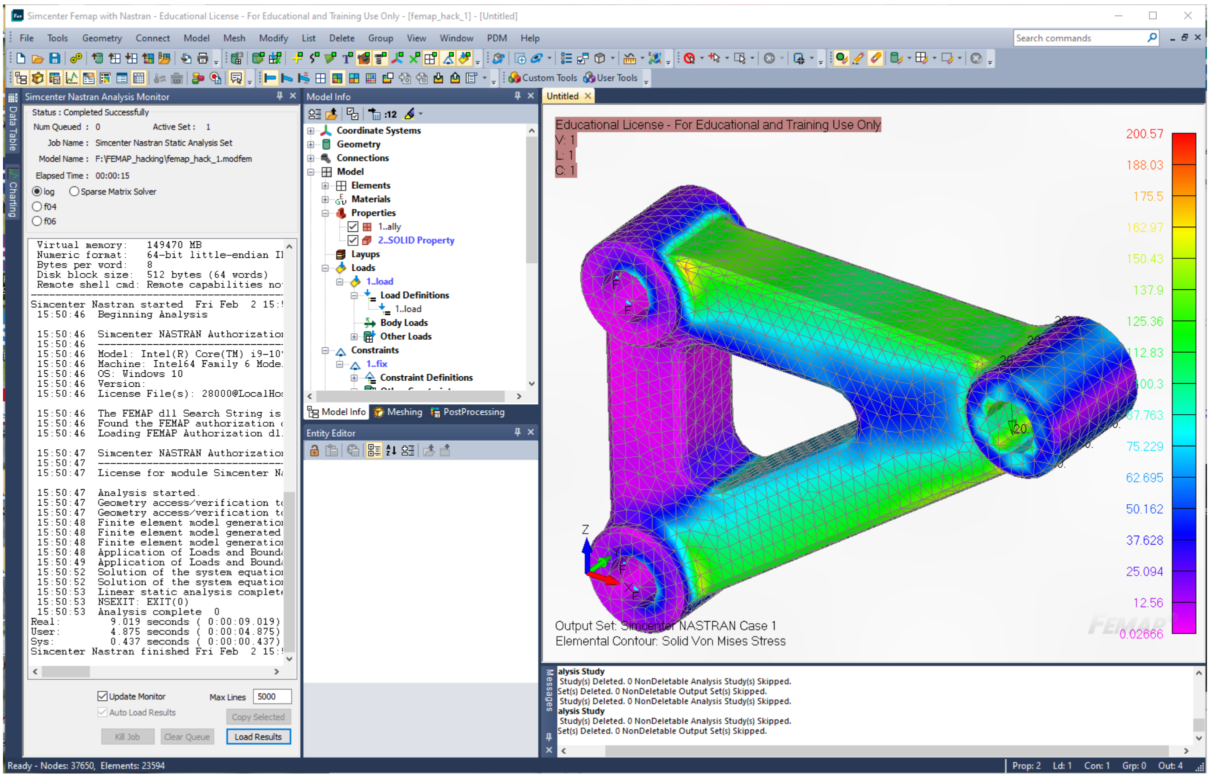
DRAFT

FEMAP Tutorial 1: Simple Bracket Analysis using the NX Nastran Solver

Laurence Marks

January 2024



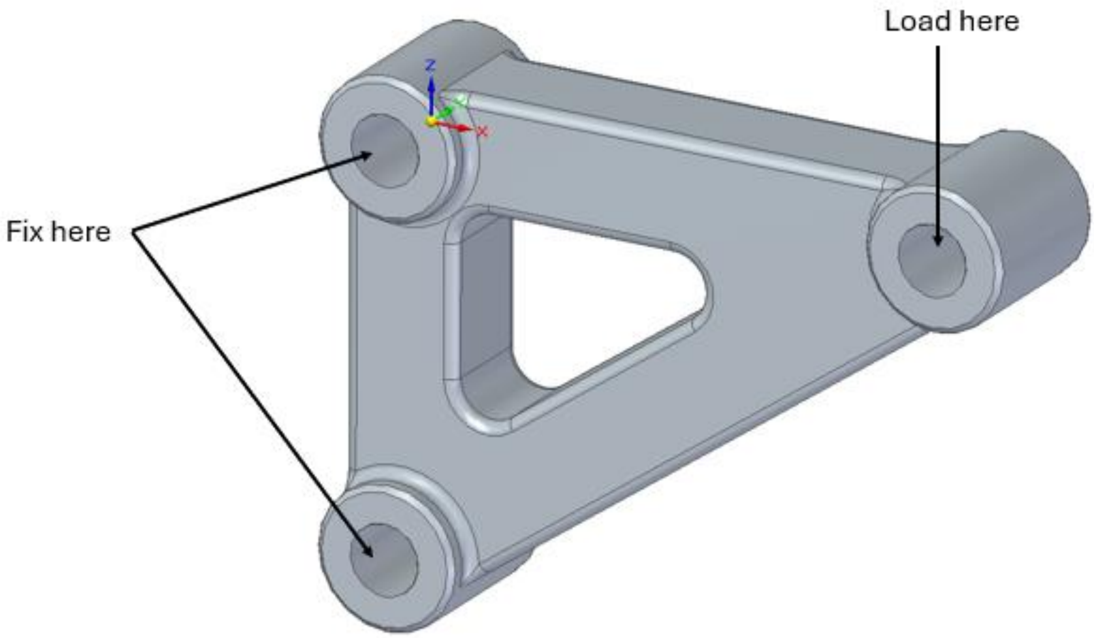
This tutorial is an introduction to the FEMAP interface and workflow. In this study we import a CAD model of a bracket, prepare it for analysis, run the model and post process the results.

