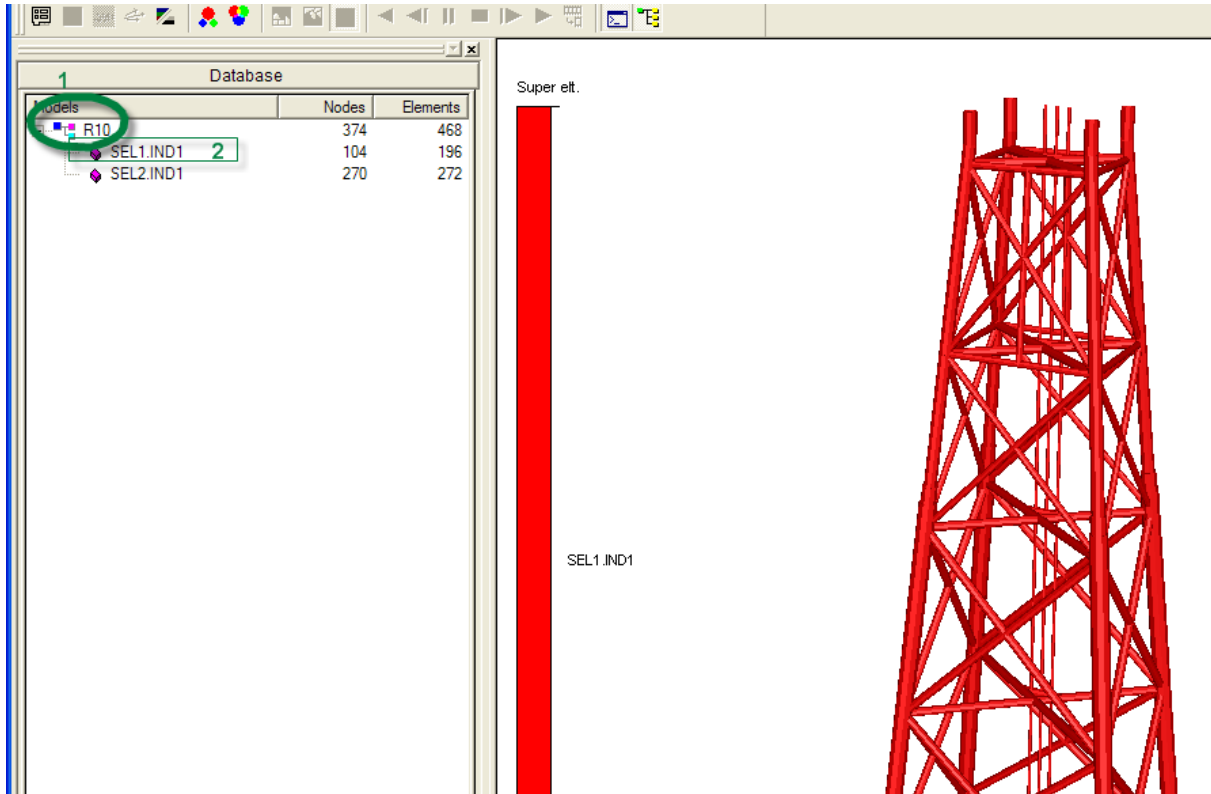


Now we will do a comparison study with Xtract to see if the results are matching.

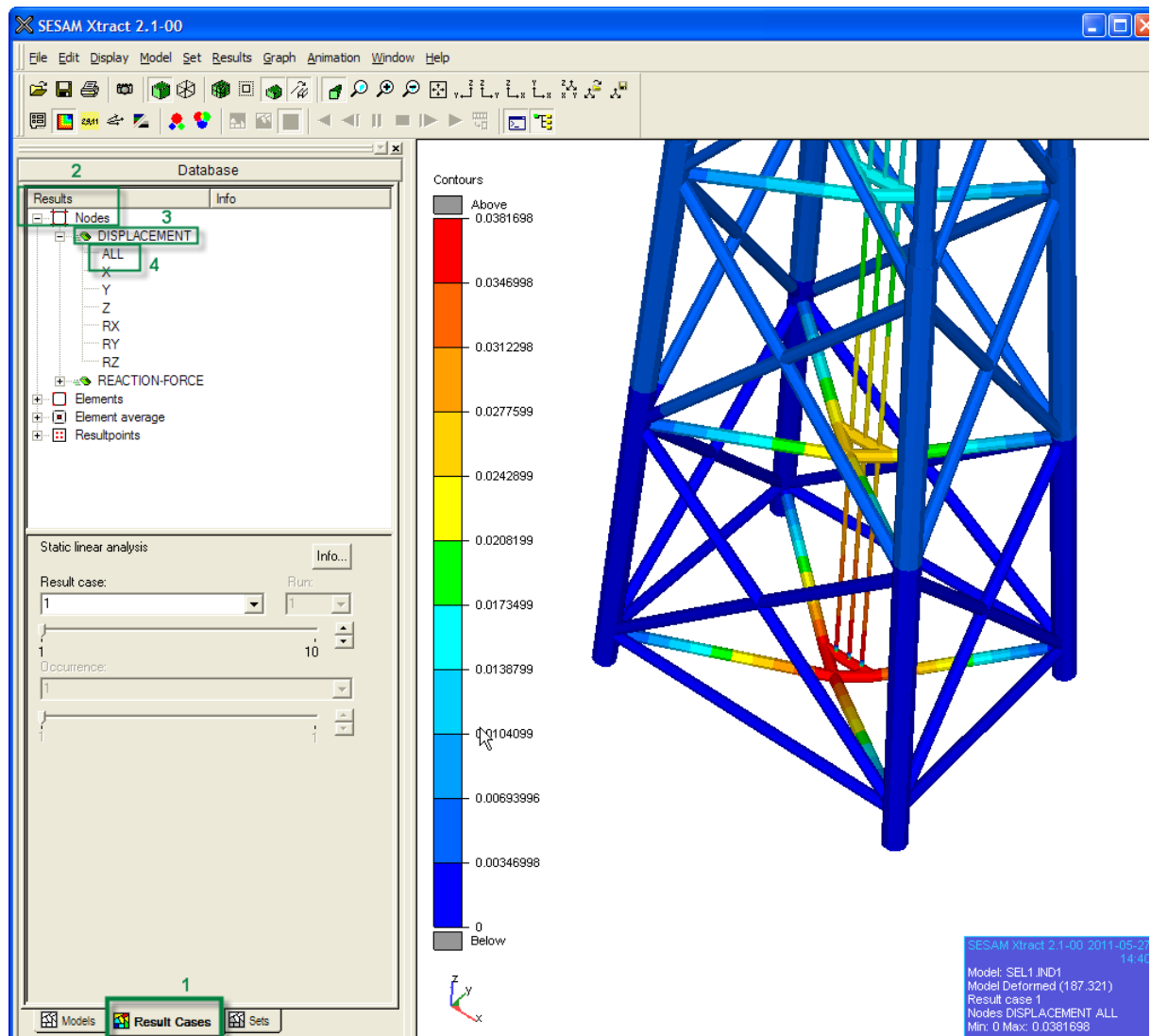
To start Xtract, go to Sesam Explorer window and click on *Xtract* and then click on *Run*,



