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

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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.

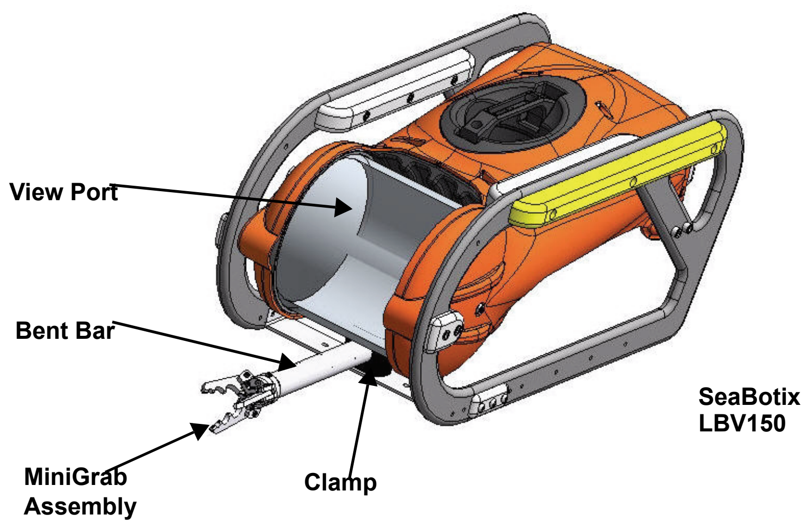
The SeaBotix LBV150

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.

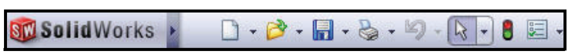
User Interface

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

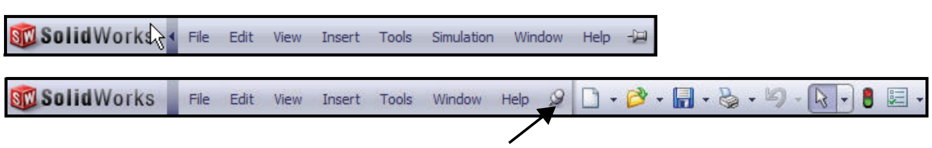
The Menu Bar toolbar contains a set of the most frequently used tool buttons. The available tools are: **New** - Creates a new document, **Open** - Opens an existing document, **Save** - Saves an active document, **Print** - Prints an active document, **Undo** - Reverses the last action, Select - Selects 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.



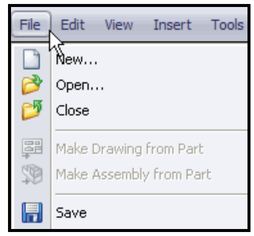
Menu Bar Menu

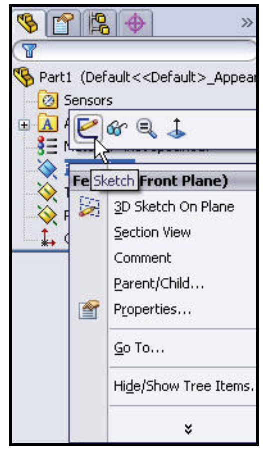
Click the SolidWorks name in the Menu Bar toolbar to display the default Menu Bar menu. SolidWorks provides a context-sensitive menu structure. The menu tittles remain the same for all three types of documents; part, assembly, and drawing but the menu items change depending on which type of document 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.

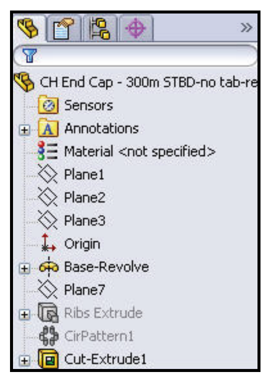
When you select, (click or right-click) items in the Graphics area or FeatureManager, Context toolbars appear and provide access to frequently performed actions for that context.

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.