This a basic workflow, which will be developed to show how an accurate model can be developed, but for the moment this just illustrates a very basic workflow.

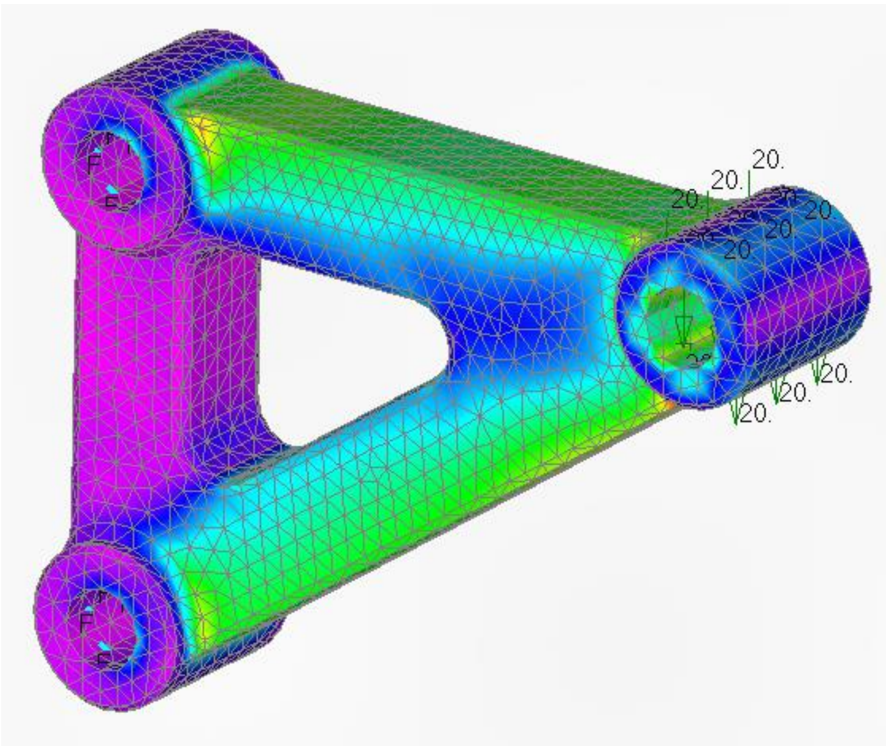
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Problem Description

In this example we are going to take a CAD model of a bracket, restrain it in 2 locations, add a load at a third, and calculate the structural response.



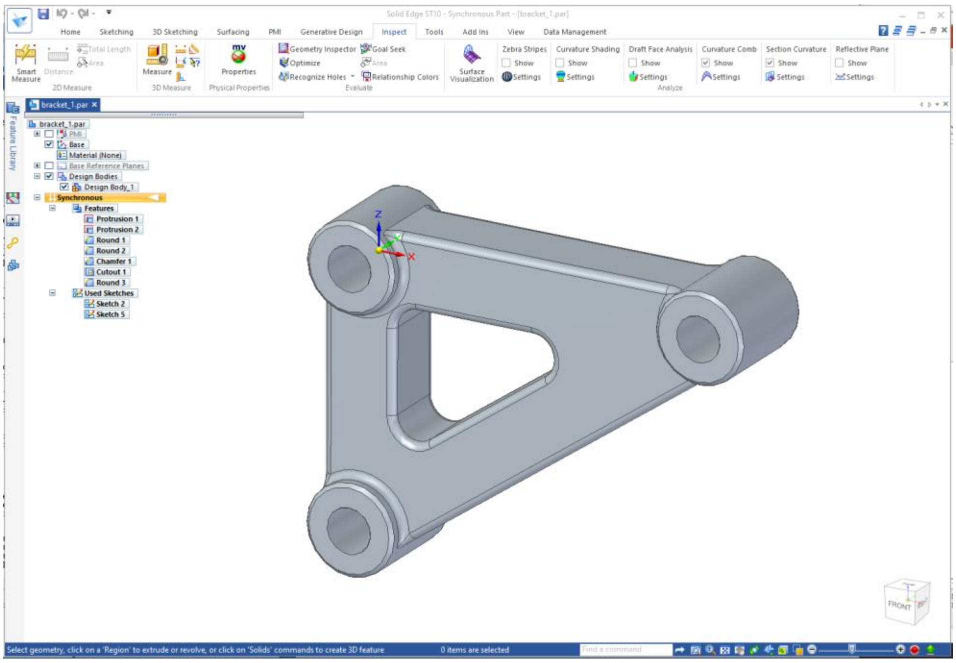
Which will ultimately give us a result that looks like this.



