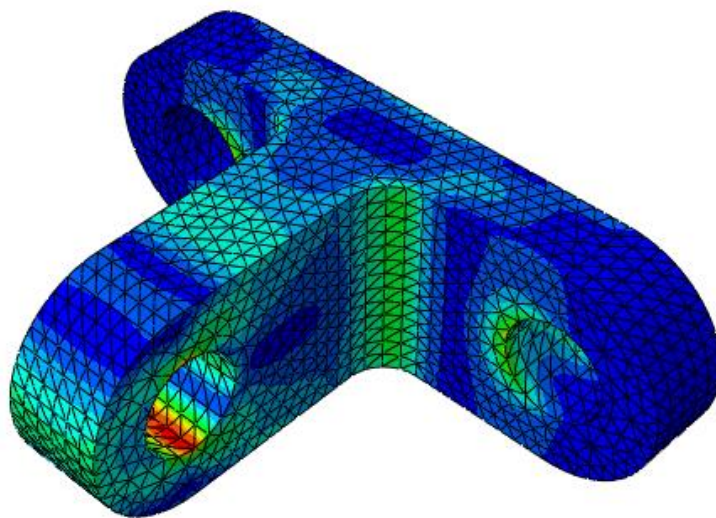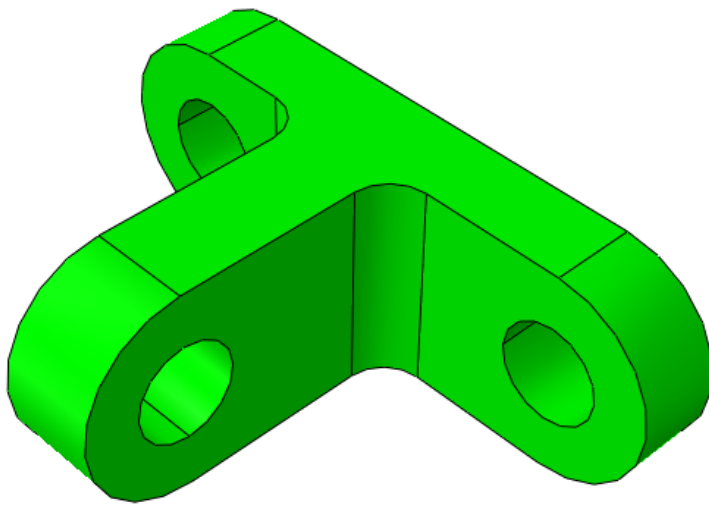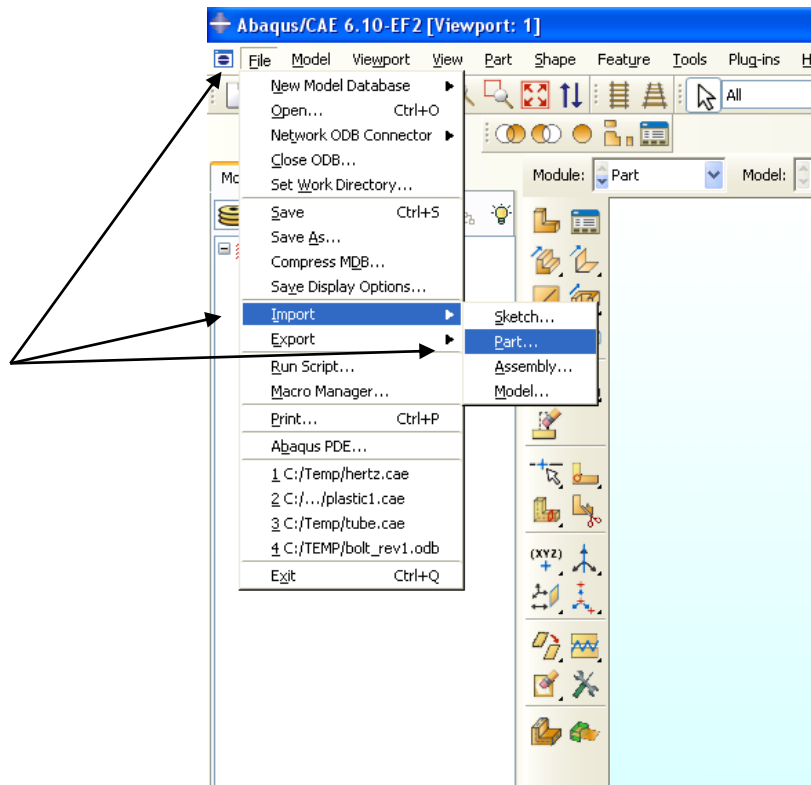# Tutorial 1 (Basic):
Simple Bracket
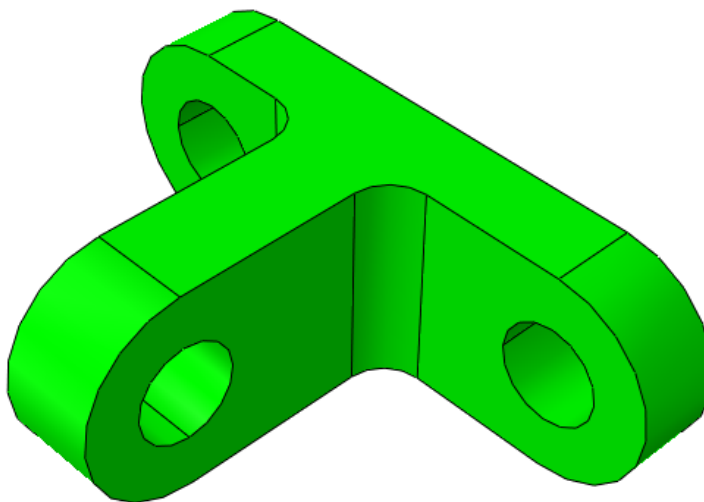
Laurence Marks

1. Geometry Import

Import the geometry in the form of a step file – Tutorial 1.stp

Accept all default settings.

This reads the AP203 file of the geometry as shown below.

2. Material and section properties

Start to define the material properties – go into the properties module.

