SolidWorks® 2010

SolidWorks Simulation

Dassault Systèmes SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742 USA

© 1995-2009, Dassault Systèmes SolidWorks Corporation, a Dassault Systèmes S.A. company, 300 Baker Avenue, Concord, Mass. 01742 USA.

All Rights Reserved.

The information and the software discussed in this document are subject to change without notice and are not commitments by Dassault Systèmes SolidWorks Corporation (DS SolidWorks).

No material may be reproduced or transmitted in any form or by any means, electronic or mechanical, for any purpose without the express written permission of DS SolidWorks.

The software discussed in this document is furnished under a license and may be used or copied only in accordance with the terms of this license. All warranties given by DS SolidWorks as to the software and documentation are set forth in the SolidWorks Corporation License and Subscription Service Agreement, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such warranties.

Patent Notices for SolidWorks Standard, Premium, and Professional Products

U.S. Patents 5,815,154; 6,219,049; 6,219,055; 6,603,486; 6,611,725; 6,844,877; 6,898,560; 6,906,712; 7,079,990; 7,184,044; 7,477,262; 7,502,027; 7,558,705; 7,571,079, and foreign patents, (e.g., EP 1,116,190 and JP 3,517,643).

U.S. and foreign patents pending.

Trademarks and Other Notices for All SolidWorks Products

SolidWorks, 3D PartStream.NET, 3D ContentCentral, DWGeditor, PDMWorks, eDrawings, and the eDrawings logo are registered trademarks and FeatureManager is a jointly owned registered trademark of DS SolidWorks.

SolidWorks Enterprise PDM, SolidWorks Simulation, SolidWorks Flow Simulation, and SolidWorks 2010 are product names of DS SolidWorks.

CircuitWorks, DWGgateway, DWGseries, Feature Palette, FloXpress, PhotoWorks, TolAnalyst, and XchangeWorks are trademarks of DS SolidWorks.

FeatureWorks is a registered trademark of Geometric Ltd.

Other brand or product names are trademarks or registered trademarks of their respective holders.

COMMERCIAL COMPUTER SOFTWARE - PROPRIETARY

U.S. Government Restricted Rights. Use, duplication, or disclosure by the government is subject to restrictions as set forth in FAR 52.227-19 (Commercial Computer Software - Restricted Rights), DFARS 227.7202 (Commercial Computer Software and Commercial Computer Software Documentation), and in the license agreement, as applicable.

Contractor/Manufacturer:
Dassault Systèmes SolidWorks Corporation, 300 Baker Avenue, Concord, Massachusetts 01742 USA

Copyright Notices for SolidWorks Standard, Premium, and Professional Products

Portions of this software © 1990-2009 Siemens Product Lifecycle Management Software III (GB) Ltd.

Portions of this software © 1998-2009 Geometric Ltd.

Portions of this software © 1986-2009 mental images GmbH & Co. KG.

Portions of this software © 1996-2009 Microsoft Corporation. All rights reserved.

Portions of this software © 2000-2009 Tech Soft 3D.

Portions of this software © 1998-2009 3Dconnexion.

This software is based in part on the work of the Independent JPEG Group. All Rights Reserved.

Portions of this software incorporate PhysX™ by NVIDIA 2006-2009.

Portions of this software are copyrighted by and are the property of UGS Corp. © 2009.

Portions of this software © 2001-2009 Luxology, Inc. All Rights Reserved, Patents Pending.

Portions of this software © 2007-2009 DriveWorks Ltd.

Copyright 1984-2009 Adobe Systems Inc. and its licensors. All rights reserved. Protected by U.S. Patents 5,929,866; 5,943,063; 6,289,364; 6,563,502; 6,639,593; 6,754,382; Patents Pending.

Adobe, the Adobe logo, Acrobat, the Adobe PDF logo, Distiller and Reader are registered trademarks or trademarks of Adobe Systems Inc. in the U.S. and other countries.

For more copyright information, in SolidWorks see Help > About SolidWorks.

Other portions of SolidWorks 2010 are licensed from DS SolidWorks licensors.

Copyright Notices for SolidWorks Simulation

Portions of this software © 2008 Solversoft Corporation.

PCGLSS © 1992-2007 Computational Applications and System Integration, Inc. All rights reserved.

Portions of this product are distributed under license from DC Micro Development, Copyright © 1994-2005 DC Micro Development, Inc. All rights reserved.

Document Number: PMT1040-ENG

Lesson 1
The Analysis Process

Objectives

Upon successful completion of this lesson, you will be able to:

- Navigate the SolidWorks Simulation interface.
- Execute a linear static analysis using solid elements.
- Understand the influence of mesh density on displacement and stress results.
- Employ various methods to present FEA results.
- Manage SolidWorks Simulation result files.
- Access available help and assistance.

The Analysis Process

The process of analyzing models consists of the same basic steps regardless of the type of analysis or model. We must understand these steps fully to have a meaningful analysis.

Stages in the Process

Some key stages in the analysis of a model are shown in the following list:

- **Create a study**
  Each analysis we do of a model is a study. We can have multiple studies on each model.
- **Apply material**
  We apply a material which contains the physical information, such as yield strength, to the model.
- **Apply fixtures**
  Fixtures are added to represent the way the physical model is held.
- **Apply loads**
  Loads represent the forces on the model.
- **Mesh the model**
  The model is broken into finite elements.
- **Run the study**
  The solver calculates the displacement, strain and stress in the model.
- **Analyze the results**
  The results are interpreted.

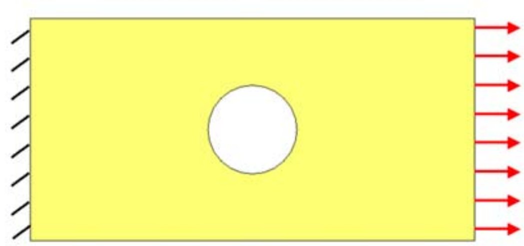
Case Study: Stress in a Plate

In this first case study, we will determine the stress in a rectangular plate, with a hole in it, under a tensile load. We will use this simple model to familiarize ourselves with all the steps and the majority of the software functionality typically used in a static analysis of solid models.

In spite of its simplicity, this is probably the most important lesson in this course. This lesson goes through all the required steps; however, after the lesson is complete, you should continue to explore other software functionality and other modeling assumptions, such as different material properties, loads, restraints, and so on.

Project Description

The rectangular plate with a hole is fixed at a short-end face. A 110,000 Newton load is uniformly distributed along the other end face.



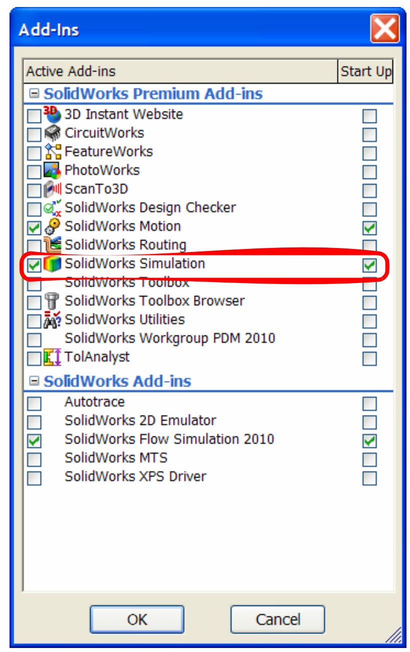
In addition to learning SolidWorks Simulation functions, our objective is to investigate the impact of different mesh densities on the results. Using FEA terminology, the objective is to investigate the effect of different discretization choices on the data of interest, in our case, on deformation and stress.

Therefore, we perform the analyses using meshes with different element sizes. Note that repetitive analysis with different meshes does not represent standard practice in FEA. We repeat the analysis using different meshes as a learning tool to gain more insight into how FEA works.

1 Open a part file.
Open `rectangular hollow plate` from the `Lesson01\Case Studies` folder. Review the dimensions of the model and note down the length, width, and thickness of the part in millimeters.

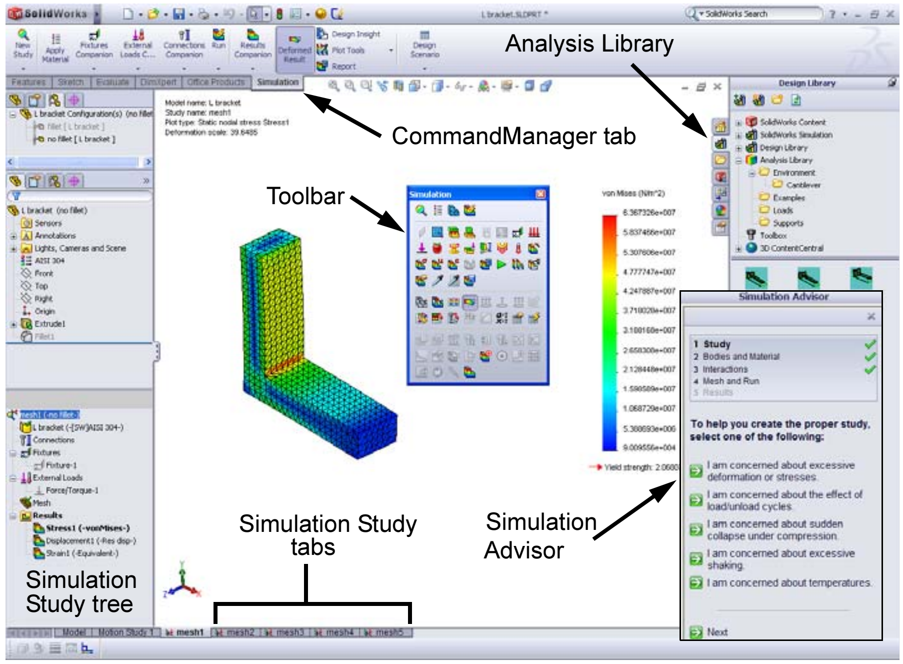
2 Start SolidWorks Simulation.
Click **Tools, Add-Ins**. Select **SolidWorks Simulation**.

Click **OK**.



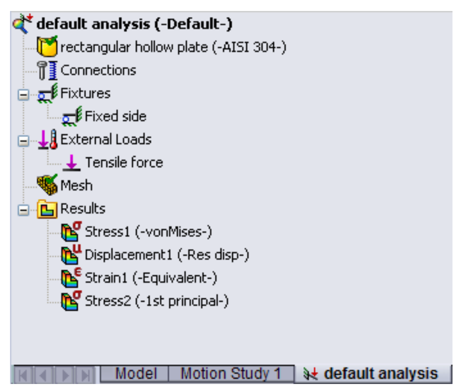
SolidWorks Simulation Interface

SolidWorks Simulation functions are accessed in the same way as core SolidWorks. To create an FEA model, solve the model, and analyze the results, we use a graphical interface in the form of icons and folders located in the SolidWorks Simulation Study tree window.



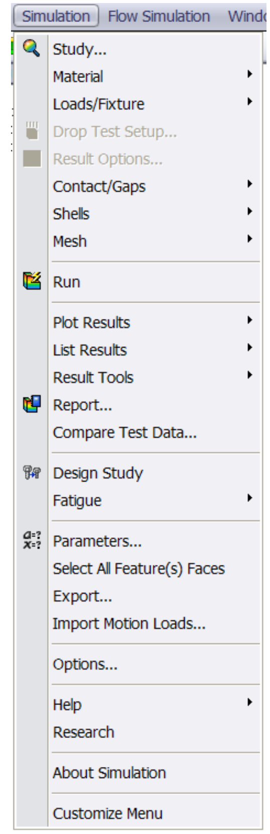
Simulation Study Tree

Once a simulation study is created, the Simulation Study tree will appear in the lower part of the FeatureManager design tree. Its visibility is controlled by a tab below the graphics area.



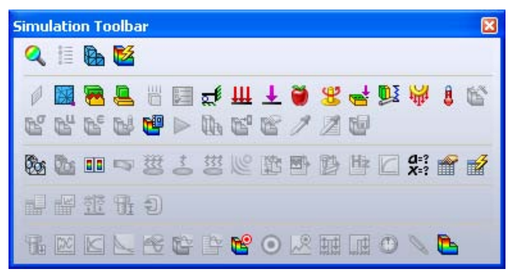
Pull-down Simulation Menu

The **Simulation** menu provides a method to access all the commands for simulation.



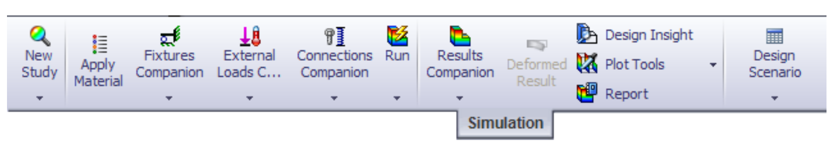
Toolbars

The Simulation toolbar contains all the commands that have toolbar buttons. It can be customized to show only those commands you use frequently.



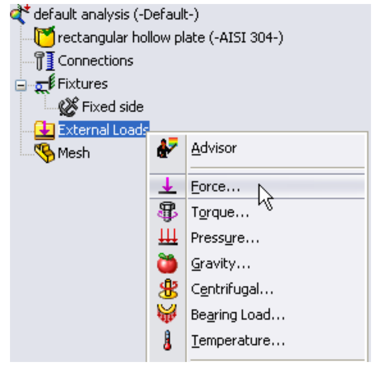
CommandManager

The CommandManager provides a universal toolbar for simulation. The **Simulation** tab provides the tools to setup a study and for analyzing the results.



Context menus

Functions can be selected by right-clicking geometry or items in the Simulation Study tree.



SolidWorks Simulation Options

Located on the Simulation menu, the Options dialog box enables you to customize the Simulation software to reflect the standards your company uses for analysis. There are two categories of options, system and default.

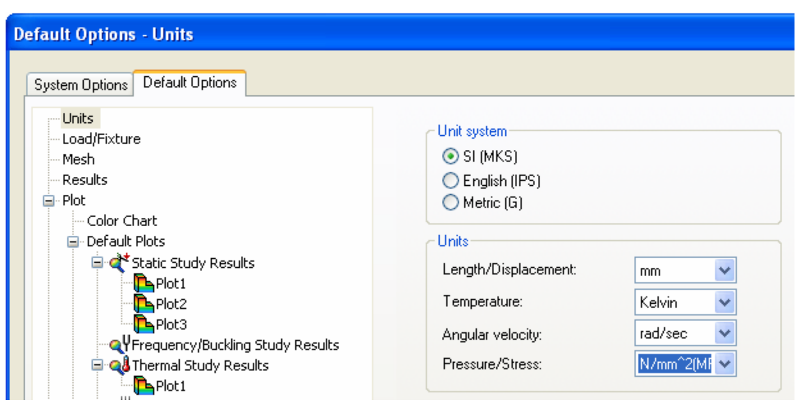
- **System Options**
  System options apply to all studies. Included are the settings for the way the errors are displayed and the location of the default libraries.
- **Default Options**
  Default options apply to new studies. As we do not use templates for simulation studies, this is where the options are set for units, default plots, etc.

Where to Find It

- Select **Options** from the **Simulation** menu.

3 **Open Options window.**
Click **Options** on the **Simulation** menu.

4 **Set default units for SolidWorks Simulation.**
Under **Default Options**, select **Units**. Make sure that the **Units** system is set to **SI (MKS)** and **Length/Displacement** and **Stress** are in **mm** and **N/mm^2(MPa)**, respectively.



5 Set default results.

Under **Default options**, select the Results folder. In this lesson, the analysis results will be created and stored in a sub-folder located in the SolidWorks document directory.

Under **Results folder**, select **SolidWorks document folder. SolidWorks document folder** is the folder where `rectangular hollow plate.SLDPRT` file resides in your computer.

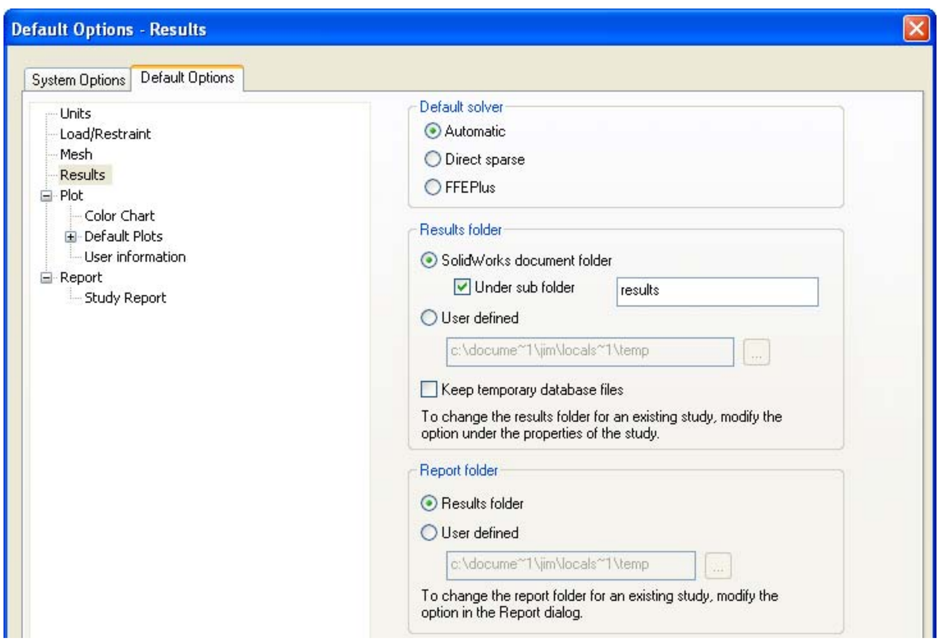
Select the **Under sub folder** check box.

In the **Under sub folder** box, enter **results**. This will automatically create a sub folder results to store SolidWorks Simulation results.

The **Report folder** (the place to store automatically generated reports) is by default the same as the Result folder.

Under **Default Solver**, select **Automatic**.

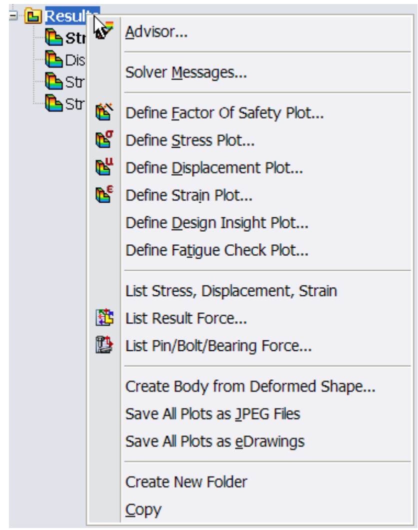
Note Solvers will be discussed later in the course.



Plot Settings

Upon completion of any static analysis, SolidWorks Simulation automatically creates the following result plots:

- **Stress1**
- **Displacement1**
- **Strain1**

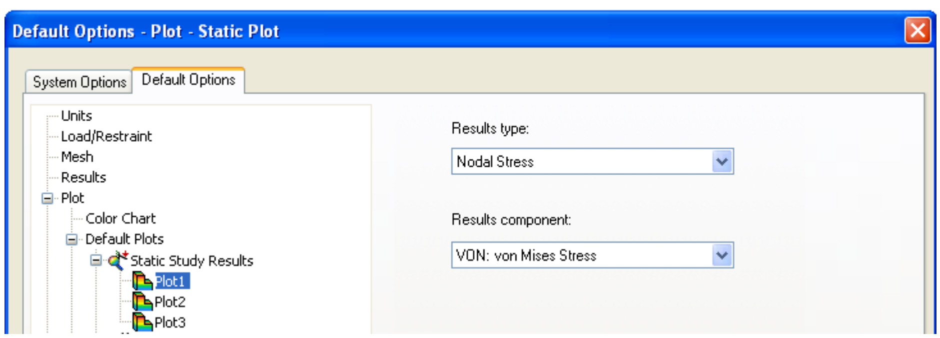


The plot settings determine which plots will be automatically created and their units. To add an additional plot, right-click Static Study Results and select the type of plot you wish to define. Each type of plot can be stored in a user-defined folder.

6 Set default plots.

Expand the Default plots subfolder located in the Plot folder. This section allows you to select default result plots to be generated after solving the analysis.

We will use the default settings in the Default plots folder for this lesson.

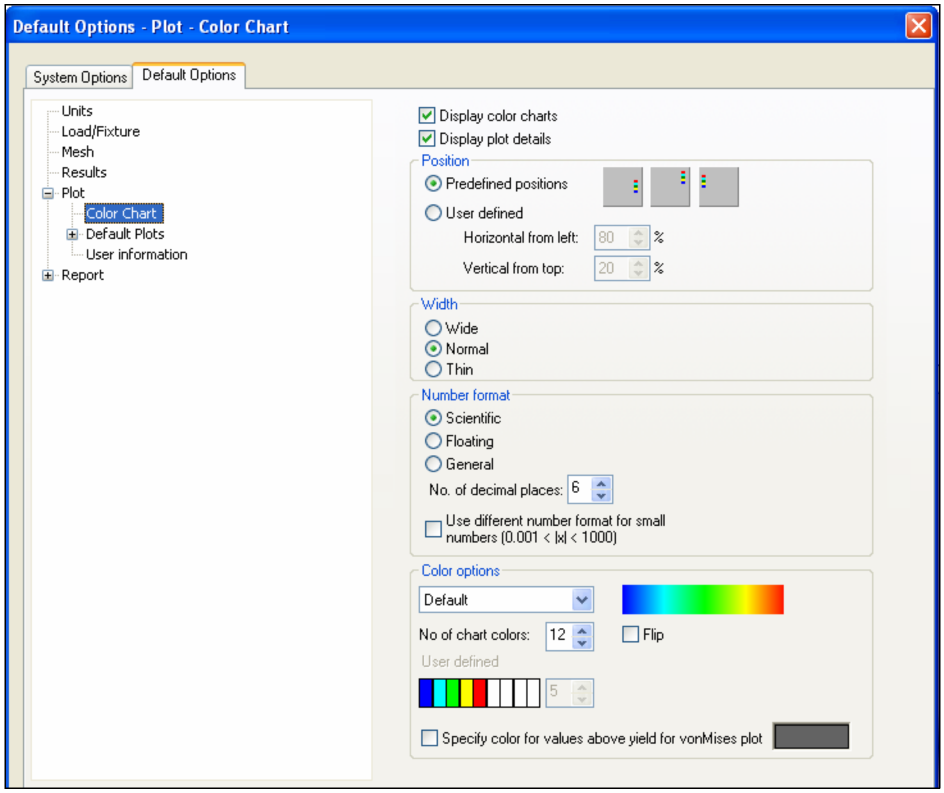


7 Specify color chart options.
Under the Plot folder, select Color chart.

Set **Number format** to **scientific (e)** and **No. of decimal places** to **6**.

