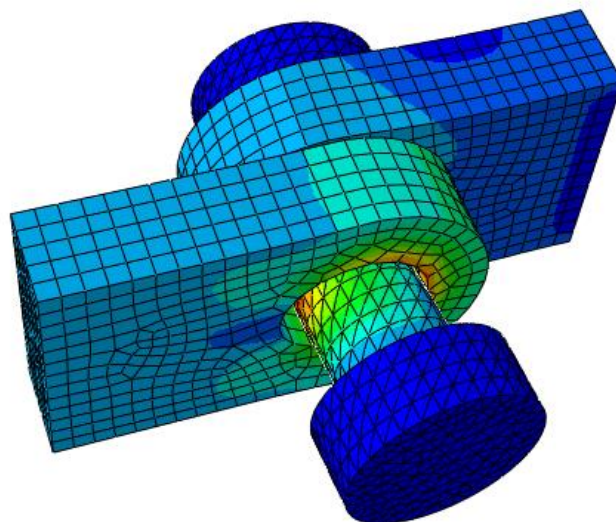
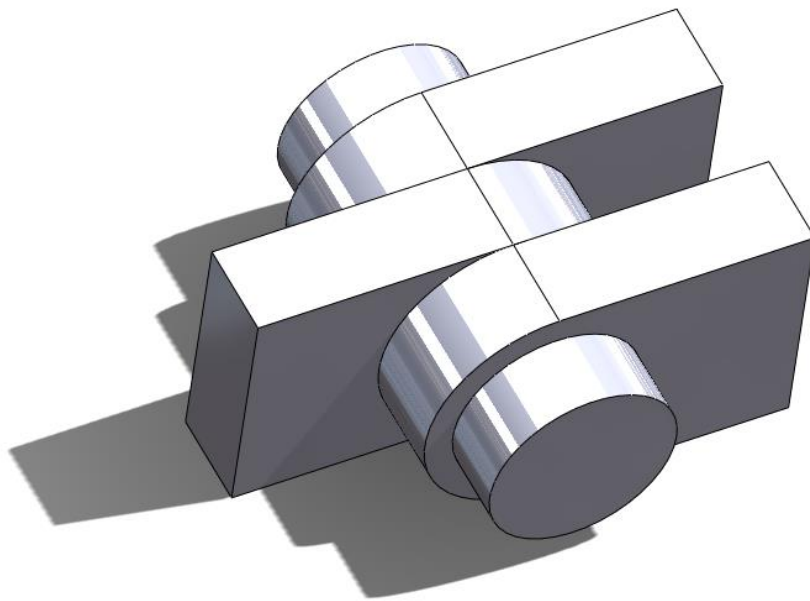