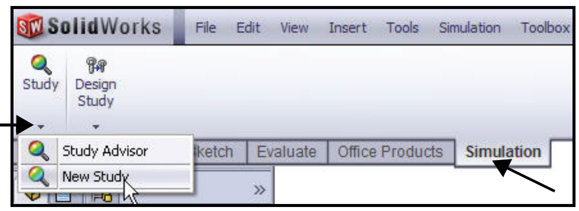
As features are created, they are added to the FeatureManager. As a result, the FeatureManager represents the chronological sequence of modeling operations. The FeatureManager also allows access to editing the features and objects that it contains. The Part FeatureManager consist of four default tabs:

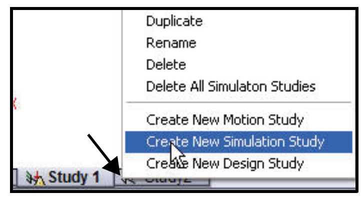
FeatureManager , **PropertyManager** , **ConfigurationManager** , and **DimXpertManager** .

SolidWorks Simulation CommandManager Tab

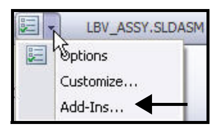
The SolidWorks Simulation CommandManager enables you to quickly create a Simulation Study. Click the SolidWorks Simulation tab in the CommandManager to create a new study. Studies are organized in tabs and are displayed in the bottom section of the Graphics area.

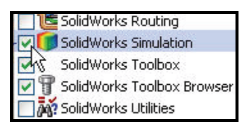
Note: Create a New Study using the **New Study** tool or right-click on a **Study** tab, click **Create New Simulation Study**.





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.



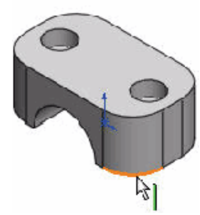


Mouse Buttons

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

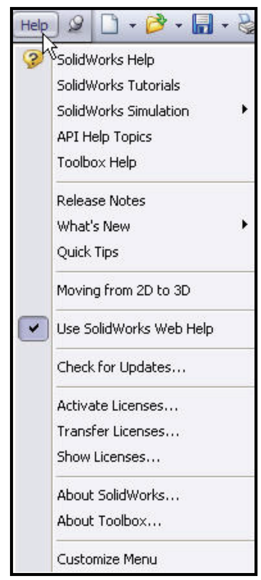
- **Left** — Selects objects such as geometry, menu buttons, and objects in the FeatureManager design tree.
- **Middle** — Holding the middle mouse button as you drag the mouse rotates the view. Holding the **Shift** key down while you use the middle mouse button zooms the view. Using the **Ctrl** key 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

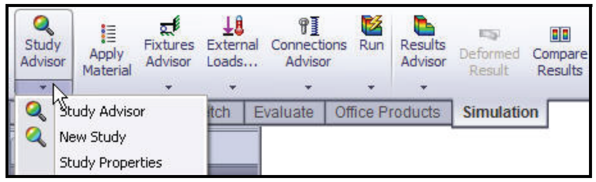


SolidWorks has a comprehensive Home help Page function that is design to assist the new and experience user. It provides information on What's New, SolidWorks Glossary, New Release notes, and more.

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.

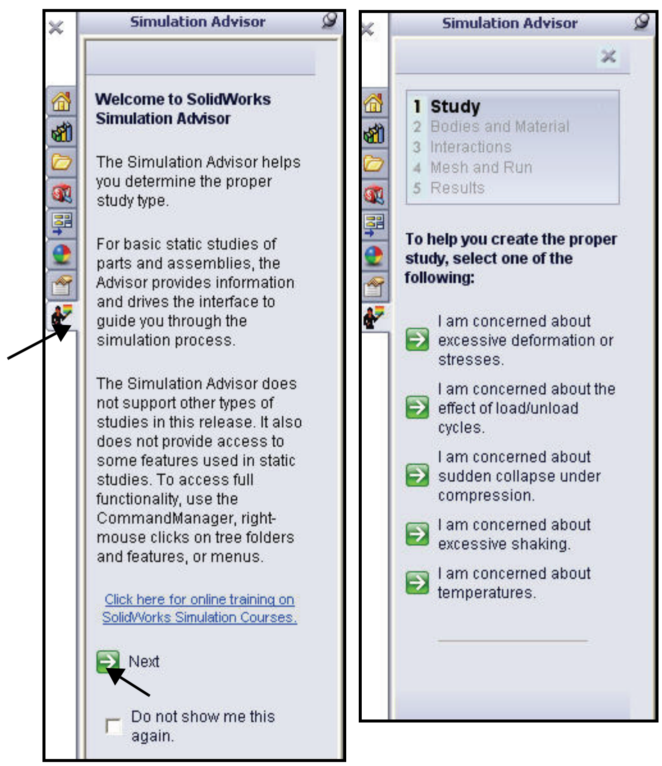


Click **Study Advisor**, **Study Advisor**  from the Simulation tab in the CommandManager with an active study to obtain the Simulation Advisor.

