

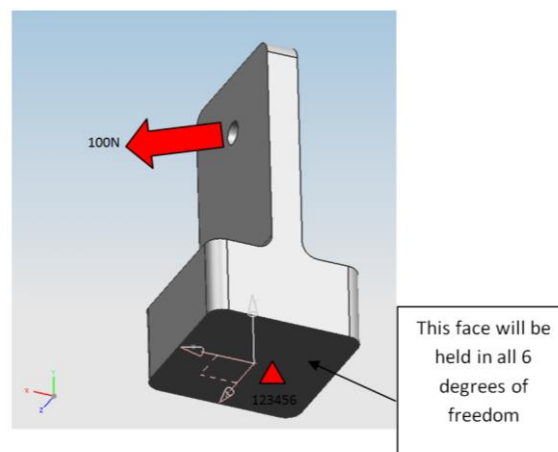
Mesh Refinement

In the previous exercises we have seen the need to refine our mesh in areas of high stress gradients, that is, where the stress levels change significantly, we need more elements to accurately capture this. In these exercises we will look at ways of refining our mesh.

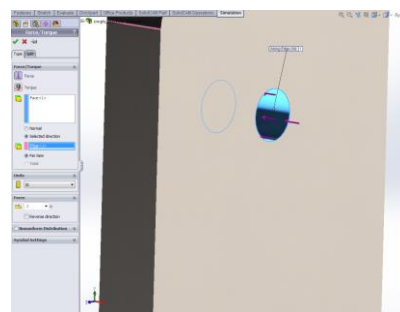
Manually refining our mesh

In this exercise we are going to load a simple bracket such that the fillets of that bracket are the areas of high stress concentration, thus we will need to refine the mesh in that region.

1. Open simple_bracket2.sldprt
2. We are going to restrain the base in all degrees of freedom and apply a load of 100N in the +x direction to the hole as shown below;

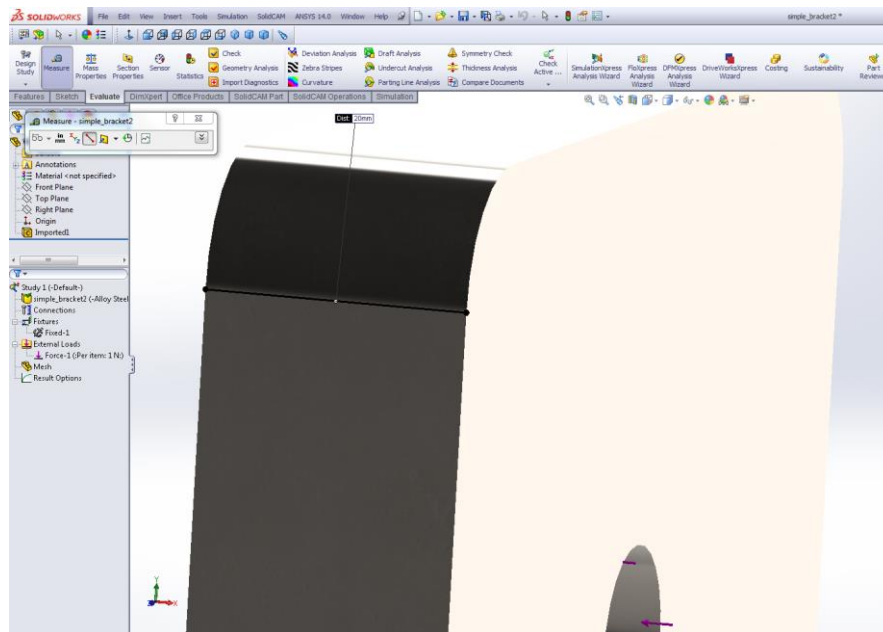


3. Create a new simulation study > Static > ok
4. Apply your material (Alloy Steel)
5. Fix the base in degrees of freedom
6. Apply your force of 100N in the x direction

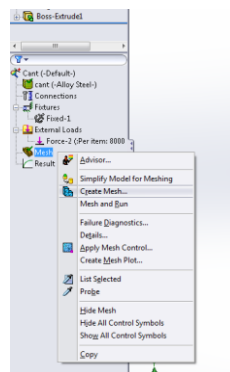


7. We now need to mesh our part. As previously discussed, in order to do this we need a least three elements through the thickness when using solid elements. Hence we must first measure the thickness of this part.