## Tutorial 2 (Basic): Pin/Lug Assembly

Laurence Marks

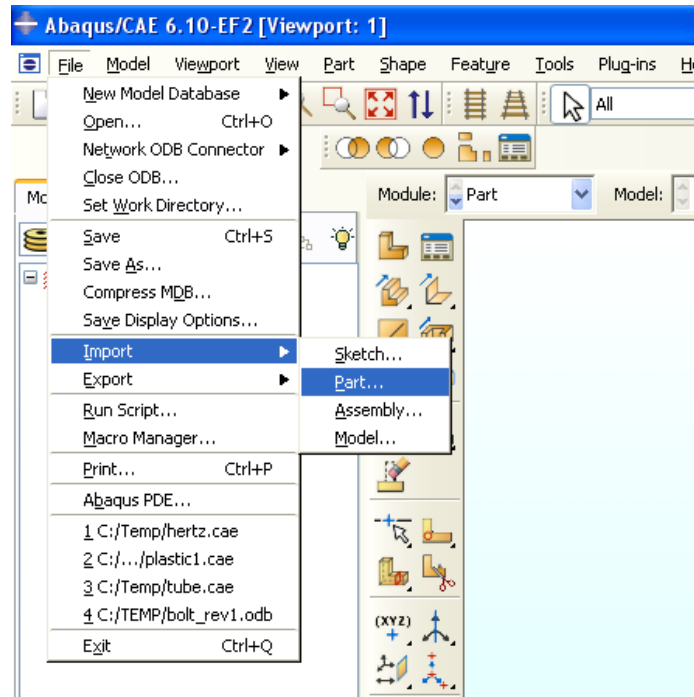
---



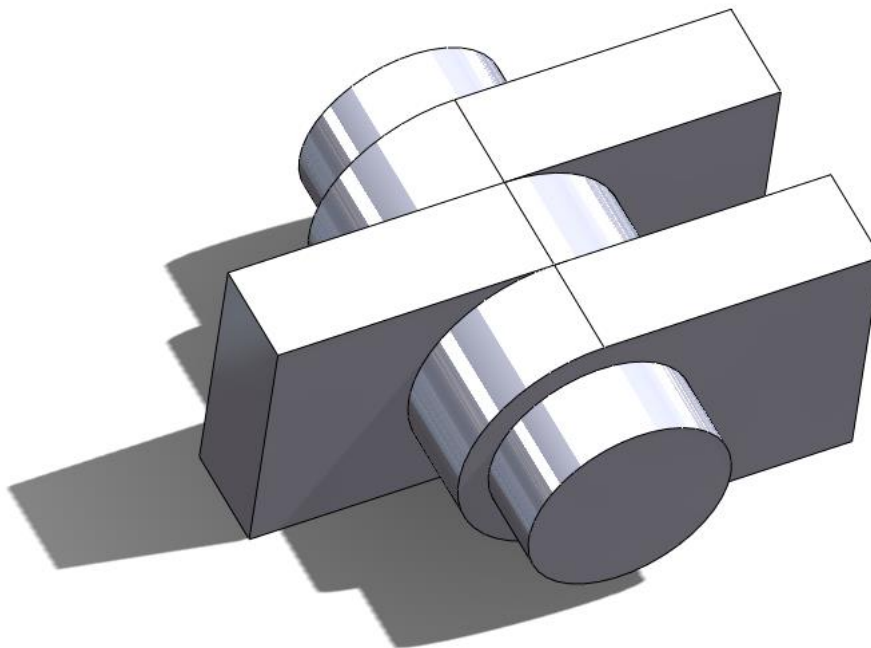
This tutorial introduces contact – you will need basic tutorial 1 on hand to cover some steps concerning materials and section definitions.