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.

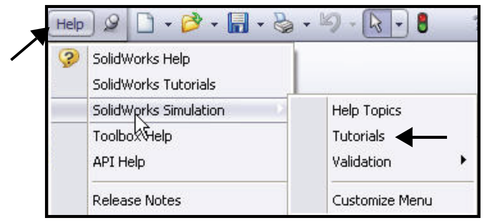
Note: The Simulation Advisor  tab is displayed in the Task Pane.

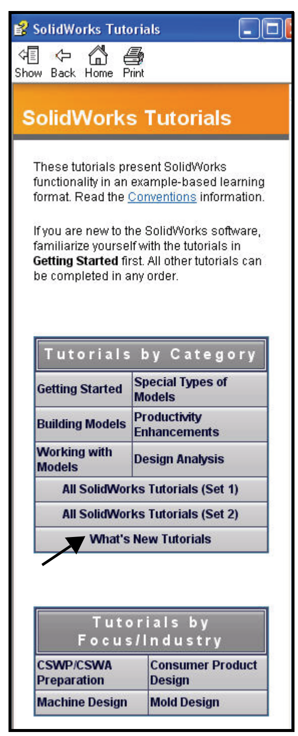


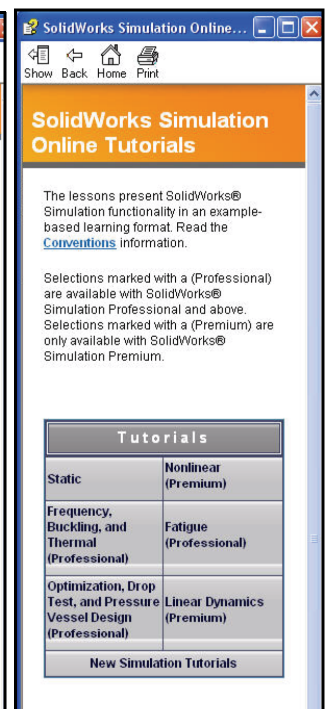
SolidWorks Tutorials and SolidWorks Simulation Tutorials

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

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

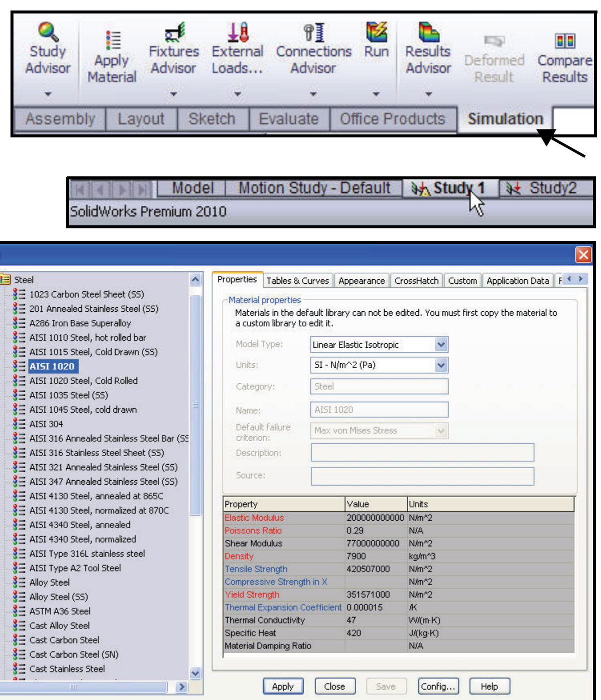
SolidWorks and SolidWorks Simulation

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.

