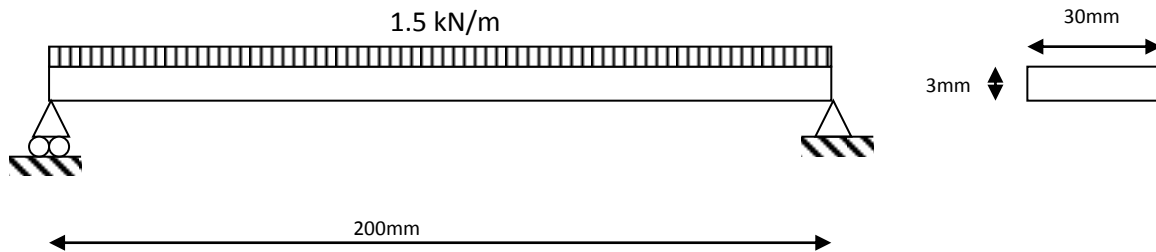


Shell Meshing and Post Processing Options

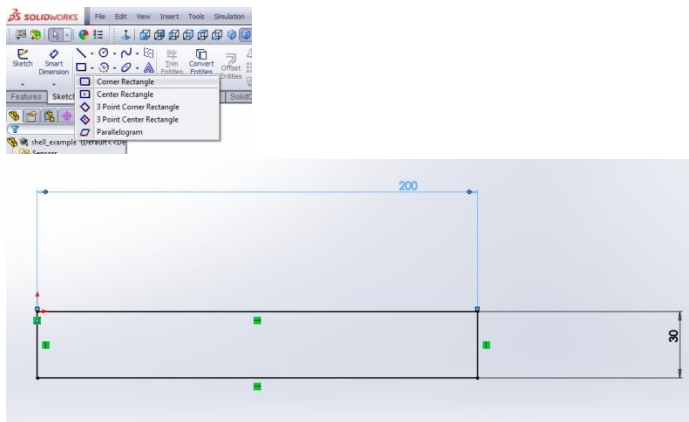
In order to demonstrate some of the fundamentals of shell meshing we are going to consider the following simple case.



A steel simply supported beam has a uniformly distributed load of 1.5kN/m across it. What is the maximum stress and what is the maximum deflection?

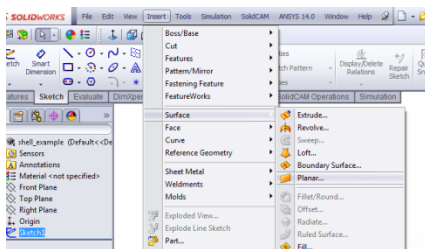
Because the height dimension of the cross section is much smaller than the other two dimensions we are going to use shell elements.

1. Start Solidworks
2. **Create a new part. OK.**
3. Save your part as shell_example locally to your hard drive.
4. Create a 30mm x 200mm sketch on the top plane

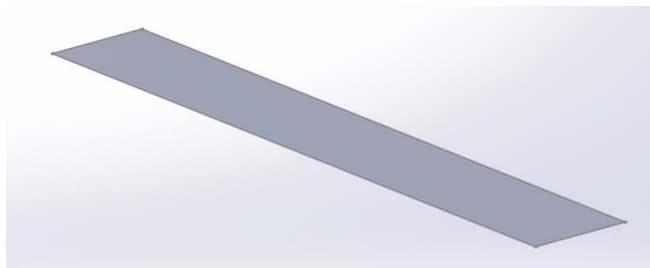


Exit sketch

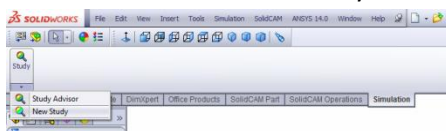
5. Use **Insert>Surface>Planar** to create a surface using this rectangle. This will be the MIDSURFACE of our part



Highlight the sketch and choose **OK**. You should now see the following;