Abaqus does not handle the units for us, so we need to take this into consideration. For example, if our geometry is in mm and our force in Newtons then our stresses will be reported in MPa.
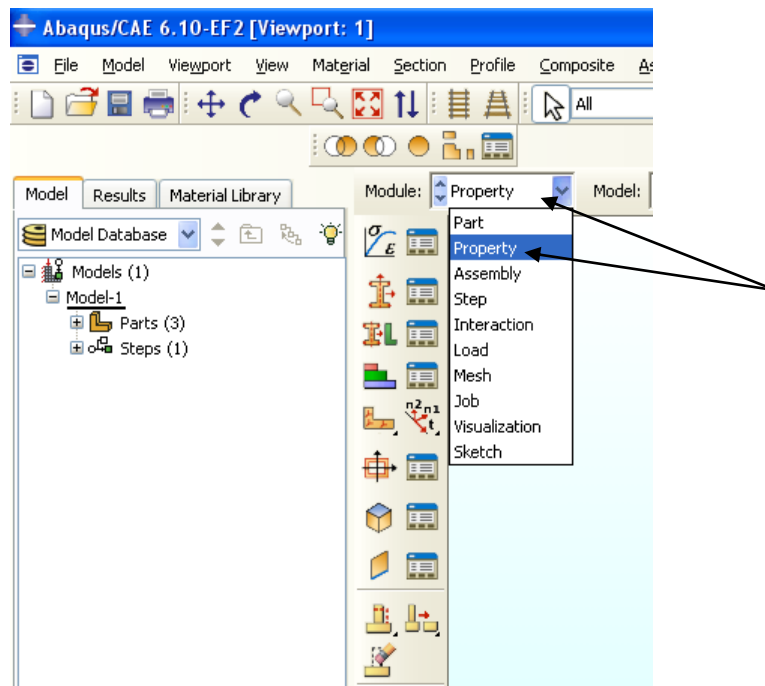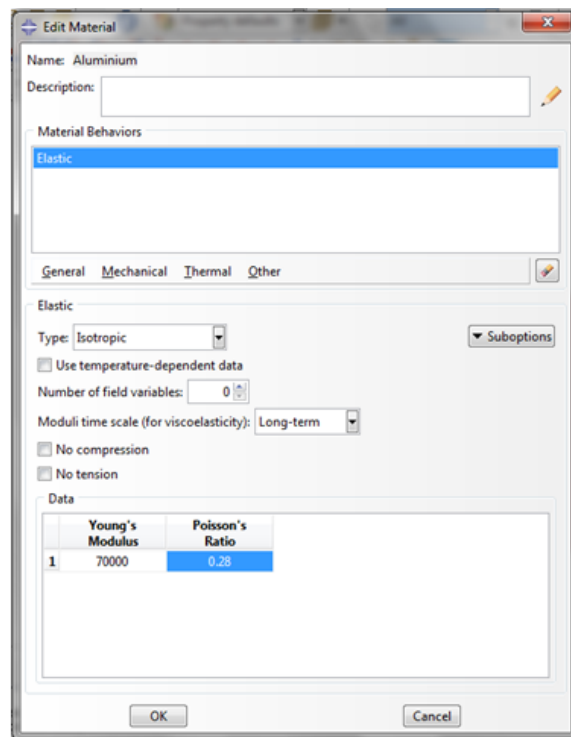

There are many options for material models, types and data, we are simply going to define the most basic sort required for a linear run.

Define some properties for aluminium...
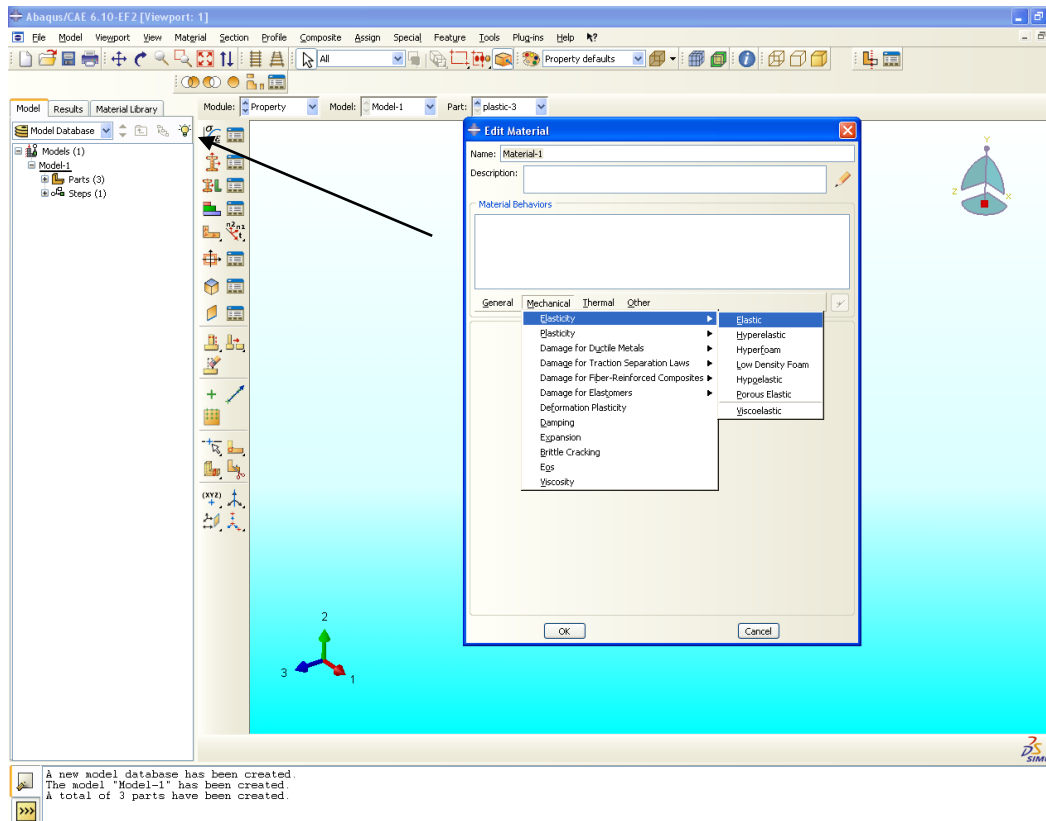
We define a modulus of 70e3 (Newton and mm units – remember to be consistent). Also define a Poisson's ratio of 0.28.
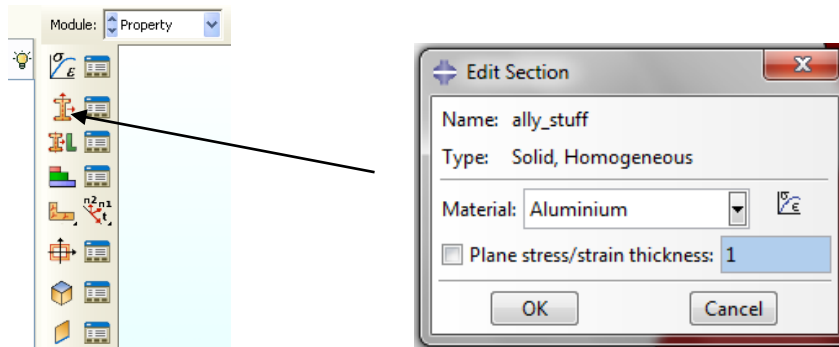
We now need to define a section. With a simple material definition and a solid element, this often seems like an unnecessary step, but it comes into its own with more complex models.

Create a solid homogenous section and apply the recently created aluminium material properties to it.

This section then needs to be applied to the geometry. This is how we tell the software which parts are made of what.

Click the assign section button, select the geometry in the viewport and apply the recently created section to it.

In Abaqus there is no such thing as a part on its own – it must always exist as part of an assembly.

Go into the assembly module to instance the part (basically put it into an assembly). Select the part to add it to the assembly and click OK.

