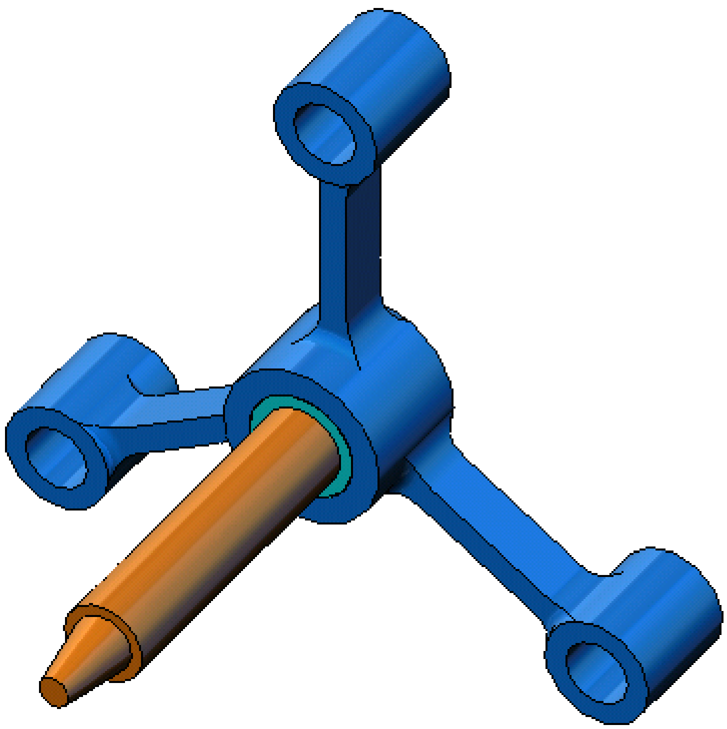
Engineering Design and Technology Series

An Introduction to Stress Analysis Applications with SolidWorks Simulation, Student Guide



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Introduction

About This Course

The *Introduction to Stress Analysis Applications with SolidWorks Simulation* and its supporting materials is designed to assist you in learning SolidWorks Simulation in an academic setting.

Online Tutorials

The *Introduction to Stress Analysis Applications with SolidWorks Simulation* is a companion resource and is supplemented by the SolidWorks Simulation Online Tutorials.

Accessing the Tutorials

To start the Online Tutorials, click **Help, SolidWorks Tutorials, All SolidWorks Tutorials**. The SolidWorks window is resized and a second window will appears next to it with a list of the available tutorials. As you move the pointer over the links, an illustration of the tutorial will appear at the bottom of the window. Click the desired link to start that tutorial.

Conventions

Set your screen resolution to 1280x1024 for optimal viewing of the tutorials.

The following icons appear in the tutorials:

Next » Moves to the next screen in the tutorial.

Represents a note or tip. It is not a link; the information is to the right of the icon. Notes and tips provide time-saving steps and helpful hints.

You can click most toolbar buttons that appear in the lessons to flash the corresponding SolidWorks button. The first time you click the button, an ActiveX control message appears: An ActiveX control on this page might be unsafe to interact with other parts of the page. Do you want to allow this interaction? This is a standard precautionary measure. The ActiveX controls in the Online Tutorials will not harm your system. If you click **No**, the scripts are disabled for that topic. Click **Yes** to run the scripts and flash the button.

Open File or **Set this option** automatically opens the file or sets the option.

Video example shows a video about this step.

A closer look at... links to more information about a topic. Although not required to complete the tutorial, it offers more detail on the subject.

Why did I... links to more information about a procedure, and the reasons for the method given. This information is not required to complete the tutorial.

Printing the Tutorials

If you like, you can print the Online Tutorials by following this procedure:

1. On the tutorial navigation toolbar, click **Show**.

   This displays the table of contents for the Online Tutorials.

2. Right-click the book representing the lesson you wish to print and select **Print** from the shortcut menu.

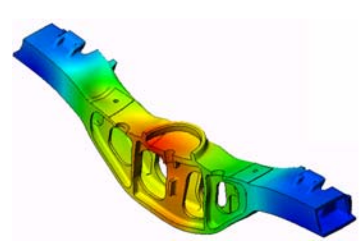
   The **Print Topics** dialog box appears.

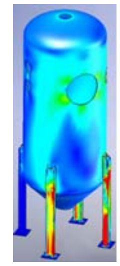
3. Select **Print the selected heading and all subtopics**, and click **OK**.
4. Repeat this process for each lesson that you want to print.

SolidWorks Simulation Product Line

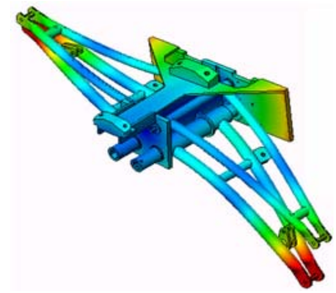
While this course focuses on the introduction to the static linear simulation of elastic bodies using SolidWorks Simulation, the full product line covers a wide range of analysis areas to consider. The paragraphs below lists the full offering of the SolidWorks Simulation packages and modules.

Static studies provide tools for the linear stress analysis of parts and assemblies loaded by static loads. Typical questions that will be answered using this study type are:
Will my part break under normal operating loads?
Is the model over-designed?
Can my design be modified to increase the safety factor?

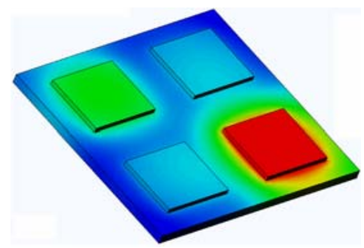




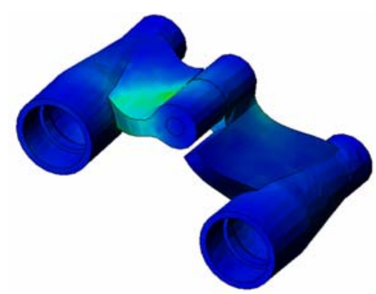
Buckling studies analyze performance of the thin parts loaded in compression. Typical questions that will be answered using this study type are:
Legs of my vessel are strong enough not to fail in yielding; but are they strong enough not to collapse due to loss of stability?
Can my design be modified to ensure stability of the thin components in my assembly?



Frequency studies offer tools for the analysis of the natural modes and frequencies. This is essential in the design or many components loaded in both static and dynamic ways. Typical questions that will be answered using this study type are:
Will my part resonate under normal operating loads?
Are the frequency characteristics of my components suitable for the given application?
Can my design be modified to improve the frequency characteristics?

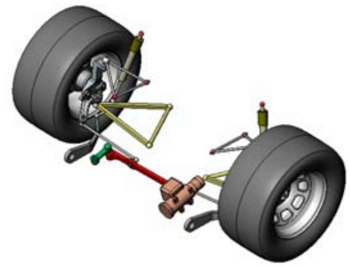


Thermal studies offer tools for the analysis of the heat transfer by means of conduction, convection, and radiation. Typical questions that will be answered using this study type are:
Will the temperatures changes effect my model?
How does my model operate in an environment with temperature fluctuation?
How long does it take for my model to cool down or overheat?
Does temperature change cause my model to expand?
Will the stresses caused by the temperature change cause my product failure (static studies, coupled with thermal studies would be used to answer this question)?



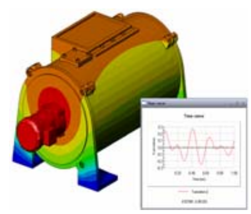
Drop test studies are used to analyze the stress of moving parts or assemblies impacting an obstacle. Typical questions that will be answered using this study type are:
What will happen if my product is mishandled during transportation or dropped?
How does my product behave when dropped on hard wood floor, carpet or concrete?

Optimization studies are applied to improve (optimize) your initial design based on a set of selected criteria such as maximum stress, weight, optimum frequency, etc. Typical questions that will be answered using this study type are:
Can the shape of my model be changed while maintaining the design intent?
Can my design be made lighter, smaller, cheaper without compromising strength of performance?

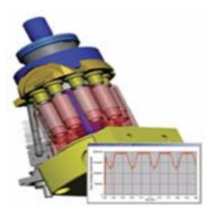


Fatigue studies analyze the resistance of parts and assemblies loaded repetitively over long periods of time. Typical questions that will be answered using this study type are:
Can the life span of my product be estimated accurately?
Will modifying my current design help extend the product life?
Is my model safe when exposed to fluctuating force or temperature loads over long periods of time?
Will redesigning my model help minimize damage caused by fluctuating forces or temperature?

Nonlinear studies provide tools for analyzing stress in parts and assemblies that experience severe loadings and/or large deformations. Typical questions that will be answered using this study type are:
Will parts made of rubber (o-rings for example) or foam perform well under given load?
Does my model experience excessive bending during normal operating conditions?



Dynamics studies analyze objects forced by loads that vary in time. Typical examples could be shock loads of components mounted in vehicles, turbines loaded by oscillatory forces, aircraft components loaded in random fashion, etc. Both linear (small structural deformations, basic material models) and nonlinear (large structural deformations, severe loadings and advanced materials) are available. Typical questions that will be answered using this study type are:
Are my mounts loaded by shock loading when vehicle hits a large pothole on the road designed safely? How much does it deform under such circumstances?



Motion Simulation enables user to analyze the kinematic and dynamic behavior of the mechanisns. Joint and inertial forces can subsequently be transferred into SolidWorks Simulation studies to continue with the stress analysis. Typical questions that will be answered using this modulus are:
What is the correct size of motor or actuator for my design?
Is the design of the linkages, gears or latch mechanisms optimal?
What are the displacemements, velocities and accelerations of the mechanism components?
Is the mechanism efficient? Can it be improved?

Composites modulus allows users to simulate structures manufactured from laminated composite materials.
Typical questions that will be answered using this modulus are:
Is the composite model failing under the given loading?
Can the structure be made lighter using composite materials while not compromising with the strength and safety?
Will my layered composite delaminate?