In Xtract, open results file *R10.SIN* (available in the Input folder). Then expand *R10.SIN* in the database window and select *SEL1.IND1* to display results for Superelement type 1 i.e. *T1.FEM*.



To display the results, for instance displacements for resultcase1, click on *Resultcases* tab, expand the result attributes *Nodes > Displacements > All*



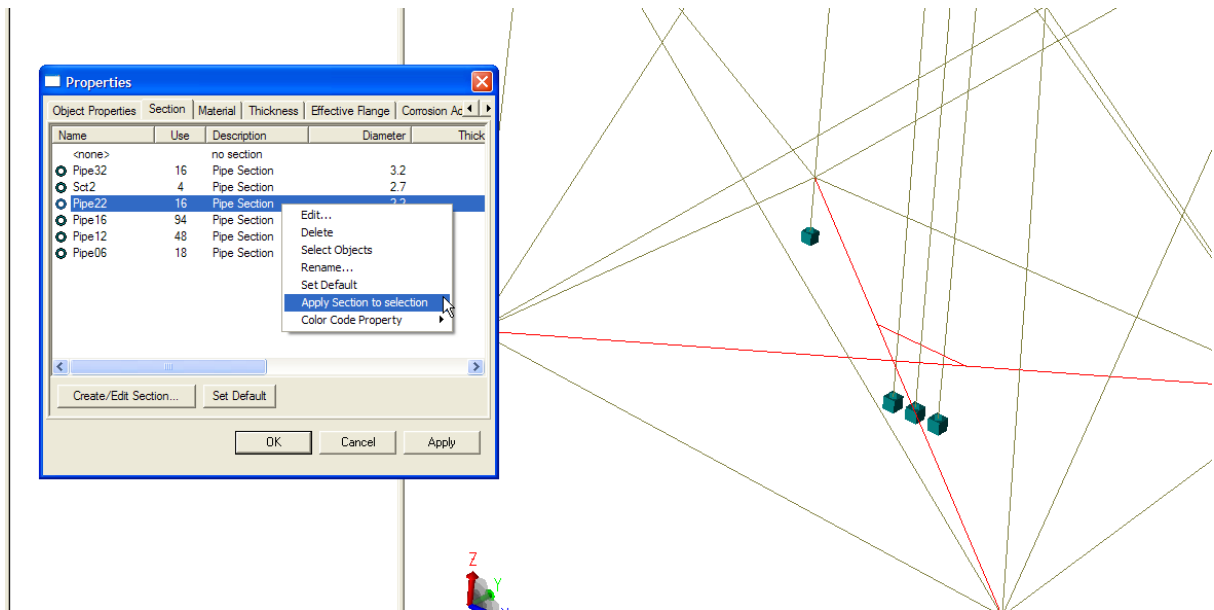
If time permits you may also label element or node numbers in Xtract and compare it with GeniE to see that node and element numbers are the same.

Now switch back to the GeniE window. We will do a property update on a few beams near the bottom of the structure. We will change the section properties of these beams.

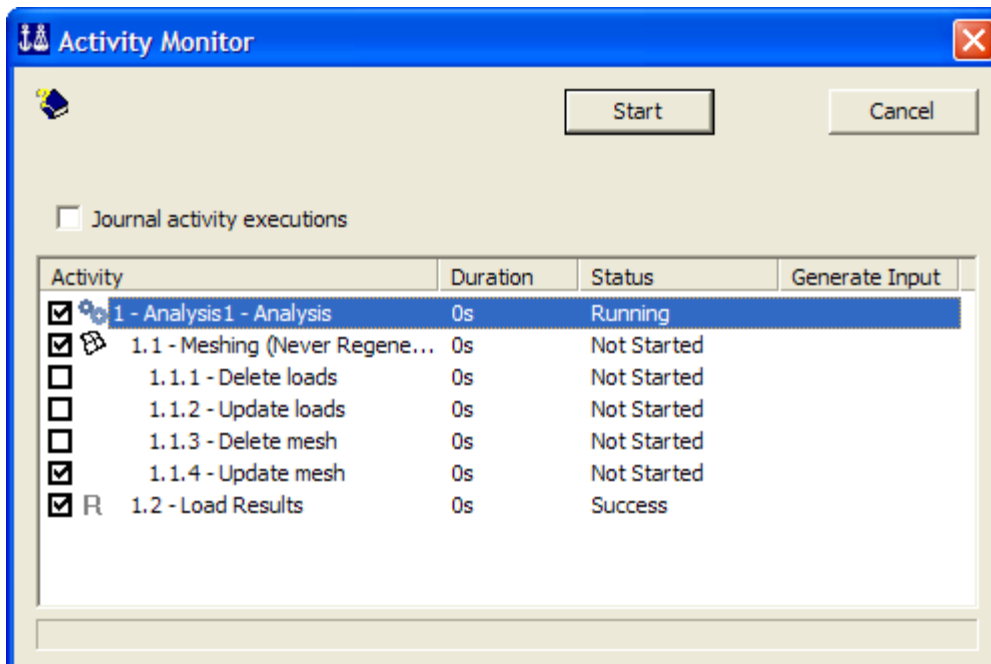
Switch to *Modelling Structure* view and select the following beams to change the properties,

Bm27
Bm34
Bm76
Bm87
Bm88
Bm105
Bm110
Bm121
Bm163
Bm170

Apply *Pipe22* section property to all these beams.

