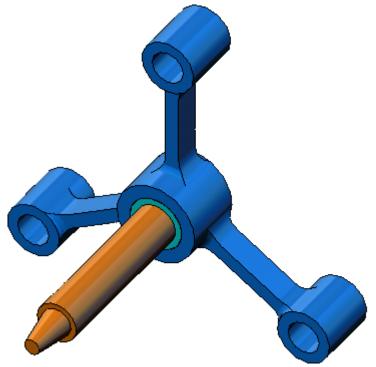
Engineering Design and Technology Series

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



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#### 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.

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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?

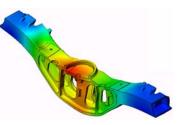
Can my design be modified to improve the frequency characteristics?

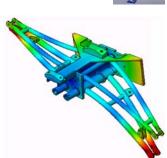
Thermal studies offer tools for the analysis of the heat transfer by means of conduction, convection, and radiation. Typical questions that will be answered using this study type are:

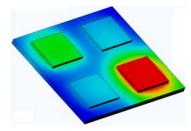
Will the temperatures changes effect my model? How does my model operate in an environment with temperature fluctuation?

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?

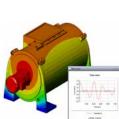
Nonlinear studies provide tools for analyzing stress in parts and assemblies that experience severe loadings and/or large deformations. Typical questions that will be answered using this study type are: Will parts made of rubber (o-rings for example) or foam perform well under given load?

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?









Introduction

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?



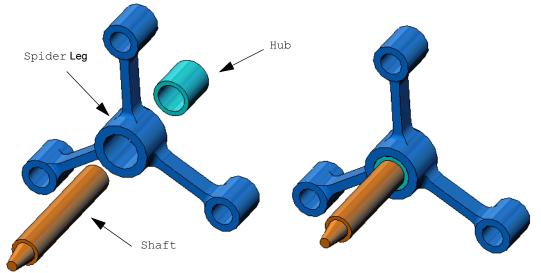


## Lesson 1: Basic Functionality of SolidWorks Simulation

#### **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 timeconsuming 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?

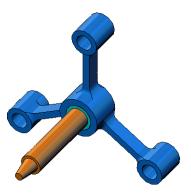
#### Answer

- □ 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.

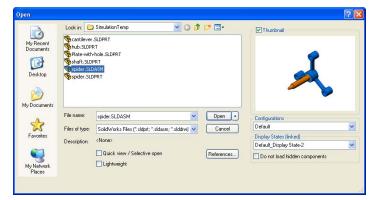
- 1 Create a temporary directory named SimulationTemp in the Examples folder of the SolidWorks Simulation installation directory.
- 2 Copy the SolidWorks Simulation Education Examples directory into the SimulationTemp directory.

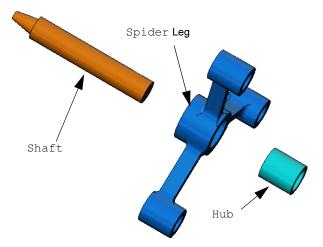
#### Opening the Spider.SLDASM Document

- Click Open *b* 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.

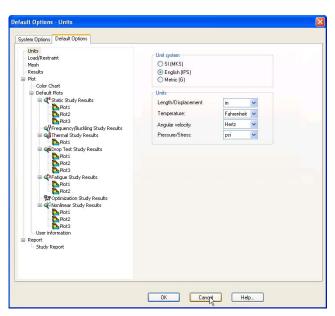
- 2 Check the checkboxes next to SolidWorks Simulation.If SolidWorks Simulation is not in the list, you need to install SolidWorks Simulation.
- 3 Click OK.

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.
- 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 <u>Model Motion Study 1 & My First Study</u> to navigate between multiple studies and your model.



#### Step 2: Assigning Material

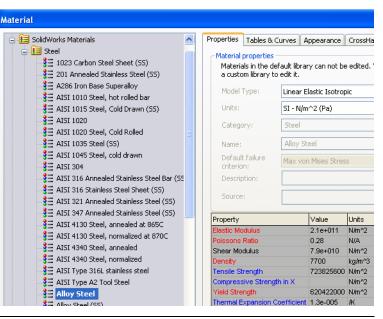
All assembly components are made of Alloy Steel.

