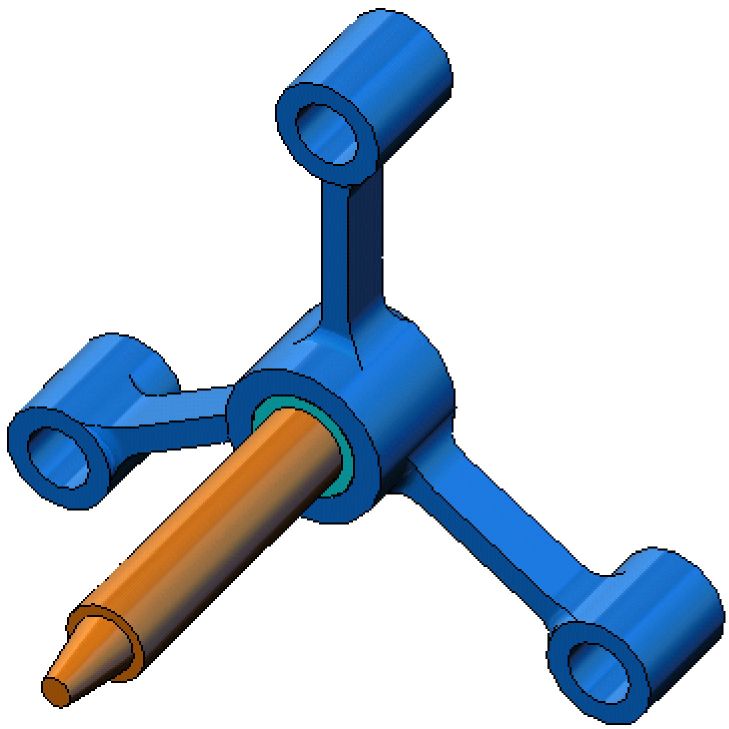
Engineering Design and Technology Series

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



Dassault Systèmes SolidWorks Corporation
300 Baker Avenue
Concord, Massachusetts 01742 USA
Phone: +1-800-693-9000

Outside the U.S.: +1-978-371-5011
Fax: +1-978-371-7303
Email: info@solidworks.com
Web: http://www.solidworks.com/education

Document Number: PME0218-ENG

i

Introduction

To the Instructor

This document introduces SolidWorks users to the SolidWorks Simulation software package. The specific goals of this lesson are to:

1. introduce the basic concepts static structural analysis and its benefits.
2. demonstrate the ease of use and the concise process for performing these analyses.
3. introduce the basic rules for static analyses and how to obtain reliable and accurate results.

This document is structured similar to lessons in the SolidWorks Instructor Guide. This lesson has corresponding pages in the *SolidWorks Simulation Student Workbook*.

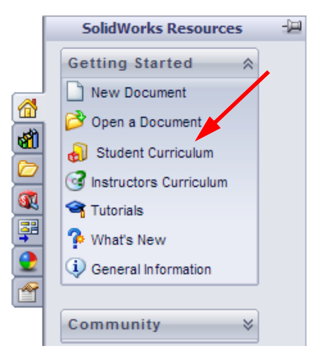
Note: This lesson does not attempt to teach all capabilities of SolidWorks Simulation. It only intends to introduce the basic concepts and rules of performing linear static analyses and to show the ease of use and the concise process of doing so.

Education Edition Curriculum and Courseware DVD

An *Education Edition Curriculum and Courseware* DVD is provided with this course.

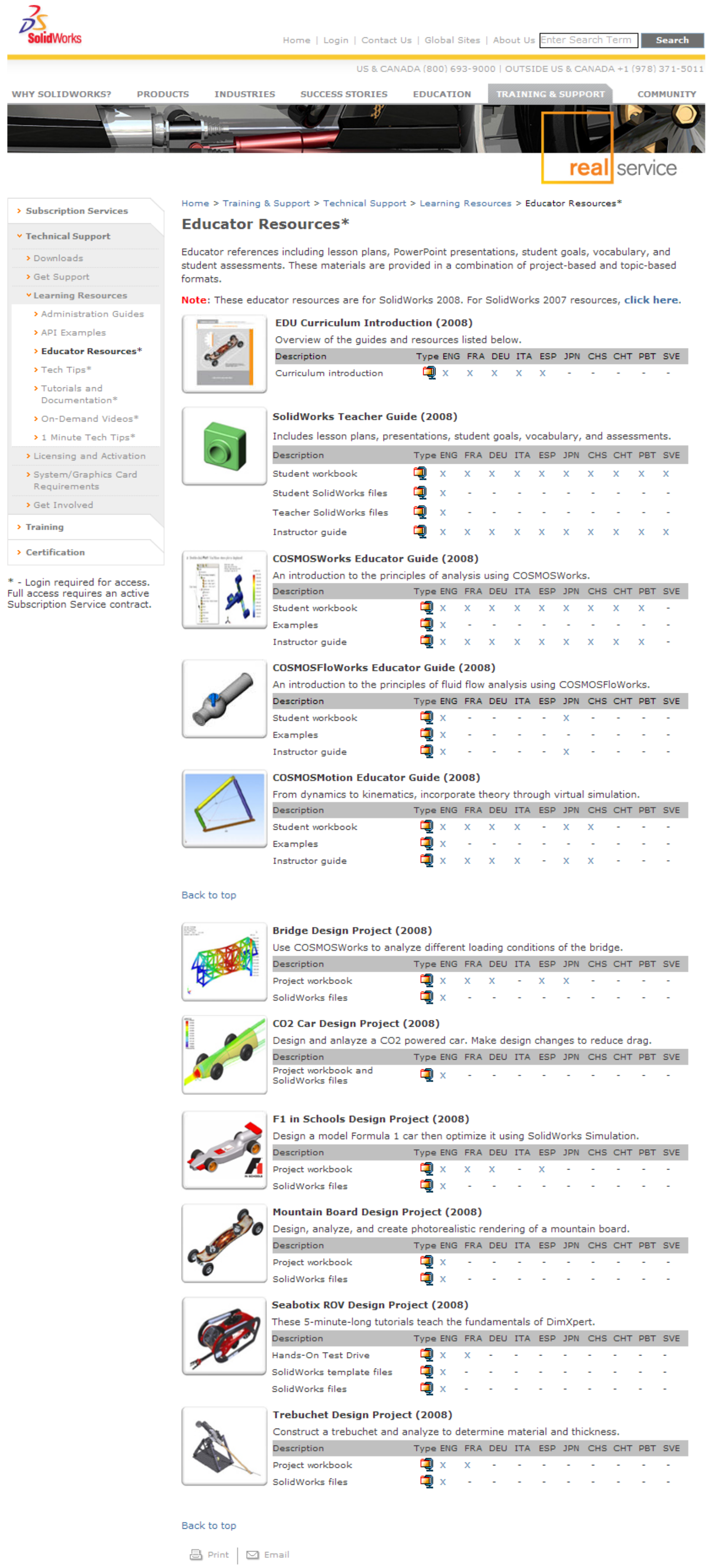
Installing the DVD creates a folder named `SolidWorks Curriculum_and_Courseware_2010`. This folder contains directories for this course and several others.

Course material for the students can also be downloaded from within SolidWorks. Click the SolidWorks Resources tab in the Task Pane and then select Student Curriculum.



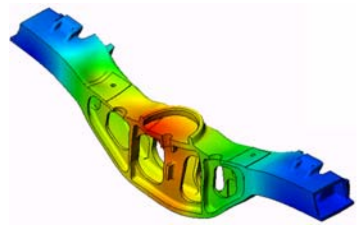
Double-click the course you would like to download. Control-select the course to download a ZIP file. The `Lessons` file contains the parts needed to complete the lessons. The `Student Guide` contains the PDF file of the course.

Course material for teachers can also be downloaded from the SolidWorks web site. Click the SolidWorks Resources tab in the Task Pane and then select Instructors Curriculum. This will take you to the Educator Resources page shown below.

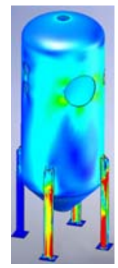


SolidWorks Simulation Product Line

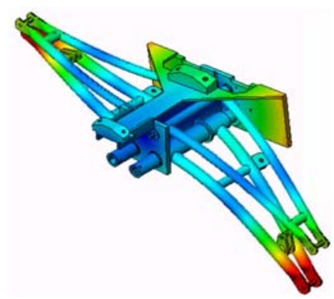
While this course focuses on the introduction to the static linear simulation of elastic bodies using SolidWorks Simulation, the full product line covers a wide range of analysis areas to consider. The paragraphs below lists the full offering of the SolidWorks Simulation packages and modules.



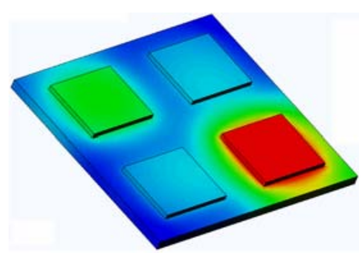
Static studies provide tools for the linear stress analysis of parts and assemblies loaded by static loads. Typical questions that will be answered using this study type are:
Will my part break under normal operating loads?
Is the model over-designed?
Can my design be modified to increase the safety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



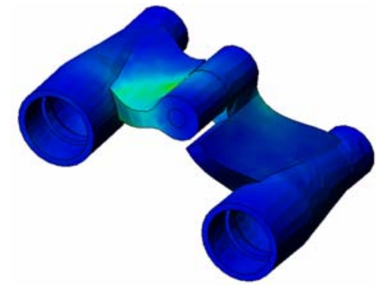
Buckling studies analyze performance of the thin parts loaded in compression. Typical questions that will be answered using this study type are:
Legs of my vessel are strong enough not to fail in yielding; but are they strong enough not to collapse due to loss of stability?
Can my design be modified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



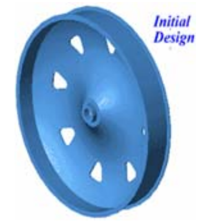
Frequency studies offer tools for the analysis of the natural modes and frequencies. This is essential in the design or many components loaded in both static and dynamic ways. Typical questions that will be answered using this study type are:
Will my part resonate under normal operating loads?
Are the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



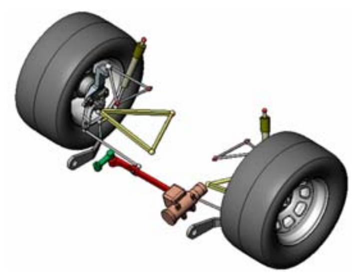
Thermal studies offer tools for the analysis of the heat transfer by means of conduction, convection, and radiation. Typical questions that will be answered using this study type are:
Will the temperatures changes effect my model?
How does my model operate in an environment with temperature fluctuation?
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?