## 1. Geometry Import

Import the geometry in the form of a step file – Tutorial 2.stp



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



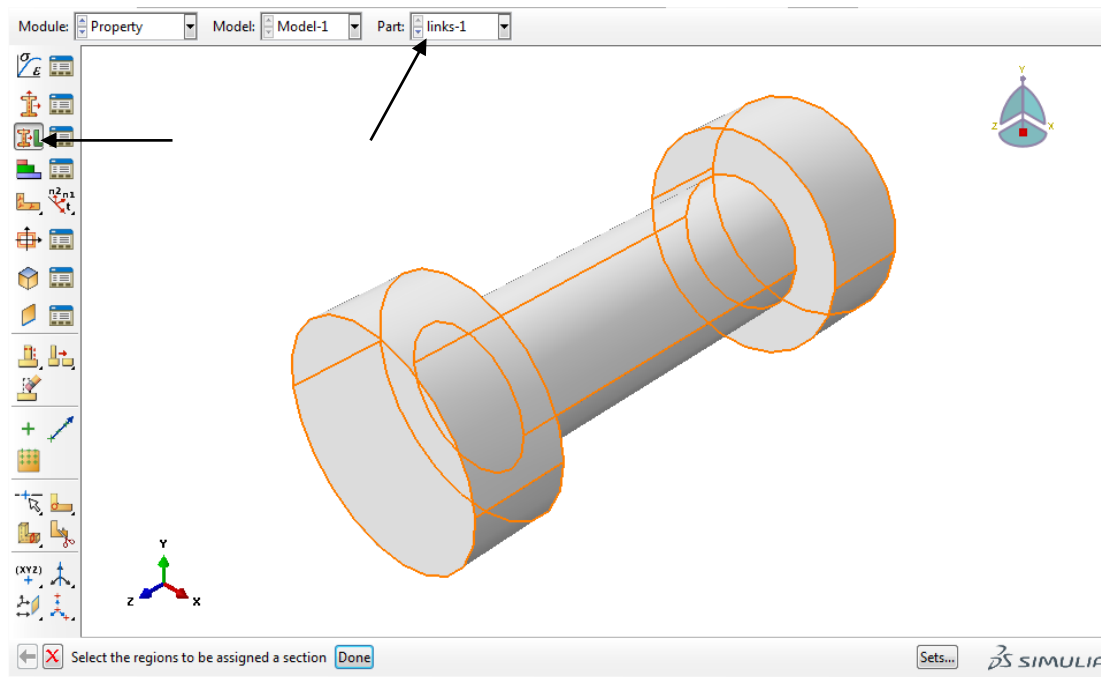
## 2. Material and section properties

Define 2 material properties

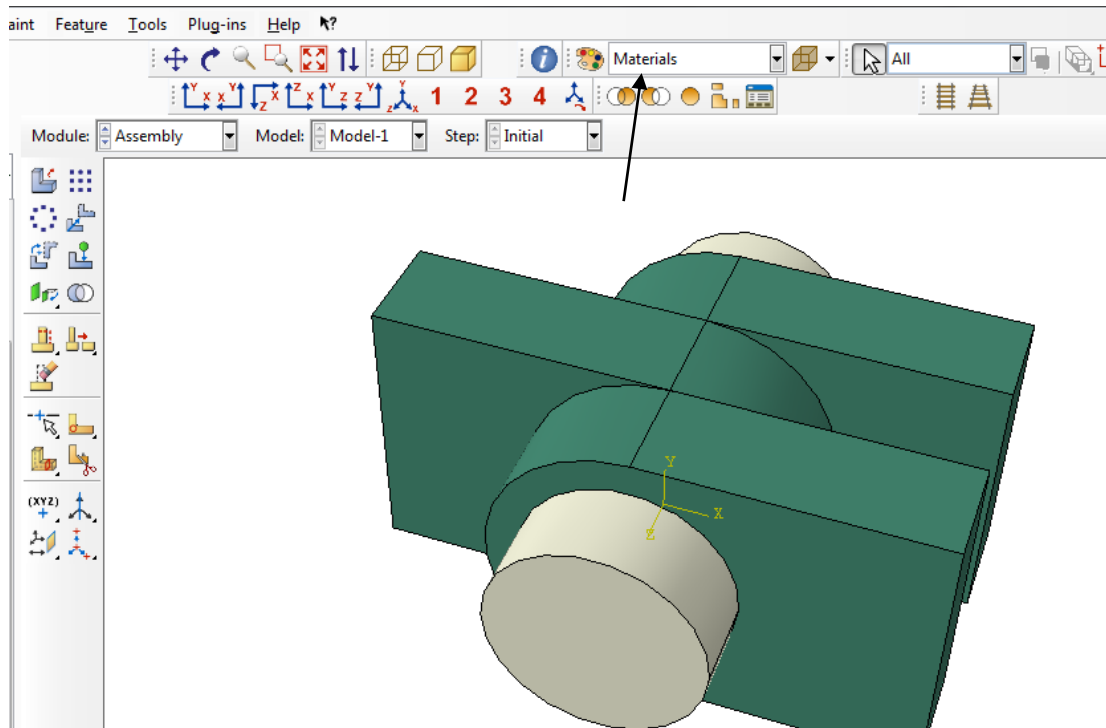
1. Aluminium – Modulus = 70e3N/mm2 and Poisson's Ratio = 0.3
2. Steel – Modulus = 210e3 N/mm2 and Poisson's Ratio = 0.3

Define 2 sections – Steel and Ally

Choose the assign section button from the vertical ribbon bar and choose which part is to be used using the drop-down part list at the top of the graphics area. Select the pin as the part and assign the Steel section to it. Repeat this process assigning Aluminium sections to the lugs.

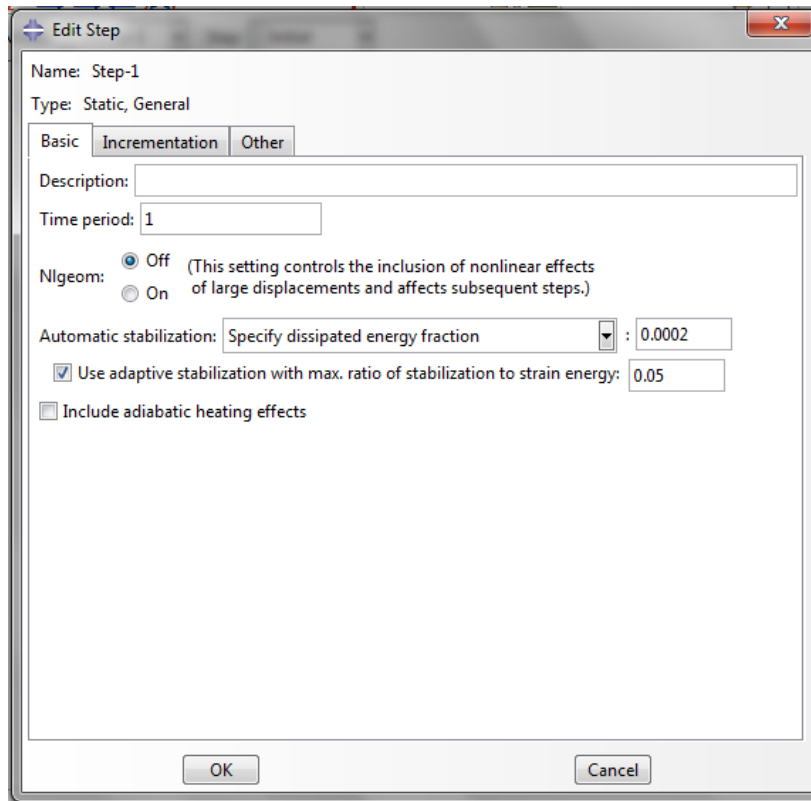


Create an assembly by instancing all the parts into the same assembly.



In the assembly module it is possible to colour code the model according to the material property applied to each part. Open up the drop down list highlighted and select 'Materials'. If no property has been to a part it will appear transparent.

