SolidWorks® 2011

SolidWorks Simulation Hands-on Test Drive

Dassault Systèmes SolidWorks Corp.
300 Baker Avenue
Concord, MA 01742 USA
Phone: 1 800 693 9000

Outside the US: 1 978 371 5011
Fax: 1 978 371 7303
info@solidworks.com

Document Number: MKSIMHOTBK1109

Table of Contents

Hands on Test Drive

When you complete this manual, you will have experienced firsthand an introduction to the capabilities of SolidWorks® Simulation products, including:

- SolidWorks® Simulation
- SolidWorks® Simulation Professional
- SolidWorks® Flow Simulation
- SolidWorks® Motion

Introduction

The SolidWorks® Simulation Hands-on Test Drive provides you with an understanding of the capabilities and benefits of using SolidWorks® Simulation analysis software to perform powerful analysis from your desktop. Only SolidWorks Simulation validation tools provide seamless integration with SolidWorks® 3D CAD software, with the benefit of the easy-to-use Windows® user interface.

Learn how you can use SolidWorks Simulations to perform stress analysis on your design; SolidWorks® Simulation Professional to perform stress, thermal, optimization, and fatigue analysis; SolidWorks® Motion to perform motion simulations; and SolidWorks® Flow Simulation to perform fluid-flow analysis on your designs.

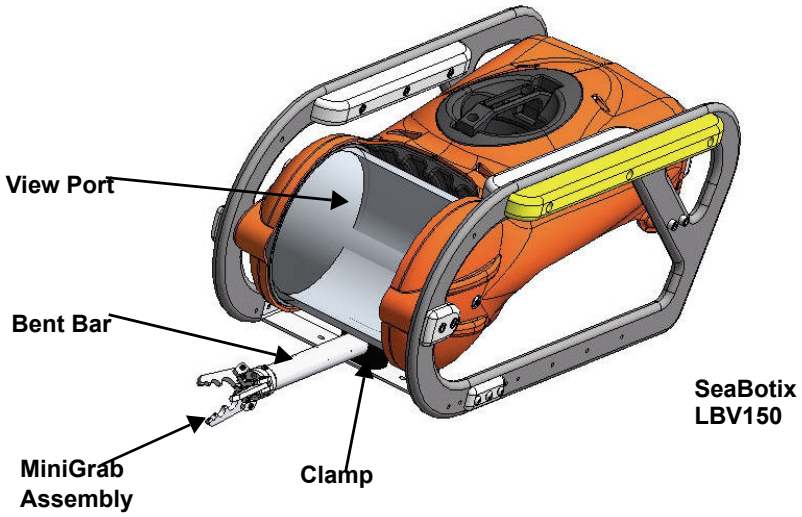
The SeaBotix LBV150

During this hands-on session, you will analyze some of the parts and assemblies that are components of the SeaBotix LBV150 assembly shown below.

SeaBotix, Inc. designed, manufactured, and introduced the first lightweight, low-cost, fully production submersible, remotely operated vehicle, the Little Benthic Vehicle. Bringing this breakthrough product to a wider market required modern 3D design and analysis tools, so product developers could shorten design cycles, validate cutting-edge technologies, and employ organic shapes and surfaces.

The company selected SolidWorks mechanical design software for the Little Benthic Vehicle project because of its ease of use, ability to model organic shapes and surfaces, SolidWorks® eDrawings® communication capabilities, and seamless integration with SolidWorks® Simulation analysis software.

The SeaBotix assembly can be remotely operated for use at depths of up to 1,500 meters. Weighing less than 25 pounds, the SeaBotix assembly represents a breakthrough in tethered submersible design.



You will have a chance to experience firsthand the ease of using SolidWorks® Simulation analysis software on the following items:

1. SeaBotix LBV150 assembly
2. Housing assembly
3. MiniGrab assembly
4. EndCap part
5. 3 Finger Jaw part

Today, you will use the SolidWorks Simulation family of products:

- **SolidWorks® Simulation** - The static analysis application that determines the stresses on the Housing assembly and the EndCap part.
- **SolidWorks® Simulation Professional** - The static, thermal, drop test, and optimization analysis application that validate the design of the Housing assembly, EndCap part, and the 3 Finger Jaw part.
- **SolidWorks® Motion** - The ridge body motion analysis application that simulates the mechanical operation of the motorized MiniGrab assembly and the physical forces it generates.
- **SolidWorks® Flow Simulation** - The fluid flow analysis application that provides insight into the SeaBotix LBV150 assembly related to fluid flow and forces on the immersed model.

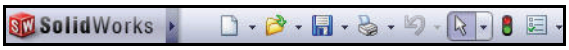
User Interface

The first thing that you notice about the SolidWorks® user interface is that it looks like Microsoft® Windows®. That is because it is Windows!

The SolidWorks 2010 (UI) is designed to make maximum use of the Graphics area space. Displayed toolbars and commands are kept to a minimum. Communicate with SolidWorks through the drop-down menus, Context document sensitive toolbars, Consolidated toolbars, or the CommandManager tabs.

Menu Bar Toolbar

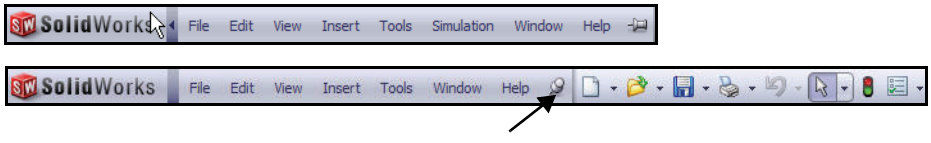
The Menu Bar toolbar contains a set of the most frequently used tool buttons. The available tools are: **New** - Creates a new document, **Open** - Opens an existing document, **Save** - Saves an active document, **Print** - Prints an active document, **Undo** - Reverses the last action, Select - Selects sketch entities, faces, edges and so on, **Rebuild** - Rebuilds the active part, assembly, or drawing, **Options** - Changes system options, document properties, and Add-Ins for SolidWorks.



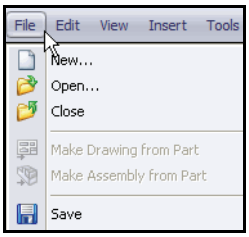
Menu Bar Menu

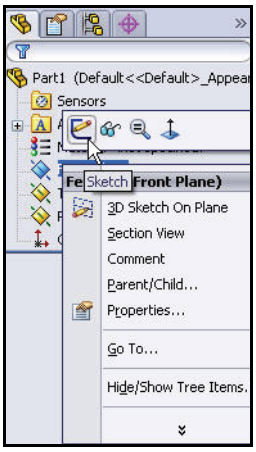
Click the SolidWorks name in the Menu Bar toolbar to display the default Menu Bar menu. SolidWorks provides a context-sensitive menu structure. The menu tittles remain the same for all three types of documents; part, assembly, and drawing but the menu items change depending on which type of document is active. The display of the menu is also dependent on the work flow customization that you have select. The default menu items for an active document are: **File**, **Edit**, **View**, **Insert**, **Tools**, **Window**, **Help**, and **Pin**.

Note: The Pin option displays both the Menu Bar toolbar and the Menu Bar menu.



Drop-down menu / Context Toolbar





Communicate with SolidWorks either thought the Drop-down menu or the Pop-up Context toolbar. The Drop-down menu from the Menu Bar toolbar or the Menu Bar menu provides access to various commands.

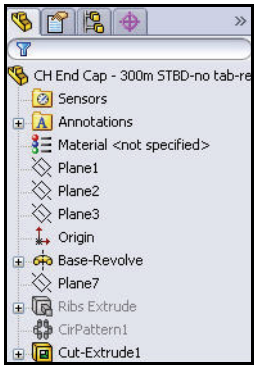
When you select, (click or right-click) items in the Graphics area or FeatureManager, Context toolbars appear and provide access to frequently performed actions for that context.

Keyboard Shortcuts

Some menu items indicate a keyboard shortcut like this: 

. SolidWorks conforms to standard Windows conventions for shortcuts such as **Ctrl+O** for **File, Open**; **Ctrl+S** for **File, Save**; **Ctrl+X** for **Cut**; **Ctrl+C** for **Copy**; and so on. In addition, you can customize SolidWorks by creating your own shortcuts.

FeatureManager Design Tree



The FeatureManager® design tree is a unique part of the SolidWorks software that employs patented SolidWorks technology to visually display all of the features in a part, assembly, or drawing.

As features are created, they are added to the FeatureManager. As a result, the FeatureManager represents the chronological sequence of modeling operations. The FeatureManager also allows access to editing the features and objects that it contains. The Part FeatureManager consist of four default tabs:

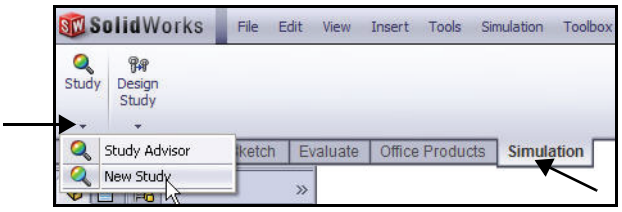
FeatureManager  , **PropertyManager**  ,

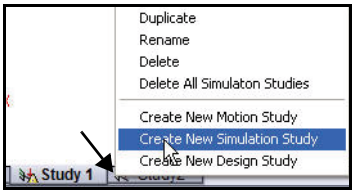
ConfigurationManager  , and **DimXpertManager**  .

SolidWorks Simulation CommandManager Tab

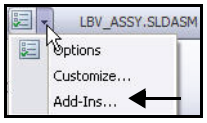
The SolidWorks Simulation CommandManager enables you to quickly create a Simulation Study. Click the SolidWorks Simulation tab in the CommandManager to create a new study. Studies are organized in tabs and are displayed in the bottom section of the Graphics area.

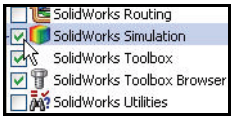
Note: Create a New Study using the **New Study** tool or right-click on a **Study** tab, click **Create New Simulation Study**.





Note: To activate SolidWorks Simulation, click the **Options** drop-down arrow from the Menu bar toolbar. Click **Add-Ins**. The Add-Ins dialog box is displayed. Check the **SolidWorks Simulation** box. Click **OK** from the Add-Ins dialog box. The Simulation tab is displayed in the CommandManager.





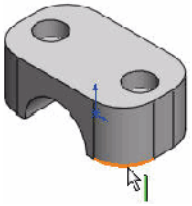
Mouse Buttons

The left, middle, and right mouse buttons have specific uses in SolidWorks.

