# Tutorial 1: Stressing a simple bracket

This tutorial talks you through the process of building a simple structural finite element model in freecad, but uses this to teach key concepts about the process along the way.
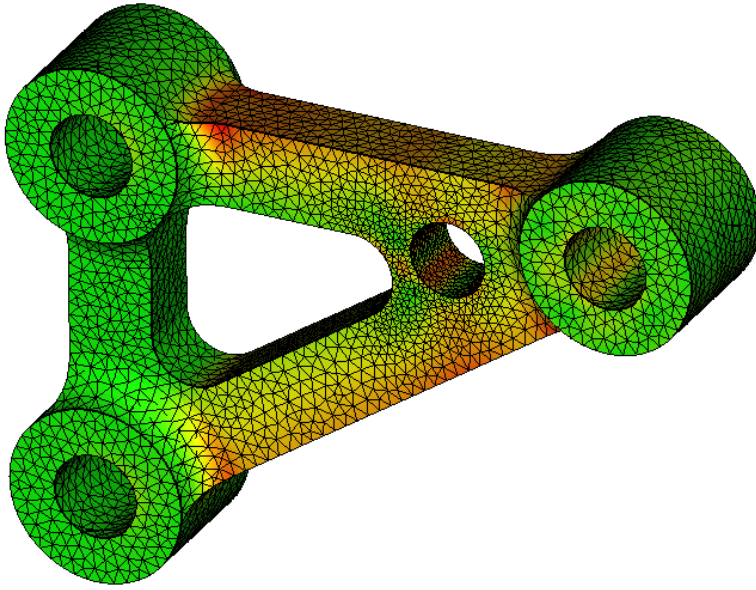
What is a finite element model?

A finite element model is an array of theoretical springs that are used to represent the way in which a component or assembly will react to a load. So in its most simple form an FE model is what is shown below.

We have a spring and we fix one end. We pull the other end. It gets longer. Hooke's law says how much longer it gets depends on how hard you pull it and how stiff the spring is.
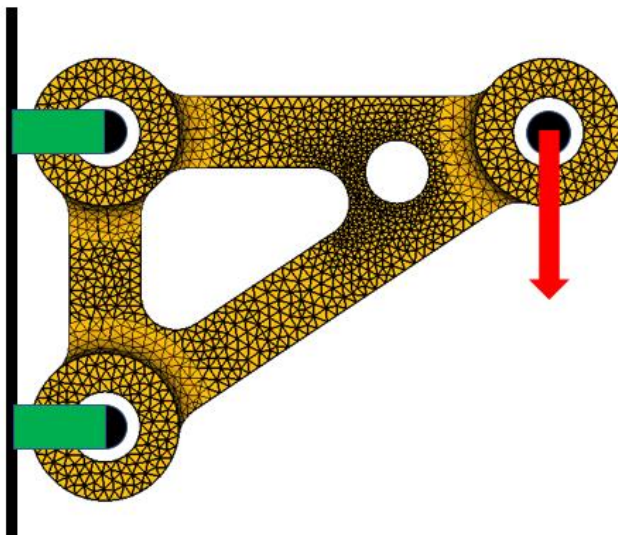
In the finite element model we are going to make we've replaced the single spring with an array of springs (each line is a kind of spring, each nodal connection point a calculation point); as before the harder we pull them the more they deflect. And the deflection is also dependent on how stiff they are. The computer does a lot of work calculating how stiff the springs are and how much they deflect under load. Once we know how the part deforms we can work out what the stress is in each of the sub-volumes which have previously been used to calculate the stiffness.

Step by Step Analysis Process

The process of turning a CAD part into a finite element model is simply that of defining where and how we fix it, where and how we load it, and how to define the spring stiffnesses that represent the structural stiffness of the part. Simple. Essentially. Though people have devoted lifetimes of effort to each one of the steps listed below.