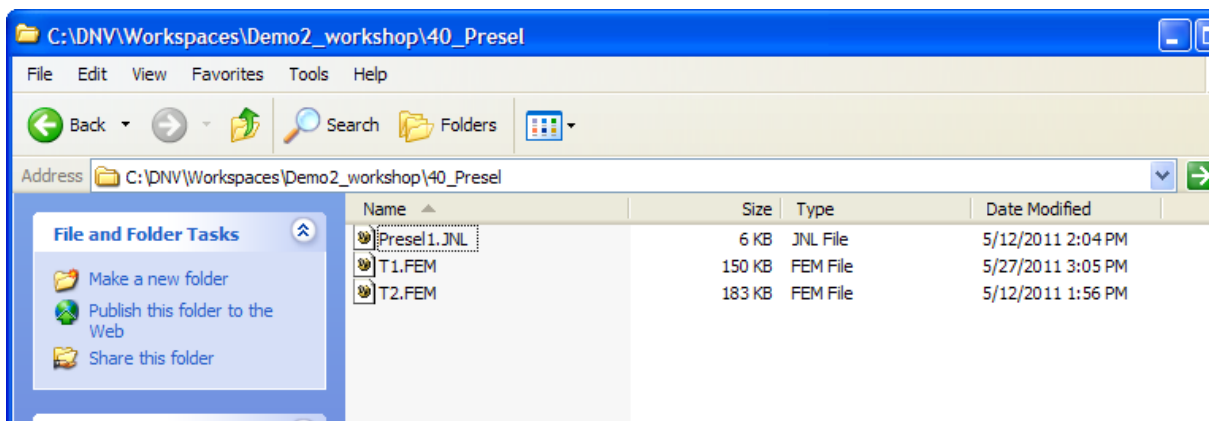


Now press Alt+D on keyboard to open the Activity Monitor. Make sure that under the *Meshing* activity, only *Update Mesh* is checked on. Click on Start to update the mesh.

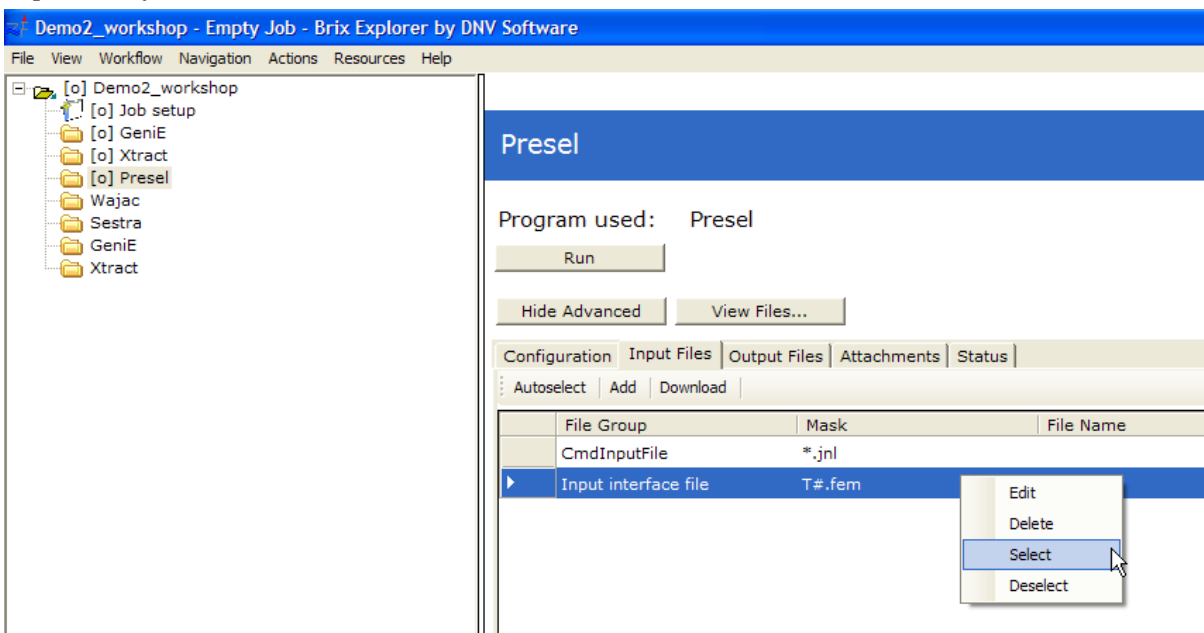


You may find an updated *T1.FEM* file in the *Analysis1* folder in the GeniE folder. Copy this *T1.FEM* file and copy it to the *Presel* folder.
Now we will assemble both *T1.FEM* and *T2.FEM* models into higher order Superelement *T10.FEM* file. Also we will create load combinations in Presel for the Top level Superelement.
For this purpose, we will copy the *T1.FEM* file from GeniE, Analysis folder. Also, copy the *T2.FEM* file and *Presel1.jnl* file into the *Presel* folder at the project root.