- **Left** — Selects objects such as geometry, menu buttons, and objects in the FeatureManager design tree.
- **Middle** — Holding the middle mouse button as you drag the mouse rotates the view. Holding the **Shift** key down while you use the middle mouse button zooms the view. Using the **Ctrl** key scrolls or pans the view.
- **Right** — Activates context-sensitive pop-up menus. The contents of the menu differ depending on what object the cursor is over. These right-mouse button menus give you shortcuts to frequently used commands.

System Feedback

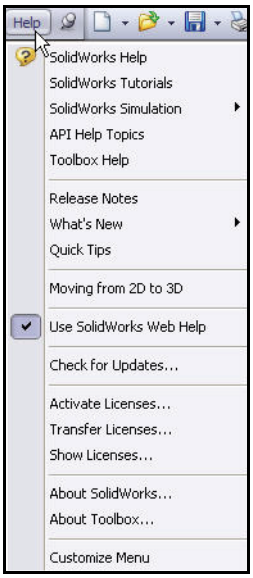
System feedback is provided by a symbol attached to the cursor arrow indicating what you are selecting or what the system is expecting you to select. As the cursor floats across the model, feedback comes in the form of symbols riding next to the cursor arrow.



Getting SolidWorks Help

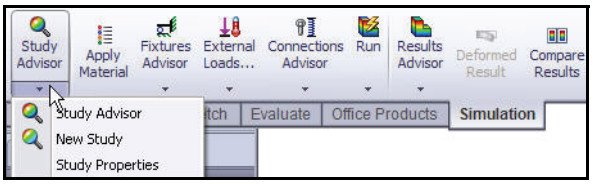
SolidWorks has a comprehensive Home help Page function that is design to assist the new and experience user. It provides information on What's New, SolidWorks Glossary, New Release notes, and more.

Click **Help**, **SolidWorks Help** from the Menu bar menu to view the comprehensive SolidWorks online Home help Page.



Note: Use SolidWorks Web Help is checked by default.

Getting SolidWorks Simulation Help.

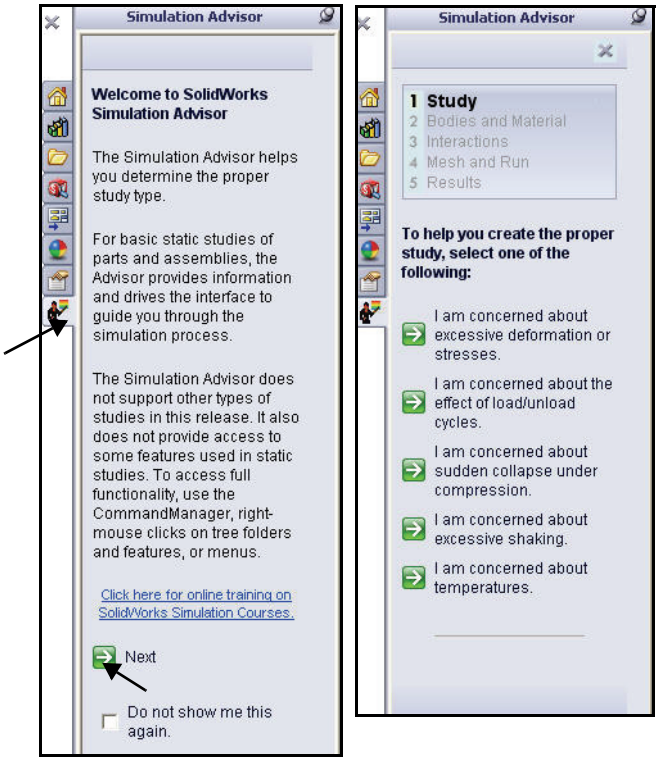


Click **Study Advisor**, **Study Advisor** from the Simulation tab in the CommandManager with an active study to obtain the Simulation Advisor.

The Simulation Advisor is a tool to help the user to determine how to create the proper study. It is broken into the following categories: *Study, Bodies and Material, Interactions, Mesh and Run, and Results*.

The Simulation Advisor walks you through by asking basic questions to lead to the correct action. By default, when you click on a tool in the Simulation CommandManager, it launches the relevant advisor. Deactivate the Simulation Advisor in the Simulation Options section.

Note: The Simulation Advisor tab is displayed in the Task Pane.



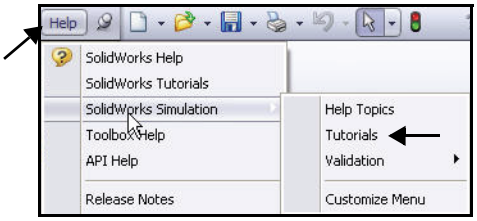
SolidWorks Tutorials and SolidWorks Simulation Tutorials

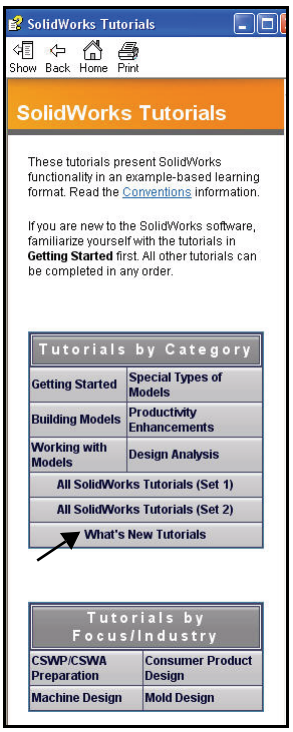
The SolidWorks Tutorials provide step-by-step lessons with sample files covering SolidWorks terminology, concepts, functions, features, and many Add-Ins. Work or view the lesson tutorials to learn and strengthen your skills.

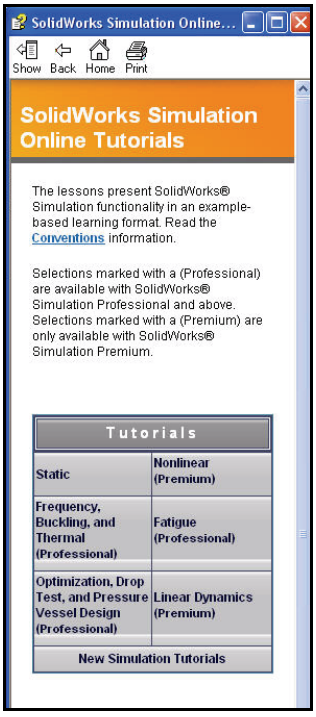
Click **Help**, **SolidWorks Tutorials** or click **SolidWorks Simulation**, **Tutorials** from the Menu Bar menu. View the results. The Tutorials are displayed by category.

Note: You can also access the SolidWorks Tutorials, click the SolidWorks **Resources** tab from the Task Pane and click **Tutorials**. View the available tutorials.

Note: Use the What's New Tutorials to view whats new in SolidWorks 2010.







SolidWorks Simulation

SolidWorks® Simulation is a design analysis application fully integrated with SolidWorks. It provides a one-screen solution for stress analysis and also enables you to solve large problems quickly using your personal computer. In this section of SolidWorks Simulation, you will address the following:

- SolidWorks Simulation User Interface
- The integration between SolidWorks Simulation and SolidWorks
- Creating Design Studies
- Understanding the Analysis Steps
- Assigning Materials
- Applying Fixtures and Loads
- Meshing the Model
- Running the Analysis
- Viewing the Results

Time: 55 - 60 minutes

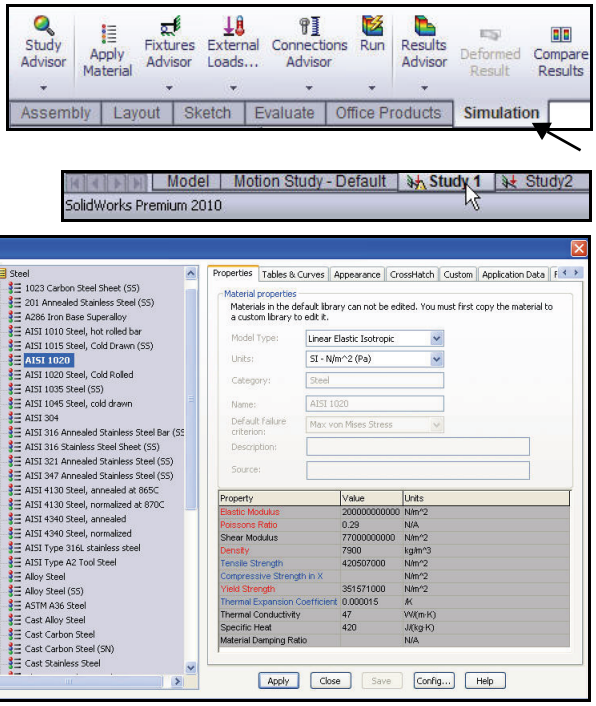
SolidWorks and SolidWorks Simulation

SolidWorks Simulation allows you to test a design and run multiple analysis iterations without ever leaving SolidWorks.

SolidWorks Simulation utilizes the SolidWorks FeatureManager tab, PropertyManager tab, and ConfigurationManager tab, the CommandManager, Motion Study tabs, Material Library, etc. and many of the same mouse and keyboard commands.

Anyone who can design a model in SolidWorks can analyze it without having to learn a new user interface. SolidWorks Simulation utilizes the power of SolidWorks configurations to test multiple designs. Plus, since SolidWorks Simulation uses native SolidWorks geometry, design changes made in one application are automatically updated in the other.

Regardless of the industry application, from aerospace to medical, SolidWorks Simulation provides significant product quality benefits, enabling engineers and designers to go beyond hand calculations and verify proof of concept for their designs.



Analyze the Housing

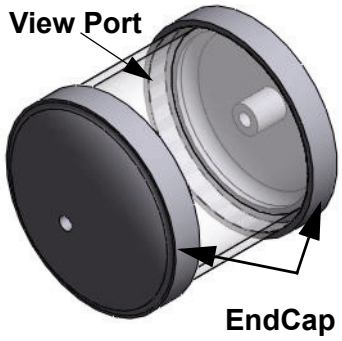
For your first analysis, explore the design validation of the Housing components in the SeaBotix LBV150 assembly using SolidWorks Simulation.

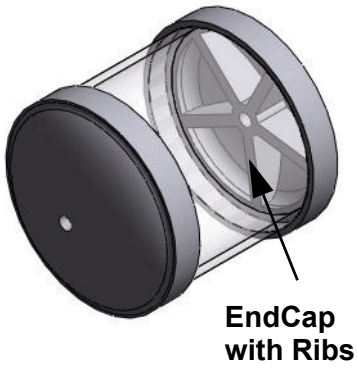
The Housing was simplified for today's class due to limited time. The Housing consists of two EndCaps and a View Port. The support tube, camera, and other components have been removed.