Part geometry

This tutorial is based on a simple solid part. It could be imported, (bracket.stp) or drawn in Freecad. For reasons that will become apparent it shouldn't be too complex and should be well drawn.
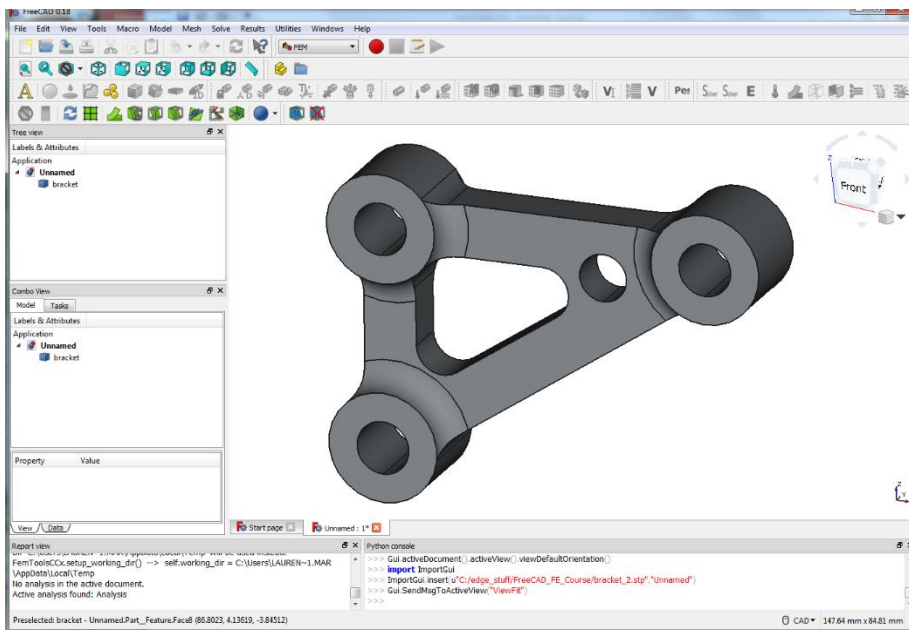
FEM Workbench and Analysis Container
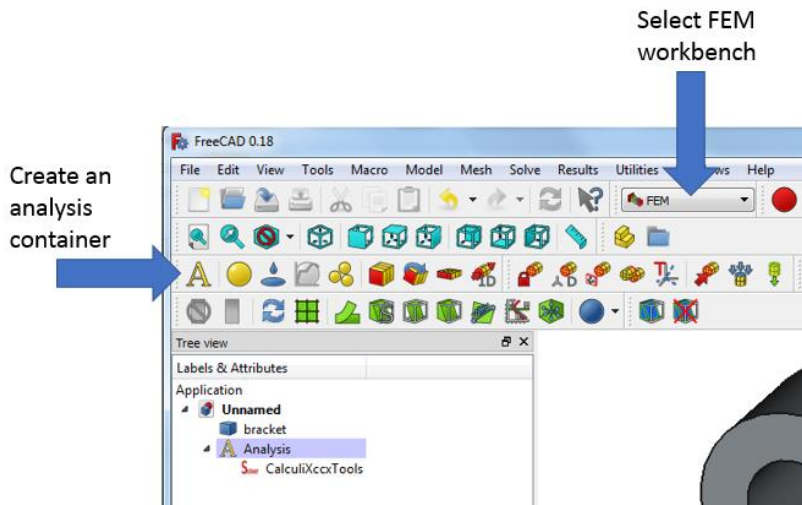
Select the FEM workbench and create an analysis container by clicking on the A button. This sets up the study so that you can group and manage all the simulation data

Materials

Because initially we are only calculating how the structure will deform we only need to input modulus and poissons ratio. But we can also achieve this by selecting a material from the database.
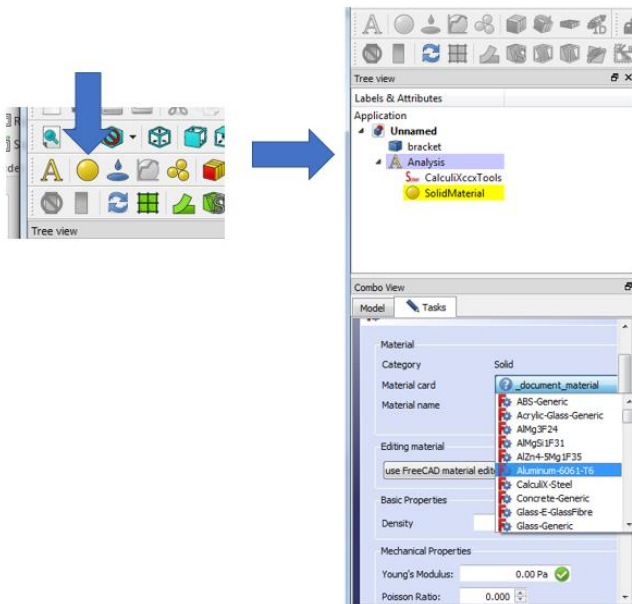