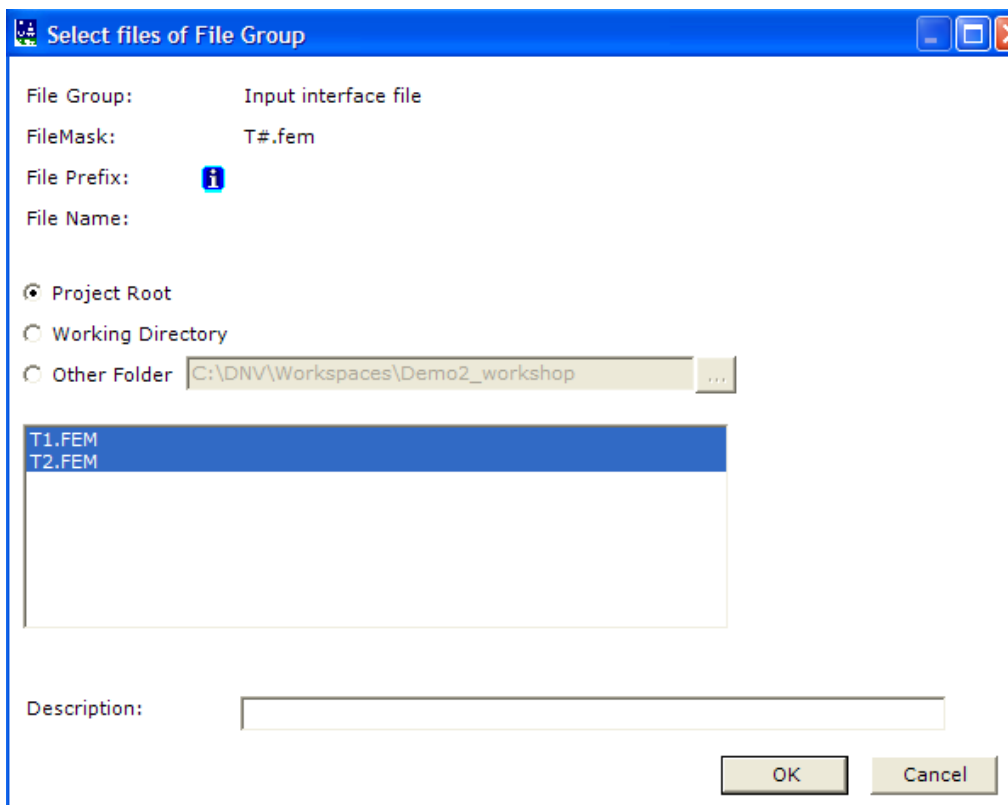
The Presel activity folder should look like below:



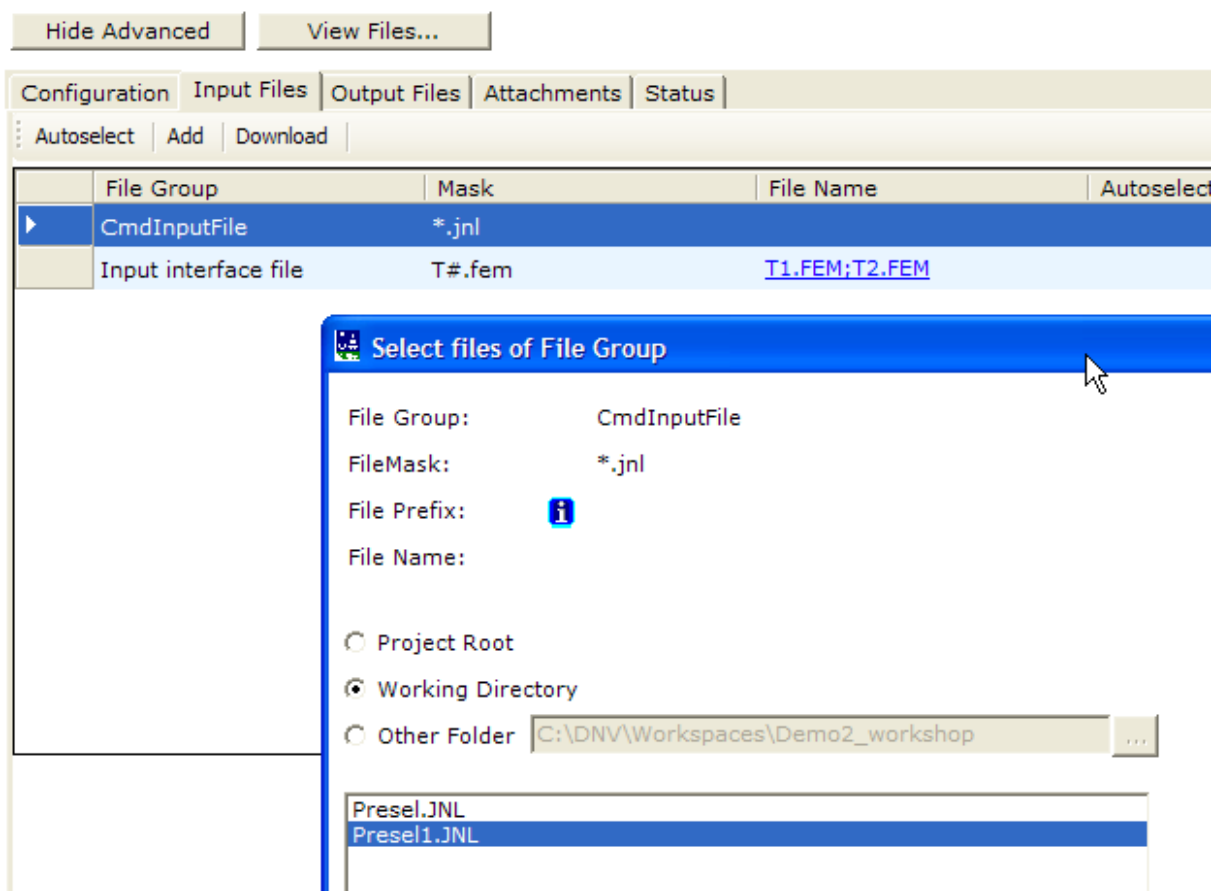
To guide Presel to pick the required files, click on *Input Files* tab and then right click on *Input Interface File* and click on *Select*.



Click on *Project Root* and select both *T1.FEM* and *T2.FEM* files, finish by clicking on *Ok*.