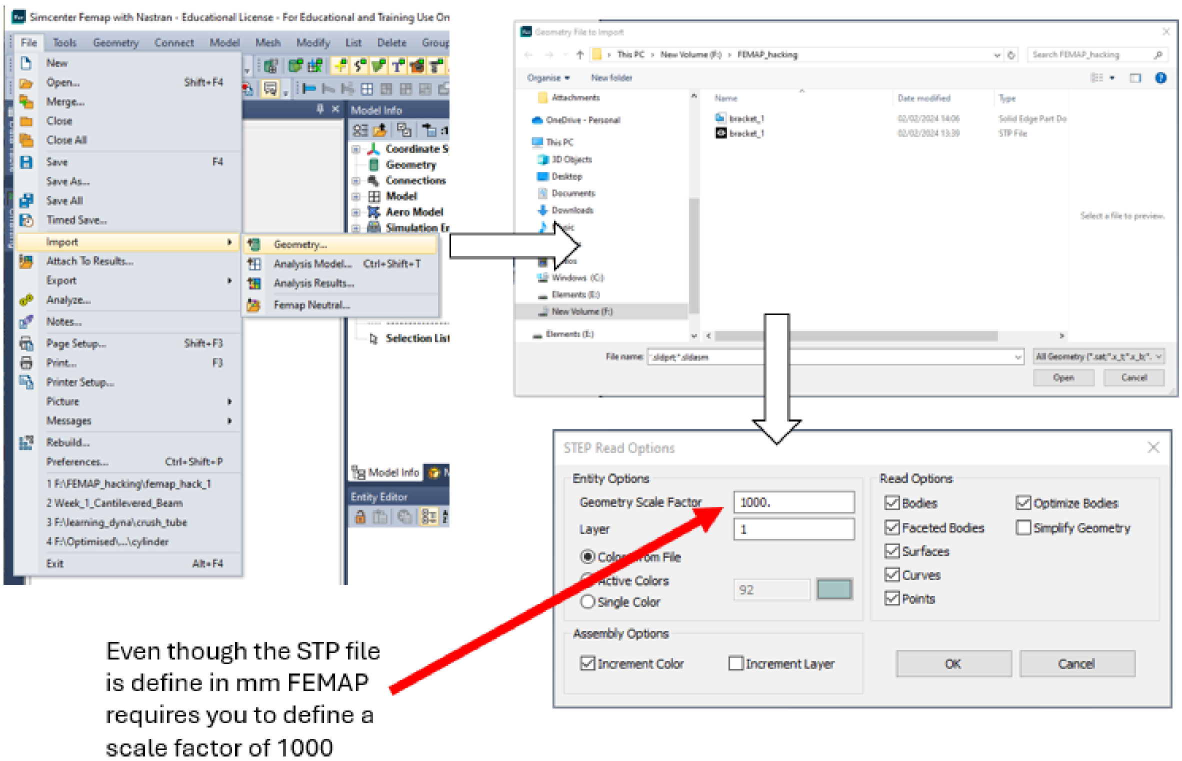
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Step one: Import the CAD model.

The model was created in Solid Edge, and a STP file was written out. The stp file is defined in mm.



We import this in to FEMAP using the Import command on the file menu. We need to define a scale factor of 1000. Otherwise the part will import 1000 times too small.