Your design goal in this section is to obtain a Factor of Safety (FOS) greater than one. You will first perform a static analysis on the Housing assembly containing the EndCaps without structural ribs as illustrated.

You will then perform a second static analysis on the Housing assembly containing the EndCaps with the addition of structural ribs as illustrated in hopes that the addition of the structural ribs will obtain your design goal of an FOS greater than one.

You will then compare the two studies side-by-side for a final design comparison.





Starting a SolidWorks Session

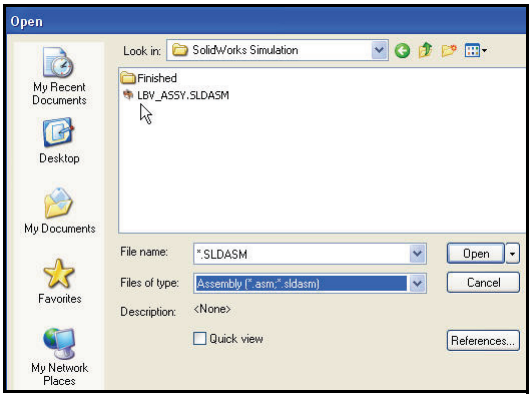
1 **Start a SolidWorks Session.**

- Click the **Start** menu.
- Click **All Programs**, **SolidWorks 2010**, **SolidWorks 2010**.

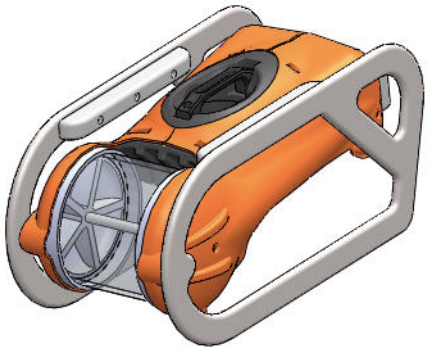
Note: You can quickly start a SolidWorks 2010 session by double-clicking the left mouse button on the desktop shortcut, if there is a shortcut icon on the system desktop.

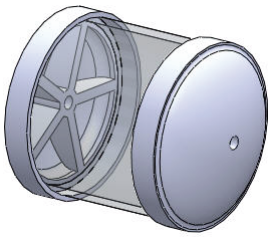
2 **Open the SeaBotix LBV150 Assembly.**

- Click **Open** from the Menu bar toolbar.
- Double-click **LBV_ASSY** from the SeaBotix\SolidWorks Simulation folder. A simplified sub-assembly is displayed in the Graphics area. View the FeatureManager.



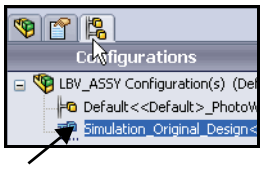
Note: The FeatureManager design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing. This makes it easy to see how the model or assembly was constructed or to examine the various sheets and views in a drawing.

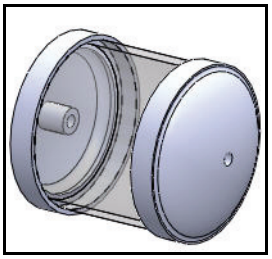




3 Select the Simulation_Original_Design Configuration.

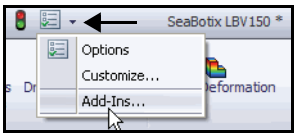
- Click the **ConfigurationManager** tab. The various configurations are displayed.
- Double-click the **Simulation_Original_Design** configuration. The Housing assembly (No Ribs) is displayed in the Graphics area.

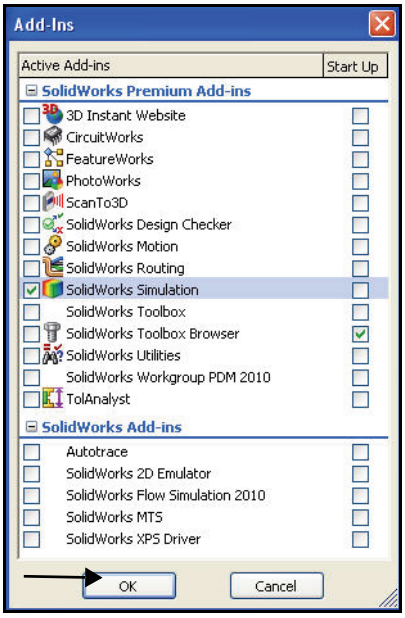




4 Activate SolidWorks Simulation.

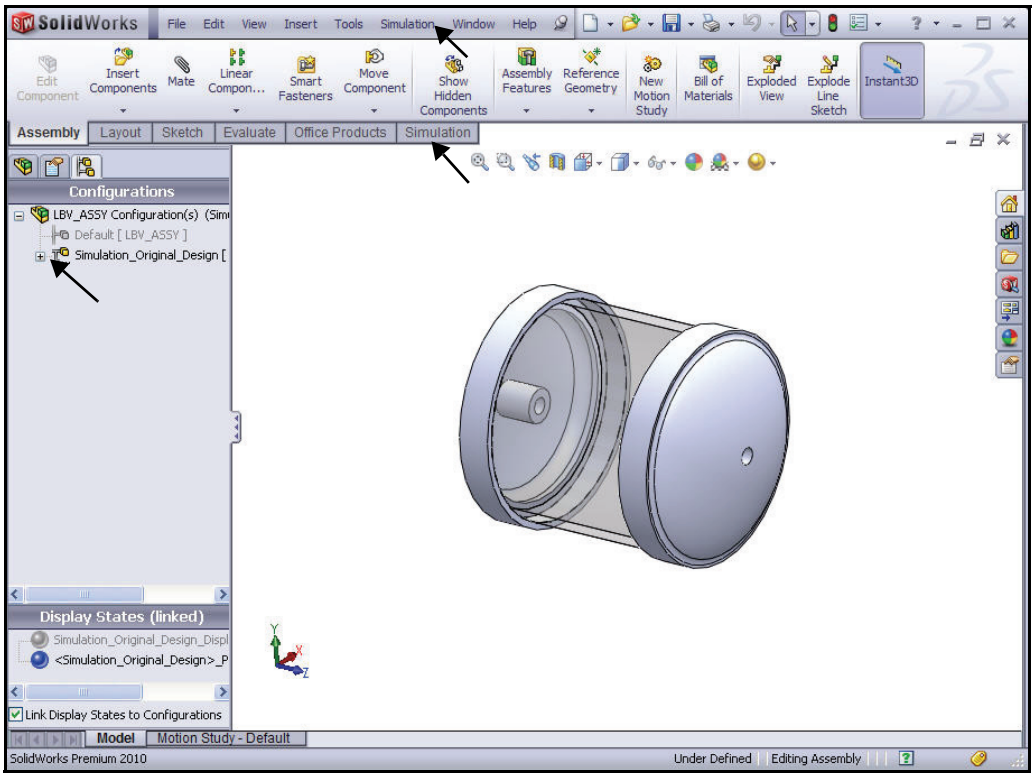
- Click the **Options** drop-down arrow as illustrated from the Menu bar toolbar.
- Click **Add-Ins**. The Add-Ins dialog box is displayed.
- Check the **SolidWorks Simulation** box.
- Click **OK** from the Add-Ins dialog box.





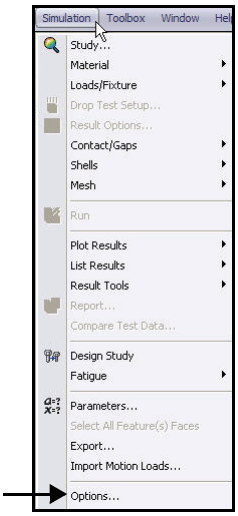
Note: Displayed Add-Ins may vary per system setup.

A Simulation tab is added to the CommandManager and a Simulation button is added to the Menu bar menu.

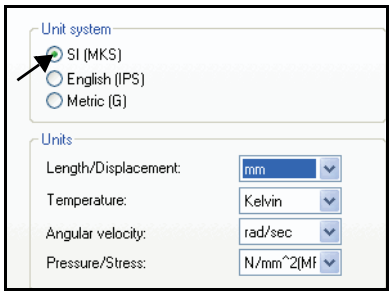


5 Set Default Options in SolidWorks Simulation.

- Click the **Simulation** button from the Menu bar menu.
- Click **Options** from the drop-down menu. The System Options - General dialog box is displayed.

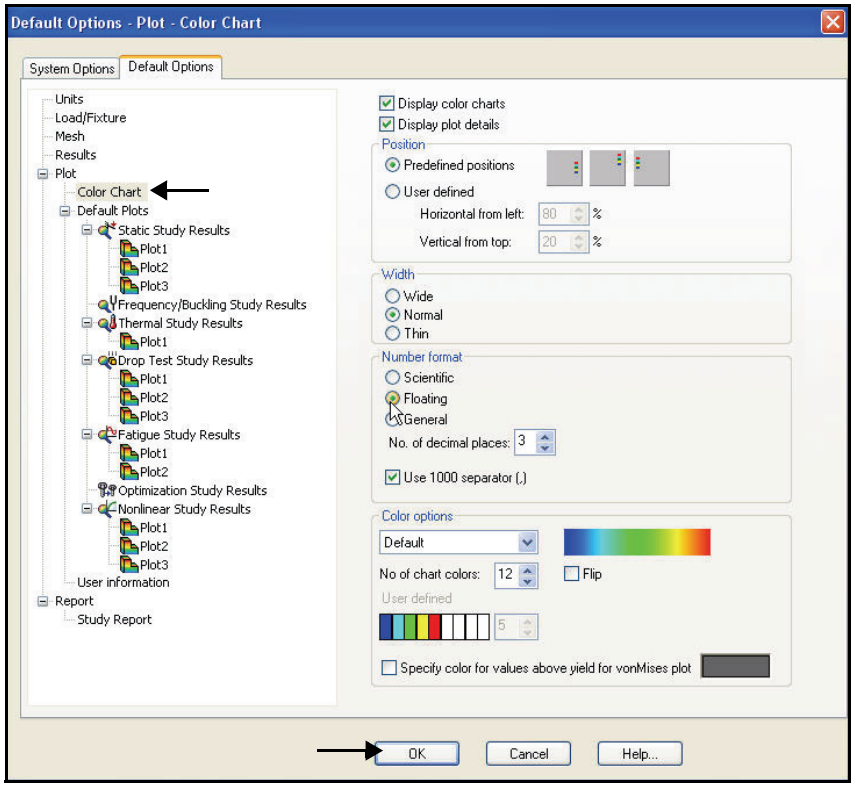


- Click the **Defaults Options** tab. View the Default Options - Unit dialog box.
- Click the **Units** folder.
- Click the **SI (MKS)** Unit system box.
- Select **mm** for Length/Displacement.
- Select **Kelvin** for Temperature.
- Select **rad/sec** for Angular velocity.
- Select **N/mm^2(MPa)** for Pressure/ Stress.

