Achieving Convergence in SolidWorks Simulation

This guide is written for SolidWorks version 2010. If you are using an earlier version of SolidWorks, you may encounter some differences in the interface for SolidWorks Simulation and discrepancies between menu screen-shots and your version.

Simulation Overview

SolidWorks Simulation is a series of products that enable users to analyze their SolidWorks models for several types of failure, including simple static analysis, thermal analysis, frequency, advanced non-linear analysis, and more. All of these analysis tools are variations of **finite element analysis (FEA)**.

A common design question when using analysis tools such as SolidWorks Simulation is: "When do I know that I can trust my results?" This is often referred to as **convergence**. This document is intended to explain the concept of convergence, and enable users to determine whether their results have achieved convergence in SolidWorks Simulation. This ensures that simulation results are reliable and can be justifiably used when making design decisions.

This is not a troubleshooting guide. If you have any technical issues with SolidWorks please contact a support technician through our **eSupport** website at www.hawksupport.com or call us in the U.S. at **877-266-4469** Canadian customers dial **866-587-6803**

Contents

What is Convergence? 2
How do I determine if Convergence is achieved? 4
Manual Inspection 4
Trend Tracker 5
Adaptive Meshing 6
Convergence Criteria 6
h-Adaptive Mesh 7
p-Adaptive Mesh 8
Design Study 10
Summary 13
Appendix A: Example Problem 14
Added Fillet 16

What is Convergence?

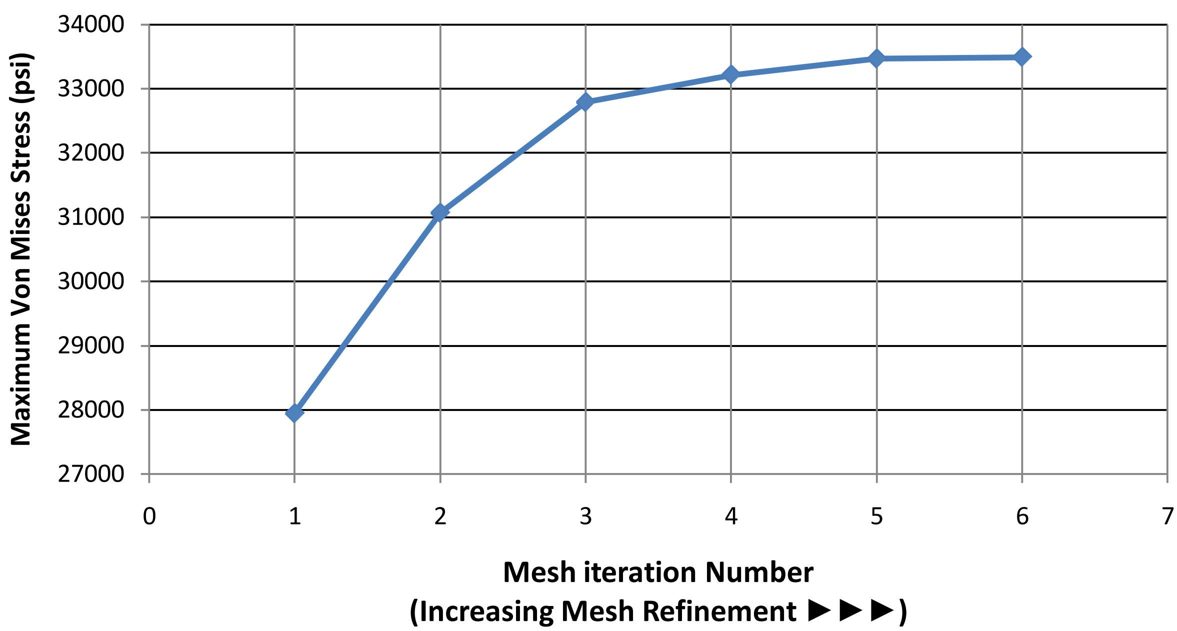
In **finite element analysis (FEA)**, **convergence** is defined as the process of simulation results gradually approaching a finite value as the **mesh** is refined.

All FEA analysis is based on the concept of subdividing a physical model into individual elements, which together make up the mesh. This process is known as **discretization**, and inherently adds a level of error into the results of any simulation.

This **discretization error** is the difference between the calculated results obtained from SolidWorks simulation and the theoretical results which would be obtained from an **exact solution,** such as the Unit dummy force method or Castigliano's method, which require numerical integration. Successful convergence is achieved when, via mesh refinement, discretization error has been reduced to a level such that the change in results is no longer deemed significant by the user. It is important to remember that due to discretization, FEA results will never be identical to an exact solution. In terms of FEA, an exact solution would be analogous to a mesh of infinitely small element size.

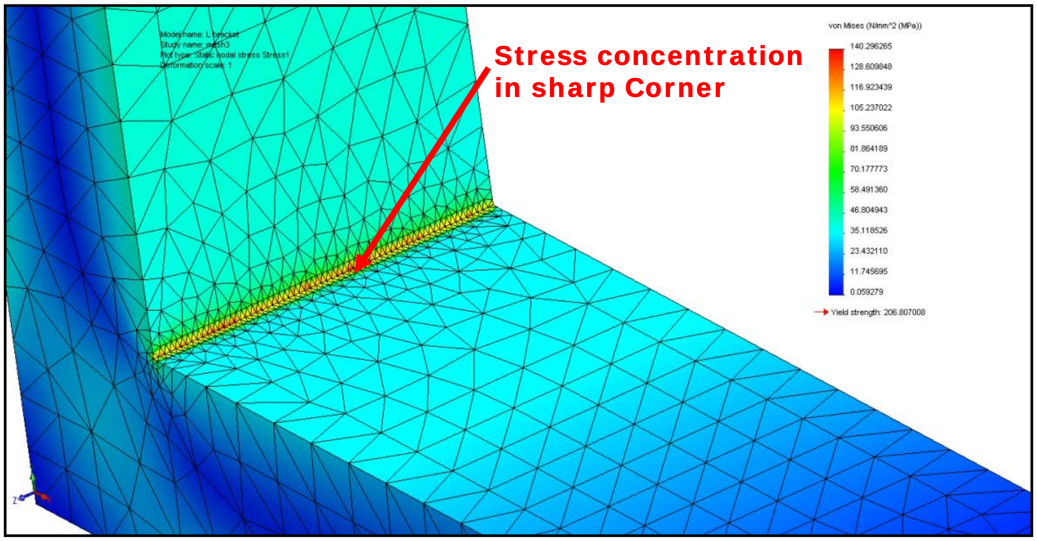
It is also important to note that convergence does not provide any information about the true accuracy or realism of a simulation. Any assumptions, boundary conditions, or material properties must be consistent between an FEA analysis and an exact solution, and thus convergence cannot eliminate error that may be caused by such incorrect assumptions. *Convergence only addresses error due to meshing*, so it extremely important to verify all assumptions of a simulation before attempting to achieve convergence.