Simuleon B.V.
Sint Antoniestraat 7 5314 LG Bruchem
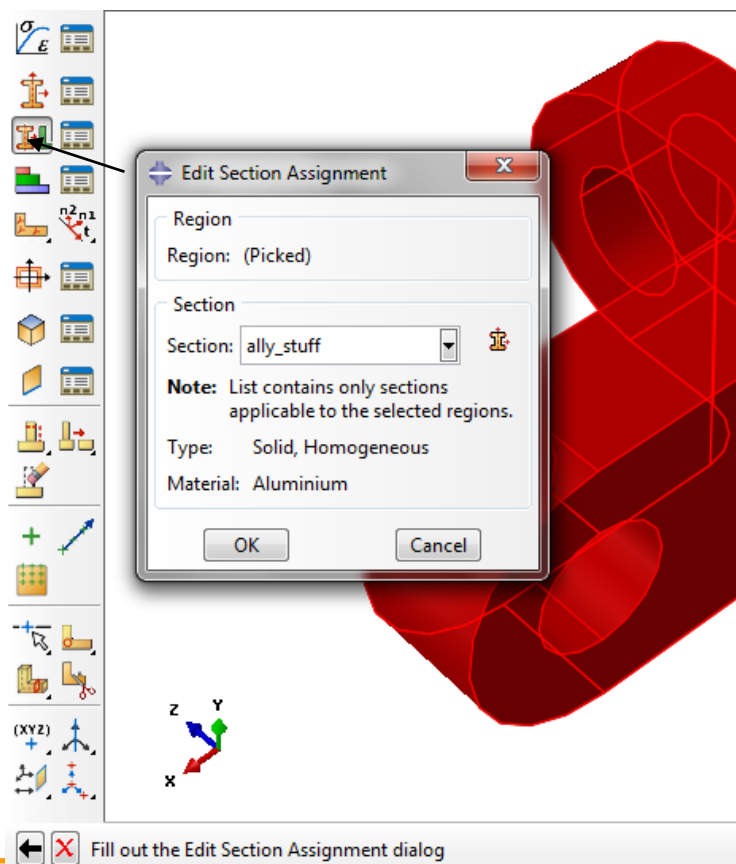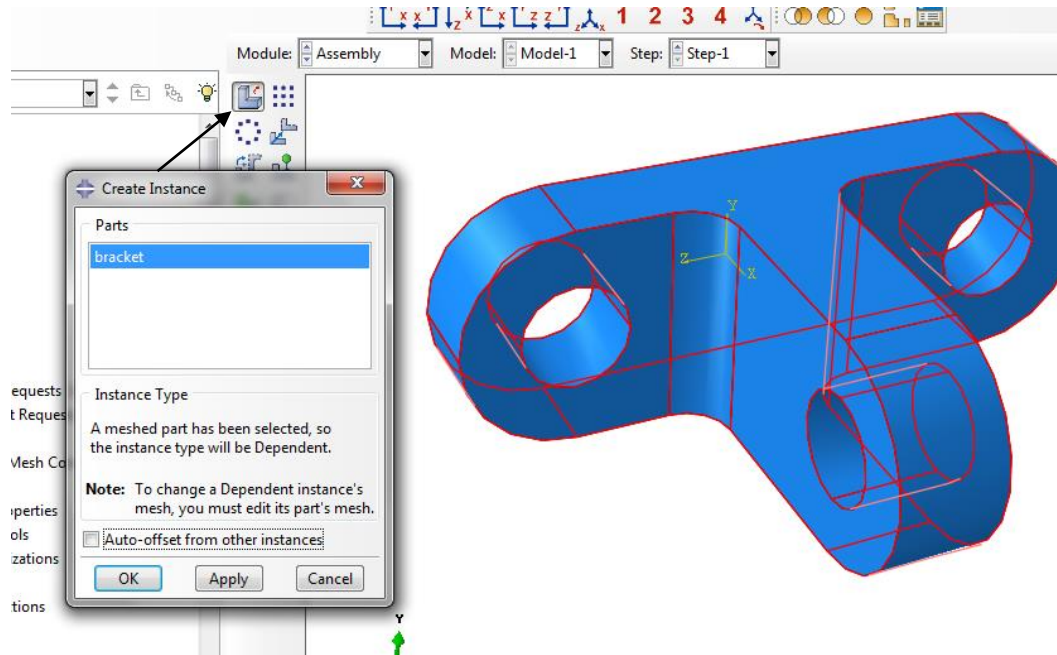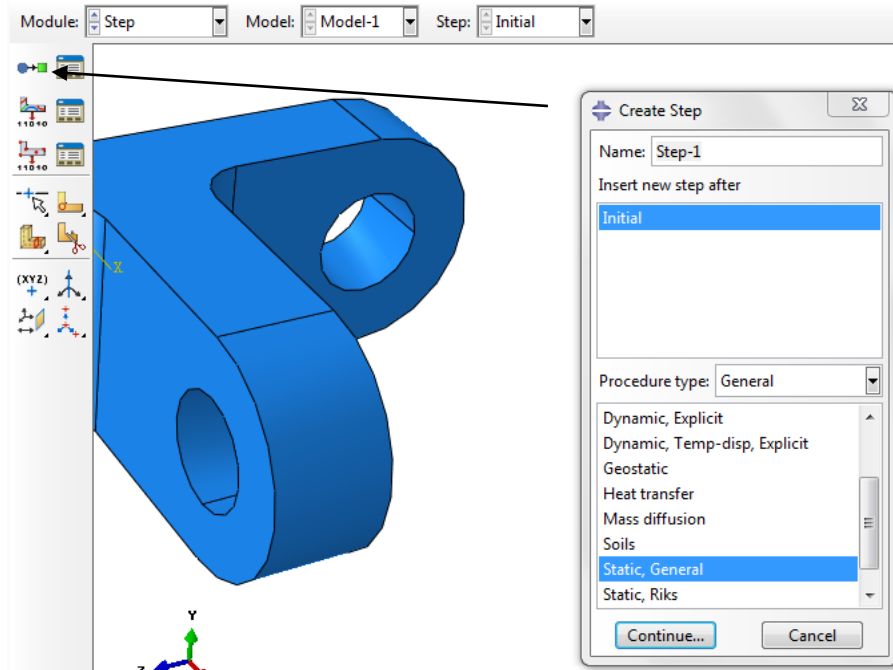T. +31(0)418-644699 F. +31(0)418-644690 E. info@simuleon.nl W. www.simuleon.nl

6

3. Define the first analysis step

Abaqus has a wide range of solution types and options – we are going to define the simplest, a single step static solution.

Go to the step module, create a new Static - General step and click continue.