### 3. Define the analysis step

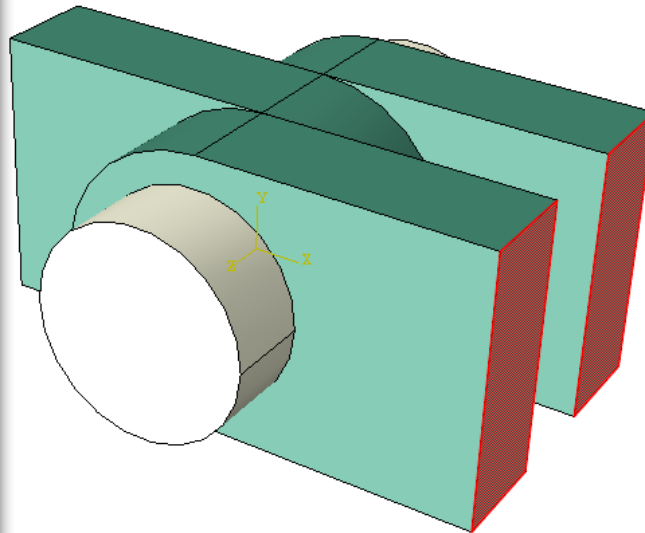
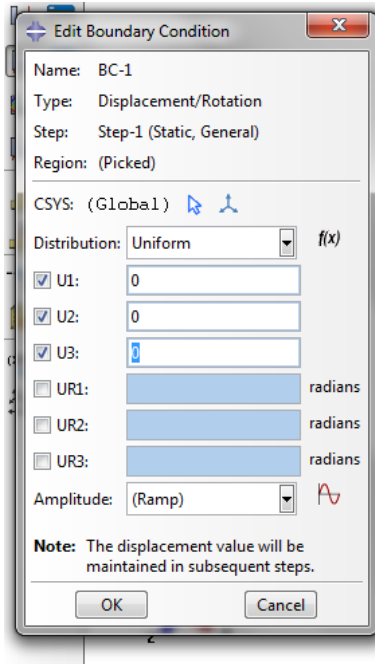


In this analysis we take all the defaults for a static general step, but we also include automatic stabilisation. It is important to gain an understanding of the automatic stabilisation function before applying it to your own model, but for now we'll assume that it's sensible.

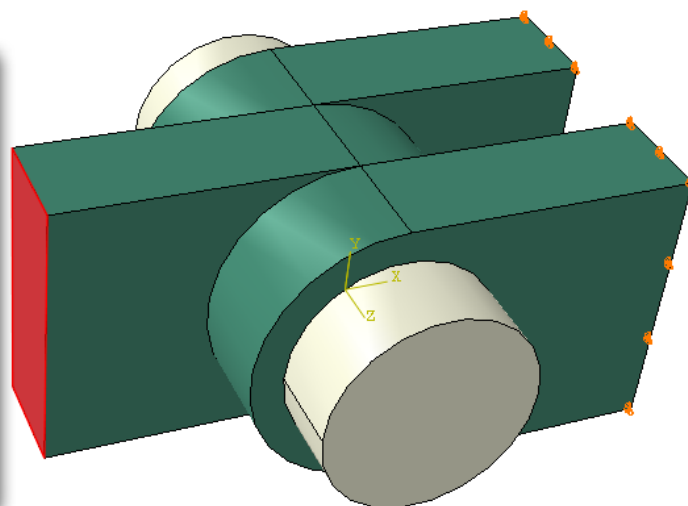
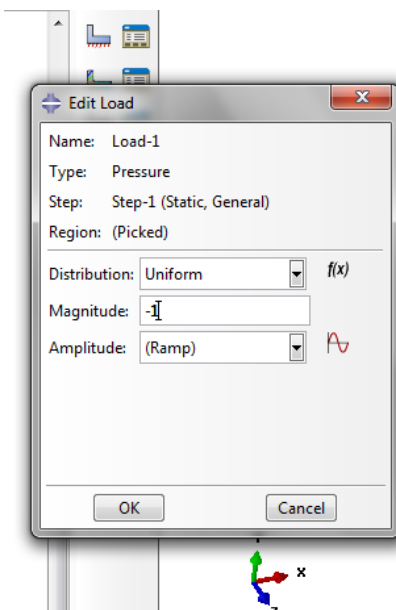
## 4. Loads and restraints

The loads and restraints are applied in a similar manner to the bracket model.

Fully fix the back faces of the lugs.