Also note that in SolidWorks Simulation, the FFEPLus solver has an option to show its own convergence plot, which displays information about the internal solver progress. This plot is wholly separate from the concept of mesh convergence. *Convergence in this document refers to mesh convergence only.*

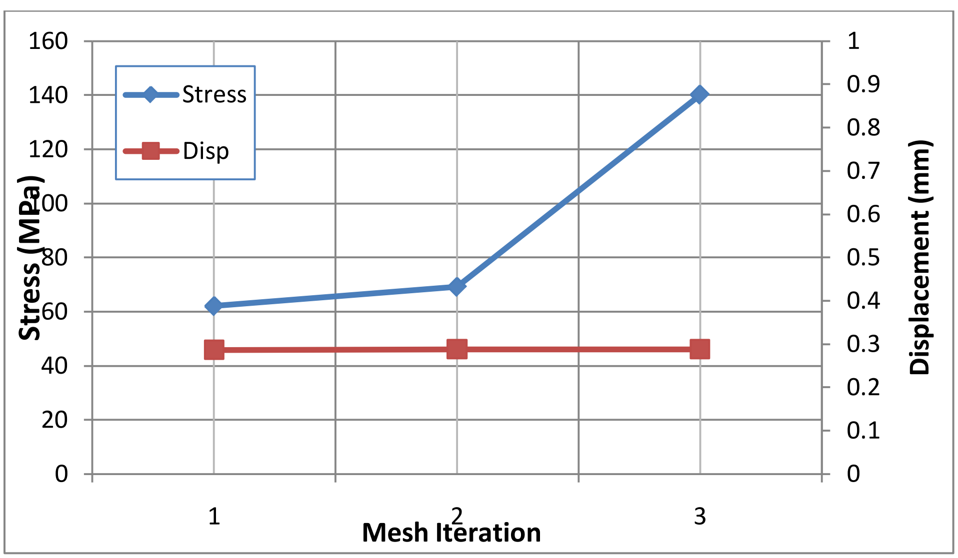


Stress Concentrations

When checking for convergence in Solidworks Simulation, it is important to recognize the impact of **stress concentrations** (or singularities) in finite element analysis. Due to the nature of the stress calculation (which can be generalized as force per area), sharp corners in an FEA model will cause a phenomenon where refining the mesh near the corner causes ever-increasing stress. This is due to the fact that a truly sharp corner would have 0 area and thus infinite stress. This phenomenon is an inherent part of the discretization process and occurs in any FEA program.



The existence of stress concentrations in a study will produce highly **mesh-dependent** results, and can prevent successful convergence. It is therefore necessary to eliminate stress concentrations when trying to achieve convergence of stress results. Note that displacement results are not dependent on area, and so do not produce singularities in FEA. As such they can sometimes be more useful for examining convergence.



How do I determine if Convergence is achieved?

In FEA, there are no universally defined criteria for when results can be said to have converged. The level of acceptable error in any simulation is subjective, and dependent upon the user's preference. However, the *process* of convergence can be recognized as the progression of results slowly approaching a certain value (diminishing error), as opposed to divergence, which shows ever-increasing error.

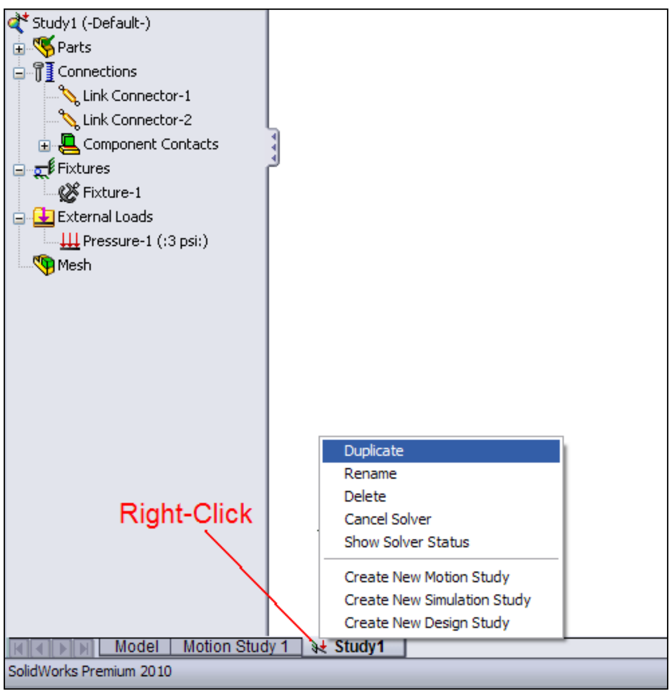
There are three methods in SolidWorks Simulation of examining convergence. They are:

A. Manual Inspection
B. Adaptive Meshing
C. Design Study

Manual Inspection

Checking convergence by **manual inspection** can be performed by manually changing the mesh settings of a study and examining the corresponding change in simulation results. A common practice is to use the **Duplicate** command which will create a copy of an existing study, including all inputs and mesh settings.

- To copy a study, Right-click on the **Study name** and click **Duplicate**.

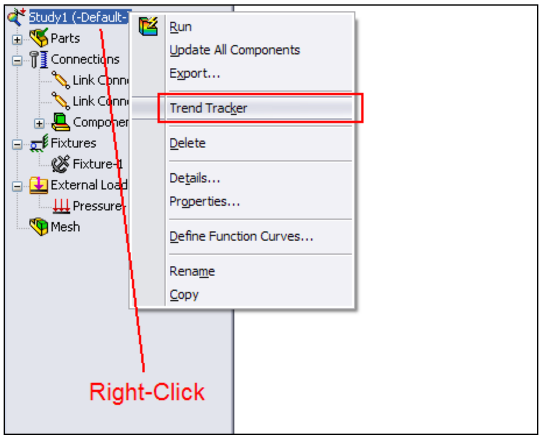


The mesh is then manually modified by such methods as reducing **Global Element Size**, adding **Mesh Controls**, or changing other mesh settings with the goal of increasing detail to better capture the model geometry. In this scenario, the results (such as stress or displacement) can be tracked either by inspection or using sensors, and manually plotted in a spreadsheet program such as Excel. The trends can then be analyzed until the change in results after each iteration seems sufficiently small for the user.