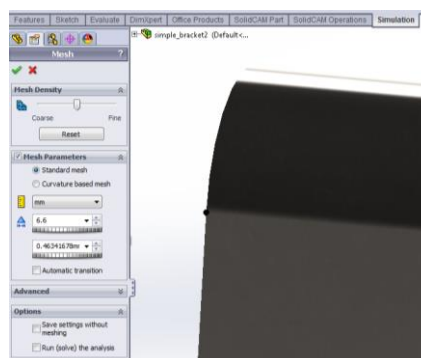
8. Choose the 'evaluate' tab then 'measure'. As you can see, this part is 20mm thick, hence our starting point for our element size will be 6.6mm.



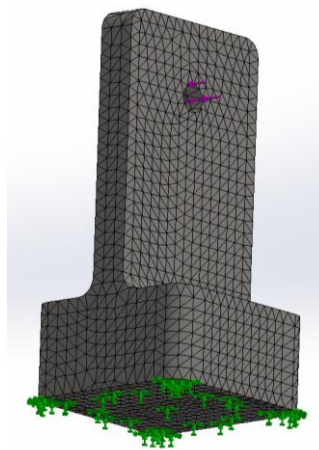
9. RMB on your mesh and choose **Create mesh**



Set your mesh size to 6.6mm and tick ok.



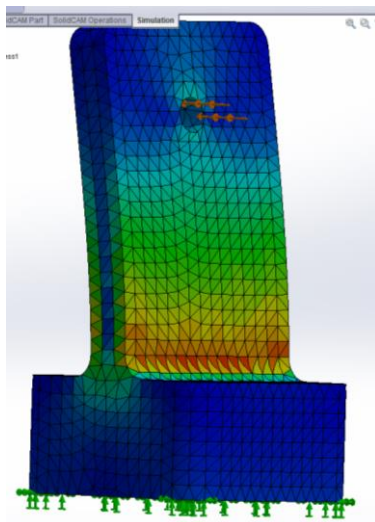
10. Your model should now look as below and be ready to solve.



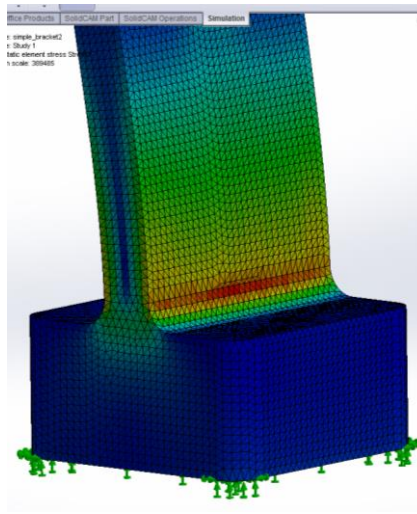
11. Save

12. Solve

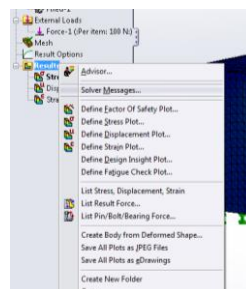
13. Your solution should look like the following. Change your display so that you are looking at the underlying mesh and so that the stress results are not averaged. Note that the maximum stress is approximately 1.4MPa. Note also the fact that the mesh clearly needs refining around the fillet.



14. One solution to this would be to simply remesh the part with a much smaller mesh as shown below with a 3mm mesh. You can simply right click on the mesh and click on create mesh and choose 3mm this time. Notice that when you do this the results for the 6.6mm mesh are automatically deleted.



Notice that the maximum stress has increased and there is less of a jump from one element to the next suggesting that these results are much more accurate. Notice, also however that the job has taken much longer to solve. You can check this by right clicking on the results and choosing solver messages. Re-mesh your part using a 6.6mm mesh and compare the results.