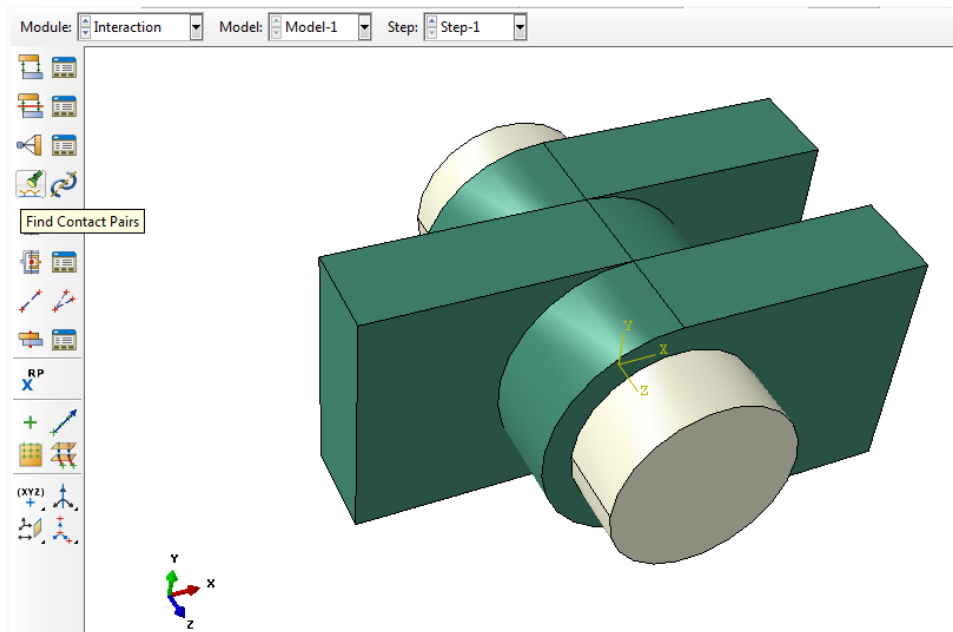


Apply a negative pressure of 1N/mm<sup>2</sup> to the other lug face.