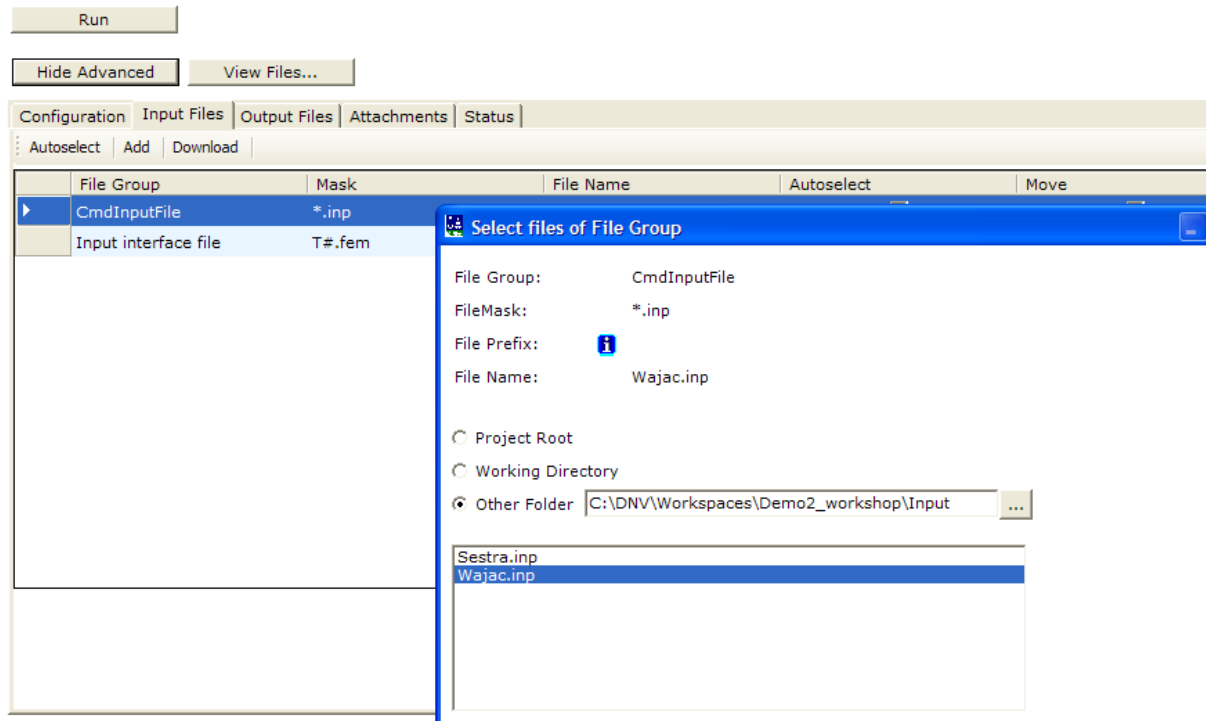


Similarly, browse for the journal file *Presel1.jnl*.



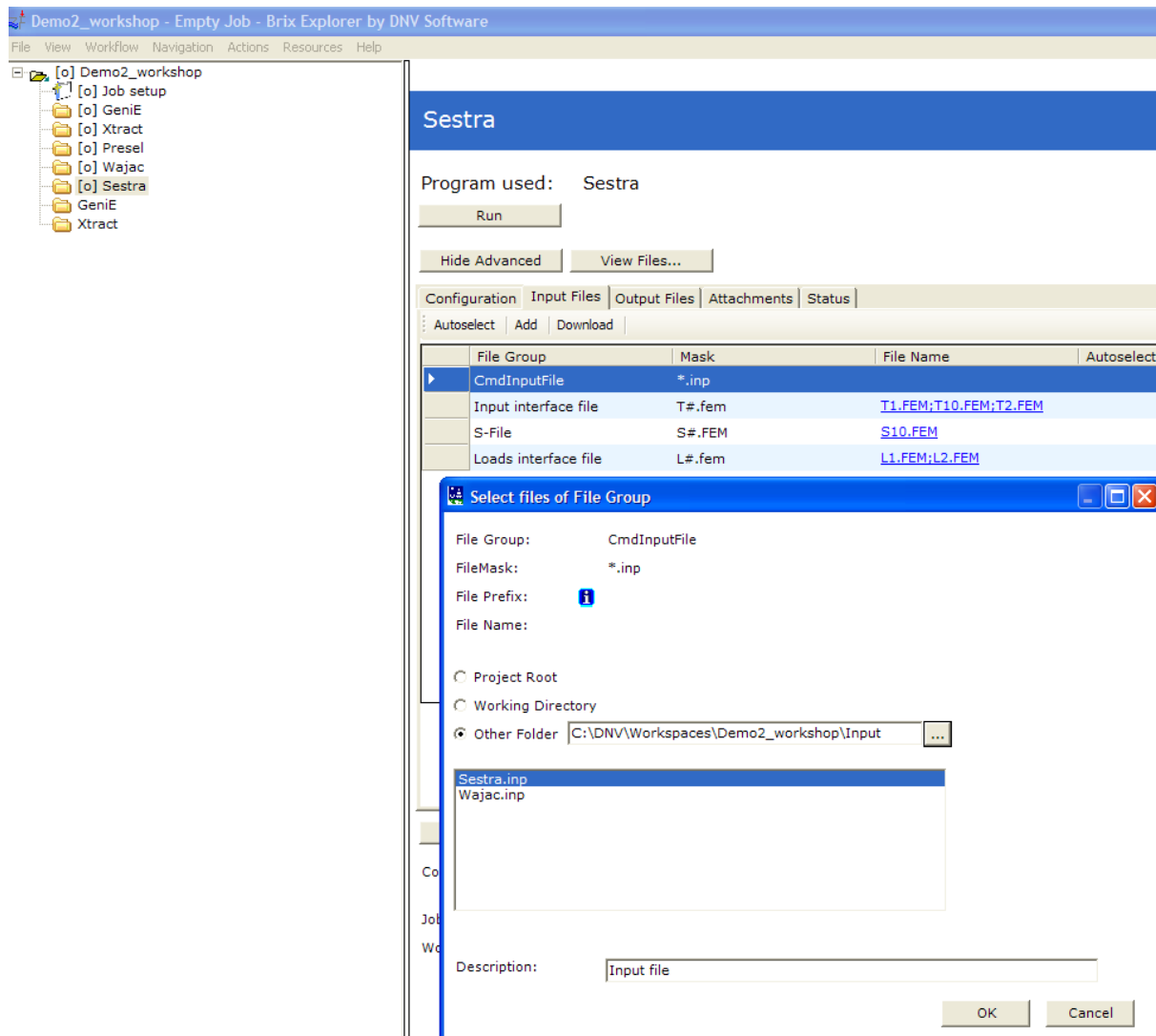
Now click on *Hide advanced* and *Run Presel* activity. Close Presel by clicking on *Exit*. Now in the project root, you will be able to see all the 3 fem files.