Restraints

We have to fix the model somehow. If we don't do that properly applying a load makes it move off in an unrestrained manner to infinity, and that blows the solver up. So we'll pick the two rear bores of the bracket and fully fix them, in all directions. FEA is all about assumptions and educated

guesses. And this is the first of them. Think about it. You can't fully fix something in any direction. It'll move a bit. So this will be one reason the model isn't exactly right – if its right at all.
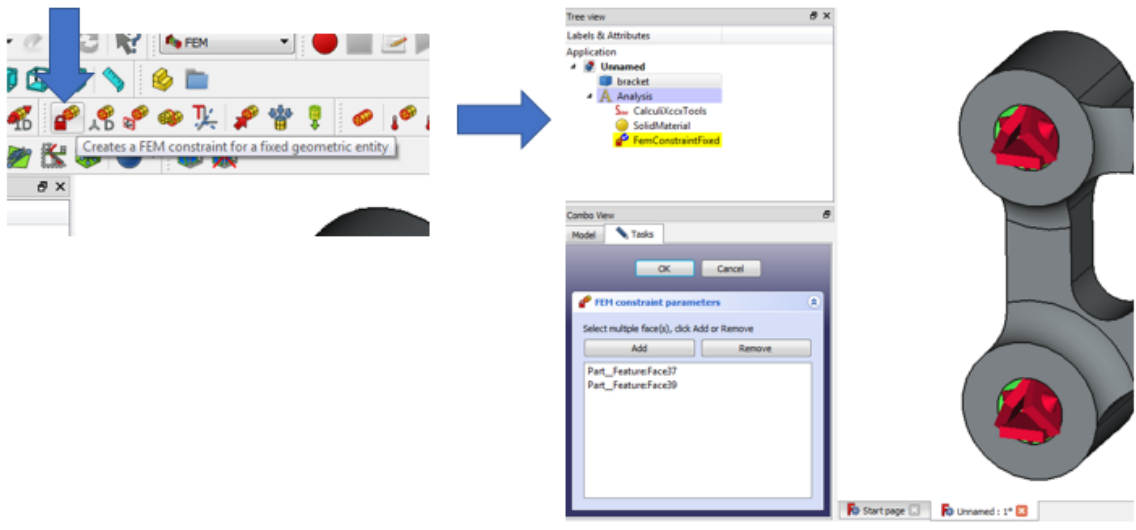