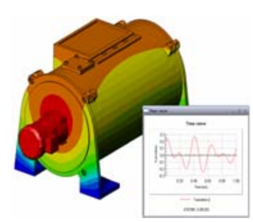


Optimization studies are applied to improve (optimize) your initial design based on a set of selected criteria such as maximum stress, weight, optimum frequency, etc. Typical questions that will be answered using this study type are:
Can the shape of my model be changed while maintaining the design intent?
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?

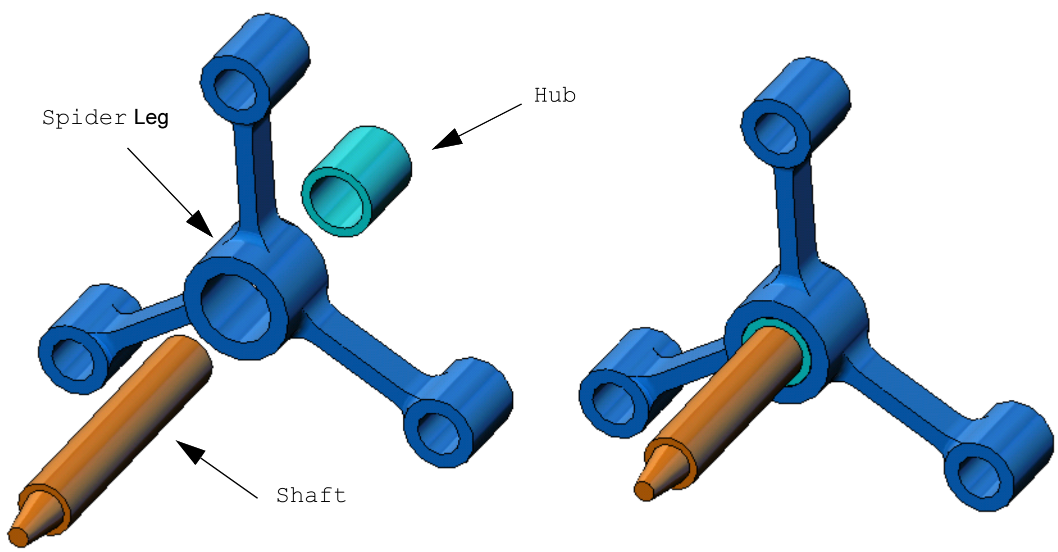
1

Lesson 1: Basic Functionality of SolidWorks Simulation

Goals of This Lesson

- Introduce design analysis as an essential tool to compliment 3D modeling using SolidWorks. Upon successful completion, the students should be able to understand the basic concepts of design analysis and how SolidWorks Simulation implements them. The students should see how analysis can save time and money by reducing time-consuming and expensive design cycles.
- Introduce design analysis using an Active Learning Exercise. The Active Learning Exercise in this lesson is designed to break the ice by having the students go through few steps to complete an analysis. With this concept in mind, the steps are performed with minimal description.
- Introduce the concept of meshing the model. The generated mesh depends on the active meshing preferences. These options are not explained in this lesson. The lesson goes through setting meshing options so that all students get a similar mesh and consequently similar results. A description of these options is available by clicking the Help button in the PropertyManager where they are specified.

  The results of analysis may slightly vary depending on versions/builds of SolidWorks and SolidWorks Simulation.



Outline

- In Class Discussion
- Active Learning Exercise — Performing Static Analysis
  - Opening the `spider.SLDASM` Document
  - Checking the SolidWorks Simulation Menu
  - Switching to SolidWorks Simulation Manager
  - Setting the Analysis Units
  - Step 1: Creating a Static Study
  - Step 2: Assigning Materials
  - Step 3: Applying Fixtures
  - Step 4: Applying Loads
  - Step 5: Meshing the Assembly
  - Step 6: Running the Analysis
  - Step 7: Visualizing the Results
  - Visualizing von Mises Stress
  - Animating the Plot
  - Visualizing Resultant Displacements
  - Is the Design Safe?
  - How Safe Is the Design?
  - Generating a Study Report
  - Saving Your Work and Exiting SolidWorks
- 5 Minute Assessment
- In Class Discussion-Changing Material Assignments
- More to Explore-Modifying the Geometry
- Exercises and Projects-Deflection of a Beam Due to an End Force
- Lesson Summary

In Class Discussion

Ask the students to identify objects around them and what loads and fixtures to specify. For example, ask the students to estimate the stress on the legs of their chair.

Answer

- ❑ Stress is force per unit area or force divided by area. The legs support the weight of the student plus the weight of the chair. The chair design and how the student is sitting determine the share of each leg. The average stress is the weight of the student plus the weight of the chair divided by the area of the legs.

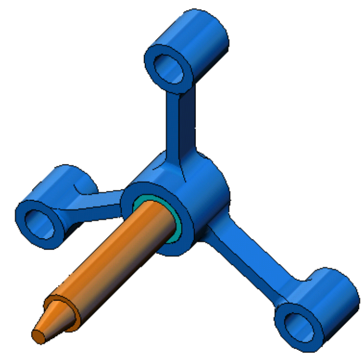
More to explore

The purpose of this section is to encourage students to think about the applications of stress analysis. Ask the students to estimate the stress on their feet when they stand up. Is the stress the same at all points? What happens if the student leans forward, backward, or to the side? How about the stress on the knee and ankle joints? Is this information useful in designing artificial joints?

Answer

- ❑ Stress is force per unit area or force divided by area. The force is the weight of the student. The area that supports the weight is the area of the foot in contact with the shoes. The shoes redistribute the load and transmit it to the floor. The reaction force from the floor should be equal to the student's weight.
- ❑ When standing upright, each foot approximately takes half the weight. When walking, one foot supports the whole weight. The student could feel that the stress (pressure) is higher at some points. When standing upright, the students can move their toes indicating that there is little or no stress on the toes. As the students lean forward, the stress is redistributed with more stress on the toes and less on the heel. The average stress is the weight divided by the area of the feet in contact with the shoes.
- ❑ We can estimate the average stresses on the knee and ankle joints if we know the area that carry the weight. Detailed results require performing stress analysis. If we can build the knee or ankle joint assembly in SolidWorks with the proper dimensions and if we know the elastic properties of the various parts, then static analysis can give us the stresses at every point of the joint under different support and load scenarios. The results can help us improve designs for artificial joint replacements.
- ❑ Students may ask whether SolidWorks Simulation can model bones. The answer is yes and some problems of this type have been solved by SolidWorks Simulation users and used to design artificial joint replacements.

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.

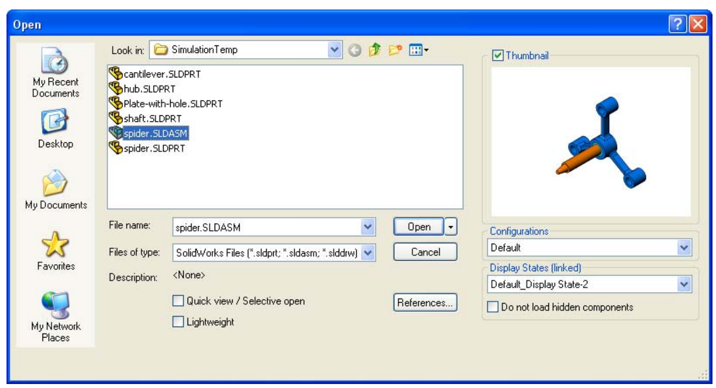
Creating a `SimulationTemp` directory

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.

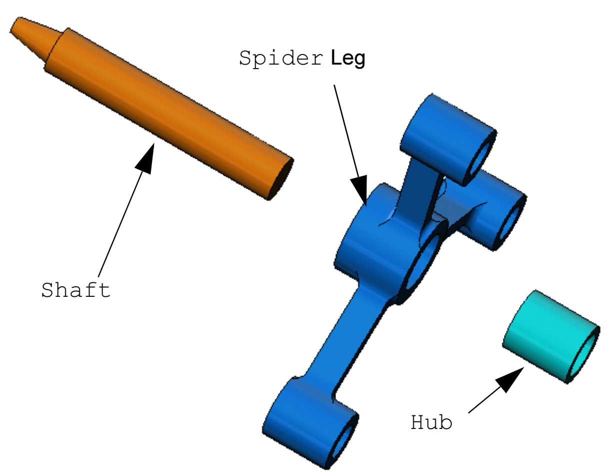
2 Navigate to the `SimulationTemp` folder in the `SolidWorks Simulation` installation directory.

3 Select `Spider.SLDASM`

4 Click **Open**.

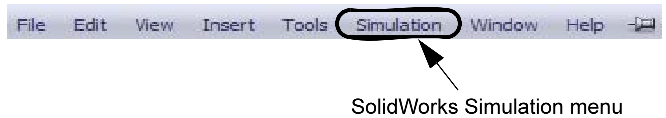
The `spider.SLDASM` assembly opens.

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:



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.

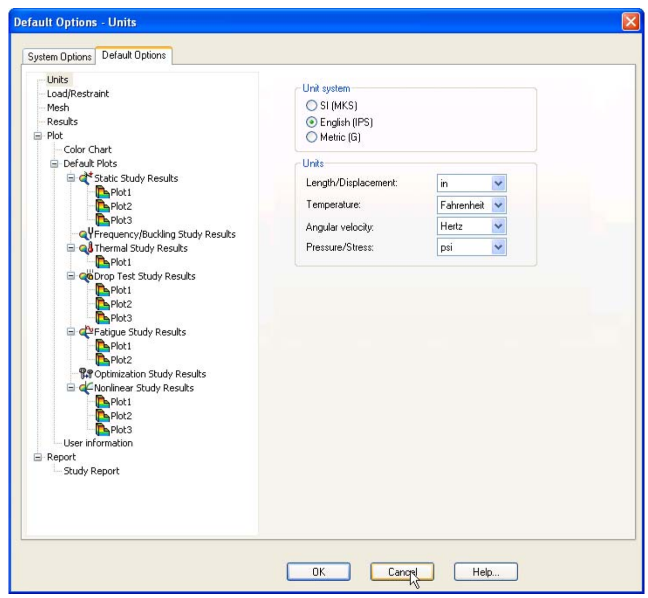
1 On the SolidWorks menu bar click **Simulation**, **Options**.