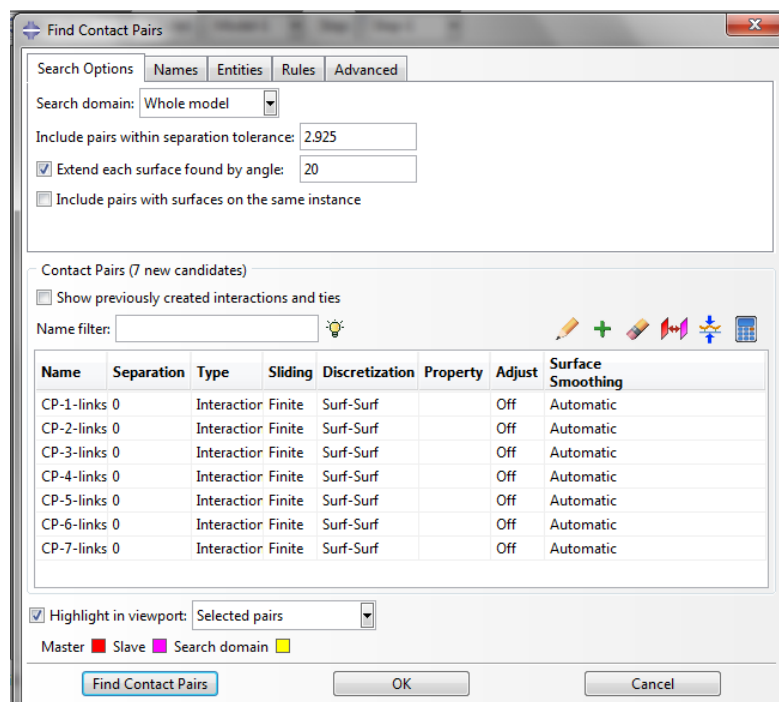


## 5. Contact Definitions

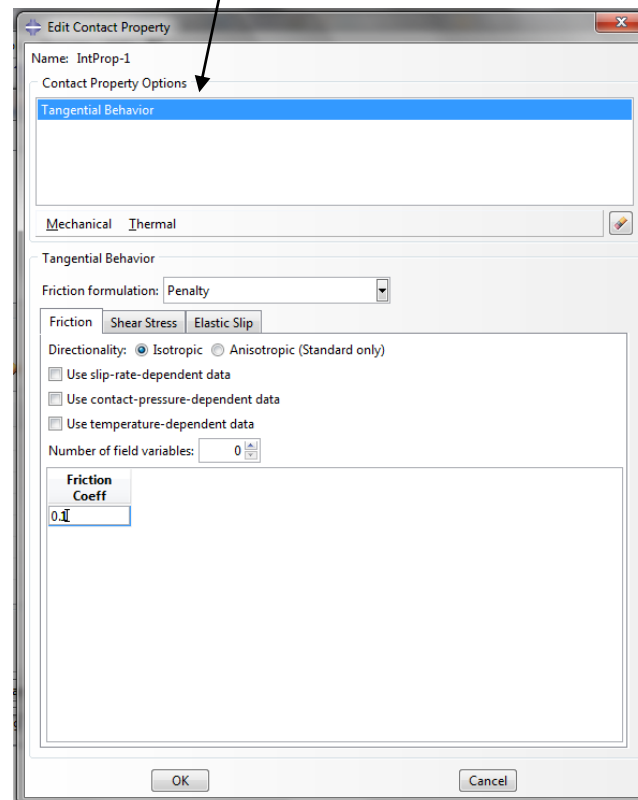
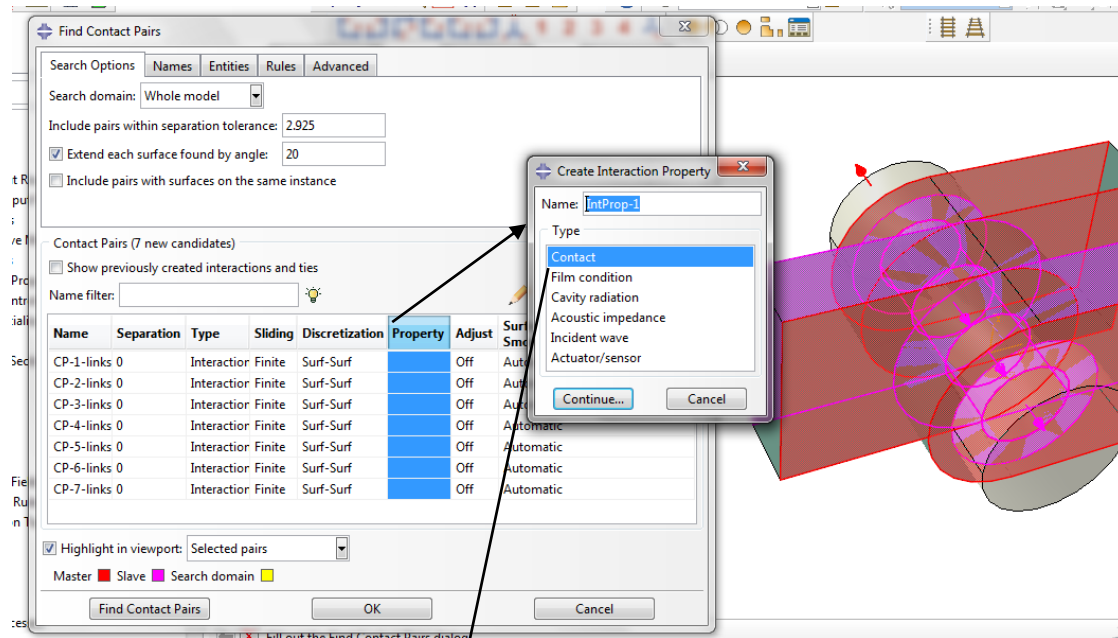
We shall now use a feature of Abaqus which automatically generates the contact definitions. Go to the interaction module and select the “find contact pairs” button.



If you take the defaults and click on find contact pairs the system finds 7 contact pairs we may be interested in. If we click down the list of names the contact surfaces are highlighted in the viewport window.



All the default settings are fine for an assembly in which all the overlapping faces are contact faces. We simply need to define a property for the contact which we do by double clicking the properties text in the table.



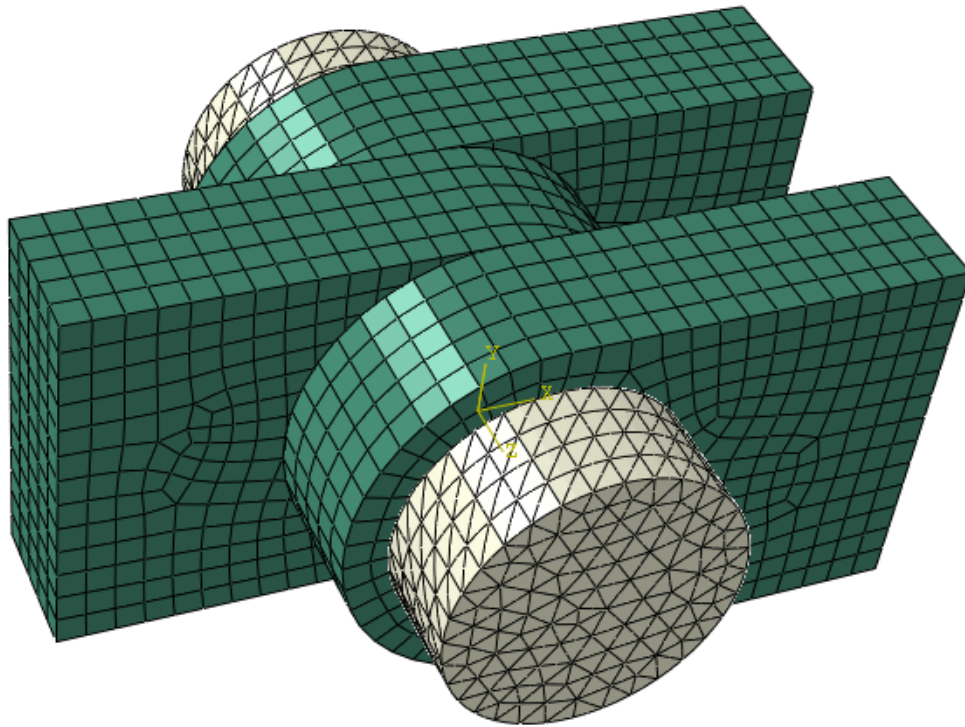
We will select tangential behaviour, a penalty method, and define a friction coefficient of 0.1.