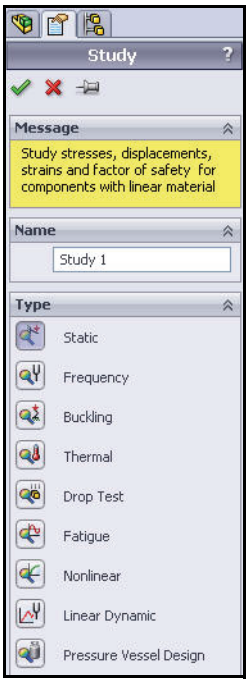


6 Set Number format.

- Click the **Color Chart** folder as illustrated.
- Click **Floating** for number format. View your options.
- Click **OK** from the Default Options - Plot Color Chart dialog box.



Create a Static Analysis Study



Create a Static study today. Static studies calculate displacements, reaction forces, strains, stresses, and factor of safety distribution.

Factor of safety calculations are based on common failure criteria.

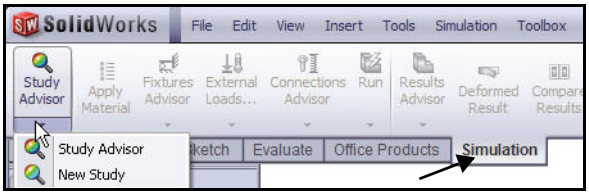
The first default Study name is Study 1.

SolidWorks Simulation offers six different results options. They are:

- Stress
- Displacement
- Strain
- Deformation
- Factory of Safety
- Design Insight

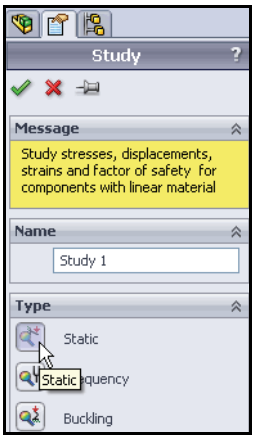
Static studies can help you avoid failure due to high stresses. A factor of safety less than one indicates likely material failure. Large factors of safety in a continuous region indicate that you can probably remove some material from this region.

Creating a Static Analysis Study



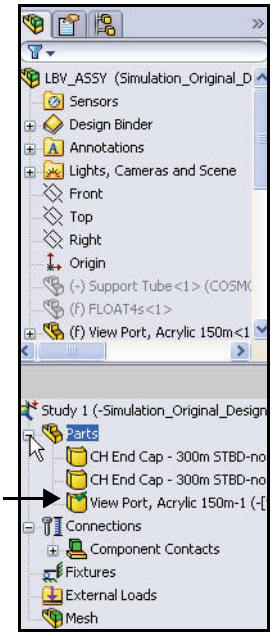
1 Create a Static Analysis Study.

- Click **Simulation** tab in the CommandManager.
- Click the **Study Advisor** drop-down arrow as illustrated.
- Click **New Study**. The Study PropertyManager is displayed. Study 1 is the default name for the first study. Accept the default Study name.
- Click the **Static** button for Type.



2 Display the Study.

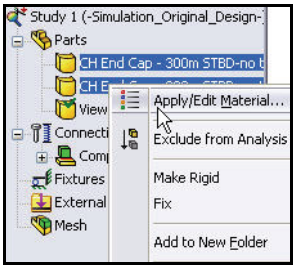
- Click **OK** from the Study PropertyManager. Study 1 (-Simulation_Original_Design-) is displayed. View the default folders.



Note: A green check mark on a Study folder indicates that material is assigned.

Note: If needed, return to the FeatureManager.

Assigning Materials in SolidWorks Simulation

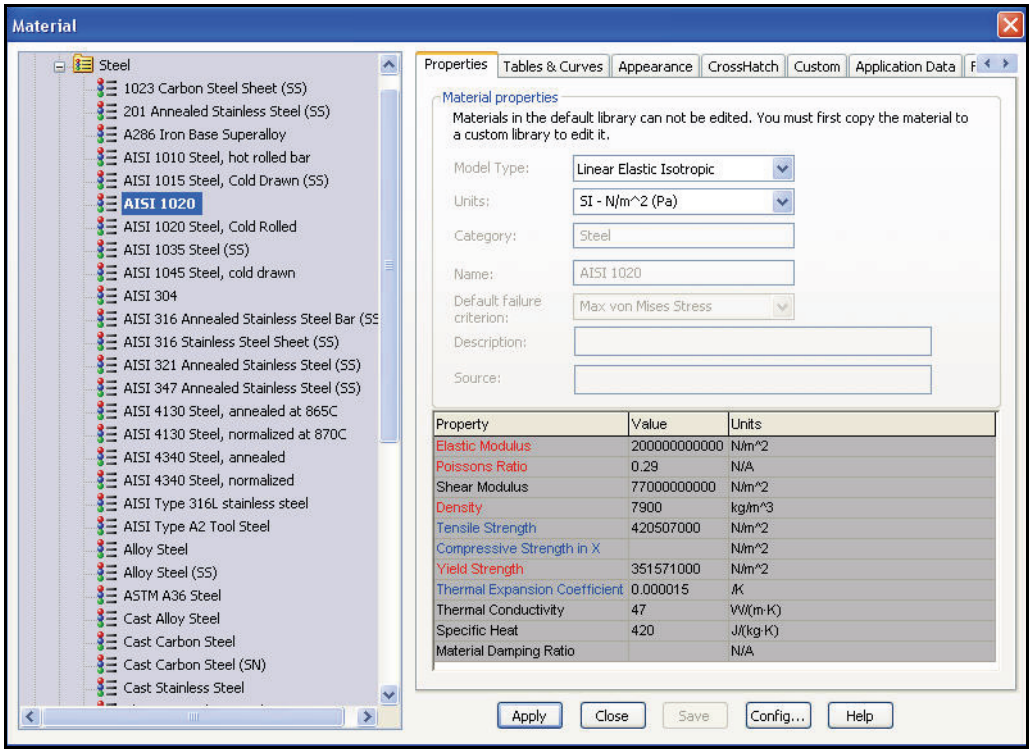


You can apply a material to a part, and create or edit a material with the SolidWorks Simulation Material dialog box.

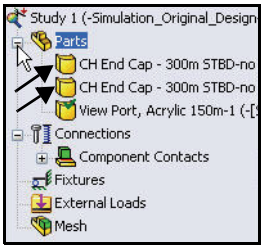
The Properties tab in the Material dialog box allows you to define a material source, material model, and material properties. You can define constant or temperature-dependent properties.

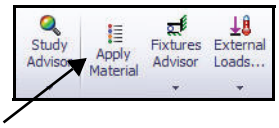
Defining materials in SolidWorks Simulation does not update the material assigned to the model in SolidWorks.

Define and apply material to the two EndCaps in the Housing assembly in the next section.



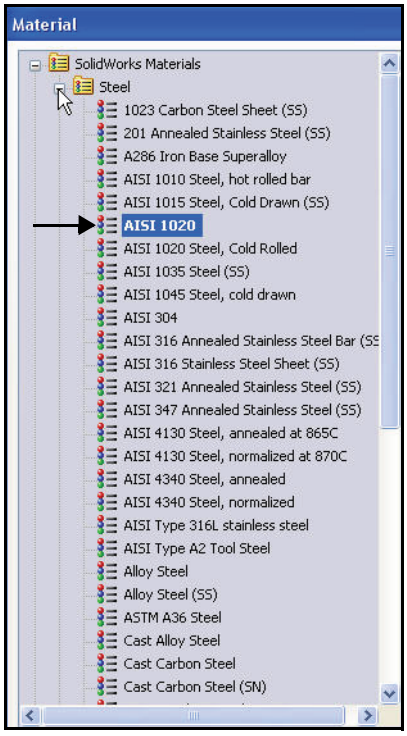
Selecting parts and Applying Material in SolidWorks Simulation

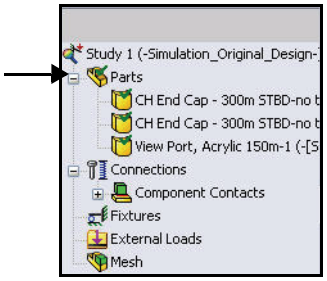




1 **Select the two EndCaps.**
   - Expand the **Parts** folder.
   - Click the first **CH EndCap** part.
   - Hold the **Ctrl** key down.
   - Click the second **CH EndCap** part.
   - Release the **Ctrl** key.
   - Click **Apply Material** from the Simulation tab in the CommandManager. The Material dialog box is displayed.

2 **Assign Material**.
   - Expand the **Steel** folder.
   - Click **AISI 1020**. View the available material properties and information.
   - Click **Apply**.
   - Click **Close** from the Material dialog box. View the results in the Study tree.





Note: A green check mark on the Parts folder indicates that material is assigned to the parts.

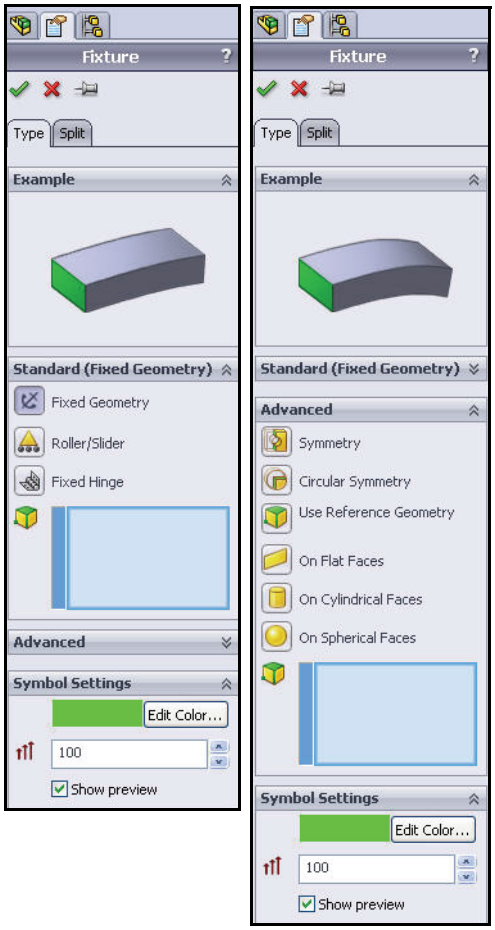
Applying Fixtures

A component that is not fixed will travel indefinitely in the direction of the applied load as a rigid body. Fixtures and loads define the environment of the model.

