# SimSolid “Real World” Validation Manual For Stress Accuracy

Richard King 04/09/19

## Abstract

This manual contains a series of SolidWorks models with static simulation studies that were run in Altair SimSolid™. Some of them are complex, but an accurate solution was obtained in SolidWorks by the conventional procedure of placing mesh controls in the regions of highest stress gradient and successively refining until the solution converged. This provides a suite of accurate stress results to compare SimSolid’s results against. In all cases SimSolid’s stress results compared to the SolidWorks comparison results within 10 percent.

## Introduction

SimSolid is a very easy to use and fast product for structural analysis using novel alternative technology- a meshless approach that based on similar theory to FEA. This approach can be shown theoretically to converge to correct solutions, as discussed in the whitepaper on the technology [1], where it is shown that the mathematical foundation for the method used in SimSolid is on as solid of a foundation as FEA. But in stress analysis the quantity being calculated is the derivative of the directly calculated variables (displacements) so is more difficult to achieve accurate results. This is true of FEA codes as well as SimSolid. In FEA codes, the solution is to have a fine enough mesh in regions of high stress gradients. This can sometimes be done adaptively using automatic mesh refinement, but often requires one or more user interventions, for example to create mesh controls, as described in appendix 1. On complex models this can be tedious and time consuming.

SimSolid has an automated approach to address this issue. In a first step, the model is analyzed with automatically chosen displacement functions to accurately calculate displacements, assure continuity at all interfaces and satisfaction of boundary conditions. This captures overall load paths and reaction forces accurately [2]. In the second step, it adapts to local features where there are high stress gradients by introducing additional functions it those areas. From a user’s standpoint, this “two stage workflow”, as it is called in the training material, is very easy:

1. Assign materials, loads, and constraints directly to the CAD model. Hit the “Solve” button to run the model with defaults settings. This run is typically very fast. When complete, examine the results to make sure the deformation looks reasonable.
2. Turn on solution settings to adapt to features by clicking a single checkbox shown in the figure. If the model contains thin shell or sheetmetal type parts, an additional button, “adapt to thin solids” can be checked. This is all done for the entire model for a single part model. For an assembly, the user picks specific parts in which high stress has been observed to use for adapting to features or thin solids.

I have used various finite element codes with adaptivity, including SolidWorks Simulate, Autodesk Fusion 360, and Mechanica (of which I was co-inventor and one of the authors). In my opinion SolidWorks Simulate is a good quality commercial code, with a robust automesher, so it was a good choice for comparing the accuracy of SimSolid. But all FEA codes have the drawback of difficulties associated with meshing, so none of these is as easy to use in practice as the two stage workflow for SimSolid described in the Altair SimSolid Fast Start Training Guide [2].

## Solutions for Validation

Most FEA codes have validation manuals, and SimSolid does as well, but they typically contain test cases that tend to be simple, so that theoretical solutions are available. I consider them “necessary but not sufficient”. To further investigate accuracy, this manual uses a variety of more complicated models for which in most cases there is not theoretical solution for comparison. Instead, an accurate solution for all these models was obtained in SolidWorks, by using automatic mesh refinement and by the conventional procedure of placing mesh controls in the regions of highest stress gradient and successively refining until the solution converged. This provides a suite of accurate stress results to compare SimSolid's results against. The procedure for obtaining accurate FEA results in SolidWorks simulation is described in appendix 1. The specific procedure used to obtain accurate stresses in SimSolid is described in appendix 2.

## Note on Singular Stresses

Many situation such as reentrant corners lead to stresses that are singular (theoretically infinite). Some examples are shown in appendix 3. If the maximum stress in your model occurs at one of these it is best to fix the situation, for example by creating a fillet. Alternatively, if you are interested in the stress elsewhere in the model, it can be examined using a probe. This was necessary, for example, for the hanger-beam tutorial.

## Model Location

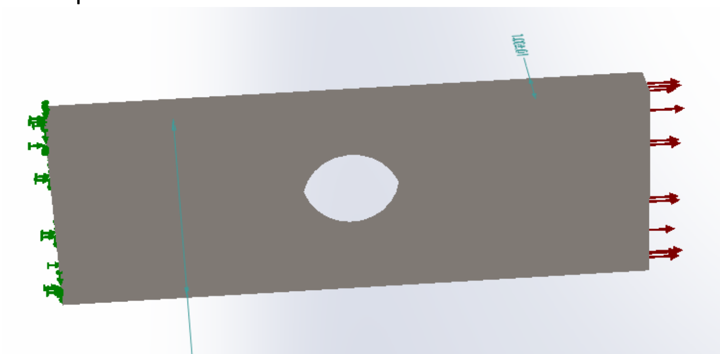
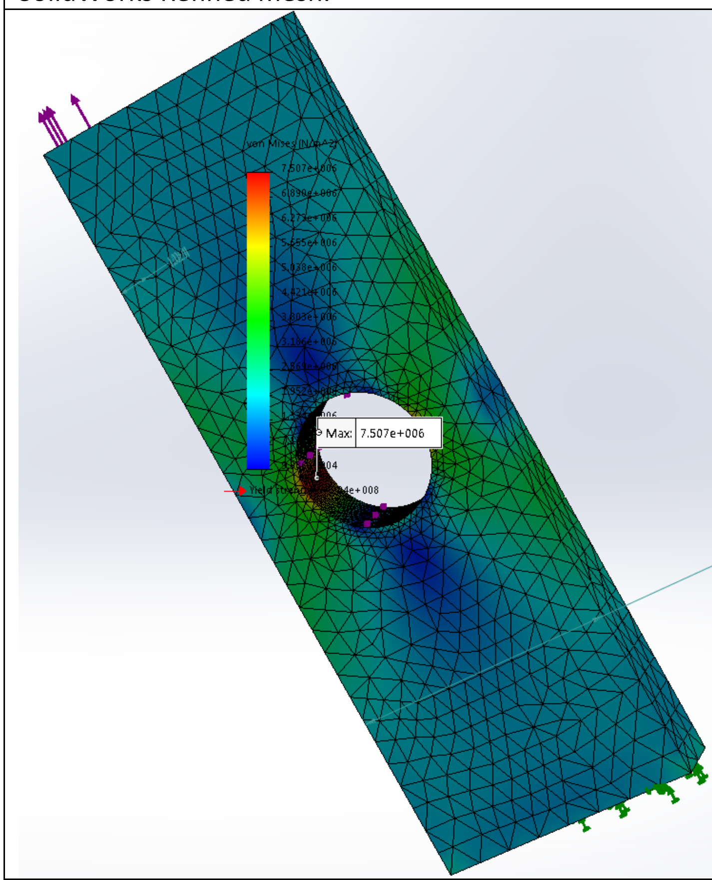
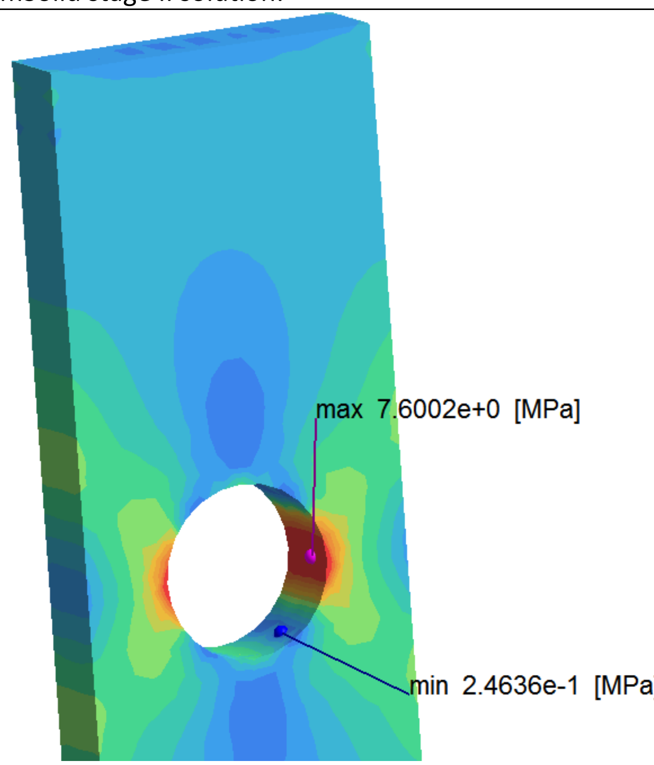
All the models described in this document are in the folder "valmanModels". There is a SimSolid workspace (.ssp) file for each, as well as SolidWorks part and/or assembly files.

## Summary

| Model Description/name | error in maximum stress |
|---|---|
| Single Part models | |
| 1. Plate with hole (pwh) | 1.2% |
| 2. Bar with fillet in bending (petebendfillet) | 0.8% |
| 3. plate with narrow elliptical hole (Ingliss) | 1.5% |
| 4. model with thin slot (model with thin slot) | 1% |
| 5. bar with edge slits (edgeslit) | 1.6% |
| 6. stepped model with fillet and slot (stepfillet) | 3.2% |
| 7. Crank (crank) | 5.3% |
| 8. Tube joint with thin solids (newtj) | 2.0% |
| 9. Spherical pressure vessel with hatch (spherehatch) | 4.4% |
| 10. Canoe paddle (canoepaddle) | 0.6% |
| 11. rectangular tank (recttank) | 5.9% |
| Assembly Models | |
| 12. multiple cylindrical columns with fillets (railcylassy) | 0.7% |
| 13. multiple bars with edge slits (edgeslitassy) | 2.5% |
| 14. testing machine and specimen (clevisassy) | 4.1% |
| 15. multiple plates with steps and fillets (filletassy) | 0.2% |
| 16. structure with thin plates and girders (floorsystem) | 8.6% |
| 17. pullup bar (SimSolid tutorial) | 3.5% |
| 18. hanger-beam (SimSolid tutorial) | 2.6% |

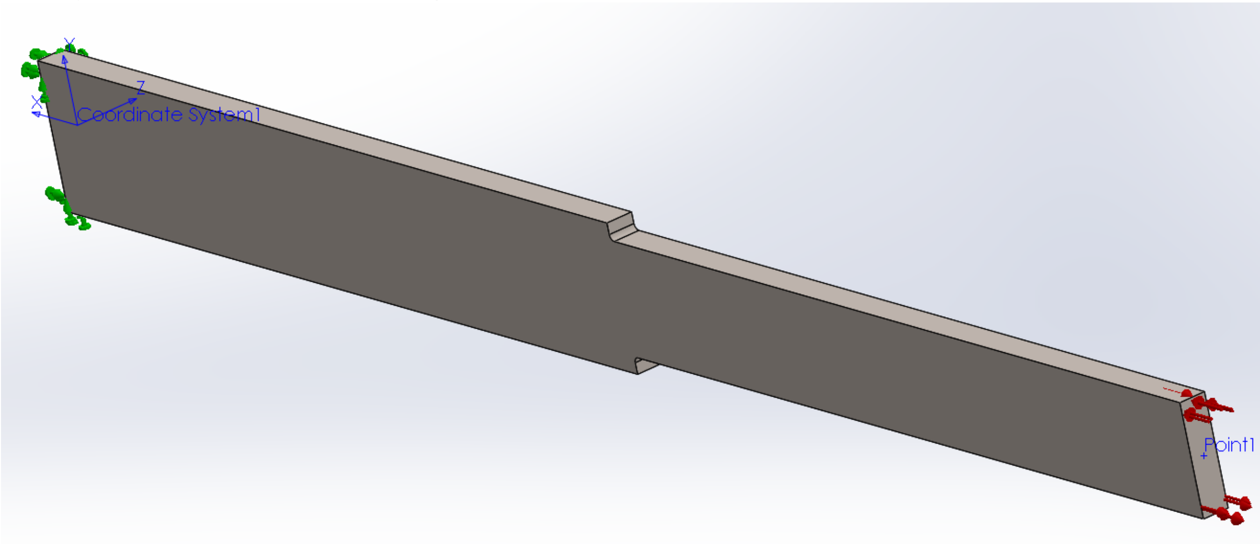
Solution Timing: Solution time was similar for all the smaller models. The last 3 models are relatively large and the timing for SolidWorks vs SimSolid is described below. For these models the total solution time (including meshing time in SolidWorks) is significantly less in SimSolid. For all models, considerably less user intervention was require in SimSolid to achieve an accurate result.

## Single Part Models

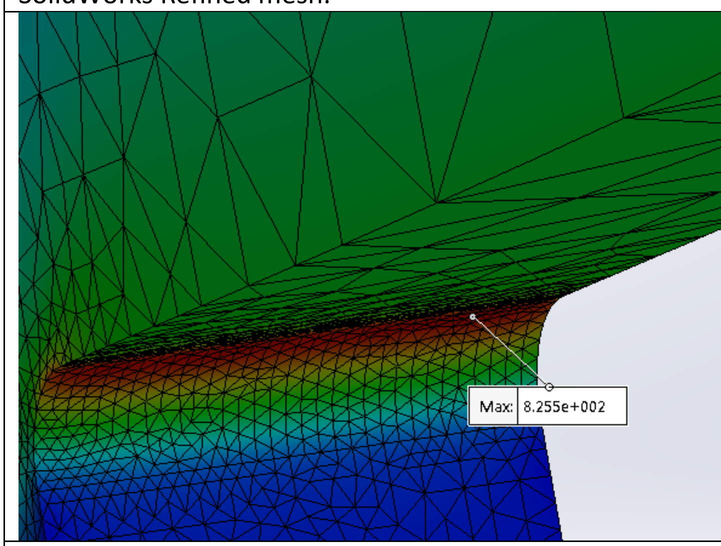
<table>
<tr><td colspan="3">Model: pwh</td></tr>
<tr><td colspan="3">Description: Plate with hole<br></td></tr>
<tr><td>SolidWorks Refined mesh:</td><td colspan="2">SimSolid stage II solution:</td></tr>
<tr><td><br></td><td colspan="2"><br></td></tr>
<tr><td colspan="3">Maximum Stress Result</td></tr>
<tr><td>SolidWorks Refined mesh:</td><td>SimSolid Stage II Solution</td><td>Error</td></tr>
<tr><td>7.5 Mpa</td><td>7.6 Mpa<br>(3 solution passes)</td><td>1.2%</td></tr>
<tr><td colspan="3">Notes:<ul><li>Theoretical Solution for this model is 7.6 Mpa [3]</li><li>The SimSolid results did not change significantly when “adapt to features” was turned on with 3 solution passes, so that was taken as the converged result.</li><li>SolidWorks adaptive mesh refinement result was accurate for this model</li></ul></td></tr>
</table>