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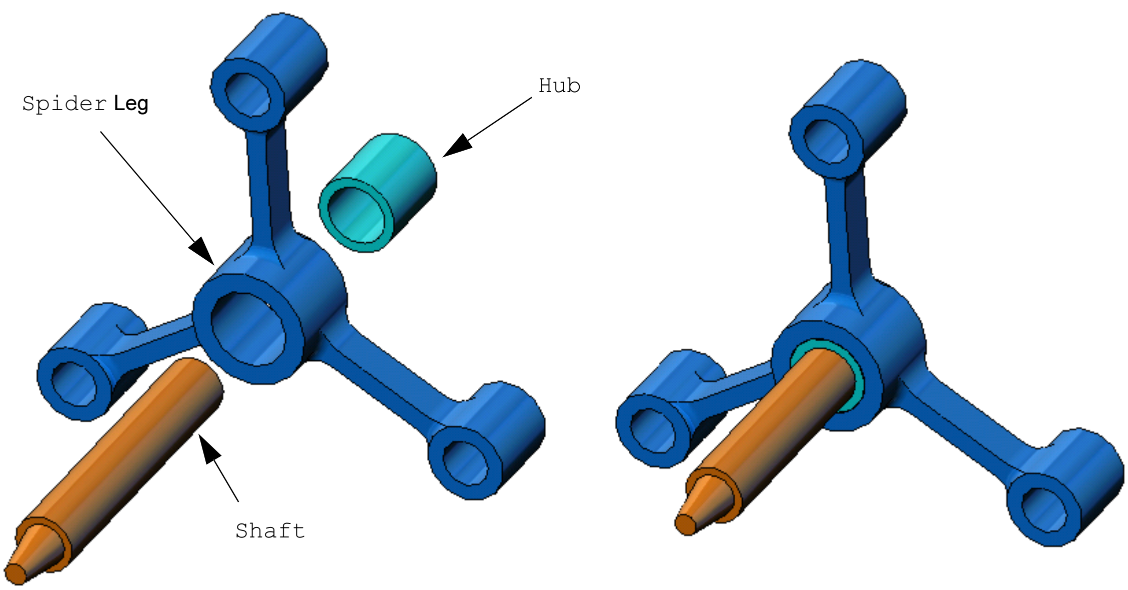
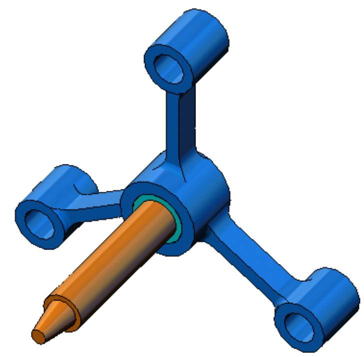
Lesson 1: Basic Functionality of SolidWorks Simulation

Upon successful completion of this lesson, you will be able to understand the basic functionality of SolidWorks Simulation and perform static analysis of the following assembly.



Active Learning Exercise — Performing Static Analysis



Use SolidWorks Simulation to perform static analysis on the Spider.SLDASM assembly shown to the right.

The step-by-step instructions are given below.

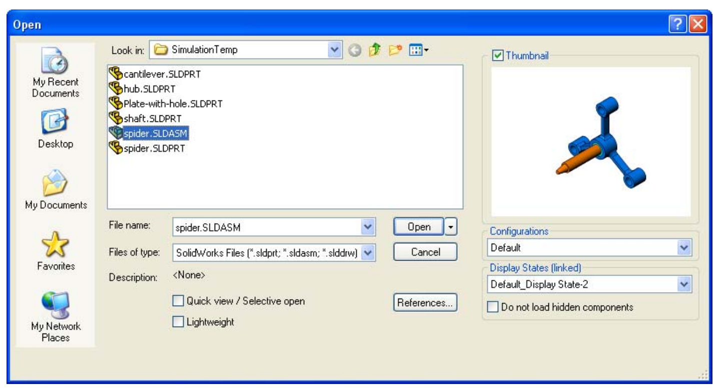
Creating a SimulationTemp directory

We recommend that you save the SolidWorks Simulation Education Examples to a temporary directory to save the original copy for repeated use.

1 Create a temporary directory named SimulationTemp in the Examples folder of the SolidWorks Simulation installation directory.

2 Copy the SolidWorks Simulation Education Examples directory into the SimulationTemp directory.

Opening the Spider.SLDASM Document



1 Click **Open**  on the Standard toolbar. The **Open** dialog box appears.

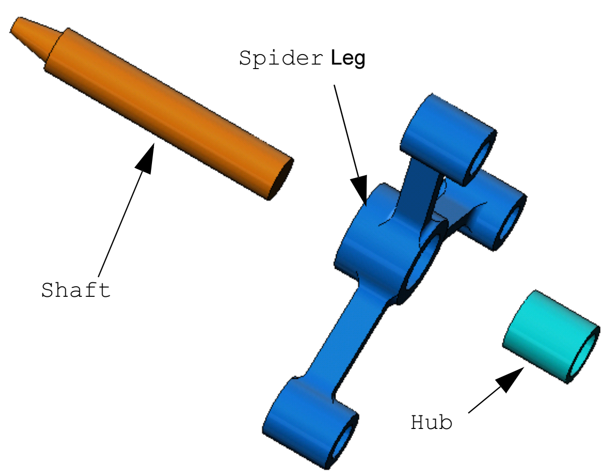
2 Navigate to the SimulationTemp folder in the SolidWorks Simulation installation directory.

3 Select Spider.SLDASM

4 Click **Open**.

The spider.SLDASM assembly opens.

The spider assembly has three components: the shaft, hub, and spider leg. The figure below shows the assembly components in exploded view.



Checking the SolidWorks Simulation Menu

If SolidWorks Simulation is properly installed, the SolidWorks Simulation menu appears on the SolidWorks menu bar. If not:

SolidWorks Simulation menu

1 Click **Tools, Add-Ins**.

  The **Add-Ins** dialog box appears.

2 Check the checkboxes next to SolidWorks Simulation.
  If SolidWorks Simulation is not in the list, you need to install SolidWorks Simulation.

3 Click **OK**.

  The Simulation menu appears on the SolidWorks menu bar.

Setting the Analysis Units

Before we start this lesson, we will set the analysis units.

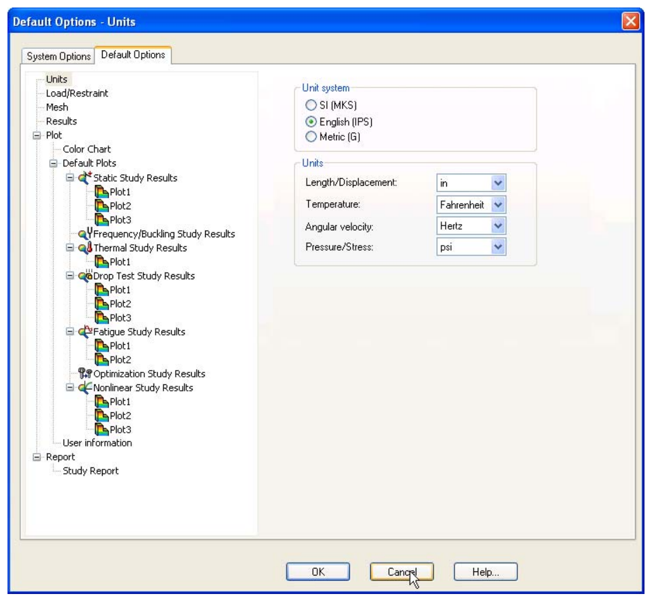
1 On the SolidWorks menu bar click **Simulation**, **Options**.

2 Click the **Default Options** tab.

3 Select **English (IPS)** under **Unit system.**

4 Select **in** and **psi** from the **Length/ Displacement** and **Pressure/ Stress** fields, respectively.

5 Click **OK**.



Step 1: Creating a Study

The first step in performing analysis is to create a study.

1 Click **Simulation**, **Study** in the main SolidWorks menu on the top of the screen.

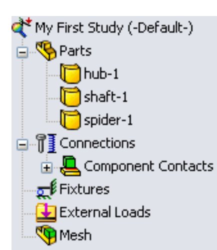
  The **Study** PropertyManager appears.

2 Under **Name**, type `My First Study`.

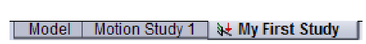
3 Under **Type**, select **Static**.

4 Click **OK**.

  SolidWorks Simulation creates a Simulation study tree located beneath the FeatureManager design tree.



  A tab is also created at the bottom of the window for you to navigate between multiple studies and your model.



Step 2: Assigning Material

All assembly components are made of Alloy Steel.

Assign Alloy Steel to All Components

1. In the SolidWorks Simulation Manager tree, right-click the `Parts` folder and click **Apply Material to All**.

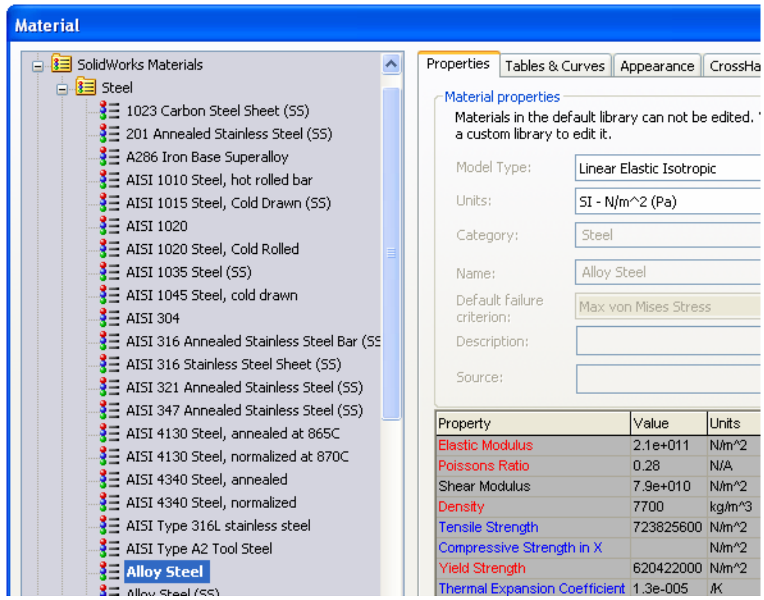
   The `Material` dialog box appears.

2. Do the following:

   a) Expand the `SolidWorks Materials` library folder.

   b) Expand the `Steel` category.

   c) Select **Alloy Steel**.



Note: The mechanical and physical properties of Alloy Steel appear in the table to the right.

3. Click **Apply**.

4. Close the **Materials** window.

   Alloy steel is assigned to all components and a check mark appears on each component's icon. Note that the name of the assigned material appears next to the component's name.

Step 3: Applying Fixtures

We will fix the three holes.

