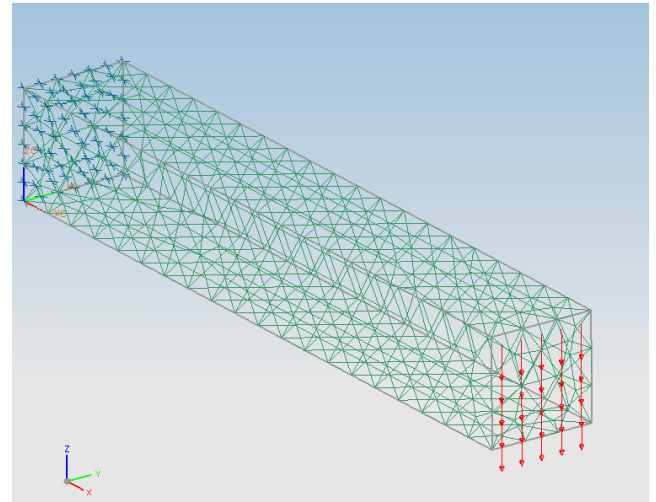


FEA v Hand Calculations Example

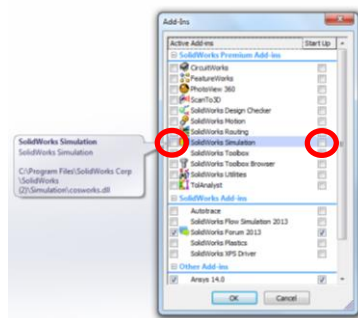


In this problem we are going to assess a steel beam which is 30mmx30mm in cross section and 200mm long. This beam will be fully fixed at one end and a load of 8kN will be applied to the opposite end in the vertical direction. We can then compare this to the theoretical hand calculations to demonstrate the similarities (and differences).

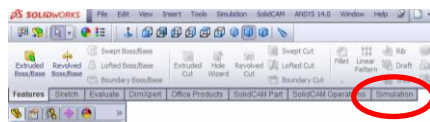
1. Start Solidworks
2. **File>New**
3. Make sure that Part is selected. OK.
4. **File>Save As** (Create a directory locally on the C drive and save your work here)
5. Give your model a meaningful name
6. **Save**
7. The first thing that you must do is make sure that the Simulation Add-in is activated



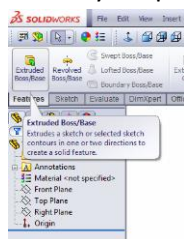
Add solidworks simulation and also tick the box so that it is added at start-up in future



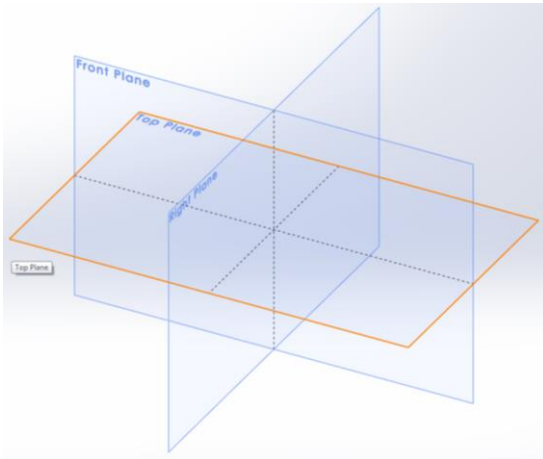
Note that a simulation tab is now added



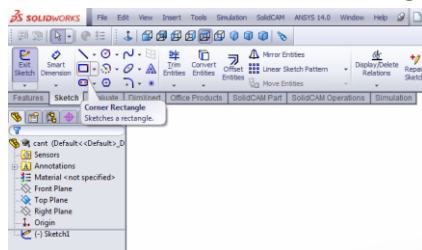
8. Create your part using **Extruded Boss/Base**



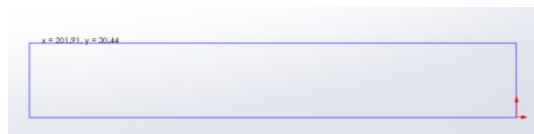
Choose the top plane on the screen



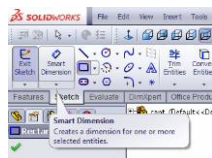
Go into sketch and draw a rectangle by choosing two corners