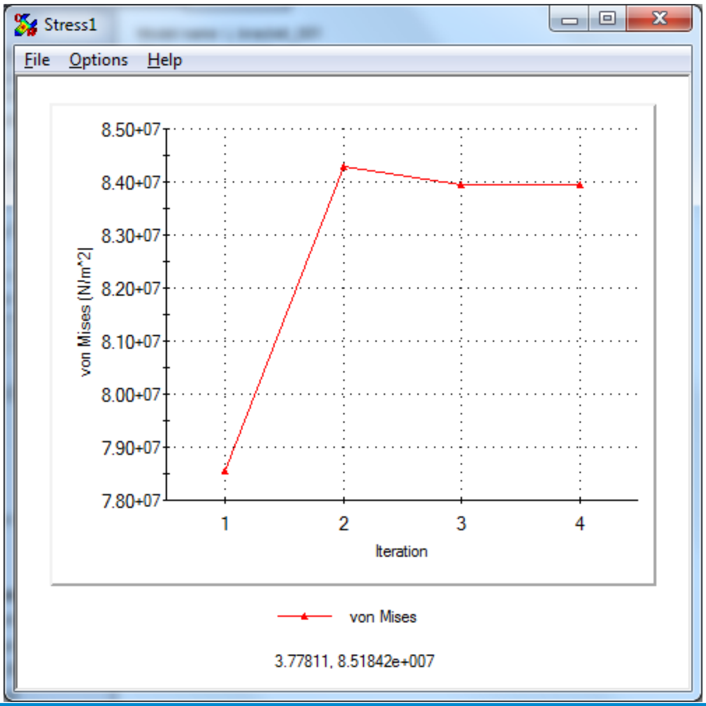
Trend Tracker

Another alternative for manual inspection is to use the **Trend Tracker**. The Trend tracker can record results and automatically produce trend graphs for a series of solver iterations without needing to duplicate the study each time. *Note: Trend Tracker is available in SolidWorks Simulation Professional.*

- To enable **Trend Tracker**, right-click on the **Study name** and select **Trend Tracker**. After the initial study run, right-click on the **Trend Tracker** and click **"Set Baseline."**



Any result plots that have been created will have graphs automatically generated in the Trend Tracker, showing the history of results through all iterations. After each run of the solver, the mesh can be refined and run again, until the desired level of accuracy is reached. The main limitation of this method is that while previous iterations of a study can be restored, this will cause the loss of any subsequent iteration.



Adaptive Meshing

For more robust convergence checking, SolidWorks has **adaptive meshing** tools which can prove useful in obtaining a more mesh-independent solution. These adaptive mesh tools can automatically iterate a study to refine the mesh until the desired convergence criteria are met. The two varieties are **h-adaptive** and **p-adaptive** mesh. Adaptive meshing is only available for linear static studies with solid (tetrahedral) elements.

- To access the adaptive mesh options, Right-Click on the study name and go to **Properties**, then click the **Adaptive** tab

Convergence Criteria

Each type of adaptive mesh has different methods of determining when and where the mesh needs to be refined. The h-adaptive method checks the Strain Energy normalized error throughout the model. Strain Energy norm error is calculated based on stress error, which is defined as:

$$\boldsymbol{e_{\sigma_i} = (\sigma_{exact} - \sigma_i)}$$

where $\boldsymbol{\sigma_{exact}}$ is the stress from an exact solution and $\boldsymbol{\sigma_i}$ is the nodal stress. Elemental error is calculated by integrating the nodal errors over the element's domain.

Since SolidWorks Simulation is only a finite element analysis package, an exact solution is not available. Instead, $\boldsymbol{\sigma_{exact}}$ is substituted for the average values of the neighboring elements. Since a node at the corner of a tetrahedron is commonly shared by all the adjacent elements, the elemental stress from these elements can be averaged to approximate what the nodal stress "should" be.

A global error estimate is calculated as the square root of the sum of all elemental errors in the model squared:

$$\boldsymbol{e_\sigma} = \sqrt{\sum_{\boldsymbol{i=1}}^{\boldsymbol{n}} \boldsymbol{e_\sigma}^2{}_{\boldsymbol{i}}}$$

SolidWorks then calculates the Strain Energy normalized error percentage for each element as:

$$\boldsymbol{\eta_i} = \sqrt{\frac{\boldsymbol{e_\sigma}^2{}_{\boldsymbol{i}}}{(\boldsymbol{u}^2 + \boldsymbol{e_\sigma}^2)}}\ \boldsymbol{x\ 100}$$

where $\boldsymbol{u}$ is twice the total strain energy in the model.

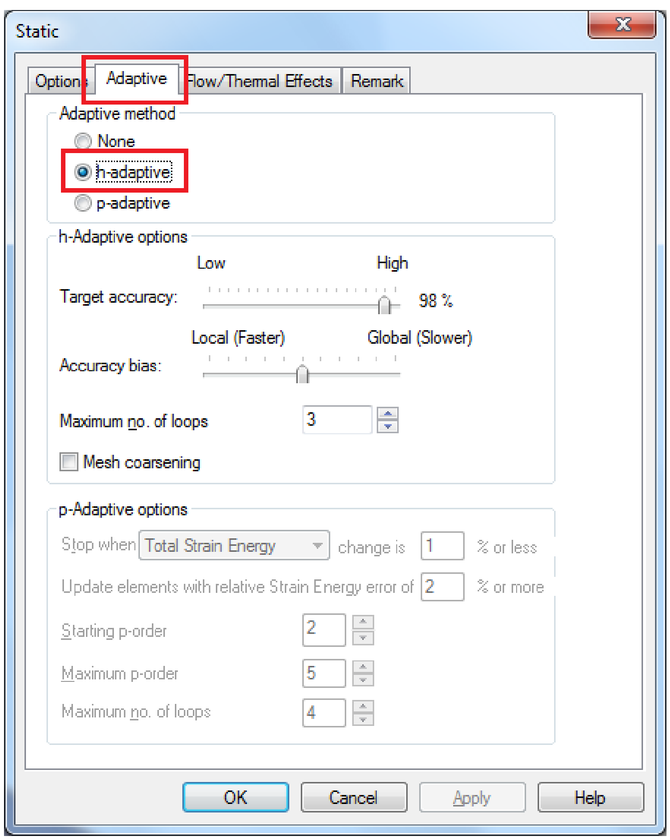
When using the p-adaptive method, multiple parameters are available for specifying the convergence criteria, unlike the h-adaptive mesh which is limited to Strain Energy normalized error. They are:

1. **Total Strain Energy,** the sum of strain energy for all elements in the mesh
2. **RMS Resultant Displacement,** the root mean square of resultant displacement for all elements in the mesh
3. **RMS Von Mises stress,** the root mean square of Von Mises Stresses for all elements in the mesh

h-Adaptive Mesh

The h-adaptive mesh method will automatically adjust individual element size throughout the model to refine the mesh in critical areas. The variable *h* is the element size, analogous to the diameter of a sphere around the tetrahedron. The h-adaptive mesh also features a **"Mesh Coarsening"** option which can increase element size above the global size in areas of low error. This can help offset the performance hit from the additional elements which are added by the adaptive meshing process.