Loads

This isn't a very realistic loadcase, but lets go with it anyway. We'll apply a 1000N to the end bore, and by default it will act vertically downwards. In the real world you can apply 1000N to the end of the bracket, but its unlikely to be uniform over the bore of a hole. But it should be good enough for most design purposes.
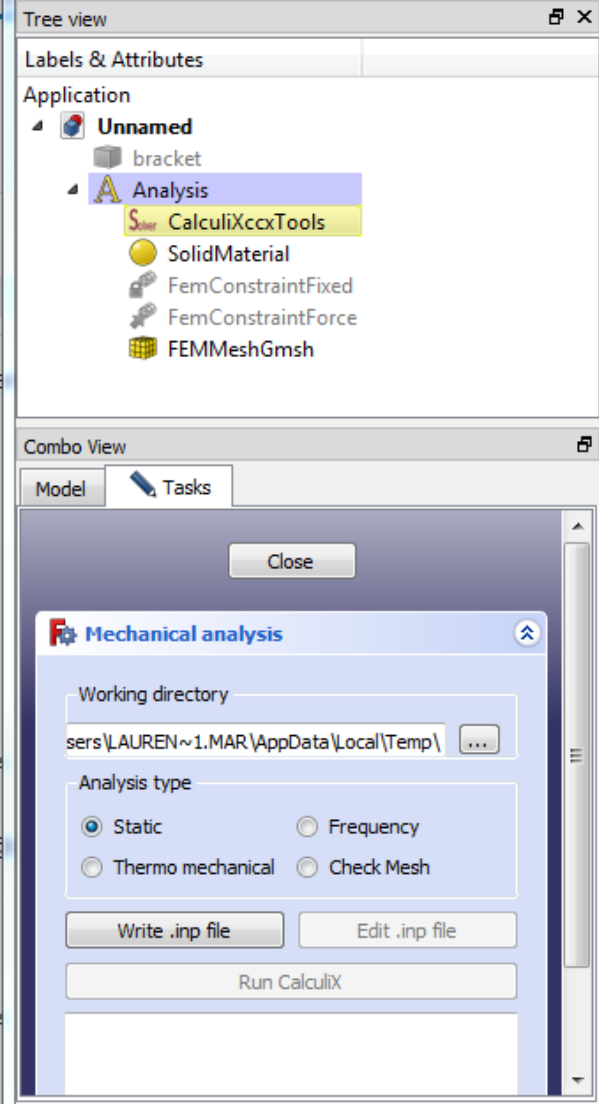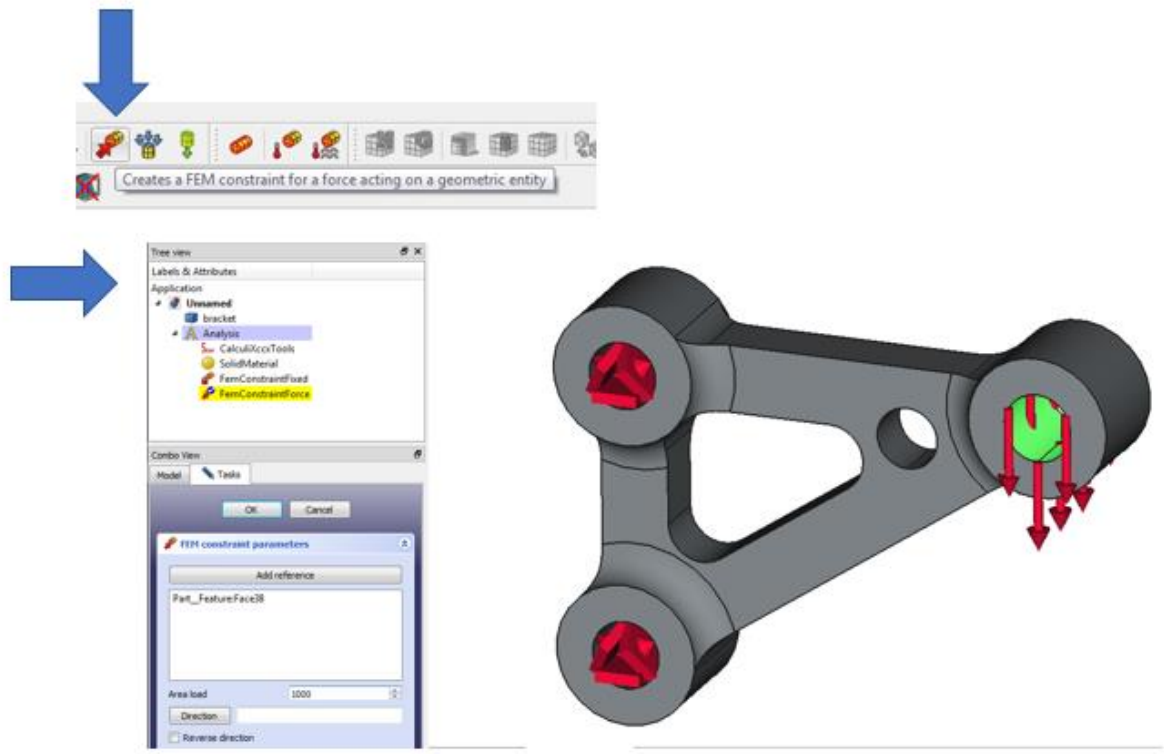
Meshing