Model: petebendfillet

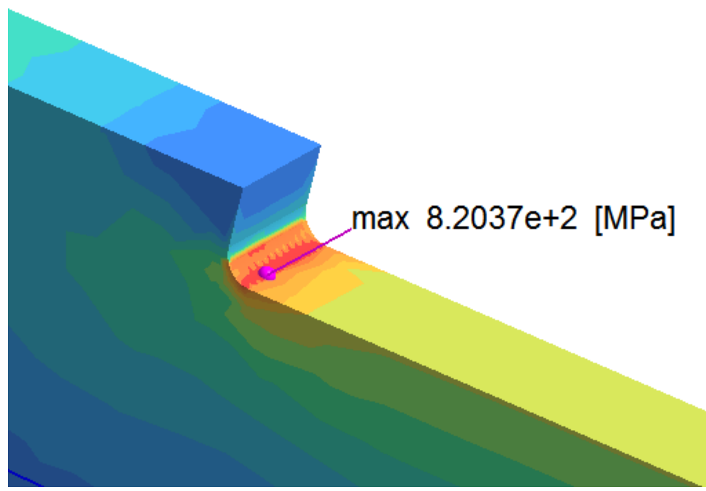
Description: Bar with fillet in bending




SolidWorks Refined mesh:




SimSolid stage II solution:




Maximum Stress Result

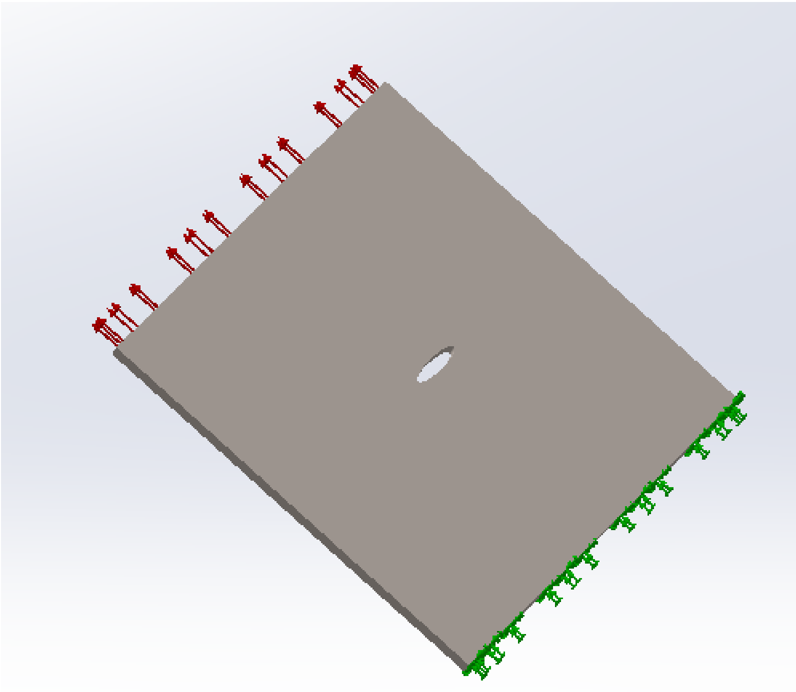
| SolidWorks Refined mesh: | SimSolid Stage II Solution (4 solution passes) | Error |
|---|---|---|
| 826 Mpa | 819 Mpa | 0.8% |

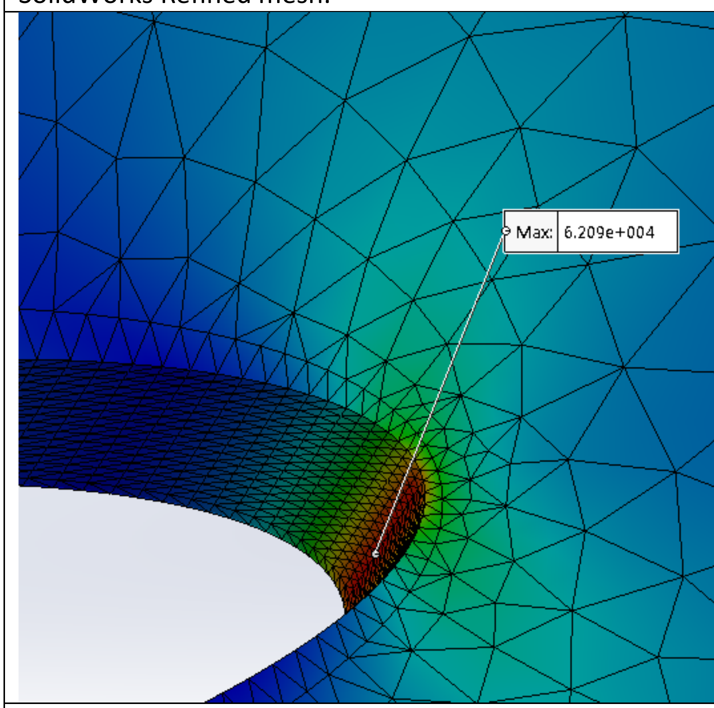
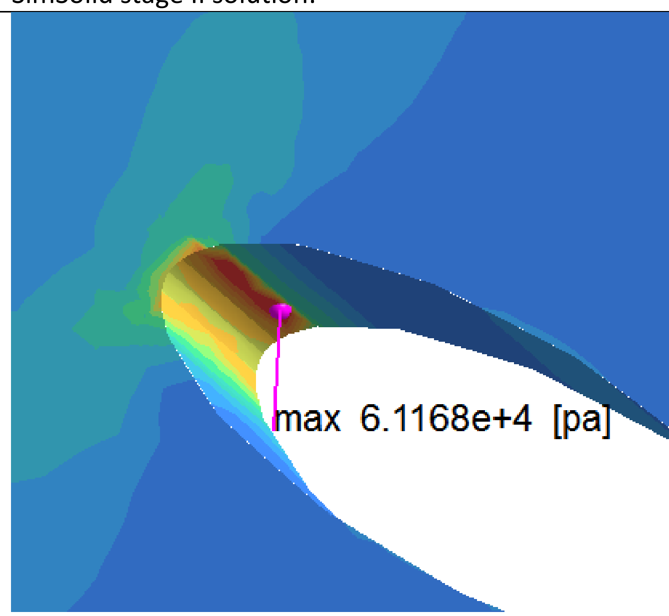
Notes:

- The handbook solution for this model is 823 Mpa [4]
- SimSolid Stage II Solution did not change appreciable between runs with 4 and 5 solution passes so the result from 4 passes was taken as the converged result
- SolidWorks adaptive mesh refinement result for this model was 771 Mpa which is 6.6% low. The SolidWorks Refined mesh result was obtained by applying a mesh control at the fillet and successively refining it manually.

Model: Ingliss

Description: plate with narrow elliptical hole



| SolidWorks Refined mesh: | SimSolid stage II solution: |
| --- | --- |
|  |  |

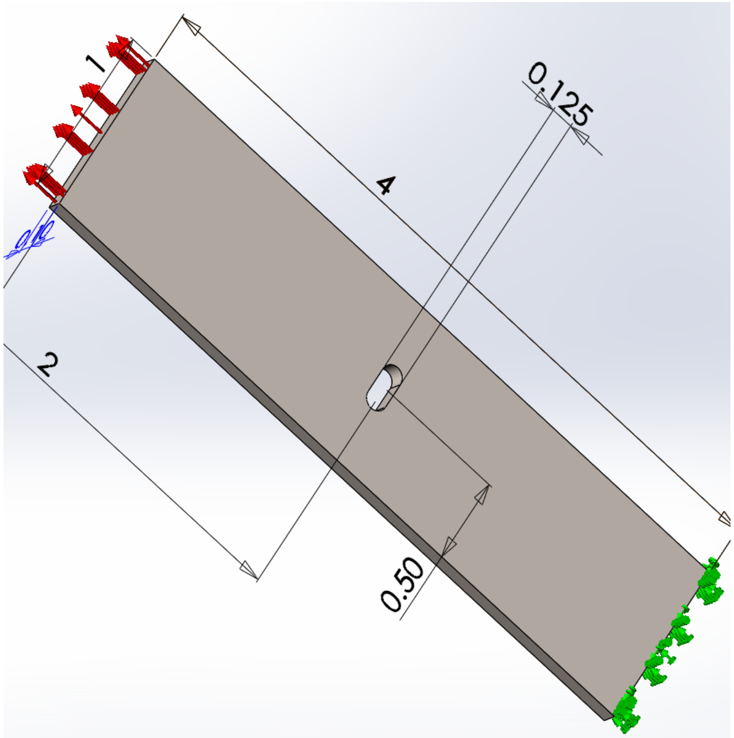
Maximum Stress Result

| SolidWorks Refined mesh: | SimSolid Stage II Solution (6 solution passes) | Error |
| --- | --- | --- |
| 6.21E4 Pa | 6.12E4 Pa | 1.5% |

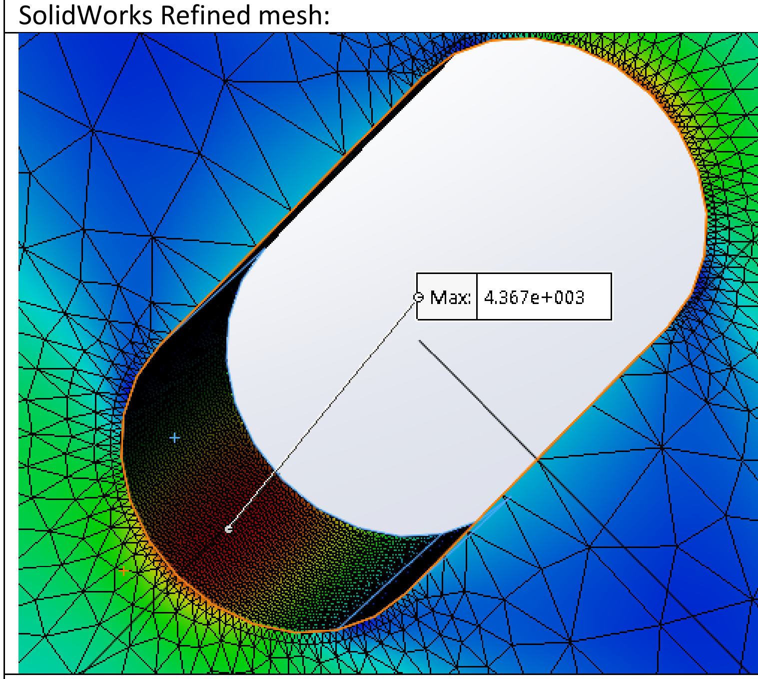
Note: SolidWorks adaptive mesh refinement result for this model was 5.66E4 Pa which is 8.9% low. The SolidWorks Refined mesh result was obtained by applying a mesh control at the hole and successively refining it manually

Model: thinslot

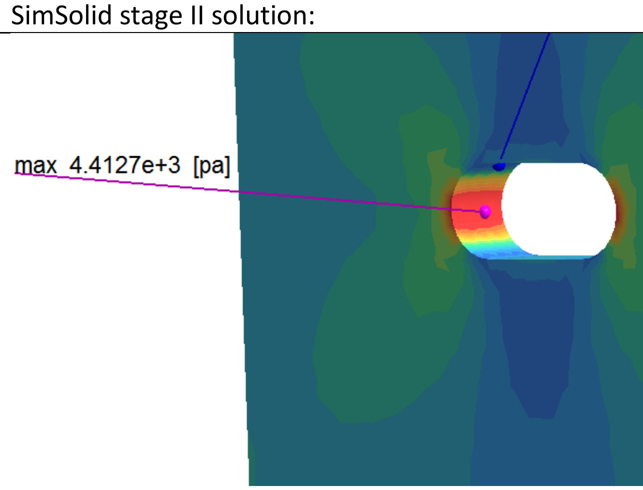
Description: model with thin slot




SolidWorks Refined mesh:

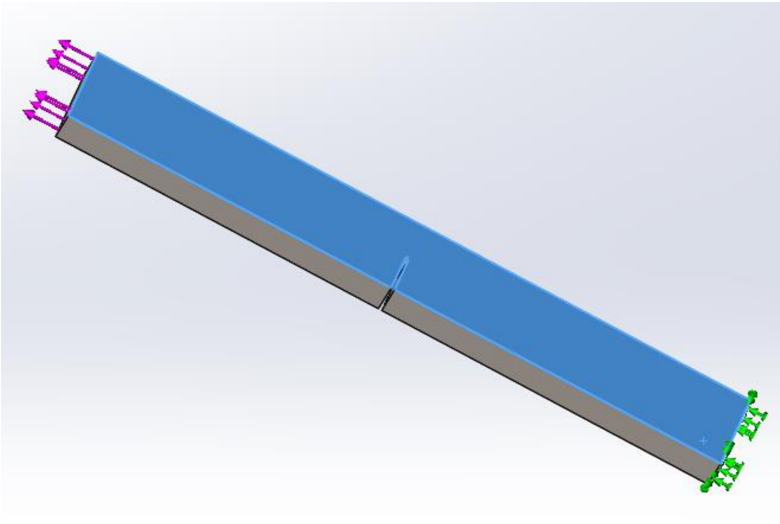
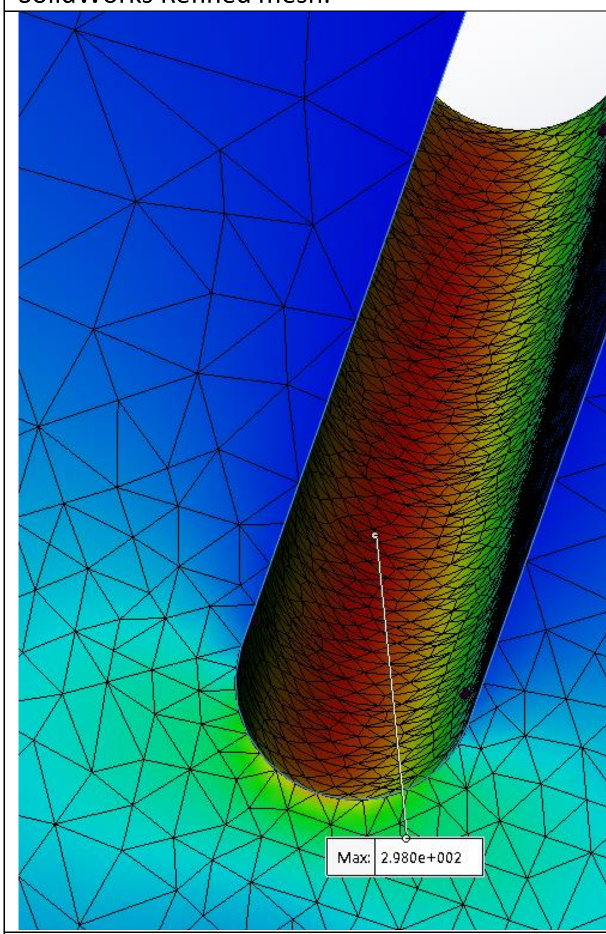
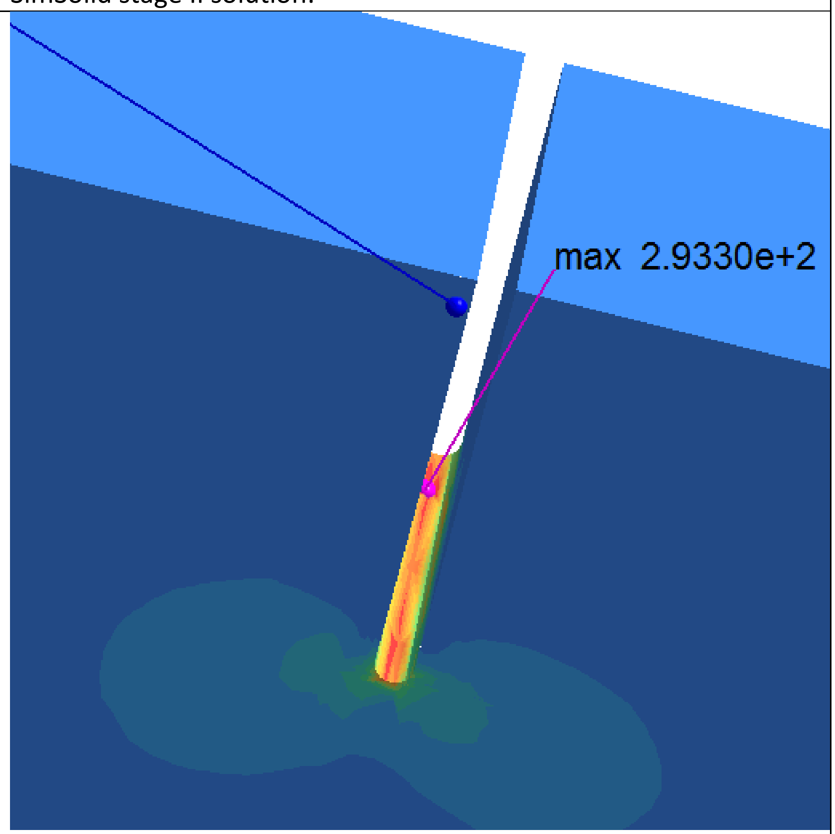