Explore all the chart options in this dialog.

Click **OK** to close the **Options** window.



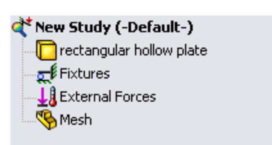
Preprocessing

In the following steps, we will prepare the model for analysis. The preprocessing steps include:

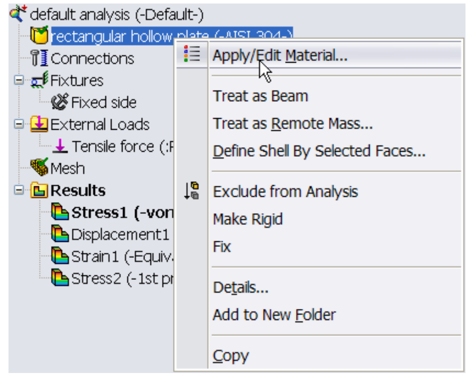
- **Create a study**
- **Apply material**
- **Apply fixtures**
- **Apply external forces**
- **Mesh the model**

New Study

Creation of an FEA model always starts with the definition of a study.



The study definition is where we enter information about the kind of analysis we wish to perform.



Each analysis we do is a separate study. When a study is defined, SolidWorks Simulation automatically creates a study folder (named in this case, `default analysis`) and places several icons in it.

Some of the icons are folders that contain other icons.

We use the `Parts` folder to define and assign material properties, the `External Loads` folder to define loads, the `Fixtures` folder to define fixtures, and the `Mesh` icon to create the finite element mesh.

The `Connections` folder is not used in this lesson.

There is only one component, named `rectangular hollow plate`, in the `Parts` folder. If an assembly (and not a part) is analyzed, then the `Parts` folder contains as many components as there are parts in the assembly.

Renaming Studies

The name of the study can be changed at any point by click-pause-clicking on the study name (similarly to renaming files and folders in Windows).

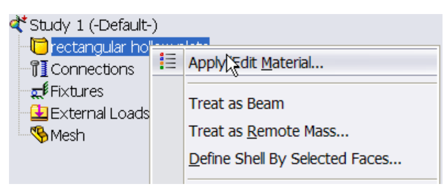
Assigning Material Properties

We can assign material to the model in either the SolidWorks or the SolidWorks Simulation window.

If a material was assigned in the SolidWorks window, then the material definition will be transferred automatically to SolidWorks Simulation.

In this lesson, we assign material to the part in the SolidWorks Simulation window, not because this is the preferred way, but to demonstrate this option.
To assign a material:

- Select **Material** on the Simulation menu, then click **Apply material to all**.
- Select the component, then click **Apply Material** in the **Simulation Main** toolbar.
- Right-click the `body/part/assembly` icon in the Simulation Study tree and select **Apply/Edit Material**.



Note

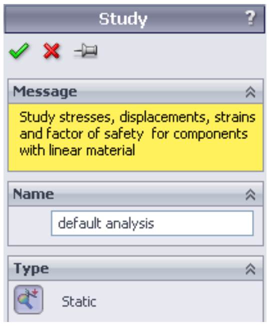
The first method assigns the same material properties to all components in the model. The second method assigns material properties to one particular component and all the multi-bodies associated to the component. The third method assigns material properties to one particular body: in this lesson, the `rectangular hollow plate`. Because we are not working with an assembly but with a single part which contains only one body (i.e. this is not a multi body part) any of the above three ways of material assignment can be used.

8 Create a study.

Click **Study** on the **Simulation** menu.

9 Name the study.

Studies can have any name; here we name the study `default analysis`. Type `default analysis` for the **Name**.



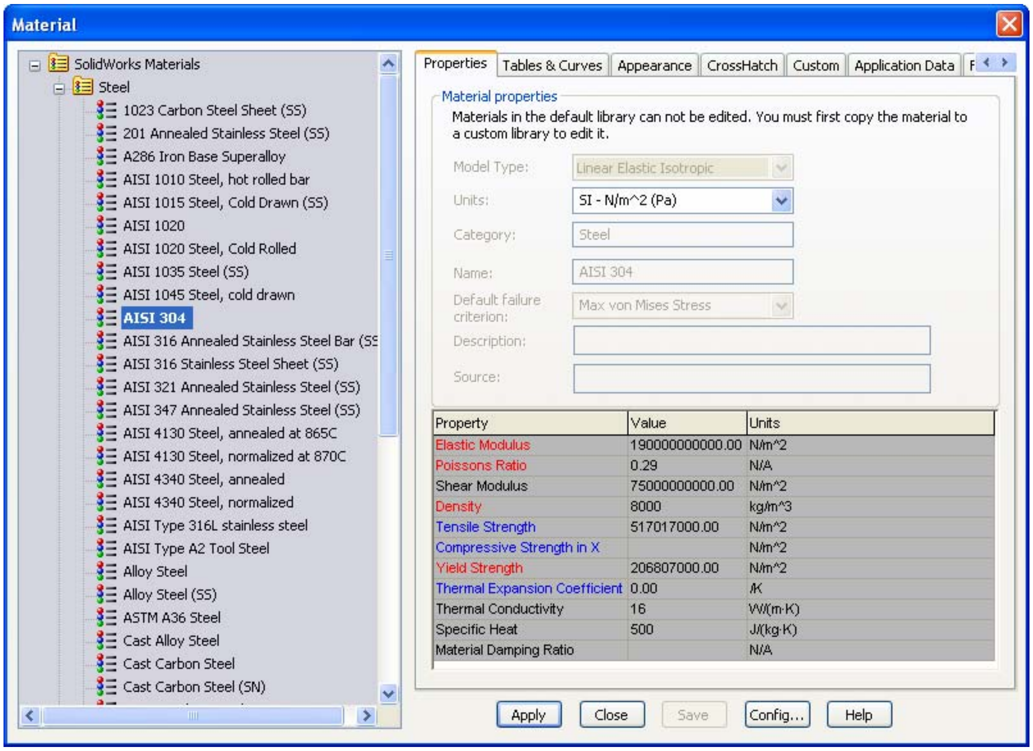
Select **Static** for the **Type** of study.

Click **OK**.

10 Assign material properties.

Click **Material** in the **Simulation** menu. Click **Apply Material to All**.

Expand `Solidworks Materials` and assign **AISI 304** from the `Steel` folder.



The required material constants are in red. The constants shown in blue may be required if specific load types are used (for example, the **Temperature** load would require the **Thermal expansion coefficient**).

Note that you may add a new material library by right clicking any folder or existing material in the **Material** dialog window. The new material can be added by copying the existing material into a new location and editing its properties.

Click **Close**.

The `rectangular hollow plate` icon in the `Parts` folder now displays a green check mark and the name of the selected material to indicate that a material has successfully been assigned.

Fixtures

To do a static analysis, the model must be properly restrained so that it cannot move. SolidWorks Simulation provides various fixtures that can be used to restrain the model. Generally, fixtures can be applied to faces, edges, and vertices using various methods.

Fixture Types

The fixtures and restraints are grouped as **Standard** and **Advanced**. Their properties are summarized below:

Standard Fixtures

Fixture Type	Definition
Fixed Geometry	Also called a rigid support, all translational and all rotational degrees of freedom are constrained. **Fixed Geometry** does not require any information on the direction along which restraints are applied.
Roller/Slider	Use the **Roller/Slider** restraint to specify that a planar face can move freely in its plane but cannot move in the direction normal to its plane. The face can shrink or expand under loading.
Fixed Hinge	Use the hinge restraint to specify that a cylindrical face can move only about its axis. The radius and the length of the cylindrical face remain constant under loading.

Advanced Fixtures

Fixture Type	Definition
Symmetry	This option is available for use on flat face; in-plane displacements are allowed and rotation in the direction normal to the plane is allowed.
Circular Symmetry	This option is used to restrain segments which, if periodically revolved around a specified axis of revolution, would form a rotationally symmetrical body.
Use Reference Geometry	This option restrains a face, edge, or vertex only in desired direction(s), while leaving the other directions free to move. You can specify the desired direction(s) of restraint in relation to the selected reference plane, axis, edge, or face. The SolidWorks Flyout FeatureManager is useful for selecting reference geometry (plane and axis).
On Flat Faces	This option provides restraints in selected directions, which are defined by the three principal directions of the flat face where restraints are being applied.
On Cylindrical Faces	This option is similar to **On flat face** except that the three principal directions of a cylindrical reference face define the directions in a cylindrical coordinate system; this option is very useful because you can apply a restraint that allows for rotation about the axis associated with the cylindrical face.
On Spherical Faces	Similar to **On flat faces** and **On cylindrical faces**; the three principal directions of a spherical face define the directions of the applied restraints in a spherical coordinate system.

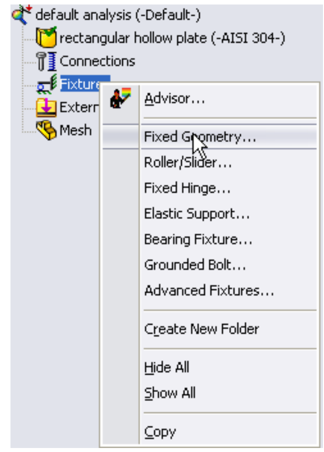
Display/Hide Symbols

Fixture and External Forces symbols can be displayed or hidden by doing one of the following actions:

- Right-click `Fixtures` or `External Forces` and select **Hide All** or **Show All**.
- Right-click a `Fixture` or `External Forces` symbol for each restraint individually, and then select **Hide** or **Show**.

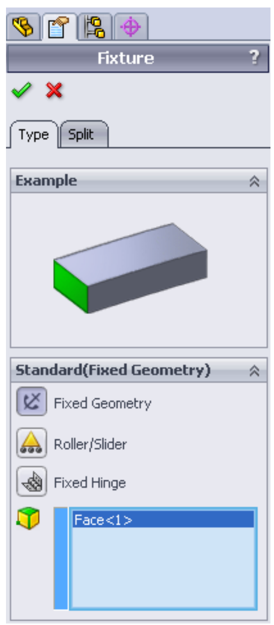
11 Define Fixed Restraints.

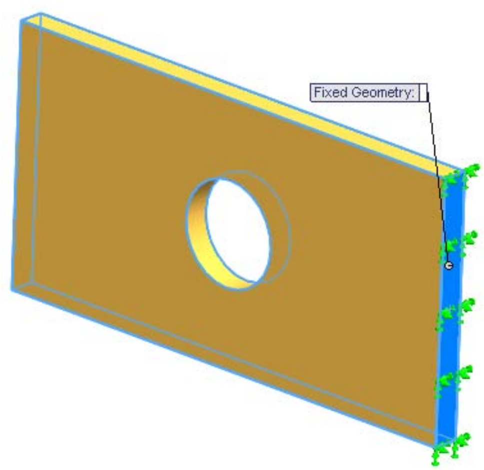
In the Simulation Study tree, right-click `Fixtures` and select **Fixed Geometry**.



Rotate the model and select the face to apply restraints. The Flyout FeatureManager is available in the upper left corner of the graphics area to make selection easier for parts, bodies or features.

In the **Type** box, select **Fixed Geometry**, and then click **OK** ✓ to close the **Fixture** PropertyManager.

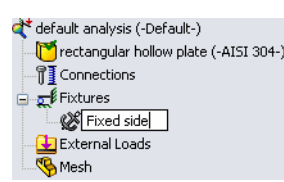




Having defined fixtures, we have fully restrained the model in space. Therefore, the model cannot displace without elastic deformation. In FEA terminology, we say that the model does not have any rigid body modes.

Renaming

Each boundary condition can be renamed to help us decipher the meaning later on.



Window's standard click-pause-click technique can be used to rename fixtures, loads and connectors.

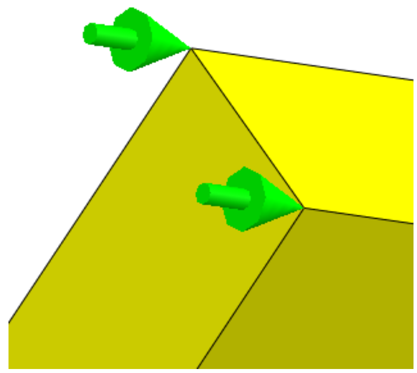
Fixture Symbols

Fixture symbols are displayed on the face where they have been applied.

In this case study, we select **Fixed Geometry** as the fixture type, meaning that all six degrees of freedom (three translations and three rotations) have been restrained.

The fixture symbols are arrows to indicate translational restraints and discs to indicate rotational restraints in respective directions. In this lesson, the fixtures are defined by the directions of the global coordinate system visible in the lower-left corner of the model window.

If, instead of selecting **Fixed Geometry** as the type of fixture, we selected **Roller/Slider**, then the rotational degrees of freedom would not be constrained and the corresponding fixture symbols would feature only arrows, not discs.



External Loads

Once the model is restrained, we must apply external loads, or forces, to the model. SolidWorks Simulation provides various external forces that can be used to load the model. Generally, forces can be applied to faces, edges, and vertices using various methods. These external forces and their properties are summarized below:

Standard External Forces

Force Type	Definition
Force	This option applies a force or moment to a face, edge, or vertex in the direction defined by selected reference geometry (plane, edge, face, or axis). Note that a moment can only be applied if shell elements are used. Shell elements have six degrees of freedom per node (translations and rotations) and can assume a moment load. Solid elements have only three degrees of freedom per node (translations only) and, therefore, cannot assume a moment load directly. If you need to apply a moment to solid elements, it must be represented by appropriately distributed forces, or remote loads.
Torque	This option applies torque about a reference axis using the Right-hand Rule. This option requires that the axis be defined in the SolidWorks.

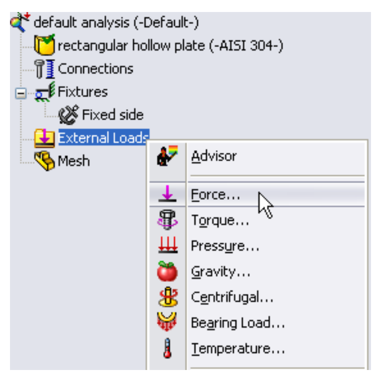
Advanced External Forces

Force Type	Definition
Pressure	Applies a pressure to a face. Can be directional and variable, such as hydrostatic pressure.
Gravity	Applies linear accelerations to parts or assemblies.
Centrifugal Force	Applies an angular velocity and acceleration to a part or assembly.
Bearing Load	Bearing loads are defined between contacting cylindrical faces.
Remote Load/ Mass	Remote loads apply loads that would normally be transferred by connecting structure.
Distributed Mass	Distributed masses are applied to selected faces to simulate the mass of components that are suppressed or not included in the model.

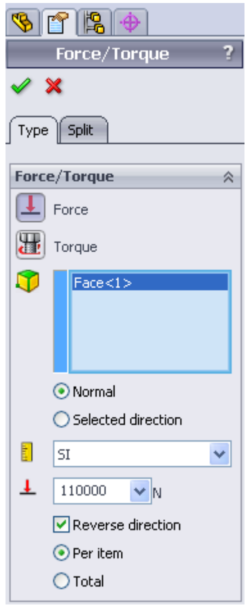
The presence of an external force is indicated by arrows symbolizing the load and by the corresponding icon.

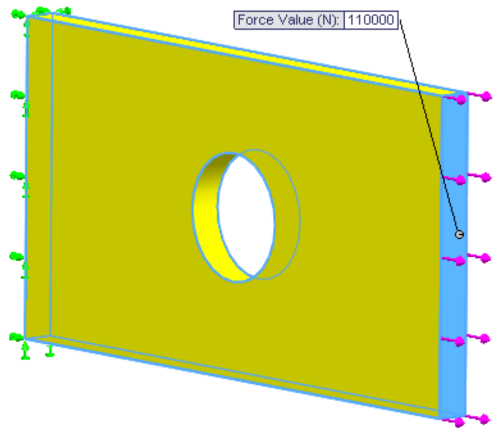
12 Rename the fixture.
Use the Windows click-pause-click method to rename the fixture called `Fixture-1` to `Fixed side`.

13 Define Force.
Rotate the model to reveal the face where the 110,000 N [24,729 lbf] tensile force is to be applied and select this face.



Right-click External Loads and select **Force** to list the available options for defining loads. This action opens the **Force/Torque** PropertyManager.





In the **Type** area, select **Normal**, in the **Units** dialog make sure that **SI** is selected, and in the **Force Value** box, type **110,000**.

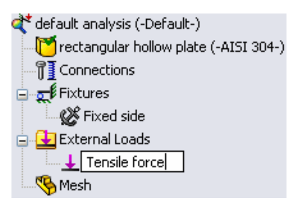
Select **Reverse direction**. This is required to define a tensile force. Clearing the **Reverse direction** check box would result in a compressive force.

When defining a normal force we do not need to use any reference geometry. Load direction is sufficiently defined by the orientation of the loaded face when **Normal** is in effect.

Click **OK** .

14 Rename the force.

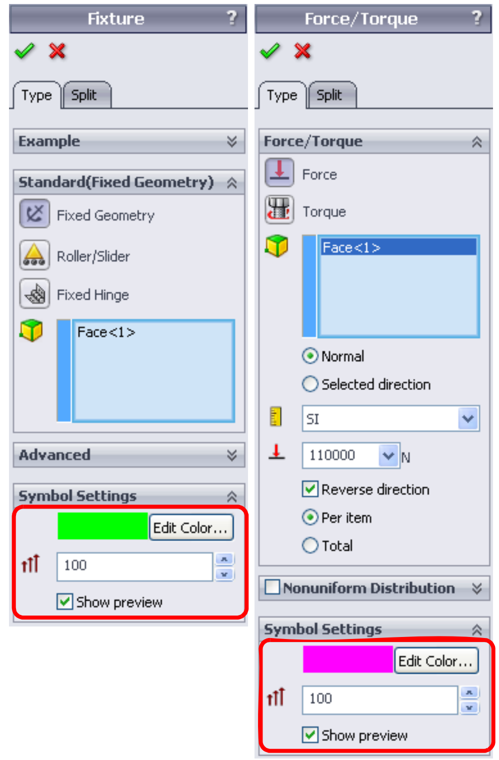
Rename this force definition to Tensile force.



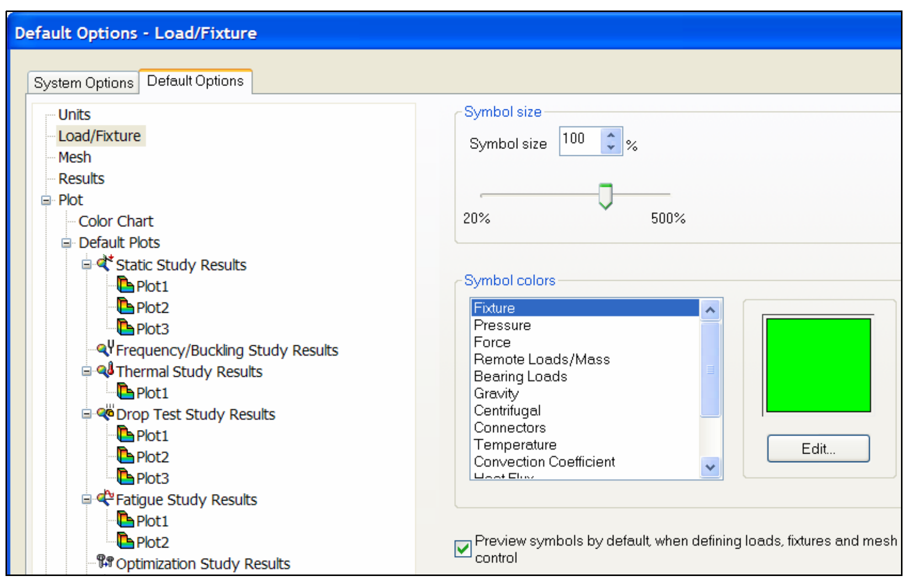
Size and Color of Symbols

The size and color of restraint and load symbols can be controlled both locally and globally.

The local settings of the symbols are controlled from the **Symbol settings** dialog in the **Fixtures** and **External Loads** PropertyManagers.



The global definitions for the symbols can be controlled by the SolidWorks Simulation Options in the External Loads folder.



Display/Hide Symbols

The model now shows both loads and restraints symbols. To hide or show the symbols:

- Right-click a particular restraint or load icon in the Fixtures or External Forces folder and choose **Show** or **Hide**.
- Right-click the Fixtures or External Loads folder to globally display or hide loads and restraints and choose **Show All** or **Hide All**.

Preprocessing Summary

Now that we have assigned the material properties, fixtures, and external loads, we have fully defined the mathematical model, which we intend to solve with FEA.

The mathematical model must be discretized into a finite element model. Before creating the finite element model, let us make a few observations about the following terms:

- Geometry preparation
- Material properties
- External forces definition
- Fixtures definition

Geometry Preparation

Geometry preparation is a well-defined step with few uncertainties. Geometry that is simplified for analysis can be checked visually by comparing it with the original CAD model.