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.

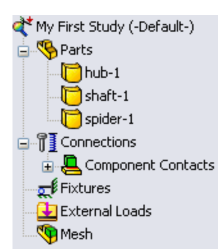
The **Study** PropertyManager appears.

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

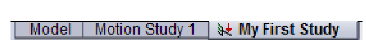
3 Under **Type**, select **Static**.

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**.

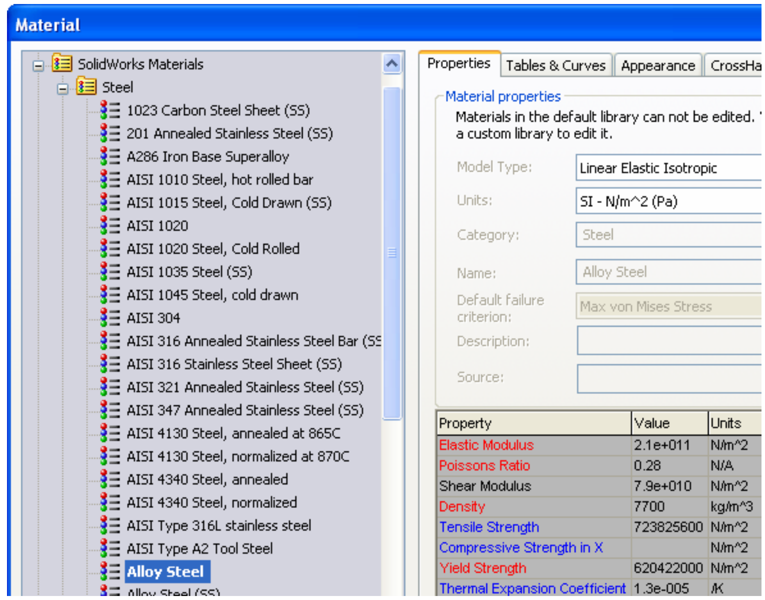
The `Material` dialog box appears.

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**.

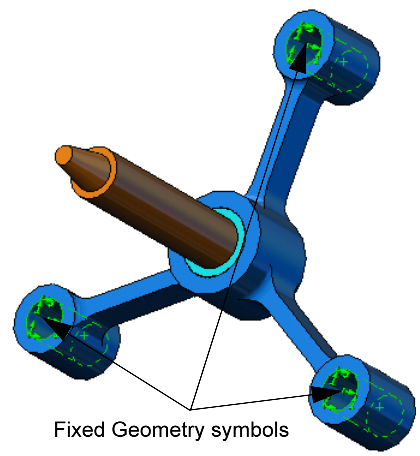
4 In the graphics area, click the faces of the three holes, indicated in the figure below.

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

5 Click ✓.

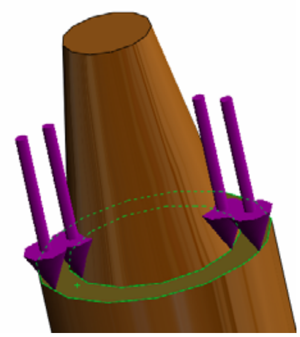
Fixed fixture is applied and its symbols appear on the selected faces.

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**.

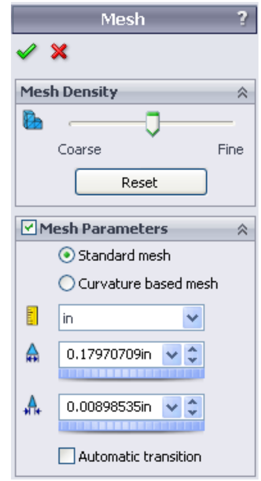
7 Click ✓.

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**.

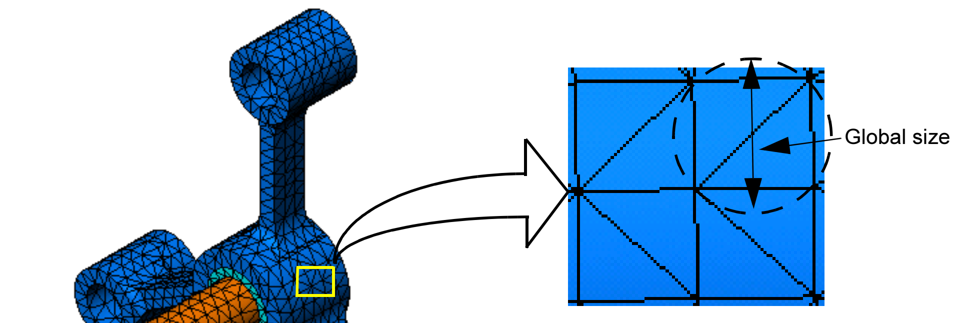
   The **Mesh** PropertyManager appears.

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

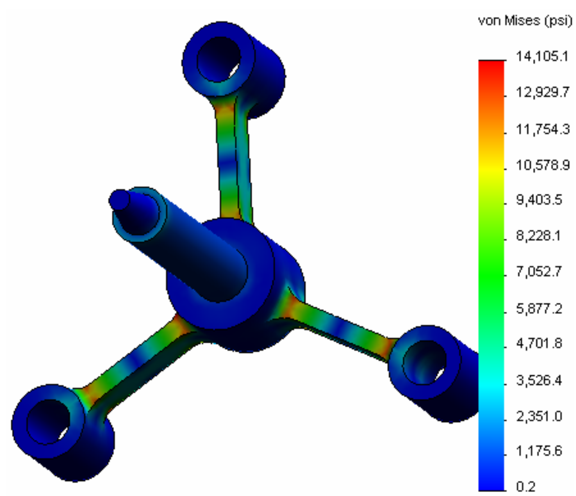
von Mises stress

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.

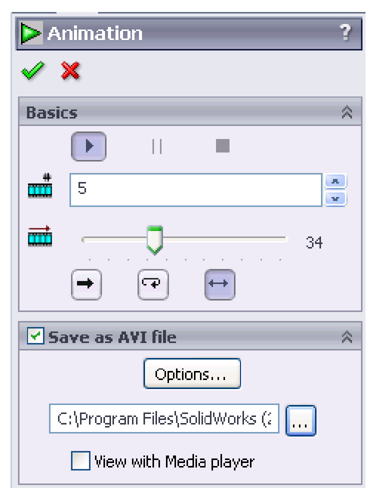
3 Check **Save as AVI File**, then click to browse and select a destination folder to save the AVI file.

4 Click to **Play** the animation.

The animation is played in the graphics area.

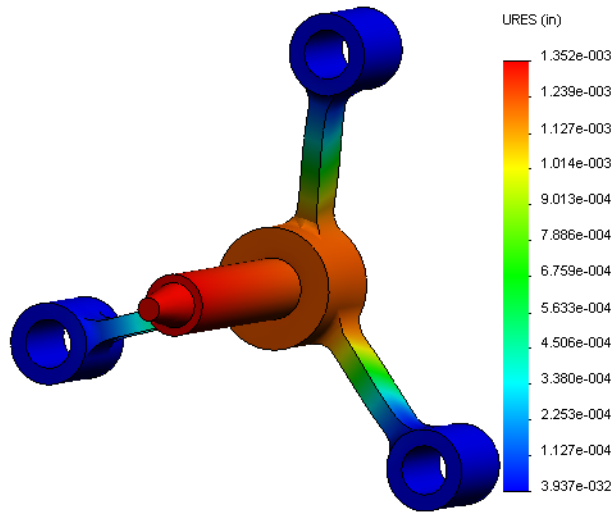
5 Click to **Stop** the animation.

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



Visualizing Resultant Displacements

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



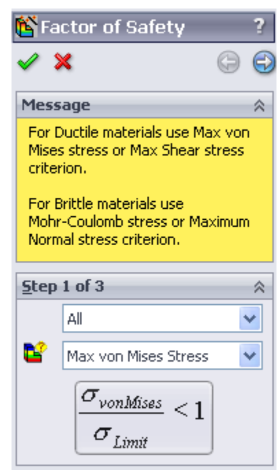
Is the Design Safe?

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.

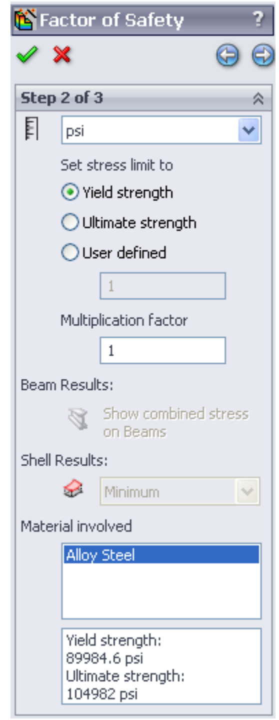
1 Right-click the `Results` folder and select **Define Factor of Safety Plot.**

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.**

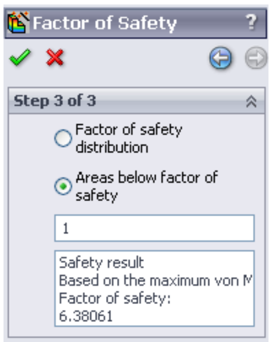
Note: When material yields, it continues to deform plastically at a quicker rate. In extreme case it may continue to deform even if the load is not increased.

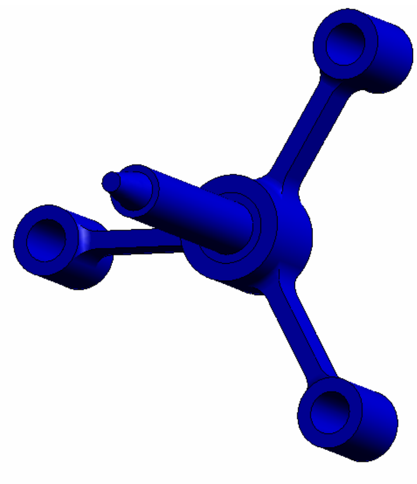
6 Click **Next.**

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.

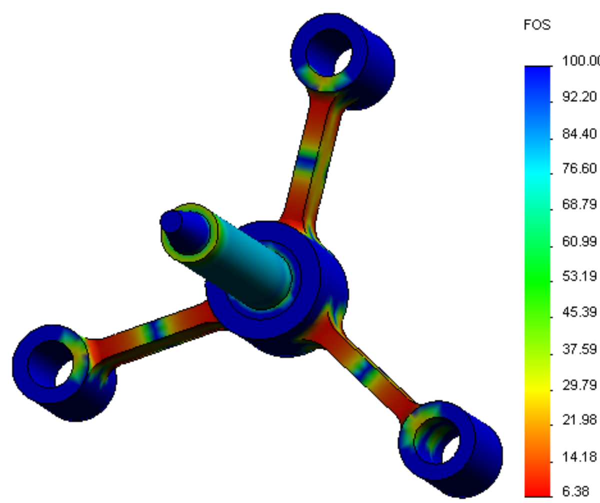
2 In the **Criterion** list, select **Max von Mises stress**.