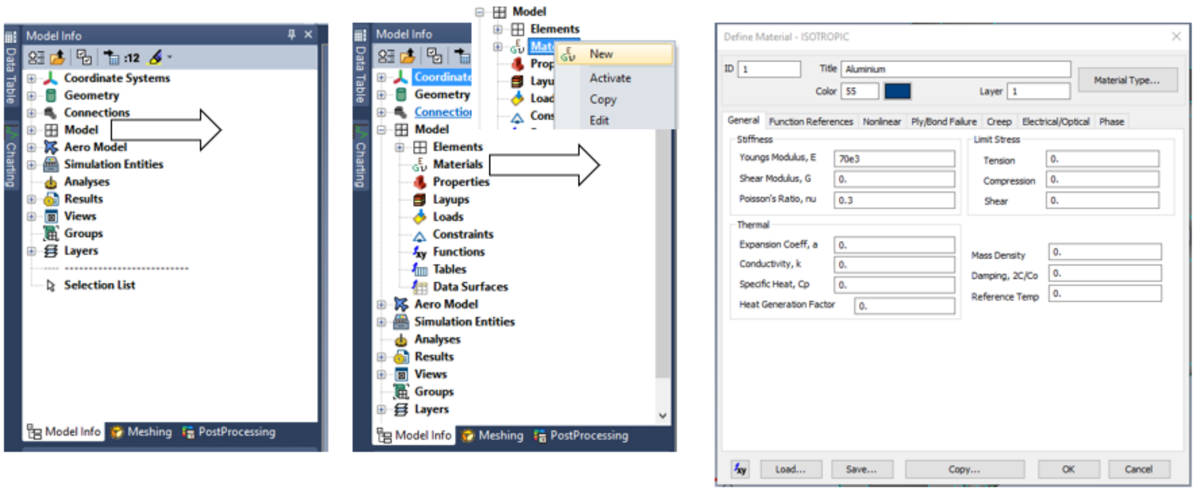


It is wise to check the dimensions of the imported part using tools>measure.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

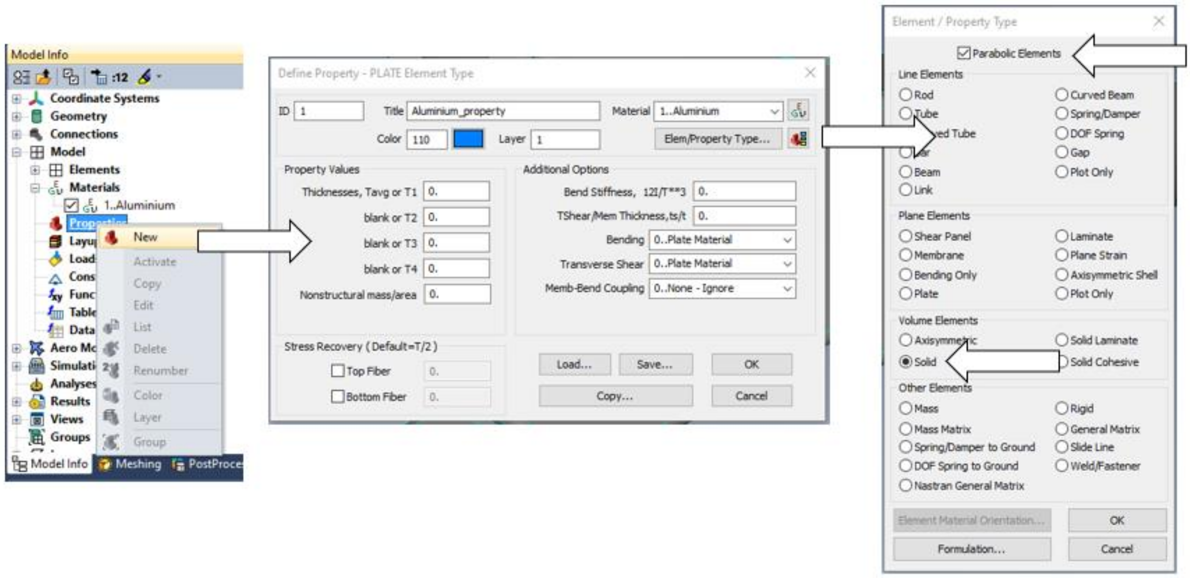
Step 2: Define a Material

We need to define a material for the bracket. We'll call it aluminium and give it a modulus of 70e3N/mm2 and a poisons ratio of 0.3. Expand the model node on the model info feature tree and right click over materials to create a new material. Define the 2 required properties.



Step 3: Define a Property

For reasons that become clearer as you build more complex models you need to create a property to apply the material to the part. (Just go with it now.) Define a new property, give it a name, and select aluminium as the material.