Material Properties

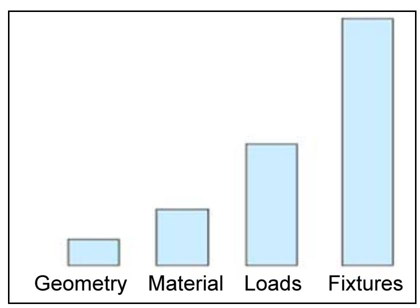
Material properties are most often selected from the material library and do not account for local defects, surface conditions, and so on. Generally, material definition has more uncertainties than geometry preparation.

External Loads Definition

External loads definition, even though done in a few quick menu selections, involves many background assumptions because in real life, load magnitude, distribution, and time dependence are often known only approximately and must be roughly estimated in FEA with many simplifying assumptions. Therefore, significant idealization errors can be made when defining loads. Nonetheless, loads can be expressed in numbers, which makes loads easier for FEA users to relate to.

Fixtures Definition

Defining restraints is where severe errors are most often made. A common error is over-constraining the model, which results in an overly stiff structure that underestimates deformations and stresses.



The relative level of uncertainties in defining geometry, material, loads, and fixtures is qualitatively shown.

Idealizations and Assumptions

Geometry is the easiest to define while fixtures are the most difficult, but the level of difficulty has no relation to the time required for each step, so the message in this bar graph may be counterintuitive. In fact, preparing CAD geometry for FEA may take hours, while defining material, and applying loads and fixtures involves only a few mouse clicks.

In all examples here, we assume that material properties, external forces, and supports are known with certainty, and that the way they are defined in the model represents an acceptable idealization of real conditions. However, we need to emphasize that it is the responsibility of the user of the FEA software to determine if all those idealized assumptions made during the creation of the mathematical model are indeed acceptable. The best automesher and the fastest solver do not help if the mathematical model submitted for analysis with FEA is based on erroneous assumptions.

Meshing

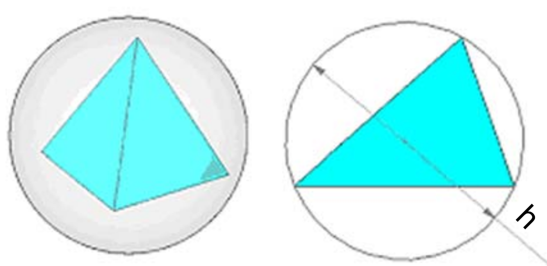
The last step before processing the FEA model is to mesh the geometry. In this step, the geometry will be divided into finite elements by an automesher. While the automesher will take care of the tedious part of the problem, we have input into the process to control the size and quality of the mesh.

Mesh Size

SolidWorks Simulation will suggest medium mesh density as the default that SolidWorks Simulation will use for meshing our model.

The element size and the element size tolerance are established automatically, based on the geometric features of the SolidWorks model. SolidWorks Simulation uses the units of length specified in the SolidWorks model for the element size. Remember, however, that we can enter analysis data and analyze results in any one of three unit systems: SI, Metric and English.

The element size represents the characteristic element size in the mesh and is defined as the diameter of a sphere circumscribing the element (on the left in the following figure). This representation is easier to illustrate with the 2-D analogy of a circle circumscribing a triangle (on the right in the following figure).



Mesh density directly affects the accuracy of results. The smaller the elements, the lower the discretization errors, but the longer the meshing and solution times.

In the majority of analyses with SolidWorks Simulation, the default mesh settings produce a mesh that provides acceptable discretization errors while keeping solution times reasonably short.

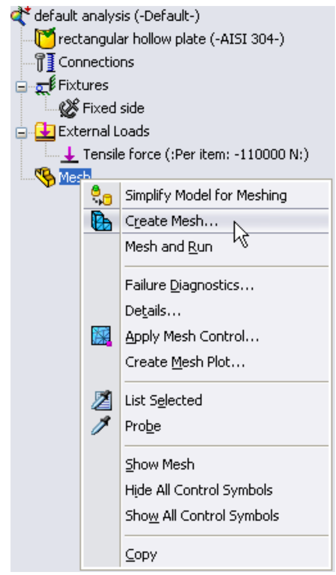
Tolerance

This field sets the tolerance in the global size of finite elements. The default value is equal to 5% of the global element size. In some instances, when the mesher fails to mesh the model, increasing the tolerance may help.

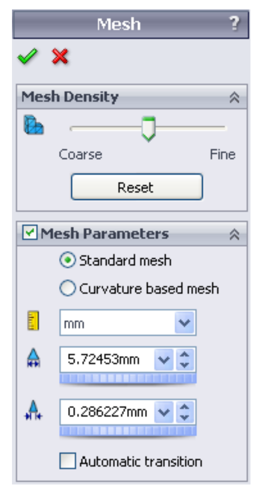
15 Generate the mesh.
Right-click `Mesh` and select **Create Mesh**.

The model will be meshed using **High** quality elements.

Expand all the sections of the PropertyManager to see all the available choices.



16 Set the mesh density.
The default mesh density will have the slider at mid-scale. Under **Mesh Parameters**, the size of the mesh is shown as **5.72453 mm** [0.2254 in] and the Tolerance is 5% of that value at **0.286227 mm** [0.01126 in]. For the initial analysis, we will use the default settings.



Mesh Quality

The mesh can be created with either a **High** or **Draft** mesh quality. The default is to use a High quality mesh. To use a draft quality mesh, you must select it in the PropertyManager under **Advanced** options.

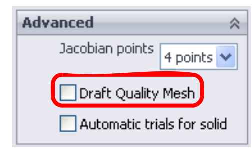
The difference between High and Draft quality is that:

- Draft quality mesh uses first order elements.
- High quality mesh uses second order elements.

The differences between first and second order elements are discussed in *Element Types Available in SolidWorks Simulation* in the *Introduction to FEA* chapter.

17 Set mesh quality.

In the **Advanced** section, clear **Draft Quality Mesh**.

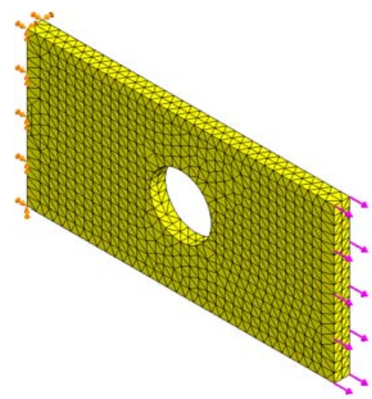


We will review the other mesh options as we proceed with the class.

Click **OK** to generate the mesh.

The mesh appears after mesh generation is completed.

The `Mesh` icon in the SolidWorks Simulation Study tree window now displays a green check mark to indicate that meshing has been successfully completed.



Note

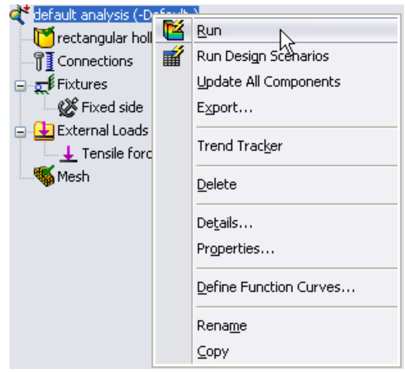
We named this study default analysis with the intention of using the default mesh size. Later on in this lesson the problem will be solved again with coarse and fine meshes.

Display/Hide Mesh

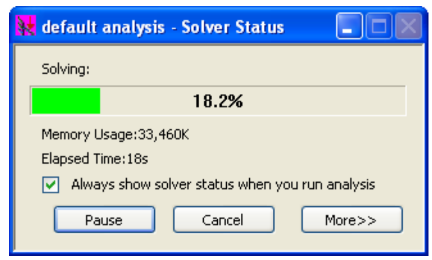
Mesh visibility can be controlled by right-clicking `Mesh`, and then doing one of the following:

- Select **Hide Mesh**.
- Select **Show Mesh**.

18 Run the analysis.
Right-click the study icon, `default analysis`, and select **Run**.



You can monitor or pause the solution in the solver window while the analysis is running.



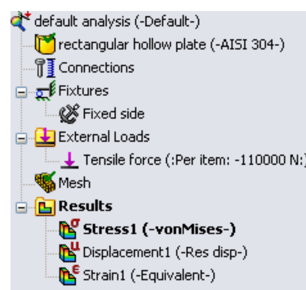
Postprocessing

After the analysis is complete, SolidWorks Simulation automatically creates the `Results` folder with the default results plots that we specified at the beginning of the lesson: `Stress1 (-vonMises-)`, `Displacement1 (-Res disp-)`, and `Strain1 (-Equivalent-)`.

Result Plots

Each result plot can be displayed by doing one of the following:

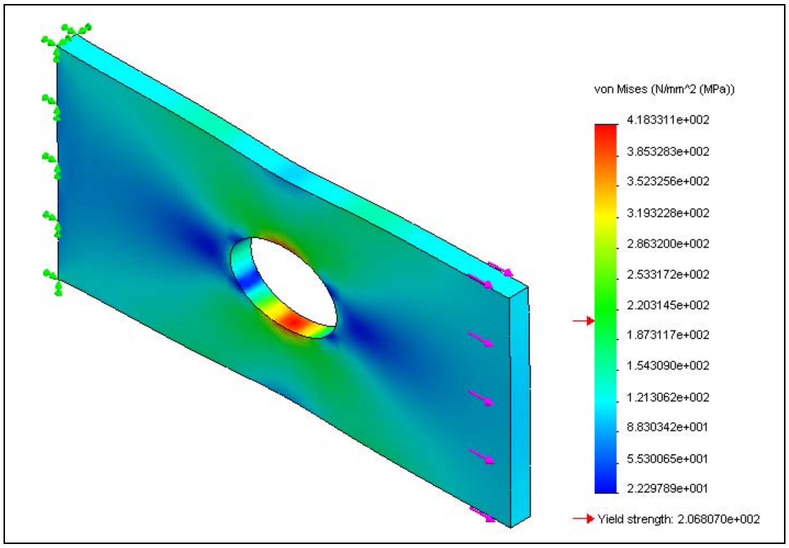
- Double-click the desired plot icon (`Stress1`, for example).
- Right-click the desired plot icon (`Stress1`, for example) and select **Show** under any folder.



While a plot is active (appears in the model window) you can right-click the plot icon again to examine the plot control options.

19 Show and edit `Stress1 (-vonMises-)` plot.
Double-click on `Stress1 (-vonMises-)` under the `Results` folder to display the plot.

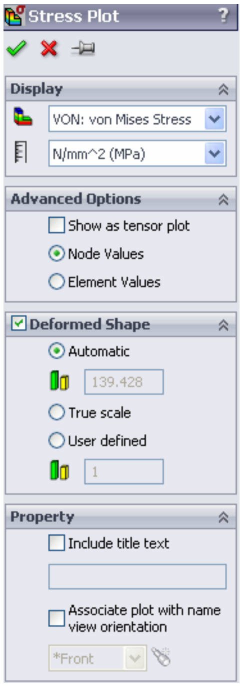
Notice that the stress plot is in Mega-pascals (N/mm^2) units and the legend features scientific numbers with six digits, just as we requested in the **Options** at the beginning of the lesson.



We observe that the maximum value of Von Mises stress is 418 MPa, which significantly exceeds the yield stress of the material, 206 MPa, indicated by the red marker in following the chart.

Editing Plots

To edit a plot, right-click on the plot and select **Edit definition**.



The **Display** dialog lets you specify a stress component, units, and the type of plot.

The **Advanced Options** dialog lets you choose to plot either **Node** or **Element values** which is discussed below.

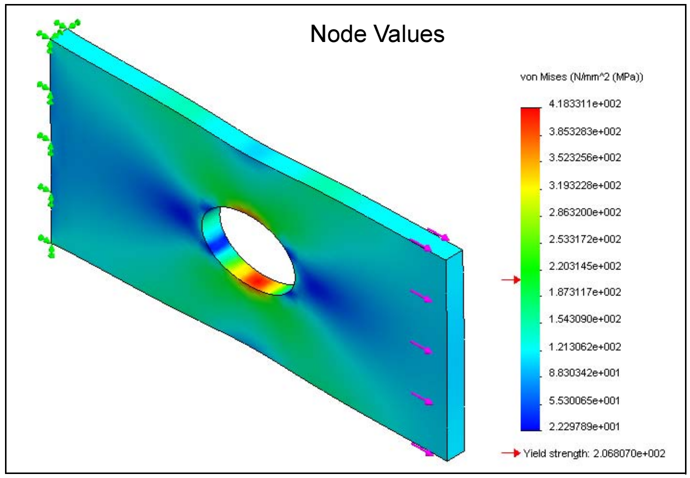
The **Show as tensor plot** option lets users plot the orientation as well as the magnitudes of the principle stresses (shown in the discussion below).

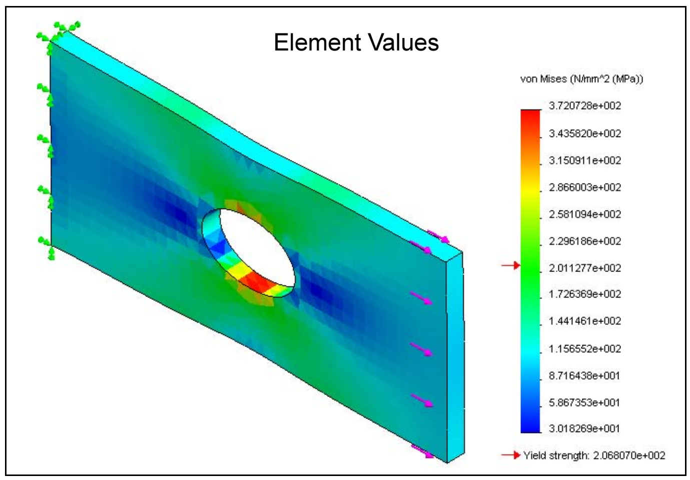
The **Deformed Shape** dialog lets the user specify the deformation scale for the plot. **Automatic** (default), **True scale**, and **User Defined** scale options are available.

Students are encouraged to experiment with these options.

Nodal vs. Element Stresses

The following figures show the nodal and elemental values of the Von Mises stress for our model.





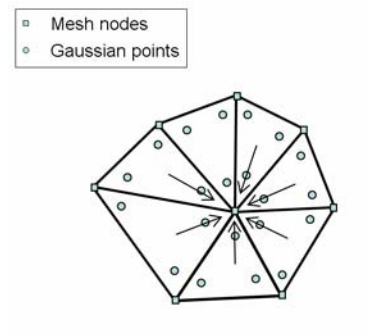
The stress plot that displays **Nodal values** appears "smooth", while the stress plot that displays **Element values** appears "rough".

To understand the reasons for these different appearances, we need to explain the differences between nodal and element stresses.

During the solution process, in each element, stress results are calculated at certain locations called Gauss points. First order tetrahedral elements (draft quality) have one Gauss point in their volume. Second order tetrahedral elements have four Gauss points. First order shell elements have one Gauss point. Second order shell elements have three Gauss points.

Nodal Values

Stresses in Gauss points can be extrapolated to element nodes. Most often, one node is shared by several elements, and each element reports different stresses at the shared node. Reported values from all adjacent elements are then averaged to obtain a single value. This method of stress averaging produces averaged (or nodal) stress results.



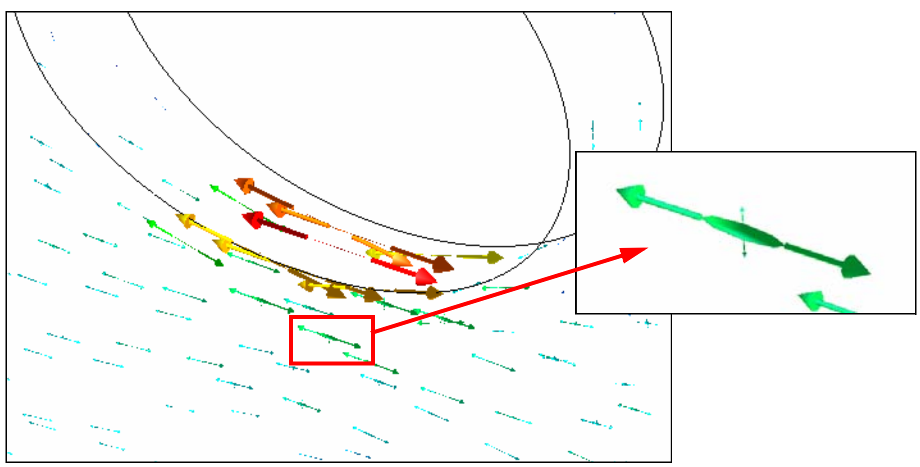
Element Values

Alternately, the stress values from all Gauss points within each element can be averaged to report a single elemental stress. Although these stresses are averaged between Gauss points, they are called non-averaged stresses (or element stresses) because the averaging is done internally within the same element only.

Element stresses and nodal stresses are always different, but too large a difference indicates that the mesh is not sufficiently refined in that location. See the exercise *Exercise 1: Bracket* on page 71 for the practical use of these quantities.

Show as Tensor Plot Option

This plot type helps visualize the directions as well as the magnitudes of the principal stresses P1, P2, and P3. Due to the considerable differences in magnitudes between these stress values, one must zoom in substantially to see all three arrows.



Modifying Result Plots

The Results plots can be modified in several ways to suit your needs. There are three primary functions to control the content, units, display and annotations of the plots.

- **Edit Definition**
  Edit Definition controls the component (von Mises, 1st principal stress, X normal stress) and units to be displayed.
- **Chart Options**
  Chart Options control the annotations. Options include which annotations are shown as well as the color, type of units (scientific, floating, general) and the number of decimal places shown in the legend. The position of the legend and titles can also be adjusted.
- **Settings**
  Settings are used to control the display of the model.

Where to Find It

- Right-click a plot and select either **Edit Definition**, **Chart Options** or **Settings**.

20 Modify the chart.

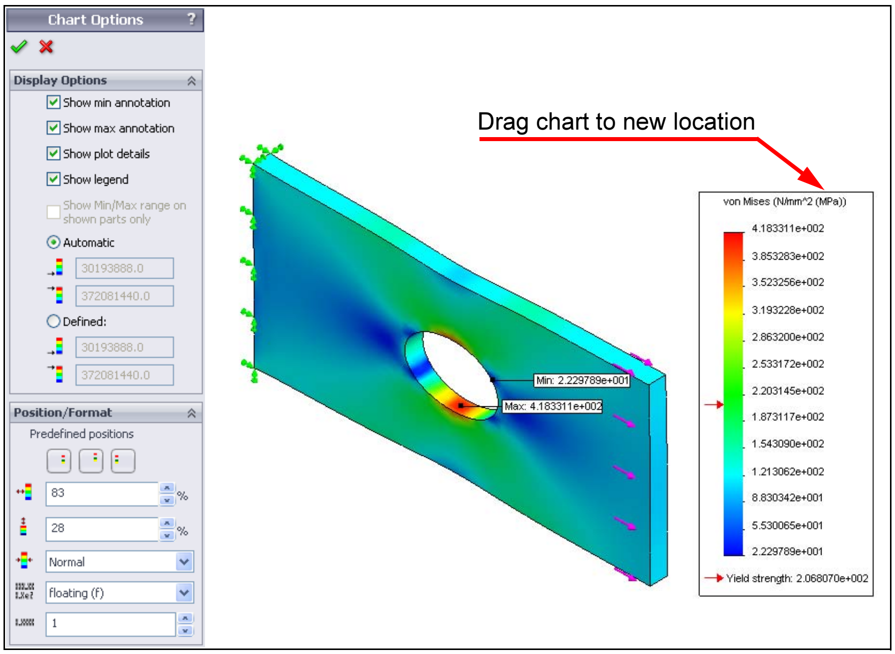
Right-click `Stress1 (-vonMises-)` and select **Chart Options**.

Check **Show min annotation** and **Show max annotation** boxes to show the markers in the plot.

Note that you can also modify the limiting values in the chart, format of the numbers, and the color options.

If you select the chart, it will be framed. You can then drag the chart to any location on the plot.

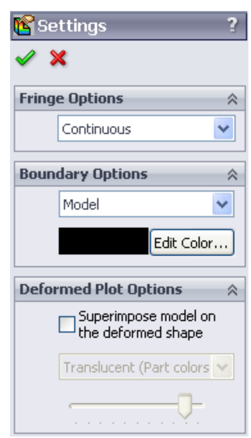
Click **OK** to save new settings.



21 Modify settings of stress plot.

Right-click on `Stress1 (-vonMises-)` and select **Settings**.

Explore the **Fringe**, **Boundary**, and **Deformed Plot Options** in this dialog.



Other Plot Controls

There are several other plot types available to display specific results of the analysis.

Introducing: Section Plot

Sections plots allow a cutting plane to be positioned at any point in the model and the plotted results shown at the plane location.

Where to Find It

- Right-click an existing plot and select **Section Clipping**.

Introducing: Iso Plots

Iso plots show that part of a model where the plotted parameter is a certain value or between certain values.

Where to Find It

- Right-click an existing plot and select **Iso Clipping**.

Introducing: Probe

A probe allows you to select a point or points on the model and display the plot parameter in both tabular and plotted form.

Where to Find It

- Right-click a plot and select **Probe**.