1. Use the **Arrow** keys to rotate the assembly as shown in the figure.

2. In the Simulation study tree, right-click the `Fixtures` folder and click **Fixed Geometry**.

   The **Fixture** PropertyManager appears.

3. Make sure that **Type** is set to **Fixed Geometry**.

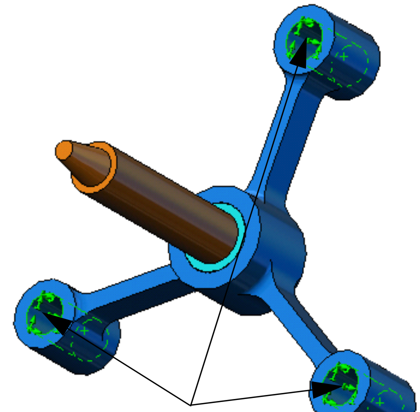
4. In the graphics area, click the faces of the three holes, indicated in the figure below.

   `Face<1>`, `Face<2>`, and `Face<3>` appear in the **Faces, Edges, Vertices for Fixture** box.

5 Click .

Fixed fixture is applied and its symbols appear on the selected faces.

Also, `Fixed-1` item appears in the `Fixtures` folder in the Simulation study tree. The name of the fixture can be modified at any time.



Step 4: Applying Loads

We will apply a 500 lb force normal to the face shown in the figure.

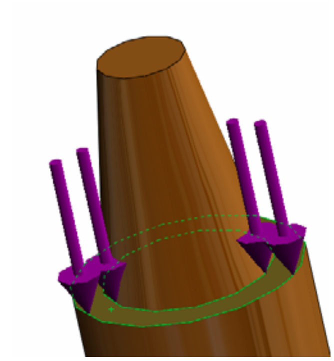
1 Click **Zoom to Area** icon on the top of the graphics area and zoom into the tapered part of the shaft.

2 In the SolidWorks Simulation Manager tree, right-click the `External Loads` folder and select **Force**.

The **Force/Torque** PropertyManager appears.

3 In the graphics area, click the face shown in the figure.

`Face<1>` appears in the **Faces and Shell Edges for Normal Force** list box.



4 Make sure that **Normal** is selected as the direction.

5 Make sure that **Units** is set to **English (IPS)**.

6 In the **Force Value** box, type **500**.

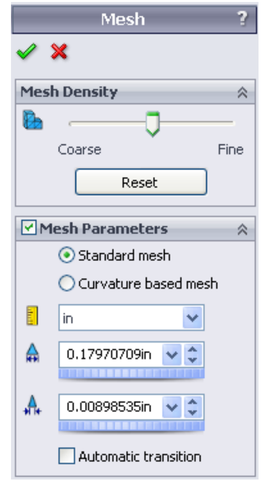
7 Click .

SolidWorks Simulation applies the force to the selected face and `Force-1` item appears in the `External Loads` folder.

To Hide Fixtures and Loads Symbols

In the SolidWorks Simulation Manager tree, right-click the `Fixtures` or `External Loads` folder and click **Hide All**.

Step 5: Meshing the Assembly



Meshing divides your model into smaller pieces called elements. Based on the geometrical dimensions of the model SolidWorks Simulation suggests a default element size (in this case 0.179707 in) which can be changed as needed.

1 In the Simulation study tree, right-click the `Mesh` icon and select **Create Mesh**.

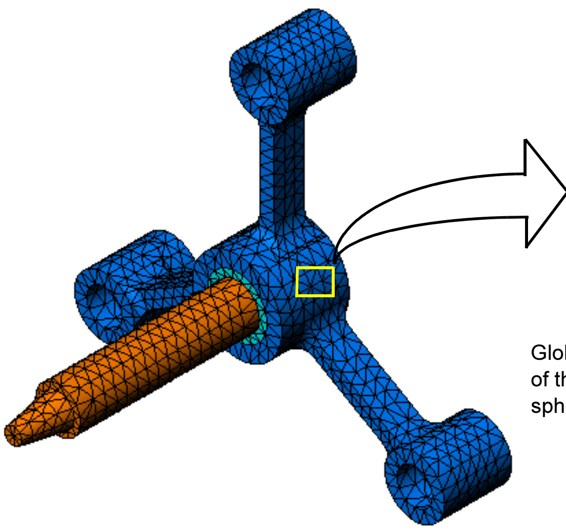
   The **Mesh** PropertyManager appears.

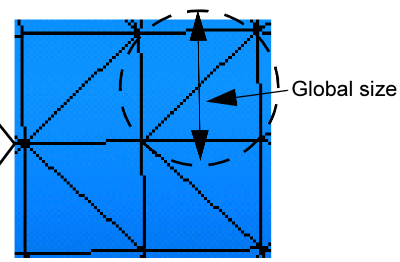
2 Expand **Mesh Parameters** by selecting the check box.

   Make sure that **Standard mesh** is selected and **Automatic transition** is not checked.

   Keep default **Global Size**  and **Tolerance**  suggested by the program.

3 Click **OK** to begin meshing.





Global element size is a measure of the average diameter of a sphere circumscribing the element.

Step 6: Running the Analysis

In the Simulation study tree, right-click the `My First Study` icon and click **Run** to start the analysis.

When the analysis completes, SolidWorks Simulation automatically creates default result plots stored in the `Results` folder.

Step 7: Visualizing the Results

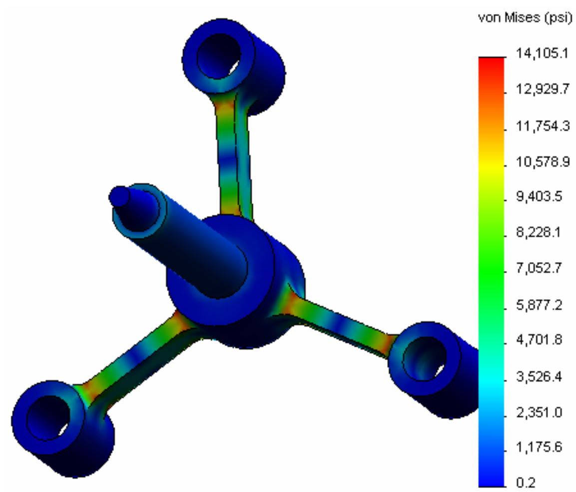
von Mises stress

1 Click the plus sign beside the `Results` folder.

All the default plots icons appear.

Note: If no default plots appear, right-click the `Results` folder and select **Define Stress plot.** Set the options in the PropertyManager and click.

2 Double-click `Stress1 (-vonMises-)` to display the stress plot.



Note: To show the annotation indicating the minimum and the maximum values in the plot, double-click the legend and check **Show min annotation** and **Show max annotation** check boxes. Then click.

Animating the Plot

1 Right-click `Stress1 (-vonMises-)` and click **Animate**.

The **Animation** PropertyManager appears and the animation starts automatically.

2 Stop the animation by clicking the **Stop** button. The animation must be stopped in order to save the AVI file on the disk.

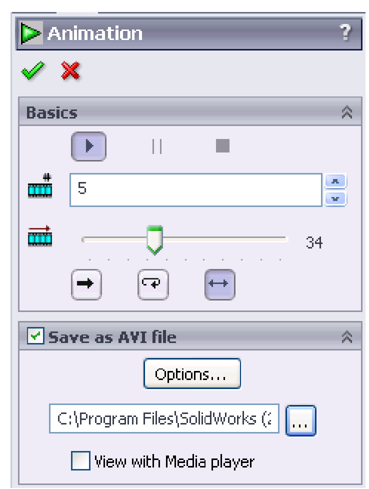
3 Check **Save as AVI File**, then click to browse and select a destination folder to save the AVI file.

4 Click to **Play** the animation.

The animation is played in the graphics area.

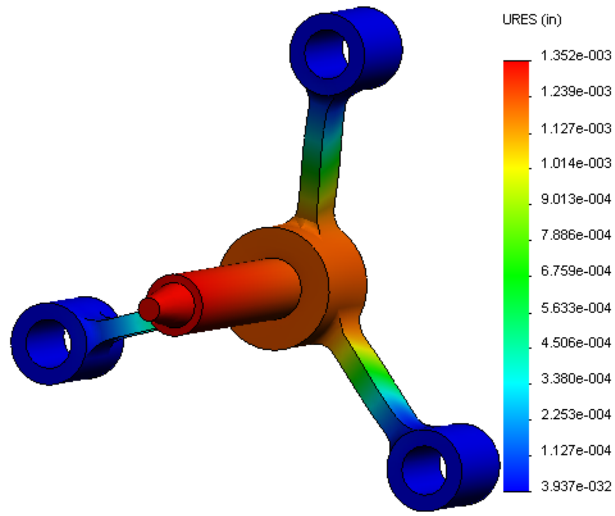
5 Click to **Stop** the animation.

6 Click to close the **Animation** PropertyManager.



Visualizing Resultant Displacements

1 Double-click `Displacement1` `(-Res disp-)` icon to display the resultant displacement plot.



Is the Design Safe?

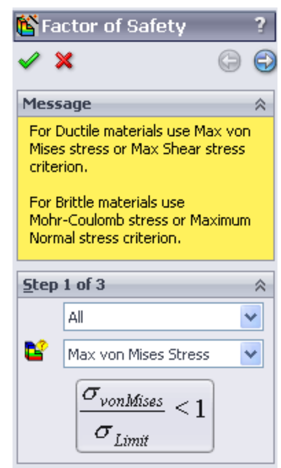
The **Factor of Safety** wizard can help you answer this question. We will use the wizard to estimate the factor of safety at every point in the model. In the process, you will need to select a yielding failure criterion.

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

Factor of Safety wizard **Step 1 of 3** PropertyManager appears.

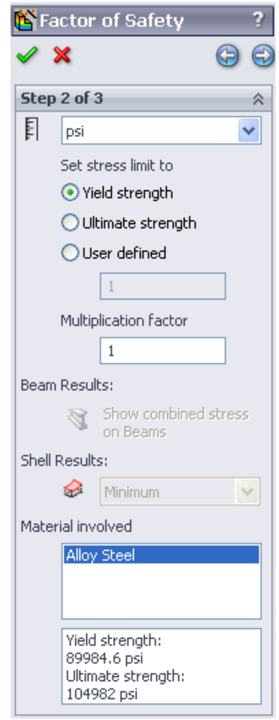
2 Under **Criterion**, click **Max von Mises stress**.