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.

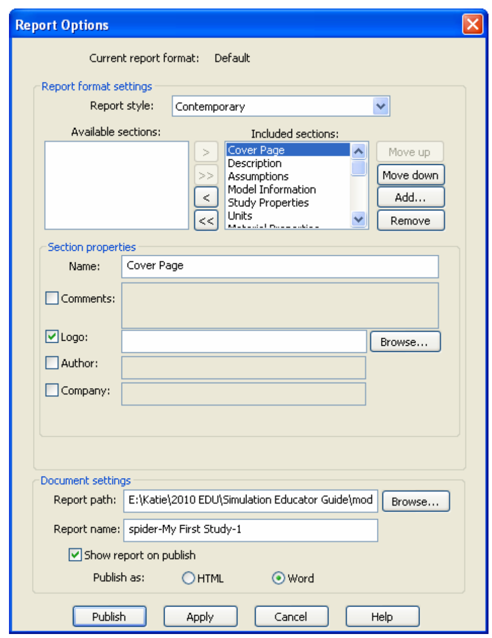
3 Click **OK**.

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 – Answer Key

1 How do you start a SolidWorks session?

Answer: On the Windows task bar, click **Start, Programs, SolidWorks, SolidWorks Application**. The SolidWorks application starts.

2 What do you do if SolidWorks Simulation menu is not on the SolidWorks' menu bar when a file is opened?

Answer: Click **Tools, Add-Ins**, check the checkboxes next to SolidWorks Simulation, and click **OK**.

3 What types of documents can SolidWorks Simulation analyze?

Answer: SolidWorks Simulation can analyze parts and assemblies.

4 What is analysis?

Answer: Analysis is a process to simulate how your design performs in the field.

5 Why is analysis important?

Answer: Analysis can help you design better, safer, and cheaper products. It saves you time and money by reducing traditional, expensive design cycles.

6 What is an analysis study?

Answer: An analysis study represents a scenario of analysis type, materials, loads and fixtures.

7 What types of analysis SolidWorks Simulation can perform?

Answer: SolidWorks Simulation can perform static, frequency, buckling, thermal, drop test, fatigue, optimization, pressure vessel, nonlinear static, linear and nonlinear dynamic analyses.

8 What does static analysis calculate?

Answer: Static analysis calculates stresses, strains, displacements, and reaction forces in your model.

9 What is stress?

Answer: Stress is the intensity of force or force divided by area.

10 What are the main steps in performing analysis?

Answer: The main steps are: create a study, assign materials, apply fixtures, apply loads, mesh the model, run the analysis, and visualize the results.

11 How can you change the material of a part?

Answer: Under the `Parts` folder of your study, right-click the part icon and click **Apply Material to All**, then select the new material and click **OK**.

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

Answer: No. The minimum factor of safety should not be less than 1.0 for a safe design.

In Class Discussion — Changing Material Assignment

Ask the students to assign different materials to the assembly components according to the following table and run the analysis.

Component	**Material Name**
Shaft	Alloy Steel
Hub	Gray Cast Iron
Spider	Aluminum 6061 Alloy

Answer

To assign different materials to the assembly components do the following:

Assign Gray Cast Iron to the `hub`

1 In the Simulation study tree, right-click the `hub-1` icon located inside the `Parts` folder and click **Apply/Edit Material**.

The **Material** dialog box appears.

2 In `SolidWorks Materials`, under `Iron` category, select **Gray Cast Iron**.

3 Click **Apply** and **Close**.

Assign Aluminum 6061 Alloy to the `spider` Leg

1 In the Simulation study tree, right-click the `spider-1` icon located inside the `Parts` folder and click **Apply/Edit Material**.

The **Material** dialog box appears.

2 In `SolidWorks Materials`, under `Aluminum Alloys` category, select **6061 Alloy**.

3 Click **Apply** and **Close**.

Run the study again and visualize results

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

1 In the Simulation study tree, right-click the `Study` icon and click **Run**.

Note: To get the new results, you do not need to remesh the model.

2 In the SolidWorks Simulation Manager tree, click the plus sign beside the `Results` folder.

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

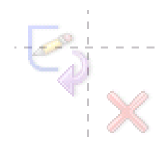
3 Double-click `Stress1 (-vonMises-)` icon to plot the von Mises stress plot.

More to Explore — Modifying the Geometry

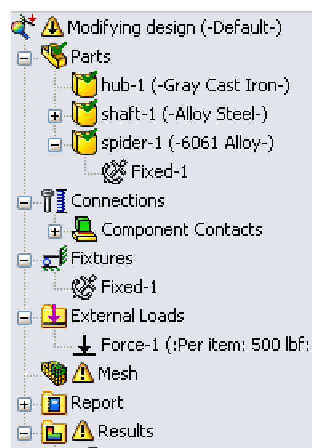
After visualizing the results, you may want to make changes in your design. Ask the students to make a change in the geometry and recalculate the results. It is important to emphasize that they need to remesh the model and rerun the study after any change in geometry. The following procedures describe how to change the diameters of the three holes and re-evaluate the results.

Answer

- Click the FeatureManager tab .
- Click the plus sign `(+)` beside `(-) spider<1>`.
- Click the plus sign `(+)` beside `Cut-Extrude2`. The `Sketch7` icon appears.
- Right-click `Sketch7` icon and select **Edit Sketch** . The sketch opens.
- Press `spacebar` and select ***Front** in the **Orientation** menu.
- Double-click the dimension **0.60**. The **Modify** dialog box appears.
- Enter **0.65** in the **Modify** dialog box and click .
- Click **OK** on the confirmation corner.



- Click the **Edit Component** icon to exit the edit mode.
- A warning icon appears next to `My First Study` and next to `Mesh`. A warning icon also appears next to `Results` folder indicating that the results are invalid.

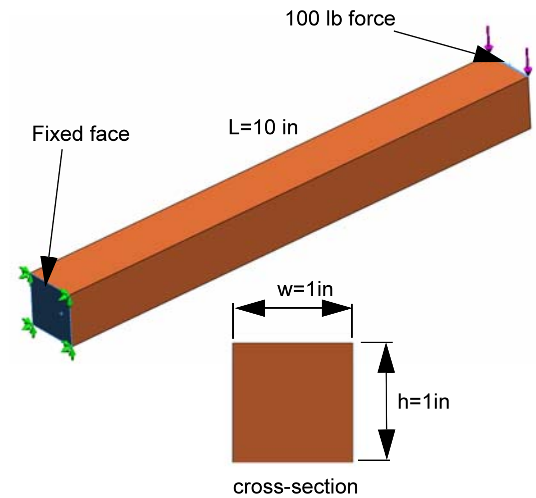


- To remesh the model, right-click the `Mesh` icon and click **Create Mesh**. A warning message appears to inform you that remeshing deletes the current results. Click **OK**.
- Use the default **Global Size** and **Tolerance** values. Note that these values are different from before.
- Check **Run (solve) the analysis** and click .
- When analysis is completed, view the default von Mises stress, displacement, strain and other results as described earlier.

Exercises and 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 (use the **Measure** tool).

   Answer: The width is 1.0 inch, the height is 1.0 inch, and the length is 10.0 inches.
3. Save the part to another name.
4. Create a **Static** study.

   Answer: Do the following:
   - Click **Simulation, Study**.
   - Enter a name for the study.
   - Set **Analysis type** to **Static**.
   - Click **OK**.
5. Assign Alloy Steel to the part. What is the value of the elastic modulus in psi?

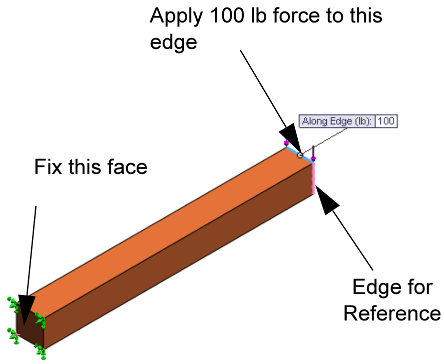
   Answer: Do the following:
   - In the SolidWorks Simulation Manager tree, right-click the `Front_Cantilever` icon select **Apply/Edit Material**. The **Material** dialog box appears.
   - Expand the `SolidWorks Materials` library.
   - Expand `Steel` category and select **Alloy Steel**.
   - In the **Units** menu, select **English (IPS)**. Notice that the value of **Elastic Modulus in X** is **30,457,919 psi**.
   - Click **Apply** and **Close**.
6. Fix one of the end faces of the cantilever.

   Answer: Do the following:
   - In the Simulation study tree, right-click the `Fixtures` folder and click **Fixed Geometry**. The **Fixture** PropertyManager appears.
   - Under **Type**, select **Fixed Geometry**.
   - Click the end face of the bar shown in the figure.
   - Click .

7 Apply a downward force to the upper edge of the other end face with magnitude of 100 lb.

Answer: Do the following:

- Right-click `External Loads` folder and click **Force**. The **Force/Torque** PropertyManager appears.
- Under **Type**, click **Force**.
- Click the edge shown in the figure.
- Make sure that `Edge<1>` appears in the **Face, Edge, Plane, Axis for Direction** box.



- Click **Selected direction** and choose the side edge of the beam as the **Face, Edge, Plane, Axis for Direction**.
- Select **English (IPS)** from the **Units** menu.
- Under **Force**, type **100** in the value box. Check the **Reverse direction** box. This is a vertical downward force.
- Click .

8 Mesh the part and run the analysis.

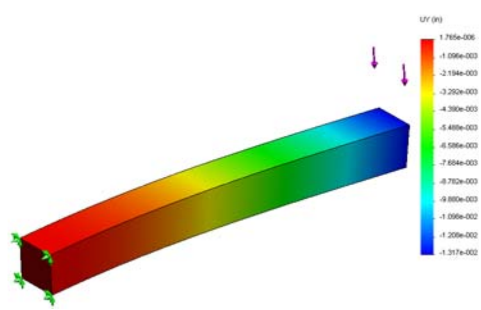
Answer: Do the following:

- In the Simulation study tree, right-click the `Mesh` icon.
- Use the default **Global size** and **Tolerance** .
- Check **Run (solve) the analysis**.
- Click .