Now we will compute waveloads on the model. After clicking on *Wajac* folder, click on *Show Advanced* and click on *Input Files* tab as shown in image above. We have to select the Wajac input file. In the browsing window, select *Other folder*, and select *wajac.inp* from the *Input* folder.



Click on *Hide Advanced* and Run Wajac program. After Wajac finishes running, you will find the additional load files *L1.FEM* and *L2.FEM* in the Project root for first level Superelement and *S10.FEM* file which will be used by Sestra to read in the loadcase numbering for seastates.

Now we will run Sestra for structural analysis. We will follow the same procedure as we did for Wajac above. The *T*.FEM* files are selected by Sesam explorer by default along with load files. We have to guide it to pick the input file which is located in the *Input* folder. So follow the same procedure as above after clicking on *Sestra* folder in the *Sesam Explorer*.



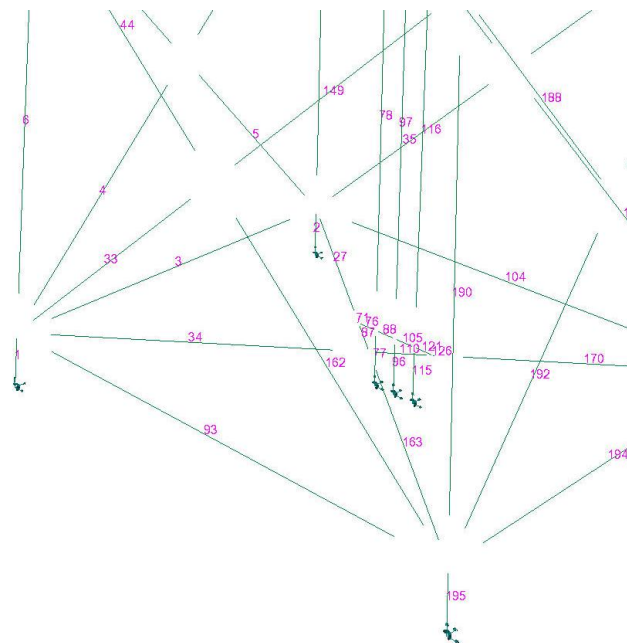
Now run Sesra and we will get an updated results file *R10.SIN*

We now have two options

1. We start a new GeniE workspace and import the updated *T1.FEM* file found on the project root and then read in the results file *R10.SIN* (latest one, located on the project root).
2. We import the *R10.SIN* file (latest one) in the existing GeniE workspace which is already opened i.e. the first activity in the workflow.

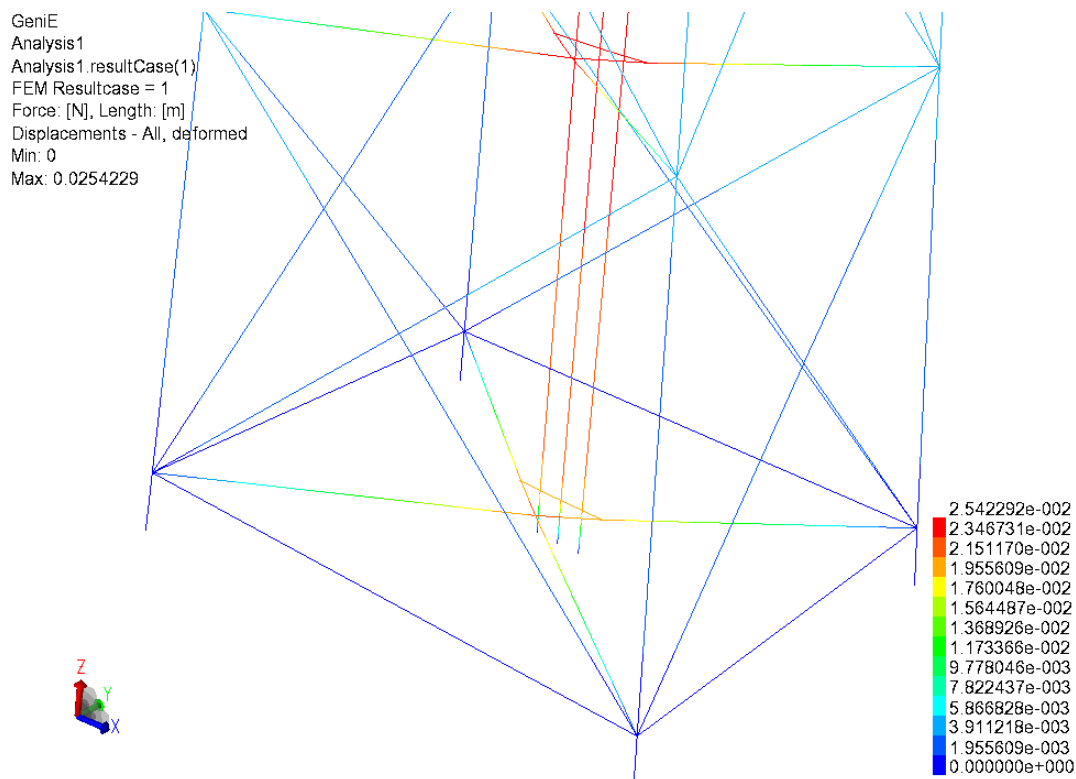
We will follow option 2 in this workshop (if you have closed the GeniE workspace then you can start it again on a condition that you have saved the previous database, or you can follow option 1).

Label element numbers in GeniE, to see that the numbers are still the same.