Note: Several yielding criteria are available. The von Mises criterion is commonly used to check the yielding failure of ductile materials.



3 Click **Next**.

Factor of Safety wizard **Step 2 of 3** PropertyManager appears.



4 Set **Units** to **psi**.

5 Under **Set stress limit to**, select **Yield strength.**

Note: When material yields, it continues to deform plastically at a quicker rate. In extreme case it may continue to deform even if the load is not increased.

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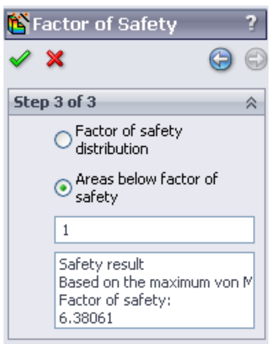
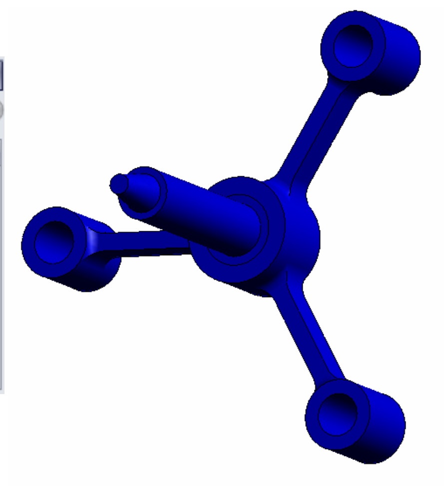
6 Click **Next.**

Factor of Safety wizard **Step 3 of 3** PropertyManager appears.

7 Select **Areas below factor of safety** and enter **1**.

8 Click to generate the plot.





Inspect the model and look for unsafe areas shown in red color. It can be observed that the plot is free from the red color indicating that all locations are safe.

How Safe is the Design?

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

Factor of Safety wizard **Step 1 of 3** PropertyManager appears.

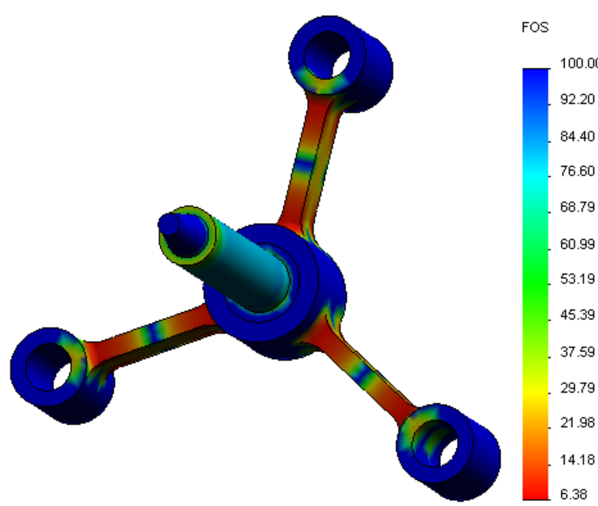
2 In the **Criterion** list, select **Max von Mises stress**.

3 Click **Next**.

Factor of Safety wizard **Step 2 of 3** PropertyManager appears.

4 Click **Next.**

Factor of Safety wizard **Step 3 of 3** PropertyManager appears.



5 Under **Plot results**, click **Factor of safety distribution**.

6 Click .

The generated plot shows the distribution of the factor of safety. The smallest factor of safety is approximately 6.4.

Note: A factor of safety of 1.0 at a location means that the material is just starting to yield. A factor of safety of 2.0, for example, means that the design is safe at that location and that the material will start yielding if you double the loads.

Saving All Generated Plots

1 Right-click `My First Study` icon and click **Save all plots as JPEG files**.

The **Browse For Folder** window appears.

2 Browse to the directory where you want to save all result plots.

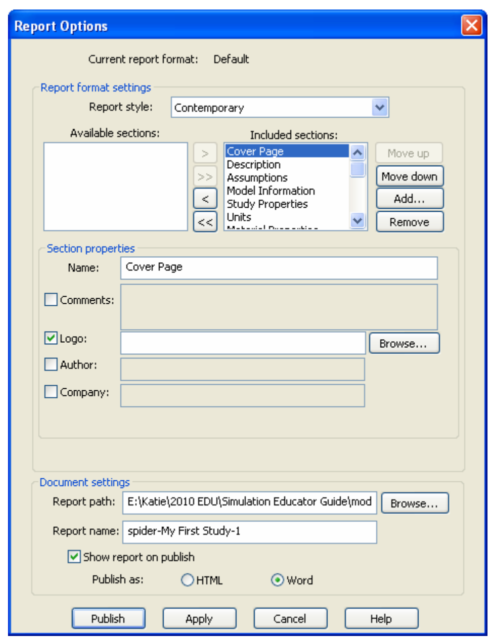
3 Click **OK**.

Generating a Study Report

The **Report** utility helps you document your work quickly and systematically for each study. The program generates structured Internet-ready reports (HTML files) and Word documents that describe all aspects related to the study.

1 Click **Simulation**, **Report** in the main SolidWorks menu on the top of the screen.

The **Report Options** dialog box appears.



The **Report format settings** section allows you to select a report style and choose sections that will be included in the generated report. You may exclude some of the sections by moving them from the **Included sections** field to the **Available sections** field.

2 Each report section can be customized. For example, select the **Cover Page** section under **Included sections** and fill the **Name, Logo, Author** and the **Company** fields.
Note that the acceptable formats for the logo files are **JPEG Files (*.jpg), GIF Files (*.gif)**, or **Bitmap Files (*.bmp)**.

3 Highlight **Conclusion** in the **Included Sections** list and enter conclusion of your study in the **Comments** box.

4 Select the **Show report on publish** check box and the **Word** option.

5 Click **Publish**.

The report opens in your word document.

Also, the program creates an icon in the `Report` folder in the SolidWorks Simulation Manager tree.

To edit any section of the report, right-click the report icon and click **Edit Definition**. Modify the section and click **OK** to replace the existing report.

Step 8: Save Your Work and Exit SolidWorks

1 Click on the **Standard** toolbar or click **File, Save**.

2 Click **File, Exit** on the main menu..

5 Minute Assessment

1 How do you start a SolidWorks Simulation session?

____

____

2 What do you do if SolidWorks Simulation menu is not on the SolidWorks menu bar?

____

____

3 What types of documents can SolidWorks Simulation analyze? ____________
____

____

4 What is analysis? ______________________________

____

5 Why is analysis important?______________________

____

6 What is an analysis study? ______________________

____

7 What types of analysis can be perfomed in SolidWorks Simulation? __________
____

8 What does static analysis calculate?_____________________

____

9 What is stress?________________________________

____

10 What are the main steps in performing analysis?__________________

____

11 How can you change the material of a part? __________________

____

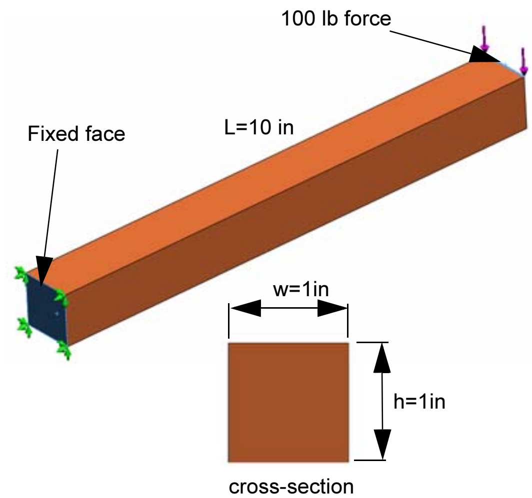
12 The Design Check wizard shows a factor of safety of 0.8 at some locations. Is your design safe? ______________________________

____

Projects — Deflection of a Beam Due to an End Force

Some simple problems have exact answers. One of these problems is a beam loaded by force at its tip as shown in the figure. We will use SolidWorks Simulation to solve this problem and compare its results with the exact solution.

Tasks



1. Open the `Front_Cantilever.sldprt` file located in the `Examples` folder of the SolidWorks Simulation installation directory.
2. Measure the width, height, and length of the cantilever.
3. Save the part to another name.
4. Create a **Static** study.
5. Assign **Alloy Steel** to the part. What is the value of the elastic modulus in psi?

   Answer: ______________________________
6. Fix one of the end faces of the cantilever.
7. Apply a downward force to the upper edge of the other end face with magnitude of **100 lb**.
8. Mesh the part and run the analysis.
9. Plot the displacement in the Y-direction. What is the maximum Y-displacement at the free end of the cantilever?

   Answer: ______________________________
10. Calculate the theoretical vertical displacement at the free end using the following formula:

$$UY_{Theory} = \frac{4FL^3}{Ewh^3}$$

where F is the force, L is the length of the beam, E is the modulus of elasticity, w and h are the width and height of the beam, respectively.

Answer: ______________________________

____

11. Calculate the error in the vertical displacement using the following formula:

$$ErrorPercentage = \left(\frac{UY_{Theory} - UY_{COSMOS}}{UY_{Theory}}\right)100$$

Answer: ______________________________

____

Lesson 1 Vocabulary Worksheet

Name________________________________Class: _________ Date:_______________

Fill in the blanks with the proper words.

1 The sequence of creating a model in SolidWorks, manufacturing a prototype, and testing it: ________________

2 A *what-if* scenario of analysis type, materials, fixtures, and loads: _________

3 The method that SolidWorks Simulation uses to perform analysis: ______________

4 The type of study that calculates displacements, strains, and stresses: ___________

5 The process of subdividing the model into small pieces: __________

6 Small pieces of simple shapes created during meshing: ___________

7 Elements share common points called: _________

8 The force acting on an area divided by that area: __________

9 The sudden collapse of slender designs due to axial compressive loads: _________

10 A study that calculates how hot a design gets: ____________

11 A number that provides a general description of the state of stress: ______________

12 Normal stresses on planes where shear stresses vanish: _________________

13 The frequencies that a body tends to vibrate in: _____________________

14 The type of analysis that can help you avoid resonance: __________________

Lesson 1 Quiz

Name: ______________________________Class: _________ Date:_______________

Directions: Answer each question by writing the correct answer or answers in the space provided.