9 When analysis is completed, plot the displacement in the Y-direction. The Y-direction is the same as dir 2 of `Plane1`. What is the maximum Y-displacement at the free end of the cantilever?

Answer: Do the following:

- In the Simulation study tree, right-click the `Results` folder and select **Define Displacement Plot.** The **Displacement Plot** PropertyManager appears.
- Select **in** for **Units** .
- Select **UY: Y Displacement** for **Component** .



- Click .
- The vertical displacement at the free end is **- 0.01317** in.

10 Calculate the theoretical vertical displacement at the free end using the following equation:

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

Answer: For this problem we have:
F = the end load = -100 lb,
L = the length of the beam = 10 in,
E = the Elastic modulus = 30,457,919 psi,
w = width of the bar = 1in,
h = height of the bar = 1 in.
Upon substituting the numerical values into the previous equation we obtain:
UY_{Theory} = -0.01313 inches.

11 Calculate the error in the vertical displacement using the following equation:

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

Answer: The error percentage in the maximum vertical displacement is 0.3%. In most design analysis applications, an error of about 5% is acceptable.

Lesson 1 Vocabulary Worksheet – Answer Key

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: **traditional design cycle**
2. A *what-if* scenario of analysis type, materials, fixtures, and loads: **study**
3. The method that SolidWorks Simulation uses to perform analysis: **finite element method**
4. The type of study that calculates displacements, strains, and stresses: **static study**
5. The process of subdividing the model into small pieces: **meshing**
6. Small pieces of simple shapes created during meshing: **elements**
7. Elements share common points called: **nodes**
8. The force acting on an area divided by that area: **average stress**
9. The sudden collapse of slender designs due to axial compressive loads: **buckling**
10. A study that calculates how hot a design gets: **thermal study**
11. A number that provides a general description of the state of stress: **von Mises Stress**
12. Normal stresses on planes where shear stresses vanish: **principal stresses**
13. The frequencies that a body tends to vibrate in: **natural frequencies**
14. The type of analysis that can help you avoid resonance: **frequency analysis**

Lesson 1 Quiz — Answer Key

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?

Answer: A study is a *"what-if"* scenario that defines the analysis type, materials, fixtures, and loads.

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

Answer: Static, frequency, buckling, thermal, drop test, fatigue, optimization, pressure vessel, nonlinear static, linear and nonlinear dynamic studies.

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

Answer: No. You only need to run the study again.

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

Answer: Yes. You must mesh the model after changing the geometry.

5 How do you create a new static study?

Answer: To create a new static study:

- Click **Simulation, Study**. The **Study** dialog box appears.
- Under **Study name**, type the name of the study. Use a meaningful name!
- Under **Study type**, select **Static**.
- Click **.**

6 What is a mesh?

Answer: A mesh is the collection of elements and nodes generated by meshing the model.

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

Answer: There will be one icon for each body. A component can have multiple bodies.

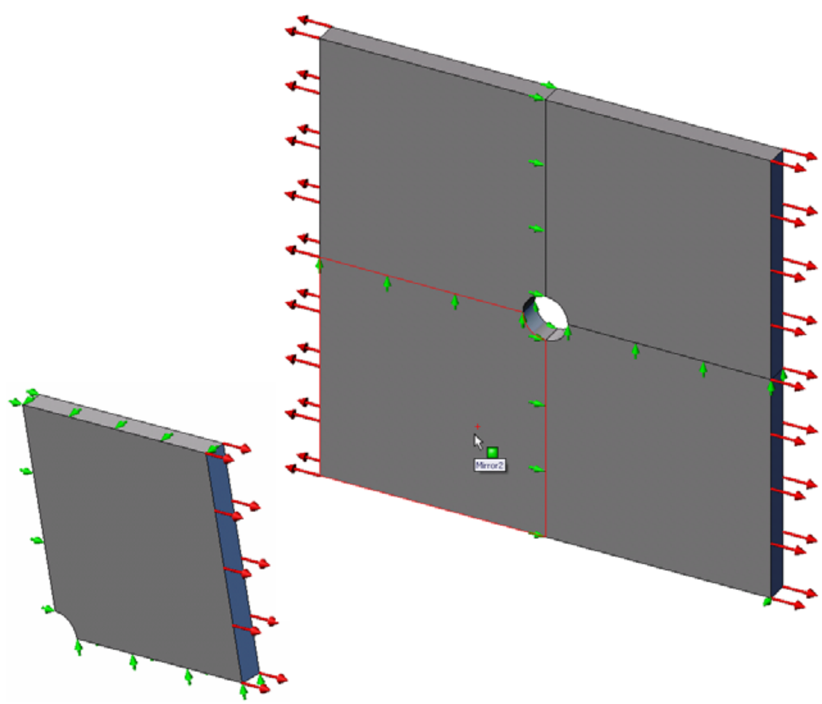
Lesson Summary

- ❑ SolidWorks Simulation is a design analysis software fully integrated in SolidWorks.
- ❑ Design analysis can help you design better, safer, and cheaper products.
- ❑ Static analysis calculates displacements, strains, stresses, and reaction forces.
- ❑ Frequency analysis calculates the natural frequencies and associated mode shapes.
- ❑ Buckling analysis calculates buckling loads for compressed parts.
- ❑ Drop test analysis calculates the impact loads on objects dropped on a rigid or flexible surface.
- ❑ Thermal analysis calculates the temperature distribution under thermal loads and thermal boundary conditions.
- ❑ Optimization analysis optimizes your model based on objective functions (i.e, minimize volume or mass).
- ❑ Materials start to fail when stresses reach a certain limit.
- ❑ von Mises stress is a number that gives an overall idea about the state of stresses at a location.
- ❑ The Factor of Safety Wizard checks the safety of your design.
- ❑ To simulate the model, SolidWorks Simulation subdivides the model into many small pieces of simple shapes called elements. This process is called *meshing*.
- ❑ The steps to perform analysis in SolidWorks Simulation are:
  - Create a study.
  - Assign material.
  - Apply fixtures to prevent rigid body motion.
  - Apply loads.
  - Mesh the model.
  - Run analysis, and
  - Visualize the results.

Lesson 2: Adaptive Methods in SolidWorks Simulation

Goals of This Lesson

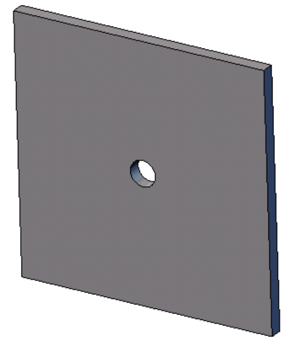
- Introduce the concept of adaptive methods for static studies. Upon successful completion of this lesson, the students should be able to understand the basic concepts behind adaptive methods, and how SolidWorks Simulation implements them.
- Analyze a portion of the model instead of the whole model. In the second part of this lesson, the students will analyze a quarter of the original model using symmetry fixtures. They should be able to recognize under which conditions they can apply symmetry fixtures without jeopardizing the accuracy of the results.
- Introduce the concept of shell meshing. The differences between a shell and solid mesh are highlighted in the project discussion. The students should be able to recognize which models are better suited for shell meshing.
- Compare SolidWorks Simulation results with known theoretical solutions. A theoretical solution exists for the problem described in this lesson. For the class of problems that have analytical solutions, the students should be able to derive the error percentages and decide if the results are acceptable or not.



Outline

- Active Learning Exercise — Adaptive Methods in SolidWorks Simulation
  - **Part 1**
  - Opening the `Plate-with-hole.SLDPRT` Document
  - Checking the SolidWorks Simulation Menu
  - Saving the Model to a Temporary Directory
  - Setting the Analysis Units
  - Step 1: Creating a Static Study
  - Step 2: Assigning Materials
  - Step 3: Applying Fixtures
  - Step 4: Applying Pressure
  - Step 5: Meshing the Model and Running the Analysis
  - Step 6: Visualizing the Results
  - Step 7: Verifying the Results
  - **Part 2**
  - Modeling a Quarter of the Plate Applying Symmetry Fixtures
  - **Part 3**
  - Applying the h-adaptive Method
- 5 Minute Assessment
- In Class Discussion-Creating a Frequency Study
- Exercises and Projects-Modeling the Quarter Plate with a Shell Mesh
- Lesson Summary

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:

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 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`.

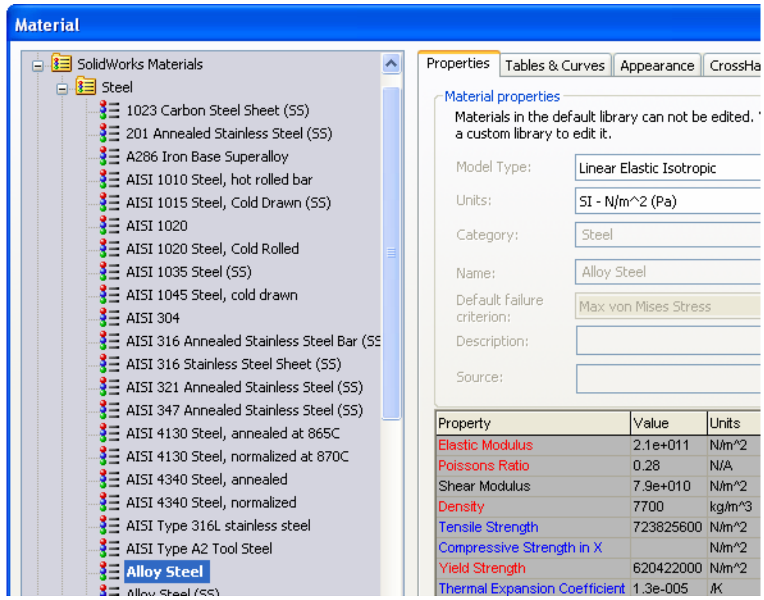
3 Under **Type**, select **Static**.

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.

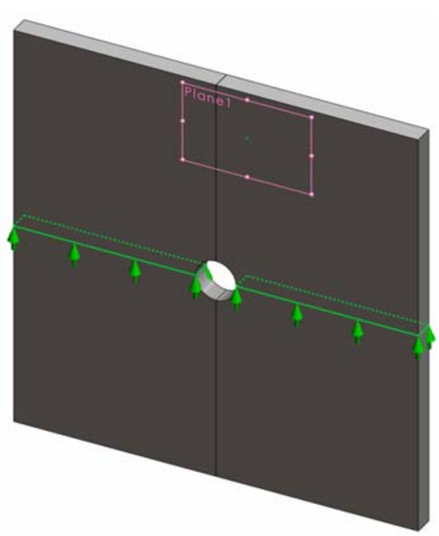
c) Select **Alloy Stee**.