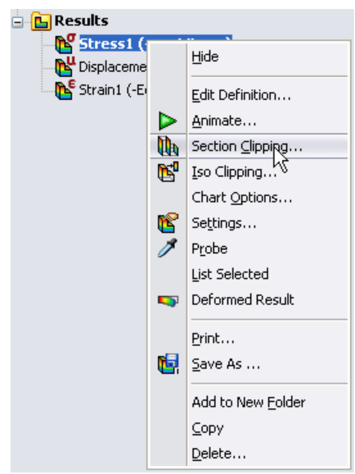
---

22 Create section plot.

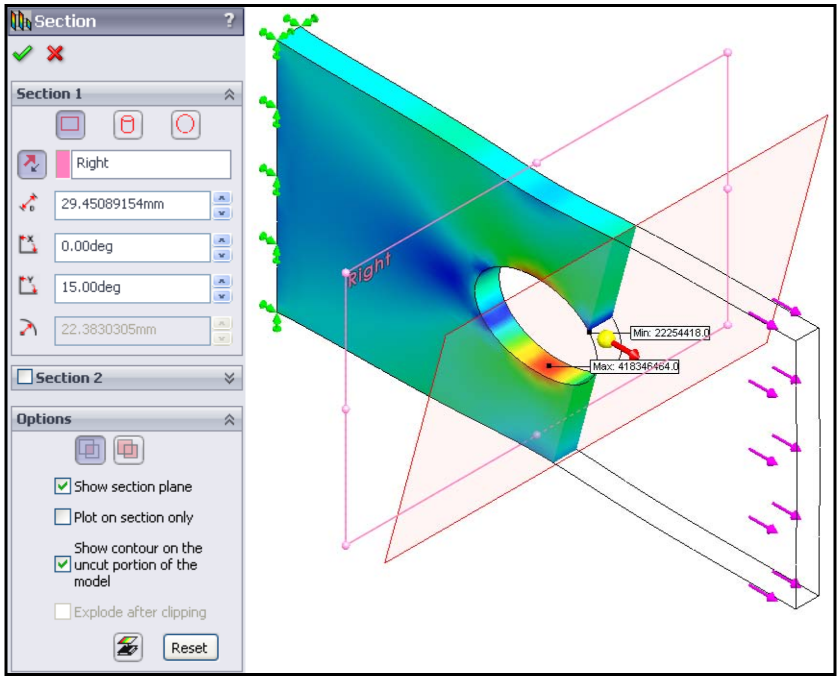
In many applications it is useful to cut the model and look at the distribution of the result quantity in the through-thickness direction.

Right-click `Stress1` (-vonMises-) and select **Section Clipping**.

From the SolidWorks fly-out menu, select `Right` plane as a **Reference entity**.



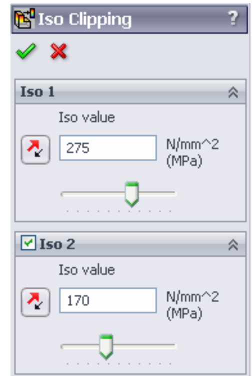
Students are encouraged to explore all the options and parameters in the **Section** dialog. Note that the user can also drag the triad to easily move the cut plane through the model.



Use **Reverse Clipping Direction** and **Clipping On/Off** to control the cutting direction and to disable the section plot.

Click **OK** to close the **Section** dialog.

23 Create Iso plot.



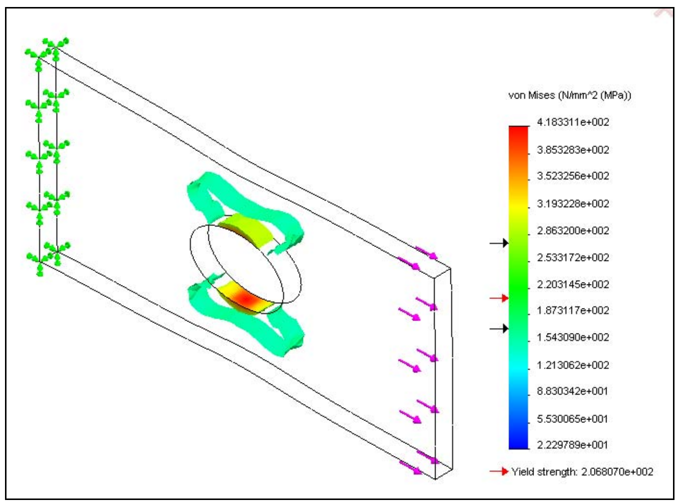
Suppose that we wish to display portions of the model where the von Mises stress is between 170 MPa and 275 MPa.

Right-click on `Stress1 (-vonMises-)` and select **Iso Clipping**. This opens the **Iso Clipping** PropertyManager.

In the **Isovalue** box, under the **Iso1** dialog, enter **275 N/mm^2 [MPa]** [39,886 psi].

Check **Iso 2** and in the **Isovalue** box, enter **170 N/mm^2 [MPa]** [24,657 psi].

Click **OK**.



The black arrows on the stress legend indicate the values defined for the two iso surfaces.

Experiment with the **Iso Clipping** window options using different numbers of iso surfaces and different cutting directions.

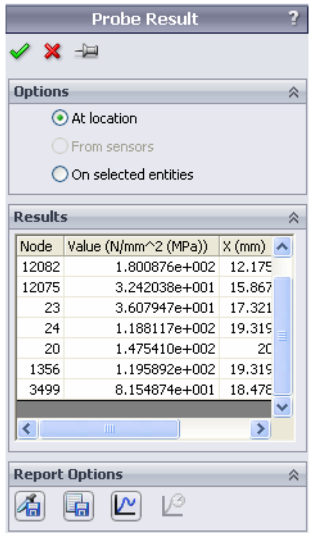
Use **Reverse Clipping Direction** and **Clipping On/Off** to control the cutting direction and to reset the plot.

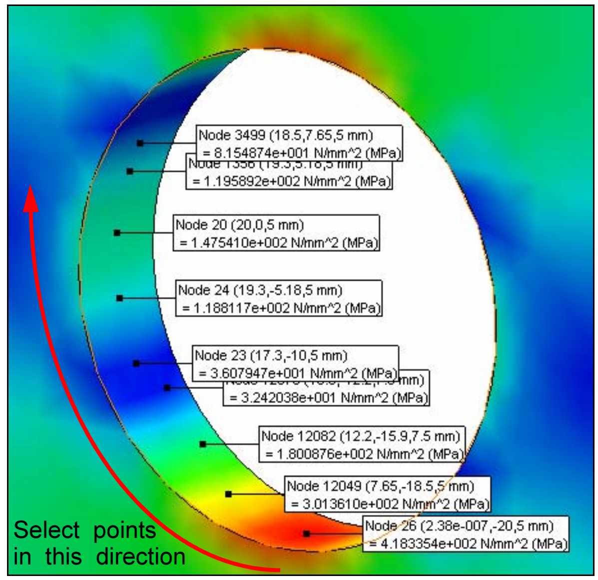
24 Probe stress results.

Right-click on `Stress1 (-vonMises-)` and select **Probe**.

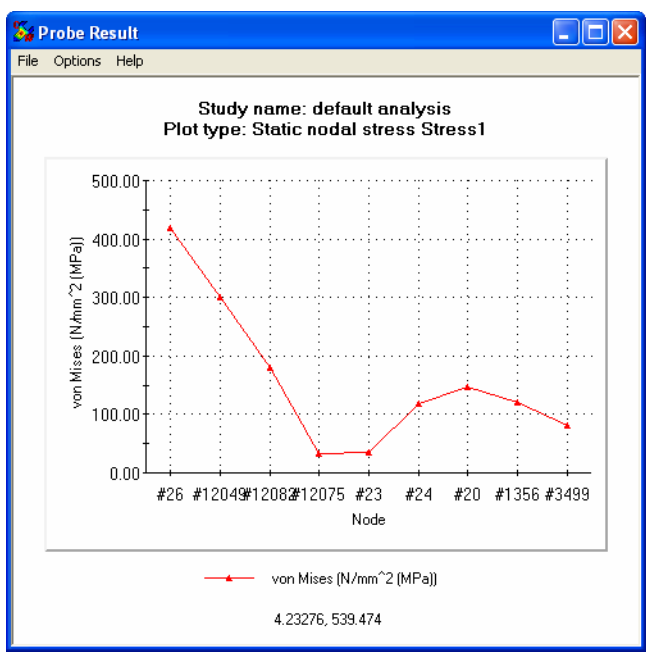
Using the pointer, click the desired locations on the plot. It helps to zoom in on the area.

The stress results are listed in the **Results** dialog table and in the plot at the selected locations.





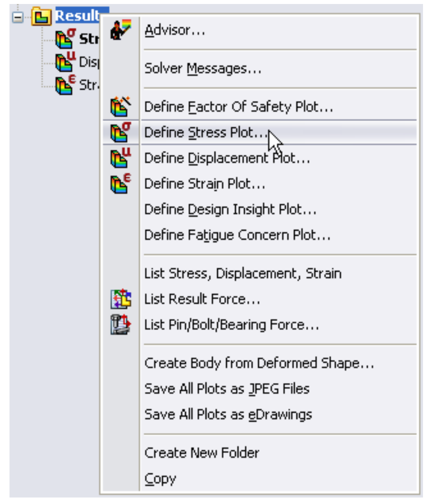
Under **Report Option**, you can save the results in a file, plot the path-graph, or save the locations as sensors. (Sensors are discussed in detail later on in the class.)



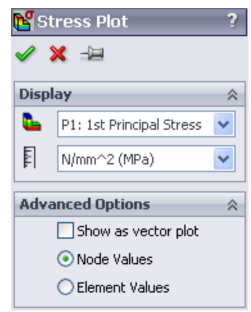
The figure above shows a Von Mises stress path plot for the selected locations.

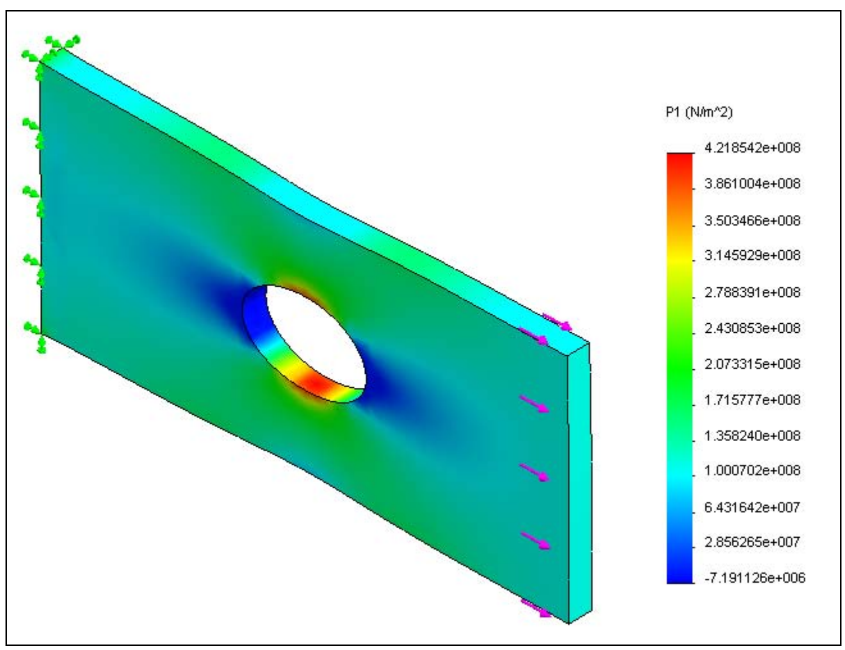
25 Define P1: 1st Principle Stress plot.

Define a new stress plot. Right-click the Results folder and select **Define Stress Plot**.



Select **P1: 1st Principle Stress** as the stress component, keep all other default options, and click **OK**.



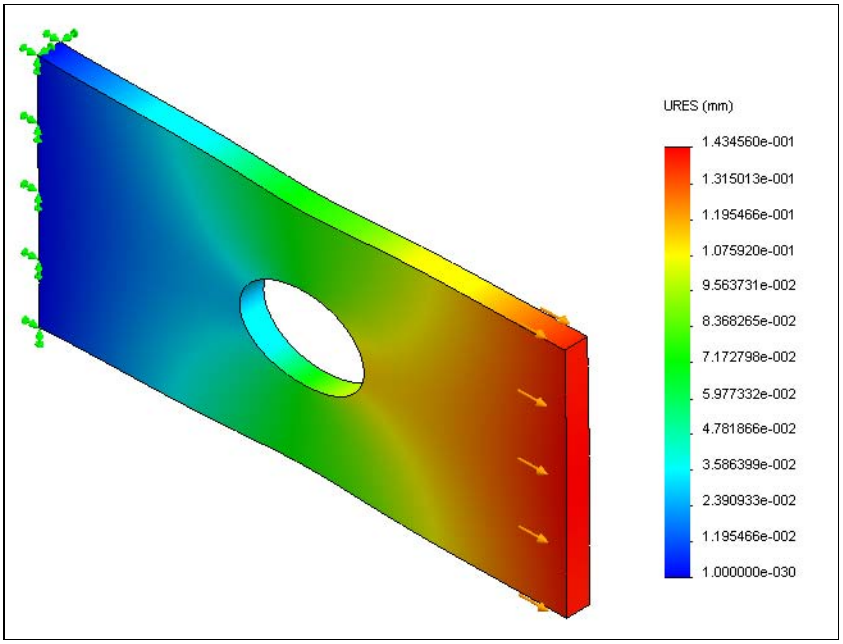


We observe that the maximum value of the 1st principle stress, 421 MPa [61,062 psi], is very close to the maximum value of the Von Mises stress, 418 Mpa [60,627 psi]. This is because the specified Tensile load is the only dominant load component resulting in predominantly tensile stress along the longitudinal direction of the plate.

26 Define displacement plot.
Double-click the `Displacement1 (-Res disp-)` plot icon.

The post processing features that we practiced in the case of `Stress1 (-vonMises-)` are applicable to all other result quantities, such as Displacement.

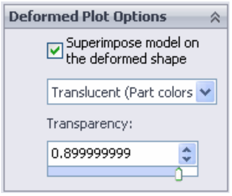
The displacement shows a maximum resultant displacement of 0.1435 mm [0.00565 in].



Note

We record the displacement result with 6 digits only to practice the plot options and to compare results from studies with different meshes. The uncertainties and simplifying assumptions used to create the model do not justify this accuracy.

27 Superimpose undeformed shape.
Right-click on `Displacement1(-Res disp-)` and select **Settings**.

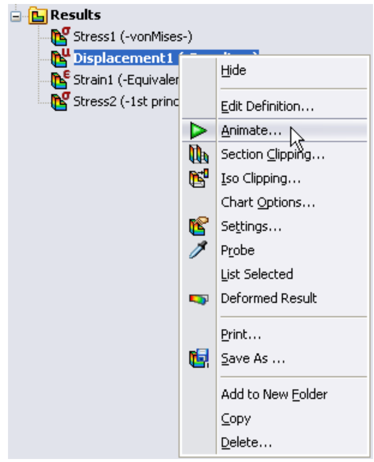


Select **Superimpose model on the deformed shape**. You can also adjust the transparency of the undeformed image.

Click **OK**.

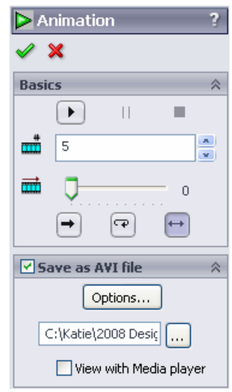
28 Animate displacement plot.

To animate the displacement plot, right-click on `Displacement1 (-Res disp-)` and select **Animate**.



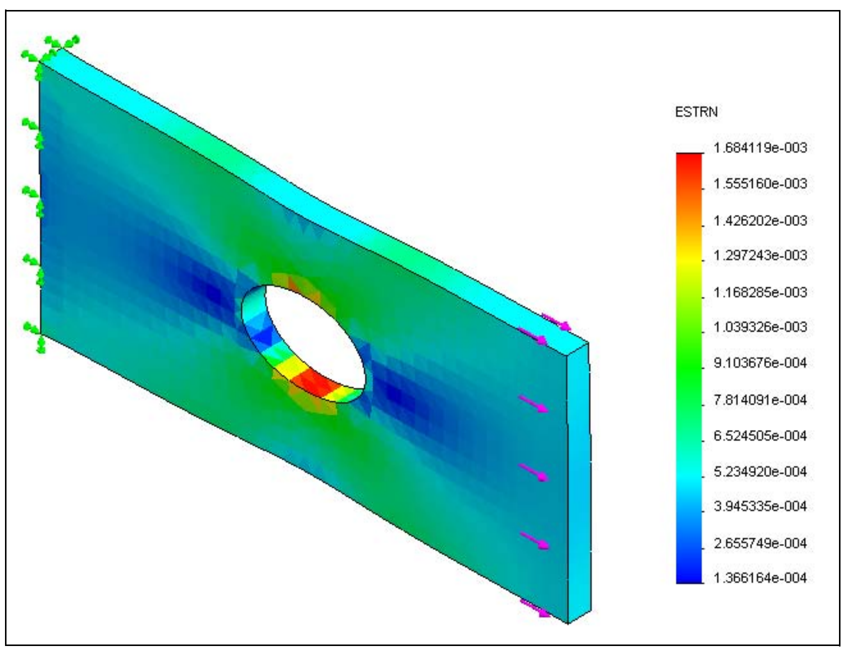
In the **Animation** PropertyManager you can start and stop the animation, set the number of frames, control the speed, and save the animation as an `*.avi` file.

Try the options of the animation feature.



29 Plot strain results.

Double-click the `Strain1 (-Equivalent-)` plot icon to show the plot.



Note that strain results are dimensionless.

Strain results are shown as non-averaged (element values) by default as opposed to stress results, which are shown as averaged (node values) by default.

Examine the strain plot showing **Element Values**.

To review the averaged strain plot, right-click on `Strain1 (-Equivalent-)` and select **Edit Definition**, and then select **Node Values**.
To examine the available chart options, right-click `Strain1 (-Equivalent-)` and select **Edit Definition**.
All post processing features that we practiced for the stress plot are available for strain plots as well.

Other Plots

There are several other postprocessing quantities available to view at the end of the analysis.

Introducing: Factor of Safety Plot

Factor of Safety Plot show the safety of the design based on the design strength of the material (typically the yield strength). This plot is fully introduced in *Lesson 7*.

Where to Find It

- Right-click the `Results` folder and select **Define Factor of Safety Plot**.

Introducing: Fatigue Check Plot

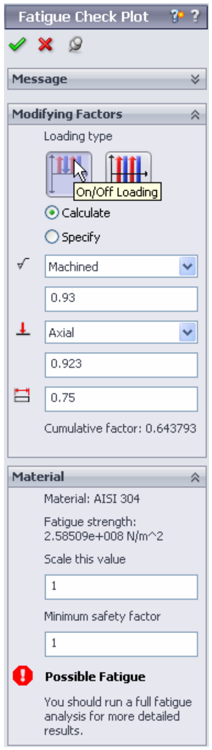
Fatigue Check Plot serves as a quick indicator if the fatigue may be of any concern in the design of the component.

Where to Find It

- Right-click the `Results` folder and select **Define Fatigue Check Plot**.

30 Plot Fatigue Check Plot.

Right-click on the Results folder and select **Define Fatigue Check Plot**.

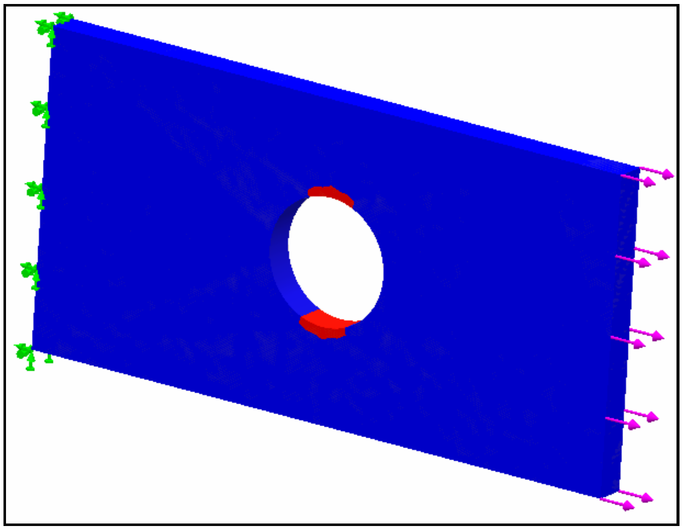


Set the **Loading type** to **On/Off Loading** to indicate that the Tensile force may oscillate between 0 and 110,000 N.

Set the **Surface Finish Factor** to **Machined**. Keep the **Loading Factor** and **Size Factor** at their default values of **Axial** and **0.75**.

Under **Material** keep the **Scale this value** and **Minimum safety factor** fields at their default values of **1**.

Click **OK**.



The areas in red indicate potential fatigue problems. Note that accurate calculations using the SolidWorks Simulation Professional fatigue modulus may be required.

Multiple Studies

We have completed the analysis of rectangular hollow plate with a coarse mesh and now wish to see how a change in mesh density affects the results. For this reason, we will repeat the analysis two more times using both coarser and finer density meshes.

To repeat the analysis with coarsened mesh, we can create a new mesh while still in the `default analysis` study, but this action would overwrite the old results.

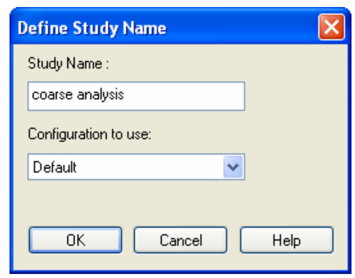
To preserve the results of the study, we will create a new study, `coarse analysis`. Creating a new study can be done in several ways.

Creating New Studies

New studies can be created in one of two ways:

- Create a new study from scratch.
- Duplicate an existing study. Right-click the tab for the study you want to duplicate and click Duplicate. This is essentially the same as copying a study and pasting it into a blank study.

When we duplicate a study, SolidWorks Simulation displays the **Define Study Name** window. This will allow us to name the duplicated study and choose the model configuration to use.