1 You test your design by creating a study. What is a study? ______________________

____

2 What types of analyses can you perform in SolidWorks Simulation? ______________

____

3 After obtaining the results of a study, you changed the material, loads, and/or fixtures. Do you have to mesh again?____

____

4 After meshing a study, you changed the geometry. Do you need to mesh the model again? ____

____

5 How do you create a new static study? _____________________________________

____

6 What is a mesh? ____

____

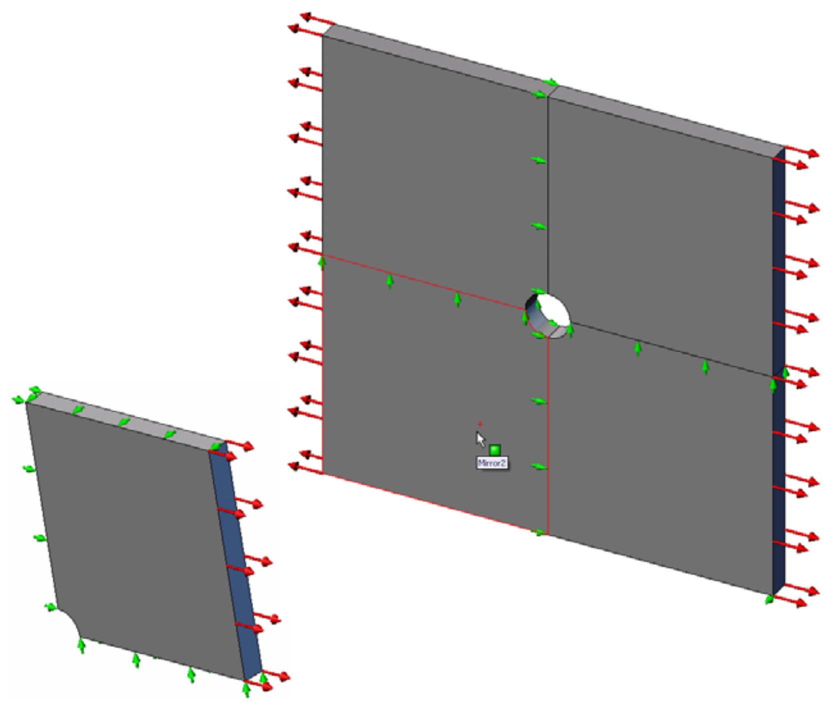
7 In an assembly, how many icons you expect to see in the `Solids` folder? ____________

____

2

Lesson 2: Adaptive Methods in SolidWorks Simulation

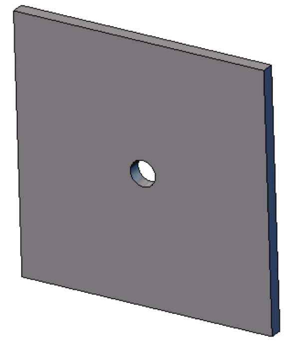
Upon successful completion of this lesson, you will be able to (a) use adaptive methods to improve accuracy of the results and (b) apply symmetry fixtures to analyze a quarter of your original model.



You will calculate the stresses of a 20 in x 20 in x 1 in square plate with a 1 inch radius hole at its center. The plate is subjected to a 100 psi tensile pressure.

You will compare the stress concentration at the hole with known theoretical results.

Active Learning Exercise — Part 1



Use SolidWorks Simulation to perform static analysis on the `Plate-with-hole.SLDPRT` part shown to the right.

You will calculate the stresses of a 20 in x 20 in x 1 in square plate with a 1 inch radius hole at its center. The plate is subjected to a 100 psi tensile pressure.

You will compare the stress concentration at the hole with known theoretical results.

The step-by-step instructions are given below.

Creating `Simulationtemp` directory

We recommend that you save the `SolidWorks Simulation Education Examples` to a temporary directory to save the original copy for repeated use.

1. Create a temporary directory named `Simulationtemp` in the `Examples` folder of the `SolidWorks Simulation` installation directory.
2. Copy the `SolidWorks Simulation Education Examples` directory into the `Simulationtemp` directory.

Opening the `Plate-with-hole.SLDPRT` Document

1. Click **Open** on the Standard toolbar. The **Open** dialog box appears.
2. Navigate to the `Simulationtemp` folder in the SolidWorks Simulation installation directory.
3. Select `Plate-with-hole.SLDPRT`.
4. Click **Open**.

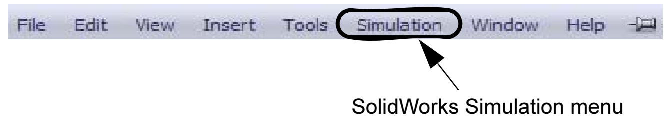
   The `Plate-with-hole.SLDPRT` part opens.

   Notice that the part has two configurations: (a) `Quarter plate`, and (b) `Whole plate`. Make sure that `Whole plate` configuration is active.

Note: The configurations of the document are listed under the ConfigurationManager tab at the top of the left pane.

Checking the SolidWorks Simulation Menu

If SolidWorks Simulation is added-in, the SolidWorks Simulation menu appears on the SolidWorks menu bar. If not:



1 Click **Tools, Add-Ins**.

The **Add-Ins** dialog box appears.

2 Check the checkboxes next to SolidWorks Simulation.

If SolidWorks Simulation is not in the list, you need to install SolidWorks Simulation.

3 Click **OK**.

The SolidWorks Simulation menu appears on the SolidWorks menu bar.

Setting the Analysis Units

Before we start this lesson, we will set the analysis units.

1 Click **Simulation, Options**.

2 Click the **Default Options** tab.

3 Select **English (IPS)** in **Unit system** and **in** and **psi** as the units for the length and stress, respectively.

4 Click .

Step 1: Creating a Study

The first step in performing analysis is to create a study.

1 Click **Simulation, Study** in the main SolidWorks menu on the top of the screen.

The **Study** PropertyManager appears.

2 Under **Name,** type `Whole plate`.

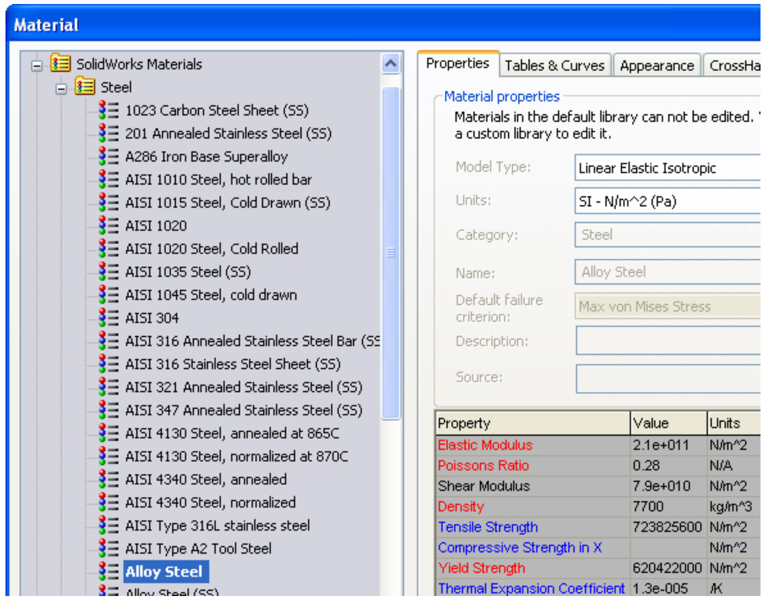
3 Under **Type**, select **Static**.

4 Click .

SolidWorks Simulation creates a Simulation study tree located beneath the FeatureManager design tree.

Step 2: Assigning Material

Assign Alloy Steel



1 In the SolidWorks Simulation Manager tree, right-click the `Plate-with-hole` folder and click **Apply Material to All Bodies**.

   The **Material** dialog box appears.

2 Do the following:

   a) Expand the `SolidWorks Materials` library folder.

   b) Expand the `Steel` category.

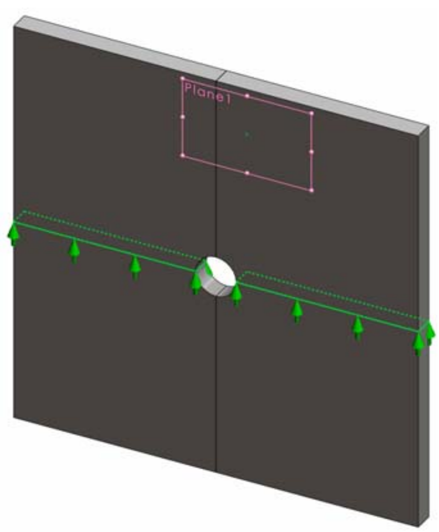
   c) Select **Alloy Stee**.

Note: The mechanical and physical properties of Alloy Steel appear in the table to the right.

3 Click **OK**.

Step 3: Applying Fixtures

You apply fixtures to prevent the out of plane rotations and free body motions.



1 Press `spacebar` and select ***Trimetric** in the **Orientation** menu.

   The model orientation is as shown in the figure.

2 In the Simulation study tree, right-click the `Fixtures` folder and click **Advanced Fixtures.**

   The **Fixture** PropertyManager appears.

3 Make sure that **Type** is set to **Use Reference Geometry**.

4 In the graphics area, select the 8 edges shown in the figure.

   `Edge<1>` through `Edge<8>` appear in the **Faces, Edges, Vertices for Fixtures** box.

5 Click in the **Face, Edge, Plane, Axis for Direction** box and select `Plane1` from the flyout FeatureManager tree.

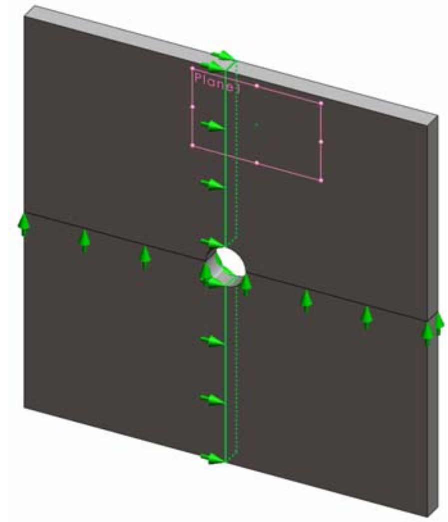
6 Under **Translations**, select **Along plane Dir 2** .

7 Click ✓.

The fixtures are applied and their symbols appear on the selected edges.

Also, a fixture icon (`Fixed-1`) appears in the `Fixtures` folder.