- To access the h-adaptive options, select the **Adaptive** tab and then select **h-adaptive**.



The **Target Accuracy** option under the **h-adaptive** mesh options controls when the adaptive mesh process should stop. The default value is 98% accuracy, which means that the mesh will no longer be refined once the RMS Strain Energy norm error over the whole model is less than 2%. Note that this is not the same as a 2% error in maximum **Von Mises stress**, however achieving this accuracy indicates reliable stress results. Strain Energy norm error is used because it calculates the total Strain Energy error throughout the model and always leads to **monotonic convergence**. In other words, this avoids mistakenly stopping refinement due to convergence of stress results in only a small area of the model.

Users can also adjust the **Accuracy Bias** slider to control what parts of the model are favored for convergence. Setting the slider towards **Local** will place more weight on local Strain Energy norm errors, giving better results in high-stress areas. Moving the slider towards **Global** will emphasize the total Strain Energy norm error, which can help to average out the high stress values arising from any singularities.

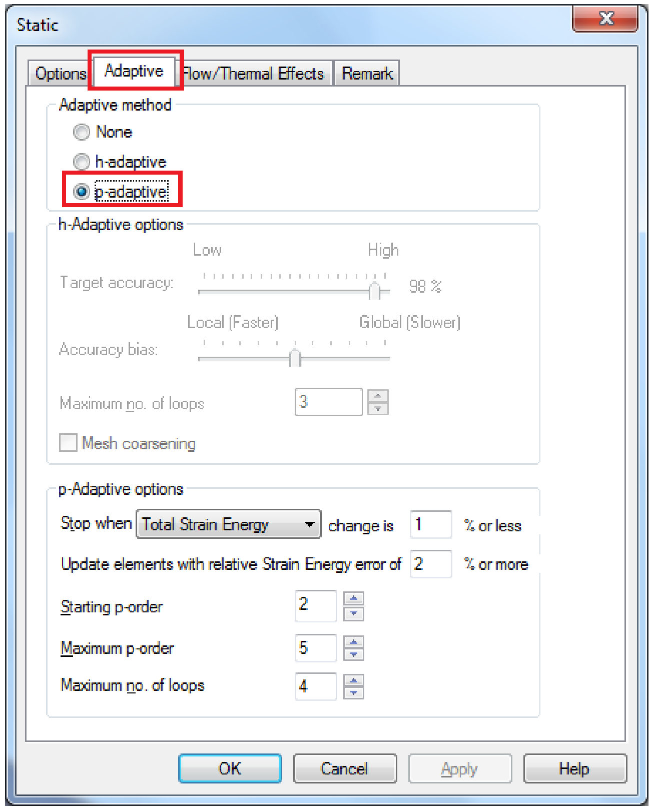
The **Maximum no. of loops** will control how many times the h-adaptive mesh will create a new iteration and refine the mesh. The maximum number of loops which can be run is 5. Please note that the h-adaptive mesh refinement can be run repeatedly, so the overall number of refinement loops from the baseline mesh is not limited.

Finally, as mentioned above, the **Mesh Coarsening** option will automatically increase element size above the global size in areas which experience small or no change in results. This can help offset the performance hit from the additional elements which are added by the adaptive meshing process.

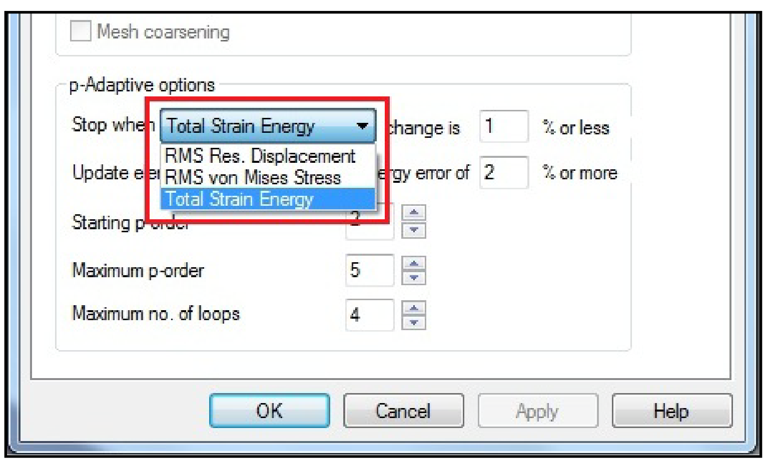
p-Adaptive Mesh

The p-adaptive mesh uses a method where mesh element sizes remain unchanged, but the **polynomial order** of mesh elements is increased, allowing them to match detailed or highly curved geometry with more detail. The variable *p* refers to the order of the equation describing each side of the element. By default, mesh elements in SolidWorks simulation are 2nd-order, having one additional node in the middle of each edge of a tetrahedral element, while **Draft-quality** mesh elements are first order. The p-adaptive mesh can increase this up to a maximum of 5th-order elements.

- To access the p-adaptive options, select the **Adaptive** tab and then select **p-adaptive**.



For each of these global parameters, a **percent change** value can be specified which will cause the p-adaptive solution to stop once the change in between two iterations is below this value.



The level of refinement in each iteration can also be controlled. In the second percentage box ("Update elements with Strain Energy norm error of"), the user can control how broad the refinement is by setting which elements will have their order increased, based on Strain Energy normalized error. As this value is decreased, a greater percentage of elements will be converted to the next higher order, which will increase accuracy at the expense of additional degrees of freedom (and additional solve time).

Since the p-adaptive mesh method increases order of mesh elements, and the maximum order is 5, the p-adaptive solution can only loop up to a maximum of 4 times and will then stop even if convergence has not been achieved. The **Starting p-order** and **Maximum p-order** set which order the mesh will start at and which order the adaptive mesh will stop at. Unlike the h-adaptive mesh which can be repeated to reduce element size without limit, the p-adaptive mesh must stop once all elements have been upgraded to 5^{th}-order, since no further refinement is possible.