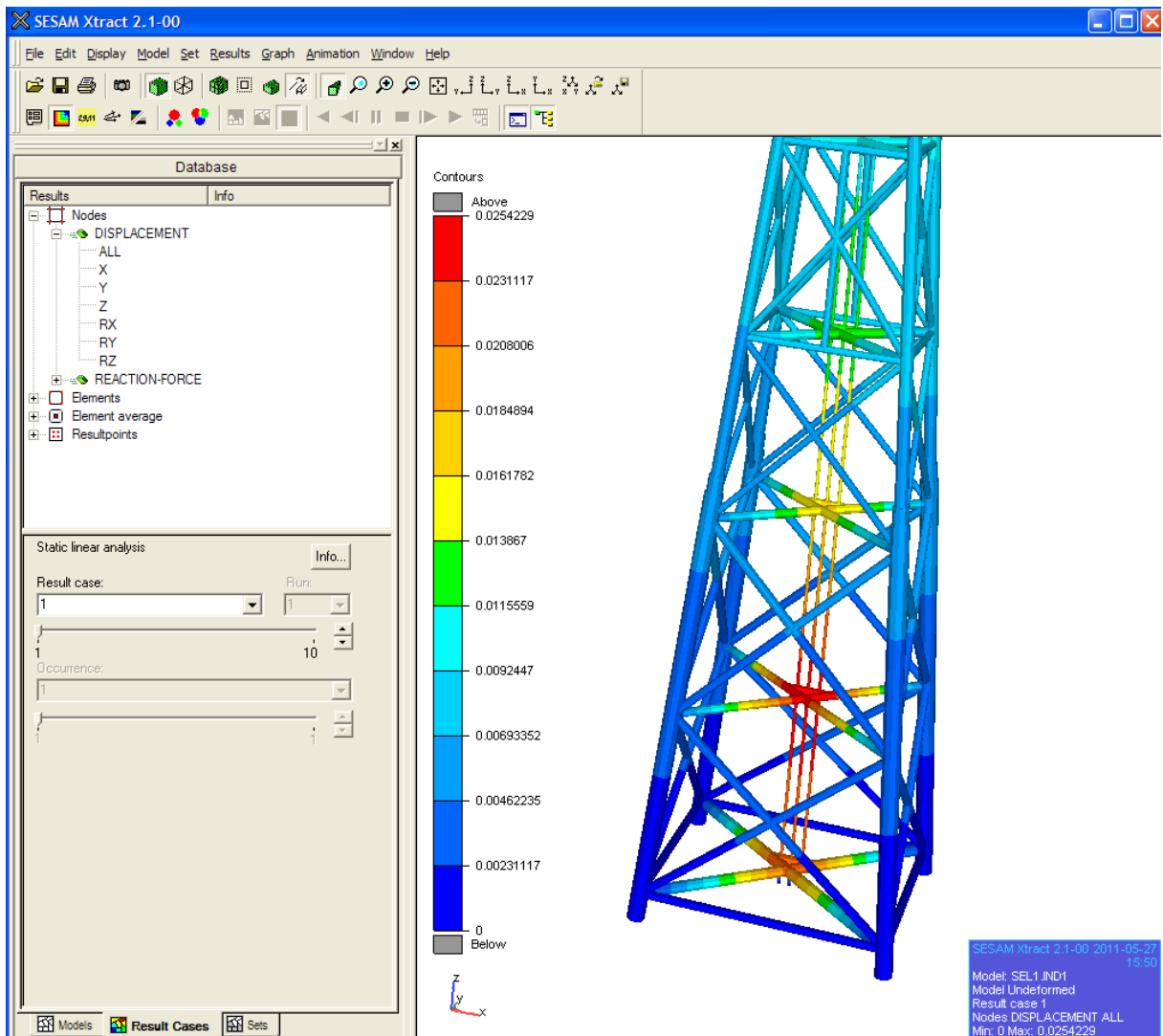


Select *File> Import> External Results SIN*
File and browse for the *R10.SIN* on the project root.

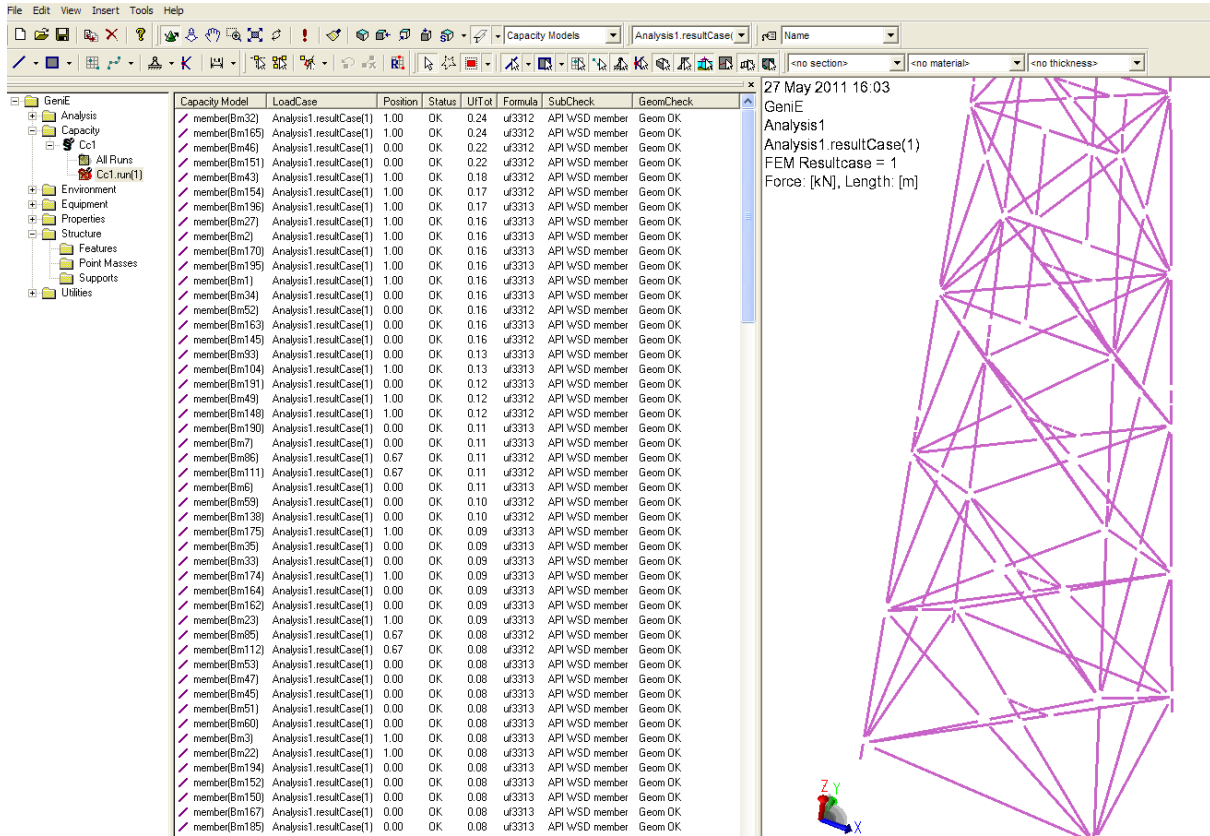
See results for Resultcase1. You will find that the results are updated.