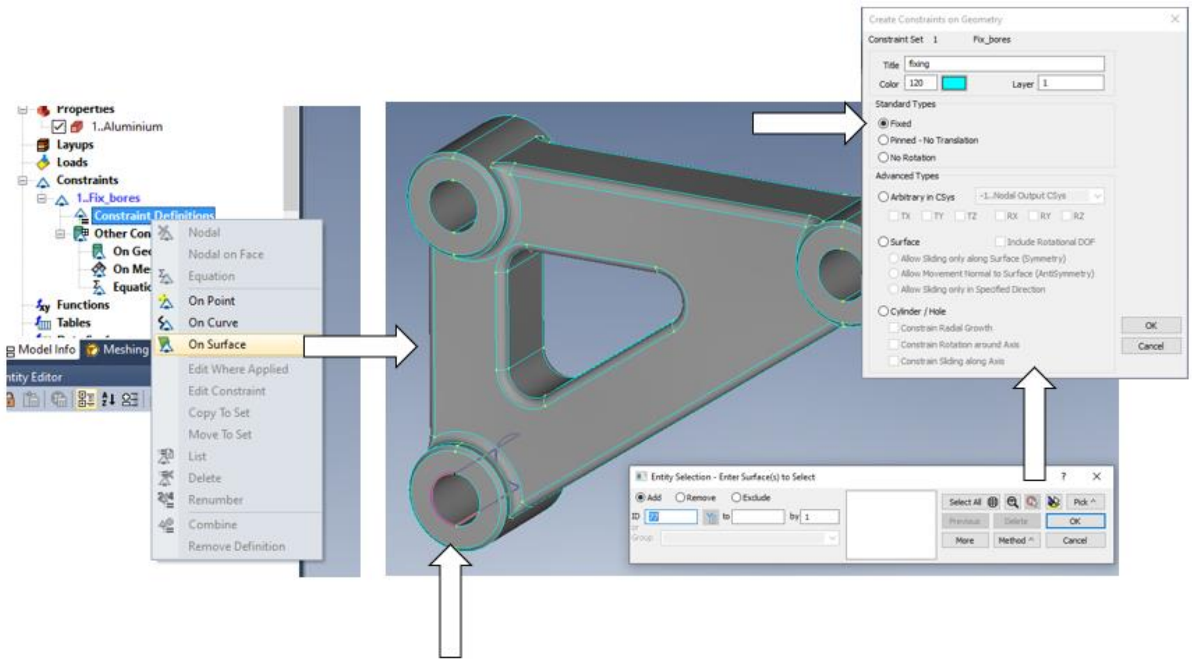
When you click OK at the end of the process you'd be presented with another property dialog so you can define additional sets. We don't need to so click cancel. The subsequent steps only ask us for info relevant to solid elements.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Step 4: Apply Restraints

We need to define the way in which the bracket is fixed. Firstly we defined a restrain set. Which we do by clicking on the constraints node of the feature tree.

We then need to define an actual restraint definition. Right click constrain definition and use the on surface option. We can then pick the surfaces on the two hole bores (FEMAP is a bit odd in the way it allows you to select these.)



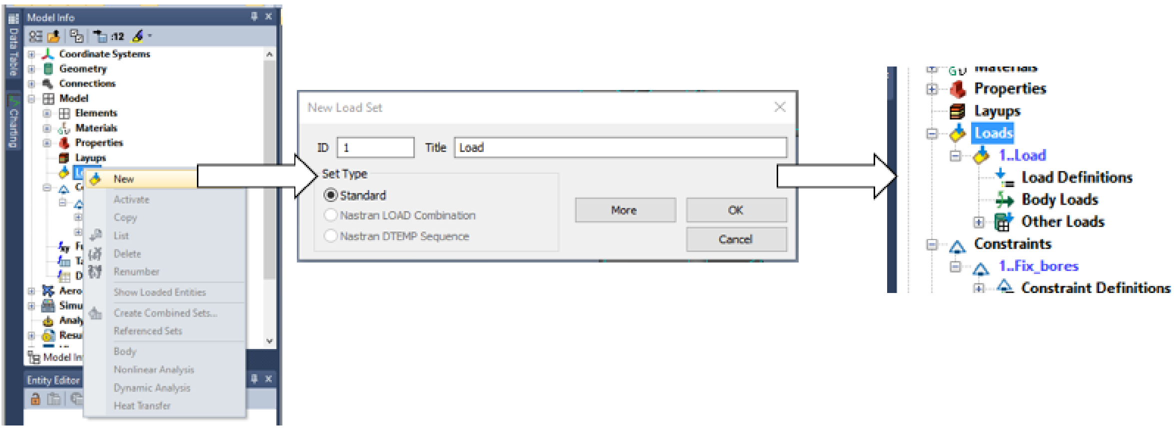
We use the default "fixed" option.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