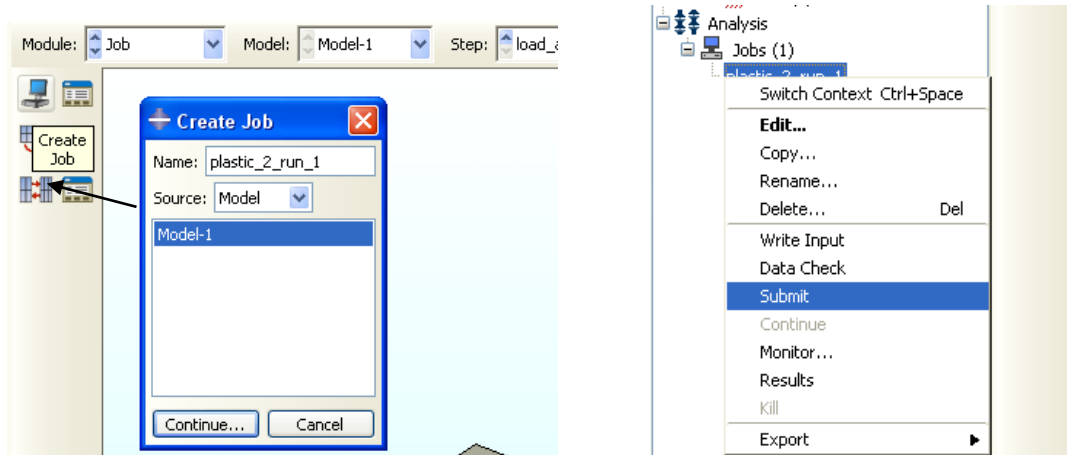
## 6. Meshing

We mesh the parts in exactly the same manner as we meshed the bracket. The lugs will mesh using a brick (hex) mesh approach and don't need the meshing approach modified. The pin (as it stands at this stage) needs to be meshed using tetrahedrons.

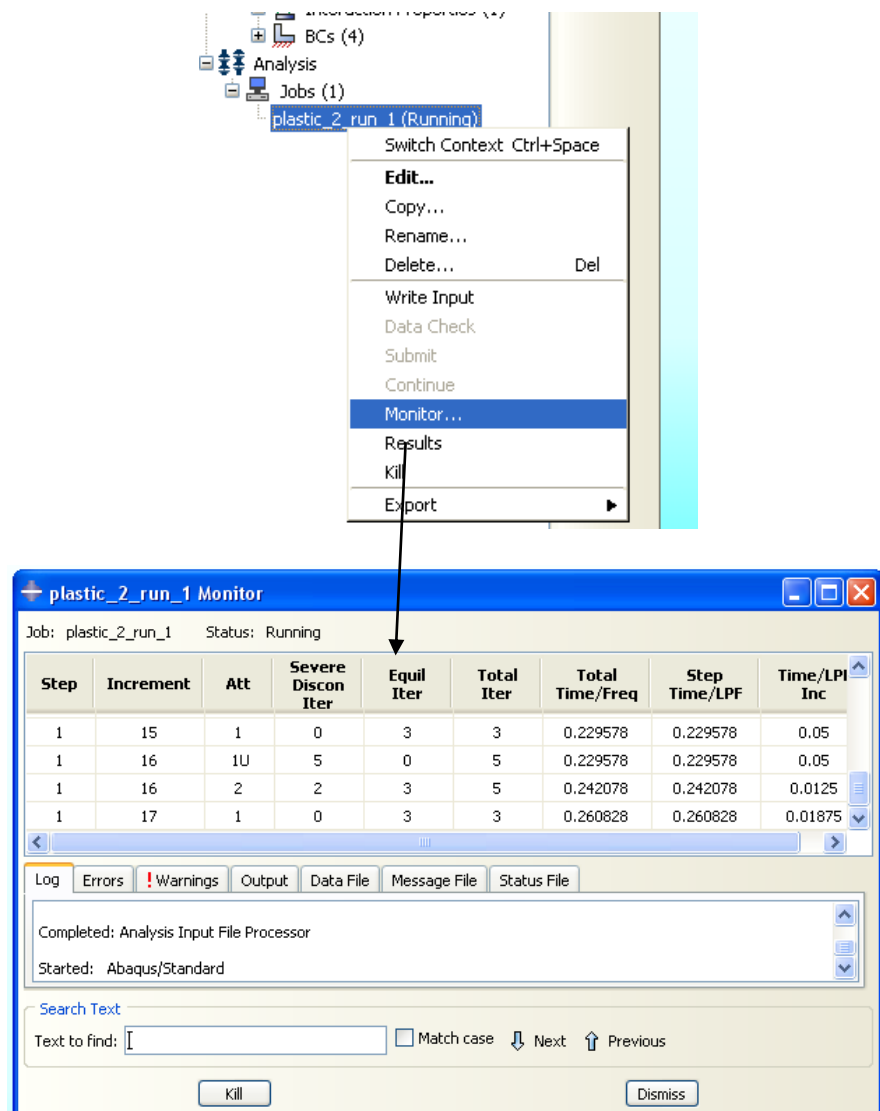
When meshing an assembly it needs to be done part by part, just like the materials/property process.