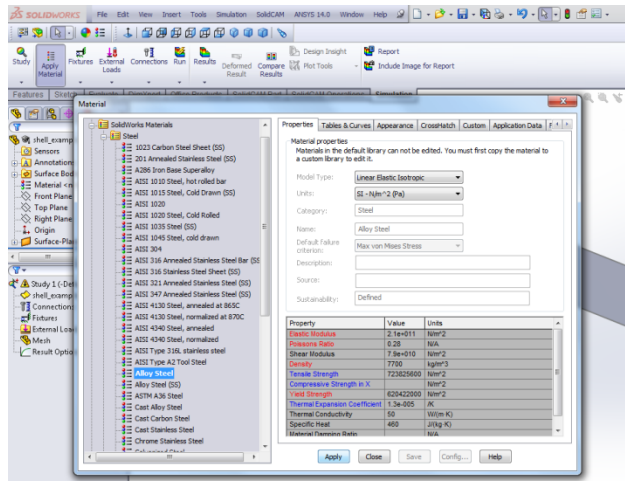


6. **File>Save**
7. Create a new simulation study

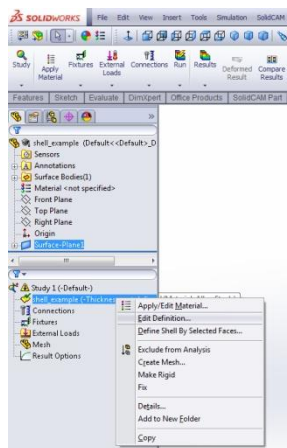


8. Choose a static analysis. **OK.**

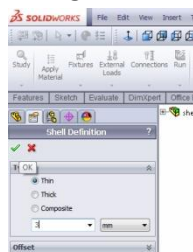
9. Apply steel this part. **Close**



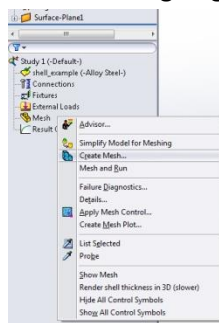
10. Now, RMB on the 'Shell example' icon in the simulation tree and choose 'Edit definition'



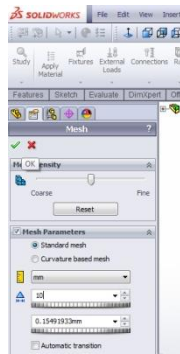
Change the thickness to 3mm. **OK.**



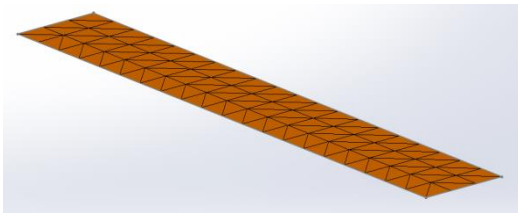
11. We are now going to mesh our part.



Set the mesh size to 10mm. **OK**

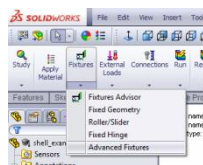


You should now see the following;

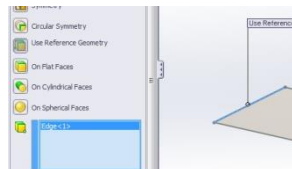


12. We will now set up our constraints. If the part was created on the top surface, then we need to constrain one end of the beam in degrees of freedom 1,2,3,4 and 5. i.e. degree of freedom 6 is free. (degrees of freedom 1,2 and 3 refer to translations in the global x,y and z axis respectively, whilst degrees of freedom 4,5 and 6 refer to rotations about x,y and z respectively).