For now we will accept the default settings. This will solve the entire analysis in a single step. This is possible for a very basic model like this, but more complex analyses will require hundreds or even thousands of time steps to reach a solution. Click OK.
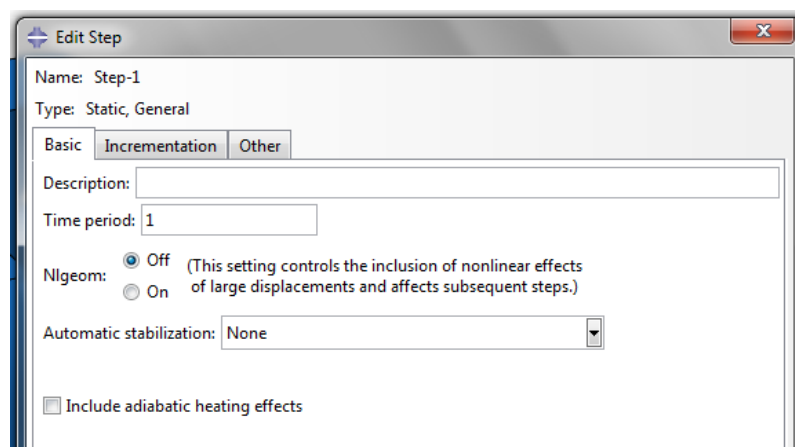
4.   Loads and restraints

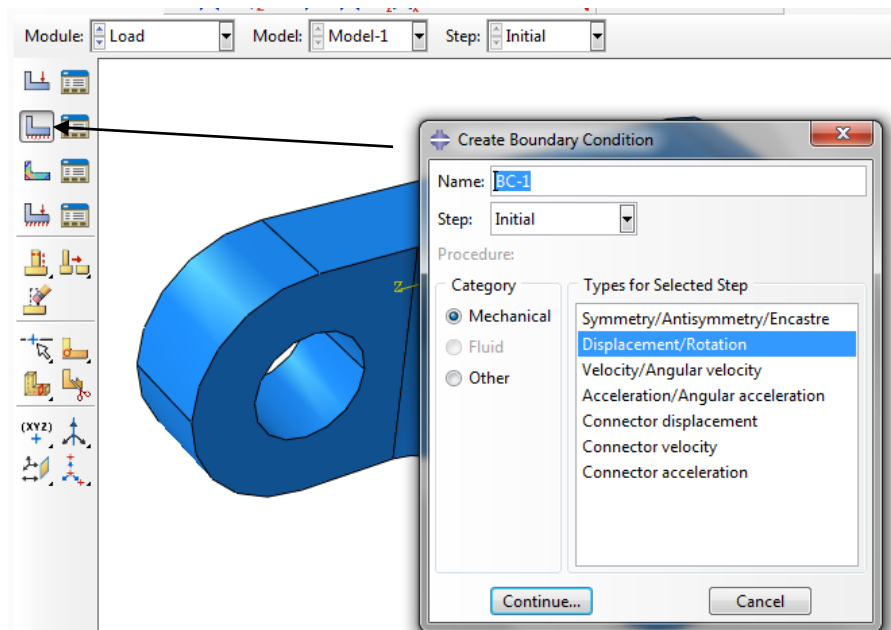As there are no interactions in this model, we jump straight to the load module to define loads and restraints. Firstly we define a full fixing at the pair of holes on the back of the bracket.

In the load module select the Create Boundary Condition option and choose Displacement/Rotation.

Select all the internal faces on the bore holes (hold down shift to select multiple faces), when all faces have been selected click Done.

In the Edit Boundary Condition window that appears check all three translational degrees of freedom and then click Ok.
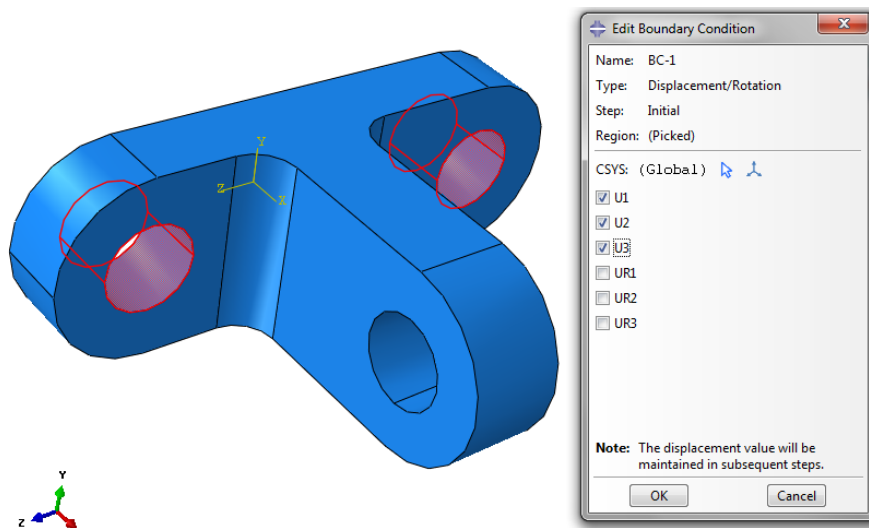
Now we define a load in the form of a surface traction – this is a force per unit area in a defined direction.
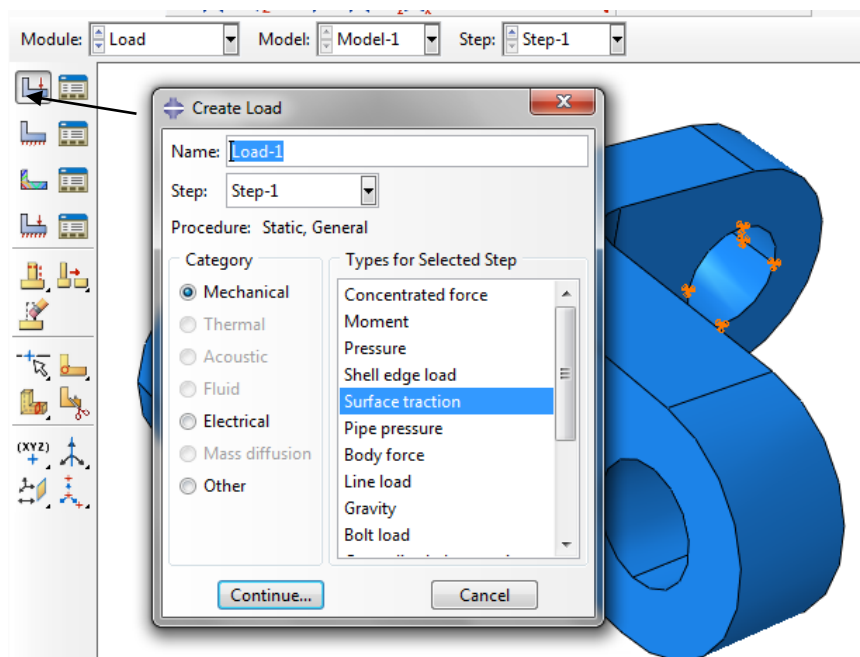
Select the Create Load option and choose Surface Traction from the type list.

Using the same method as before select the face highlighted in the image below and then enter a magnitude of 1 and set the traction type to general in the Edit Load window.