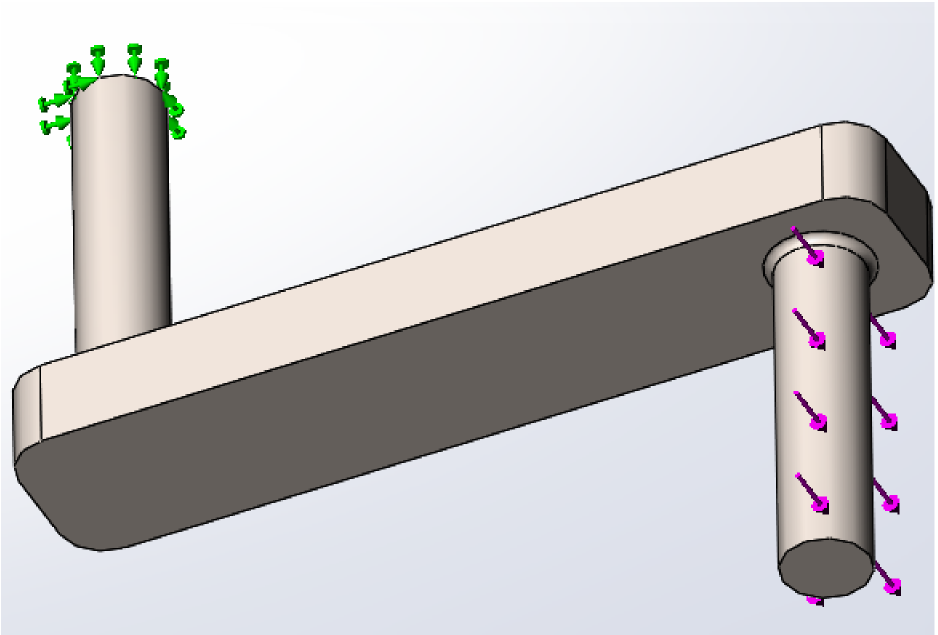
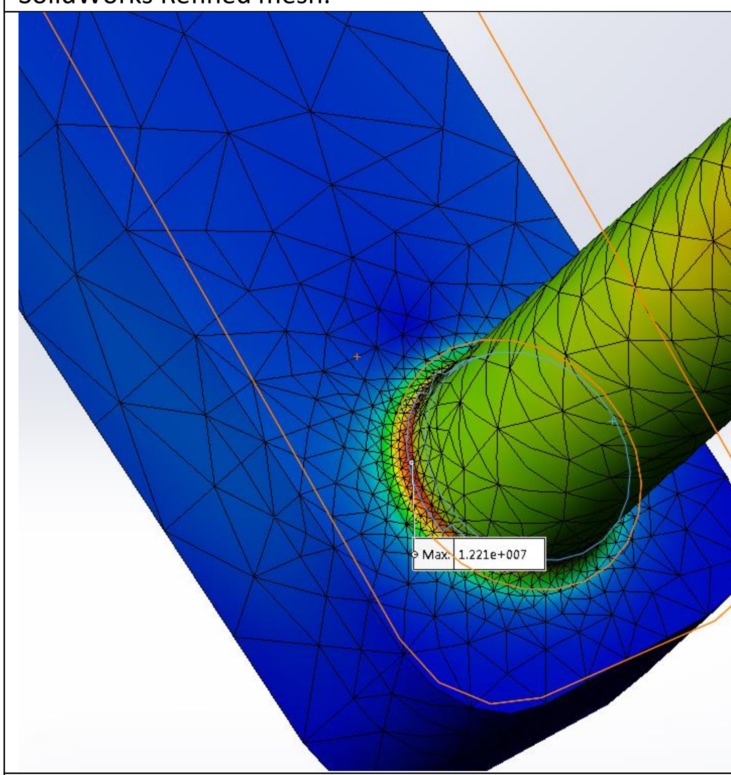
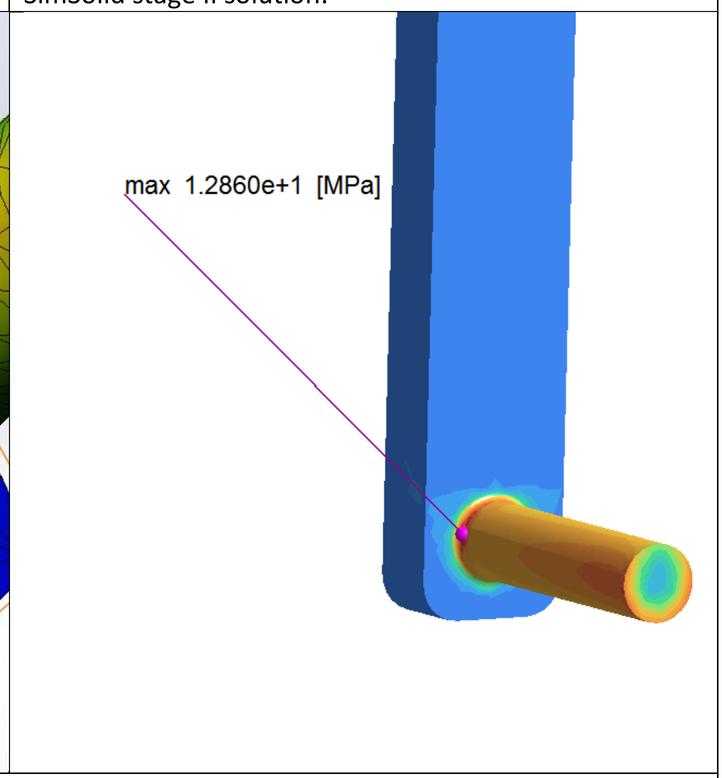

SimSolid stage II solution:




| Maximum Stress Result | | |
|---|---|---|
| SolidWorks Refined mesh: | SimSolid Stage II Solution (4 solution passes) | Error |
| 4367 Pa | 4412 Pa | 1% |

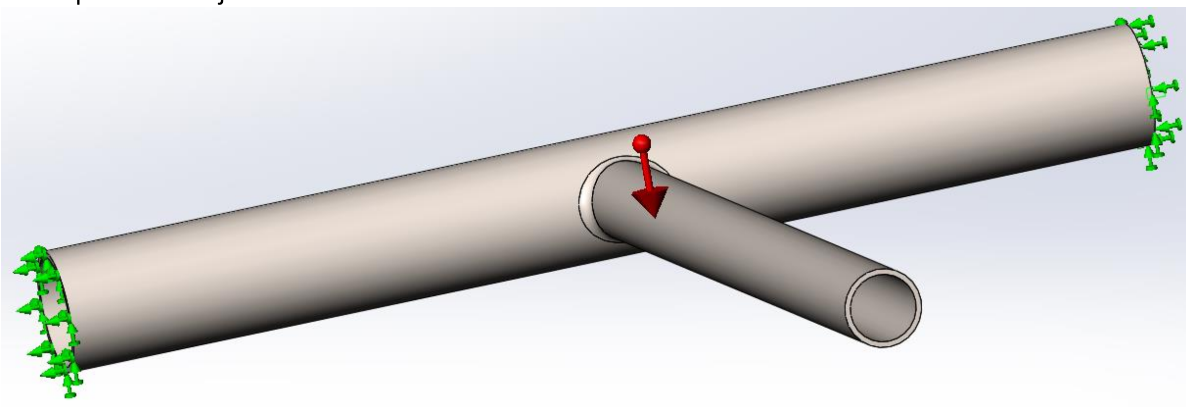
Note: SolidWorks adaptive mesh refinement was accurate for this model and agreed with the solution obtained by successive manual mesh refinement

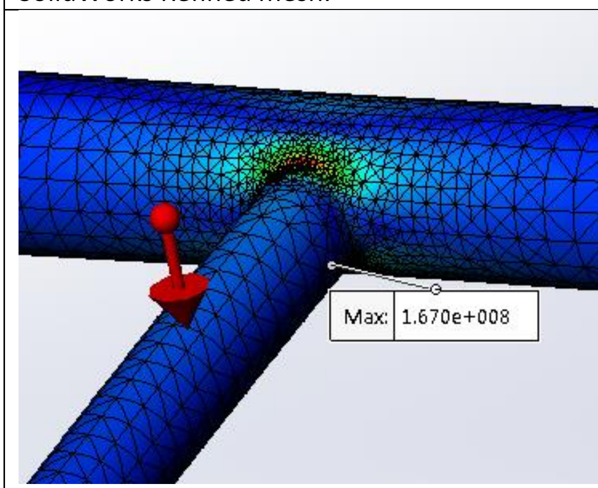
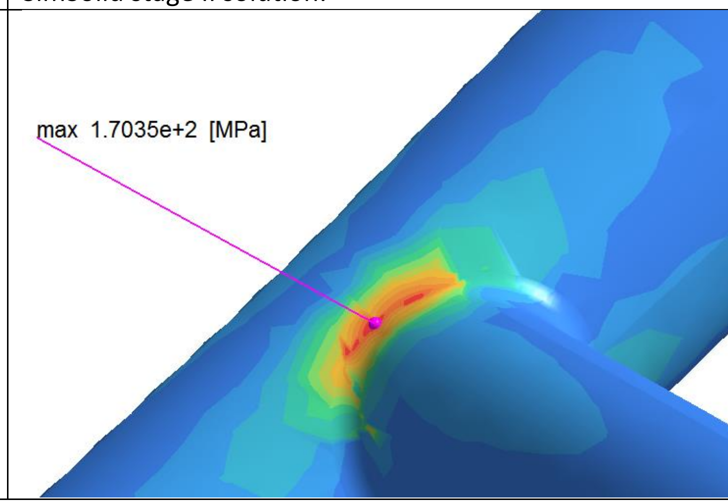
<table>
<tr><td colspan="3">Model: edgeslit</td></tr>
<tr><td colspan="3">Description: bar with edge slit<br></td></tr>
<tr><td>SolidWorks Refined mesh:</td><td colspan="2">SimSolid stage II solution:</td></tr>
<tr><td>
</td><td colspan="2">
</td></tr>
<tr><td colspan="3">Maximum Stress Result</td></tr>
<tr><td>SolidWorks Refined mesh:</td><td>SimSolid Stage II Solution<br>(5 solution passes)</td><td>Error</td></tr>
<tr><td>298 ksi</td><td>293 ksi</td><td>1.6%</td></tr>
</table>

<table>
<tr><td colspan="3">Model: Crank</td></tr>
<tr><td colspan="3">Description: Crank<br></td></tr>
<tr><td>SolidWorks Refined mesh:</td><td colspan="2">SimSolid stage II solution:</td></tr>
<tr><td>
</td><td colspan="2">
</td></tr>
<tr><td colspan="3">Maximum Stress Result</td></tr>
<tr><td>SolidWorks Refined mesh:</td><td>SimSolid Stage II Solution<br>(4 solution passes)</td><td>Error</td></tr>
<tr><td>12.21 Mpa</td><td>12.86 Mpa</td><td>5.3%</td></tr>
<tr><td colspan="3">Note: SolidWorks adaptive refinement gets an accurate result for this model.</td></tr>
</table>