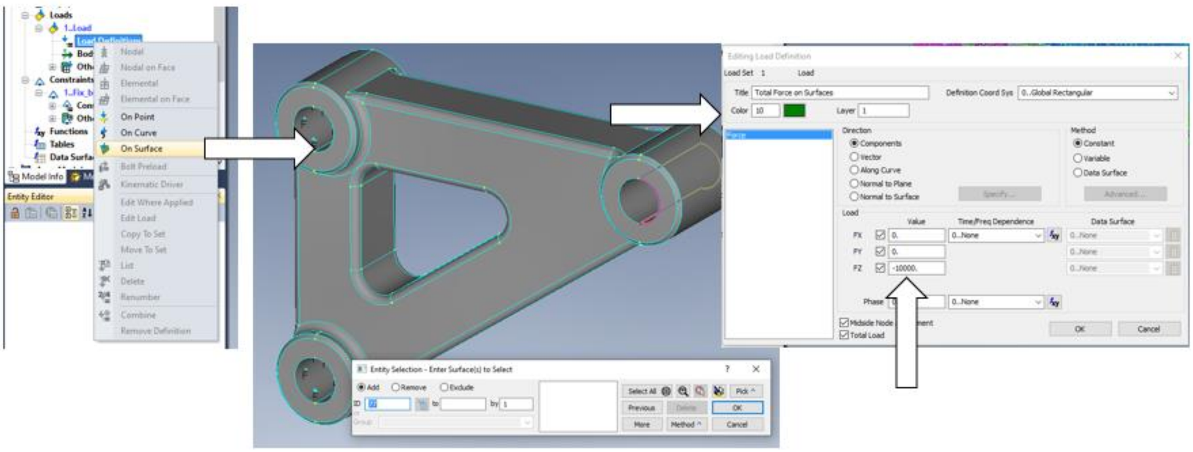
Step 5: Define the load

The process for this is very similar to the restraint definition.

We define a new load by right clicking on the loads node of the feature tree.



With that complete we then define the actual load. Select the on surface option. We then select the geometry that we are going to apply the load to. In this case it will be the surfaces on the internal faces of the end bore.

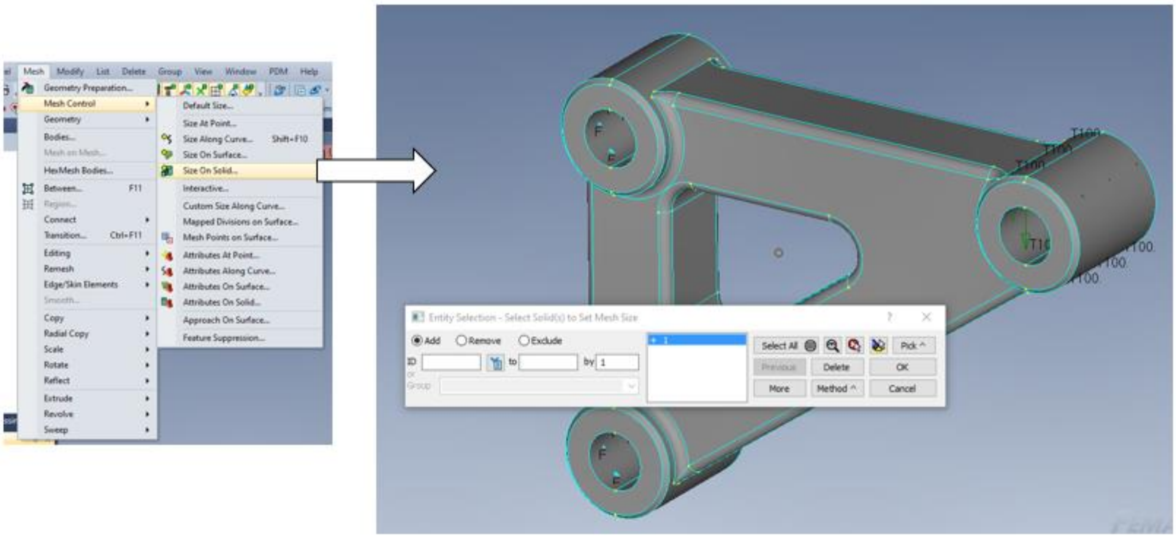


We will apply a load of -10,000N.

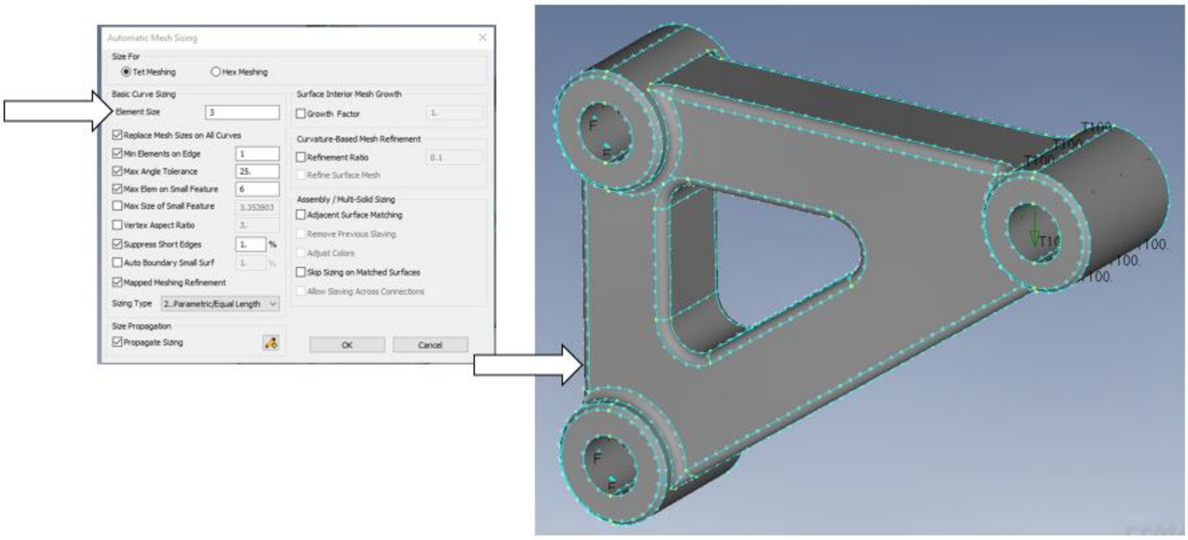
Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Step 6: Defining the finite element mesh