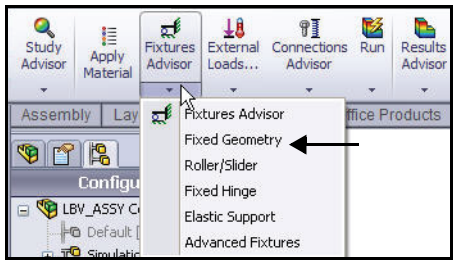
A rigid body contains six degrees of freedom, three rotational and three translational. You apply restraints to remove degrees of freedom.

Each load or fixture condition is represented by an icon in the Study.

In this section, address an On cylindrical face fixture.

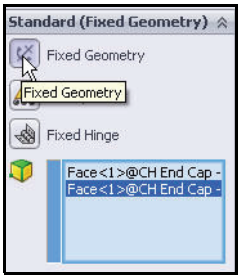


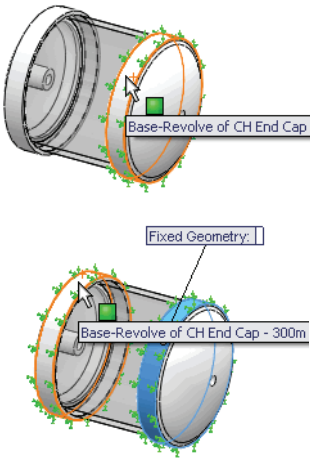
Applying a Fixture



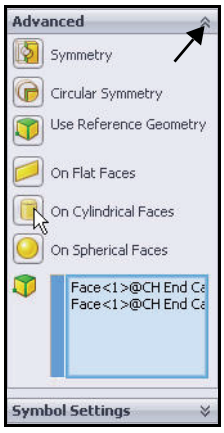
1 **Apply a Fixture.**
- Click the **Fixtures Advisor** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Fixed Geometry**. The Fixture PropertyManager is displayed. The Fixed Geometry option is selected by default. Fix the model to simulate how the two EndCaps are mounted to the Housing.

2 **Select the Faces to be Fixed.**
- Click the cylindrical face of the **right EndCap** as illustrated. Face<1> is displayed in the Standard (Fixed Geometry) box.
- Click the cylindrical face of the **left EndCap** as illustrated.

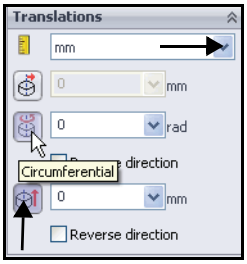


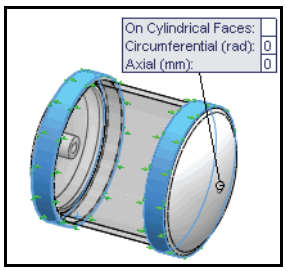


3 **Set Fixture Type.**
- **Expand** the Advanced dialog box.
- Click the **On Cylindrical Faces** box. The Translations dialog box is displayed.



4 Select Units and Displacement Components.

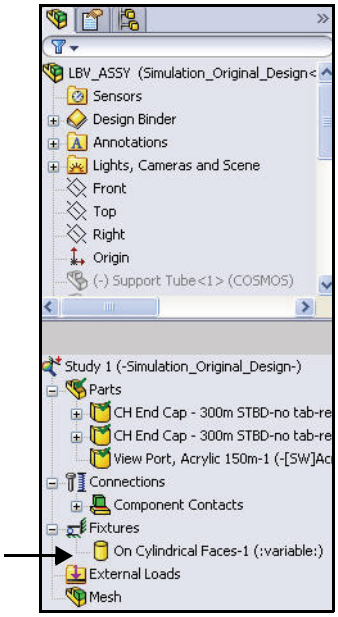


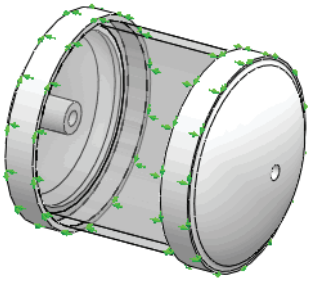


- Select **mm** from the Unit drop-down menu.
- Click the **Circumferential** box.
- Click the **Axial** box. View the results in the Graphics area.

5 Apply the Fixture.

- Click **OK** from the Fixture PropertyManager. An icon named On Cylindrical Faces-1 is displayed in the Fixtures folder.

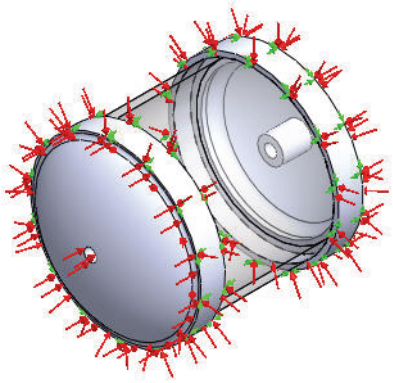




Note: Press the **f** key to fit the model to the Graphics area.

Applying Loads

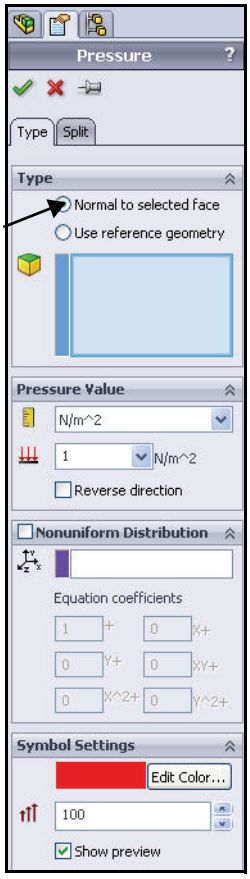
Loads are forces and pressures applied to faces, edges, and vertices of the model. In SolidWorks Simulation you can apply uniform and variable force and pressure, torque, bearing loads, and body forces such as gravity and centrifugal force.



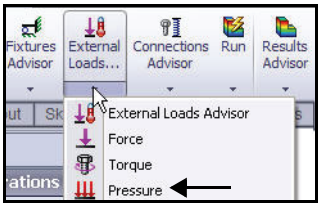
- You will apply a Pressure load to the Housing. The Pressure load will simulate approximately 3,400 feet of seawater.

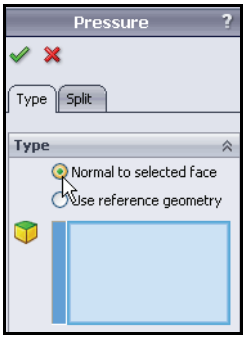
Note: You will use English (IPS) units in this section. Each 33.3 ft. of seawater is approximately equivalent to 1 ATM or 14.7 PSI.

- Apply the **Normal to selected face** option for Pressure Type.
- Select all **exposed faces** of the Housing to apply a pressure load to simulate the seawater depth pressure.



Applying a Pressure Load

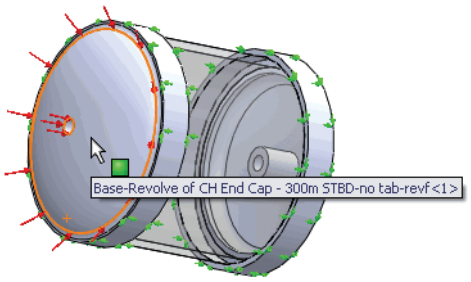




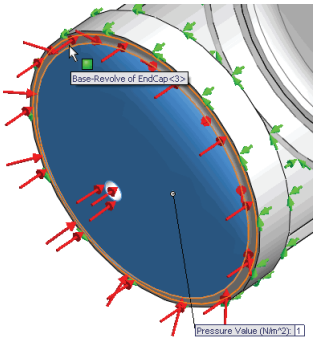
1 **Apply a Pressure Load.**

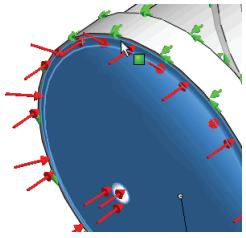
- Click the **External Loads** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Pressure** . The Pressure PropertyManager is displayed. The Type tab is selected by default.
- Click the **Normal to selected face** box.

2 **Select the Faces to Apply the Load.**

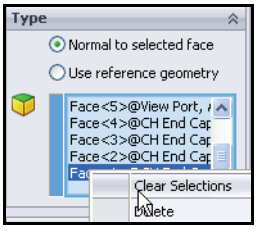


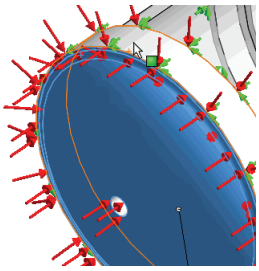
- **Rotate** the model with the middle mouse button as illustrated.
- Click the **front EndCap** as illustrated. Face<1> is displayed in the Faces for Pressure box.





- Zoom in on the **front EndCap** as illustrated.
- Click the other **three faces** of the front EndCap. Face<2>, Face<3>, and Face<4> are displayed in the Faces for Pressure box.



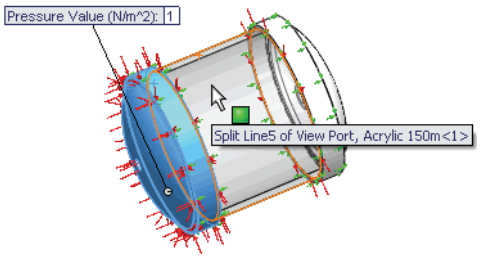


Note: If you select an incorrect face, right-click inside the Faces for Pressure box and click **Delete** if deleting a single face or click **Clear Selections** if you want to clear all entries.

Note: Face ID's in list may vary.