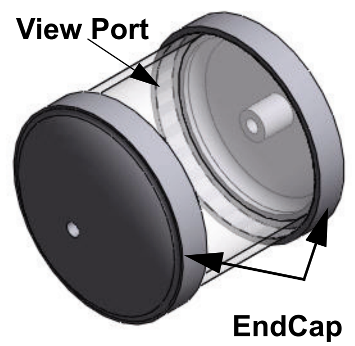
Anyone who can design a model in SolidWorks can analyze it without having to learn a new user interface. SolidWorks Simulation utilizes the power of SolidWorks configurations to test multiple designs. Plus, since SolidWorks Simulation uses native SolidWorks geometry, design changes made in one application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



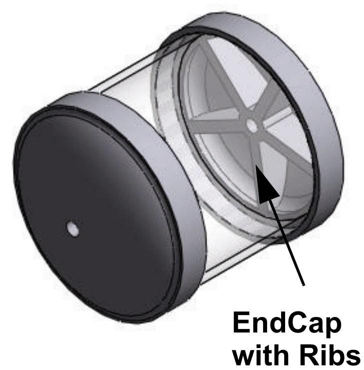
Analyze the Housing

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

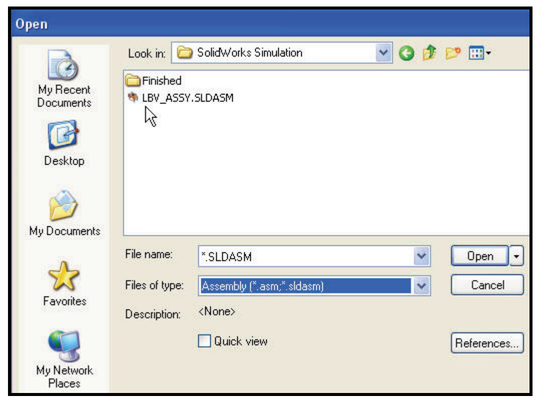
1 Start a SolidWorks Session.

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

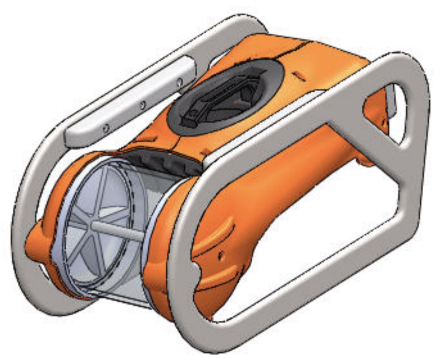
Note: You can quickly start a SolidWorks 2010 session by double-clicking the left mouse button on the desktop shortcut, if there is a shortcut icon on the system desktop.

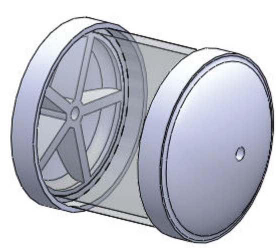
2 Open the SeaBotix LBV150 Assembly.

- 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.

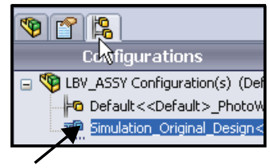


Note: The FeatureManager design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing. This makes it easy to see how the model or assembly was constructed or to examine the various sheets and views in a drawing.

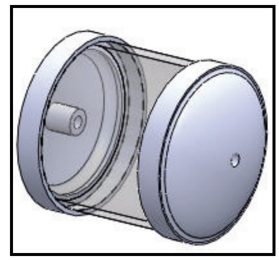




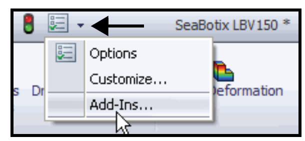
3 Select the Simulation_Original_Design Configuration.



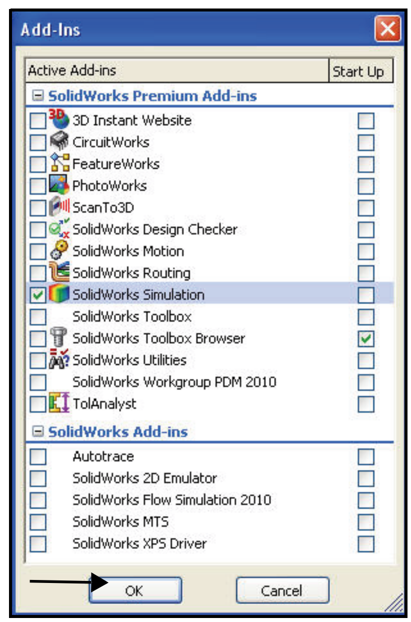
- Click the **ConfigurationManager** tab. The various configurations are displayed.
- Double-click the **Simulation_Original_Design** configuration. The Housing assembly (No Ribs) is displayed in the Graphics area.