Copy Parameters

When we create a new study, we can copy material, fixtures and external forces from existing studies rather than recreating them in the new study. To copy parameters, drag the parameter from the Simulation Study tree to the tab of the new study.

Note

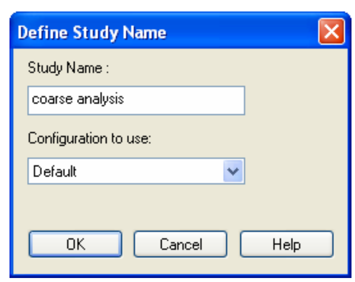
When a study is duplicated, the study settings, Fixtures, External Forces, Mesh, and the study results will be copied as well.

---

31 Duplicate the study.

Right-click the default analysis tab and click **Duplicate**.

Type `coarse analysis` for the study name. The model only has a `Default` configuration, so we cannot change it.



32 Create new mesh in `coarse analysis` study.

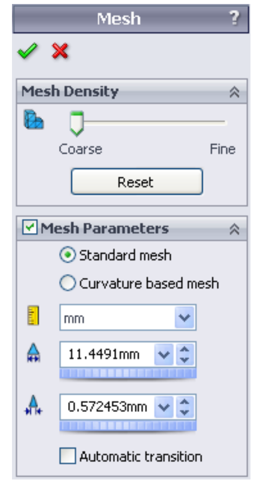
In the `coarse analysis` study, right-click `Mesh` and select **Create Mesh**. A warning window appears.

`Remeshing will delete the results for study: coarse analysis.`

Click **OK** to open the **Mesh** window.

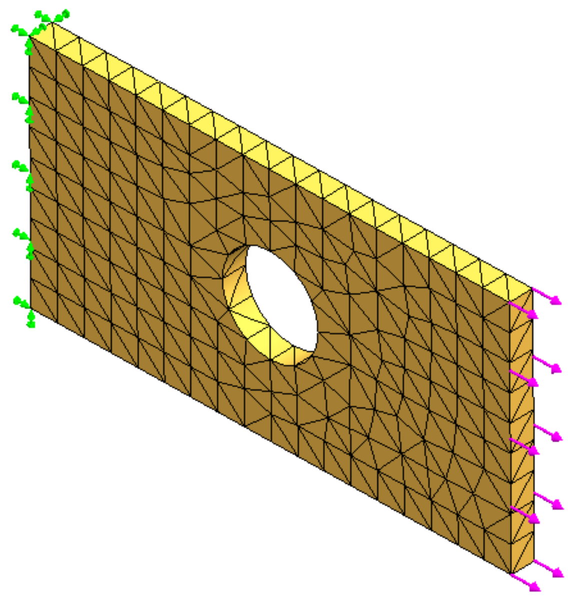
Move the **Mesh Factor** slider all the way to the left. The **Element size** and **Tolerance** should read **11.4491 mm** [0.4508 in] and **0.572453 mm**, [0.0225 in] respectively.

Click **OK**.



The generated mesh is displayed to the right.

Notice that there is only one element across the thickness of the part. In the default analysis there were two elements across the thickness.



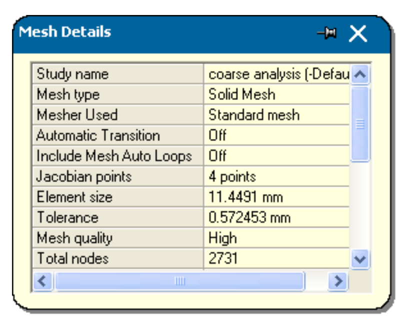
Note

The mesh is rather coarse. Later, we will discuss why this sort of mesh is not acceptable for reliable analysis results.

33 Display mesh details.

Having created the mesh, we can access the detailed mesh information by right-clicking `Mesh` and selecting **Details**.

The same detailed information can of course be displayed for the "old" mesh in the `default analysis` study.



Mesh Details

Study name	coarse analysis (-Defau
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Element size	11.4491 mm
Tolerance	0.572453 mm
Mesh quality	High
Total nodes	2731

Many of the items in this list will be discussed in later lessons.

34 Run the analysis.

35 View displacement and stress results.
Record the maximum displacement (0.143 mm / 0.00563 in) and the maximum von Mises stress (388 Mpa / 56,276 psi).

Note

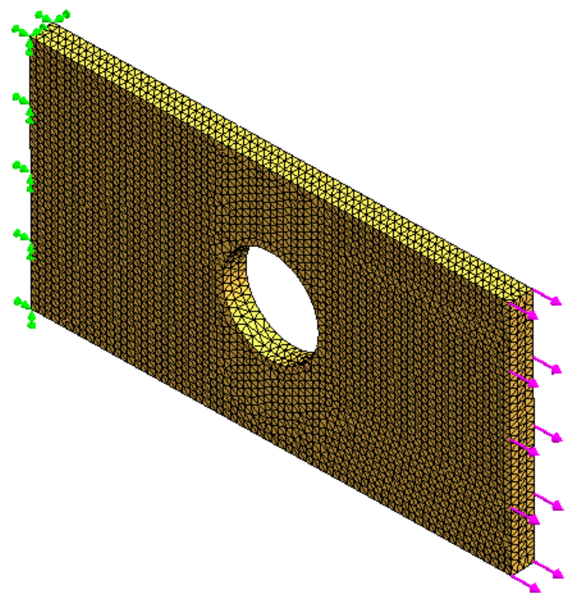
All plot settings remain the same as the `default analysis` study because the plot definitions are copied from that study.

36 Re-run the analysis with fine mesh.
Repeat steps **31** - **34** to generate a new study with fine mesh named `fine analysis`.

When re-generating the mesh, move the slider all the way to the right. The **Element size** and **Tolerance** should read **2.86227 mm** [0.1127 in] and **0.143113 mm** [0.00563 in], respectively.

The fine mesh generated using the above settings is shown to the right.

Notice that we now have three elements in the through-thickness direction. You will later learn that this mesh is acceptable for reliable analysis results.



37 View displacement and stress results.
Record the maximum displacement (0.143 mm / 0.00653 in) and the maximum von Mises stress (416 Mpa / 60,047 psi).

Check Convergence and Accuracy

Now we must collect information from all of the studies (`default`, `coarse` and `fine analysis`) to compare the displacement and maximum von Mises stress results for the various mesh refinements. We can determine the maximum displacement and the maximum von Mises stress results in plots.

We must also determine the number of elements and the number of nodes in each mesh. These can be found in the **Mesh Details** window of each respective mesh.

Reports

Results may need to be recorded in report form for review, presentation or archive purposes.

Reports can be published in either HTML or Microsoft Word formats using any of three pre-defined styles.

Different sections can be added to the report from a list of predefined commonly used topics.

Predefined sections include:

- Cover Page
- Assumptions
- Study Properties
- Material Properties
- Connector Definitions
- Mesh Information
- Sensor Results
- Free-Body Forces
- Pin Forces
- Study Results
- Description
- Model Information
- Units
- Loads and Restraints
- Contact
- Design Scenario Results
- Reaction Forces
- Bolt Forces
- Beams
- Conclusion

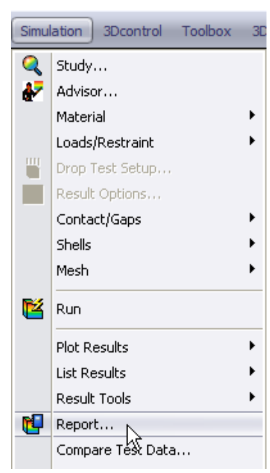
To edit the content of a section, select the section in the Included sections and fill in the appropriate section properties.

Where to Find It

- Click **Reports** in the **Simulation** menu.
- Click **Report** on the Simulation toolbar.
- Click **Report** on the Simulation tab of the CommandManager.

---

39 Generate report in Microsoft Word format.
Under **SolidWorks Simulation** menu item, select **Report.**

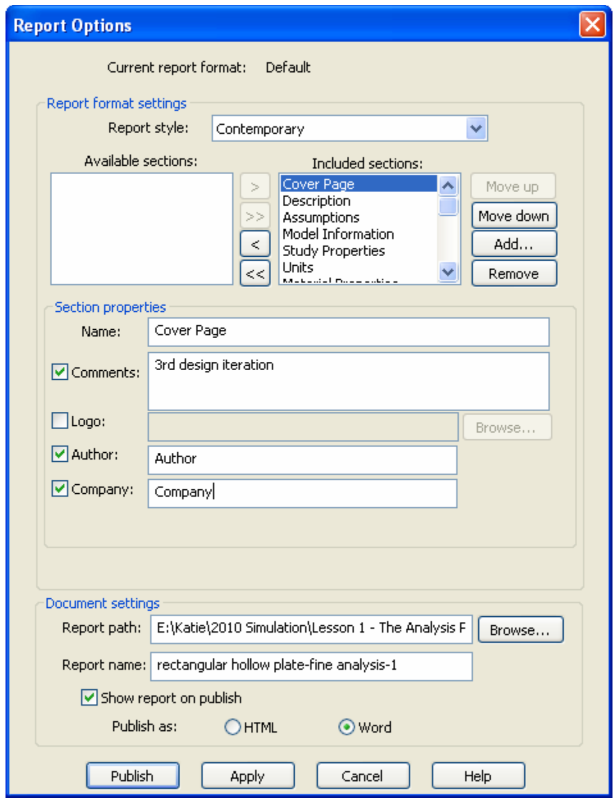


40 Add sections.

Under **Included sections**, select the required report parts. (For example, deselect the option **Contact**, as we do not have any in this analysis.)

Use **Move up** and **Move down** to arrange the sections in the order you want to include them in the report. When you select each section to be included, you can add information under **Section Properties**.

Select **Word** under **Document settings**, enter the **Report name** and click **Publish**.



41 Examine the report.

Open the report in Microsoft Word and examine the results.

42 Save and Close the file.

Summary

We used a simple model of a hollow rectangular plate to introduce the SolidWorks Simulation interface and, at the same time, to go through all major steps in the FEA process.

We created multiple studies to execute a linear static analysis with three different meshes.

While preparing models for analysis and examining results obtained with different meshes, we introduced the concept of modeling error and discretization error.

This first lesson was intended to provide an understanding of FEA methodology and the software skills necessary to complete the lessons that follow.

Questions

- The pre-processing stage of the FEA includes the following steps:
  1.____
  2.____
  3.____
  4.____
  5.____
- The density of finite element mesh (does / does not) have considerable impact on the analysis results.
- In general, we would favor (finer / coarser) meshes to obtain reliable analysis results. Therefore, the time required to solve the analysis will (increase / decrease), but this is an unavoidable consequence.

  Ultimately, we will try to design optimum meshes providing reasonable accuracy levels and resulting in acceptable run times.
- The primary unknown in finite element analysis is (displacements / strains / stresses). This quantity is therefore the most accurate.
- The accuracy levels of (displacements / strains / stresses) and (displacements / strains / stresses) are approximately the same, but significantly worse than that of (displacements / strains / stresses). Therefore, to obtain good (displacement / strain / stress) results, the mesh must be reasonably fine.
- (Refining / Coarsening) the mesh results in solutions approaching the analytical solution of a mathematical model.

Exercise 2: Compressive Spring Stiffness

In this exercise, we will use SolidWorks Simulation to determine the compressive stiffness of a coil spring.

This exercise reinforces the following skills:

- *New Study* on page 31.
- *Fixtures* on page 34.
- *External Loads* on page 38.
- *Meshing* on page 43.
- *Result Plots* on page 46.

Procedure

The stiffness of the helical spring can be determined as follows:

1 **Open a part file.**
Open `spring` from `Lesson01\Exercises` folder.

Note

For convenient application of fixtures and external loads, disks have been added to both ends of the spring. The distance between the disks corresponds to the active length of the un-compressed spring.

2 **Set SolidWorks Simulation options.**
Set the system of **Units** to **SI (MKS)** and the units of **Length** and **Stress** to **mm** and **N/m^2 (Pa)**.

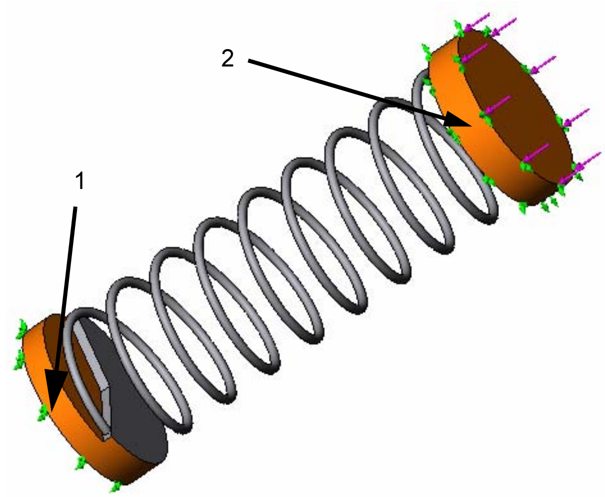
3 **Create study.**
Create a **Static** study named `spring stiffness`.

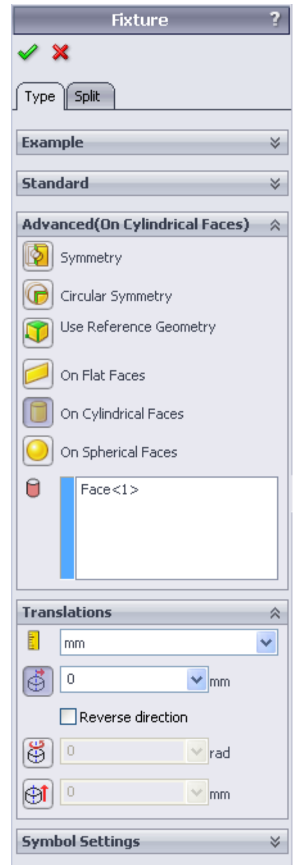
4 **Review material properties.**
The material properties (**Alloy Steel**) are transferred from SolidWorks.

5 **Apply Fixed restraint.**
Apply a **Fixed Geometry** fixture to the end face of one disk (item 1).

6 Apply radial restraint.
Use an advanced fixture to apply a restraint in the radial direction to the cylindrical face of the other disk (item 2).

This restraint only allows the spring to be compressed (or expanded) in its axial direction and to rotate about the longitudinal axis.





7 Apply compressive force.
Apply a **0.1 N** compressive force to the end face of the disk with the cylindrical face constrained in the radial direction.

8 Mesh the model and run the analysis.
Use **High** quality elements with the default **Element size** and **Tolerance** of **1.394 mm** and **0.07 mm**, respectively.

Lesson 2
Mesh Controls, Stress Concentrations and Boundary Conditions

Objectives

Upon successful completion of this lesson, you will be able to:

- Illustrate the differences between modeling and discretization errors.
- Use Automatic transition option to mesh models.
- Use mesh controls.
- Describe when the lack of convergence of FEA results may occur.
- Understand stress concentrations.
- Analyze model in different SolidWorks configurations.
- Run multiple studies in a batch mode.
- Extract reaction forces.

Mesh Control

Meshes are rarely uniform in practical problems. It would be very inefficient to uniformly reduce the mesh size in a large model because of a local stress concentration. We would create large number of elements in areas of uniform or slowly changing stress resulting in an increase of computational time that in the end tells us little about the model.

Using different methods to control the mesh, we can use a small mesh in areas of rapid changing stress and a large mesh in areas with little change.

Case Study: The L Bracket

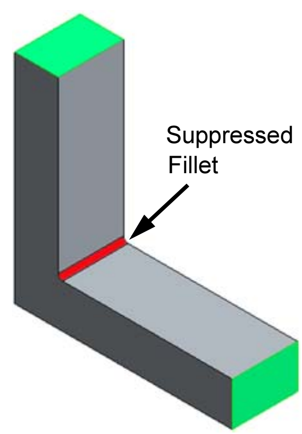
In this case study, we will determine the stress in an L bracket, under load. The L bracket presents the problem of stress at sharp corners and the effects of fillets and local mesh refinement.

The corner of the bracket is rounded by a small fillet. Since the radius of the fillet is small compared to the overall size of the model, it may be suppressed. We will solve the model with and without fillet, discuss the differences and the applicability of each approach.

We will also investigate the effect of different mesh sizes on the maximum displacement and stress results. Rather than refining the mesh uniformly in the entire model, which is called *global mesh refinement*, we refine the mesh locally, where high stresses are located. This is called *local mesh refinement*.

Project Description

An L-shaped steel bracket is fixed at the top and a 900 N load is applied to the lower end face. We will evaluate the displacements and stresses in the model.



Stages in the Process

Some key stages in the analysis of this part are shown in the following list:

- **No fillet**
  The fillet will be suppressed to simplify the geometry and to observe the stress at the sharp corner.
- **Add fillet**
  The fillet will be unsuppressed to determine the effect of the fillet on the maximum stress in this part.

- **Mesh refinement**
  As the fillet is small compared to the rest of the model, we will use different techniques to reduce the mesh size only in the area of the fillet.

Procedure

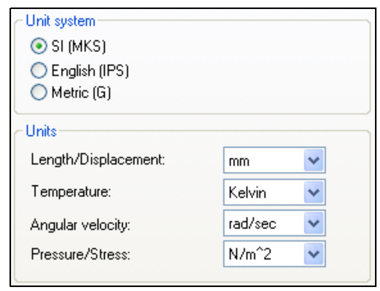
In the first part of this case study, we will examine the stress on this part without the fillet.

1 Open a part file.
Open `L bracket`.from `Lesson02\Case Studies` folder.

In the SolidWorks ConfigurationManager, examine the two configurations: `fillet` and `no fillet`.

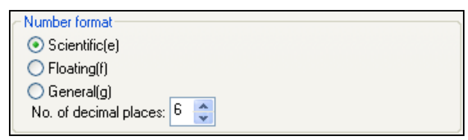
Make the `no fillet` configuration active.

2 Set the simulation options.
Click **Options** from the **Simulation** menu. Select the **Default Options** tab.



Select **Units**, then select **SI (MKS)** for the Unit system. Select **mm** for **Length/ Displacement** and **N/m^2** (Pascals) for **Pressure/Stress**.

Select **Color Chart**. For **Number format**, select **Scientific (e)** and **6** decimal places.

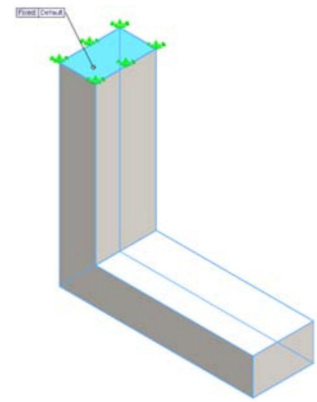


3 Define static study.
Create a new study named `mesh1`.

In the analysis **Type** list, select **Static**.

Click **OK**.

4 Examine the Simulation Study tree.
The `L bracket` icon already has a check mark next to the name of the assigned material because the material definition (AISI 304 steel) has been transferred from SolidWorks. Also, note that a sharp re-entrant corner takes the place of the suppressed fillet.



5 Apply a fixture.

Now apply a **Fixed Geometry** fixture to the top face of the `L bracket`.

Right-click `Fixtures` and select **Fixed Geometry**.

In the **Standard** list select **Fixed Geometry**.

Click **OK**.

6 Apply an external load.

Right-click `External Loads` and select **Force**.

Select **Force**.

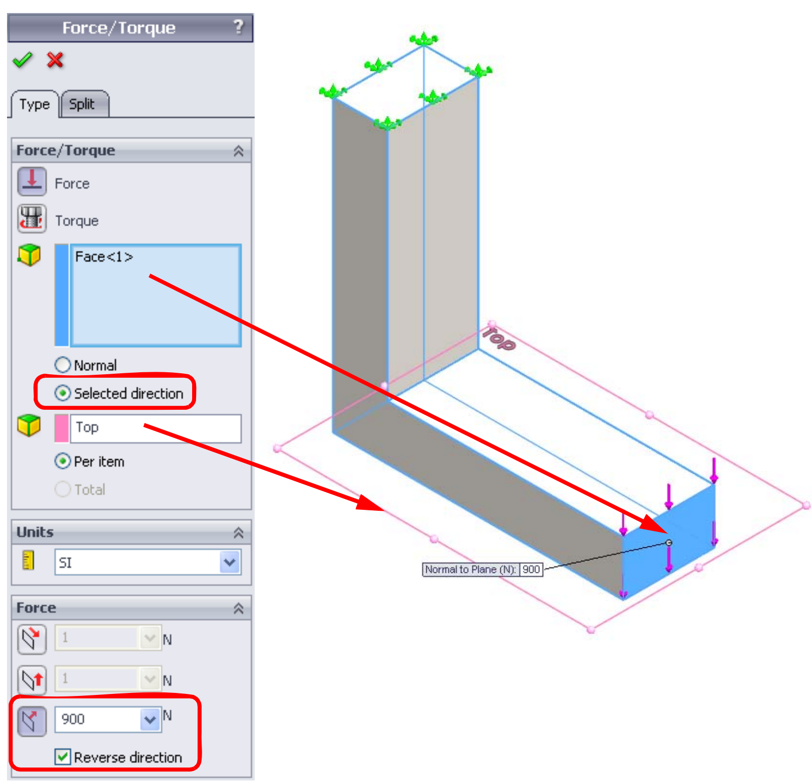
We want to apply a shearing force and not a normal force, so we must define the direction of the force. Select **Selected direction**.

Select the indicated face to apply the force and the `Top` plane to specify the direction.

Type `900` N [202.33 lbf] for the force.

Select **Reverse direction** to make sure the force is pointing as shown.

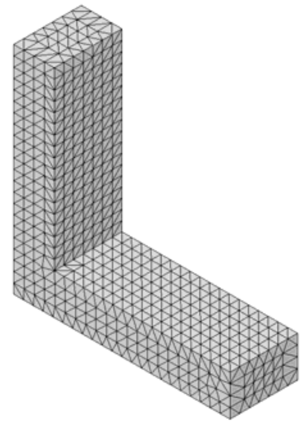
Click **OK**.