You could write chapters and chapters on this and I have in other places. Select the icon showing the bracket geometry. That will highlight the geometry and the meshing icons will activate. In this tutorial we are going to use the Gmesh mesher.
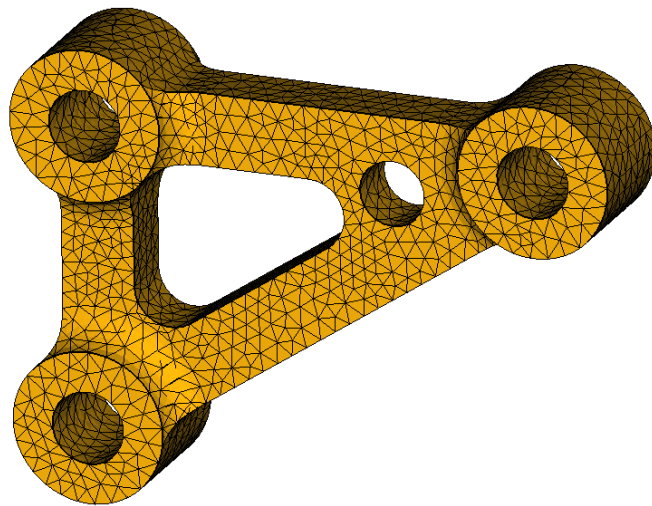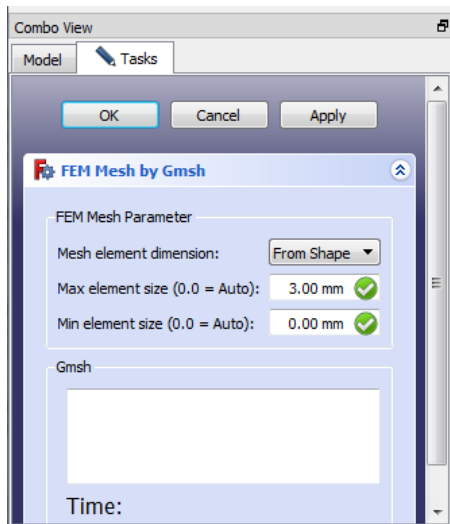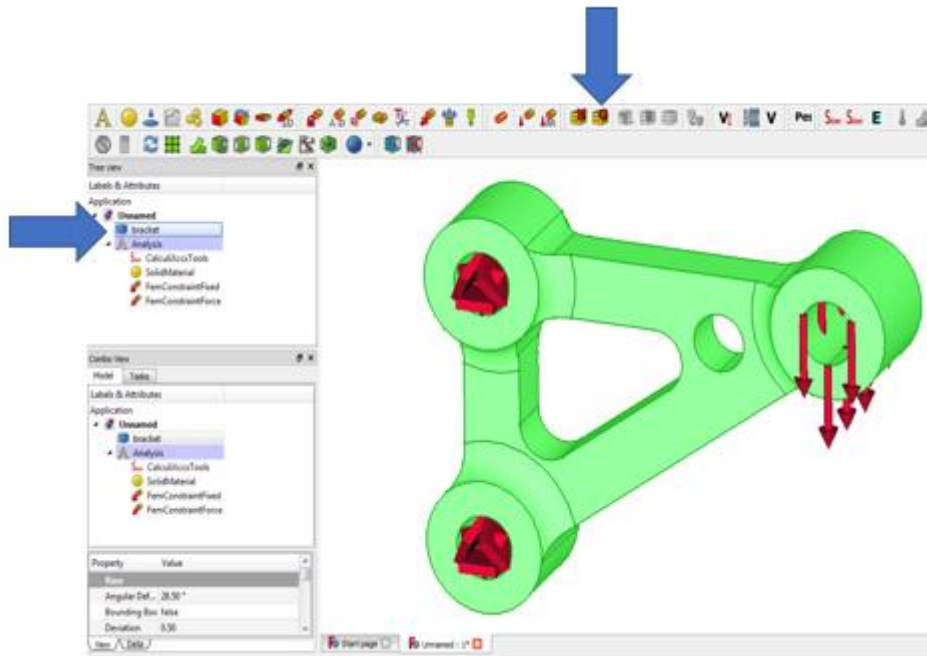
In this initial study I've chosen a max element size of 3mm. Its generally a good idea to get the model running with a moderate mesh before revisting any decisions made about meshing approaches and density. (For the moment you can ignore messages about element distortion.)

Run the Solution

Double click on the CalculiXcxxTools icon. This brings up a dialog box. The defaults for analysis type (static) and working directory are fine. Click on write .inp while which writes the file of mesh, loads constraints and other info for the solver. It looks like whats shown on the right, but you don't need to look at it at this stage, possibly never..

```
** written by FreeCAD inp file writer for Calculix,Abaqus meshes
** highest dimension mesh elements only.

** Nodes
*Node, NSET=Nall
1, 8.108108108108, -10, -9.513599892114
2, 9.513599892114, -10, -8.108108108108
3, 8.108108108108, 10, -9.513599892114
4, 9.513599892114, 10, -8.108108108108
5, 6, 5, -27.57779489814
6, 6, 5, -17.41126056831
7, 6, 5.869444441024, -14.08012784033
8, 6, -5.869444441024, -14.08012784033
9, 6, -5, -17.41126056831
10, 6, -5, -27.57779489814
11, -8.108108108108, 10, -9.513599892114
12, -8.108108108108, -10, -9.513599892114
13, 9.513599892114, -10, 8.108108108108
14, 9.513599892114, 10, 8.108108108108
15, 17.41126056831, -5, -6
16, 35.36669234722, -5, -6
17, 14.08012784033, -5.869444441024, -6
18, 14.08012784033, 5.869444441024, -6
19, 17.41126056831, 5, -6
20, 35.36669234722, 5, -6
```

Then click run CalculiX. After a short while you should get a message saying "CalculiX done without error!" and we are good to look at the results.