Make a cross check by reading in the results in Xtract by opening *R10.SIN* file in the last activity in the workflow.



Inside the GeniE workspace, you may proceed with Codechecking. *API WSD 2005* is selected as the Codecheck standard. All the 14 resultcases are part of the Codecheck run.



The screenshot displays the GeniE software interface. On the left is a tree view showing the project structure: GeniE, Analysis, Capacity, Cc1, All Runs, Cc1.run(1), Environment, Equipment, Properties, Structure, Features, Point Masses, Supports, and Utilities. The main window is divided into a table of analysis results and a 3D model of a structure.

Capacity Model	LoadCase	Position	Status	UITot	Formula	SubCheck	GeomCheck
member(Bm32)	Analysis1.resultCase(1)	1.00	OK	0.24	u3312	API WSD member	Geom OK
member(Bm165)	Analysis1.resultCase(1)	1.00	OK	0.24	u3312	API WSD member	Geom OK
member(Bm46)	Analysis1.resultCase(1)	0.00	OK	0.22	u3312	API WSD member	Geom OK
member(Bm151)	Analysis1.resultCase(1)	0.00	OK	0.22	u3312	API WSD member	Geom OK
member(Bm43)	Analysis1.resultCase(1)	1.00	OK	0.18	u3312	API WSD member	Geom OK
member(Bm154)	Analysis1.resultCase(1)	1.00	OK	0.17	u3312	API WSD member	Geom OK
member(Bm196)	Analysis1.resultCase(1)	1.00	OK	0.17	u3313	API WSD member	Geom OK
member(Bm27)	Analysis1.resultCase(1)	1.00	OK	0.16	u3313	API WSD member	Geom OK
member(Bm2)	Analysis1.resultCase(1)	1.00	OK	0.16	u3313	API WSD member	Geom OK
member(Bm170)	Analysis1.resultCase(1)	1.00	OK	0.16	u3313	API WSD member	Geom OK
member(Bm195)	Analysis1.resultCase(1)	1.00	OK	0.16	u3313	API WSD member	Geom OK
member(Bm1)	Analysis1.resultCase(1)	1.00	OK	0.16	u3313	API WSD member	Geom OK
member(Bm34)	Analysis1.resultCase(1)	0.00	OK	0.16	u3313	API WSD member	Geom OK
member(Bm52)	Analysis1.resultCase(1)	0.00	OK	0.16	u3312	API WSD member	Geom OK
member(Bm163)	Analysis1.resultCase(1)	0.00	OK	0.16	u3312	API WSD member	Geom OK
member(Bm145)	Analysis1.resultCase(1)	0.00	OK	0.16	u3312	API WSD member	Geom OK
member(Bm53)	Analysis1.resultCase(1)	0.00	OK	0.13	u3313	API WSD member	Geom OK
member(Bm104)	Analysis1.resultCase(1)	1.00	OK	0.13	u3313	API WSD member	Geom OK
member(Bm191)	Analysis1.resultCase(1)	0.00	OK	0.12	u3313	API WSD member	Geom OK
member(Bm49)	Analysis1.resultCase(1)	1.00	OK	0.12	u3312	API WSD member	Geom OK
member(Bm148)	Analysis1.resultCase(1)	1.00	OK	0.12	u3312	API WSD member	Geom OK
member(Bm190)	Analysis1.resultCase(1)	0.00	OK	0.11	u3313	API WSD member	Geom OK
member(Bm7)	Analysis1.resultCase(1)	0.00	OK	0.11	u3312	API WSD member	Geom OK
member(Bm86)	Analysis1.resultCase(1)	0.67	OK	0.11	u3312	API WSD member	Geom OK
member(Bm111)	Analysis1.resultCase(1)	0.67	OK	0.11	u3312	API WSD member	Geom OK
member(Bm6)	Analysis1.resultCase(1)	0.00	OK	0.11	u3313	API WSD member	Geom OK
member(Bm59)	Analysis1.resultCase(1)	0.00	OK	0.10	u3312	API WSD member	Geom OK
member(Bm138)	Analysis1.resultCase(1)	0.00	OK	0.10	u3312	API WSD member	Geom OK
member(Bm175)	Analysis1.resultCase(1)	1.00	OK	0.09	u3313	API WSD member	Geom OK
member(Bm35)	Analysis1.resultCase(1)	0.00	OK	0.09	u3313	API WSD member	Geom OK
member(Bm33)	Analysis1.resultCase(1)	0.00	OK	0.09	u3313	API WSD member	Geom OK
member(Bm174)	Analysis1.resultCase(1)	1.00	OK	0.09	u3313	API WSD member	Geom OK
member(Bm164)	Analysis1.resultCase(1)	0.00	OK	0.09	u3313	API WSD member	Geom OK
member(Bm162)	Analysis1.resultCase(1)	0.00	OK	0.09	u3313	API WSD member	Geom OK
member(Bm23)	Analysis1.resultCase(1)	1.00	OK	0.09	u3313	API WSD member	Geom OK
member(Bm85)	Analysis1.resultCase(1)	0.67	OK	0.08	u3312	API WSD member	Geom OK
member(Bm112)	Analysis1.resultCase(1)	0.67	OK	0.08	u3312	API WSD member	Geom OK
member(Bm53)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm47)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm45)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm51)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm60)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm3)	Analysis1.resultCase(1)	1.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm22)	Analysis1.resultCase(1)	1.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm194)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm152)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm150)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm167)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK
member(Bm185)	Analysis1.resultCase(1)	0.00	OK	0.08	u3313	API WSD member	Geom OK

On the right, a 3D model of a structure is shown, rendered in purple. A coordinate system (X, Y, Z) is visible at the bottom left of the model. Text on the right side of the interface indicates the date and time: 27 May 2011 16:03, and the analysis name: GeniE, Analysis1, Analysis1.resultCase(1), FEM Resultcase = 1, Force: [kN], Length: [m].