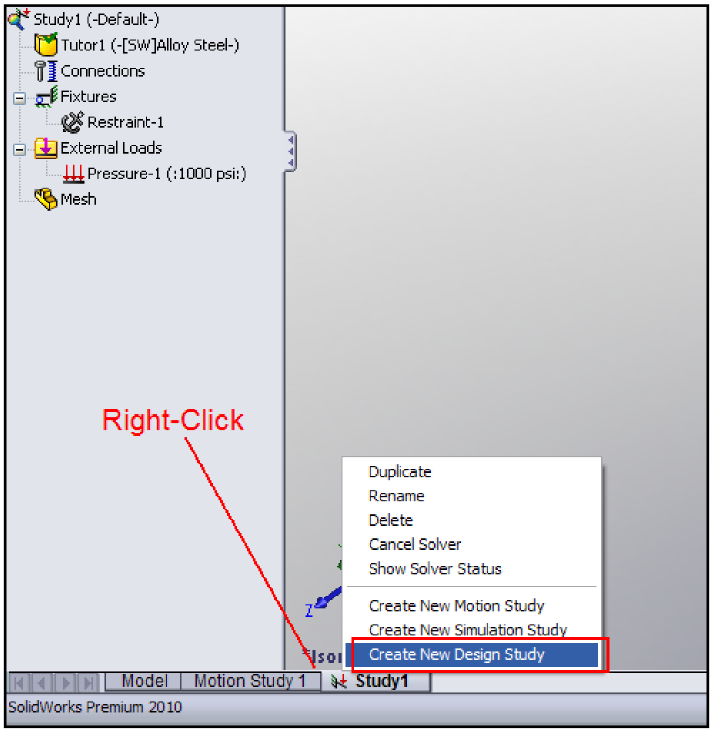
Design Study

A third method for checking convergence in SolidWorks Simulation involves using the **Design Study** functionality which can easily generate multiple simulation scenarios for comparison. One of the parameters which can be changed is the **Global Element Size**, enabling the user to quickly run multiple iterations of a study and examine the results between these global element sizes.

The Design Study method is similar in many ways to manual inspection for convergence in that no *internal* software convergence checking is performed. The results are analyzed manually by the user and thus the user determines each time whether convergence has been achieved.

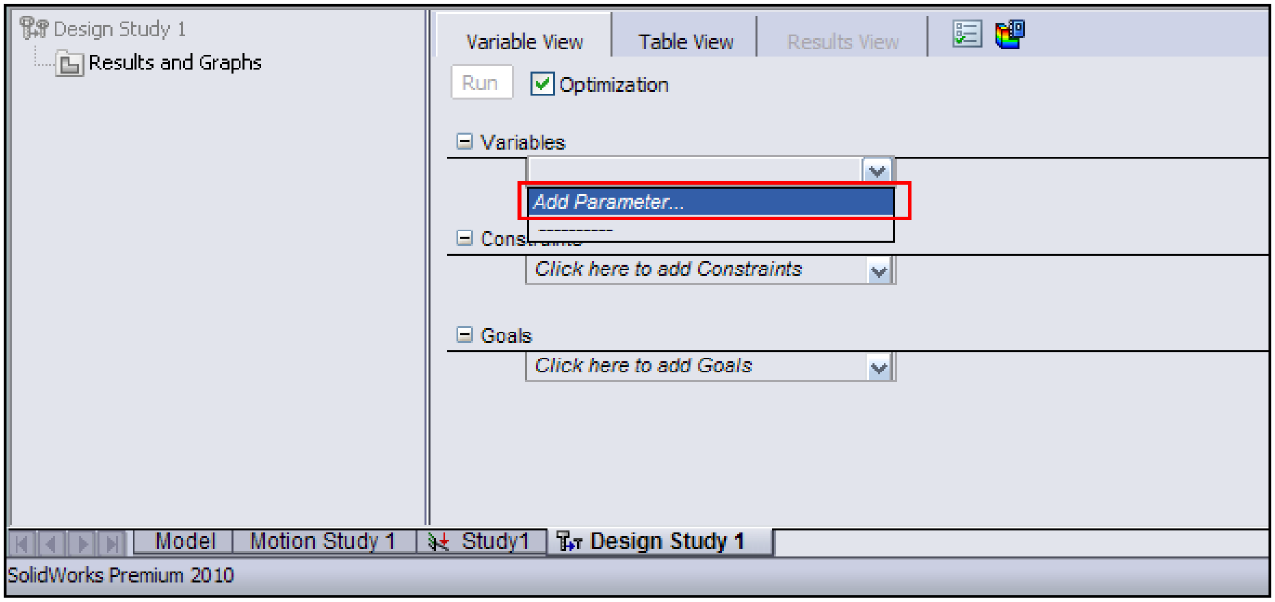
The distinction between the two methods is that whereas the user must manually refine the mesh and iterate the analysis in the first method, the Design Study method allows multiple scenarios to be run simultaneously which can significantly reduce setup and evaluation time.

- To create a new Design Study, right-click on the study name tab and choose **Create New Design Study.**

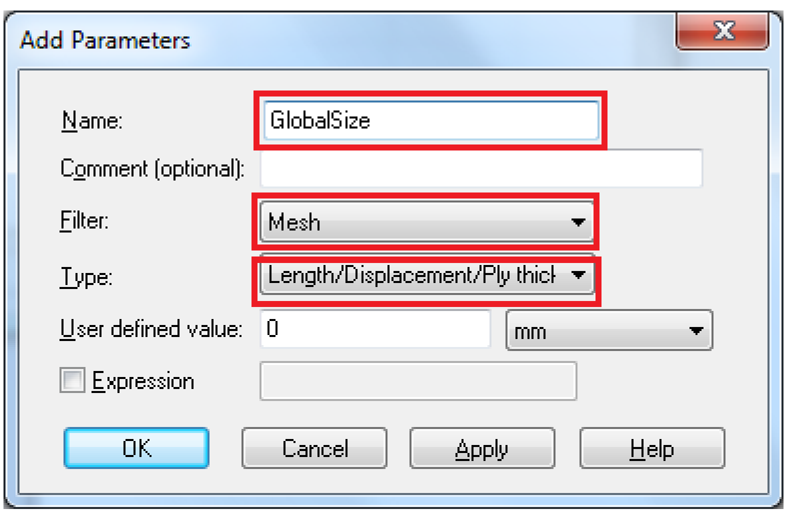


In order to set up the Design Study to run with varying mesh sizes, a **parameter** for the global element size must be added.

- To add global element size as a parameter, click the drop-down menu under **Variables** and select **Add Parameter**.



- Type in a name for the parameter. Under **Filter**, select **Mesh**, and under **Type**, select **Length/Displacement/Ply Thickness**.