#### 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.

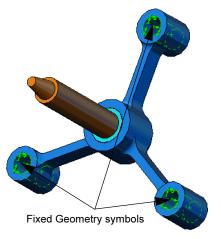
Face<1>, Face<2>, and Face<3> appear in the **Faces, Edges, Vertices for Fixture** box.



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** (2) 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.

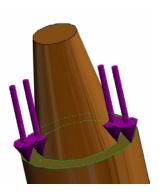
The Force/Torque PropertyManager appears.

- 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  $\perp$  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.

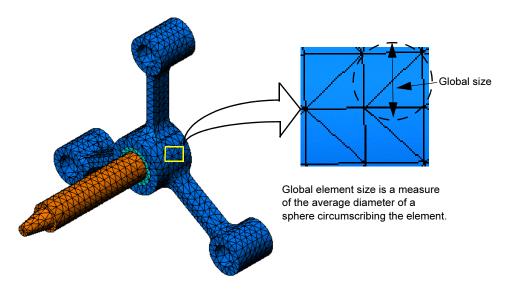
2 Expand **Mesh Parameters** by selecting the check box.

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

Keep default **Global Size**  $\triangle$  and **Tolerance** A suggested by the program.

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3 Click **OK** to begin meshing.



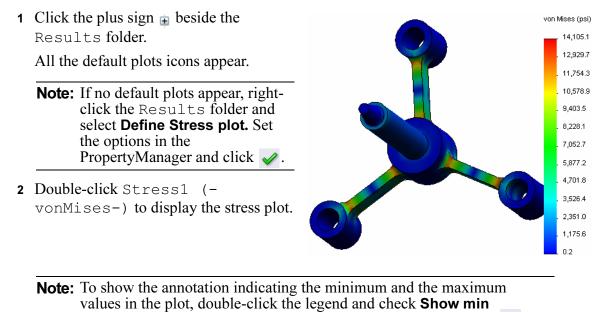
#### Step 6: Running the Analysis

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



annotation and Show max annotation check boxes. Then click  $\checkmark$ .

#### 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 **I**. 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.

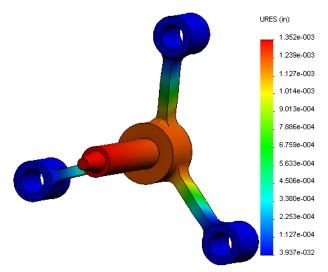
The animation is played in the graphics area.

- **5** Click **I** to **Stop** the animation.
- 6 Click 🖌 to close the Animation PropertyManager.

Animation ?					
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#### **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.

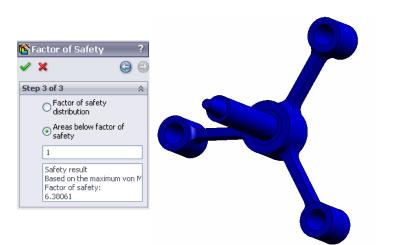
- 4 Set Units to psi.
- 5 Under Set stress limit to, select Yield strength.

**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  $\checkmark$  to generate the plot.



隆 Fa	ctor	of Safety	?			
<ul> <li>\$</li> </ul>	×		99			
Step	2 of 3	3	~			
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Material involved						
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	8998 Ultim	strength: :4.6 psi ate strength:				
	1049	182 psi				

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.

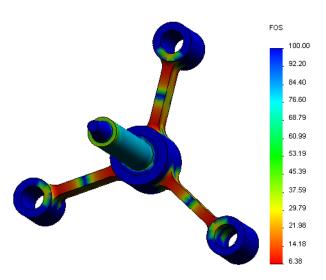
**Factor of Safety** wizard **Step 1 of 3** PropertyManager appears.

- 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 IncludedSections list and enter conclusion of your study in the Comments box.

Report Options	×
Current	: report format: Default
Report format set	
Report	style: Contemporary
Available se	Included Sections.
	Cover Page     Description     Assumptions     Mode Information     Study Properties     Units     Wood Units     Wood Owner     Add     Remove
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	Cover Page
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L	
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Report path:	E:\Katie\2010 EDU\Simulation Educator Guide\mod Browse
Report name:	spider-My First Study-1
Show rep	port on publish
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Publish	Apply Cancel Help

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?