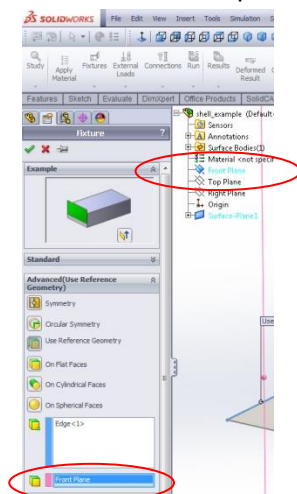
13. Choose **Fixtures>Advanced Fixtures**



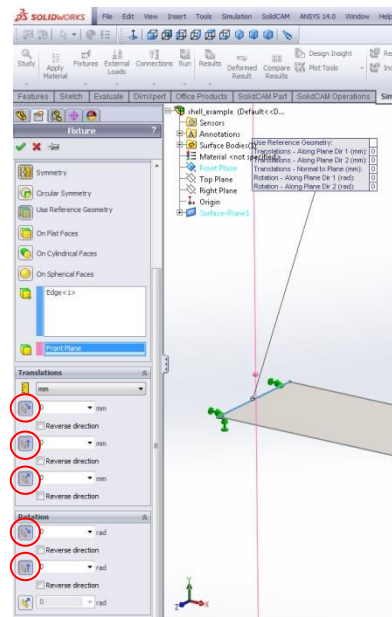
Choose the edge that you want to constrain



For the reference plane, choose the **Front Plane** so that you are using the global coordinates

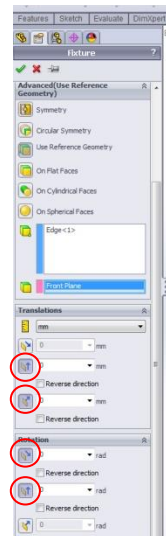


Choose all translations (x, y and z), and the first two rotations (x and y)



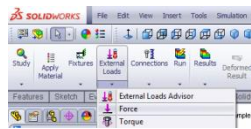
OK

14. Now set the other end so that degrees of freedom 2,3,4, and 5 are fixed to simulate the rolling pin joint

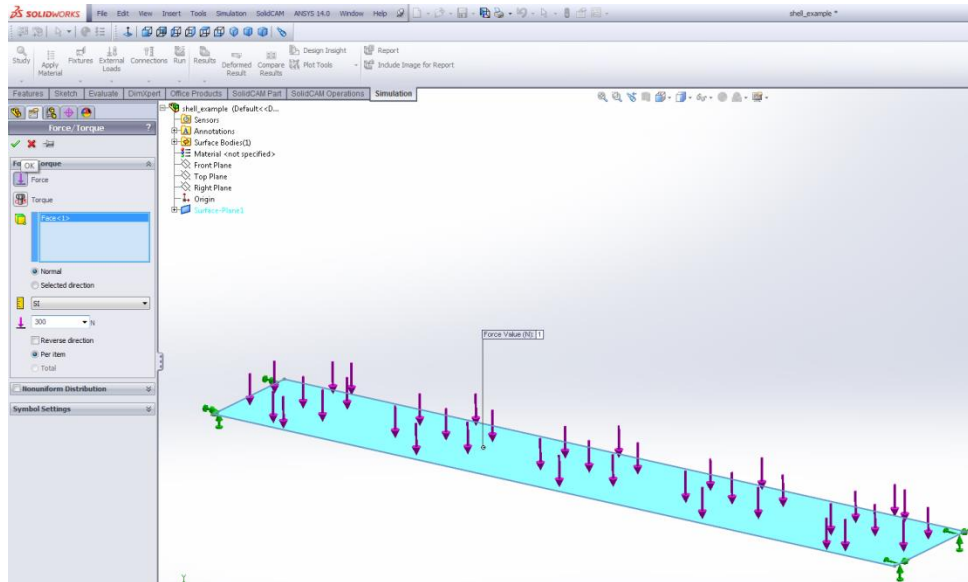


15. We are now going to put a TOTAL LOAD of 0.3kN onto our surface. As our surface is 0.2m long, this makes a udl of 1.5kN/m ($0.3/0.2$)

External Loads>Force



Choose the surface of your component and apply 300N normal to this surface.