Similarly, you follow steps 2 to 7 to apply fixtures to the vertical set of edges as shown in the figure to restrain the 8 edges **Along plane Dir 1** of `Plane1`.



To prevent displacement of the model in the global Z-direction, a fixture on the vertex shown in the figure below must be defined.

1 In the SolidWorks Simulation Manager tree, right-click the `Fixtures` folder and click **Advanced Fixtures**.

The **Fixture** PropertyManager appears.

2 Make sure that **Type** is set to **Use reference geometry**.

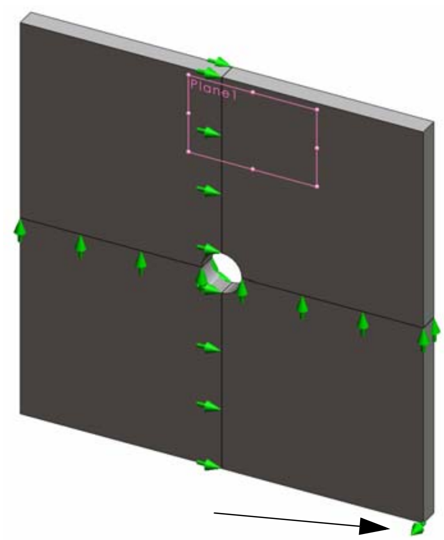
3 In the graphics area, click the vertex shown in the figure.

`Vertex<1>` appears in the **Faces, Edges, Vertices for Fixture** box.

4 Click in the **Face, Edge, Plane, Axis for Direction** box and select `Plane1` from the flyout FeatureManager tree.

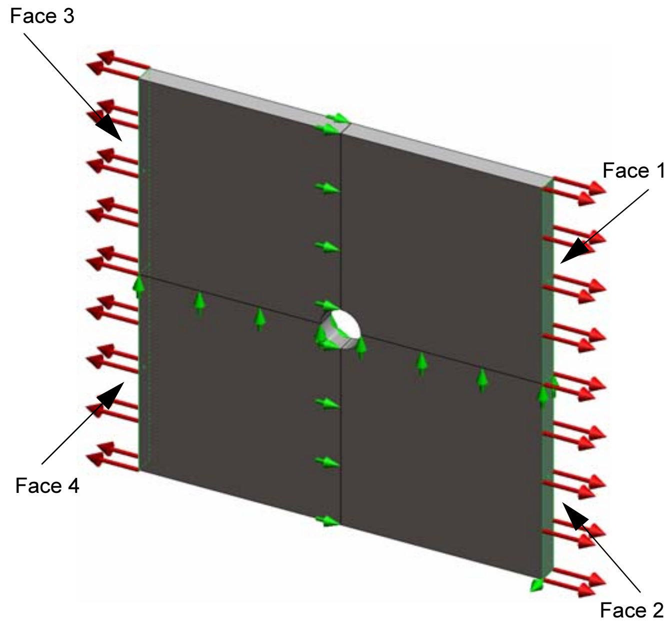
5 Under **Translations**, select **Normal to Plane** .

6 Click ✓.



Step 4: Applying Pressure

You apply a 100 psi pressure normal to the faces as shown in the figure.



1. In the SolidWorks Simulation Manager tree, right-click the `External Loads` folder and click **Pressure**.

   The **Pressure** PropertyManager appears.

2. Under **Type**, select **Normal to selected face.**
3. In the graphics area, select the four faces as shown in the figure.

   `Face<1>` through `Face<4>` appear in the **Faces for Pressure** list box.

4. Make sure that **Units** is set to **English (psi)**.
5. In the **Pressure value** box, type **100**.
6. Check the **Reverse direction** box.
7. Click .

   SolidWorks Simulation applies the normal pressure to the selected faces and `Pressure-1` icon appears in the `External Loads` folder.

To Hide Fixtures and Loads Symbols

In the SolidWorks Simulation Manager tree, right-click the `Fixtures` or `External Loads` folder and click **Hide All**.

Step 5: Meshing the Model and Running the Study

Meshing divides your model into smaller pieces called elements. Based on the geometrical dimensions of the model SolidWorks Simulation suggests a default element size which can be changed as needed.

1. In the SolidWorks Simulation Manager tree, right-click the `Mesh` icon and select **Create Mesh**.

   The **Mesh** PropertyManager appears.

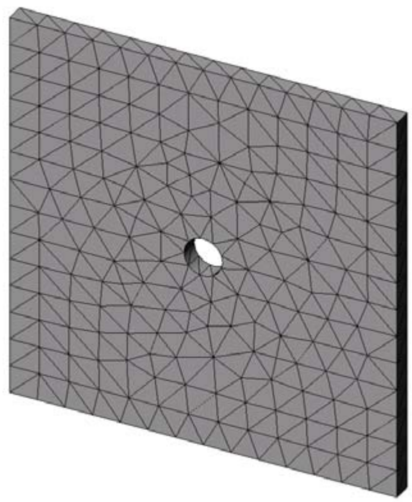
2. Expand **Mesh Parameters** by selecting the check box.

   Make sure that **Standard mesh** is selected and **Automatic transition** is not checked.

3. Type **1.5** (inches) for **Global Size** and accept the **Tolerance** suggested by the program.

4 Check **Run (solve) the analysis** under **Options** and click .

Note: To see the mesh plot, right-click Mesh folder and select **Show Mesh**



Step 6: Visualizing the Results

Normal Stress in the global X-direction.

1 Right-click the Results folder and select **Define Stress Plot.**

The **Stress Plot** PropertyManager appears.

2 Under **Display**

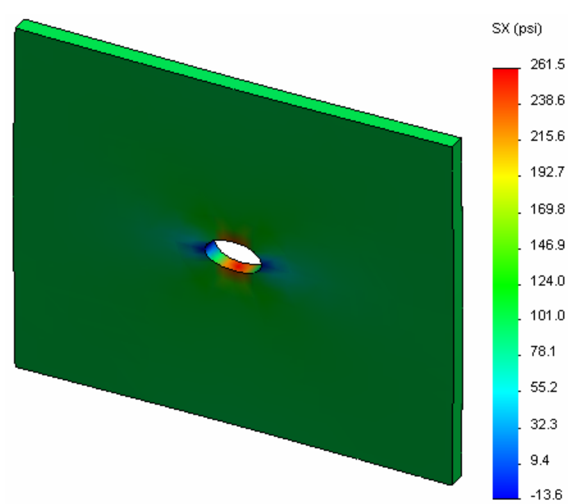
a) Select **SX: X Normal stress** in the **Component** field.

b) Select **psi** in **Units**.

3 Click .

The normal stress in the X-direction plot is displayed.

Notice the concentration of stresses in the area around the hole.



Step 7: Verifying the Results

The maximum normal stress σ_{max} for a plate with a rectangular cross section and a central circular hole is given by:

$$\sigma max = k \cdot \left(\frac{P}{t(D-2r)}\right)$$

$$k = 3.0 - 3.13\left(\frac{2r}{D}\right) + 3.66\left(\frac{2r}{D}\right)^2 - 1.53\left(\frac{2r}{D}\right)^3$$

where:

D = plate width = 20 in

r = hole radius = 1 in

t = plate thickness = 1 in

P = Tensile axial force = Pressure * (D * t)

The analytical value for the maximum normal stress is σ_{max} = 302.452 psi

The SolidWorks Simulation result, without using any adaptive methods, is SX = 253.6 psi.

This result deviates from the theoretical solution by approximately 16.1%. You will soon see that this significant deviation can be attributed to the coarsness of the mesh.

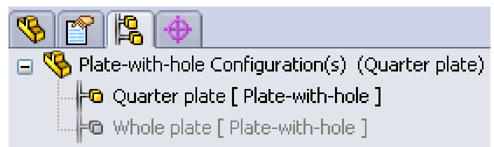
Active Learning Exercise — Part 2

In the second part of the exercise you will model a quarter of the plate with help of the symmetry fixtures.

Note: The symmetry fixtures can be used to analyze a portion of the model only. This approach can considerably save the analysis time, particularly if you are dealing with large models.

Symmetry conditions require that geometry, loads, material properties and fixtures are equal across the plane of symmetry.

Step 1: Activate New Configuration



1 Click the ConfigurationManager tab .

2 In the **Configuration Manager** tree double-click the `Quarter plate` icon.

The `Quarter plate` configuration will be activated.

The model of the quarter plate appears in the graphics area.

Note: To access a study associated with an inactive configuration right-click its icon and select **Activate SW configuration**.

Step 2: Creating a Study

The new study that you create is based on the active `Quarter plate` configuration.

1 Click **Simulation, Study** in the main SolidWorks menu on the top of the screen.

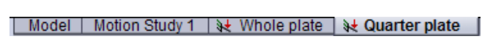
The **Study** PropertyManager appears.

2 Under **Name**, type `Quarter plate`.

3 Under **Type**, select **Static**.

4 Click .

SolidWorks Simulation creates a representative tree for the study located in a tab at the bottom of the screen.



Step 3: Assigning Material

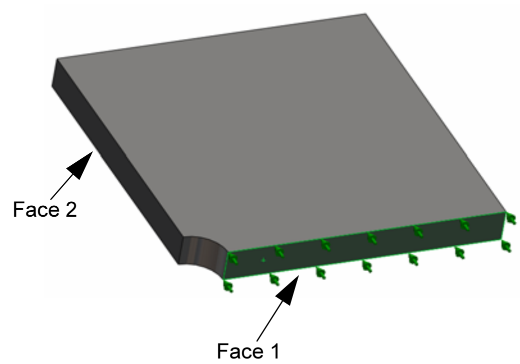
Follow the procedure described in Step 2 of Part 1 to assign **Alloy Steel** material.

Step 4: Applying Fixtures

You apply fixtures on the faces of symmetry.

1. Use the **Arrow** keys to rotate the model as shown in the figure.
2. In the Simulation study tree, right-click the `Fixtures` folder and select **Advanced Fixtures**.

   The **Fixtures** PropertyManager appears.
3. Set **Type** to **Symmetry**.
4. In the graphics area, click the `Face 1` and `Face 2` shown in the figure.



   `Face<1>` and `Face<2>` appear in the **Faces, Edges, Vertices for Fixture** box.
5. Click ✓.