Model: newtj

Description: Tube joint with thin solids



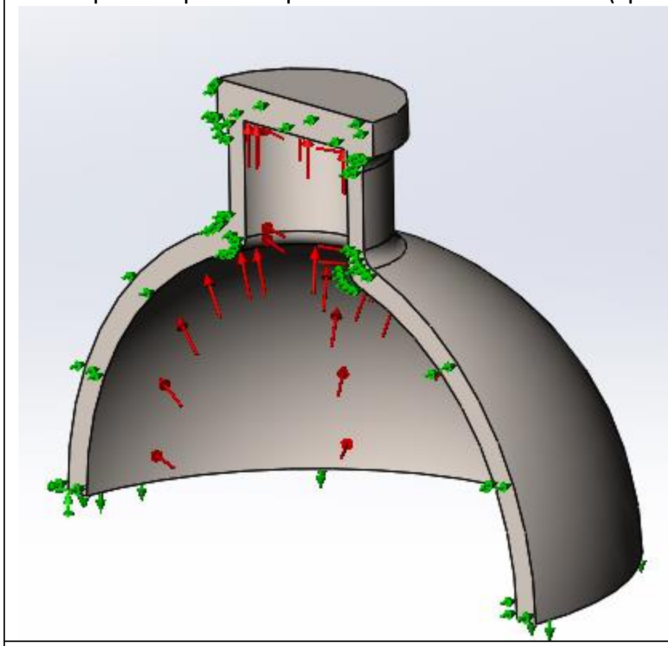
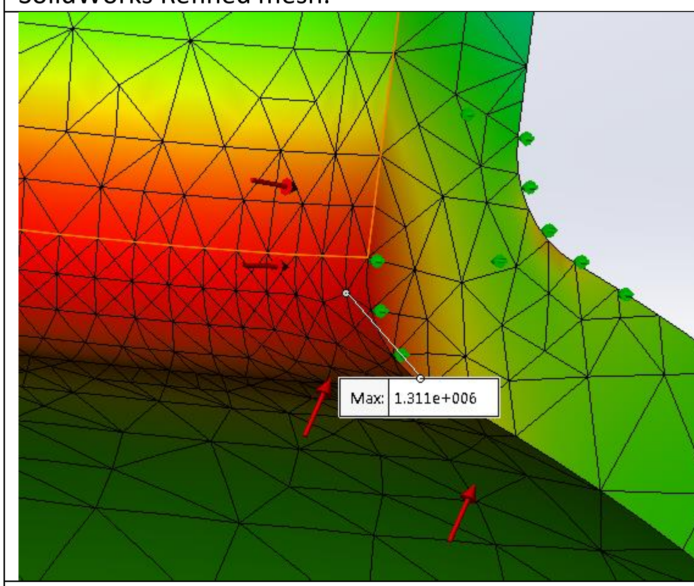
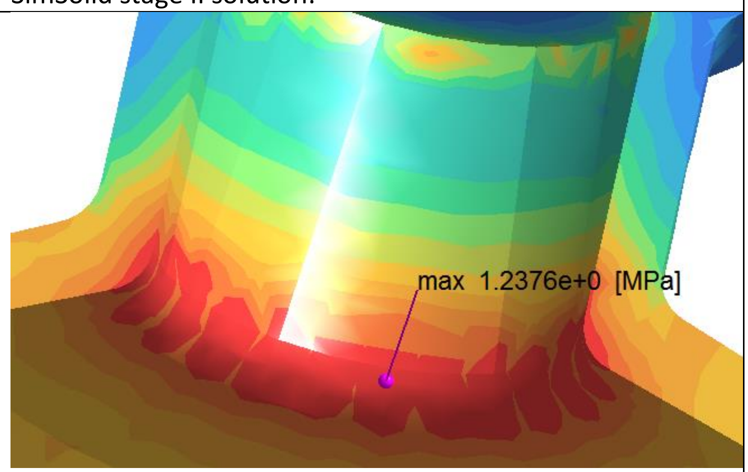
| SolidWorks Refined mesh: | SimSolid stage II solution: |
|---|---|
|  |  |

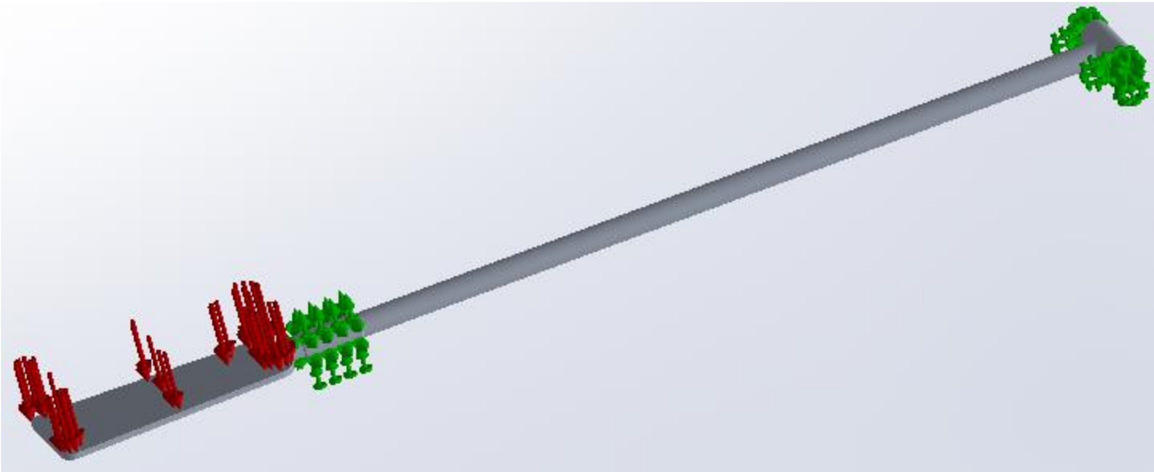
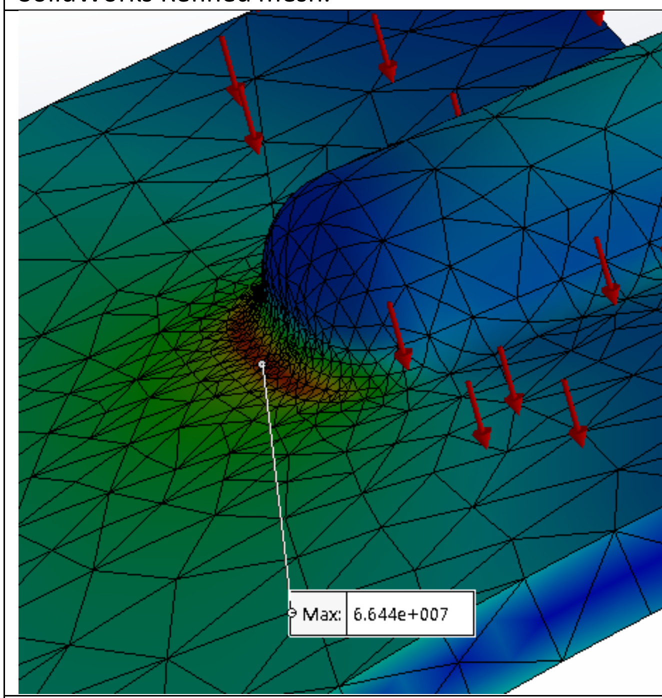
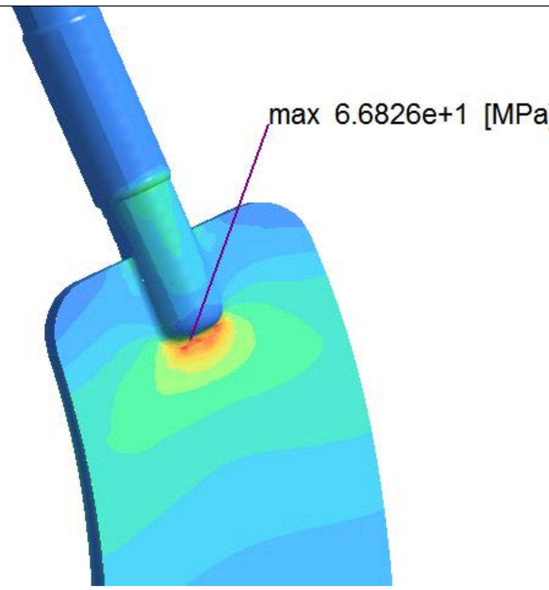
**Maximum Stress Result**

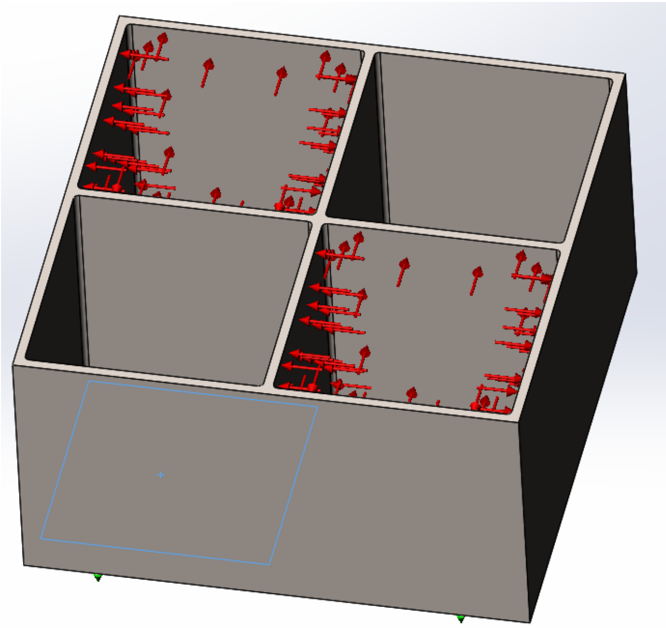
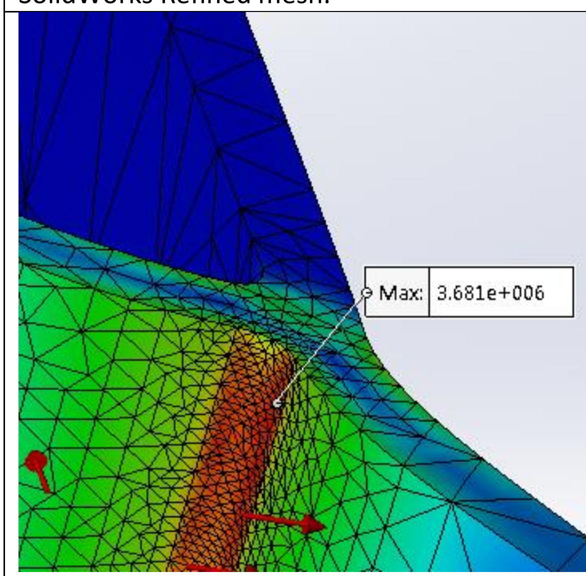
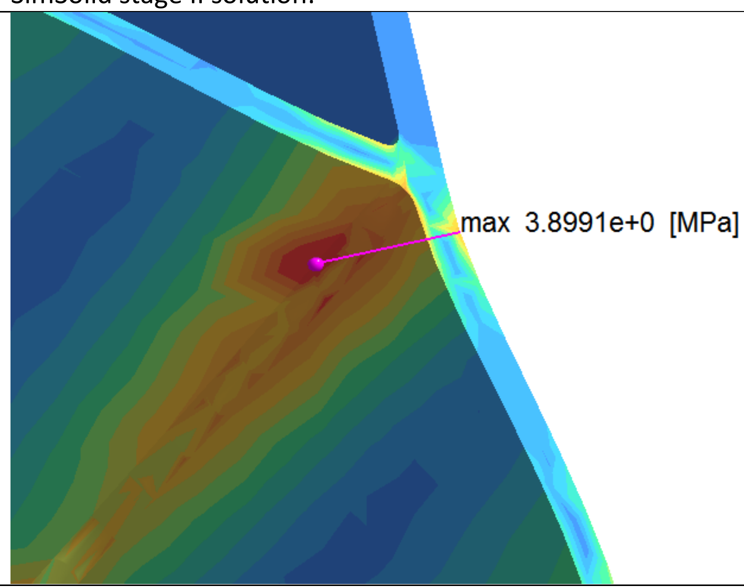
| SolidWorks Refined mesh: | SimSolid Stage II Solution<br>(6 solution passes) | Error |
|---|---|---|
| 167 Mpa | 170 Mpa | 2% |

Notes:

- SolidWorks adaptive refinement gets an accurate result for this model.
- In SimSolid I used “adapt to features” and “adapt to thin solids”

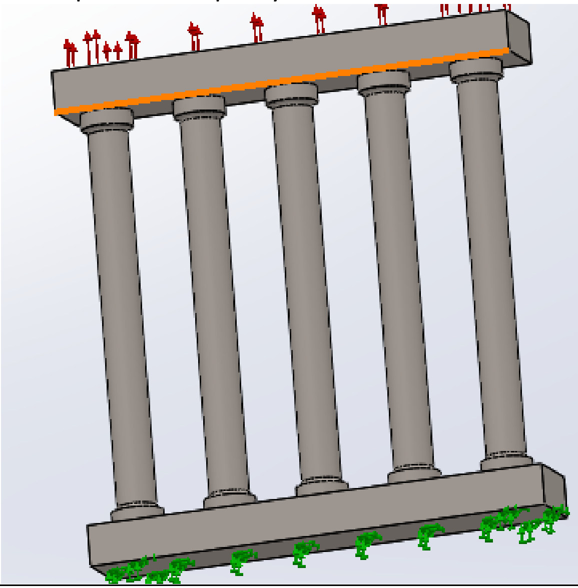
<table>
<tr><td colspan="3">Model: spherehatch</td></tr>
<tr><td colspan="3">Description: Spherical pressure vessel with hatch (spherehatch)<br></td></tr>
<tr><td>SolidWorks Refined mesh:</td><td colspan="2">SimSolid stage II solution:</td></tr>
<tr><td>
</td><td colspan="2">
</td></tr>
<tr><td colspan="3">Maximum Stress Result</td></tr>
<tr><td>SolidWorks Refined mesh:</td><td>SimSolid Stage II Solution<br>(5 Solution passes)</td><td>Error</td></tr>
<tr><td>1.31 Mpa</td><td>1.24 Mpa</td><td>5.5%</td></tr>
<tr><td colspan="3">Note: SolidWorks adaptive refinement gets an accurate result for this model.</td></tr>
</table>

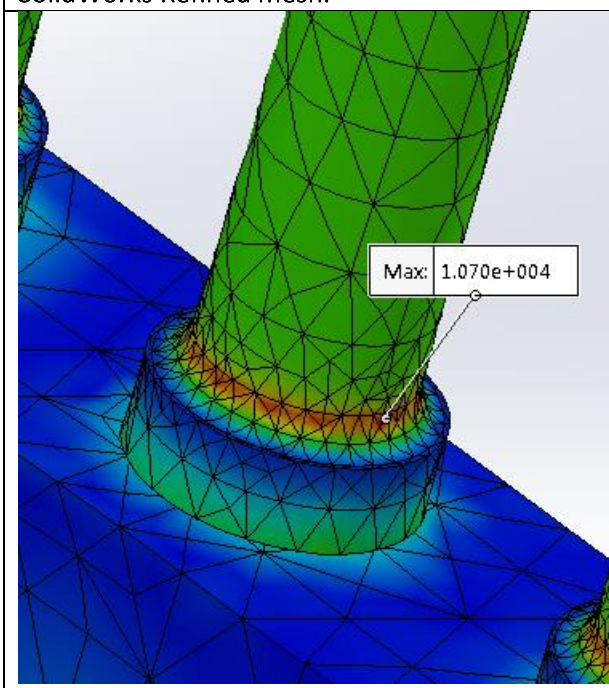
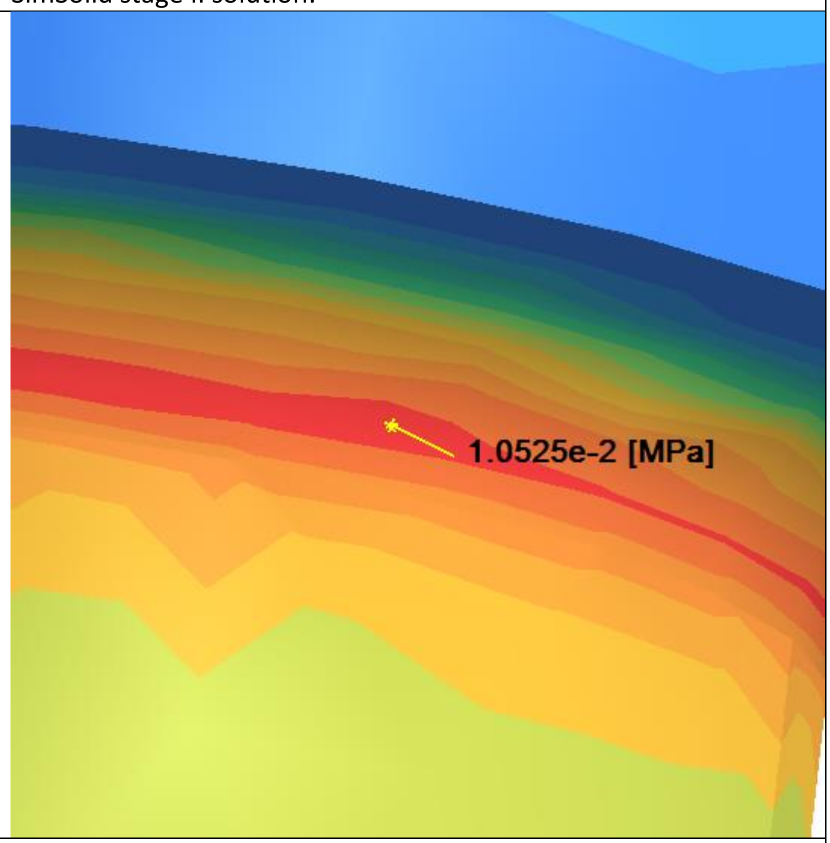
<table>
<tr><td colspan="3">Model: canoepaddle</td></tr>
<tr><td colspan="3">Description: Canoe paddle<br></td></tr>
<tr><td>SolidWorks Refined mesh:</td><td colspan="2">SimSolid stage II solution:</td></tr>
<tr><td>
</td><td colspan="2">
</td></tr>
<tr><td colspan="3">Maximum Stress Result</td></tr>
<tr><td>SolidWorks Refined mesh:</td><td>SimSolid Stage II Solution</td><td>Error</td></tr>
<tr><td>66.4 Mpa</td><td>66.8 Mpa<br>(4 solution passes)</td><td>0.6%</td></tr>
<tr><td colspan="3">Note: SolidWorks adaptive refinement gets an accurate result for this model.</td></tr>
</table>