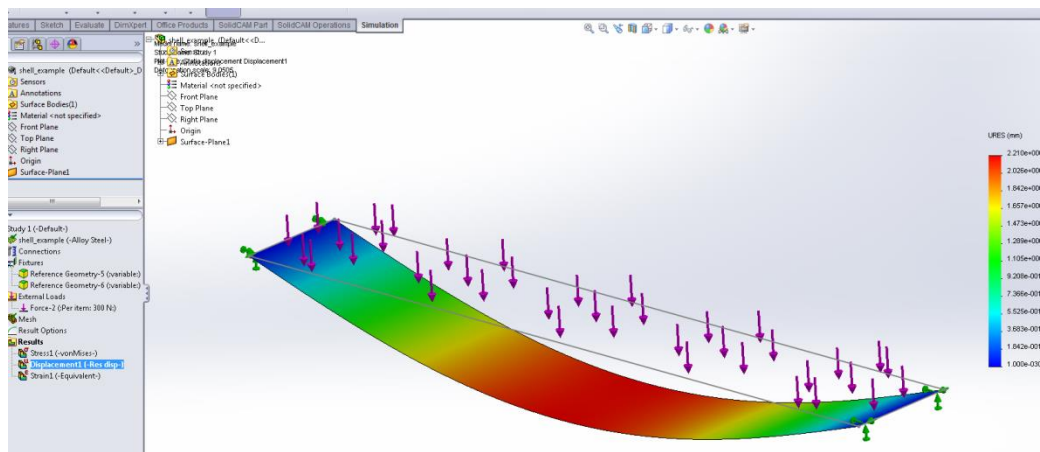


16. File>Save

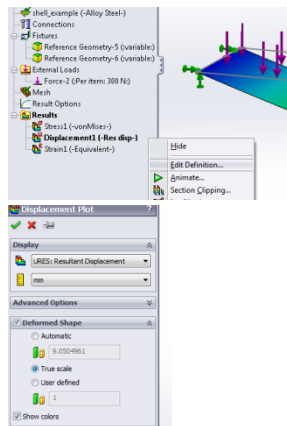
17. Run this model



18. Click on the Displacement to show these. How do these compare to the hand calculations at the back of this handout? How about the stress?

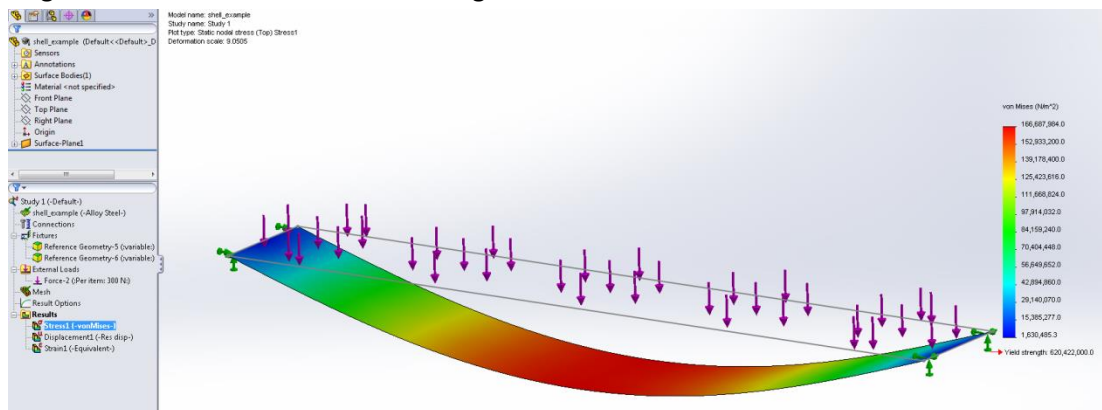


18. Note that the deformation plot is massively exaggerated. This is so that you can see what is happening and is a good way to visualize if the part is behaving as expected. You can set this to the actual value of deflection or any other value as shown below. By default, most post processors scale the maximum deflection so that it is 10% of the screen.

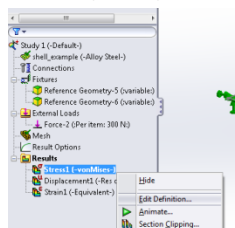


19. We are now going to look at the stresses

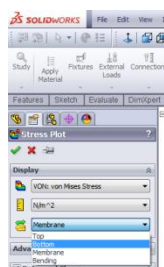
Note that by default the stress is 'averaged', it is shown for the 'top' surface and the element outlines are not shown. This can be very misleading as we have no idea what mesh was used to generate these results from this image.