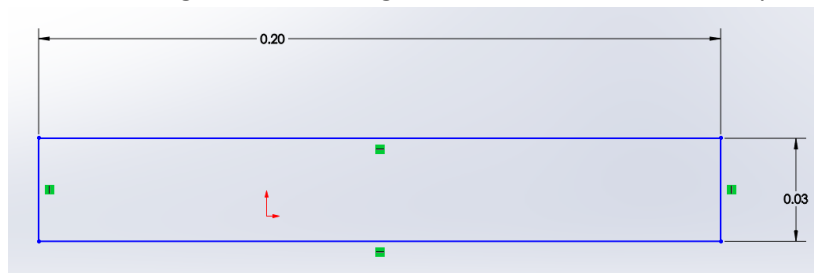
Choose the first corner at the origin, and the second corner ROUGHLY 0.2m by 0.3m.



Now choose **Smart Dimension**



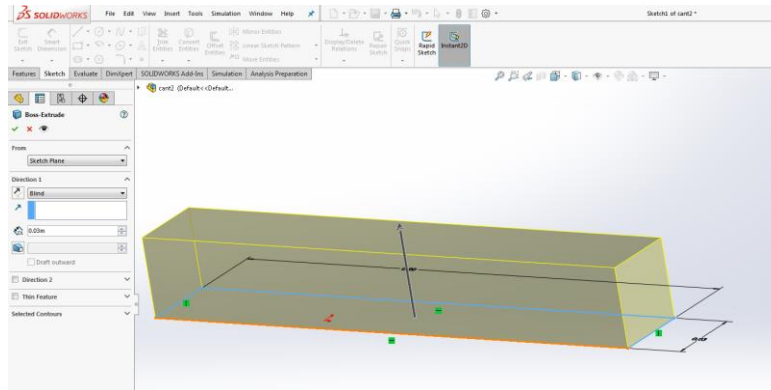
Choose the edges of the rectangle and dimension them 0.2m by 0.3m



Exit your sketch

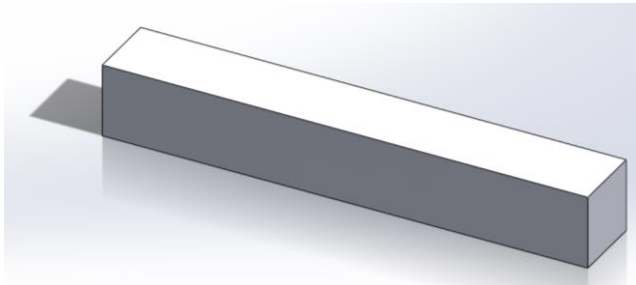


You now go back into the **Extruded Boss/Base** feature. Add your extrusion depth of 30mm.



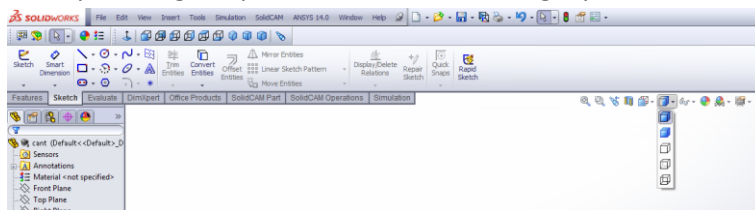
9. OK (Green tick)

Your part should now look as follows.