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.

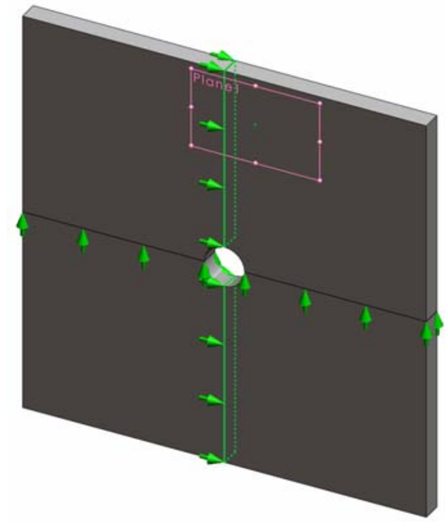
6 Under **Translations**, select **Along plane Dir 2**.

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.

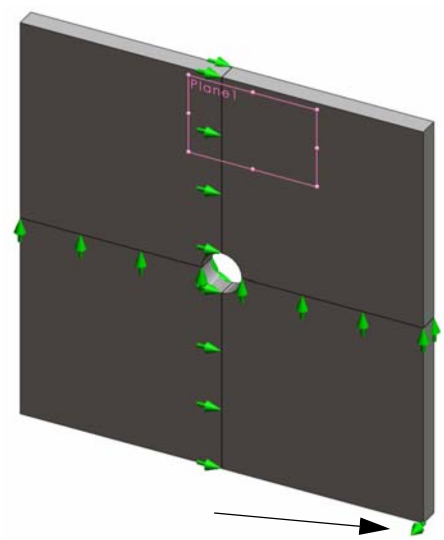
2 Make sure that **Type** is set to **Use reference geometry**.

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.

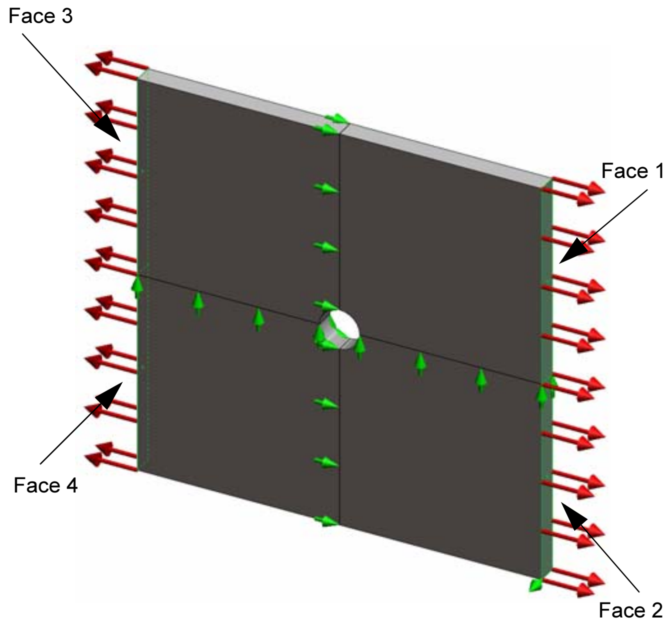
5 Under **Translations**, select **Normal to Plane** **.**



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.

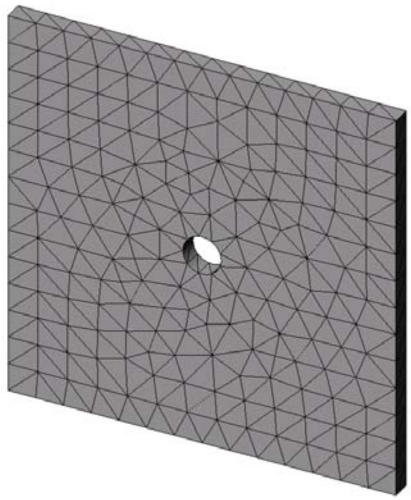
2 Expand **Mesh Parameters** by selecting the check box.

   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**

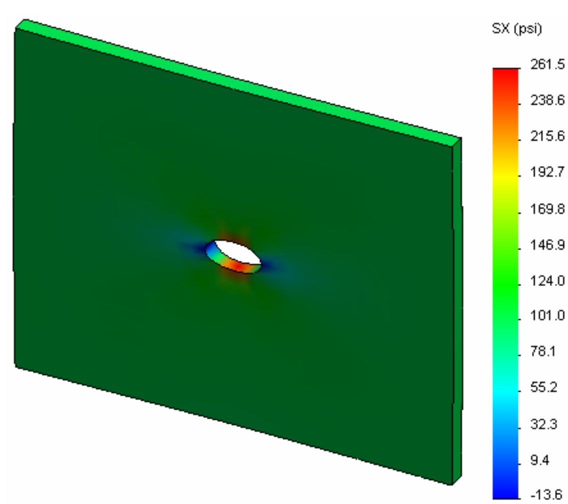
a) Select **SX: X Normal stress** in the **Component** field.

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:

$$\sigma max = k \cdot \left(\frac{P}{t(D-2r)}\right) \qquad k = 3.0 - 3.13\left(\frac{2r}{D}\right) + 3.66\left(\frac{2r}{D}\right)^2 - 1.53\left(\frac{2r}{D}\right)^3$$

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 coarseness of the mesh.

Active Learning Exercise — Part 2

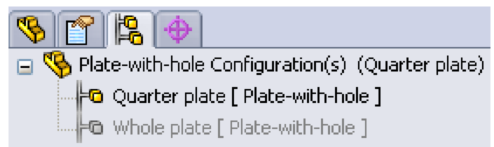
In the second part of the exercise you will model a quarter of the plate with help of the symmetry fixtures.

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

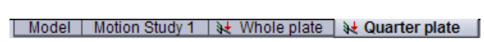
The new study that you create is based on the active `Quarter plate` configuration.

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.

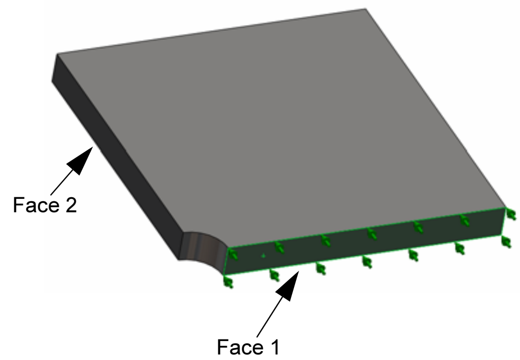


Step 3: Assigning Material

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.

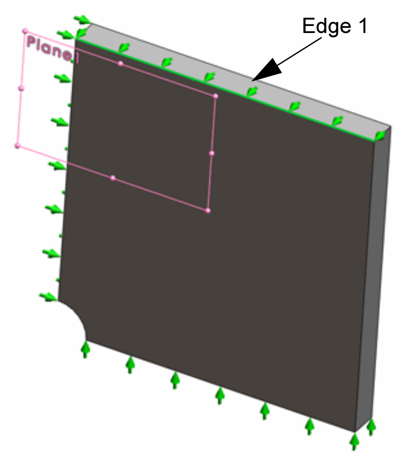
5 Click .

   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.

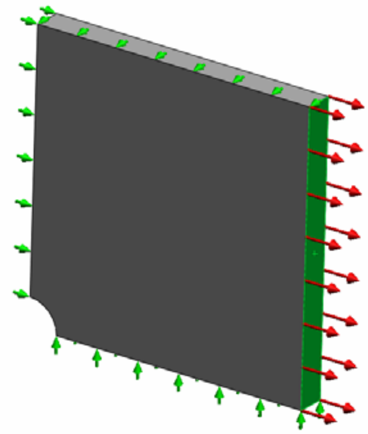
4 Under **Translations**, select **Normal to plane** . Make sure the other two components are deactivated.

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.

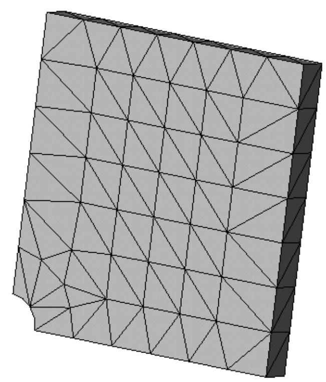
5 Click .

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**.

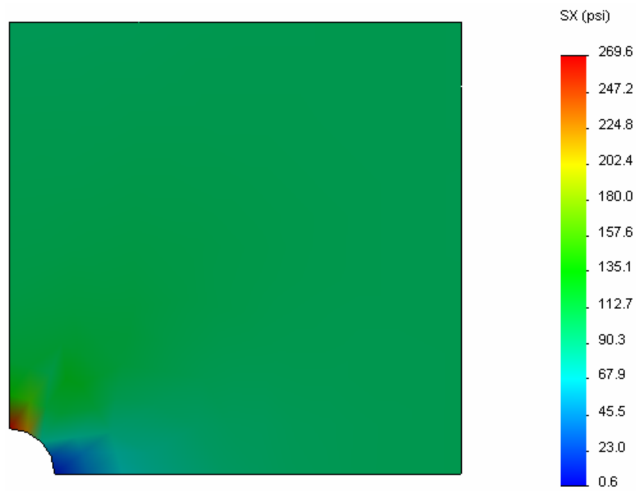
4 Under **Property:**

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

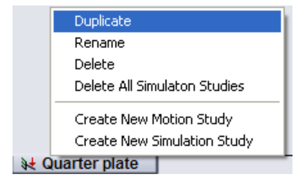
In the third part of the exercise you will apply the h-adaptive method to solve the same problem for the `Quarter plate` configuration.

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**.

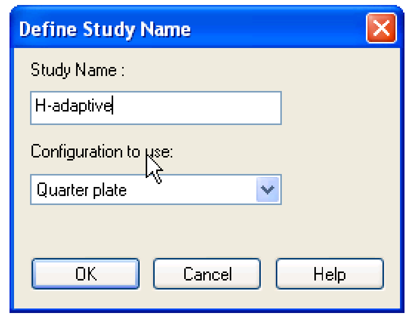


The **Define Study Name** dialog box appears.

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**.