7 Mesh the model.

Verify that the meshing option is set to **High** quality (Draft Quality Mesh is cleared), meaning that second order elements are created.

Mesh the model using the default **Element size** of **4.812 mm [0.1894 in]**.

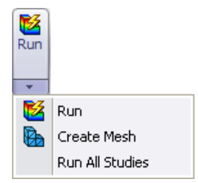


Run All Studies

Multiple studies can be run at the same time. This allows you to setup multiple studies and then run them after hours.

Where to Find It

- Select the tab **Simulation** on the CommandManager and click the down arrow under **Run** and click **Run All Studies**.



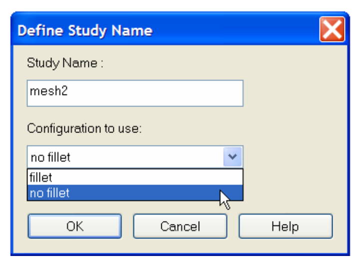
8 Create a duplicate study.

Study `mesh1` is now ready to be analyzed. However, we will create two more studies and run all three studies at the same time using the **Run All Studies** command.

Duplicate study `mesh1` into a new study `mesh2` (see see *Creating New Studies* on page 61 on how to duplicate a study).

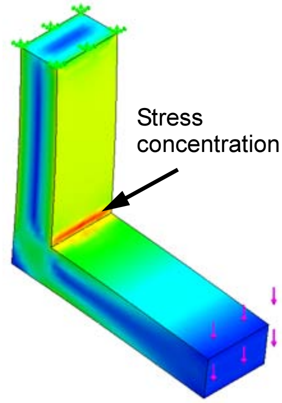
When creating the duplicate of the study make sure that the **Configuration to use** field says `no fillet`.

Click **OK**.



Analysis with Local Mesh Refinement

The second part of this case study will investigate the effect of using smaller elements in the model on the results. In Lesson 1, we refined the mesh uniformly throughout the entire model by controlling the global element size.



In this part of the case study, we will use a different technique. Note that a stress concentration is located near the sharp re-entrant corner.

Knowing the location of high stress, we can refine the mesh locally in that area by applying local mesh controls.

Mesh Control

Mesh controls allow you to control the element size locally on selected entities independent of the global element size. As compared to global mesh refinement, this is a more numerically efficient technique. Small elements are placed where needed, while portions of the model with no stress concentration are meshed with larger elements.

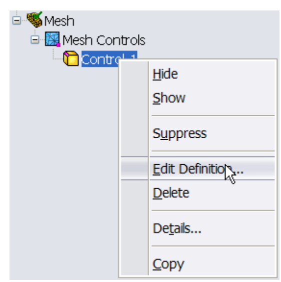
Where to Find It

- Right-click **Mesh** in the Simulation Study tree and select **Apply Mesh Control**.

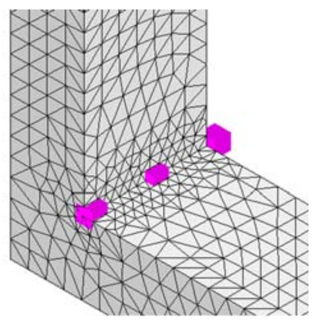
Mesh Controls

Mesh controls can also be applied to vertices, faces, or entire components of assemblies. Once mesh controls have been defined, the `Mesh` icon becomes a folder.

Mesh controls can be edited using a shortcut menu displayed by right-clicking `Control-1` and select **Edit Definition** in the `Mesh` folder, or directly by double-clicking on the `Control-1` item.



The mesh, with applied control (also called mesh bias), features localized refinement along the edges.



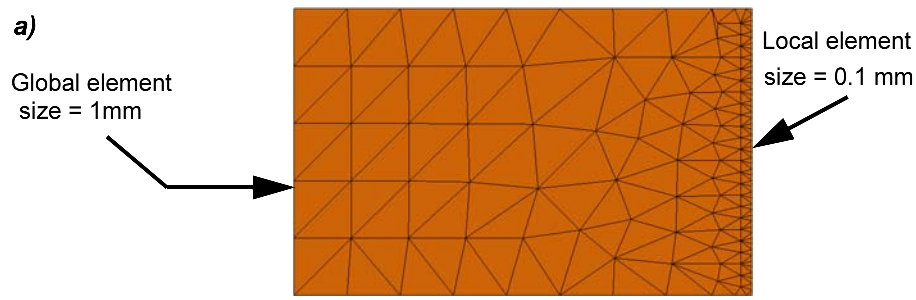
Mesh Control Symbols

Ratio

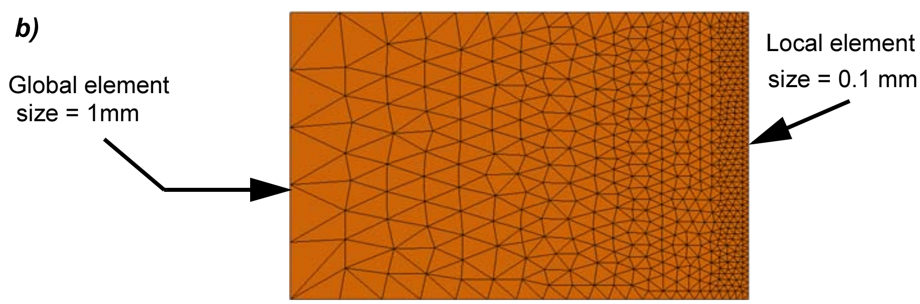
The ratio is used to refine the localized mesh.

The **Ratio** parameter specifies the ratio between element sizes in consecutive transitional element layers. In our case, the default **Ratio** of **1.5** was used.

The following shows the use of this option.



Ratio = 2.0



Ratio = 1.1

Meshing must be done after controls are defined.

Mesh control symbols are displayed along the affected edge.

Mesh control symbols

Mesh control symbols can be displayed or hidden by:

- Right-click `Mesh` and select **Hide All Control Symbols**
- Right-click `Mesh` and select **Show All Control Symbols**

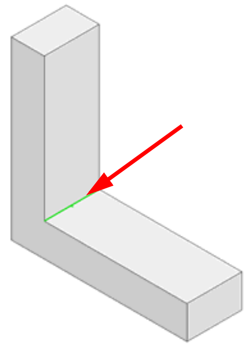
The visibility of mesh control symbols can also be controlled individually for each mesh control.

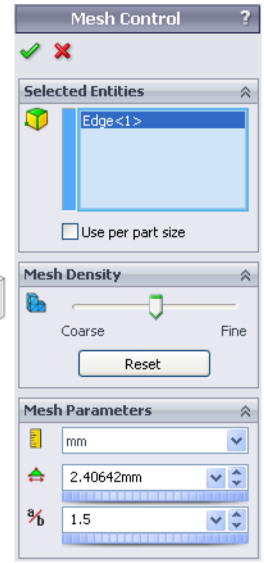
9 Apply local mesh control for study mesh2.
Select the edge shown.

Right-click Mesh and select **Apply Mesh Control**.

Use the suggested local **Element size** of **2.406 mm** and the **Ratio** of **1.5**.

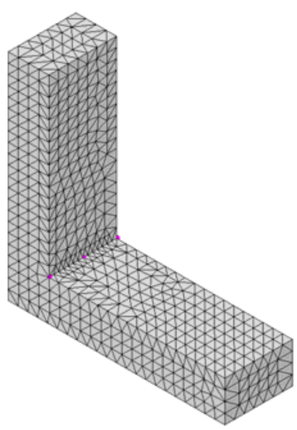
Click **OK** to close **Mesh controls** PropertyManager.



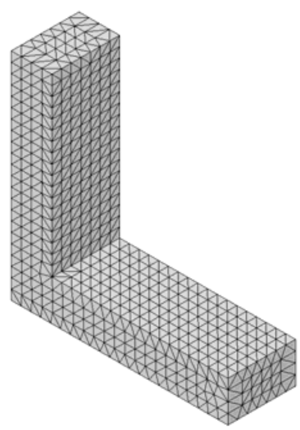


10 Create mesh.
Create **High** quality mesh with the default global **Element size** of **4.8128455 mm** and the default **Tolerance** of **0.24064227 mm**.

11 Examine the mesh.
Note that smaller elements have been created along the edge where mesh control has been just applied.



With edge mesh control



No edge mesh control

12 Duplicate study mesh2.
Name the new study **mesh3**.

13 Apply local mesh control for study mesh3.
In the mesh3 study, edit the definition of Control-1.

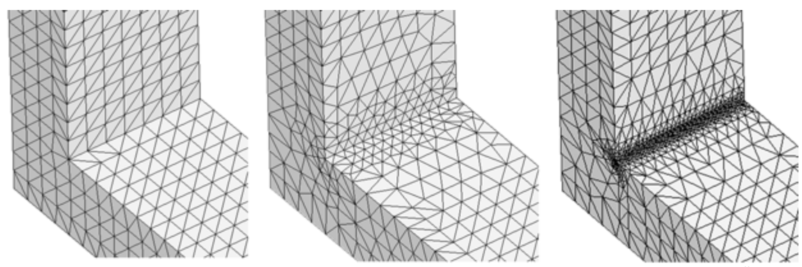
In the **Element size** box, enter **0.508 mm** to locally refine the mesh along the sharp re-entrant edge. Keep the **Ratio** at its default value of **1.5**.

With this mesh control, we will create very small elements along the sharp re-entrant edge.

Click **OK**.

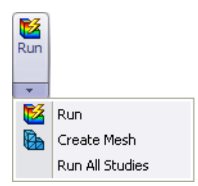
14 Mesh study mesh3.
Mesh study mesh3 with **High** quality elements and the default global **Element size** of **4.8128455 mm [0.1894 in]** and the default **Tolerance** of **0.24064227 mm [0.0095 in]**.

We now have three studies: mesh1, mesh2 and mesh3. The only difference is mesh refinement along the sharp re-entrant edge.

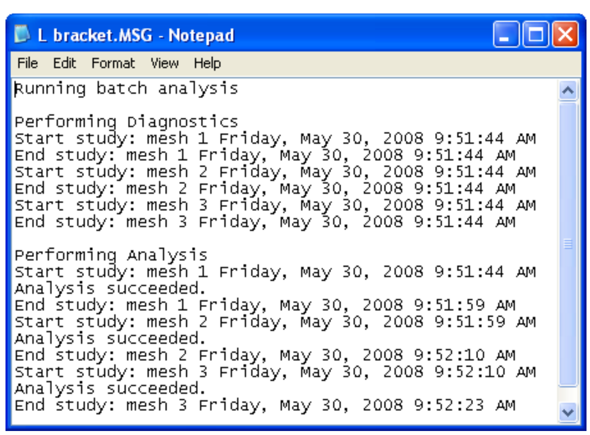


Study: mesh1 Study: mesh2 Study: mesh3

15 Run all studies.
Select the **Simulation** tab on the CommandManager. Select the down arrow under **Run Study** to flyout the other choices. Click **Run All Studies**.



16 Simulation progress log.
Once the analyses are completed, review the report in the MSG file located in the result folder.



L bracket.MSG - Notepad
File Edit Format View Help

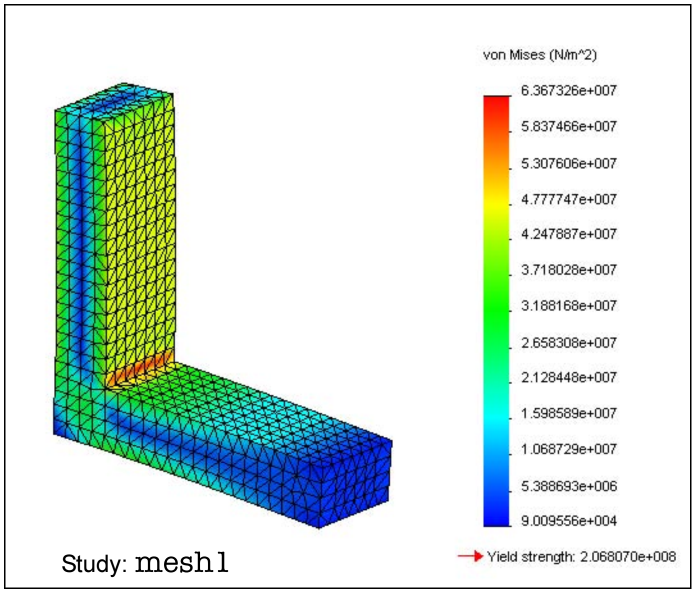
```
Running batch analysis

Performing Diagnostics
Start study: mesh 1 Friday, May 30, 2008 9:51:44 AM
End study: mesh 1 Friday, May 30, 2008 9:51:44 AM
Start study: mesh 2 Friday, May 30, 2008 9:51:44 AM
End study: mesh 2 Friday, May 30, 2008 9:51:44 AM
Start study: mesh 3 Friday, May 30, 2008 9:51:44 AM
End study: mesh 3 Friday, May 30, 2008 9:51:44 AM

Performing Analysis
Start study: mesh 1 Friday, May 30, 2008 9:51:44 AM
Analysis succeeded.
End study: mesh 1 Friday, May 30, 2008 9:51:59 AM
Start study: mesh 2 Friday, May 30, 2008 9:51:59 AM
Analysis succeeded.
End study: mesh 2 Friday, May 30, 2008 9:52:10 AM
Start study: mesh 3 Friday, May 30, 2008 9:52:10 AM
Analysis succeeded.
End study: mesh 3 Friday, May 30, 2008 9:52:23 AM
```

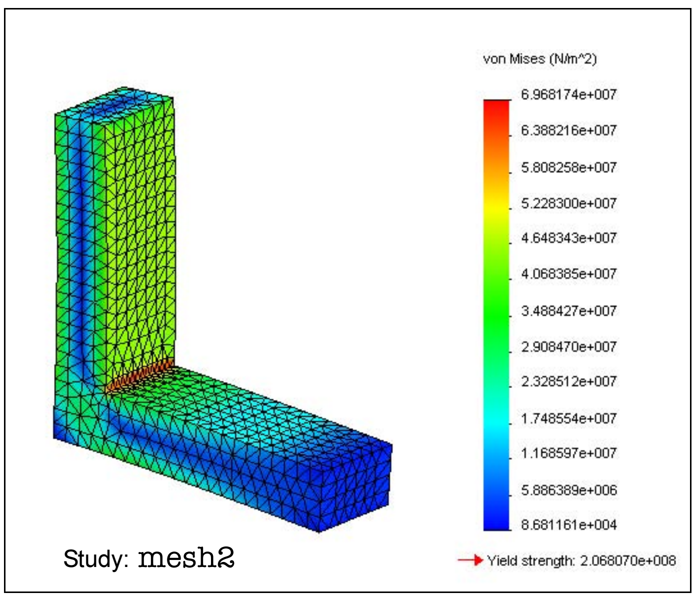
17 Plot von Mises stresses.

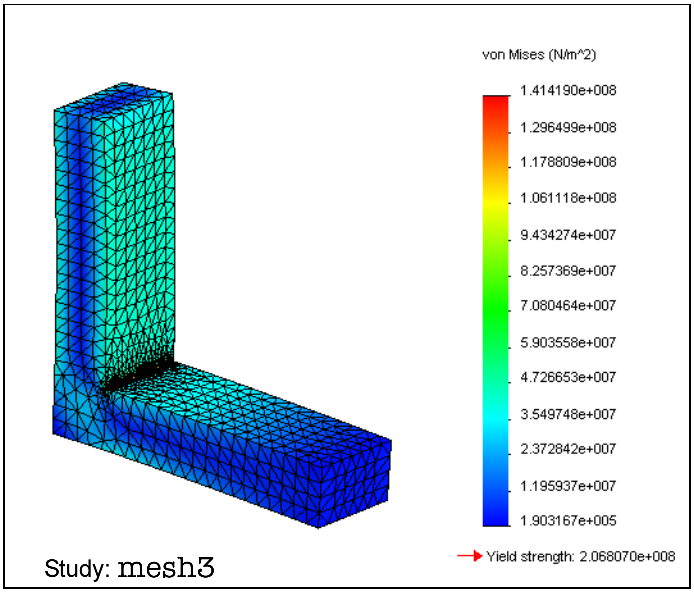
Display the mesh with the plot by right-clicking the corresponding result plot and selecting **Settings**.

Under **Boundary options**, select **Mesh**.

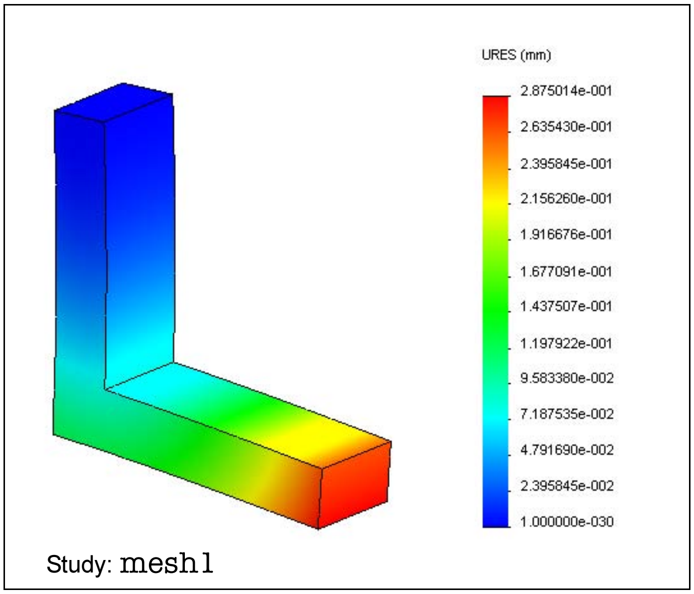
Click **OK**.

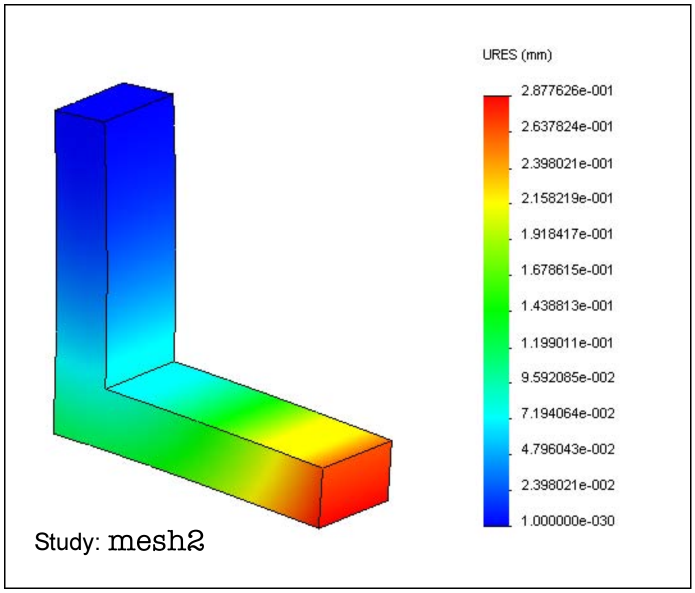


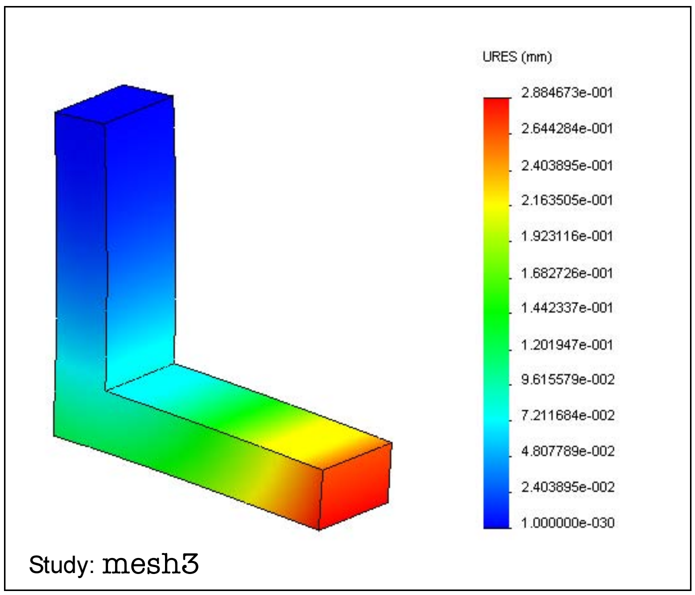




18 Plot resultant displacements.







Results

Reporting displacement results with six digits of accuracy is excessive as uncertainties in loads, restraints, and material properties definition do not normally justify this level of accuracy.

We used six digits of accuracy so that we can compare the minute differences in the displacement results calculated in the three studies we undertook in this lesson.

Results Comparison

Results for the maximum resultant displacement and maximum von Mises stress from `mesh1`, `mesh2` and `mesh3` studies are summarized in the following table:

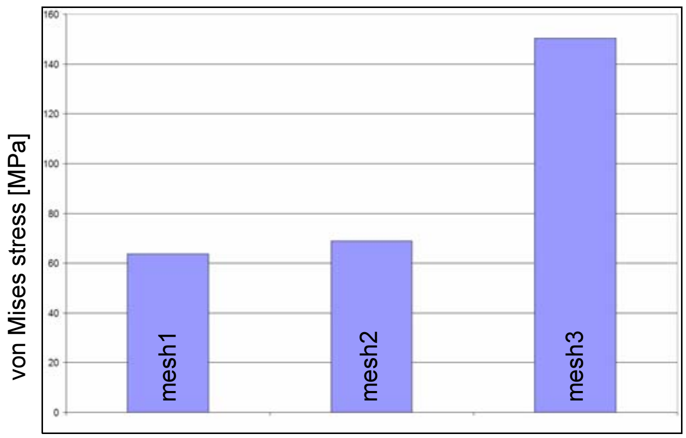
Study	Max. displ. [mm]	Increase in max. displ. [mm][%]	Max. Von Mises stress [MPa]	Increase in Von Mises stress [MPa][%]
mesh1	2.8750	-	63.67	-
mesh2	2.8779	0.0029 (0.1%)	68.89	5.22 (8.2%)
mesh3	2.8848	0.0098 (0.3%)	141.42	72.53 (105.3%)

Each mesh refinement results in an increase in both the maximum displacement and the maximum stress. The increase in the displacement results is negligible and becomes less pronounced with successive runs. From this, we can say that the displacement results converge.