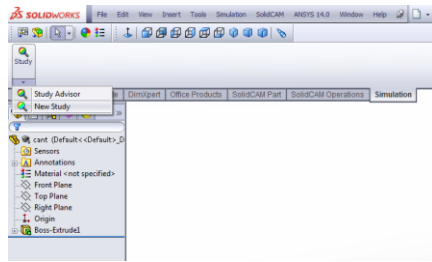
10. File>Save

11. Try viewing the part in different rendering styles

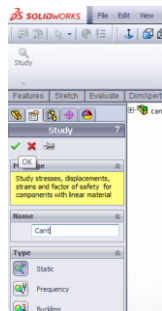


Now we have created our part we are ready to simulate our cantilever beam

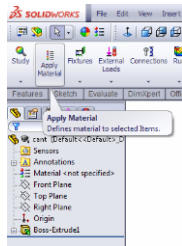
19. Simulation>Study>New study



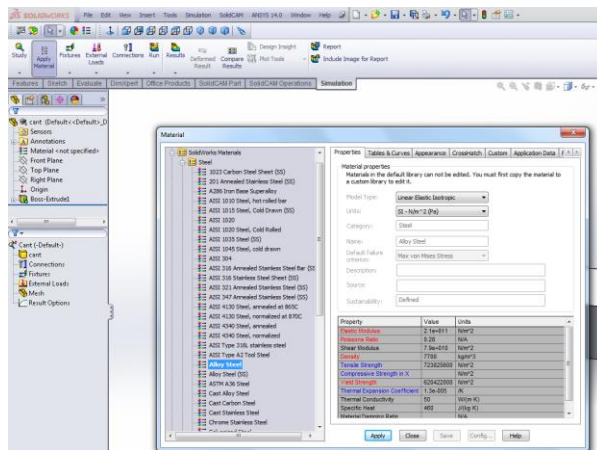
20. Call this study 'cant' and make sure that this is a 'static' analysis. **OK**.



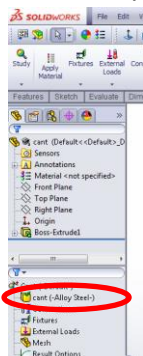
21. Apply your material



22. Choose 'Alloy steel'. **Apply>Close**.



Note that your part now has a material associated to it.

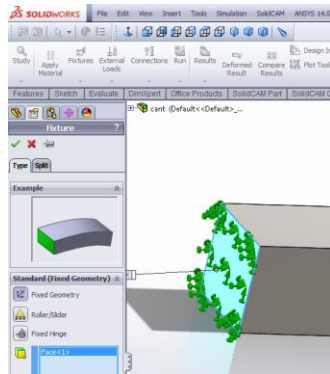


23. We are now going to fix one end of the beam in all 6 degrees of freedom.

Choose **Fixtures>Fixed Geometry**

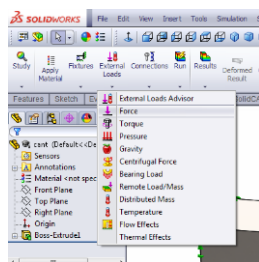


Choose the end face of your part. **OK**



24. We are now going to add our load.

Choose **External Loads>Force**



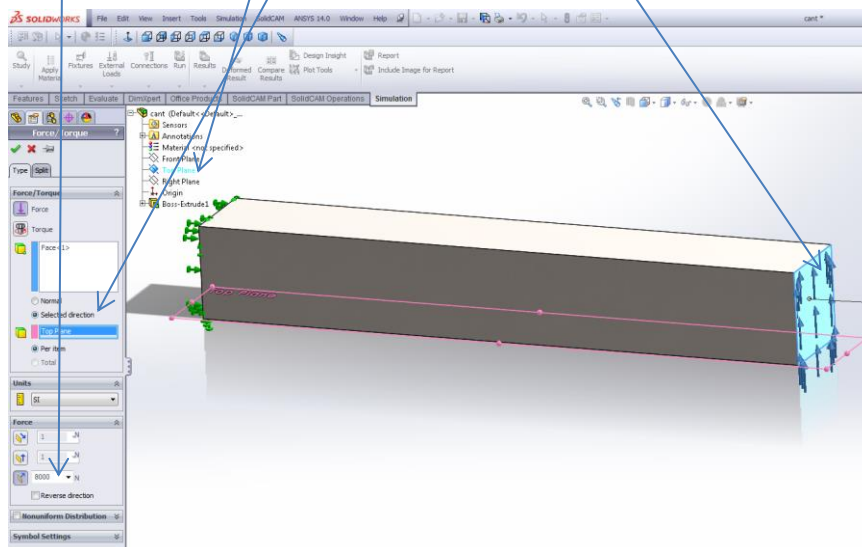
Choose your face where the load is applied

Choose a selected direction

Choose the top plane (or any plane normal to the force that you want to apply)

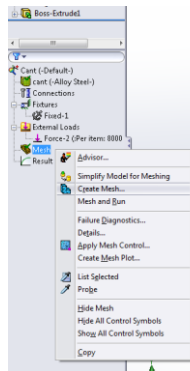
Specify 8000N

OK



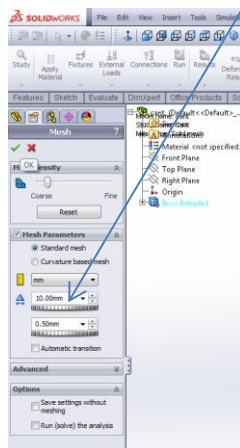
25. We are now ready to mesh our part. Initially we will use 10mm elements.