Firstly we are going to define the size of mesh we use. In FEA studies this is a super critical step. From the mesh menu select mesh control and then size on a solid.



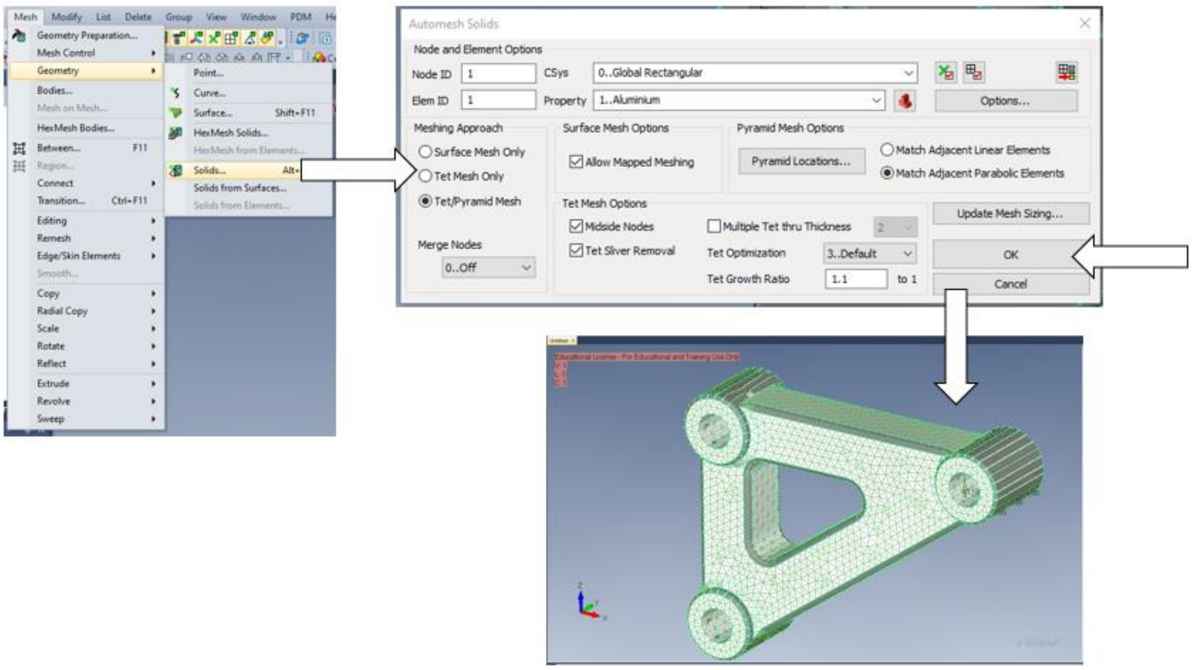
The default size is around 3mm anyway, but enter 3mm. This is how you control the global mesh size. For this initial tutorial we'll leave it at that.



Mesh seeds will appear, showing that we've got the size about right.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

To create the actual mesh we select mesh>geometry>solids and then click OK on the dialog box. No need to select anything. That produces a uniform mesh of 10 noded tet elements.