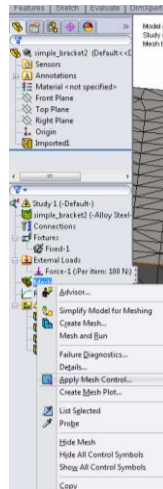


You should see the following;

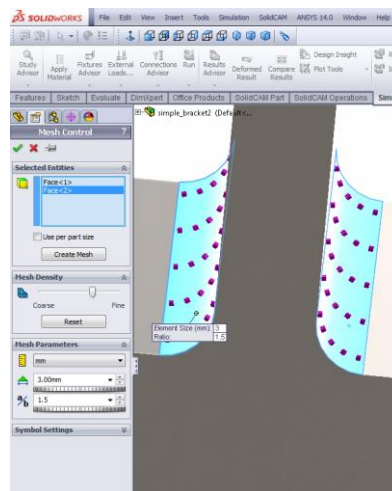
Mesh Size (mm)	No. of nodes	No. of Elements	Degrees of freedom	Time to solve (s)	Maximum stress (MPa)
6.6	29244	19173	84645	2	1.39
3	248264	173836	730989	15	1.65

Whilst we are converging on the solution, the runtime has increased by a factor of 7. Clearly we just need to refine the mesh in the areas of high stress gradient.

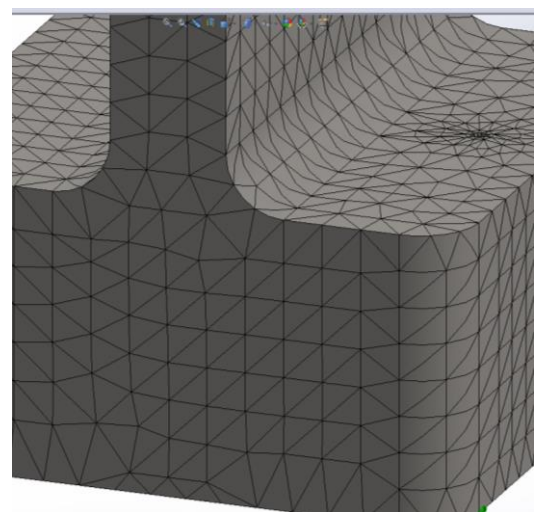
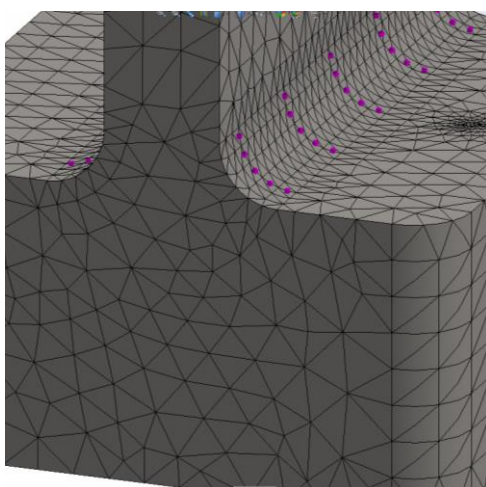
15. Remesh your part with a 6.6mm mesh, however, this time once you have done this right click on your mesh again and choose 'apply mesh control'



16. You can now click on the fillets of interest and specify a 3mm mesh just on these faces. Ok.



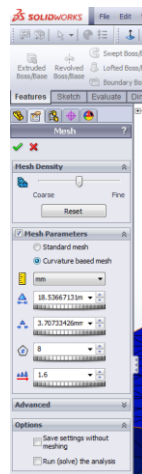
17. Once you have applied this mesh control, remesh your part with a 6.6mm mesh and notice the difference