<table>
<tr><td colspan="3">Model: recttank</td></tr>
<tr><td colspan="3">Description: rectangular tank<br></td></tr>
<tr><td>SolidWorks Refined mesh:</td><td colspan="2">SimSolid stage II solution:</td></tr>
<tr><td>
</td><td colspan="2">
</td></tr>
<tr><td colspan="3">Maximum Stress Result</td></tr>
<tr><td>SolidWorks Refined mesh:</td><td>SimSolid Stage II Solution</td><td>Error</td></tr>
<tr><td>3.68 Mpa</td><td>3.90 Mpa<br>(4 Solution passes)</td><td>5.9%</td></tr>
<tr><td colspan="3">Note: SolidWorks adaptive refinement is off by 12% on this model</td></tr>
</table>

Model: railcylassy

Description: multiple cylindrical columns with fillets



| SolidWorks Refined mesh: | SimSolid stage II solution: |
| --- | --- |
|   |   |

Maximum Stress Result

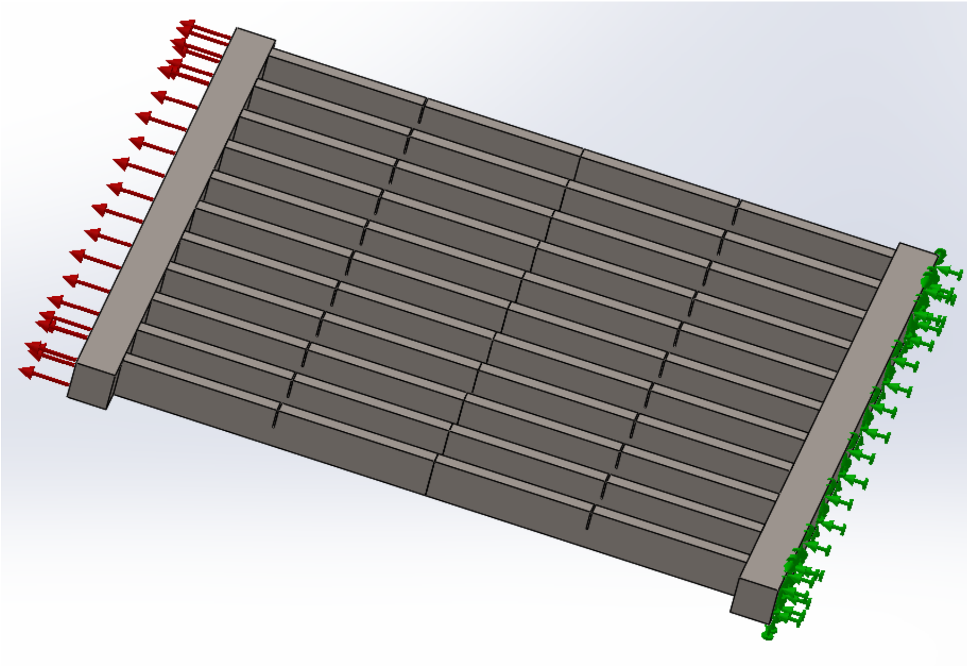
| SolidWorks Refined mesh: | SimSolid Stage II Solution<br>(4 solution passes) | Error |
| --- | --- | --- |
| 1.057E4 Pa | 1.05E4 Pa | 0.7% |

Notes:

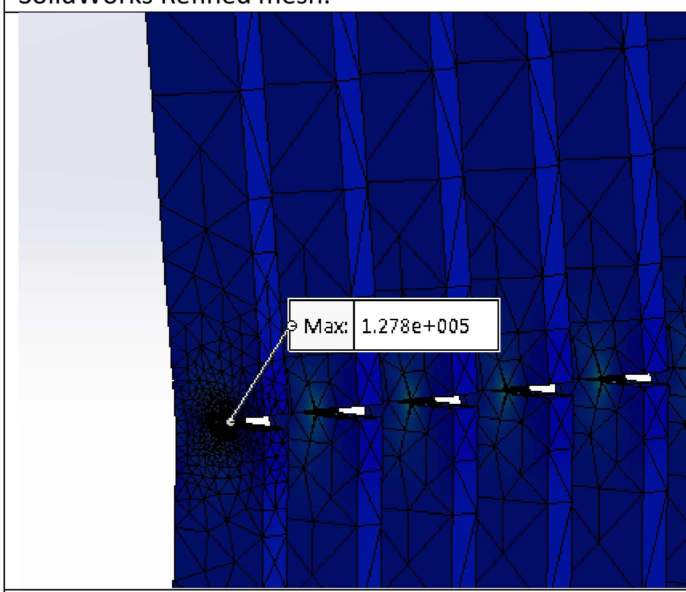
- SolidWorks adaptive refinement gets an accurate result for this model.
- For this Model the Max stress occurs at a singularity where the column has a reentrant corner with the base. Solidworks did not detect this singularity because the mesh was not refined there. The maximum meaningful stress is in a fillet on the column, and was determined in SimSolid using a probe.

Model: edgeslitassy

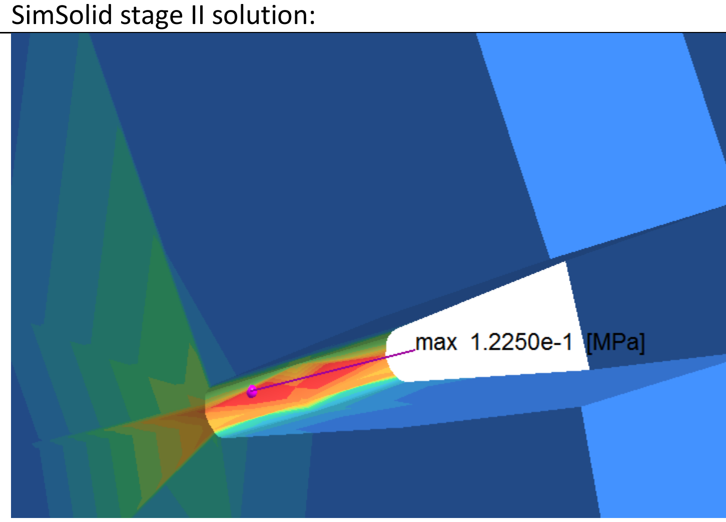
Description: multiple bars with edge slits



SolidWorks Refined mesh:




SimSolid stage II solution:

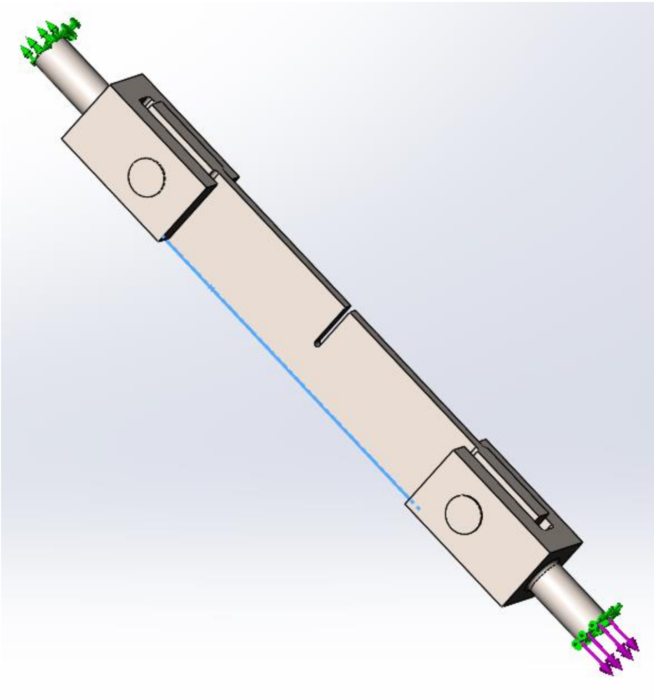


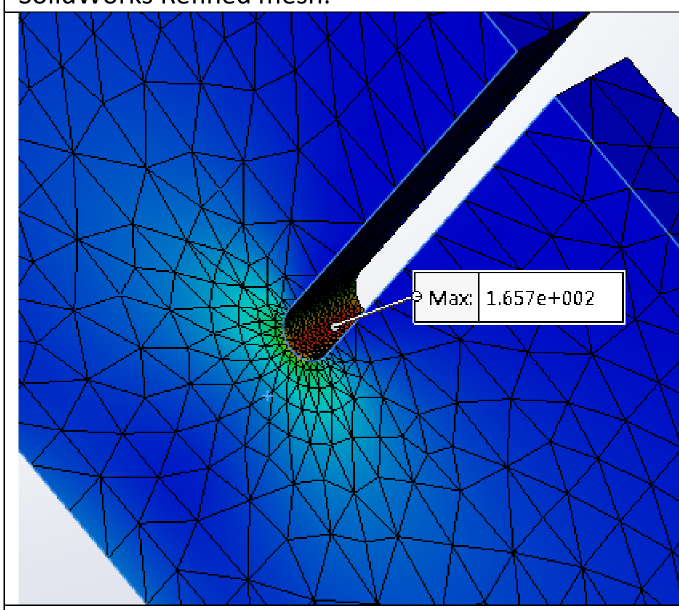
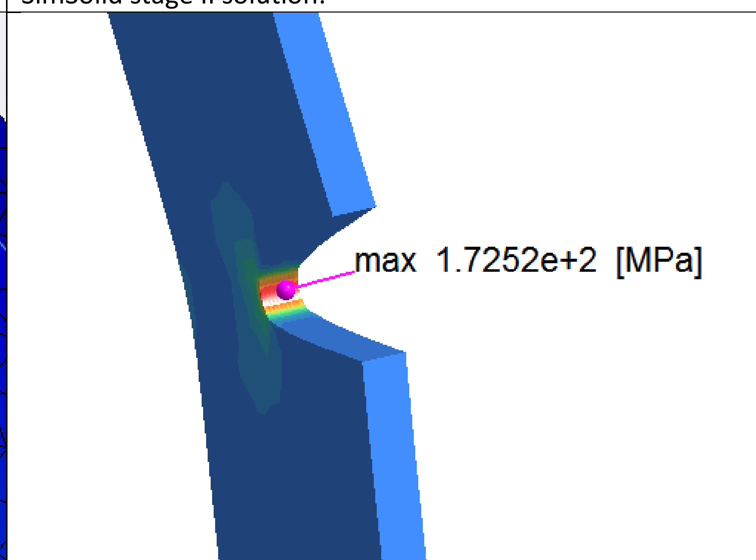

| Maximum Stress Result | | |
|---|---|---|
| SolidWorks Refined mesh: | SimSolid Stage II Solution (4 solution passes) | Error |
| 0.128 Mpa | 0.123 Mpa | 3.9% |

Note: SolidWorks adaptive refinement is off by 13% on this model

Model: clevisassy

Description: testing machine and specimen; tensile load 100000 N



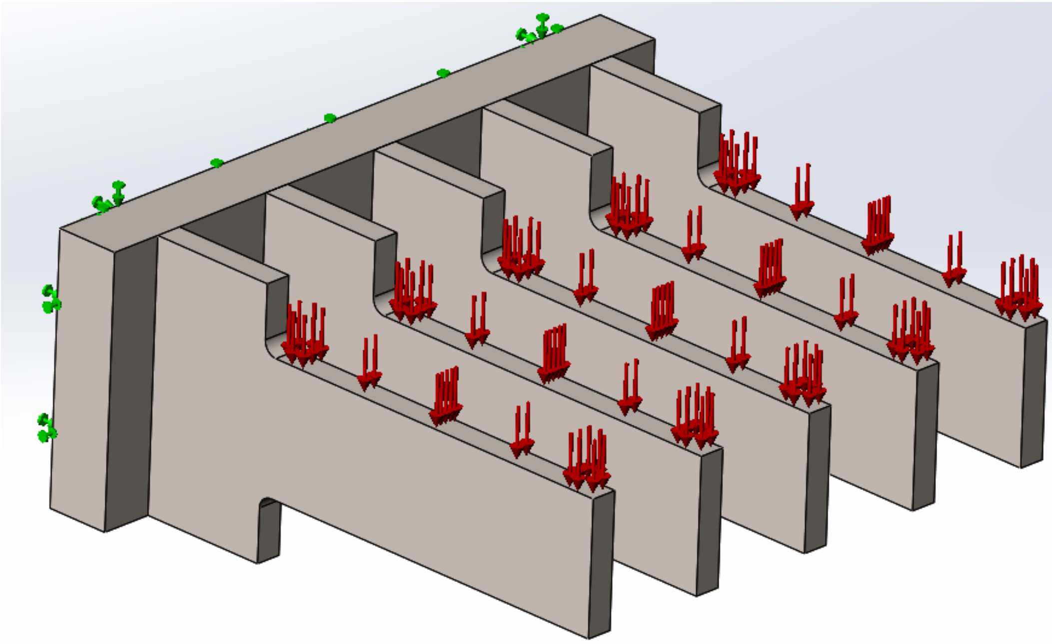
| SolidWorks Refined mesh: | SimSolid stage II solution: |
|---|---|
|   |   |

| Maximum Stress Result | | |
|---|---|---|
| SolidWorks Refined mesh: | SimSolid Stage II Solution | Error |
| 165.7 Mpa | 172.5 Mpa<br>(4 Solution passes) | 4.1% |