3 **Select the View Port Face.**

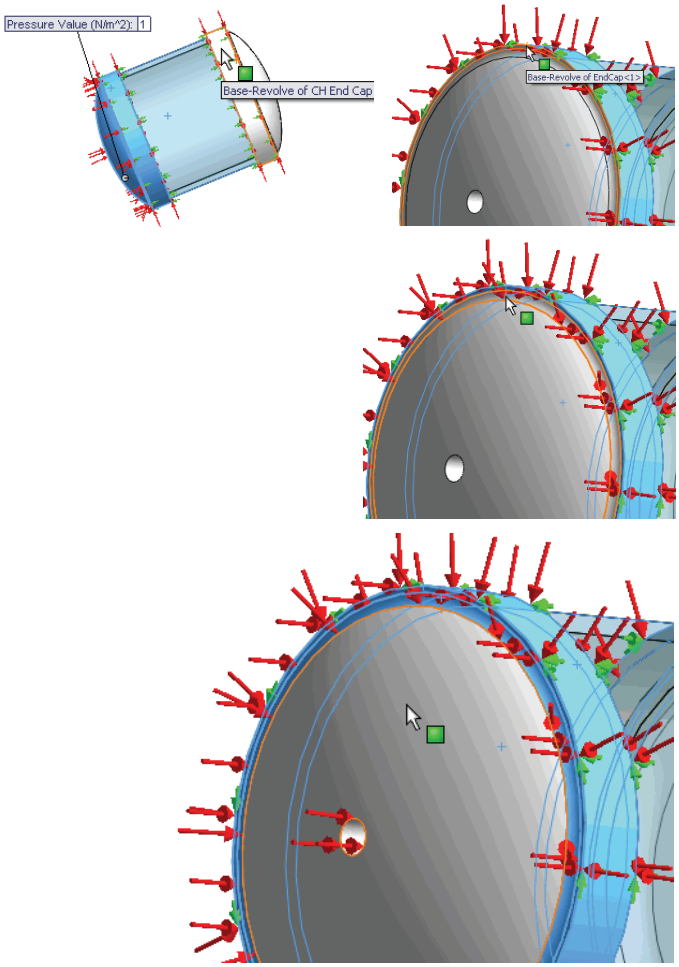
- Press the **f** key to fit the model to the Graphics area.
- **Rotate** the model with the middle mouse button as illustrated.
- Click the **View Port** face. Face<5> is displayed in the Faces for Pressure box. Note the icon feedback symbol for a face and displayed feature information.



Note: Do not select an inside face.

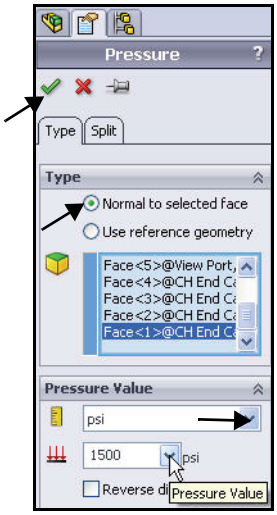
4 **Select the Faces to Apply the Load.**

- Zoom in on the **back EndCap** face as illustrated.
- **Rotate** the model with the middle mouse button to select the other four faces of the back EndCap.
- Click the four faces of the **back EndCap** as illustrated. Nine faces are displayed in the Faces for Pressure box.



5 Set the Pressure Value.

- Select **psi** from the Units drop-down menu.
- Enter **1500** in the Pressure Value box.

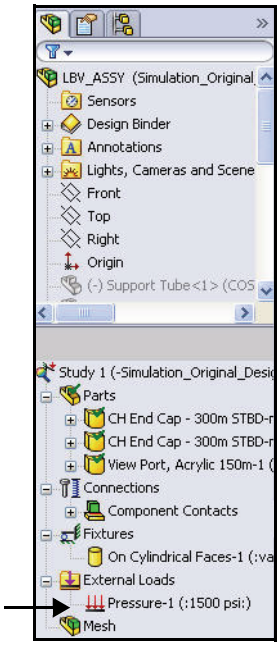


6 Apply the Pressure.

- Click **OK** from the Pressure PropertyManager. SolidWorks Simulation applies 1500 PSI pressure and creates an icon named Pressure-1 in the External Loads folder as illustrated.

7 Fit the model to the Graphics area.

- Press the **f** key. View the model in the Graphics area.



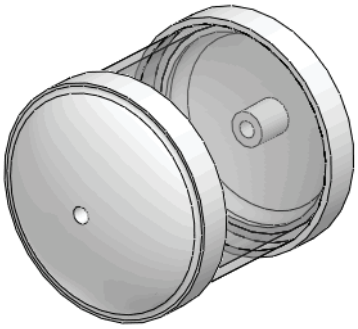
Note: If you change the units after typing a value, SolidWorks Simulation converts the value to the new units.

Creating a Mesh and Running the Analysis

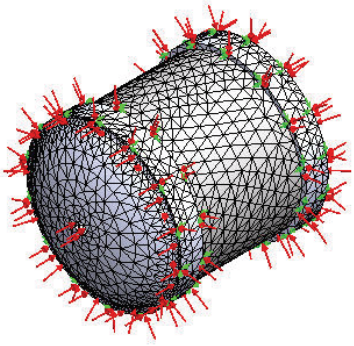
Creating a Mesh is a very crucial step in design analysis. Meshing is basically splitting the geometry into small, simply shaped pieces called finite elements. The automatic mesher in SolidWorks Simulation generates a mesh based on a global element size, tolerance, and local mesh control specifications. Mesh control lets you specify different sizes of elements for components, faces, edges, and vertices.

SolidWorks Simulation estimates a global element size for the model taking into consideration its volume, surface area, and other geometric details. The size of the generated mesh (number of nodes and elements) depends on the geometry and dimensions of the model, element size, mesh tolerance, mesh control, and contact specifications.

Meshing generates 3D tetrahedral solid elements, and 2D triangular shell elements or 1D beam elements. After the mesh is created, you can run the analysis. SolidWorks Simulation solves a series of equations based on known material properties, restraints, and loads. The Static solutions provide information on displacement, stress, and strain.

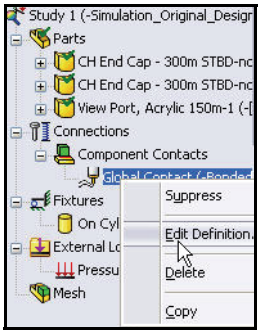


Before Meshing



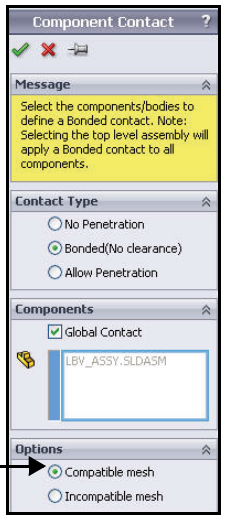
After Meshing

Creating a Compatible Mesh

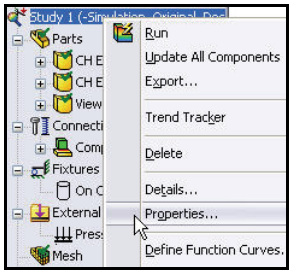


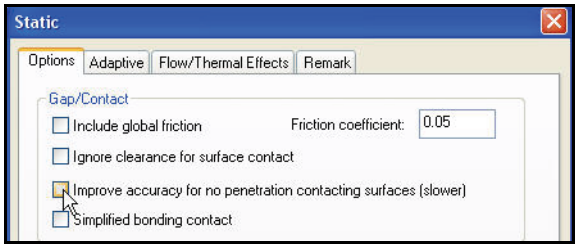
1 **Create a Compatible Mesh**

- Expand **Component Contact** from the Study tree.
- Right-click **Global Contact (-Bonded-)**.
- Click **Edit Definition**. The Component Contact PropertyManager is displayed.
- Click **Compatible mesh** from the Options box. Accept the default settings.
- Click **OK** from the Component Contact PropertyManager. In the next section, start the Meshing process.



Note: You can also right-click Study 1 and click Properties to set mesh compatibility. Check the Improve accuracy for contacting surfaces with incompatible mesh box.

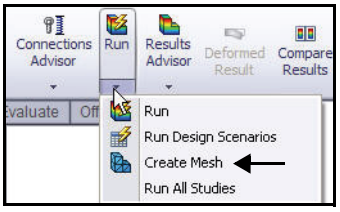




Creating a Mesh

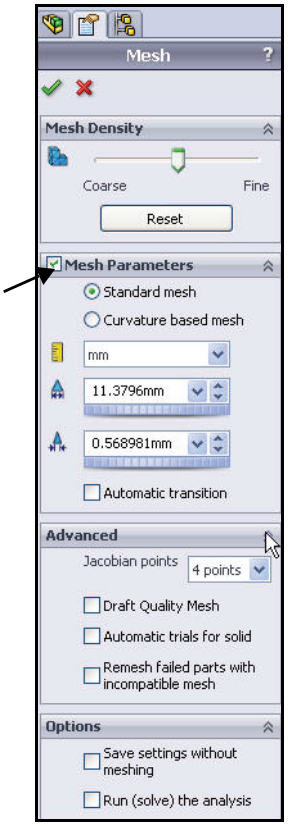
1 **Create a Mesh.**

- Click the **Run** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Create Mesh** . The Mesh PropertyManager is displayed suggesting Global Size and Tolerance values.



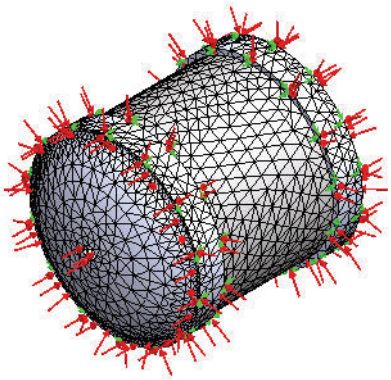
2 **Review the Meshing Options.**

- Expand the **Mesh Parameters** box. View the available options.
- Expand the **Advanced** box. View the available advanced options for additional control.



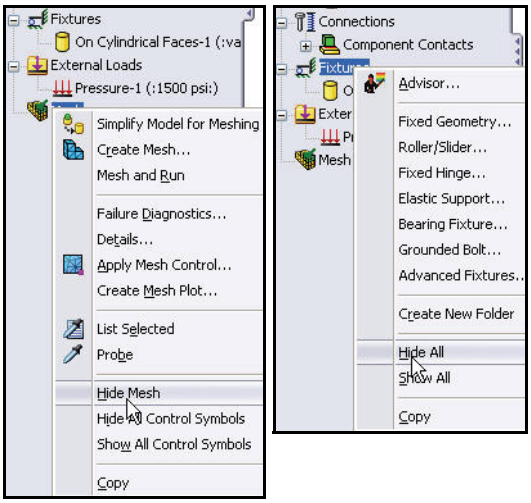
3 Start the Mesh Process.