You can change this to the bottom surface or the middle surface (membrane) by choosing **Stress (RMB)>Edit definition**

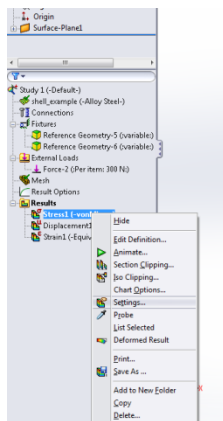


Now choose the surface that you are interested in.

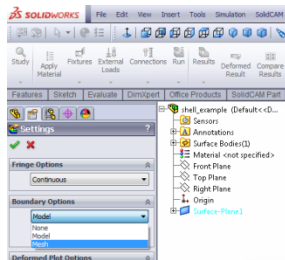


Note that if you look at the mid surface the stresses are zero because you are looking at the neutral axis. If you look at the bottom surface which is in tension the maximum value is also 166Mpa just like the top surface (as the beam is in pure bending), however this is not always the case, it is very important when using 2D elements to understand which surface you are looking at when considering any results.

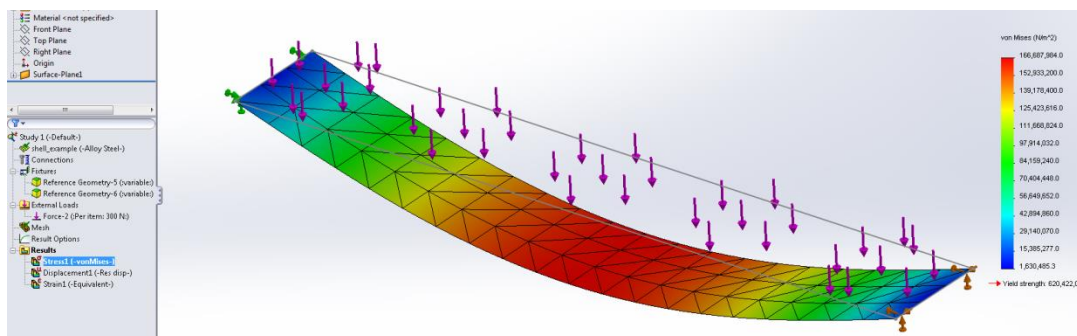
20. In order to show the underlying mesh, choose **Stress (RMB) > Settings**



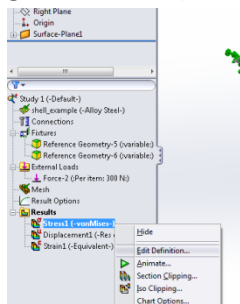
Set the **Boundary Options** to mesh



You can now see the mesh.



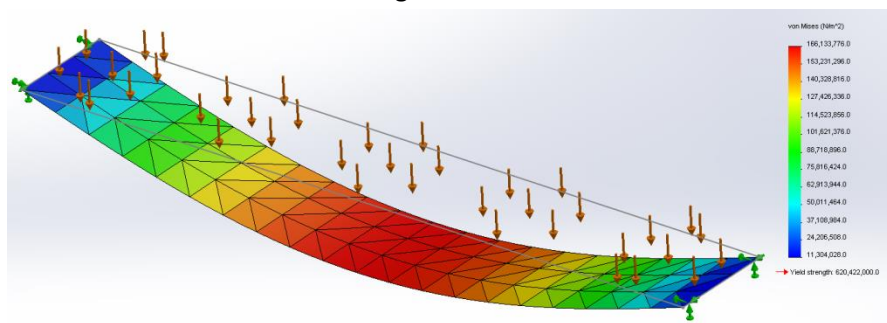
However, notice that the contours are still nodal (averaged) results. In order to change this you can go into **Stress (RMB) > Edit definition**



Then go into the advanced options and set this to **Element Values**. OK.