<u>Answer:</u> 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?

<u>Answer:</u> Click Tools, Add-Ins, check the checkboxes next to SolidWorks Simulation, and click OK.

- What types of documents can SolidWorks Simulation analyze?
   <u>Answer:</u> 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?

<u>Answer:</u> 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?

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

7 What types of analysis SolidWorks Simulation can perform?

<u>Answer:</u> 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?

<u>Answer:</u> 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?

<u>Answer:</u> 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?

<u>Answer:</u> 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

#### <u>Answer</u>

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  $\checkmark$ .

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 is 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

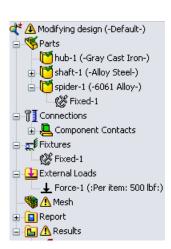
**3** Double-click Stress1 (-vonMises-) icon to plot the von Mises stress plot.

#### 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.

#### <u>Answer</u>

- □ 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 2. The sketch opens.
- □ Press spacebar and select \*Front in the Orientation menu.
- Double-click the dimension **0.60**. The **Modify** dialog box appears.
- $\Box$  Enter **0.65** in the **Modify** dialog box and click  $\checkmark$ .
- □ 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.
- $\Box$  Use the default **Global Size**  $\triangleq$  and **Tolerance**  $\clubsuit$  values. Note that these values are different from before.
- $\Box$  Check **Run (solve) the analysis** and click  $\checkmark$ .
- □ 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.

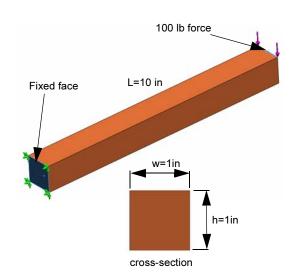
- **3** Save the part to another name.
- 4 Create a **Static** study.

<u>Answer:</u> 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?<u>Answer:</u> 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.

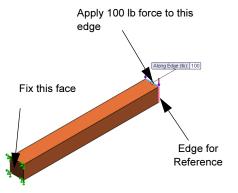
Answer: Do the following:

- 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 = 1 in,
- 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:
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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: <u>finite element</u> <u>method</u>
- 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: <u>nodes</u>
- 8 The force acting on an area divided by that area: <u>average stress</u>
- 9 The sudden collapse of slender designs due to axial compressive loads: **buckling**
- **10** A study that calculates how hot a design gets: **<u>thermal study</u>**
- 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: <u>natural frequencies</u>
- 14 The type of analysis that can help you avoid resonance: <u>frequency analysis</u>

#### Lesson 1 Quiz — Answer Key

Name: \_\_\_\_\_ Class: \_\_\_\_\_ Date:\_\_\_\_\_

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

- You test your design by creating a study. What is a study?
   <u>Answer:</u> A study is a "*what-if*" scenario that defines the analysis type, materials, fixtures, and loads.
- What types of analyses can you perform in SolidWorks Simulation?
   <u>Answer:</u> 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?

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

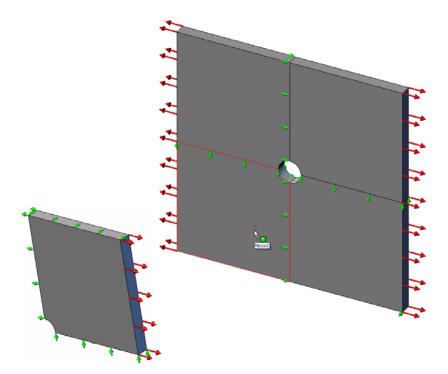
#### **Lesson Summary**

- □ 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** *in 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 **P** at the top of the left pane.



#### Checking the SolidWorks Simulation Menu

If SolidWorks Simulation is addedin, 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 Platewith-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.