Simuleon B.V.
Sint Antoniestraat 7 5314 LG Bruchem
T. +31(0)418-644699 F. +31(0)418-644690 E. info@simuleon.nl W. www.simuleon.nl

9
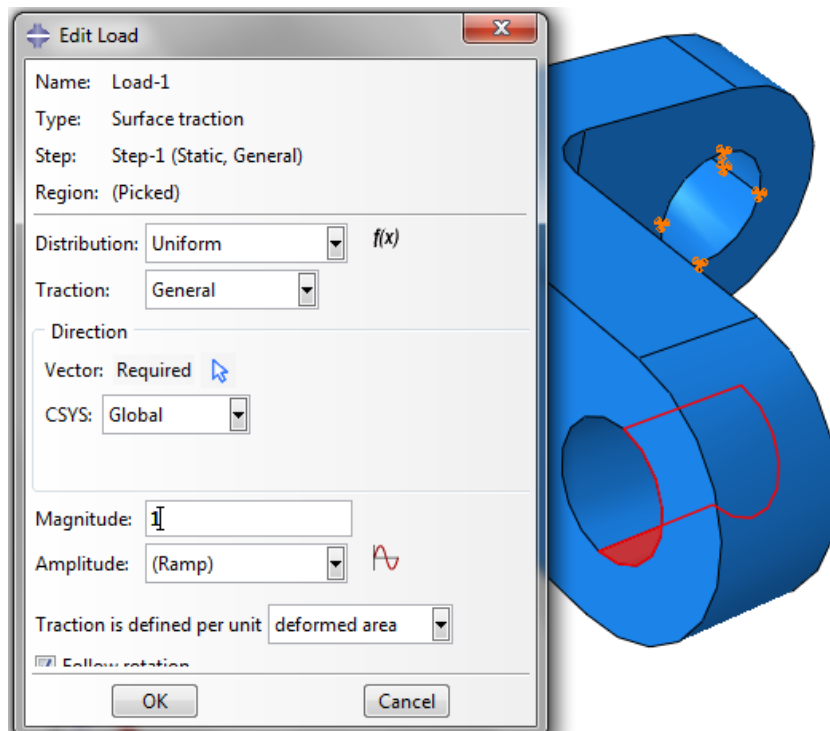
With a surface tration we need to specify a direction in which the force should act.

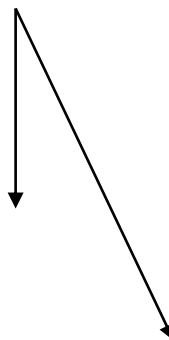In the Edit Load window click the arrow located next to Vector and in the viewport select the two points highlighted below.

Note that it is important in which order the points are selected as this denotes the direction in which the force will act. We want the force to be pulling on the bracket as if it was bolted to a wall supporting a rope.
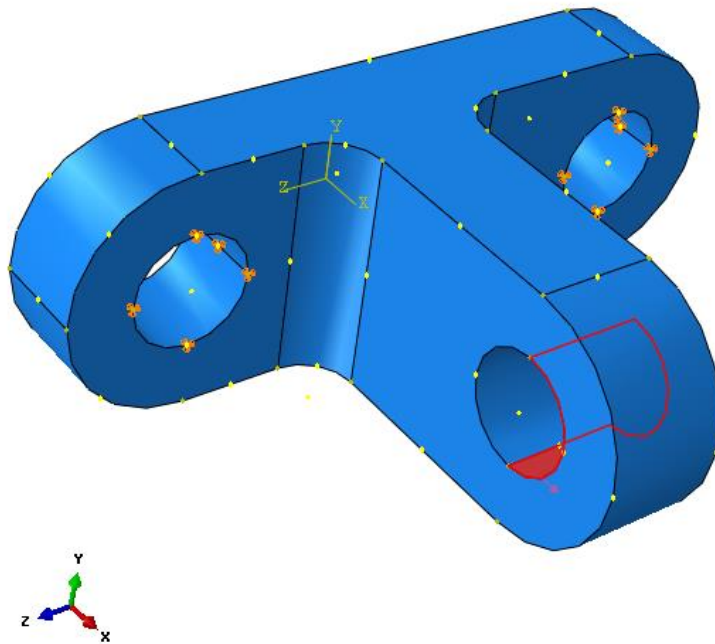
Now we have created a 1N surface traction in the X-axis direction.

5.   Meshing

The meshing module's function is self explanatory. By default Abaqus CAE will try and mesh 3D parts using brick elements, however this only works for a subset of possible geometries.

Firstly define meshing seeds – these control the mesh size and show how the mesh will appear without actually meshing the part. This is done by clicking the 'Seed Part Instance' option. Assign a global size of 2 (remember this is in mm).