If we continue this exercise of progressive mesh refinement, either locally near the sharp re-entrant corner as we did by means of the local mesh controls, or globally by reducing the global element size as we did in Lesson 1, we would note that the displacement results converge to a finite value and that even the first mesh is adequate if we are examining only displacement results.

Stress Singularities

Stresses, however, behave quite differently. Each subsequent mesh refinement produces higher stress results. Instead of converging to a finite value like the displacement results, the stress results diverge.



With enough time and patience, we can produce results that show any stress magnitude. All that is necessary is to make the element size small enough!

The reason for divergent stress results is not that the finite element model is incorrect, but that the finite element model is based on the wrong mathematical model.

According to the theory of elasticity, stress in the sharp re-entrant corner is infinite; a mathematician would say that stress there is singular. The finite element model does not produce infinite stress results due to discretization errors, and these discretization errors mask the modeling error.

However, stress results in the vicinity of the re-entrant corner are completely dependent on mesh size; therefore, they are totally meaningless at this location.

If our objective is to determine the maximum stress at this location, then the decision to suppress the fillet and analyze a model with a sharp re-entrant corner is a very serious mistake. The stress in a sharp re-entrant corner is singular, or infinite. The fillet, no matter how small it is, must be included in the model if we seek to find accurate stresses in or near that fillet.

Suppressed Configuration

When the active configuration is different from the configuration used to create the study, the study is suppressed and all items in the study are shown in grey. To unsuppress the study, the configuration must be changed to that used to do the study.

Activate SW Configuration

To change the SolidWorks configuration to the one used for a study, we can activate the configuration from the Simulation Study tree.

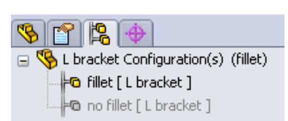
Where to Find It

- Right-click the study in the Simulation Study tree and click **Activate SW Configuration**.

Case Study: Analysis of Bracket with a Fillet

Now that we understand the problem caused by the sharp re-entrant corner, we must repeat this analysis using a model with the fillet. Obtaining the correct model requires unsuppressing the fillet.

1 Change SolidWorks configuration.
In the SolidWorks ConfigurationManager, make the configuration fillet active.



2 Examine the Simulation Study tree.
With the fillet configuration active, the `mesh1`, `mesh2` and `mesh3` studies are greyed-out. You can access them again only after activating the SolidWorks configuration corresponding to these studies.

3 Create new study.
Create a study `mesh4` by duplicating the `mesh1` study.

We copied the `mesh1` study and not the `mesh2` or `mesh3` studies for convenience because `mesh1` does not have mesh controls defined and `mesh4` does not require mesh controls.

If we use `mesh2` or `mesh3`, we have to edit or delete the mesh controls in the `mesh4` study because the geometry of the model has changed.

Automatic Transition

Because the fillet is a small feature compared to the overall size of the model, meshing with the default mesh settings produces an abrupt change in element size between the fillet and adjacent faces. To avoid this problem, we select the **Automatic transition** option in the **Mesh Parameters**. **Automatic transition** will automatically apply mesh controls to small features, details, holes and fillets.

Introducing: Automatic transition

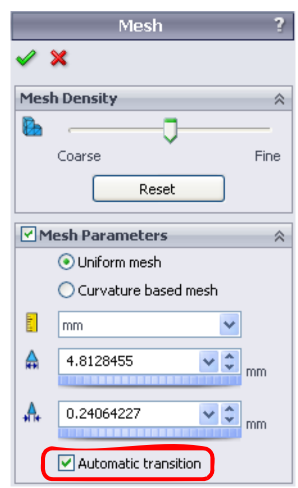
Automatic transition refines the mesh based on the curvature of the geometry. Mesh refinement in such regions is important for the accurate stress results.

Where to Find It

- Right-click `Mesh` folder and select **Create Mesh**. Expand **Mesh Parameters** and select **Automatic transition**.

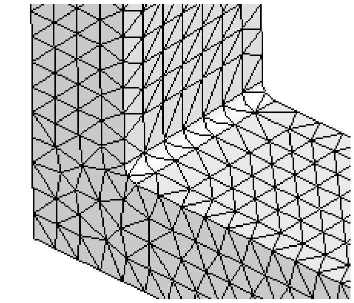
4 Mesh the model with Automatic transition option on.
Mesh the model with **High** quality elements and the default **Element size** of **4.813 mm [0.1895 in]** and the default **Tolerance** of **0.2406 mm [0.0095 in]**.

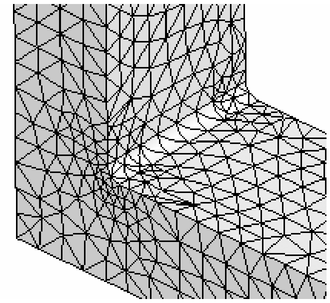
Select **Automatic transition**.



5 Compare meshes.

Compare the meshes created with and without the **Automatic transition** option.

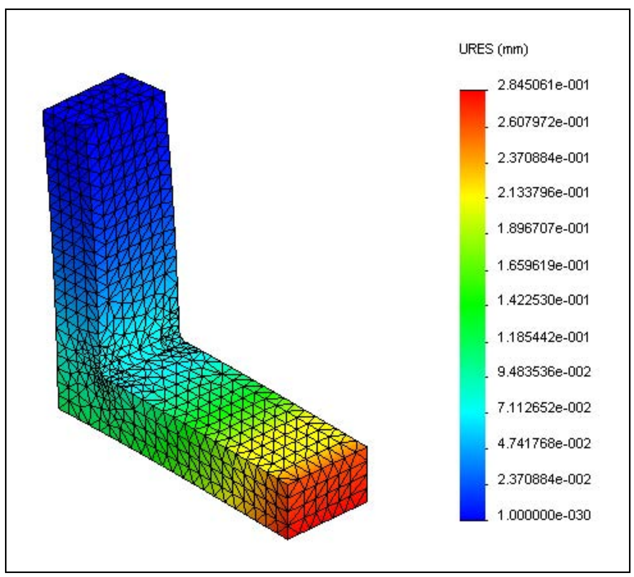

No Automatic Transition


Automatic Transition

6 Run the analysis.

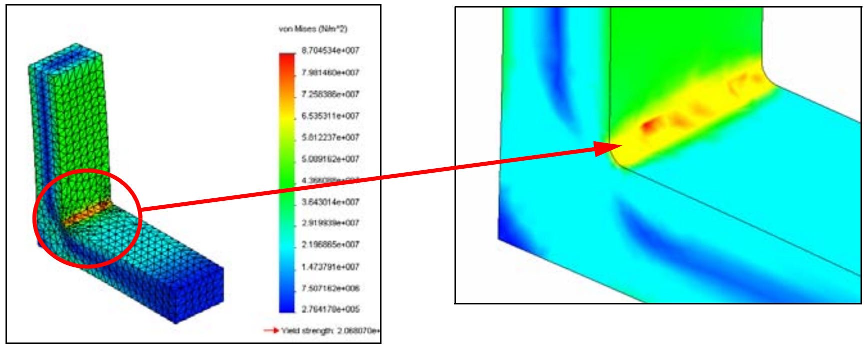
7 Plot Displacement results.

The maximum resultant displacement result (0.2845 mm) reported for the fillet study differs only insignificantly from the earlier displacement results. This small difference can be attributed to the change in the model geometry.



8 Plot von Mises stresses.

The stress results obtained by the model with the fillet indicate that the maximum von Mises stress is at the fillet location and its magnitude is 87.04 MPa.



9 Analyze the plots.

Analyzing the stress distribution uniformity at the fillet location we see rather spotty behavior and no symmetry. This is another sign of insufficient mesh resolution for stresses. The displacement results are accurate in all studies solved in this lesson.

We will therefore apply a new local mesh control on fillet and rerun the study again.

10 Apply mesh control on fillet.

To get more accurate results, we will apply a local mesh control on the fillet face.

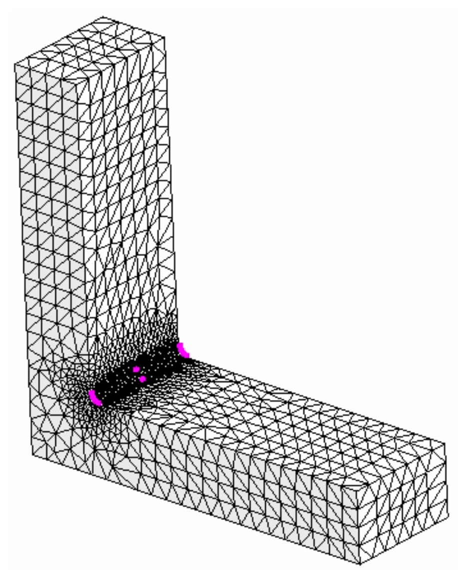
Apply mesh controls to the fillet face using **0.762 mm [0.030 in]** for the local **Element size**, **1.2** for the **Ratio**.

11 Re-mesh model.

Mesh the model with the default parameters, except clear **Automatic transition**.

The resulting mesh can be seen at right.

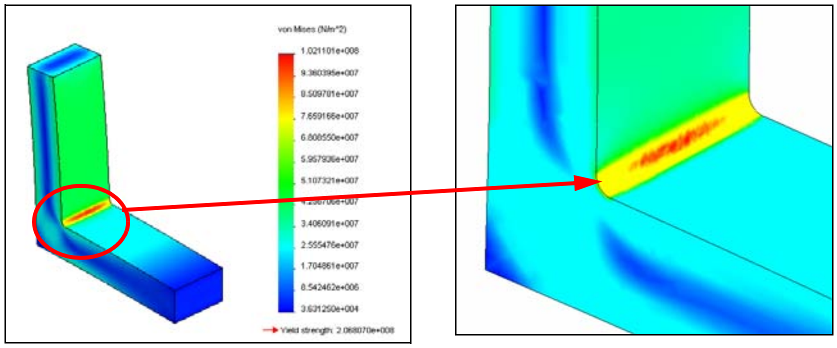
This mesh is a little excessive in its size, but given the small size of the problem we can afford it.



12 Run the study.

13 Plot von Mises stress.

We observe that the maximum stress increased to 102 MPa. The details of the stress distribution are uniform and symmetrical. We could conclude that this stress value is accurate.



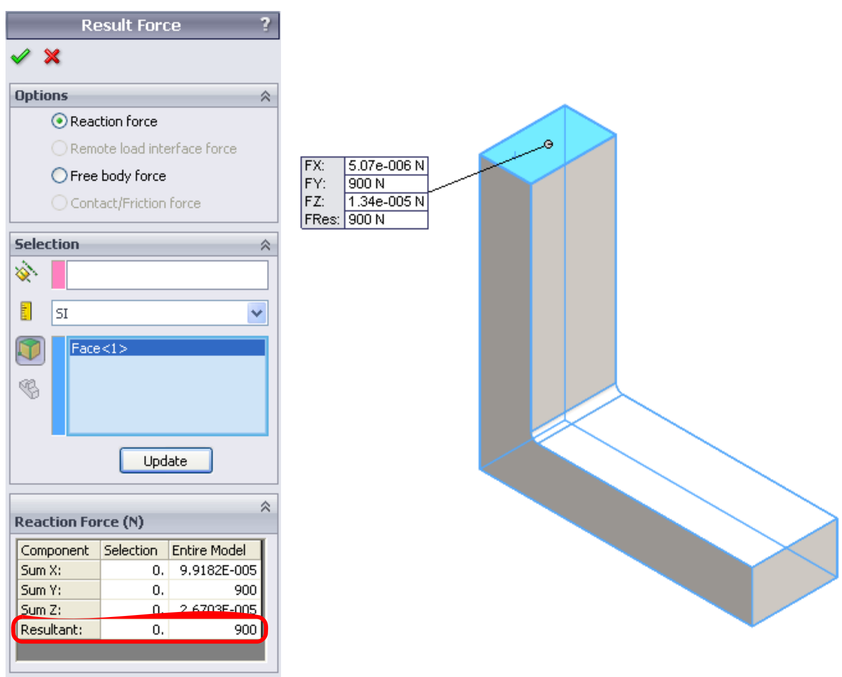
14 Extract reaction force.

Right-click on the Results folder and select **List Result Force**.

Select the face where the bracket is supported and click **Update**. Make sure the units are set to **SI**.

The **Reaction force (N)** dialog will list the resultant of the reaction on the selected face (or faces, if more supported faces exist and are selected) as well as on the entire model.

We can see that the equilibrium is satisfied; the reaction force is equal to 900 N, which confirms the equilibrium and the correctness of the solution.



Note

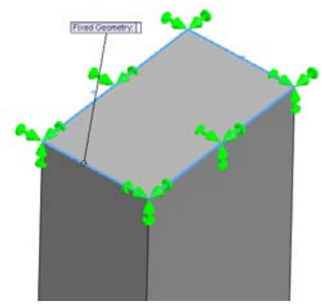
Moment reactions are not reported since solid elements feature three translational degrees of freedom only. Nodes of the solid elements do not carry any moment.

Case Study: Analysis of a Welded Bracket

Now that we understand the stress concentration in the fillet, let's repeat the analysis using a more realistic model where the edges of the faces are fixed rather than the entire face. This would more closely represent the face being welded to a plate.

1 **Create a new study.**
Create a static study named `mesh5` by duplicating the study `mesh4`.

2 **Edit the Fixture.**
Edit the fixture and remove the top face. Add the four edges surrounding that face as shown.



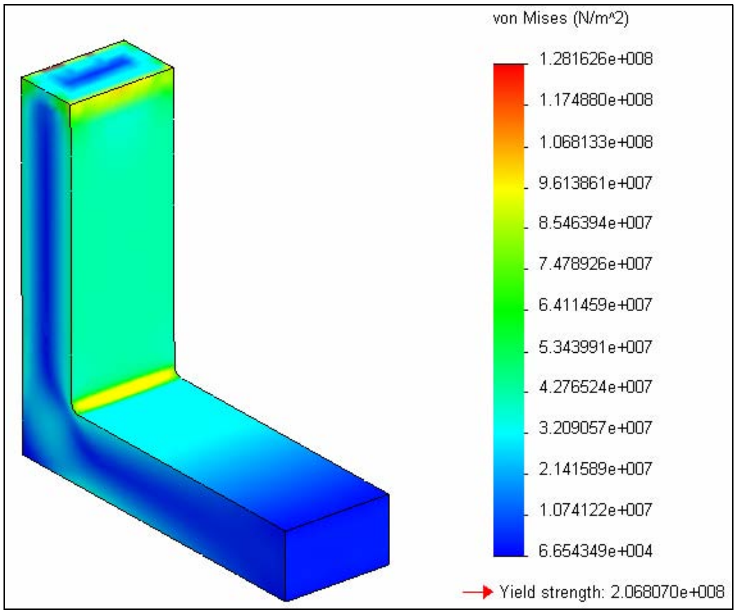
This type of restraint would simulate the part being welded to a surface when only the edges are firmly attached to the structure, and not the entire face.

3 **Run the analysis.**

4 Plot the stress results.

Another stress concentration has appeared at the edges where fixed geometry was used. Again, a singularity of stress is formed due to the fixed geometry at the sharp end. Although perhaps a more realistic finite element model, the stress concentration is an artifact of the mathematical model.

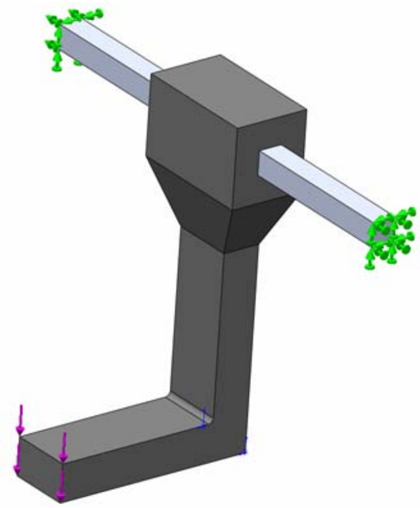
These types of effects must be understood to properly analyze model results.



5 Save and Close the file.

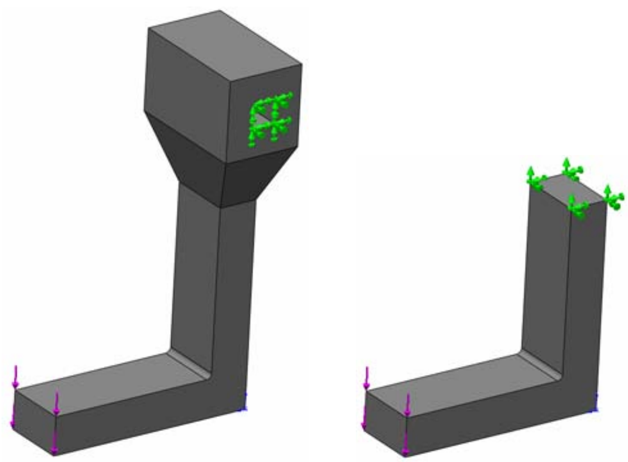
Understanding the Effect of Boundary Conditions

Boundary conditions are necessary in order to fix the model in space and solve the mathematical problem. In real life every part is connected to another and finally attached to the primary structure or the ground.



We can, however, view the boundary conditions as a means to significantly simplify our simulation. As an illustration, consider the bracket assembly shown in the figure to the right, where the bracket is part of a larger structure.

Then, before we even begin modeling in SolidWorks Simulation we have to decide whether to model the entire upper level assembly with the boundary conditions applied as shown in the figure above, or the full bracket only, or a part of the bracket (a model identical to what we had in Lesson 2). See the images below.



The decision is based on what is the objective of the analysis, i.e. what results do we truly need. The larger the model we chose, the more realistic it becomes. At the same time the size of the finite element model increases, resulting in significantly longer solution times. Boundary conditions therefore serve to express the fact how a specific part or sub-assembly is grounded or attached to another primary structure, and help us substantially reduce the size of the problem. Reduction of the problem comes at a cost, i.e. the stress results at the location of the boundary conditions may be singular and have to be ignored in such cases.

Also, we need to understand that the boundary conditions do effect our solution. In the three cases listed above the final results will be comparable, but not exactly the same. Therefore the selection, as well as the location, of the boundary condition must be done so that its effect on the results and the rest of the model is minimal.

Conclusion

The question may arise: which one study is the correct one?

The second to last study with the fillet and fixed face included in the model and the mesh control applied produce the most accurate results and is favored provided one can afford the increased size of the model due to the additional regions that must be meshed. Then what about the other studies where stress concentrations are seen?

These results are obtained by using the incorrect mathematical model. It does not make sense to debate which of the first three models produces the most accurate results and, therefore, which one was "the best" among the three. All models with sharp re-entrant edges or edges that are fixed are equally poor if we examine the stress on those edges.

Thus, if we are interested in stress at or near a sharp edge (or a sharp corner for shell models), this edge must be modeled with a fillet, even if the fillet is very small. In addition, if the edge of the model is fixed, we must realize that the appearance of the stress concentration is artificial. In general, if stresses at these singularities are of no interest, these studies still produce good results for the overall model.

Summary

In this lesson, we illustrated what can go wrong when FEA is based on an incorrectly prepared model.

Using local mesh controls rather than the global mesh controls, we obtained solutions for different meshes and revealed stress singularities at a sharp re-entrant corner and at fixed geometries.

We used this lesson to further discuss modeling and discretization error, meshing techniques, and also to illustrate the integration between SolidWorks and SolidWorks Study tree.

Questions

- Why do we often eliminate fillets and small rounds if such suppression can lead to locally inaccurate stress results? Does it imply that the stress results are inaccurate for the whole model?
- Are displacements affected by the suppression of small features (fillets, rounds) as much as stresses? Why?

Exercise 4: C-bracket

In this exercise, you will analyze a bracket with two different configurations to determine the effects of the internal fillets.

This exercise reinforces the following skills:

- *Mesh Controls* on page 90.
- *Results Comparison* on page 97.
- *Stress Singularities* on page 97.
- *Suppressed Configuration* on page 98.

Problem Statement

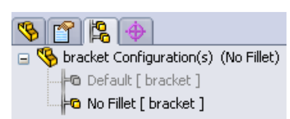
A hanging bracket mounted on the ceiling will be supporting a sign mounted on the bottom flange of the bracket. The sign will be mounted onto the bracket with a flat ribbon like cable. A 900 N [202 lb.] force will be exerted on the bracket due to the weight of the sign and ribbon. We will evaluate the displacements and stresses for the bracket due to this loading. We are also interested in how modeling the bracket with and without fillets will effect our results.

The effects of different boundary conditions will also be investigated.

Part 1: Analysis of Bracket with no Fillet

1 Open a part file.
Open `bracket` from `Lesson02\Exercises` folder.

2 Specify active configuration.
Make the configuration `No Fillet` active.



Notice that the rounded inside edges become sharp re-entrant corners. This configuration, suppresses all inner fillets.

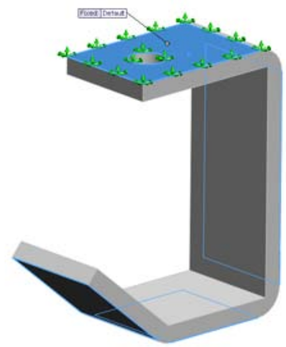
3 Define a static study.
Create a **Static** study named `no fillet 1`.