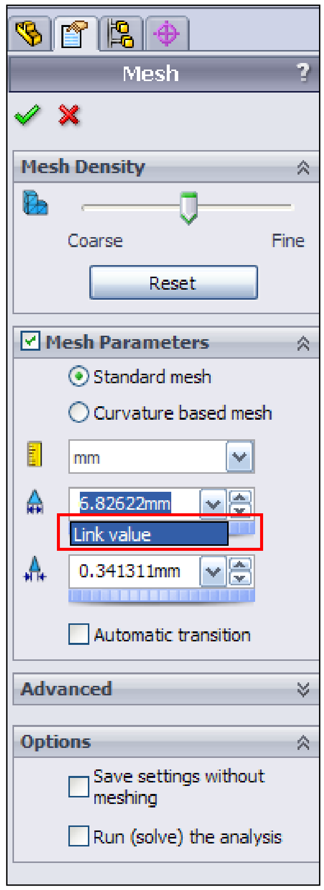


The desired scenarios can then be created by using the **Range**, **Discrete Values**, or **Range with Step** options. Each scenario will use the same setup and conditions of the original static study, except with a different value for the mesh size parameter, as specified here.

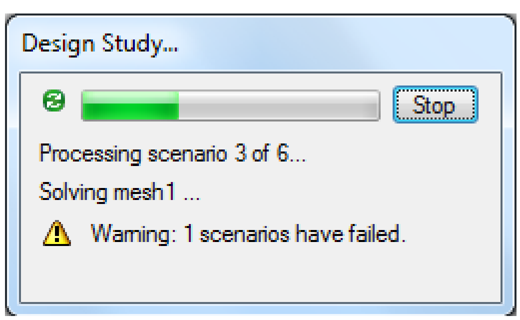
Like manual convergence checking, any results are available for viewing in the Design Study including Von Mises Stress, Resultant Displacement, and Strain. These can be specified under the **Constraints** area, and should be set to **Monitor Only**.

Before running the Design Study, the parameter must be linked from the mesh settings in the static study which the user is trying to check convergence for.

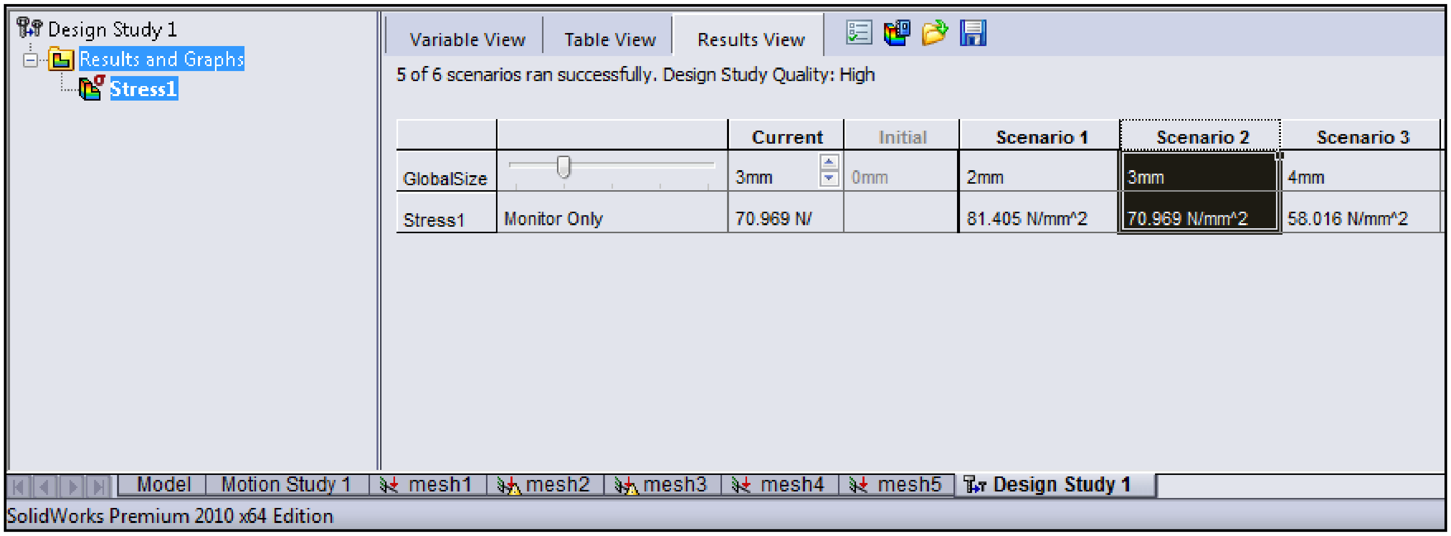
- In the **Mesh property manager**, click the drop-down next to **Global Size**, select **Link Value**, and choose the parameter that was created.



The Design Study can then be run. If any iterations of the study fail due to improper mesh settings, a warning message will appear.



Once the Design study is run, the results of each scenario are summarized under the **Results View** tab, and the scenario with the sufficiently refined mesh can be pinpointed. By clicking on each Scenario, the full model results can be analyzed in the graphics area, as well as by creating result plots in the Design Study tree on the left.



Summary

The concept of convergence is a complicated phenomenon in any finite element analysis. However, identifying and achieving convergence is important in many analysis situations where an extra level of confidence in results is necessary, especially when detailed information is necessary about stresses or factor of safety values.

SolidWorks Simulation has an array of tools to assist in convergence checking, including the Trend Tracker, Design Study, and adaptive meshing tools. These features allow the user to perform detailed convergence checking with an automated interface which reduces setup and analysis time, increasing the overall reliability and confidence or results with minimal effort.