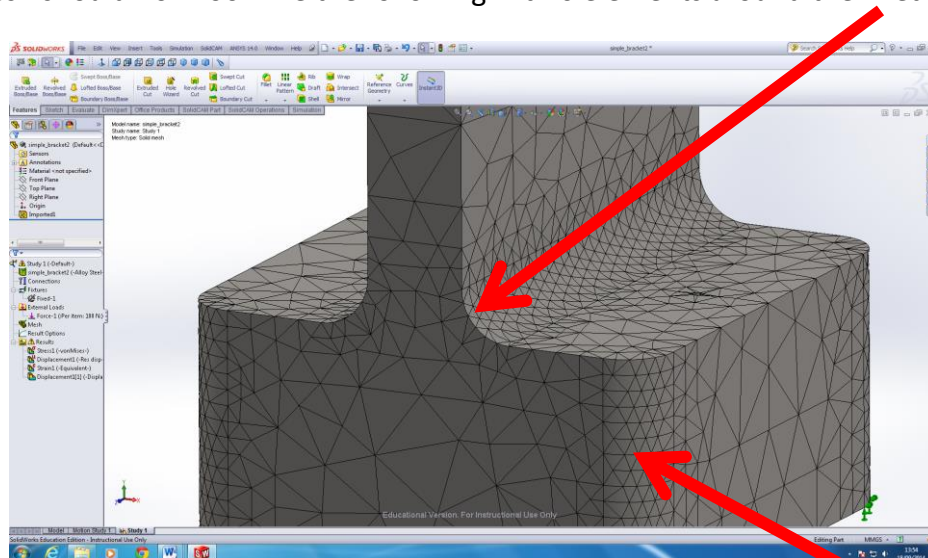
18. Now solve this model and compare the results

Mesh Size (mm)	No. of nodes	No. of Elements	Degrees of freedom	Time to solve (s)	Maximum stress (MPa)
6.6/3	49528	33595	145497	4	1.68
3	248264	173836	730989	15	1.65

19. We are now getting a similar answer in a quarter of the time. However, this method relies on you knowing where the stress concentrations are likely to be. One way to get around this could be to use a 'curvature' based mesh, rather than a 'standard' mesh that we have been using. Here we can specify how many elements would be required around a circle. In our original mesh around the fillet which is a quarter of a circle we have 2 elements, i.e. 8 elements around the circle. This is the default value as shown below. Increase this to 24 so that we have 6 elements around our fillet.

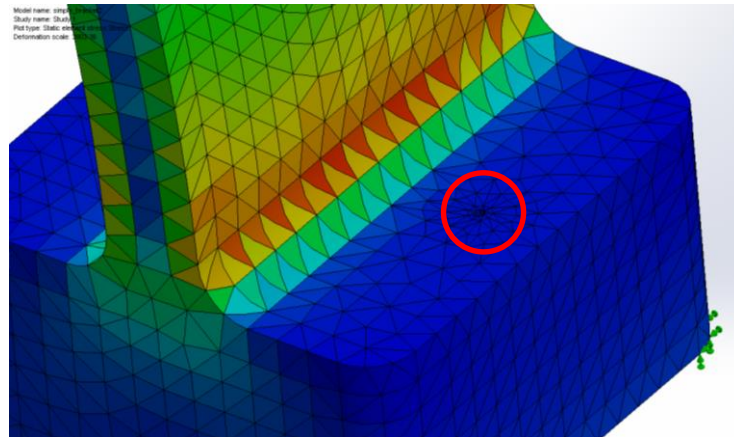


Your mesh should now look like the following with 6 elements around the fillet

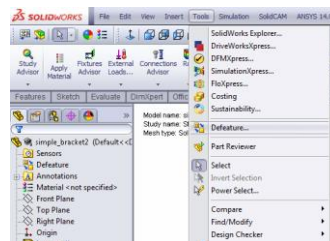


Notice also however that the mesh has been refined wherever there is a curvature, potentially leading to refinement in unnecessary areas.

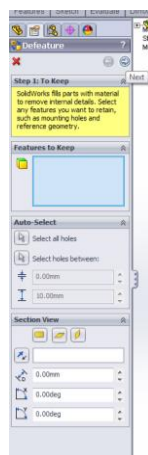
Note also that this part has a small feature in an area of low stress gradient. This forces small elements around this feature thus increasing the run time.