You should now see the following.



It is important to understand if you are looking at averaged or unaveraged results. The solver calculates the results that you are looking at each node in your model. However, one node can be attached to several elements, each potentially moving differently which in turns strains and therefore stresses the node differently. Consider the following

Averaging

For elemental and element-nodal results, you can display contours as averaged or unaveraged data. In a mesh, each element may have a different value for a given node. A node at the intersection of four elements, for instance, may have four different values.

To make the bands of color in a contour display appear smoother, the software uses the average value at the node, rather than each value. For example, in the following figure:

- For element A, displacement = 20
- For element B, displacement = 22
- For element C, displacement = 24
- For element D, displacement = 26

Therefore, the nodal average (E) is 23.

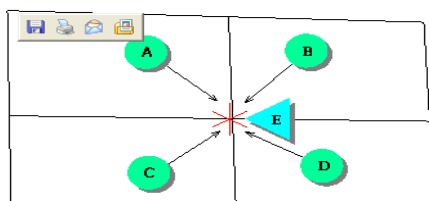
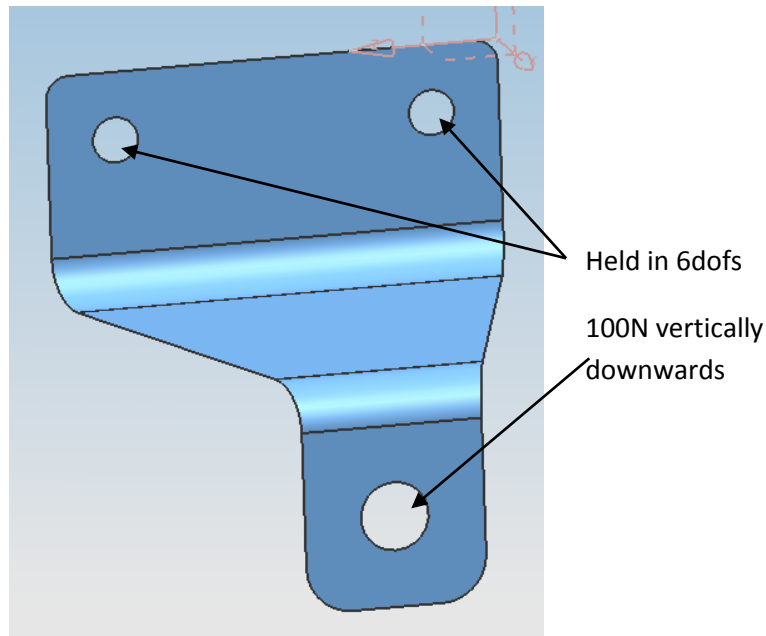


Diagram illustrating nodal averaging at a node common to four elements

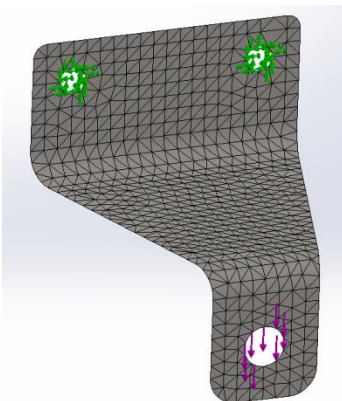
21. Looking at unaveraged results can also give you a very clear indication as to how good your mesh is. A good quality mesh should display gradual changes from one element to the next. Any significant jumps from one element to the next or plots which appear to be affected by the mesh, suggest that the mesh is not of good enough quality and is thus altering the results.