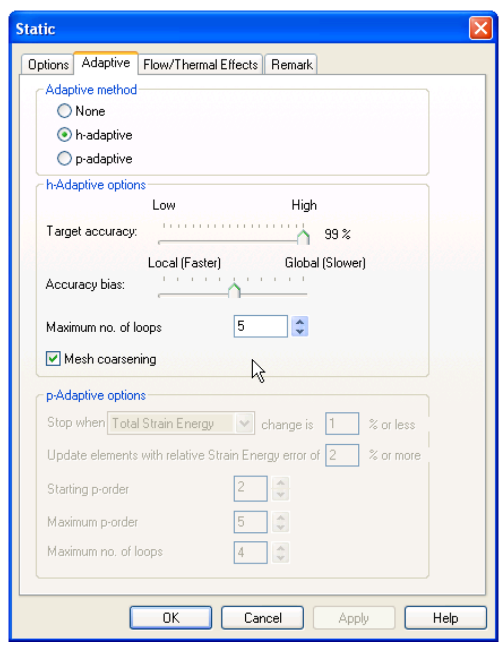
4 Under **h-Adaptive options**, do the following:

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

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

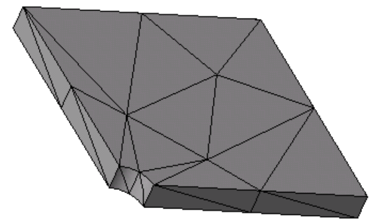
c) Check **Mesh coarsening.**

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

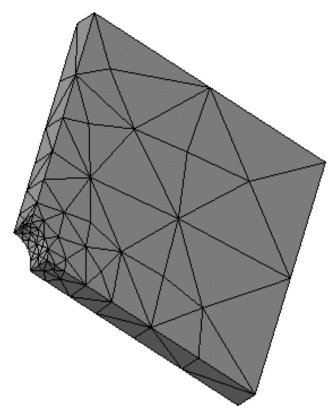
3 Type **5.0** (inches) for **Global Size** and accept the **Tolerance** suggested by the program.

   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

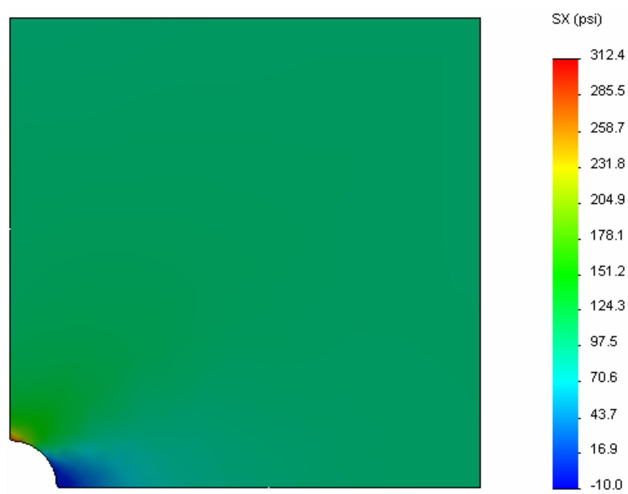


With the application of the h-adaptive method the original mesh size is reduced. Notice the transition of the mesh size from a coarser mesh (plate boundaries) to a finer mesh at the location of the central hole.

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.

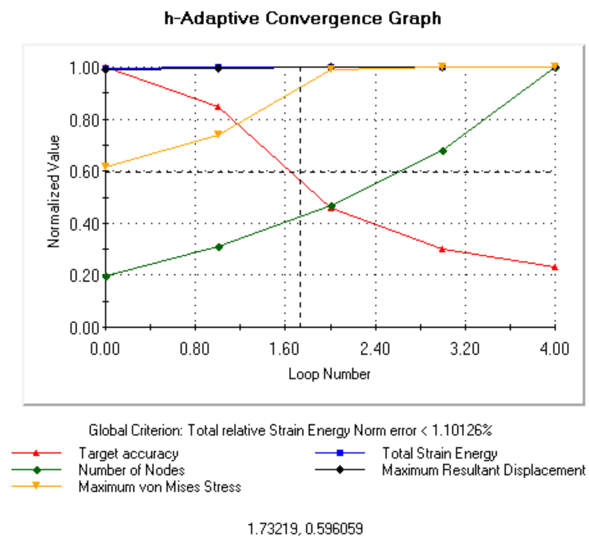
The SolidWorks Simulation result with the application of the h-adaptive method is SX = 312.4 psi, which is closer to the analytical solution (approximate error: 3.2%).

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- Answer Key

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

Answer: Material, loads and fixtures are applied to geometry. The mesh remains valid as long as geometry and mesh parameters have not changed. Results become invalid with any change in material, loads, or fixtures.

2 Does changing a dimension invalidate the current mesh?

Answer: Yes. The mesh approximates the geometry so any change in geometry requires meshing.

3 How do you activate a configuration?

Answer: Click the ConfigurationManager tab  and double-click the desired configuration from the list. You can also activate the configuration associated with a study by right-clicking the study's icon and selecting **Activate SW Configuration**.

4 What is a rigid body motion?

Answer: A rigid body mode refers to the body as a whole without deformation. The distance between any two points on the body remains constant at all times. The motion does not induce any strains or stresses.

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

Answer: The h-adaptive method is a method that tries to improve the results of static studies automatically by estimating errors in the stress field and progressively refining the mesh in regions with high errors until an estimated accuracy level is reached.

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

Answer: In mesh control, you must specify the mesh size and the regions in which you need to improve the results manually. The h-adaptive method identifies regions with high errors automatically and continues to refine them until the specified accuracy level or the maximum allowed number of iterations is reached.

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

Answer: No. The p-adaptive method increases the order of the polynomial to improve results in areas with high stress errors.

In Class Discussion — Creating Frequency Study

Ask the students to create frequency studies for the Plate-with-hole model for the `Whole plate` and `Quarter plate` configurations. To extract natural frequencies of the plate, no fixtures (except those controlling the symmetry of the quarter plate model) will be applied.

Explain that symmetry fixtures should be avoided in frequency and buckling studies since only symmetric modes are extracted. All anti-symmetric modes will be missed. Also explain the presence of the rigid body modes due to the lack of the fixtures.

Create a frequency study based on the `Whole plate` configuration

1. Activate the `Whole plate` configuration.
2. Click **Simulation, Study** in the main SolidWorks menu on the top of the screen.
   The **Study** PropertyManager appears.
3. Under **Name**, type `Freq-Whole`.
4. Under **Type**, select **Frequency**.
5. Click .

Set the properties of the frequency study

1. Right-click the `Freq-Whole` icon in the SolidWorks Simulation Manager and select **Properties.**
   The **Frequency** dialog box appears.
2. Set **Number of frequencies** to **15**.
3. Under **Solver** select **FFEPlus**.
4. Click **OK**.

Apply material

Drag-and-drop the `Plate-with-hole` folder in the `Whole plate` study to the `Freq-Whole` study.
The material properties of the `Whole plate` study are copied to the new study.

Apply loads and fixtures

Note: Both the fixtures and the pressure will not be considered in the frequency analysis. We are interested in the natural frequencies of fully unconstrained and unloaded plate.

Models without any fixture applied are allowed only in the frequency and buckling studies. In all other types of studies, proper fixtures must be applied.

Mesh the model and run the study

1. Right-click the `Mesh` icon and select **Create Mesh**.
2. Expand **Options**.
3. Check **Run (solve) the analysis**.
4. Expand **Mesh Parameters**

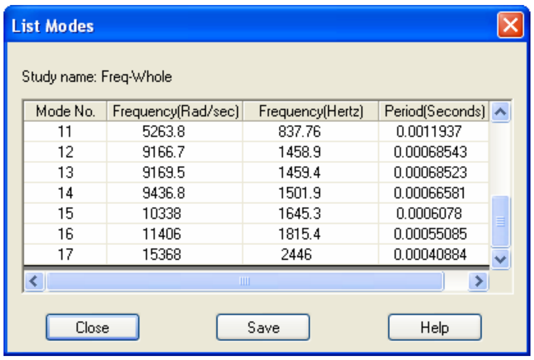
5 Make sure that **Automatic transition** is not checked.

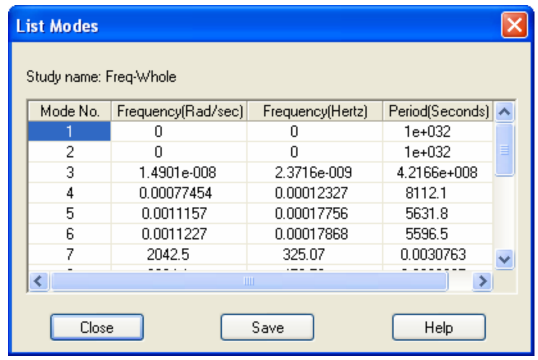
6 Click to accept the default setting for **Global Size** and **Tolerance** .

Listing resonant frequencies and viewing mode shapes

1 Right-click the `Results` folder and select **List Resonant Frequencies.**

The **List Modes** table lists the first fifteen non-zero frequencies.





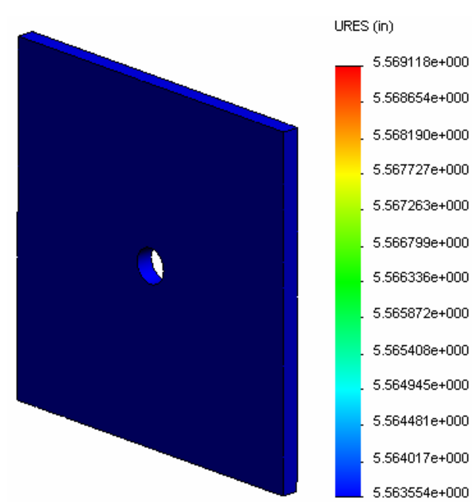
Note: The first several frequencies have zero, or near zero values. This result indicates that rigid body modes were detected and assigned very small (or zero) values. Because our model is fully unconstrained, six rigid body modes are found.

The first non-zero value corresponds to frequency #7 and has a magnitude of 2042.5 Hz. This is the first natural frequency of the unconstrained plate.

Close the **List Modes** window.

2 Expand `Results` and double-click the `Displacement1` plot.

The first rigid body mode shape appears in the graphics area.