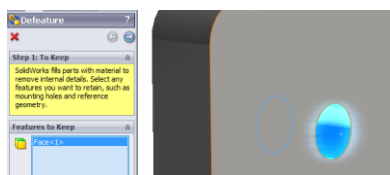
One way to simply remove this feature would be to use the 'defeature' tool



Read the instructions.

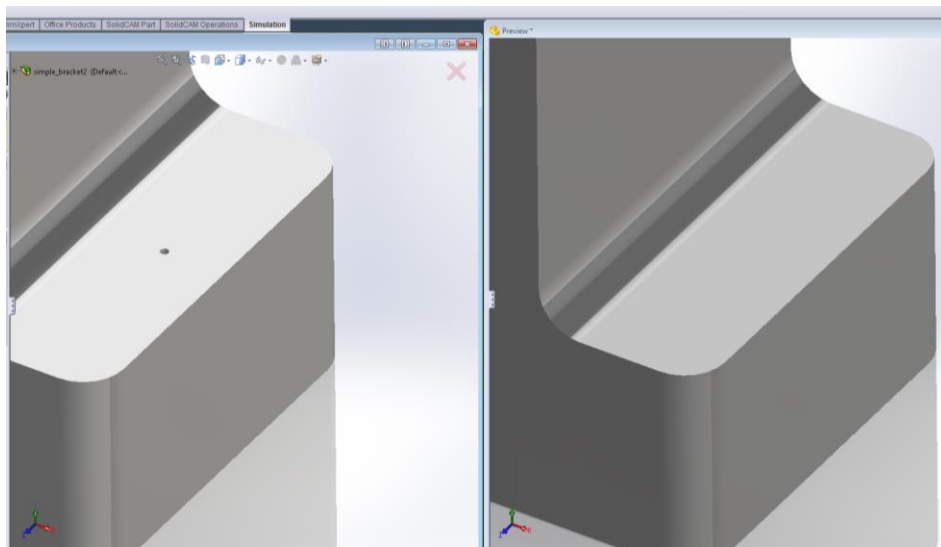


Keep the feature where we are applying the load.

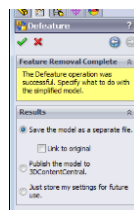


Click on next

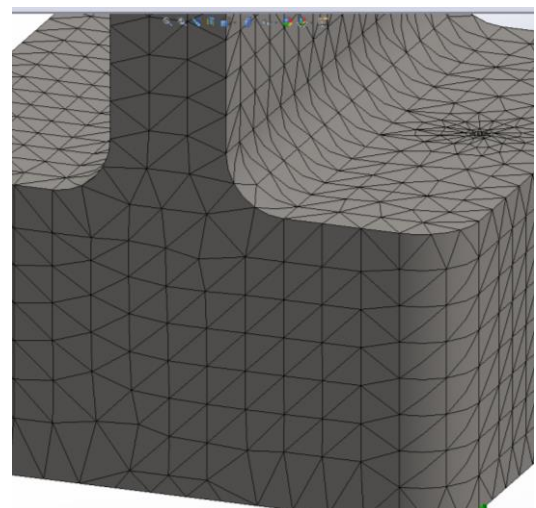
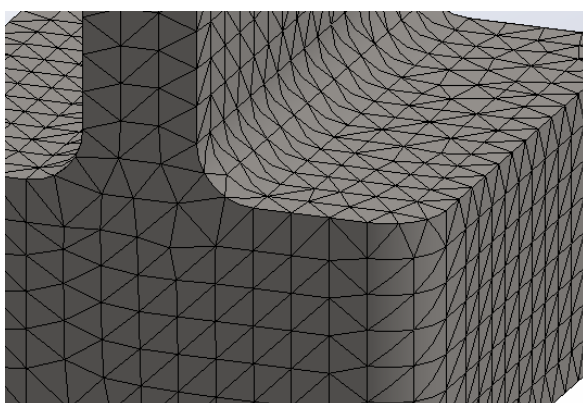
Notice the difference