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→ 📑 1023 Carbon Steel Sheet (SS)			ils in the defaul	t libra	ry can not be	e edited.	
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👌 A286 Iron Base Superalloy		Model	Europa De	6	-lashia Tashuar		
👌 AISI 1010 Steel, hot rolled bar		Model	Type:	Linear Elastic Isotropic			
AISI 1015 Steel, Cold Drawn (SS)		Units:	SI	SI - N/m^2 (Pa)			
		Catego		teel			
AISI 1020 Steel, Cold Rolled	=	Catoge	лу, <u>Б</u>	Steel			
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AISI 316 Stainless Steel Sheet (SS)		Source					
AISI 321 Annealed Stainless Steel (SS)		Source					
AISI 347 Annealed Stainless Steel (SS)		Durante			Value	Units	
📲 AISI 4130 Steel, annealed at 865C		Property Elastic Mo	alulu a		2.1e+011	Units N/m^2	
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				7.9e+010	N/m^2		
→ 📲 AISI 4340 Steel, normalized		Density		7700	kg/m^3		
📲 AISI Type 316L stainless steel		Tensile Strength		723825600	N/m^2		
ISI Type A2 Tool Steel		Compressive Strength in X			N/m^2		
Alloy Steel		Yield Stre	-		620422000		
3 Alloy Steel (SS)		Thermal E	xpansion Coeff	ficient	1.3e-005	ĸ	

**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.

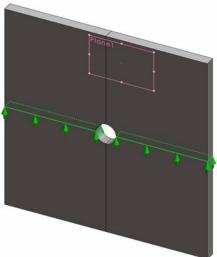
1 Press spacebar and select \*Trimetric in the Orientation menu.

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 forDirection box and select Plane1 from the flyout FeatureManager tree.
- 6 Under Translations, select Along plane Dir 2 🕅.

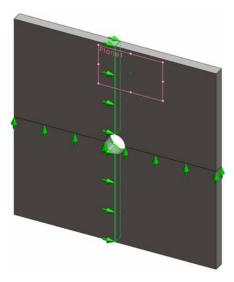


7 Click 🖌.

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.

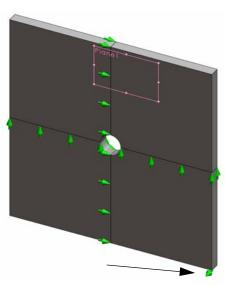
1 In the SolidWorks Simulation Manager tree, rightclick the Fixtures folder and click Advanced Fixtures.

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 Face 3 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).
- Face 4
- 5 In the **Pressure value** box  $\coprod$ , 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.

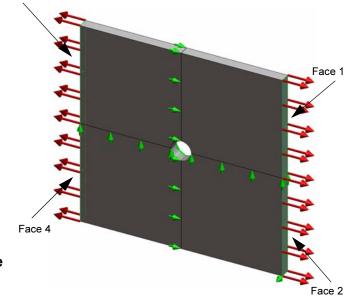
1 In the SolidWorks Simulation Manager tree, right-click the Mesh icon and select Create Mesh.

The **Mesh** PropertyManager appears.

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

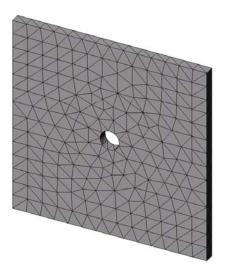
Make sure that **Standard mesh** is selected and **Automatic transition** is not checked.

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



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

Note: To see the mesh plot, right-click  ${\tt Mesh}$  folder and select Show Mesh



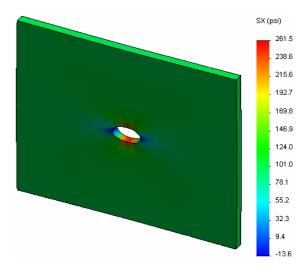
# Step 6: Visualizing the Results

Normal Stress in the global X-direction.

- 1 Right-click the Results folder in 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  $\sigma_{max}$  for a plate with a rectangular cross section and a central circular hole is given by:

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

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  $\sigma_{max} = 302.452$  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 doubleclick 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.

- 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.



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:

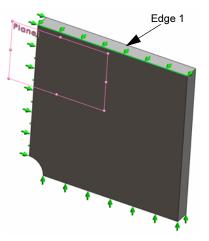
1 In the SolidWorks Simulation Manager tree, right-click the Fixtures folder and select Advanced Fixtures.

## Set Type to Use reference geometry.

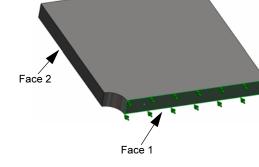
2 In the graphics area, click the upper edge of the plate shown in the figure.