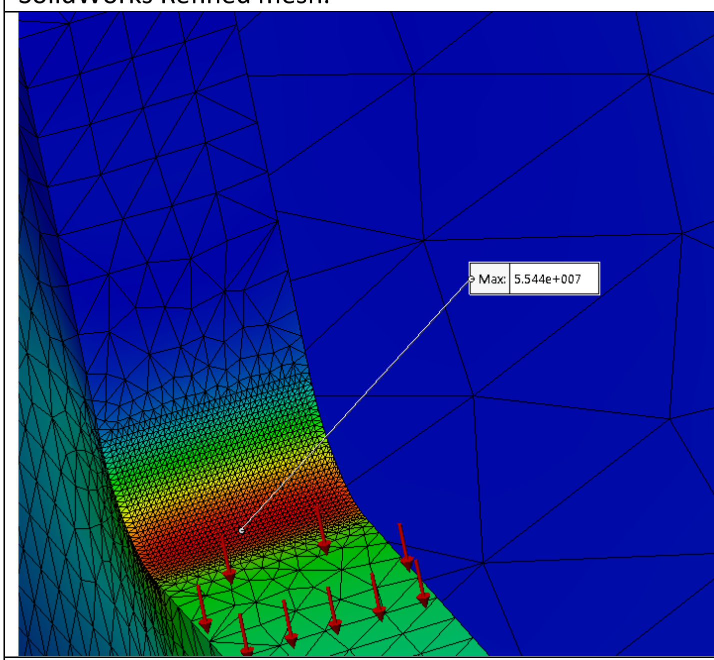
Note: SolidWorks adaptive refinement is off by 22% on this model

Model: filletassy

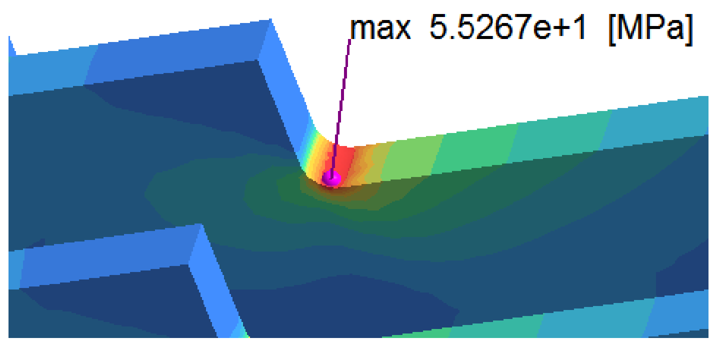
Description: multiple plates with steps and fillets



SolidWorks Refined mesh:




SimSolid stage II solution:



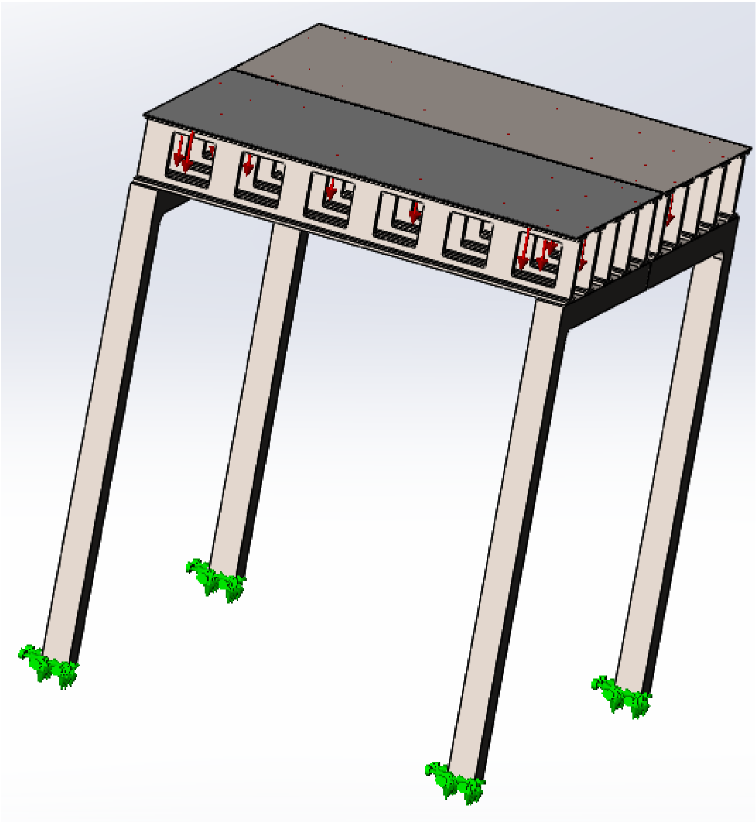

**Maximum Stress Result**

| SolidWorks Refined mesh: | SimSolid Stage II Solution | Error |
|---|---|---|
| 55.4 Mpa | 55.3 Mpa<br>(4 Solution passes) | 0.2% |

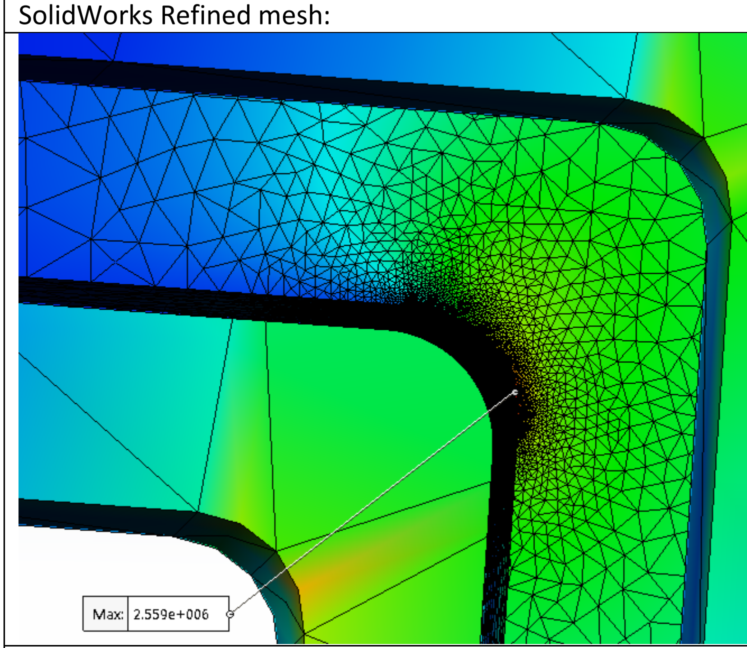
Note: SolidWorks adaptive refinement gets an accurate result for this model.

Model: floorsystem

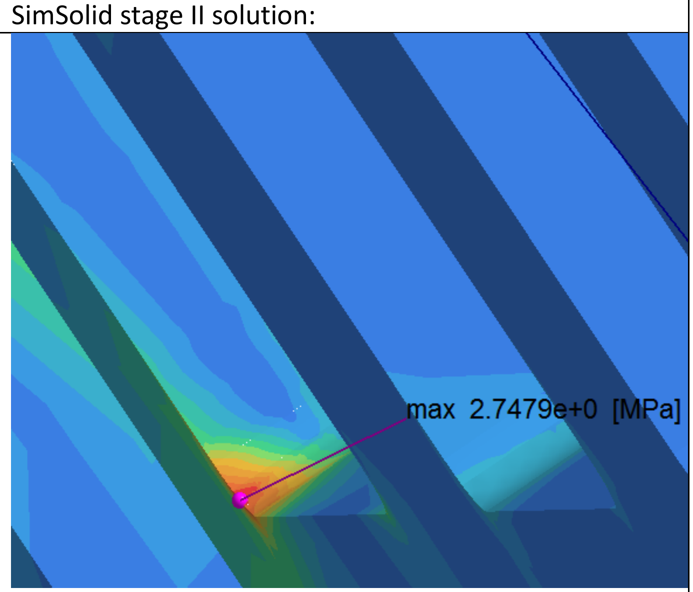
Description: structure with thin plates and girders



SolidWorks Refined mesh:




SimSolid stage II solution:




Maximum Stress Result

| SolidWorks Refined mesh: | SimSolid Stage II Solution (4 solution passes) | Error |
|---|---|---|
| 2.56 Pa | 2.75 Pa | 7.4% |

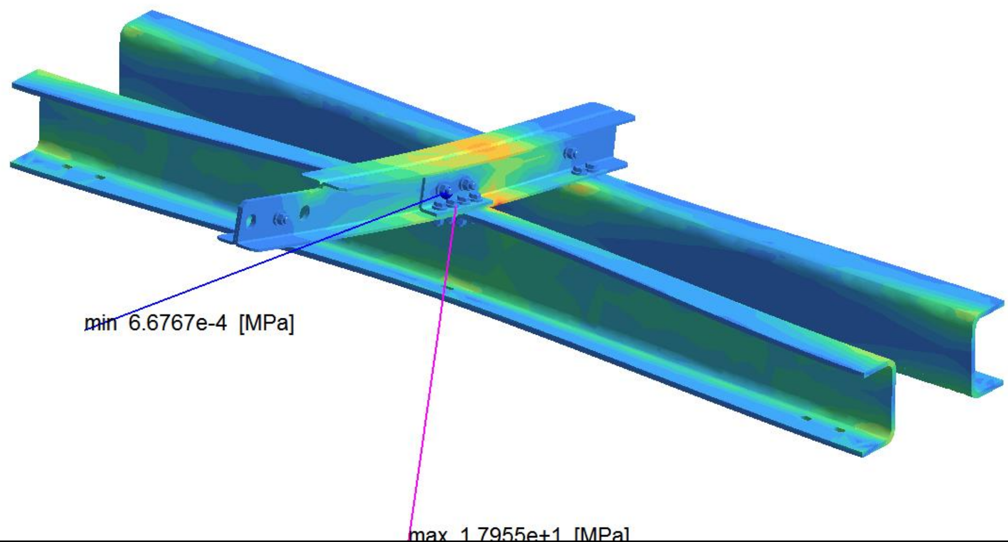
Notes:

- This model has shell elements so adaptive refinement is not available in SolidWorks

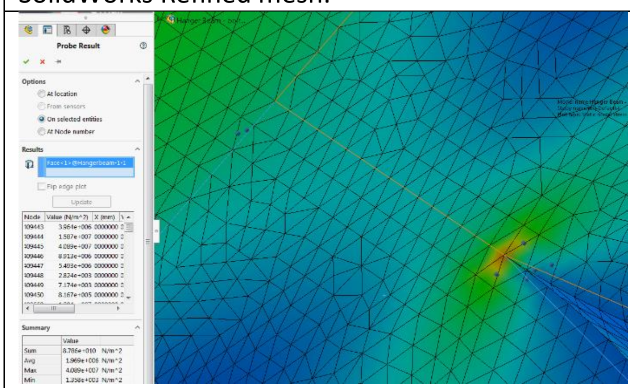
- This model took 54 seconds to solve with 4 solution passes in SimSolid. The SolidWorks solution time for 3 mesh refinement runs was 290 secs total. There was also considerable user intervention required.

Model: hanger-beam

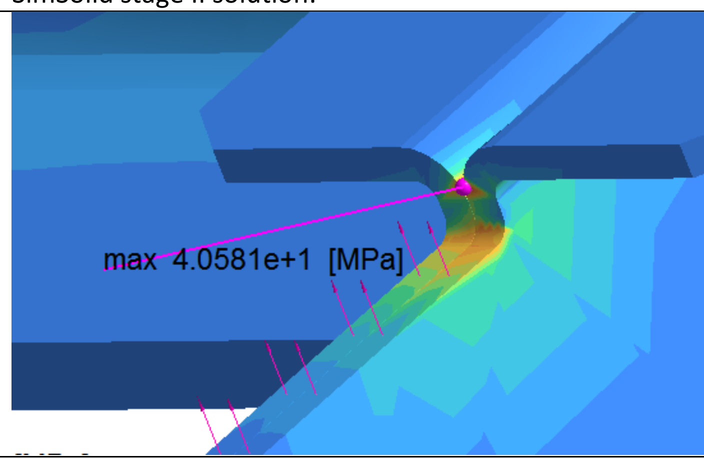
Description: hanger-beam model from SimSolid Tutorial.




SolidWorks Refined mesh:



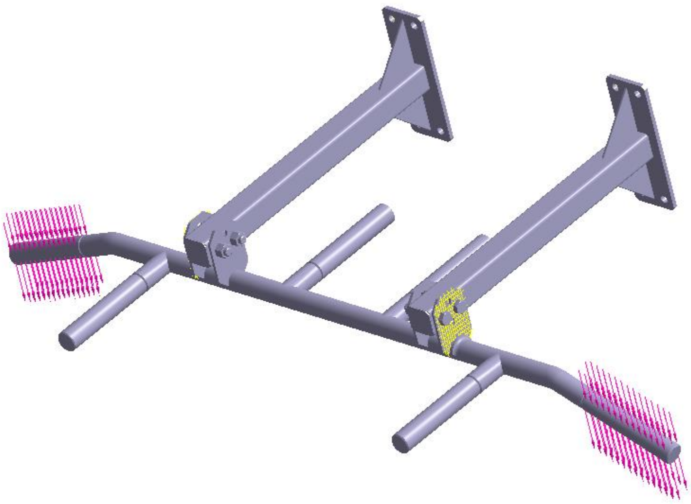
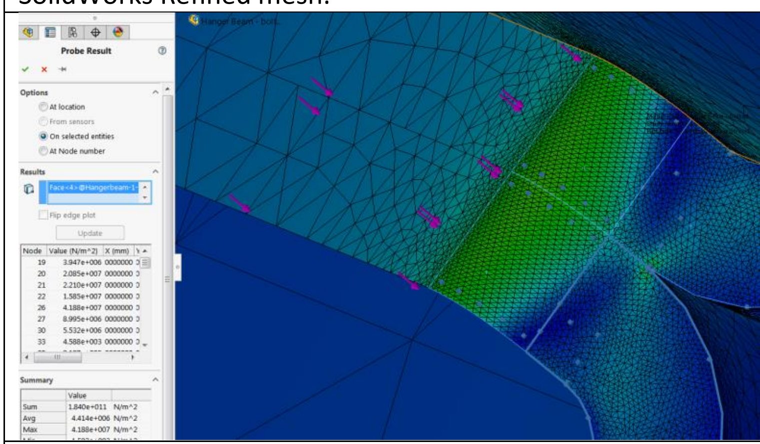
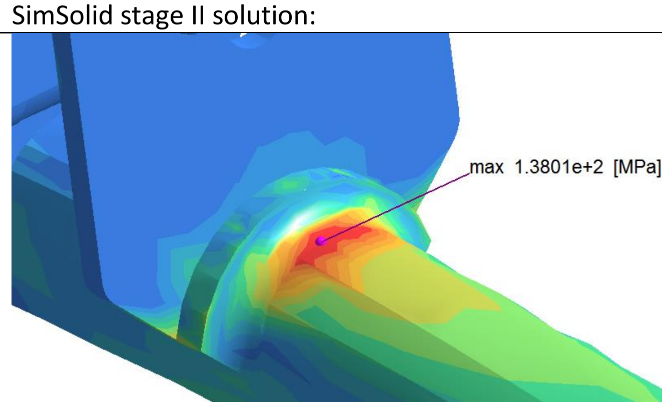

SimSolid stage II solution:




| Maximum Stress Result | | |
|---|---|---|
| SolidWorks Refined mesh: | SimSolid Stage II Solution (4 solution passes) | Error |
| 41.9 Mpa | 40.6 Mpa | 2.6% |

Notes:

- This is design study “baseline” from the tutorial
- SolidWorks picks up a singular stress result for the maximum, elsewhere in the model. For the comparison I probed the SolidWorks in the vicinity of the fillet where SimSolid detected the maximum
- SolidWorks adaptive mesh refinement gets a value of 30.8 for the maximum near the fillet which is 26% low.
- Timing: SimSolid took 14 secs to solve this model with 4 solution passes. The SolidWorks solution time for 3 mesh refinement runs was 140 secs total. There was also considerable user intervention required.