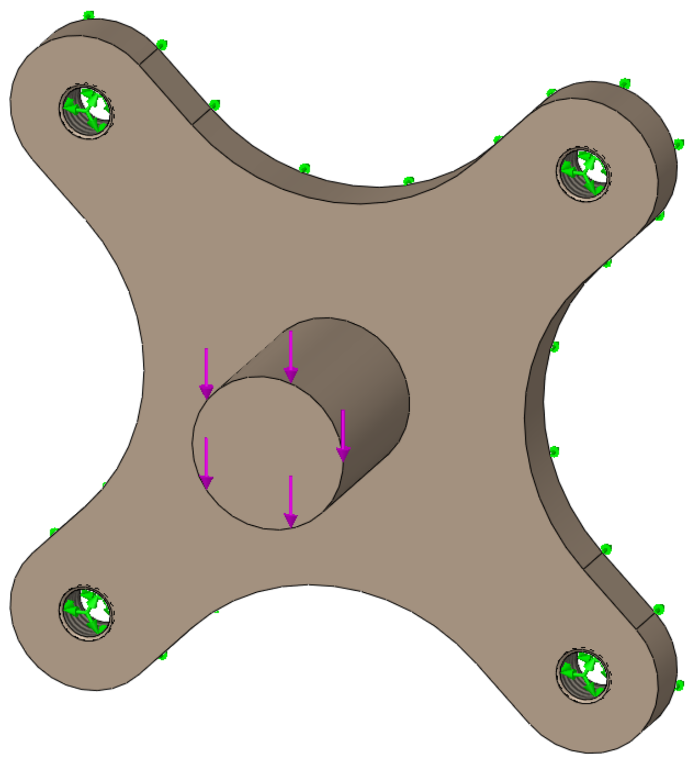
Appendix A: Example Problem

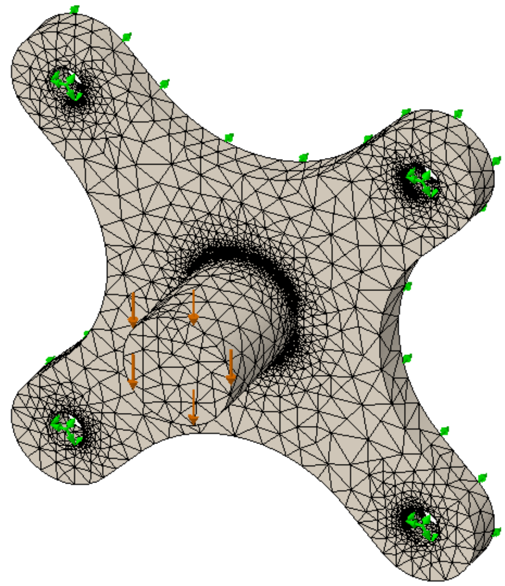
A Nickel bracket is attached flush to a wall through four M14x2.0 tapped holes. A 50-lb weight is attached to the end of a small cylinder in the bracket, resulting in a downward cantilevered force. The maximum stress in the bracket (presumed to be at the base of the cylinder) must be evaluated.

In order to ensure accurate stress results, we will try to use h-adaptive mesh convergence with a Target Accuracy of 98% (2% error).

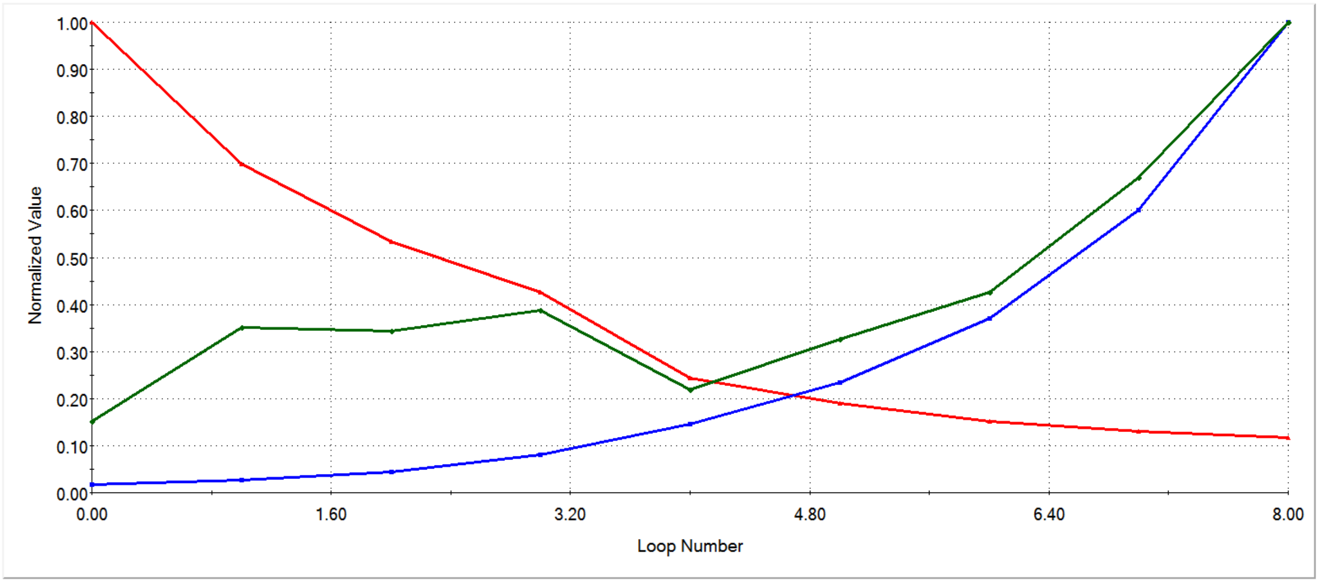
Restraint name	Selection set	Description
Fixed Hinge-2 <Convergence>	on 4 Face(s)Hinge	Bolted holes
Roller/Slider-1 <Convergence>	on 1 Face(s)Roller/Sliding	Flush contact with wall

Load name	Selection set	Loading type	Description
Force-1 <Convergence>	on 1 Face(s) apply force -50 lbf normal to reference plane with respect to selected reference Top Plane using uniform distribution	Sequential Loading	50-lb weight attached at end.





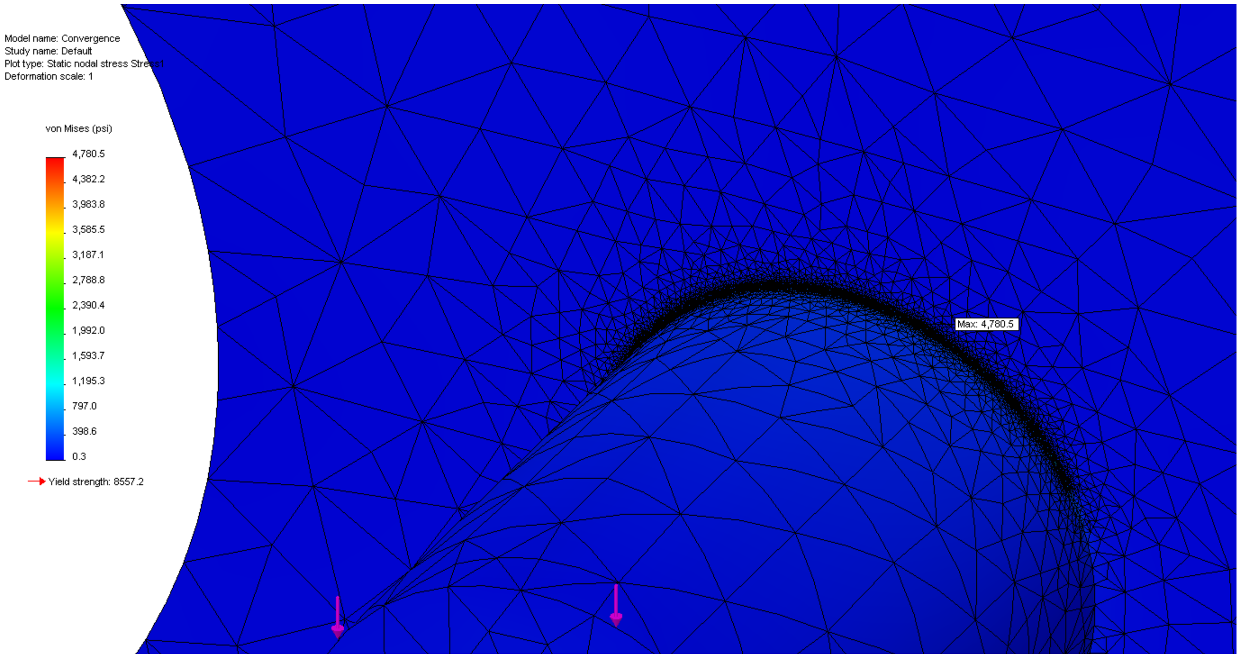
For the base design, the h- adaptive mesh required 8 loops (7 iterations beyond the initial mesh) in order to satisfy the 98% target accuracy. In this example, the maximum Von Mises stress in the corner of the bracket increases exponentially as the mesh is refined, due to a singularity.