If you instanced the geometry as 'dependant' you will need to view the object as an individual part before a mesh can be applied to it. This may seem unnecessary but in large assemblies the relevance of this functionality becomes obvious.
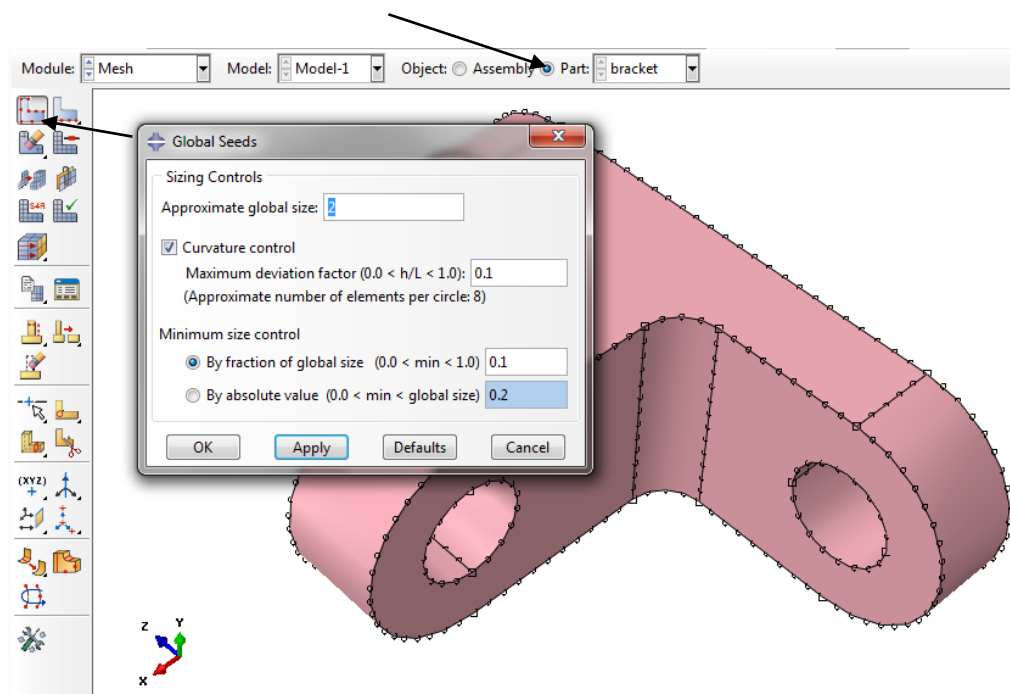
Next we define a meshing approach. Click the 'Assign Mesh Control' option, select a 'Tet' mesh and accept all the other defaults.
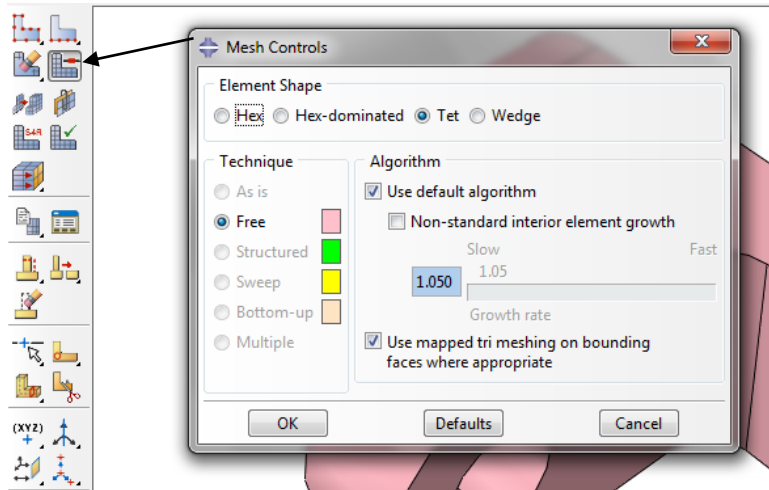
Click the 'Mesh Part' option to apply the mesh options to the part. The viewport will automatically update to display the meshed part.
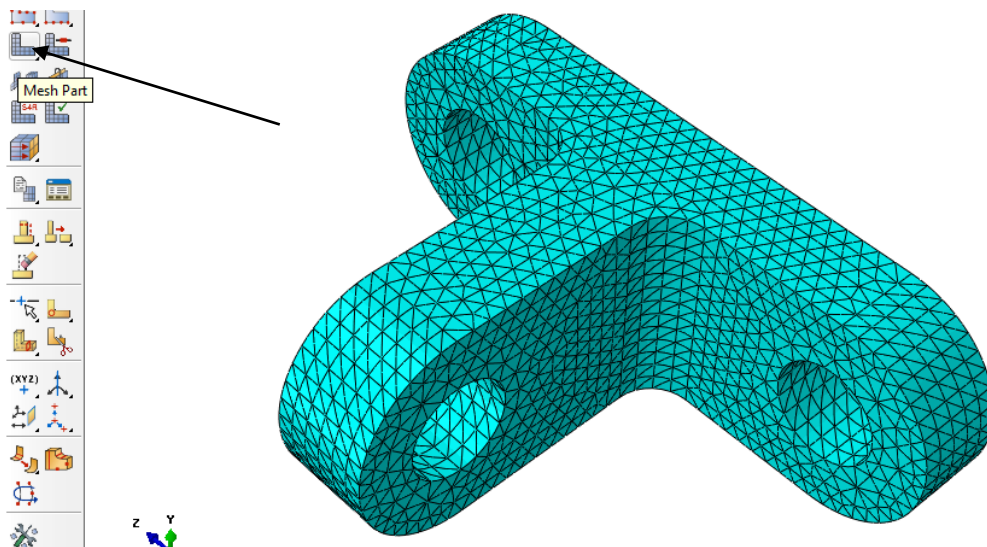
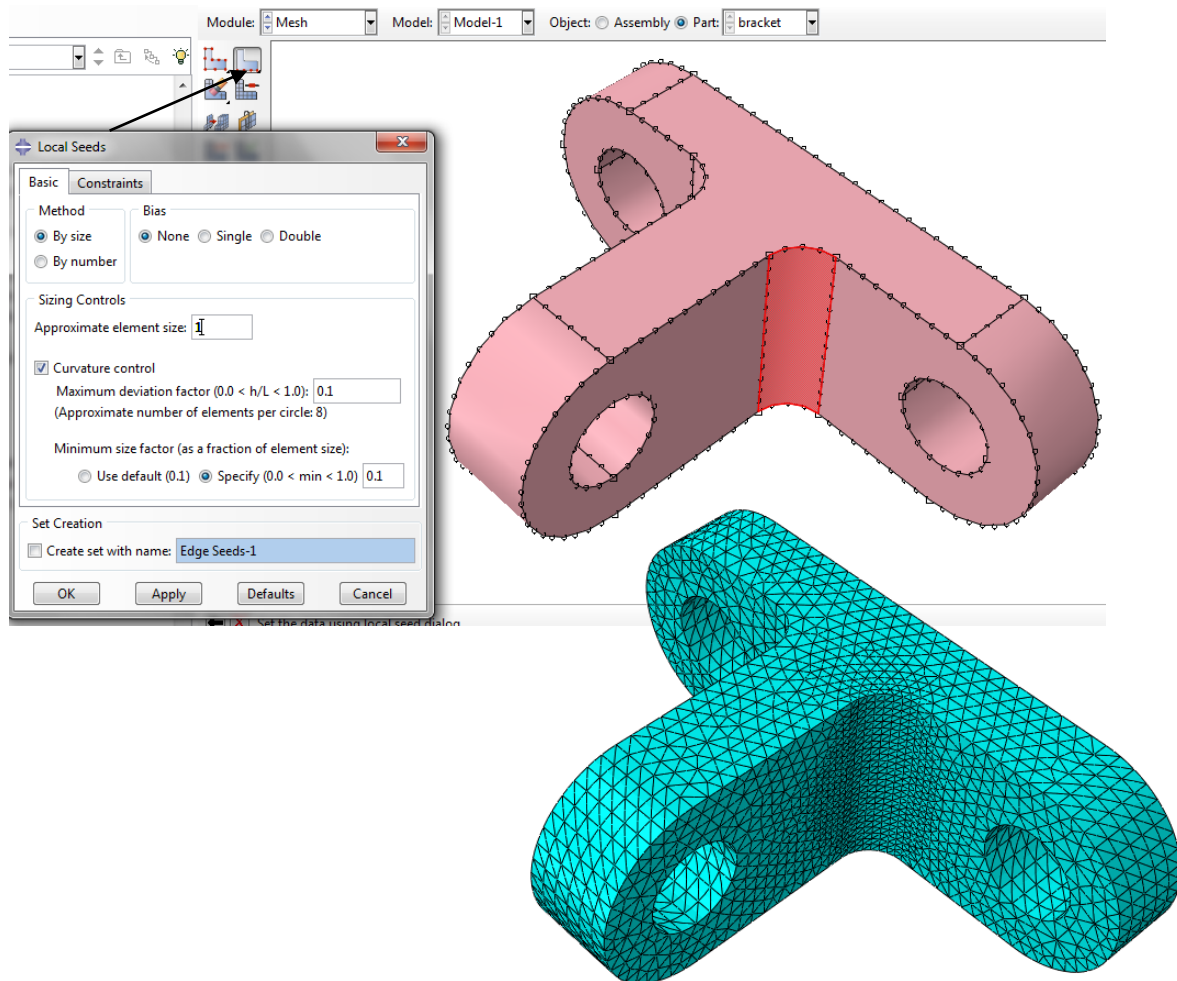If necessary the mesh can be locally refined using local mesh seeds. Before a local mesh seed can be applied we need to delete the current mesh.