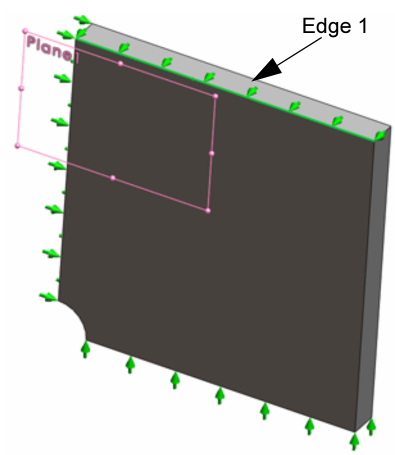
   Next you fixture the upper edge of the plate to prevent the displacement in the global Z-direction.

To restrain the upper edge:

1. In the SolidWorks Simulation Manager tree, right-click the `Fixtures` folder and select **Advanced Fixtures**.

   Set **Type** to **Use reference geometry.**
2. In the graphics area, click the upper edge of the plate shown in the figure.

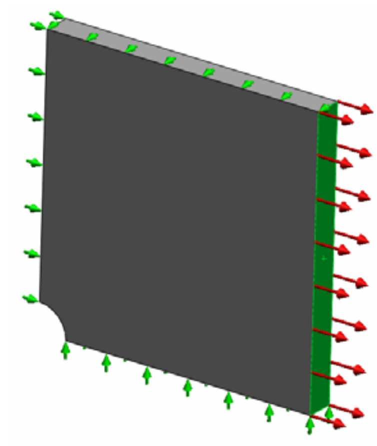
   `Edge<1>` appears in the **Faces, Edges, Vertices for Fixture** box.
3. Click in the **Face, Edge, Plane, Axis for Direction** box and select `Plane1` from the flyout FeatureManager tree.
4. Under **Translations**, select **Normal to plane**. Make sure the other two components are deactivated.
5. Click ✓**.**



After applying all fixtures, three items: `(Symmetry-1)` and `(Reference Geometry-1)` appear in the `Fixtures` folder.

Step 5 Applying Pressure

You apply a 100 psi pressure as shown in the figure below:



1 In the SolidWorks Simulation Manager tree, right-click `External Loads` and select **Pressure**.

The **Pressure** PropertyManager appears.

2 Under **Type**, select **Normal to selected face.**

3 In the graphics area, select the face shown in the figure.

1 `Face<1>` appears in the **Faces for Pressure** list box.

2 Set **Units** to **psi**.

3 In the **Pressure value** box, type **100**.

4 Check the **Reverse direction** box.

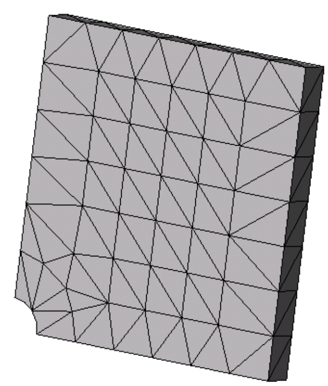
5 Click .

SolidWorks Simulation applies the normal pressure to the selected face and `Pressure-1` icon appears in the `External Loads` folder.

Step 6 Meshing the Model and Running the Analysis

Apply the same mesh settings following the procedure described in Step 5 of Part 1, Meshing the Model and Running the Study on page 2-7. Then **Run** the analysis.

The mesh plot is as shown in the figure.



Step 7 Viewing Normal Stresses in the Global X- Direction

1 In the Simulation study tree, right-click the `Results` folder and select **Define Stress Plot**.

2 In the **Stress Plot** PropertyManager, under **Display**:

a) Select **SX:X Normal stress.**

b) Select **psi** in **Units.**

3 Under **Deformed Shape** select **True Scale**.

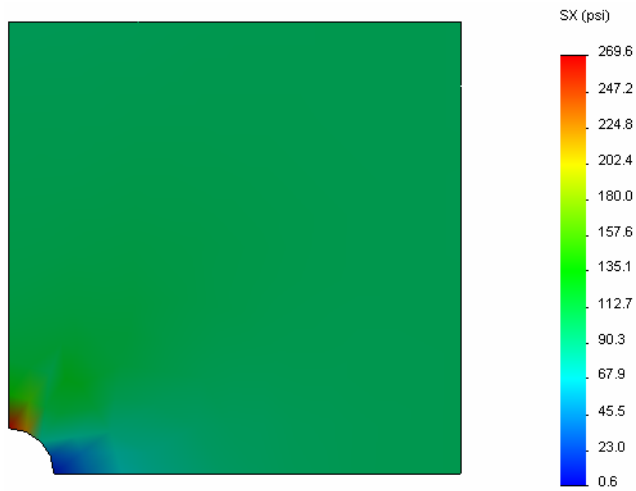
4 Under **Property:**

a) Select **Associate plot with name view orientation**.

b) Select ***Front** from the menu.

5 Click .

The normal stress in the X-direction is displayed on the real deformed shape of the plate.



Step 8 Verifying the Results

For the quarter model, the maximum normal SX stress is 269.6 psi. This result is comparable to the results for the whole plate.

This result deviates from the theoretical solution by approximately 10.8%. As was mentioned in the conclusion of Part 1 of this lesson, you will see that this deviation can be attributed to the coarsness of the computational mesh. You can improve the accuracy by using a smaller element size manually or by using automatic adaptive methods.

In Part 3 you will use the h-adaptive method to improve the accuracy.

Active Learning Exercise — Part 3

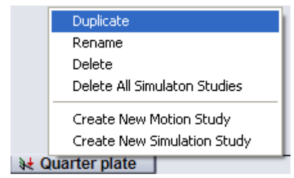
In the third part of the exercise you will apply the h-adaptive method to solve the same problem for the `Quarter plate` configuration.

To demonstrate the power of the h-adaptive method, first, you will mesh the model with a large element size, and then you will observe how the h-method changes the mesh size to improve the accuracy of the results.

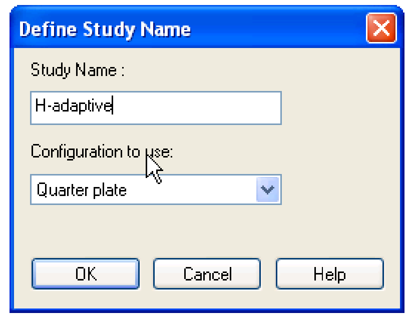
Step 1 Defining a New Study

You will create a new study by duplicating the previous study.

1 Right-click the `Quarter plate` study at the bottom of the screen and select **Duplicate**.



The **Define Study Name** dialog box appears.



2 In the **Study Name** box, type `H-adaptive.`

3 Under **Configuration to use**: select **Quarter plate.**

4 Click **OK**.

Step 2 Setting the h-adaptive Parameters

1 In the Simulation study tree, right-click `H-adaptive` and select **Properties.**

2 In the dialog box, in the **Options** tab, select **FFEPlus** under **Solver**.

3 In the **Adaptive** tab, under **Adaptive method**, select **h-adaptive**.

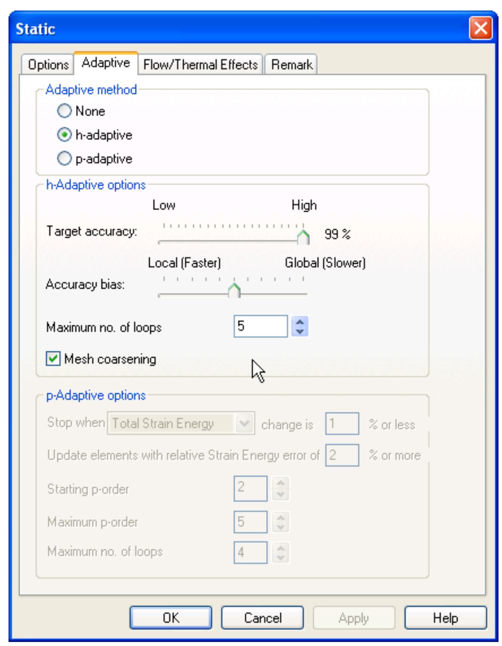
4 Under **h-Adaptive options**, do the following:

a) Move the **Target accuracy slider** to **99%**.

b) Set **Maximum no. of loops** to **5**.

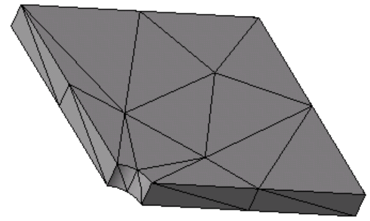
c) Check **Mesh coarsening.**

5 Click **OK**.



Note: By duplicating the study, all the folders of the original study are copied to the new study. As long as the properties of the new study remain the same, you do not need to redefine material properties, loads, fixtures, etc.

Step 3: Remeshing the Model and Running the Study



1 In the SolidWorks Simulation Manager tree, right-click the `Mesh` folder and select **Create Mesh**.

   A warning message appears stating that remeshing will delete the results of the study.

2 Click **OK**.

   The **Mesh** PropertyManager appears

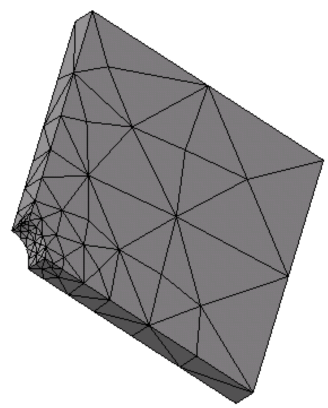
3 Type **5.0** (inches) for **Global Size** and accept the **Tolerance** suggested by the program.

   This large value for the global element size is used to demonstrate how the h-adaptive method refines the mesh to get accurate results.

4 Click. The image above shows the initial coarse mesh.

5 Right-click the **H-adaptive** icon and select **Run.**

Step 4: Viewing Results

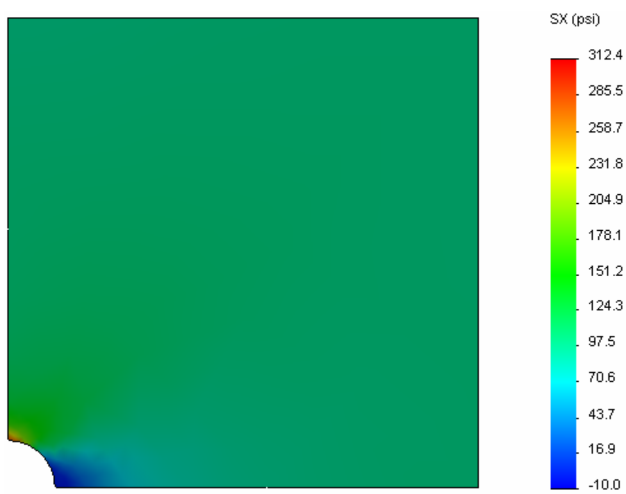


With the application of the h-adaptive method the original mesh size is reduced. Notice the transition of the mesh size from a coarser mesh (plate boundaries) to a finer mesh at the location of the central hole.