22. **File>close**

23. We are now going to consider the following



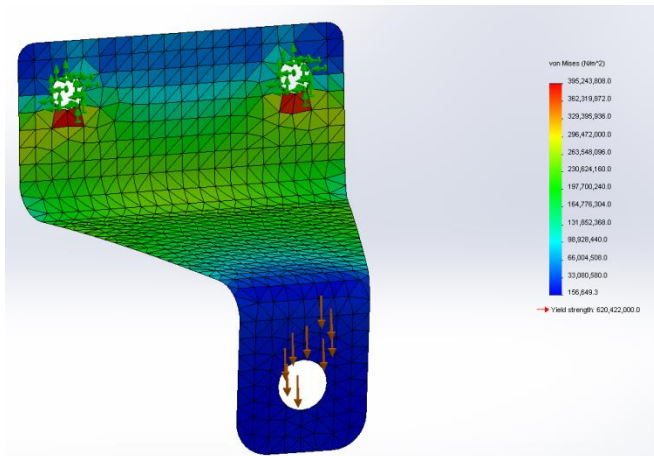
24. The above bracket is to be made from 1.5mm steel and it held such that the edges of the two small holes are held in 6dofs a force of 100N is applied in the downward direction to the larger hole. Using this handout to help you, set up this model using 5mm shell elements. When you mesh the part, simply drag a box around all the surfaces to pick them all at once. Your model should look as follows.



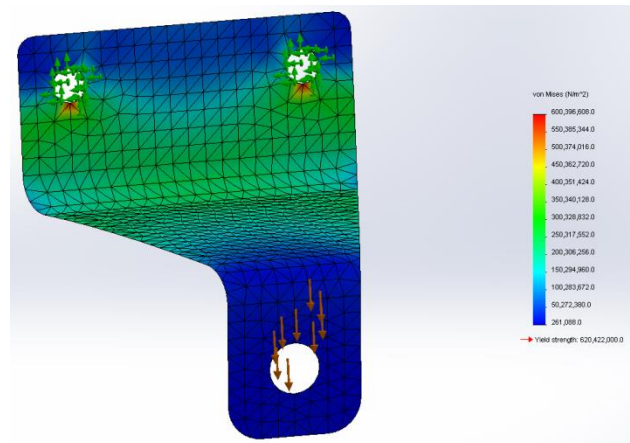
25. Solve this model

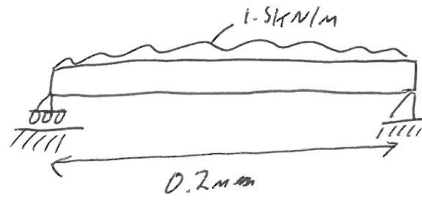
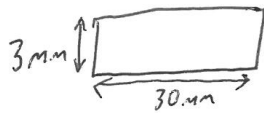
Your stress results should look as follows. Notice the areas surrounding the two constrained holes. There are large jumps in stress from one element to the next, suggesting a poor quality mesh. This will lead to poor results. If the results were viewed as averaged, this could have been missed. It is obvious then that we need to improve our mesh with the tools that are available to us in Solidworks.

Smooth, unaveraged results (top surface)



Smooth, averaged results (top surface)





$$I = \frac{bd^3}{12} = \frac{0.03 \times 0.003^3}{12} = \underline{\underline{6.75 \times 10^{-11} \text{ m}^4}}$$

IN THE CENTRE

$$\begin{aligned} M &= \frac{wL}{2} \times \frac{L}{2} - \frac{wL}{2} \times \frac{L}{4} \\ &= \frac{wL^2}{4} - \frac{wL^2}{8} \\ &= \frac{wL^2}{8} \\ &= \frac{1500 \times 0.2^2}{8} \\ &= \underline{\underline{7.5 \text{ Nm}}} \end{aligned}$$

$$\begin{aligned} \text{NOW } y_{\text{max}} &= \frac{-5wL^4}{384EI} = \frac{-5 \times 1500 \times 0.2^4}{384 \times 206.94 \times 10^9 \times 6.75 \times 10^{-11}} \\ &= -2.34 \times 10^{-3} \text{ m} \\ &= \underline{\underline{-2.34 \text{ mm}}} \end{aligned}$$

$$\begin{aligned} \text{ALSO } \sigma &= \frac{My}{I} = \frac{7.5 \times 1.5 \times 10^{-3}}{6.75 \times 10^{-11}} \\ &= \underline{\underline{166.7 \text{ MPa}}} \end{aligned}$$