Next



Save this file as 'simple_bracket2_simplified.sldprt'. Now solve this simplified model and compare the number of elements and nodes



Mesh Size (mm)	No. of nodes	No. of Elements	Degrees of freedom	Time to solve (s)
6.6	29244	19173	84645	2
6.6 defeatured	28181	18441	81456	2

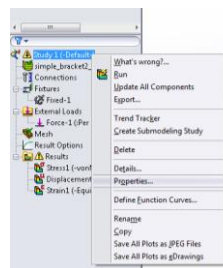
Note that the number of nodes and elements has reduced.

H-method adaptive meshing

It is possible to force the solver to look at the change in stress from one element to the next and if these are outside of a certain tolerance (called the Error Norm which is the difference between the element stress and the average of the nodal stresses), then the solver locally remeshes the model in this area and resolves the model and then rechecks the error. If the strain energy is still outside the tolerance set then the model remeshes again and so on. Obviously this automates the process but does mean that a model will potentially run several times before convergence is reached.

20. Remesh your original part with a 6.6mm mesh.

21. RMB on your study and choose properties



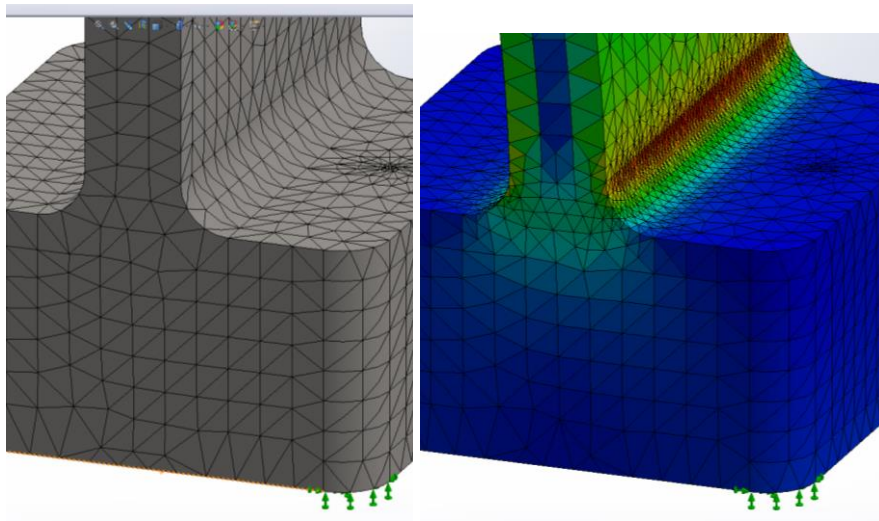
Click on the adaptive tab. Choose h-adaptive. Note that by default the accuracy is set to 98%. Set the max number of loops to 5. OK



Now resolve this. You should see that Solidworks iterates until a solution is reached and you see a message similar to the following

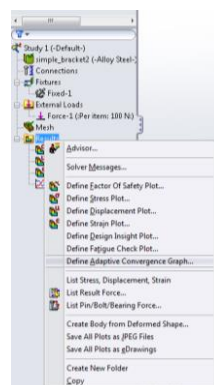


Notice how solidworks has automatically refined your mesh in the areas of high stress concentration.

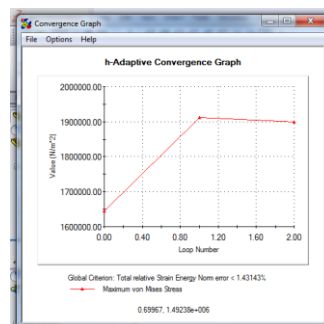


You can also show how the stress as converged.

RMB results > Define adaptive convergence graph



Choose the maximum stress > ok.



P-method adaptive meshing

Alternatively, you can specify the p-method under adaptive meshing properties in order to force the solver to increase the order of the elements locally around any areas of high stress



Try this method and compare your results.

Plotting the error norm

RMB Results > Define stress plot > Energy Norm error