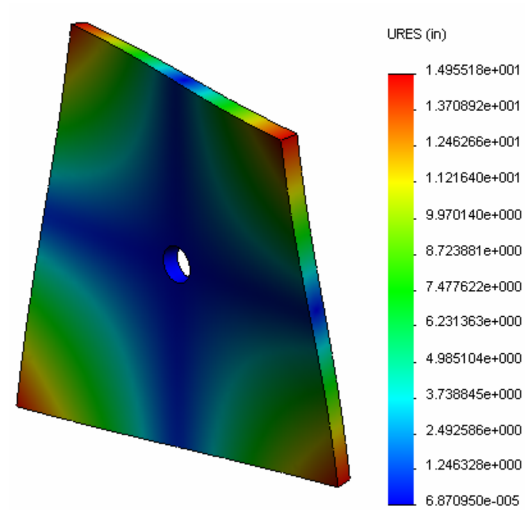


Note: The frequency #1 corresponds to the rigid body mode where the plate translates along the global X direction as a rigid body. No deformation is therefore shown.

Viewing real 1st natural frequency of the plate

1. Right-click `Results` and select **Define Mode Shape/Displacement Plot**.
2. Under **Plot Steps**, enter **7** for **Mode Shape**.
3. Click **OK**.

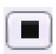
Note: Frequency #7 corresponds to the first real natural frequency of the plate.



Animating the mode shape plots

1. Double-click the mode shape icon (i.e., `Displacement6`) to activate it, and then right-click the icon and select **Animate**.

   The **Animation** PropertyManager appears.
2. Click .

   The animation is active in the graphics area.
3. Click  to stop the animation.
4. Click  to exit the animation mode.

Animating other mode shape plots

1. Double-click the mode shape icon for other frequencies (or define new mode shape plots for higher modes) and then right-click the icon and select **Animate**.
2. Also analyze the rigid body mode animations for frequencies #1 to #6.

Create a frequency study based on the `Quarter plate` configuration

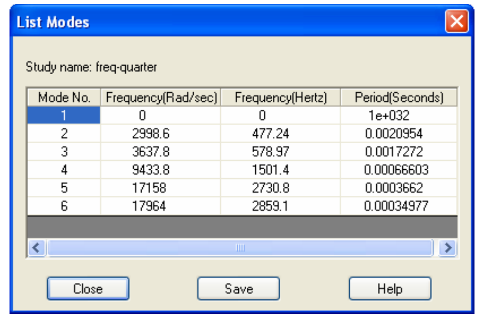
1. Activate the `Quarter plate` configuration.
2. Follow the steps described above to create a frequency study named `Freq-quarter`.

Note: Drag-and-drop the `Fixtures` folder in the `Quarter plate` study to the `Freq-quarter` study and supress the `Reference Geometry-1` fixture.

Listing resonant frequencies

The first five resonant frequencies are now listed as shown.

Animate the mode shape plots for the `Freq-quarter` study and compare them with those of the `Freq-Whole` study.



Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	0	0	1e+032
2	2998.6	477.24	0.0020954
3	3637.8	578.97	0.0017272
4	9433.8	1501.4	0.00066603
5	17158	2730.8	0.0003662
6	17964	2859.1	0.00034977

Note: Because we analyzed only a quarter of the model antisymmetric modes are not captured in the `Freq-quarter` study. For this reason, frequency analysis of the full model is strongly recommended.

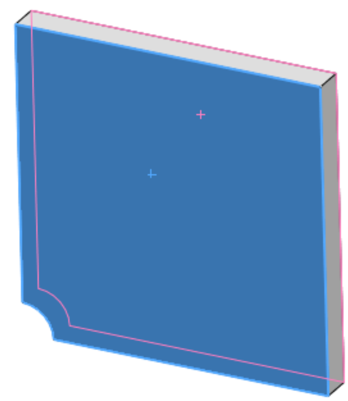
Because the `Symmetry-1` fixture restrains the model in certain directions, only one rigid body mode (zero frequency mode) is detected.

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.

Answer: Do the following:

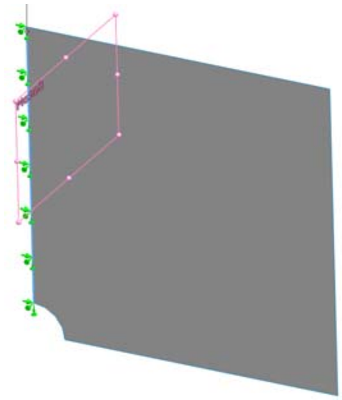
a) Right-click the `Fixtures` folder and select **Advanced Fixtures**.

b) In the **Faces, Edges, Vertices for Fixture** field select the edge indicated in the figure.

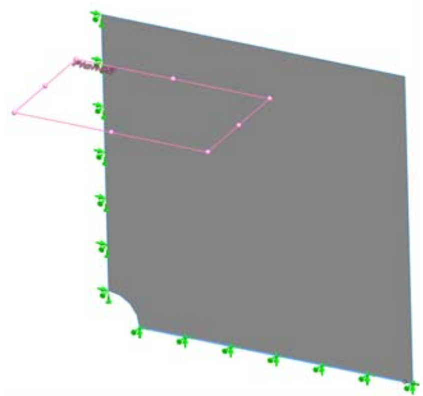
c) In the **Face, Edge, Plane, Axis for Direction** field select `Plane3`.

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 to 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.

<u>**Answer:**</u> Do the following:

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

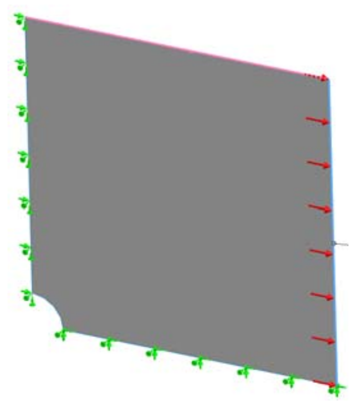
b) Under **Type** select **Use reference geometry**.

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 and check the **Reverse direction** checkbox.

f) Click .



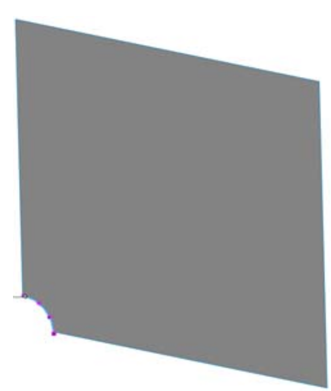
12 Apply mesh control to the edge shown in the figure.

<u>**Answer:**</u> Do the following:

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.

Answer: Do the following:

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

b) Use the default **Global size** and **Tolerance** .

c) Check **Run (solve) the analysis**.

d) Click .

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

Answer: Do the following:

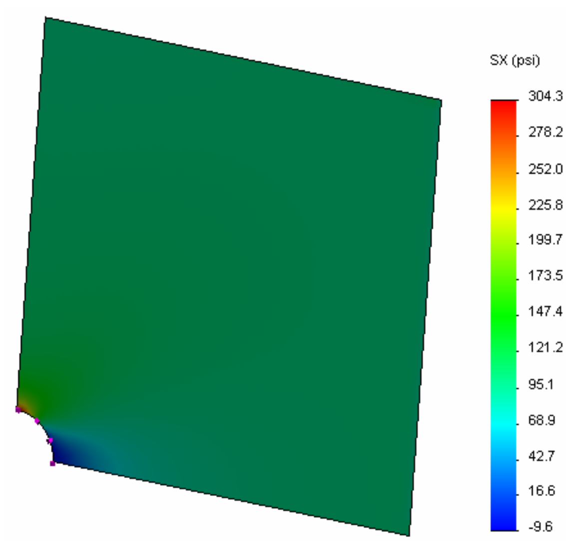
a) In the SolidWorks Simulation Manager tree, right-click the `Results` folder and select **Define Stress Plot**. The **Stress Plot** dialog box appears.

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

c) Select **psi** for **Units**.

d) Click .

e) The maximum SX normal stress is **304.3 psi**.



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

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

Answer:

The theoretical solution for the maximum SX stress is: SXmax = 302.452 psi

The error percentage in the maximum SX normal stress is 0.6%

In most design analysis applications, an error of about 5% is acceptable.

Lesson 2 Vocabulary Worksheet - Answer Key

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: **h-adaptive**
2 A method that improves stress results by increasing the polynomial order: **p-adaptive**
3 The type of degrees of freedom that a node of a tetrahedral element has: **translational**
4 The types of degrees of freedom that a node of a shell element has: **translational and rotational**
5 A material with equal elastic properties in all directions: **isotropic**
6 The mesh type appropriate for bulky models: **Solid Mesh**
7 The mesh type appropriate for thin models: **Shell Mesh**
8 The Mesh type appropriate for models with thin and bulky parts: **Mixed Mesh**

Lesson 2 Quiz - Answer Key

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?

Answer: 3 for draft and 6 for high quality

2 Does changing the thickness of a shell require remeshing?

Answer: No.

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

Answer: Adaptive methods are iterative methods that try to improve the accuracy of static studies automatically. They are based on estimating the error profile in a stress field. If a node is common to several elements, the solver gives different answers at the same node for each element. The variation of such results provides an estimate of the error. The closer these values are to each other, the more accurate the results are at the node.

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

Answer: You can experiment with your model's geometry in one document. Each study is associated with a configuration. Changing the geometry of a configuration affects only the studies associated with it.

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

Answer: Drag-and-drop the icon of an existing study onto the top icon of the SolidWorks Simulation Manager tree and then edit, add, or delete features to define the study.

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

Answer: Remesh the model with a smaller element size and rerun the study. If the changes in results are still significant, repeat the process until the results converge.

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

Answer: The program calculates displacements, strains, and stresses.

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

Displacement converges faster than stress. This is due to the fact that stress is a second derivative of displacement.

Lesson Summary

- The application of adaptive methods is based on an error-estimation of the continuity of a stress field. Adaptive methods are available for static studies only.
- Adaptive methods improve the accuracy without user interference.
- The theoretical stress at the point application of a concentrated load is infinite. The stresses keep increasing as you use a smaller mesh around the singularity or use the h-adaptive method.
- The application of Mesh control requires the identification of critical regions before the study runs. Adaptive methods do not require the user to identify critical areas.
- Symmetry can be used, when appropriate, to reduce the problem size. The model should be symmetrical with respect to geometry, fixtures, loads, and material properties across the planes of symmetry.
- No fixtures are allowed in the frequency analysis and are manifested by the presence of rigid body modes (zero, or near zero value frequencies).
- Symmetry fixtures should be avoided in frequency and buckling studies as you can extract symmetrical modes only.
- Thin parts are best modeled with shell elements. The shell elements resist membrane and bending forces.
- Bulky models should be meshed with solid elements.
- Mixed mesh should be used when you have bulky and thin parts in the same model.