<table>
<tr><td colspan="3">Model: pullup bar</td></tr>
<tr><td colspan="3">Description: pullup bar model from SimSolid Tutorial.<br></td></tr>
<tr><td>SolidWorks Refined mesh:<br>
</td><td colspan="2">SimSolid stage II solution:<br>
</td></tr>
<tr><td colspan="3">Maximum Stress Result</td></tr>
<tr><td>SolidWorks Refined mesh:</td><td>SimSolid Stage II Solution (4 solution passes)</td><td>Error</td></tr>
<tr><td>143.4 Mpa</td><td>138 Mpa</td><td>3.5%</td></tr>
<tr><td colspan="3">Notes:<br>
<ul>
<li>This is design study baseline, analysis “structural 1” from the tutorial. I had trouble getting SolidWorks to process to bolts in the same manner as SimSolid, which affected the results. To work around this I placed an additional immovable support on the face highlighted in yellow above</li>
<li>The maximum stress in the model occurs at a reentrant corner between the pullup bar and a vertical face welded to it. The stress is singular there and theoretically infinite, so cannot be used for accuracy comparisons. I modified the model to fix this by placing a fillet.</li>
<li>SolidWorks automatic mesh refinement failed for this model, with the message “Mesh Adaptation Failed”. The comparison results were obtained by placing a mesh control on the fillet and successively refining it until the stress converged.</li>
<li>Timing: The SimSolid solution, with adapt to features and 4 solution passes, took 18 secs for this model. The SolidWorks solution time for the 3 mesh refinement runs was 83 secs total. There was also considerable user intervention required.</li>
</ul></td></tr>
</table>

# Appendix 1-Obtaining Accurate Stresses With FEA

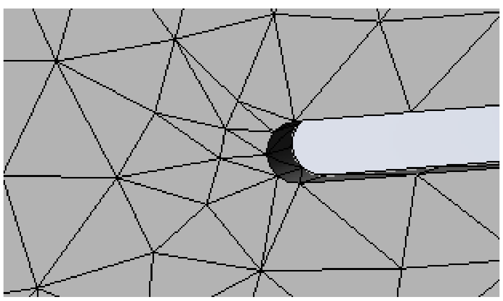
This is a topic to which an entire book can be devoted (in fact I am working on one). I'll describe here in detail the procedure I used in all the test cases in this book. If you have a FEA stress result that does not agree with SimSolid, it is recommended this procedure be followed to make sure the FEA result is accurate.

The default mesh from commercial FEA codes does a pretty good job of following the geometry of the model and will often produce accurate displacement results. Stress results are more localized and may not be accurate unless manual or adaptive mesh refinement is done in local regions of high stress gradient.

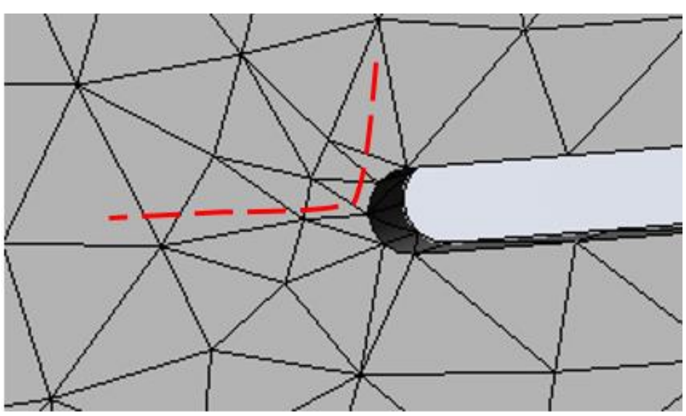
The following settings for an automesher are crucial as a prerequisite for achieving accurate stresses. They are turned on by default in Solidworks Simulate, but not always in other codes (for example they are off by default in SolidEdge Simulate), because they can make the automesher fail. But if they are not on there is little chance of accurate stresses.

- Project midside nodes on surfaces. If this is not on you may be modeling the wrong physical model, such as a plate with a hexagonal hole instead of a circular hole.
- Curvature-based mesh refinement: more elements should be placed along highly curved edges of the model.

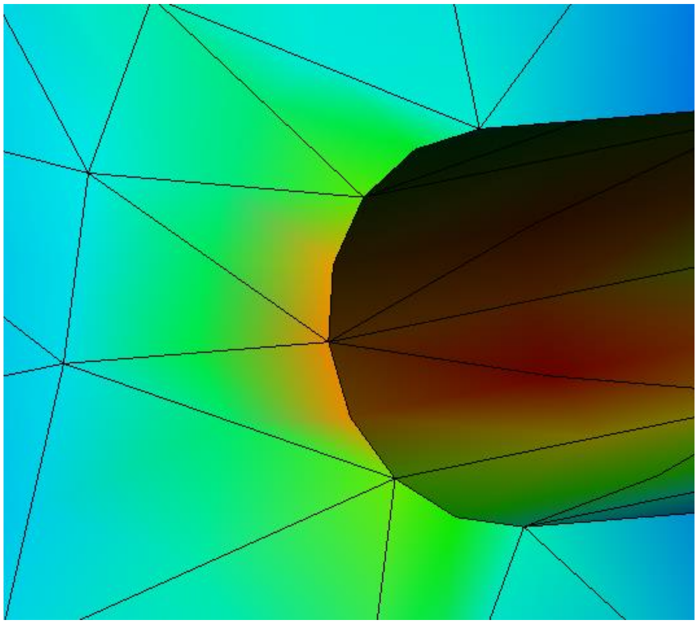
Here is an example of a decent default mesh near a curved cutout (this is actually the 14th validation model, the testing machine assembly with a notched specimen:



This is a reasonable mesh. The geometry is followed well, there are more elements where the curvature is higher, and the elements near the feature (the curved end of the notch) are not large compared the same size of the feature. They are pretty large for a region of high stress gradient, however, which raises a red flag from the standpoint of accurate stresses: elements used in FEA codes represent the displacements quadratically, so stresses, which are related to the derivatives of displacement, are approximately linearly. But the stresses near a curved feature will vary sharply, something like shown in the red line below, with distance from the edge of the notch:

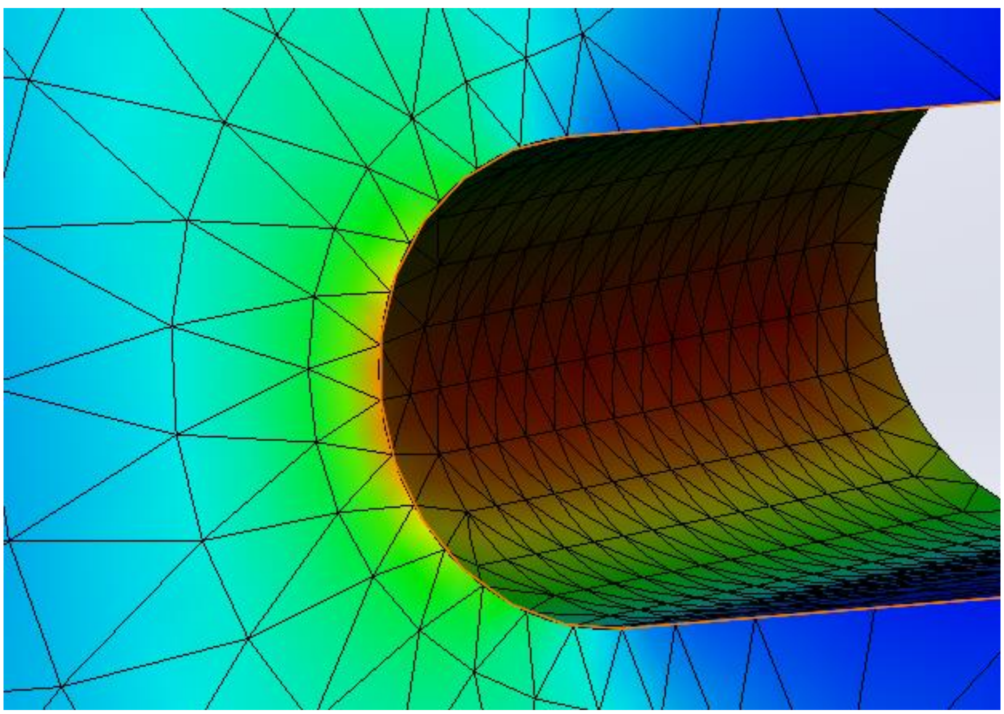


You can see this in the fringes of the FEA result:



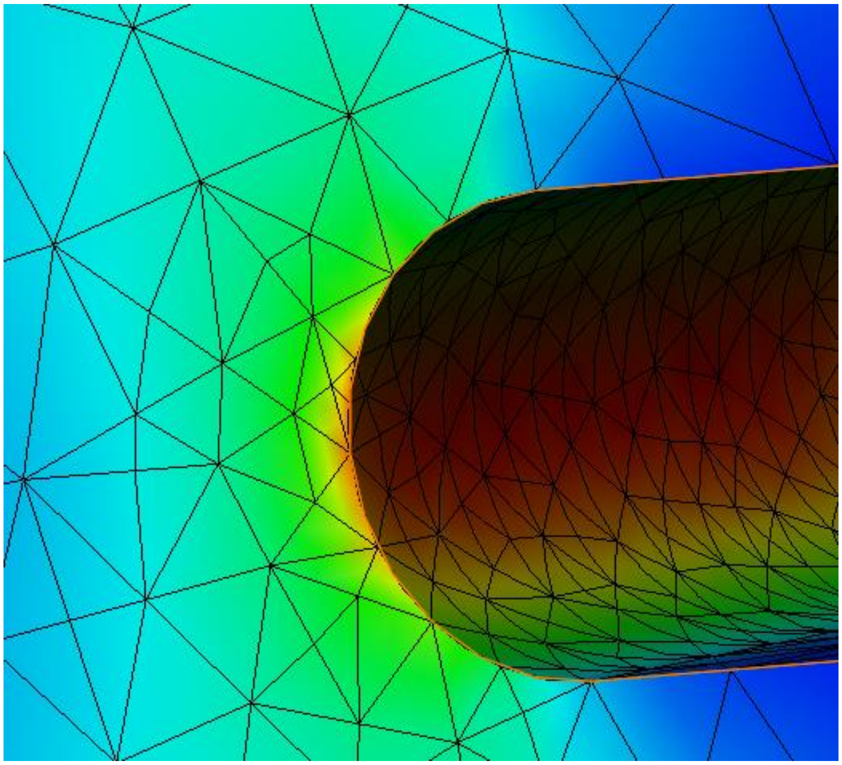
The stress goes from high to low in the distance of a single element. Mesh refinement is clearly needed in the vicinity, either using manual or adaptive mesh refinement.

Here is the mesh obtained using the manual mesh refinement procedure described below:



Now the high stress variation now occurs across several elements.

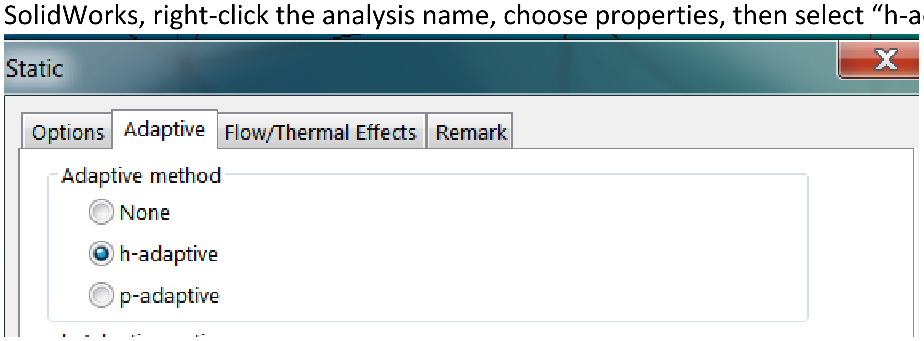
Here is the adaptive automesh for this same model:



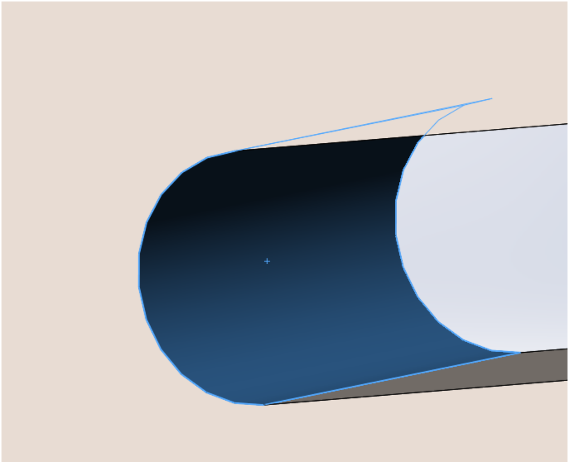
It looks reasonable also. But it turns out this result is only in error by 22%. The default mesh is in error by 41% for this model.