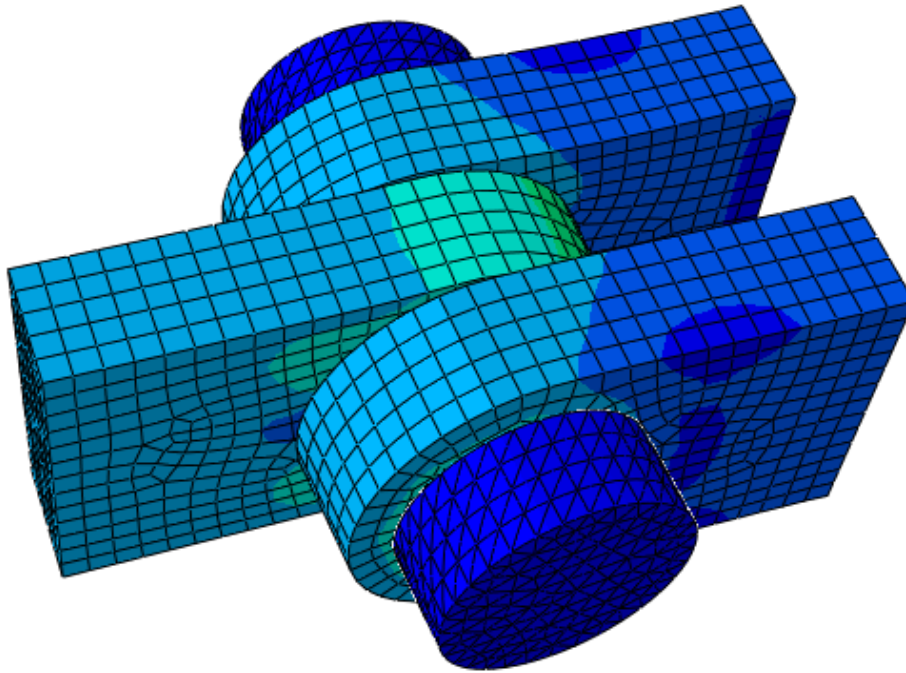
## 7. Create a job and then submit it



Monitor the progress of the solution whilst it is running.

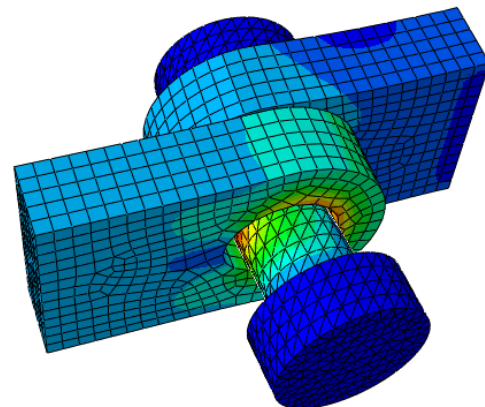
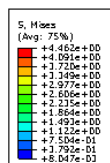
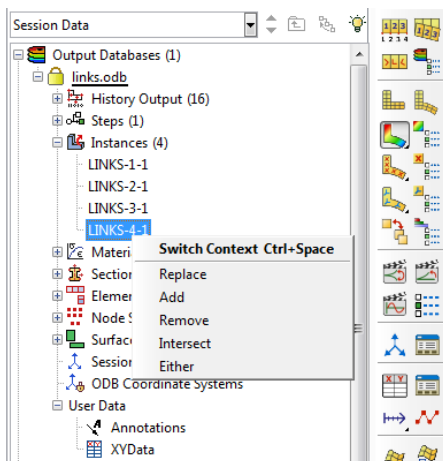


## 8. Open the results file and plot the results



We can use the post processing feature tree to remove various parts of the model to enhance visualisation.

Right click on the part under the 'Instances' header and select 'Remove'.



Try adding and removing parts to view the stresses and displacements of the pin and inside the lugs.