Global Criterion: Total relative Strain Energy Norm error < 1.98731%

Target accuracy   Number of Nodes   Maximum von Mises Stress

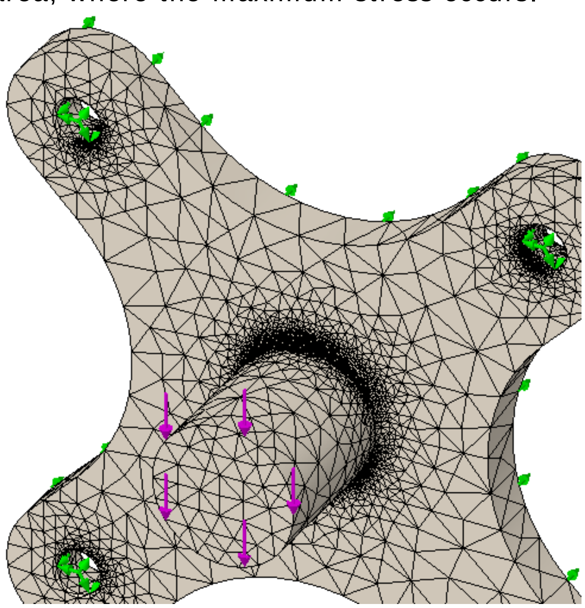
0.386124, 1.03455

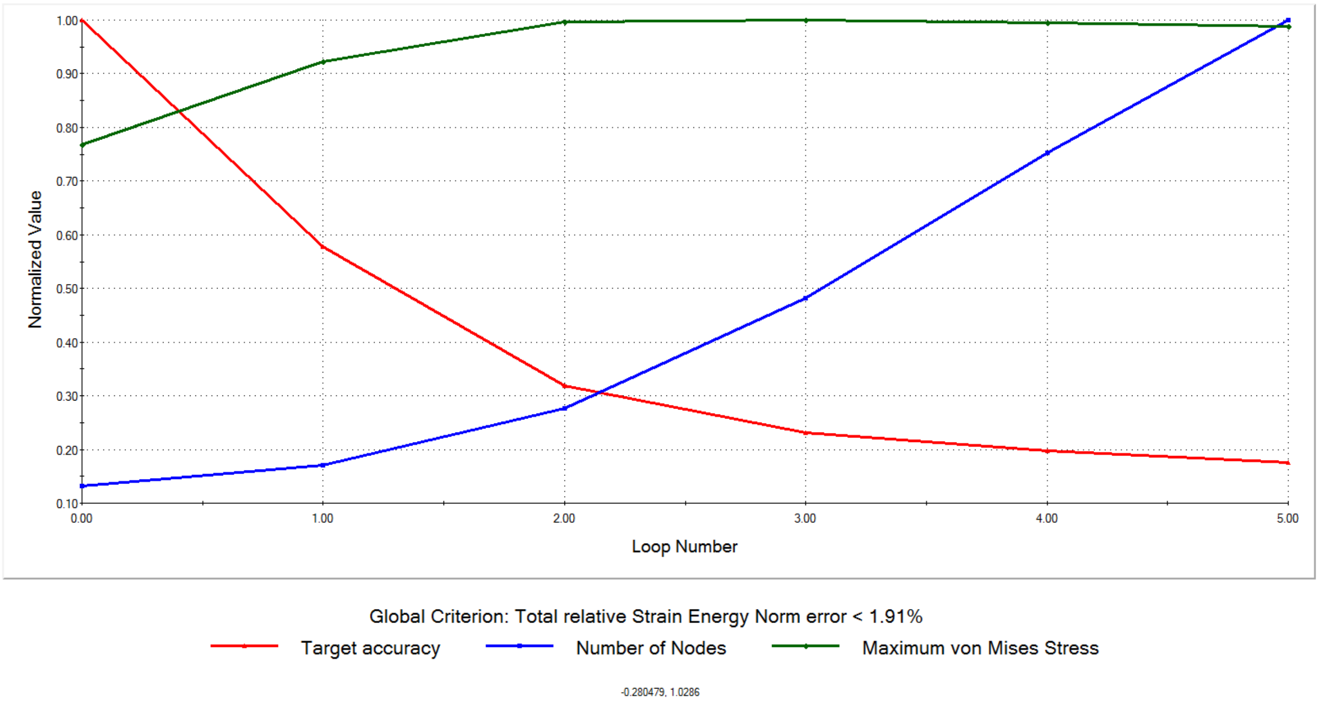


Despite achieving convergence based on strain energy norm error, the maximum Von Mises stress result of 4,780 psi is unreliable. One key indication of a singularity is that the stress is much higher at the corner than anywhere nearby in the model. Additionally, the mesh is refined so greatly that the number of nodes reaches 743,465, requiring significant computation time even for a simple one-part linear static study.

Added Fillet

In order to eliminate the distorted results caused by the stress singularity, a fillet of radius 0.1” is added to the base of the cylinder. Since an infinitely sharp corner does not exist in the real world, this will better emulate the stress results in this area, where the maximum stress occurs.





With the addition of the fillet, the h-adaptive mesh needs only 5 total loops to achieve 98% target accuracy. The h-adaptive mesh is also able to increase element size in low-stress areas of the bracket. This contributes to the much lower maximum node count of 107,289 which solves much more quickly compared to the default. The maximum Von Mises stress in the model levels off as the study converges, showing the elimination of the stress concentration. It finalizes at a more realistic value of 500.2 psi.