- Click **OK** from the Mesh PropertyManager. Meshing starts and the Mesh Progress window appears. After meshing is completed, SolidWorks Simulation displays the meshed model. A green check mark is applied next to the Mesh folder in the Study.



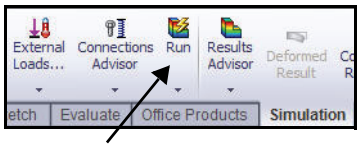
Note: Right-click **Mesh**. Click **Hide Mesh/Show Mesh** to toggle the visibility of the mesh.

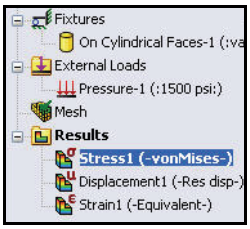
Note: Right-click **Fixtures**. Click **Hide All/Show All** to toggle the visibility of the loads and fixtures.



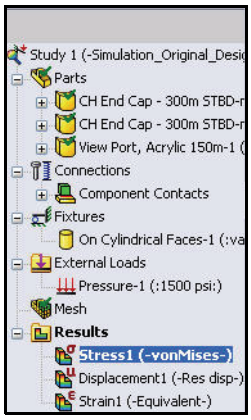
4 Run the Analysis.

- Click **Run** from the Simulation tab in the CommandManager. Three default plots are created.





Viewing the Results

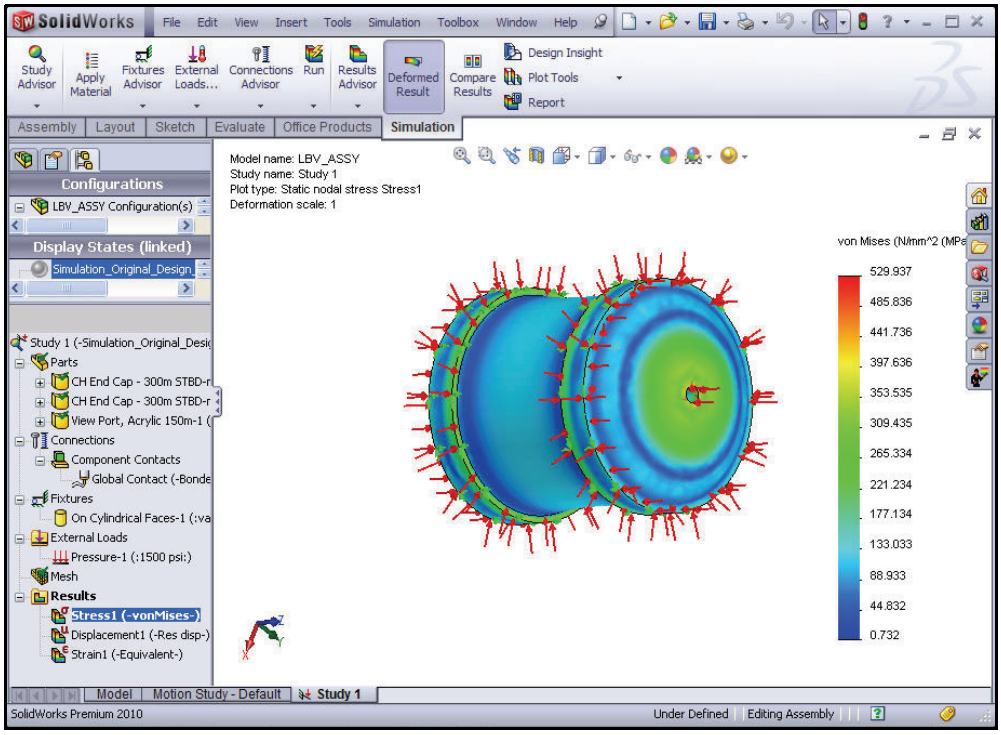


After a successful run of a Static analysis, SolidWorks Simulation creates three default plots: Stress, Displacement, and Strain.

The results are utilized with your design criteria to answer the following questions:

- Will the model fail?
- How will the model deform?
- Can you reduce material or change material without affecting performance?

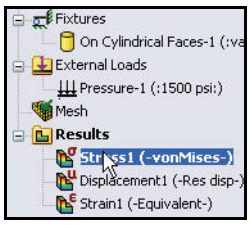
Note: Results may vary depending on the mesh speed.



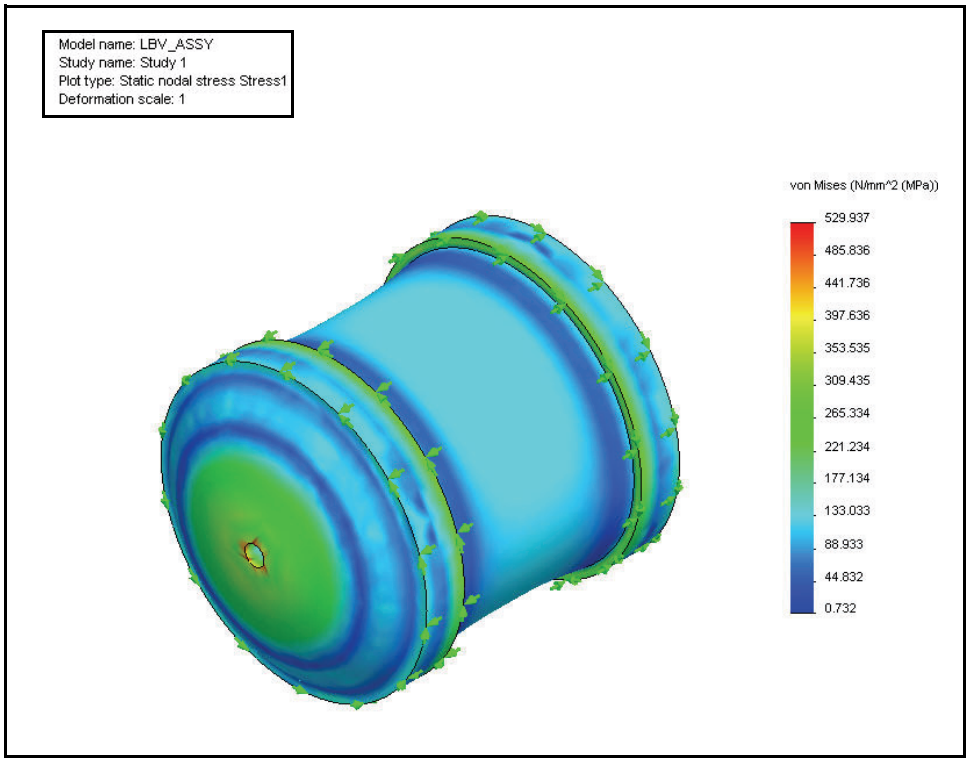
View the Results

1 **Hide the External Loads.**
   - Right-click the **External Loads** folder.
   - Click **Hide All**.

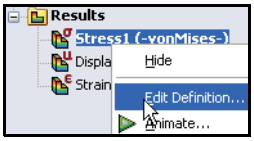
2 **View the von Mises Stresses.**
   - Double-click **Stress1 (-von Mises-)**. The Stress Plot PropertyManager is displayed. Plot units if needed can be modified from the PropertyManager.
   - Click **OK** from the Stress Plot PropertyManager.



Note: The von Mises stress indicates the internal forces in a body when subjected to external loads for ductile materials. Most engineering materials are ductile.

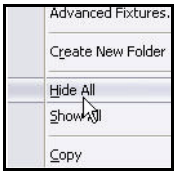


Note: To view the stress plot in a different unit system, right-click the active plot icon. Click **Edit Definition**. Set **units**. Click **OK** from the Stress Plot PropertyManager.



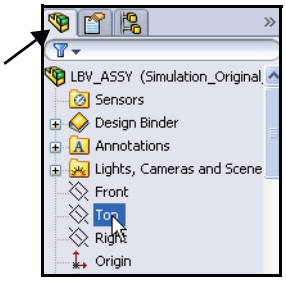
3 Hide the Fixtures.

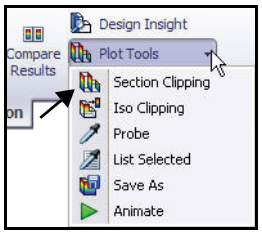
- Right-click the **Fixtures** folder.
- Click **Hide All**.

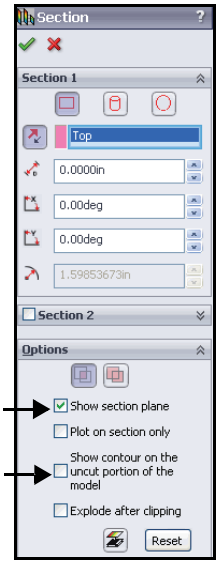


4 Display a Section View using the Top Plane.

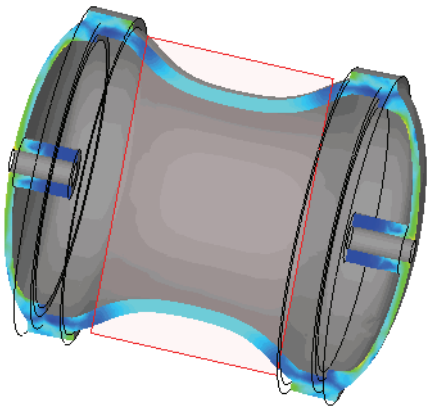
- Click the SolidWorks **FeatureManager** tab.
- Click **Top** to select Top Plane as illustrated.
- Click the **Plot Tools** drop-down arrow from the Simulation tab in the CommandManager.
- Click the **Section Clipping** tool as illustrated. The Section PropertyManager is displayed. Top is displayed in the Reference entity box.
- Check the **Show section plane** box.
- Un-check the **Show contour on the uncut portion of the model** box. View the default settings.







- Click **OK** from the Section PropertyManager.
- **Rotate** the model as illustrated with the middle mouse button to view the results.

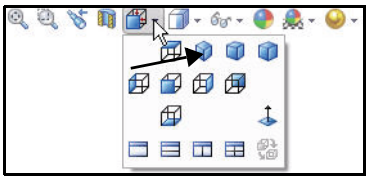


Note: Deformation is magnified for improved visibility. The deformation can be displayed at any scale.

Note: Use the **Zoom to Area** tool located in the Heads-up View toolbar to Zoom in on a section of the model.

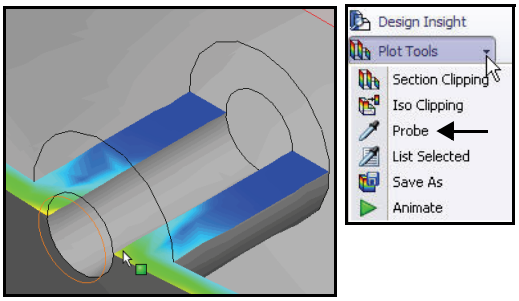
5 Display an Isometric view.