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  $\blacksquare$ , type **100**.
- 4 Check the **Reverse direction** box.
- 5 Click 🖌.

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

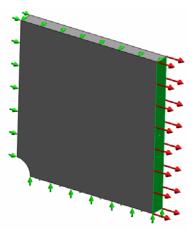
# Step 6 Meshing the Model and Running the Analysis

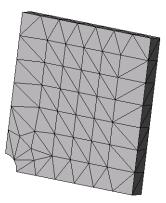
Apply the same mesh settings following the procedure described in Step 5 of Part 1, Meshing the Model and Running the Study on page 2-7. Then **Run** the analysis.

The mesh plot is as shown in the figure.

# Step 7 Viewing Normal Stresses in the Global X- Direction

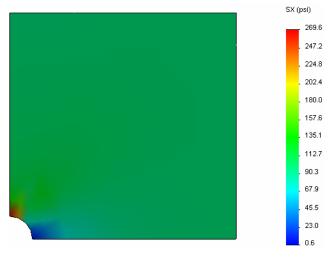
- 1 In the Simulation study tree, right-click the Results folder 🛅 and select Define Stress Plot.
- 2 In the **Stress Plot** PropertyManager, under **Display**:
  - a) Select **SX:X Normal stress**.
  - b) Select **psi** in **Units**.
- 3 Under **Deformed Shape** select **True Scale**.
- 4 Under Property:
  - 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.

itatic 🔀
Options Adaptive Flow/Thermal Effects Remark
Adaptive method
O None
In-adaptive
○ p-adaptive
h-Adaptive options
Low High
Target accuracy: 99 %
Accuracy bias:
Maximum no. of loops 5
p-Adaptive options
Stop when Total Strain Energy 💽 change is 👖 % or less
Update elements with relative Strain Energy error of $\begin{bmatrix} 2 & \ensuremath{\$} & \ensuremath{s} & \ensurem$
Starting p-order 2
Maximum p-order 5
Maximum no. of loops 4
OK Cancel Apply Help

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# 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** A and accept the **Tolerance** A 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  $\checkmark$ . The image above shows the initial coarse mesh.
- **5** Right-click the **H-adaptive** icon and select **Run**.

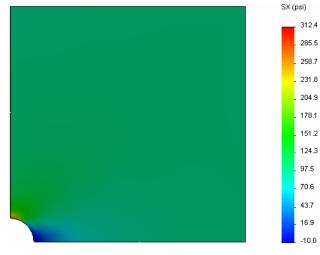
# Step 4: Viewing Results

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.

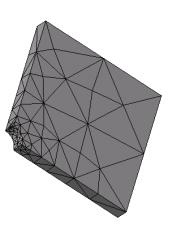
To view the converted mesh, right-click the Mesh icon and select **Show Mesh**.

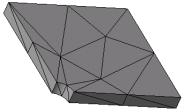
# View normal stress in the global X-direction

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



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





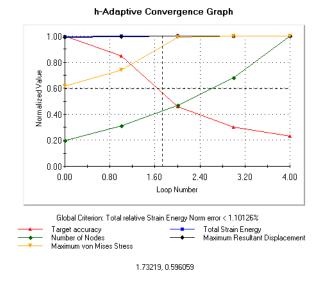
The SolidWorks Simulation result with the application of the h-adaptive method is SX = 312.4 psi, which is closer to the analytical solution (approximate error: 3.2%).

**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  $\checkmark$ .

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?

<u>Answer</u>: 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?

<u>Answer</u>: Click the ConfigurationManager tab R 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?

<u>Answer</u>: 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?

<u>Answer</u>: 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?

<u>Answer</u>: 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?

**<u>Answer</u>**: 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  $\checkmark$  to accept the default setting for Global Size  $\triangle$  and Tolerance  $\triangle$ .