RMB on your mesh and choose **Create mesh**

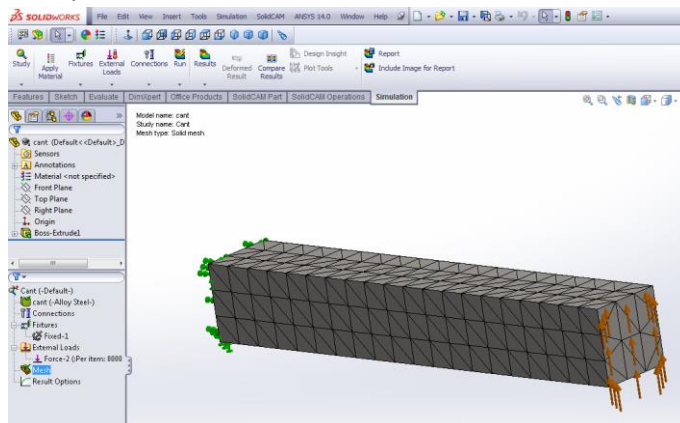


Change your global size to 10mm

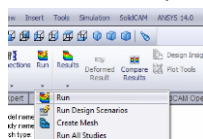
OK



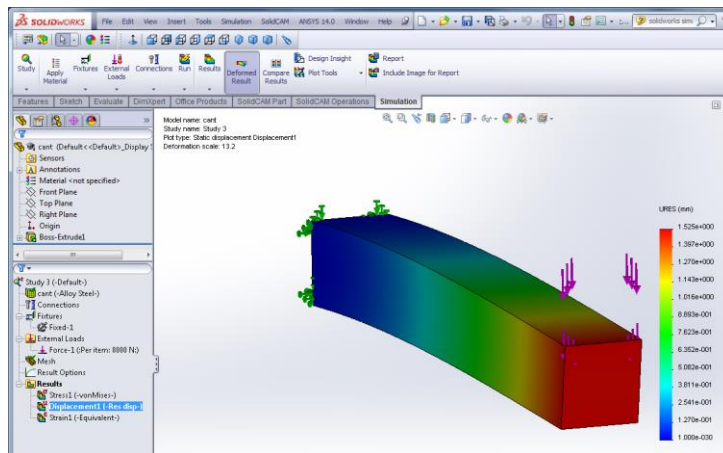
26. Your part should now look as follows



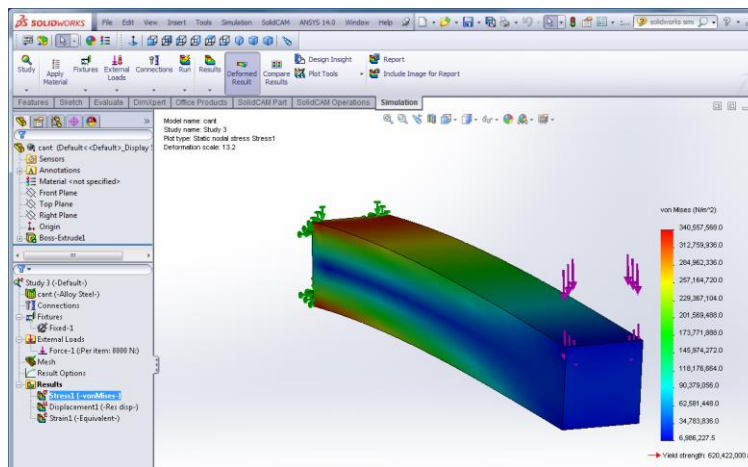
27. Run this analysis



28. What is the maximum deflection of this beam?



How about the stress? (Note the neutral axis)

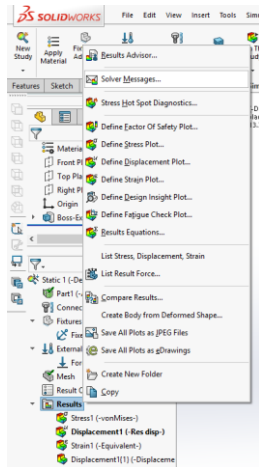


We will be looking at the differences between the FE model and the hand calculations in next weeks class. One of the things that we will be considering is the influence of the element size on the results. In order to do this fill in the following table.

Element Size (mm)	Maximum Stress (MPa)	Maximum Deflection (mm)	Solution Time (s)
1			
2			
5			
10			
20			
30			

In Order to see the solve time, nodes and DOFs:

RMB on the results and choose solver messages.



The following information is then displayed.