Click **Isometric** view from the Heads-up View toolbar.

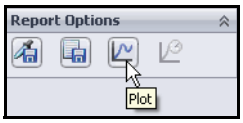


6 Probe the Model.

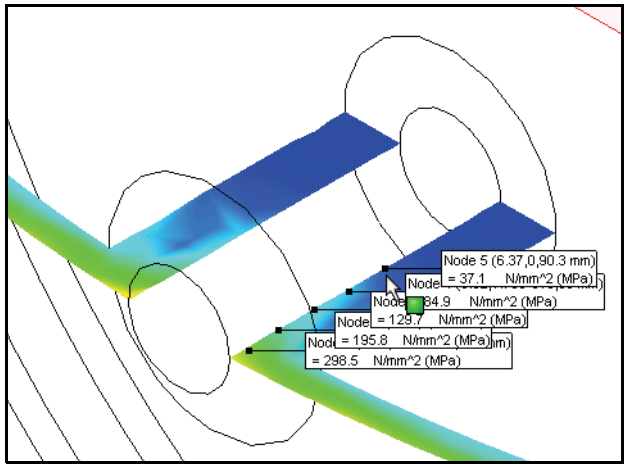
- Zoom in on the **front EndCap**.
- Click the **Plot Tools** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Probe**. The Probe Results PropertyManager is displayed.



- Click **five points** from front to back as illustrated.
- Click the **Plot** button from the Report Options box. View the results.



Note: Results will vary depending on the selected location of the points.



7 Review the Plot.

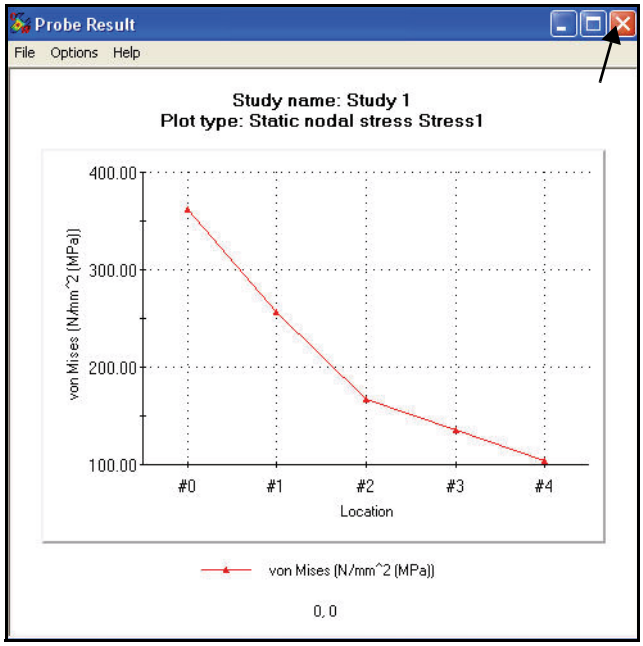
- Review the plot. This is an excellent way to examine the variation in stress across the geometry of your part.

8 Close the Probe Results dialog box.

- **Close** the Probe Results dialog box.

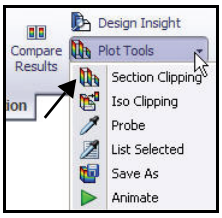
9 Close the Probe Result PropertyManager.

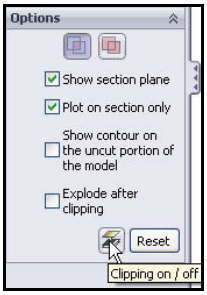
- Click **OK** from the Probe Result PropertyManager.



10 Deactivate the Section Plot.

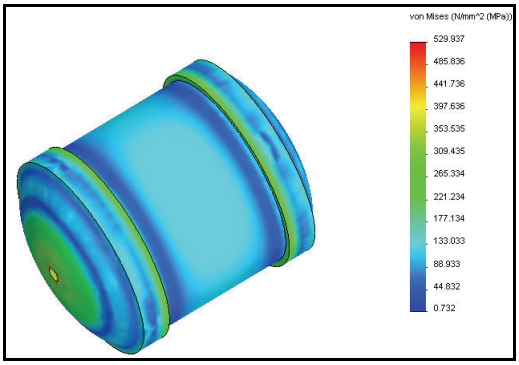
- Click the **Plot Tools** drop-down arrow from the Simulation tab in the CommandManager.
- Click the **Section Clipping** tool. The Section PropertyManager is displayed.
- Click the **Clipping on/off** button from the Options box as illustrated.
- Click **OK** from the Section PropertyManager.





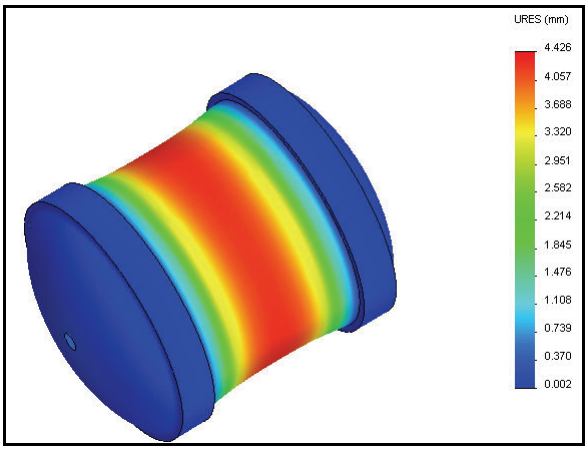
11 Fit the model to the Graphics area.

- Press the **f** key. View the results in the Graphics area.

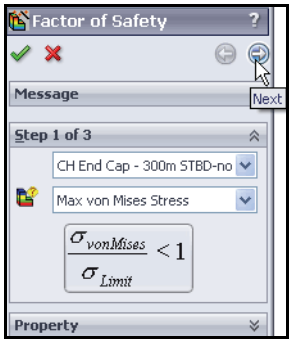


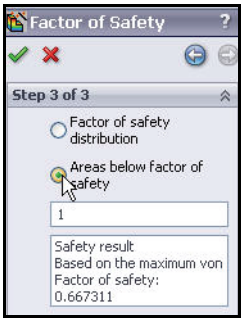
12 View the Displacement Plot.

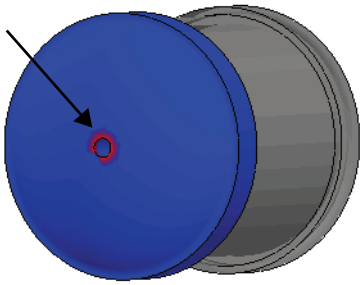
- Double-click **Displacement1 (-Res disp-)** in the Results folder. View the plot.



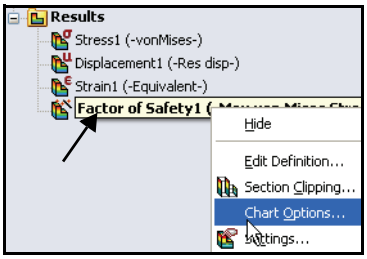
- Click **Next** to continue to step 2. Accept the defaults.
- Click **Next** to continue to step 3.
- Click the **Areas below factor of safety** box.
- Click **OK** from the Factor of Safety PropertyManager. View the model in the Graphics area.
- **Rotate** the model with the middle mouse button. The area in blue has a FOS above 1. The area in red has a FOS below 1.





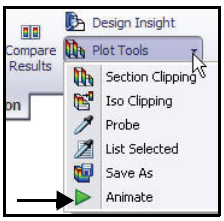


- Right-click **Factor of Safety1** as illustrated from the Results folder.
- Click **Chart Options**. The Chart Options PropertyManager is displayed.



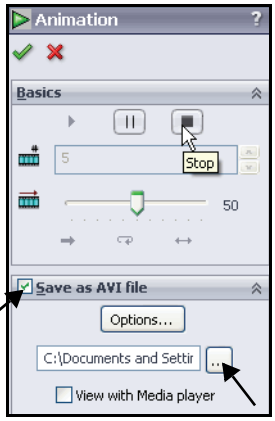
13 Animate the Displacement Plot.

- Click the **Plot Tools** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Animate** . The Animation PropertyManager is displayed. View the animation in the Graphics area.



14 Stop the Animation.

- Click **Stop** .

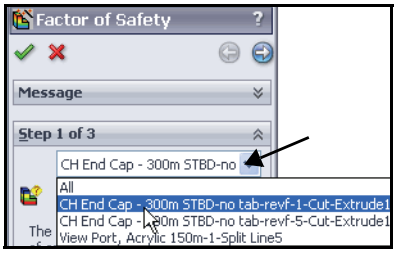


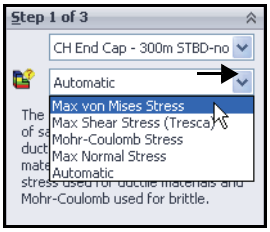
15 Save the Animation.

- Check the **Save as AVI file** box as illustrated.
- Click the **Browse** button. Accept the default location.
- Click **Save** from the Save As dialog box.
- Click **OK** from the Animation PropertyManager.

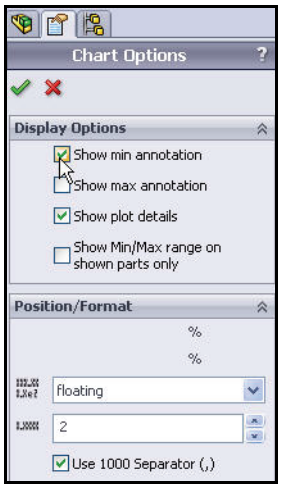
16 Calculate the Factor of Safety.

- Right-click the **Results** folder.
- Click the **Define Factor Of Safety Plot** tool. The Factor of Safety PropertyManager is displayed.
- Select the first **CH End Cap** component as illustrated from the drop-down menu.
- Select **Max von Mises Stress** from the drop-down menu as Criterion. Note your options for Criterion.





- Check the **Show min annotation** box. Accept the defaults settings. View the results in the Graphics area.
- Click **OK** from the Chart Options PropertyManager. View the results.
- **Rotate** the model with the middle mouse button. View the area in red. The area in red has a FOS below 1. The area in blue has a FOS above 1.



Note: The minimum FOS is 0.67. You did not meet the design goal, which is to obtain a FOS greater than one. In the next study, add structural ribs to the EndCap to obtain the design goal.