To view the converted mesh, right-click the `Mesh` icon and select **Show Mesh**.

View normal stress in the global X-direction

In the SolidWorks Simulation Manager tree, double-click the **Stress2 (X-normal)** plot in the `Results` folder.



The analytical value for the maximum normal stress is σ_{max}= 302.452 psi.

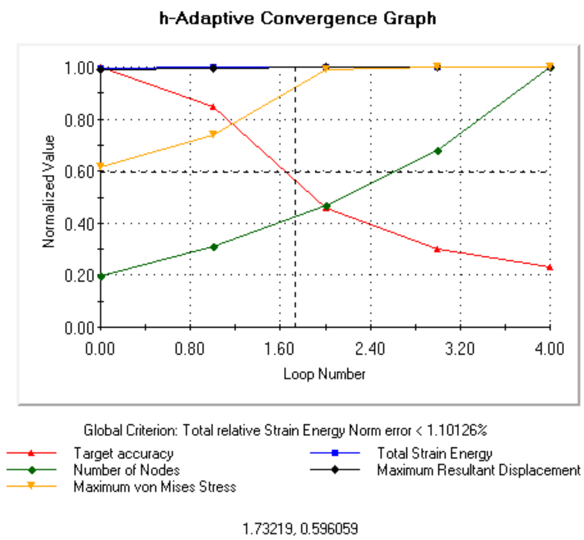
The SolidWorks Simulation result with the application of the h-adaptive method is SX = 312.4 psi, which is closer to the analytical solution (approximate error: 3.2%).

Note: The desired accuracy set in the study properties (in your case 99%) does not mean that the resulting stresses will be within the maximum error of 1%. In finite element method measures other than stresses are used to evaluate the accuracy of the solution. However, it can be concluded that as the adaptive algorithm refines the mesh, the stress solution becomes more accurate.

Step 9 Viewing Convergence Graphs

1. In the Simulation study tree , right-click the `Results` folder and select **Define Adaptive Convergence Graph**.
2. In the PropertyManager, check all options and click .

   The convergence graph of all checked quantities is displayed.



Note: To further improve the accuracy of the solution, it is possible to continue with the h-adaptivity iterations by initiating subsequent study runs. Each subsequent study run uses the final mesh from the last iteration of the previous run as the initial mesh for the new run. To try this **Run** the `H-adaptive` study again.

5 Minute Assessment

1 If you modify material, loads or fixtures, the results get invalidated while the mesh does not, why?

____

____

2 Does changing a dimension invalidate the current mesh? ________________

____

____

3 How do you activate a configuration? ______________________

____

____

4 What is a rigid body motion?______________________________

____

____

5 What is the h-adaptive method and when is it used? ______________

____

____

6 What is the advantage of using h-adaptive to improve the accuracy compared to using mesh control?

____

____

7 Does the number of elements change in iterations of the p-adaptive method?

____

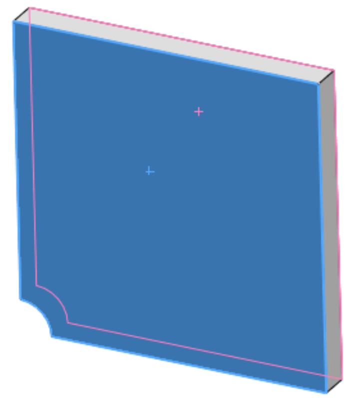
____

Projects — Modeling the Quarter Plate with a Shell Mesh

Use shell mesh to solve the quarter plate model. You will apply mesh control to improve the accuracy of the results.

Tasks

1. Click **Insert, Surface, Mid Surface** in the main SolidWorks menu on the top of the screen.
2. Select the front and back surfaces of the plate as shown.

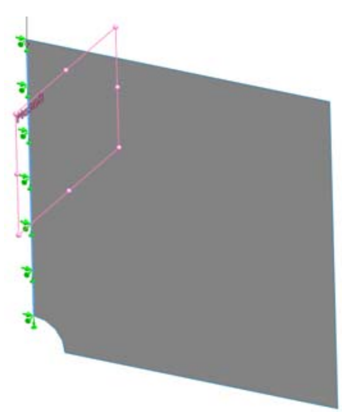


3. Click **OK**.
4. Create a **Static** study named `Shells-quarter`.
5. Expand the `Plate-with-hole` folder, right-click the `SolidBody` and select **Exclude from Analysis**.
6. In the FeatureManager design tree, expand the `Solid Bodies` folder and **Hide** the existing solid body.
7. Define **1 in** (**Thin** formulation) shell. To do this:
   a) Right-click the `SurfaceBody` in the `Plate-with-hole` folder of the Simulation study tree and select **Edit Definition**.
   b) In the **Shell Definition** PropertyManager, select **in** and type **1** in for **Shell thickness.**
   c) Click .
8. Assign **Alloy Steel** to the shell. To do this:
   a) Right-click the `Plate-with-hole` folder and select **Apply Material to All Bodies**.
   b) Expand `SolidWorks Materials` library and select **Alloy Steel** from the `Steel` category.
   c) Select **Apply** and **Close**.
9. Apply symmetry fixtures to the two edges shown in the figure.

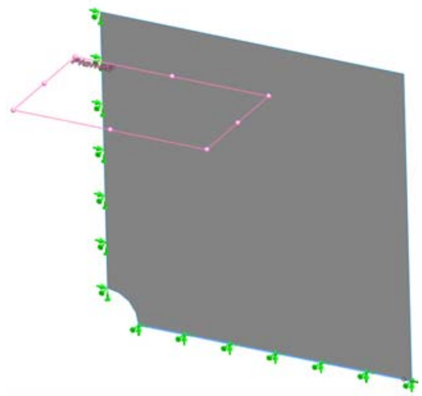
Note: For a shell mesh, it is sufficient to restrain one edge instead of the face.

---

   a) Right-click the `Fixtures` folder and select **Advanced Fixtures**.
   b) In the **Faces, Edges, Vertices for Fixture** field select the edge indicated in the figure.
   c) In the **Face, Edge, Plane, Axis for Direction** field select `Plane3`.
   d) Restrain the **Normal to Plane** translation and **Along Plane Dir 1** and **Along Plane Dir 2** rotations.
   e) Click .



10 Using the identical procedure apply a symmetry fixture to the other edge shown in the figure. This time use `Plane2` feature for **Face, Edge, Plane, Axis for Direction** field.



11 Apply **100 psi Pressure** to the edge shown in the figure.

a) Right-click the `External Loads` folder and select **Pressure**.

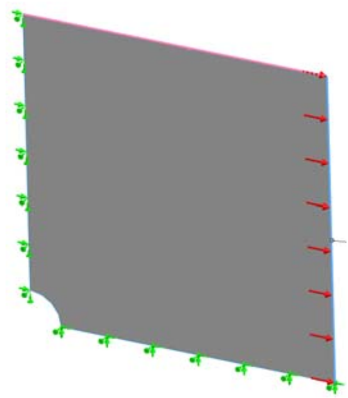
b) Under **Type** select **Use reference geometry**.

c) In the **Faces, Edges for Pressure** field select the vertical edge shown in the figure.

d) In the **Face, Edge, Plane, Axis for Direction** field select the edge indicated in the figure.

e) Specify **100 psi** in the **Pressure Value** dialog.

f) Click .

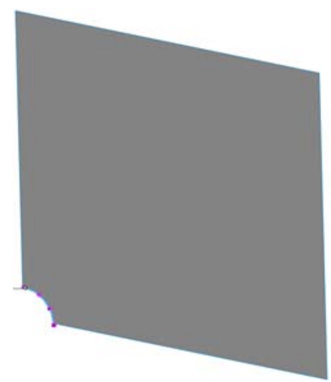


12 Apply mesh control to the edge shown in the figure.Using a smaller element size improves the accuracy.

a) In the Simulation study tree, right-click the `Mesh` icon and select **Apply Mesh Control**. The **Mesh Control** PropertyManager appears.

b) Select the edge of the hole as shown in the figure.

c) Click .



13 Mesh the part and run the analysis.

a) In the Simulation study tree, right-click the `Mesh` icon and select **Apply Mesh Control**. The **Mesh Control** PropertyManager appears.

b) Select the edge of the hole as shown in the figure.

c) Click .

14 Plot the stress in the X-direction. What is the maximum SX stress?

Answer: ________________________________

15 Calculate the error in the normal SX stress using the following formula:

$$ErrorPercentage = \left(\frac{SX_{Theory} - SX_{SIMULATION}}{SX_{Theory}}\right)100$$

Answer: ___________________________________

____

____

Lesson 2 Vocabulary Worksheet

Name________________________________Class: _________ Date:_______________

Fill in the blanks with the proper words.

1 A method that improves stress results by refining the mesh automatically in regions of stress concentration:

____

2 A method that improves stress results by increasing the polynomial order:

____

3 The type of degrees of freedom that a node of a tetrahedral element has:

____

4 The types of degrees of freedom that a node of a shell element has:

____

5 A material with equal elastic properties in all directions:

____

6 The mesh type appropriate for bulky models:

____

7 The mesh type appropriate for thin models:

____

8 The mesh type appropriate for models with thin and bulky parts:

____

Lesson 2 Quiz

Name: ______________________________ Class: _________ Date: _______________

Directions: Answer each question by writing the correct answer or answers in the space provided.

1 How many nodes are there in draft and high quality shell elements?

____

____

2 Does changing the thickness of a shell require remeshing?

____

____

3 What are adaptive methods and what is the basic idea for their formulation?

____

____

____

4 What is the benefit in using multiple configurations in your study?

____

____

5 How can you quickly create a new study that has small differences from an existing study?

____

____

6 When adaptive methods are not available, what can you do to build confidence in the results?____

____

7 In which order does the program calculate stresses, displacements, and strains?

____

____

8 In an adaptive solution, which quantity converges faster: displacement, or stress?

____

____