Click and hold the icon used to mesh the part in order to view the optioned nested beneath. Scroll along and select 'Delete Instance Native Mesh'.

Select the two fillet faces in the model, select the 'Seed Edges' option and apply a local seed of 1mm. Now re-mesh the part.

Simuleon B.V.
Sint Antoniestraat 7 5314 LG Bruchem
T. +31(0)418-644699 F. +31(0)418-644690 E. info@simuleon.nl W. www.simuleon.nl

14

6.   Create a job and then submit it

Finally we move on to the job module. Within this module we will create a run for the analysis and watch its progress.

Select the 'Create Job' option, name the job and select the model you just created.

IMPORTANT: Do not accept the default name of 'Job-1'. Two jobs of the same name in the same folder will overwrite each other. It is good practice to name the job something specific to the analysis.

The job will now appear underneath the 'Job' section in the feature tree. To run this job right click and select 'Submit'.
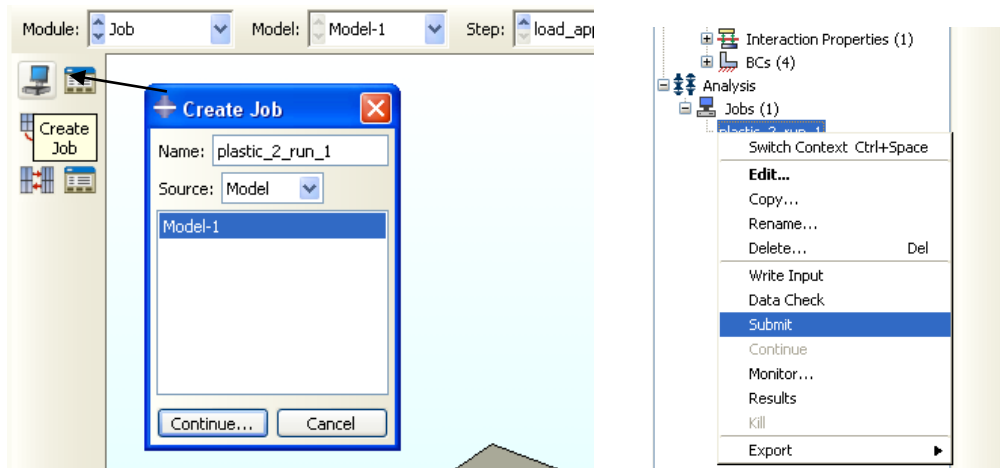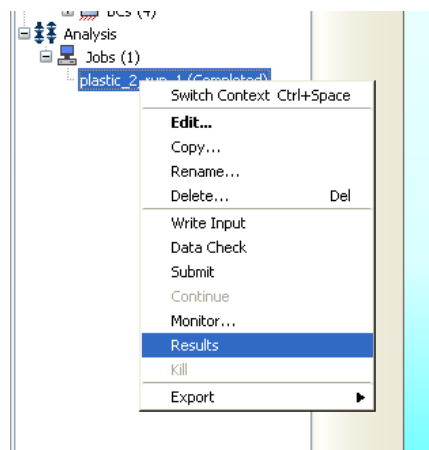

Whilst the solution is running we can monitor its progress. Right click the job and select 'Monitor'.
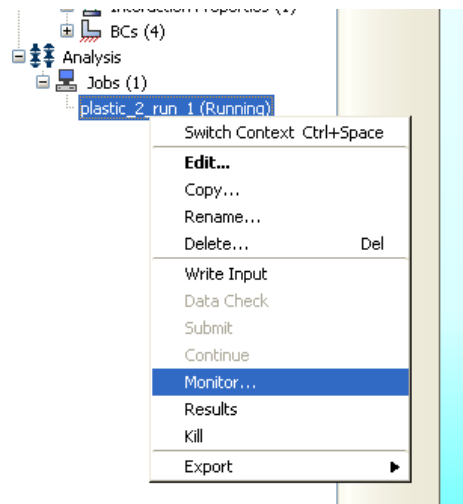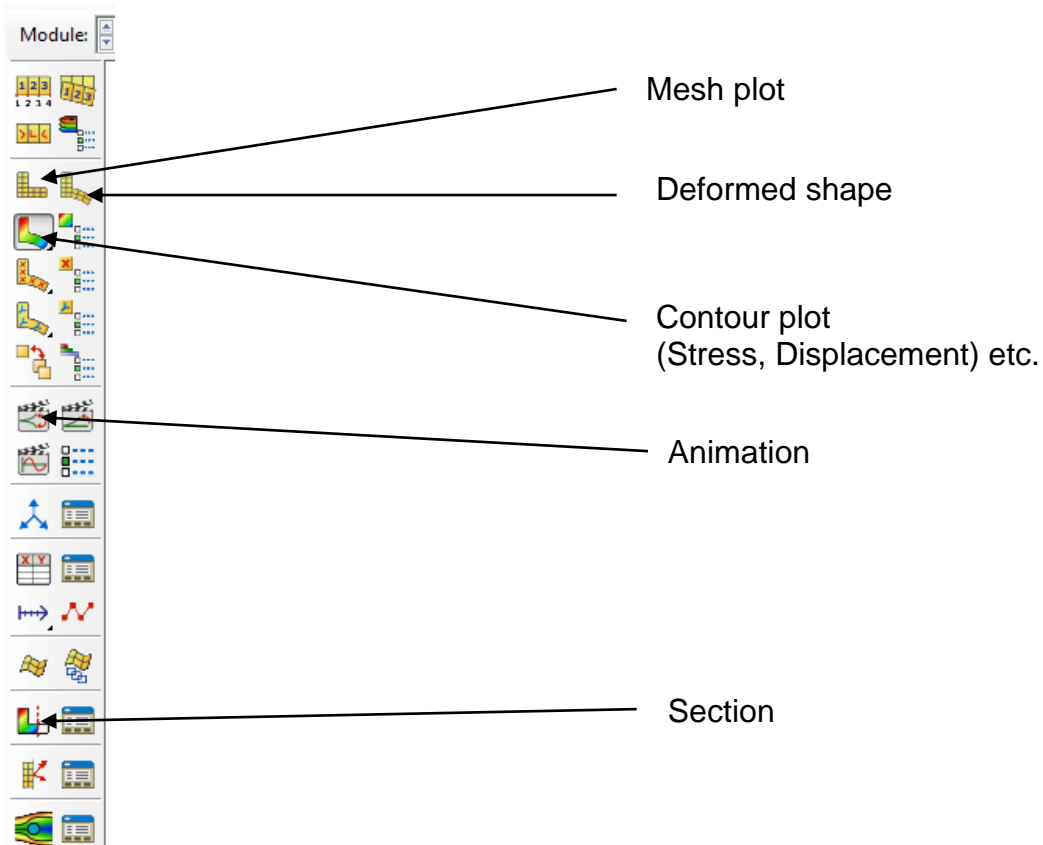
## 7.   Open the results file and plot the results

Once the job has completed we can view the results file. Right click the job again and select 'Results'.