Post Processing – viewing the results

Double clicking the Calculix_static_results icon. The dialog box allows you to select different results data to be plotted and tells you the max mean and average values. Here we've plotted deflection and Von-Mises stress levels.
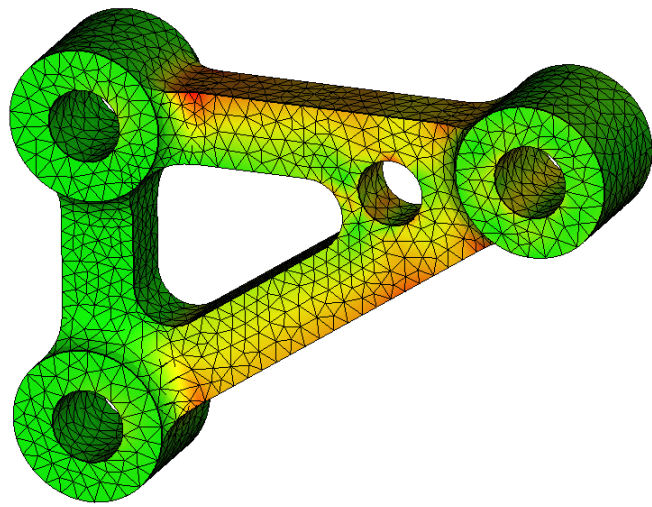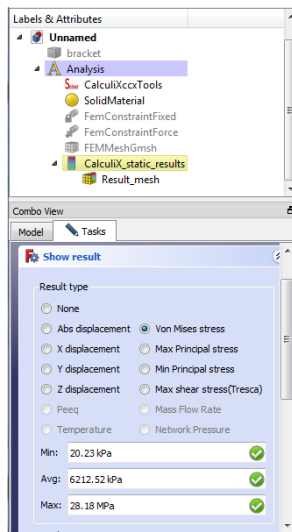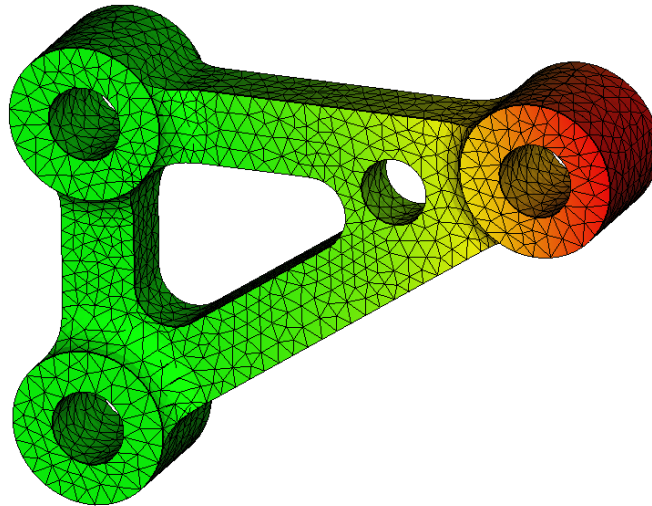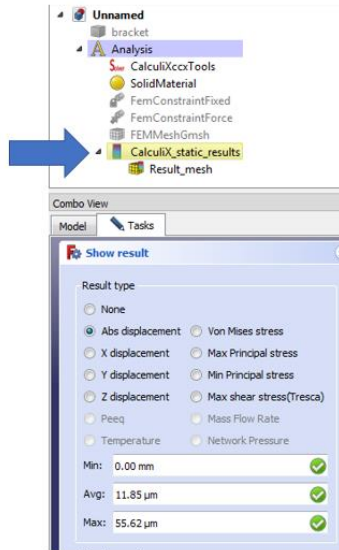
Summary

There was nothing difficult there. And the end to end process shouldn't have taken very long. The model gives us insights into how the bracket would behave under load. How much it will deflect and how near the stresses are to causing a problem. (Answer in this case, not much and not very). But the big lesson is not so obvious. Every decision we made about what to put into the model had a bearing on what we got out. (OK in this case you just did what the tutorial said, but hey go with this..)