4 Apply material properties.
Apply the material **Alloy Steel** from the `solidworks material` library.

5 Apply a fixture.

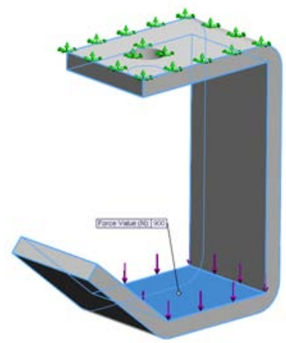
Apply a **Fixed Geometry** fixture to the top face as indicated.

We will assume that the compressive force of the screw is large enough to prevent any sliding or rotation about the screw.



6 Apply force.

Apply a **900 N [202 lb]** normal force to the top face of the bottom flange. This force is due to the weight of the sign.



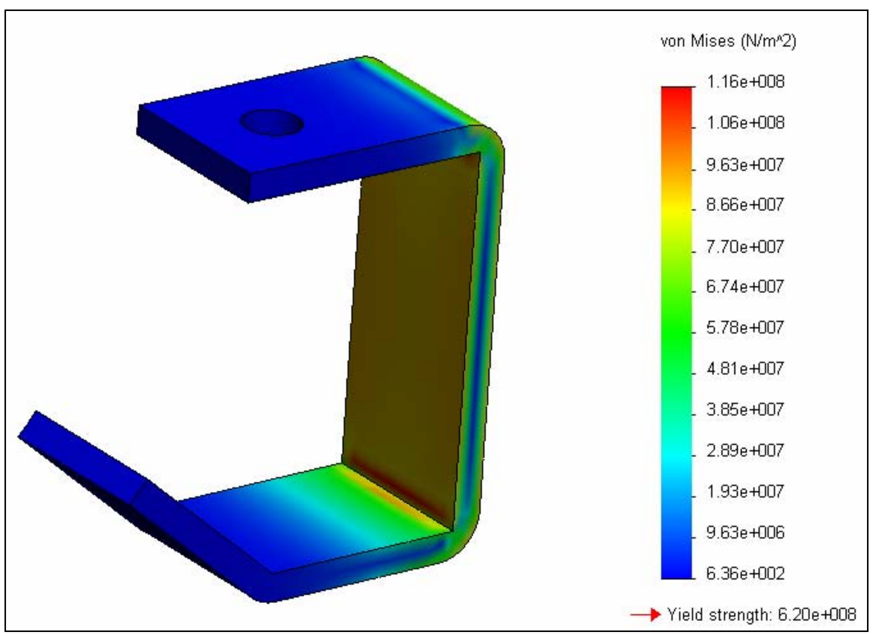
7 Mesh the model.

Mesh the model with the default element size. Use **High** quality elements.

8 Run the analysis.

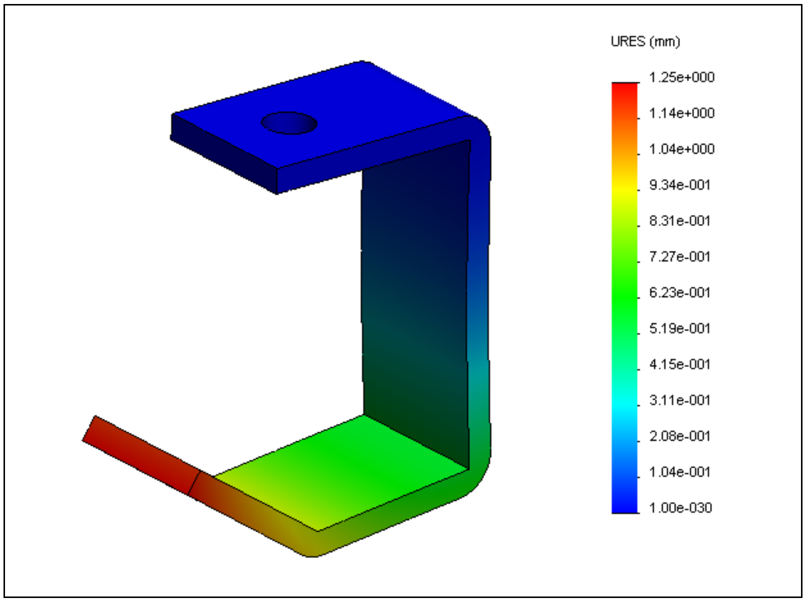
9 Plot stress results.

We find that the bracket has a maximum von Mises stress of 116 MPa [16.7 ksi] and does not yield. However, there is a high stress concentration at the sharp corners.



10 Plot displacement results.

Maximum displacement is 1.25 mm.

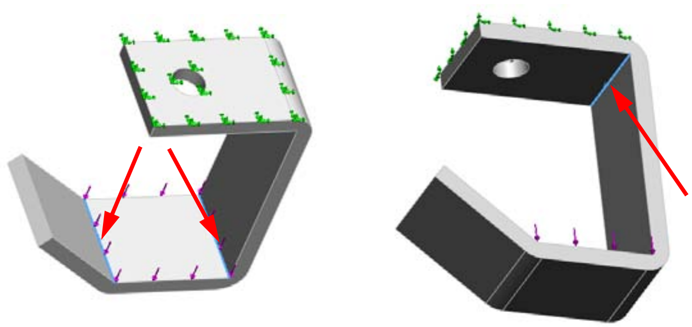


11 Create a new study.

Duplicate the existing study and name it `no fillet 2`.

12 Apply mesh control.

Apply mesh control to each of the three edges on the inner faces of the bracket. Use the default mesh control size.



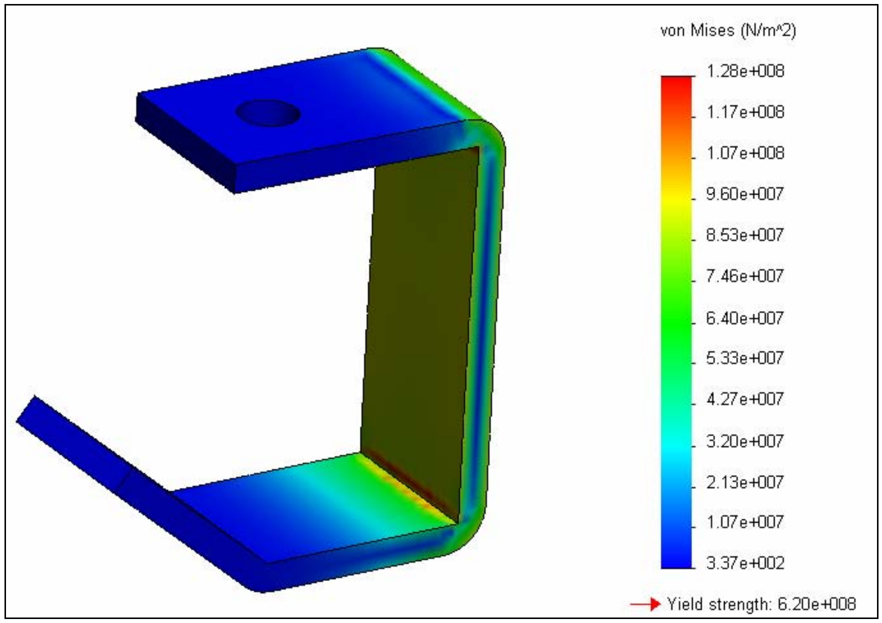
13 Mesh the model.

Mesh the model with the default element size. We have created a finer mesh at the inside edges of the bracket, while the mesh sizes are coarser at all other locations in the bracket.

14 Run the analysis.

15 Plot stress results.

The maximum von Mises stress is now 128 MPa [18,565 psi], which is slightly higher than the von Mises stress value obtained in the previous study with no mesh control.



16 Create a new study.

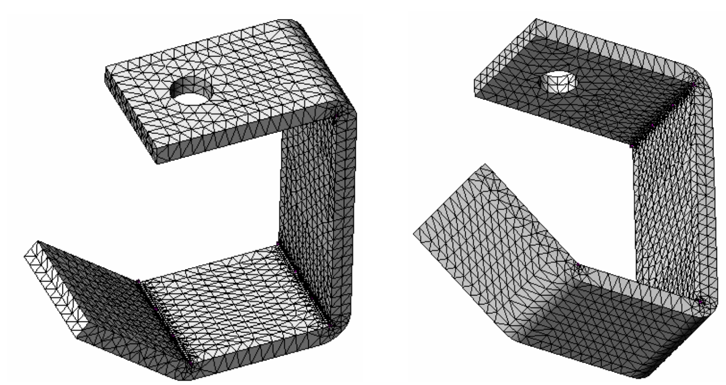
Duplicate the no fillet 1 study and name it no fillet 3.

17 Apply mesh control.

Add mesh control to the same three edges. Change the local **Element size** to **0.889 mm [0.035 in]**.

18 Mesh the model.

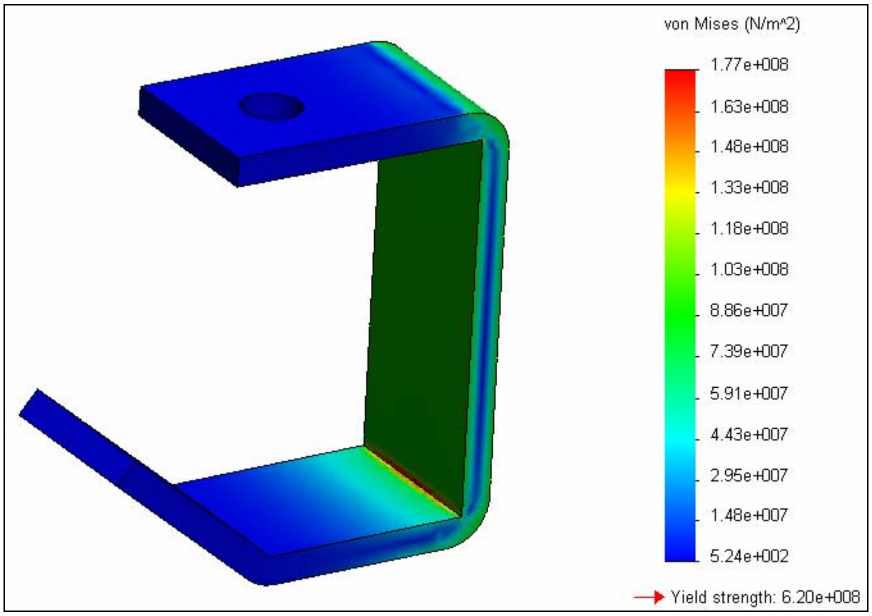
Mesh the model with the default element size. We have created a finer mesh at the inside edges of the bracket, while the mesh sizes are coarser at all other locations in the bracket.



19 Run the analysis.

20 Plot stress results.

We find that the maximum von Mises stress is significantly higher than the value obtained in the previous study with a coarser mesh control. We see that, although we are refining the mesh, the stress results are not converging. This is due to the sharp re-entrant corners.



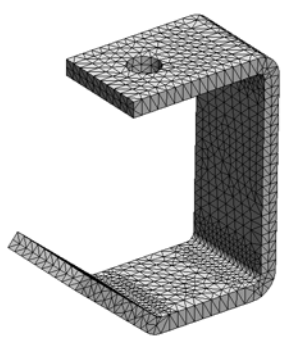
Part 2: Analysis of Bracket with Fillet

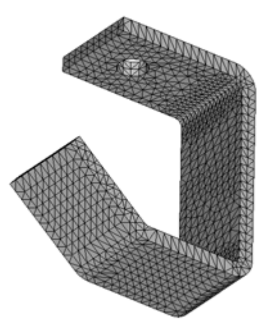
We will now look at a model with fillets and analyze its solution.

1 Change configuration.
Change the active configuration to `Default`. This configuration has the fillets unsuppressed.

2 Create a new study.
Duplicate the `no fillet 1` study and name it `fillet`.

3 Mesh the model.
Mesh the model with the default local **Element size** and the **Automatic transition** option selected.

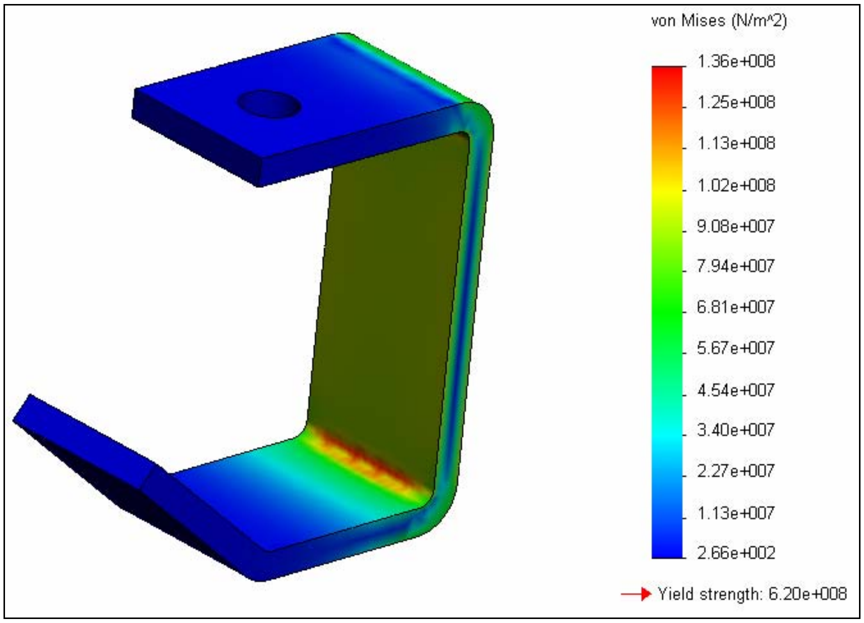




4 Run the analysis.

5 Plot stress results.

The stress results obtained from the model with the fillet indicate that the maximum von Mises stress is approximately 136 MPa [19.7 ksi]. Because no sharp edges are present in the model, this value is close to the real stress magnitudes. Further mesh refinement would improve the results and eliminate the spotty stress distribution.



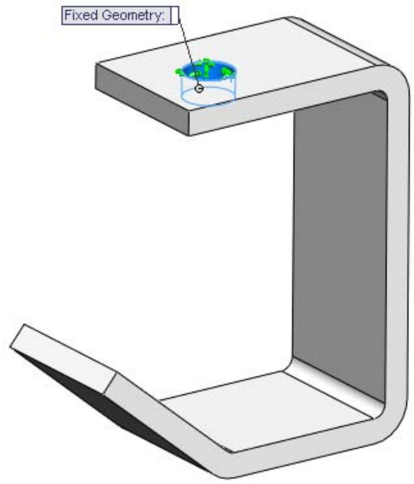
Part 3: Analysis of Bracket with Fillet and Fixed Hole

In this last study, we will change the way the part is restrained by editing the one fixture and holding the part by the cylindrical hole instead of the entire top face.

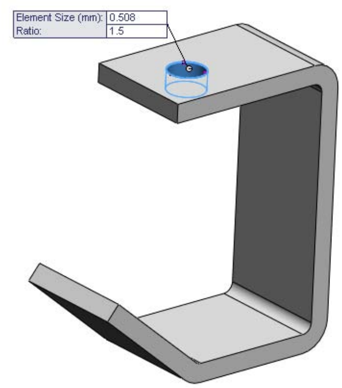
1 Create a new study.

Duplicate the `fillet` study and name it `fillet fixed hole`.

2 Use Fixed Geometry on hole.
Edit the fixture and remove the top face. Add the hole face.

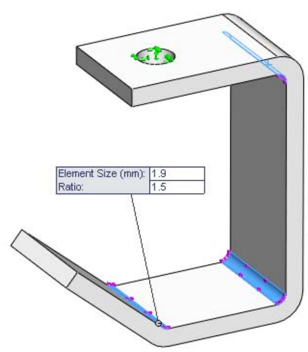


3 Apply mesh control.
Apply a mesh control with an **Element size** of **0.508 mm** to the inner cylindrical surface of the hole.

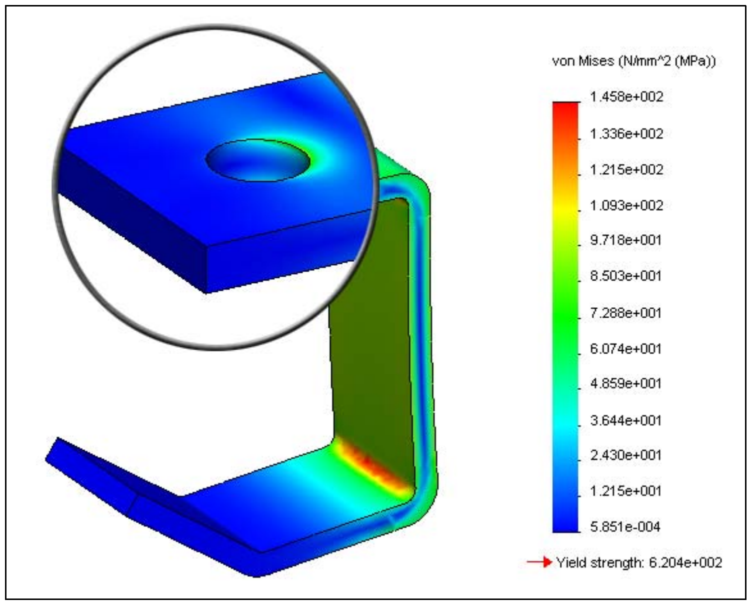


4 Mesh control on the fillets.
Apply a mesh control with an **Element size** of **1.9 mm** to the three fillets.

5 Run the analysis.
The study will mesh and solve.

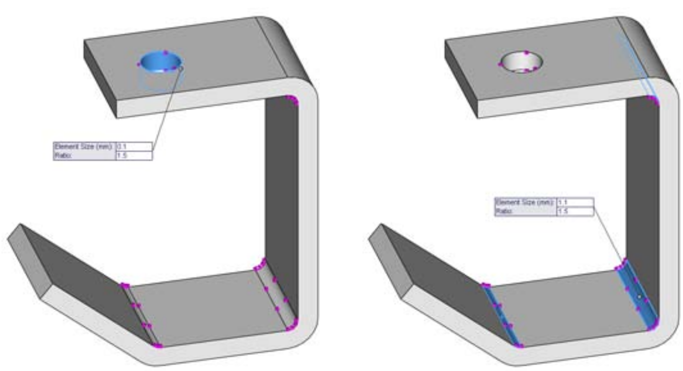


6 Plot the stress results.
The stress results obtained from the model with the fillet and the fixed geometry on the hole produce a stress concentration around the edges of the hole. This is because a singularity of stress appears in this region due to the perfectly rigid support at those edges. This is similar to the singularity seen in the fixed edges of the `L-Bracket` in *Lesson 2* and can be ignored. Change the scale of the legend to obtain a more realistic plot.



We can see that the stresses on the filleted faces increased from 136 MPa (see previous study) to nearly 146 MPa.

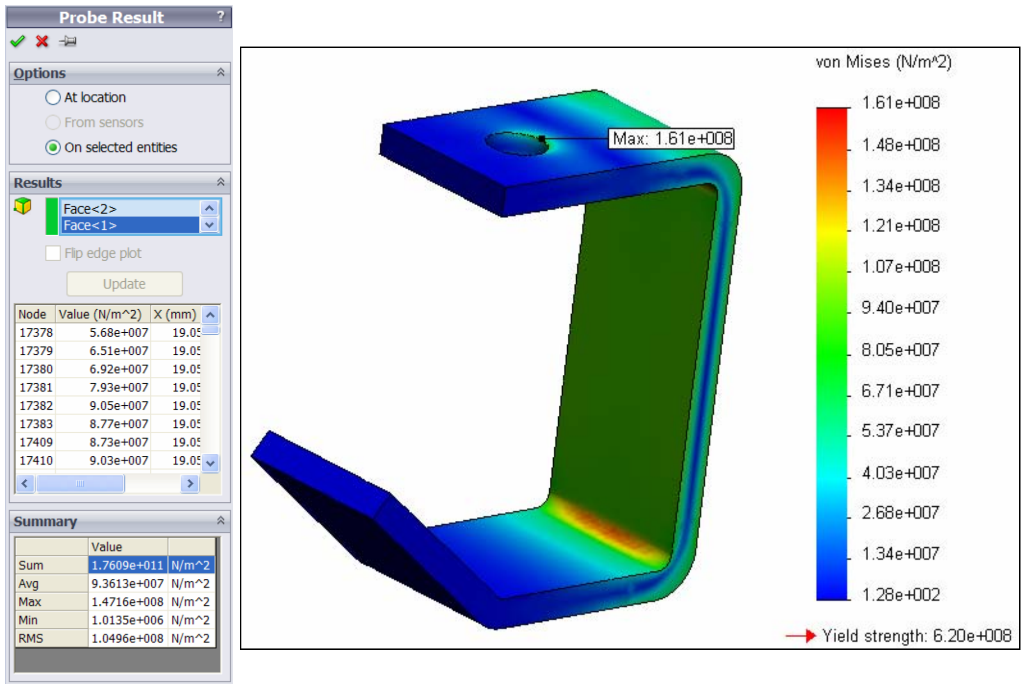
7 Modify the mesh controls.
Change the Element Size for both mesh controls to 0.1mm for the hole and 1.1mm for the fillets.



8 Run the Study.
The study will mesh and solve.

9 Examine the stress plot.

As can be seen the stress near the support increased considerably and represents the maximum stress in the model. From Lesson 2 we know that this stress is unreal and will increase as we reduce the size of the elements.



Probing on selected entities reveals the maximum stress on the filleted faces as 147 MPa, a slight increase from the 145 MPa obtained from the previous run. This stress is real and approaching a finite value (we say it converges).

10 Save and Close the file.