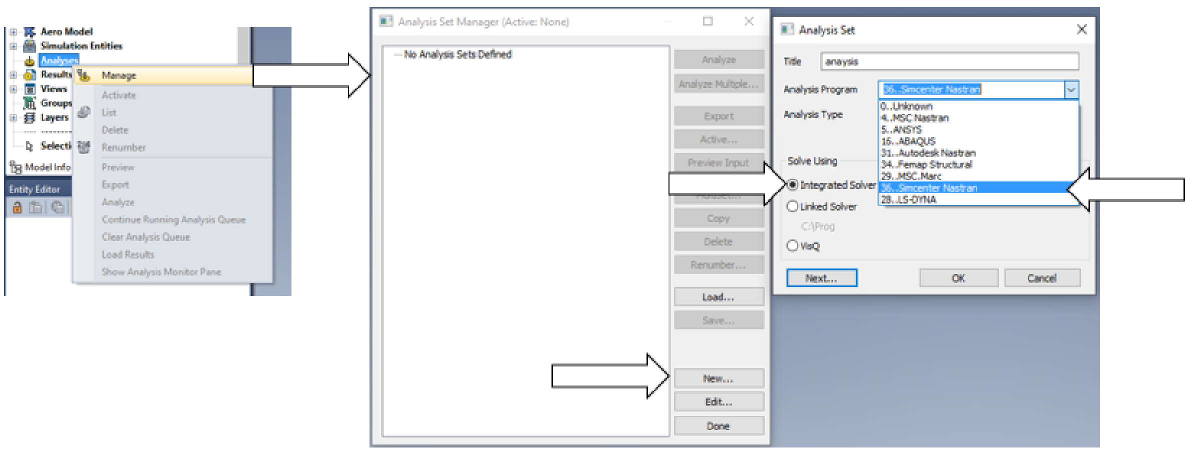


The actual model is now complete.

Step 7: Define the analysis

We need to tell the system what solver to use and what sort of analysis it needs to perform.

Create a new analysis and select the integrated simcentre Nastran solver.

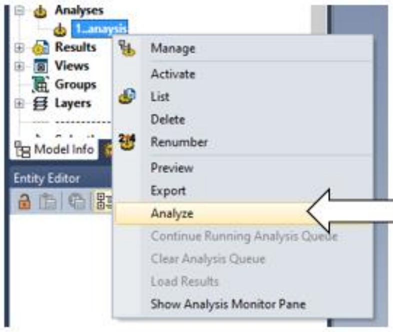


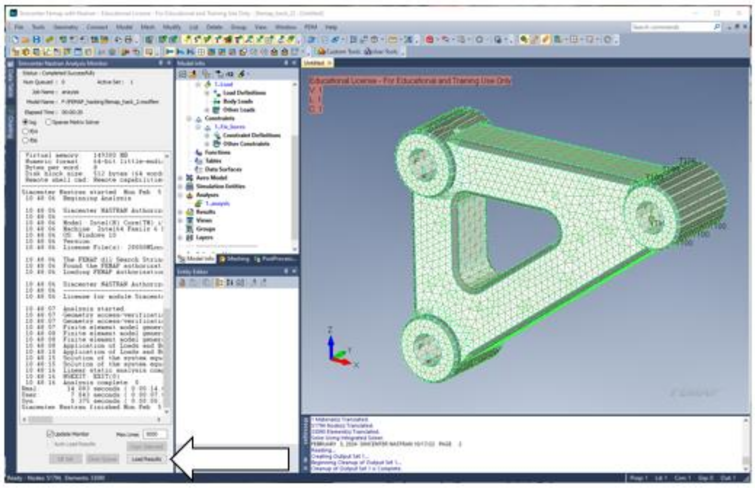
Just take all the defaults instead of clicking next. Then click done on the analysis set manager window which will now be populated with info about the simulation.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com

Step 8: Run the model

Right click the analyze option on the analysis node and the model should run. If it does a new window will appear at the side of the main FEMAP window.

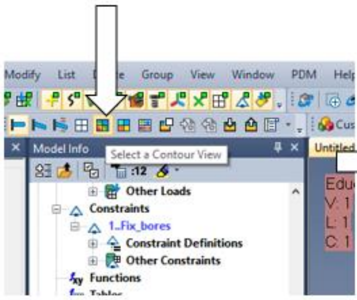


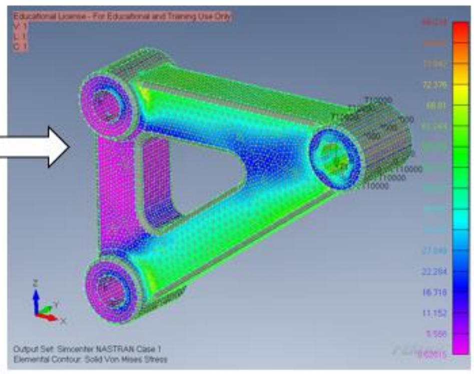


When the run has completed click the load results button.

Step 9: Plot the results

With results loaded you'll be able to see a results set in the results section of the feature tree. Click on the contour view of the post options toolbar. That will create a stress plot.





There are lots more post processing options (and model definition options) which we will look at in later tutorials. But this is the basic workflow which we will develop and build on.

Laurence Marks
l.marks@brookes.ac.uk
laurencemarks64@googlemail.com