## Suggested Refinement Procedure:

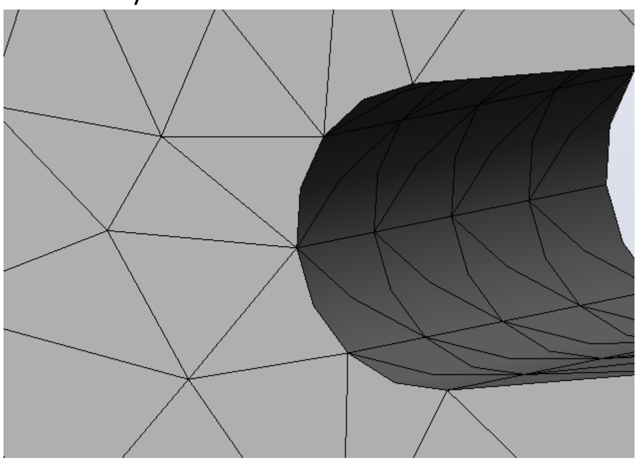
1. Run the model with the default mesh
2. If your FEA code has adaptive mesh refinement, run the model with adaptive refinement. for example, in SolidWorks, right-click the analysis name, choose properties, then select “h-adaptive” from the properties form:



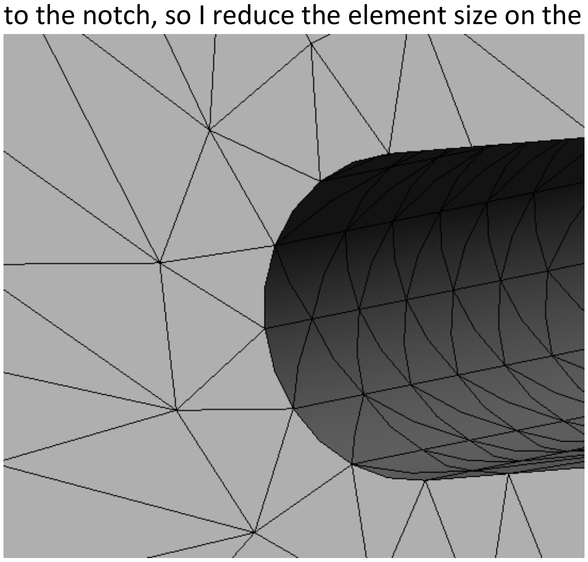

3. If the adaptive stress results agree well with the default, this is probably an accurate result. Of the 19 models in this manual, that was true for only one, the plate with a hole. Otherwise further manual refinement is needed. In the testing machine assembly, for example, the default solution and adaptive solution disagree by 12%, so I’d recommend checking further with manual refinement.
4. Examine the mesh near the vicinity of maximum stress. Place a mesh control there, for example, the notch surface:



Set the mesh density to high or "fine" in the control and remesh. Make sure the new mesh looks reasonable in that vicinity:



If not, you can manually tighten the mesh density more. Once the mesh looks reasonable, run the model again. In this case the first mesh from the mesh control still doesn't look good with fairly large elements still adjacent to the notch, so I reduce the element size on the control and try again:



This looks better, so I solved with this mesh.

5. You should repeat this process at least once more, a good rule of thumb is to increase the density of your control by a factor of two (= lower the element size by 50%), and run again. If the results of your last two runs agree within a few percent the result is probably accurate. Otherwise, you need to repeat the refinement and re-run step until the result stops changing significantly.

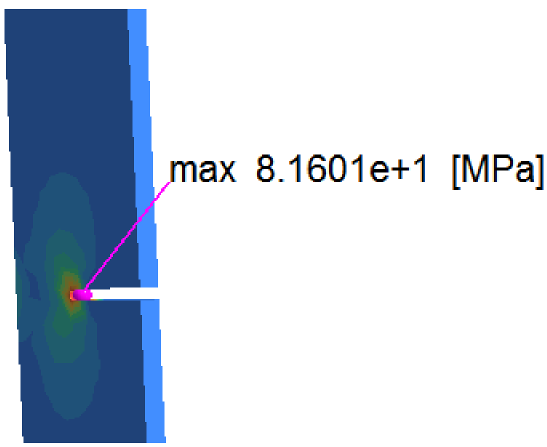
This process can be time consuming for large models but is necessary. For the 19 models run in this study, SolidWorks adaptive refinement either did not work or got an inaccurate answer in 6, or almost a third of the models. So while it is a very good first step, I recommend double-checking it with manual refinement.

The 5 step procedure I described was used for all the FEA results in this manual.

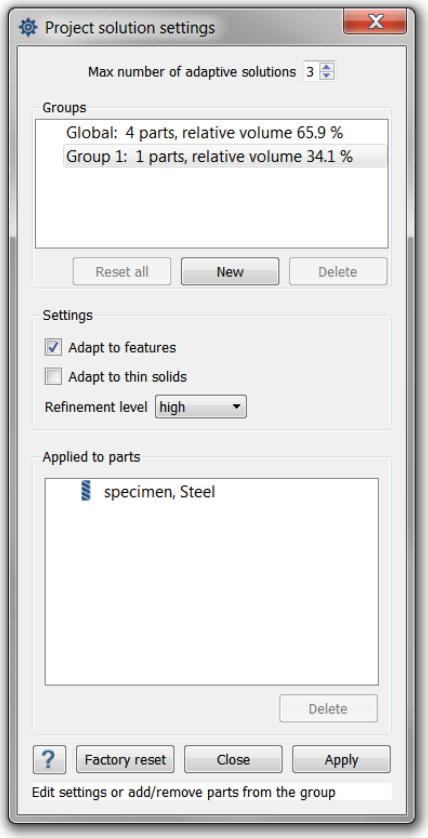
## Appendix 2- Obtaining Accurate Stresses With SimSolid

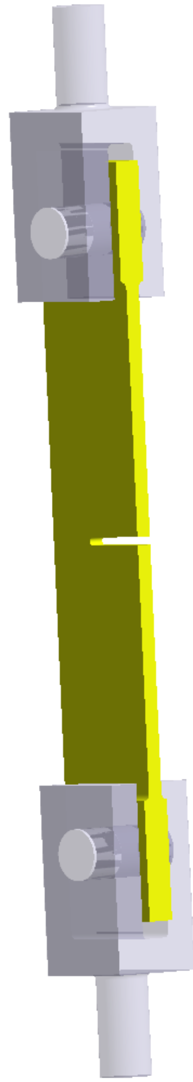
There is a good overview of the two stage procedure for obtaining accurate stress results in SimSolid in the Fast Start Training Guide [2]. With the default settings in SimSolid, the displacements and load paths through the model are usually accurate for static analyses. However, stress is more difficult to calculate accurately because it often is concentrated in regions of high gradients. The training guide suggests an excellent two stage approach for handling this: Analyze the problem with default settings (stage I), then “zoom in” on regions of interest where the might be high stresses, and use the automatic “adapt to features” setting (stage II). You can also at this point request more adaptivity passes in the solution. Then the analysis is run again with these modified settings, to obtain accurate stresses. This approach is much more efficient than simply setting “adapt to features”, and increasing the number of adaptivity passes, for the entire model. That may work fine for relatively small models but be time consuming for large assemblies. The two stage approach lets you quickly compare multiple design alternatives for large models with stage I, then verify the accuracy of stress results in a final stage II run. The user intervention required for stage II is trivial compared to having to manually do mesh refinement in FEA codes.

Now let’s look at this procedure in detail using the same example used in appendix 1, the testing machine assembly. After solving the default analysis (“stage I”) we see the max stress in the model is in the notch of the specimen:

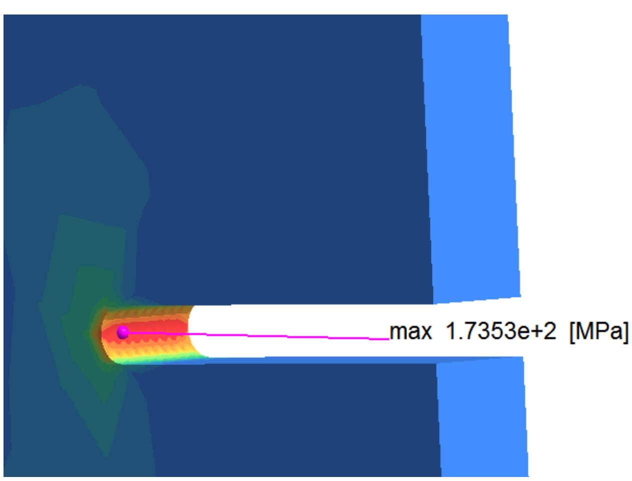



Now the procedure is to turn on adapt to features, but only for the part or parts where the stress is highest:

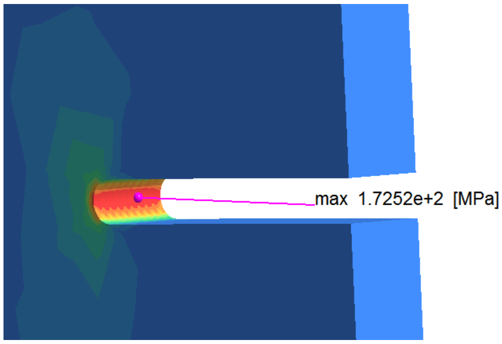


Note there is a pulldown for refinement level: standard, increased, or fine. The higher this level, the more accurate the stresses. I set it to fine for all the models in this manual. Since we're only doing this for a small subset of the model, it does not add to the solution time too much. The other option on this form is "Max number of adaptive solutions" which defaults to 3. I recommend leaving it at 3 and re-running:




If this agreed with the default solution within a few percent, we'd be confident in an accurate solution, but it is off considerably. So now I recommend bumping up the Max number of adaptive solutions to 4 and trying again:



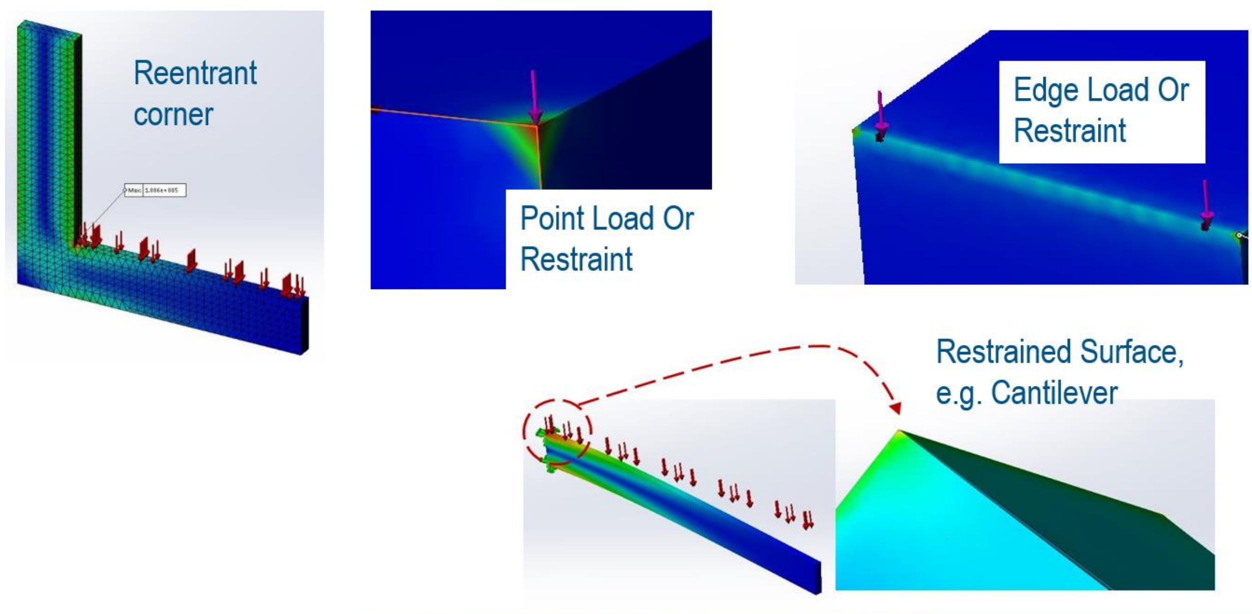

This did not change the result much so we can accept this result. Otherwise it would be necessary to adjust the number of solutions again and repeat the process.

This procedure was done for all the models in this manual. Out of 19 models, one needed only 3 for the number of solutions, thirteen needed 4, two needed 5, and two needed 6.

## Appendix 3- Avoiding Singularities

Results at singularities are not physically meaningful because the stress is theoretically infinite. They cause mesh refinement (or p-adaptivity) in FEA codes, or adaptive analysis in SimSolid, to diverge because the stress just keeps increasing towards infinity.

They can be caused by geometry, such as reentrant corners, or by loads, constraints, and special elements. Here are some examples:




Some singular stress situations

Singularities caused by loads can be avoided in SimSolid by applying the load to a “spot” or by using remote loads. Reentrant corners (or similar situations where there is a kink where two surfaces meet) can be resolved with fillets.

If the maximum stress in your model occurs at a singularity it is best to rethink the true physical situation you are trying to model and resolve it. For example, tube joints do not meet at an infinitely sharp corner, in reality there should be a fillet there. If you are interested in the stress state elsewhere in your model, you may need to use probes to evaluate the stress at that other location.

## About The Author

Richard B King, Ph.D.

I received my Ph.D. in Applied Mechanics from Stanford University in 1980. My training included theoretical stress analysis, numerical methods including the boundary element and the finite element methods, and variational methods such as the Ritz and Galerkin methods (described in the SimSolid Technology Overview [1[, these are the predecessors of both the finite element method and the external method used in SimSolid). I have decades of experience in stress analysis.

In the early part of my career I used all of these methods in investigating the mechanical reliability of pipelines and naval vessels at the National Bureau of Standards, and of disk drives at IBM research. At IBM it became clear that the state of the art of finite element analysis was found to be clearly inadequate for design purposes, which led to the idea of using the adaptive p-method of finite element analysis for this purpose. This led to my co-invented Mechanica and cofounding Rasna Corporation.

Rasna was acquired by PTC in 1995, and I continued to work on enhancing the adaptive analysis capabilities in Mechanica (later renamed Creo Simulate). This included advanced adaptive nonlinear features including snapthrough, nonlinear thermal, large strain plasticity, large displacement and frictional contact.

After leaving PTC I did freelance consulting and teaching, and developed stressRefine, an accessory for p-adaptive solution of existing FEA meshes, which is now available open-source. I also investigated the capabilities of commercial FEA adaptive mesh refinement solutions include SolidWorks Simulation, Ansys, and Autodesk Fusion 360 Simulation. More recently I have looked into SimSolid.

## References


1. Altair SimSolid Technology Overview, Altair Corporation, https://altairhyperworks.com/ResourceLibrary.aspx?category=Technical%20Papers&altair_products=SimSolid
2. Altair SimSolid Fast Start Training Guide, https://altairhyperworks.com/ResourceLibrary.aspx?category=Technical%20Papers&altair_products=SimSolid
3. Timoshenko, S, and Goodier, J, Theory of Elasticity, McGraw Hill, 1970
4. Pilkey, W. D., Pilkey, D. F., & Peterson, R. E. (2008). Petersons stress concentration factors. Hoboken, NJ: John Wiley.