## 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.

udy name:	Freq-Whole				St	udy name: f	Freq-Whole			
Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)	~		Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)	T
11	5263.8	837.76	0.0011937			1	0	0	1e+032	
12	9166.7	1458.9	0.00068543			2	0	0	1e+032	
13	9169.5	1459.4	0.00068523			3	1.4901e-008	2.3716e-009	4.2166e+008	L
14	9436.8	1501.9	0.00066581			4	0.00077454	0.00012327	8112.1	
15	10338	1645.3	0.0006078			5	0.0011157	0.00017756	5631.8	
16	11406	1815.4	0.00055085			6	0.0011227	0.00017868	5596.5	
17	15368	2446	0.00040884	~		7	2042.5	325.07	0.0030763	•
		1	>		<			1	>	

**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.

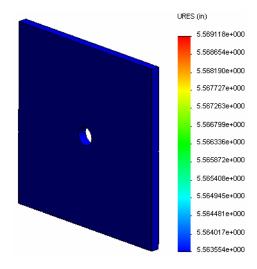
The first non-zero value corresponds to frequency #7 and has a magnitude of 2042.5 Hz. This is the first natural frequency of the unconstrained plate.

#### 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.

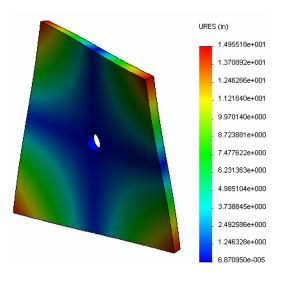
**Note:** The frequency #1 corresponds to the rigid body mode where the plate translates along the global X direction as a rigid body. No deformation is therefore shown.



#### 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 **I** to stop the animation.
- 4 Click  $\checkmark$  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 Freqquarter.

**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 Freqquarter 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
ь «	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.

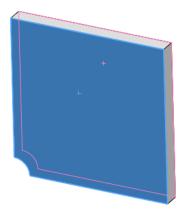
Because the Symmetry-1 fixture restrains the model in certain directions, only one rigid body mode (zero frequency mode) is detected.

# 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 Platewith-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.

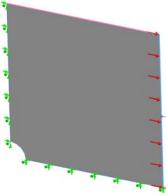
- b) In the **Faces**, **Edges**, **Vertices for Fixture** field select the edge indicated in the figure.
- c) In the Face, Edge, Plane, Axis for Direction field select Plane3.
- d) Restrain the Normal to Plane translation and Along Plane Dir 1 and Along Plane Dir 2 rotations.
- e) Click 🖌.
- 10 Using 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.

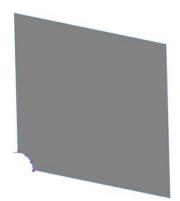
- **11** Apply **100 psi Pressure** to the edge shown in the figure.<u>Answer:</u> Do the following:
  - a) Right-click the External Loads folder and select **Pressure**.
  - b) Under Type select Use reference geometry.
  - c) In the **Faces**, **Edges for Pressure** field select the vertical edge shown 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 🖌.

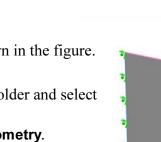
**12** Apply mesh control to the edge shown in the figure.

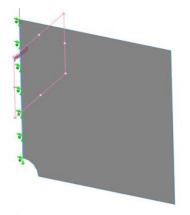
**Answer:** Do the following:

- 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.

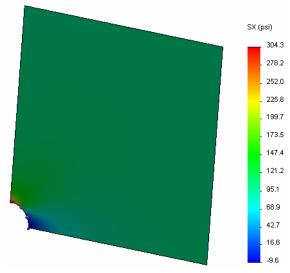
**Answer:** Do the following:

- a) In the SolidWorks Simulation Manager tree, right-click the Mesh icon and select **Create Mesh**.
- b) Use the default **Global size**  $\triangle$  and **Tolerance**  $\triangle$ .
- 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: <u>h-adaptive</u>
- 2 A method that improves stress results by increasing the polynomial order: **<u>p-adaptive</u>**
- 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: <u>translational and</u> <u>rotational</u>
- 5 A material with equal elastic properties in all directions: *isotropic*
- 6 The mesh type appropriate for bulky models: <u>Solid Mesh</u>
- 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.

- How many nodes are there in draft and high quality shell elements?
   <u>Answer:</u> 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?

<u>Answer</u>: 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?

<u>Answer</u>: 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?

<u>Answer:</u> 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.

Lesson 2: Adaptive Methods in SolidWorks Simulation