CAE has a second feature tree for results processing and another set of menus.

Module:

Mesh plot

Deformed shape

Contour plot
(Stress, Displacement) etc.

Animation

Section

Simuleon B.V.
Sint Antoniestraat 7 5314 LG Bruchem
T. +31(0)418-644699 F. +31(0)418-644690 E. info@simuleon.nl W. www.simuleon.nl

17

Now we wish to create a stress plot of the model. By selecting the 'Results' -> 'Field Output' from the menu bar we can select different post processing quantities to plot on the model. We want a Von Mises Stress plot.
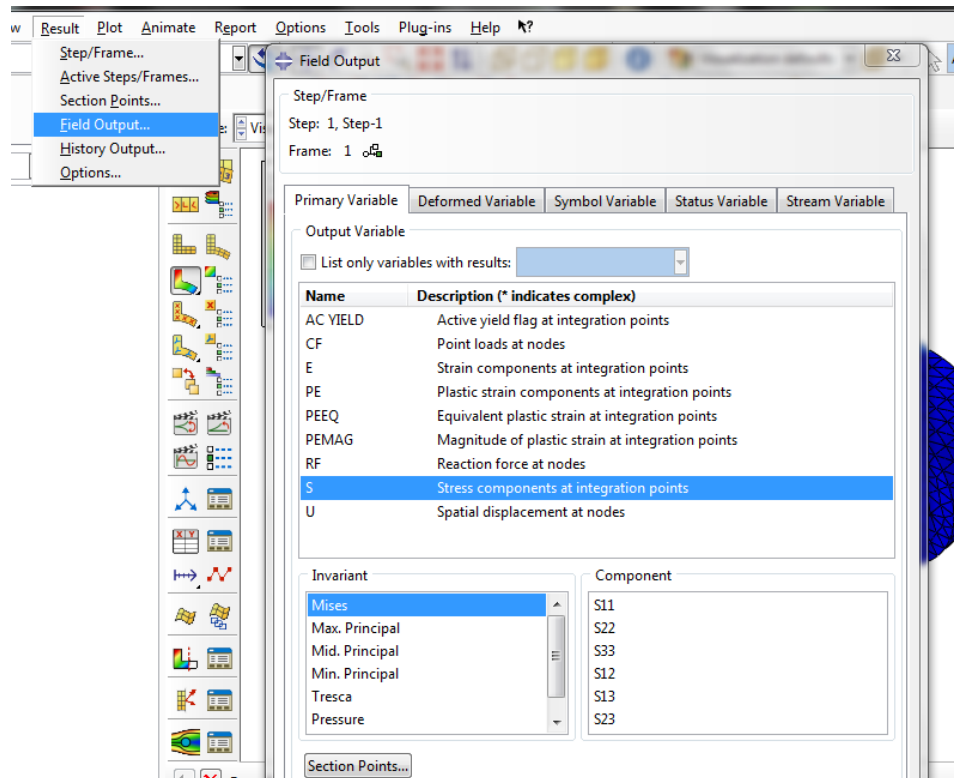
Once we have selected what we require the contour plot to show we can apply it to the model by selecting the 'Plot Contours' option.
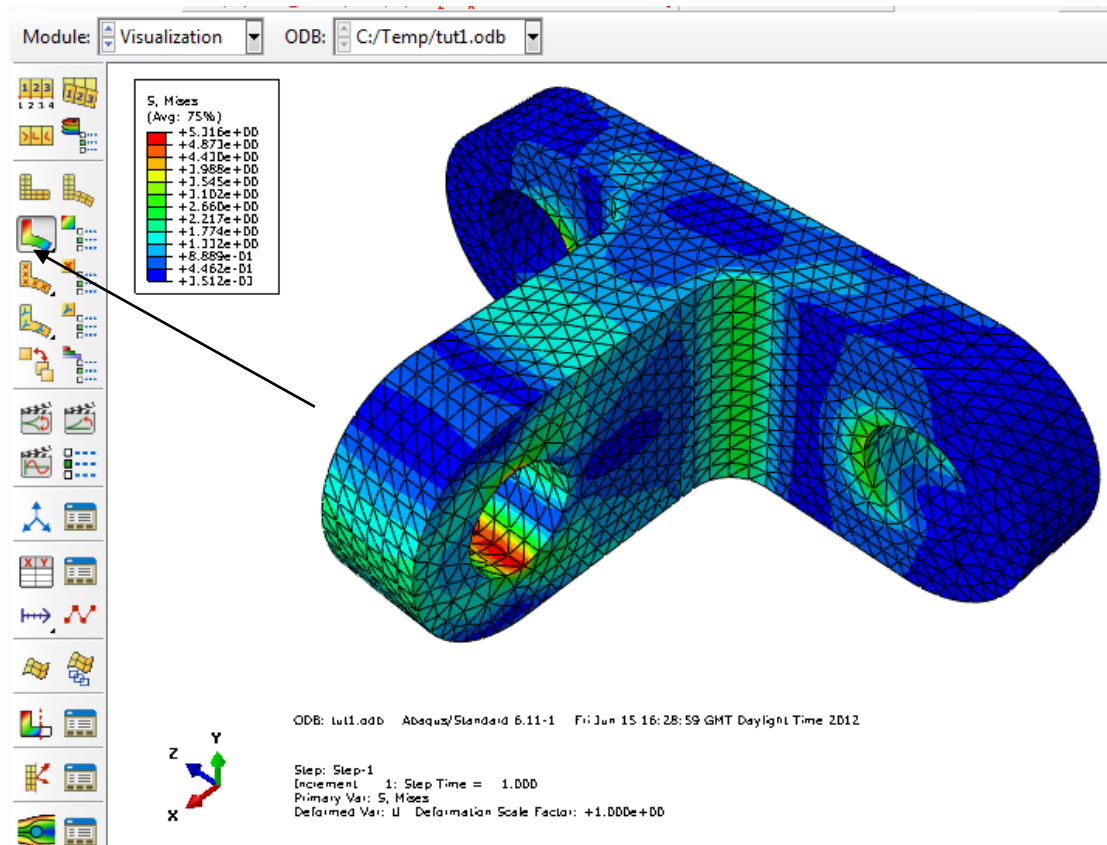
Take time to explore the contour plot post-processing options, try creating a displacement plot for the model.

Simuleon B.V.
Sint Antoniestraat 7 5314 LG Bruchem
T. +31(0)418-644699 F. +31(0)418-644690 E. info@simuleon.nl W. www.simuleon.nl

19