4 Activate SolidWorks Simulation.

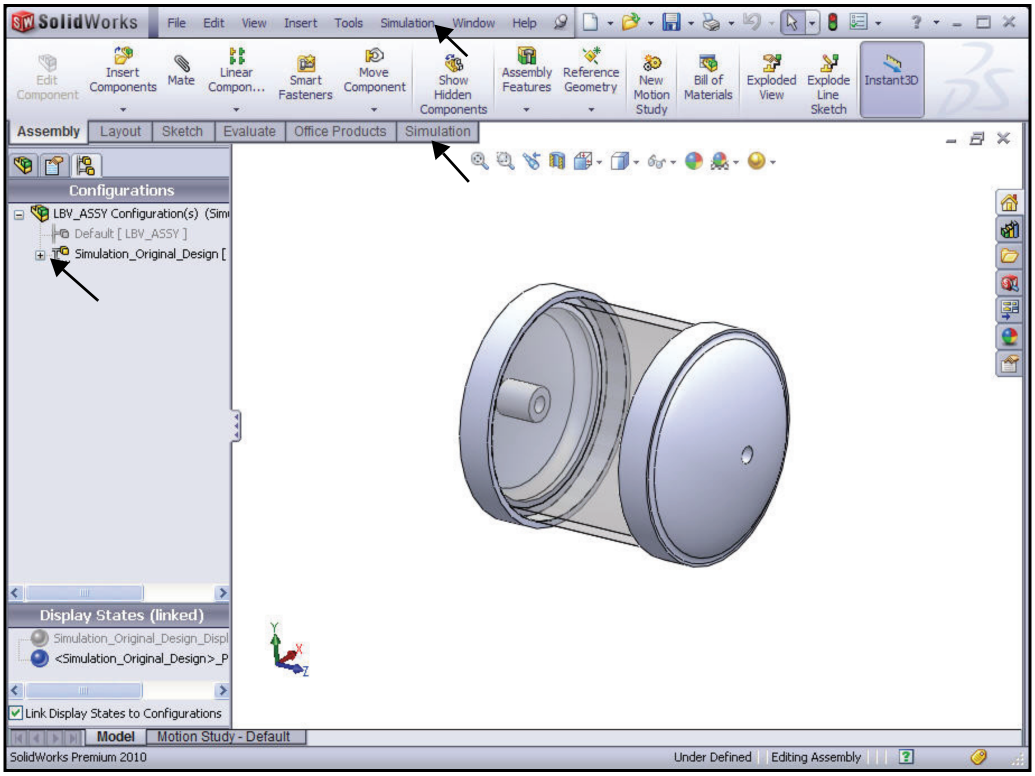


- Click the **Options** drop-down arrow as illustrated from the Menu bar toolbar.
- Click **Add-Ins**. The Add-Ins dialog box is displayed.
- Check the **SolidWorks Simulation** box.
- Click **OK** from the Add-Ins dialog box.



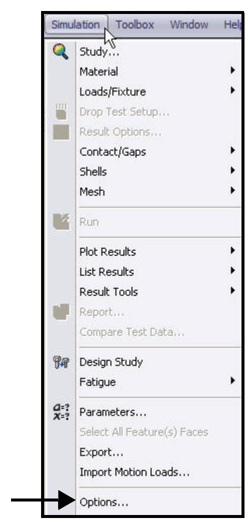
Note: Displayed Add-Ins may vary per system setup.

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

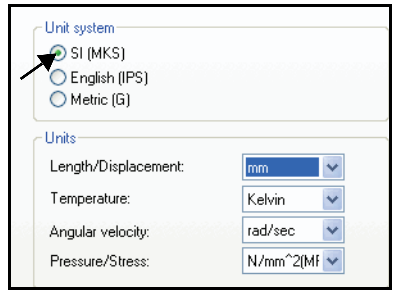


5 Set Default Options in SolidWorks Simulation.

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

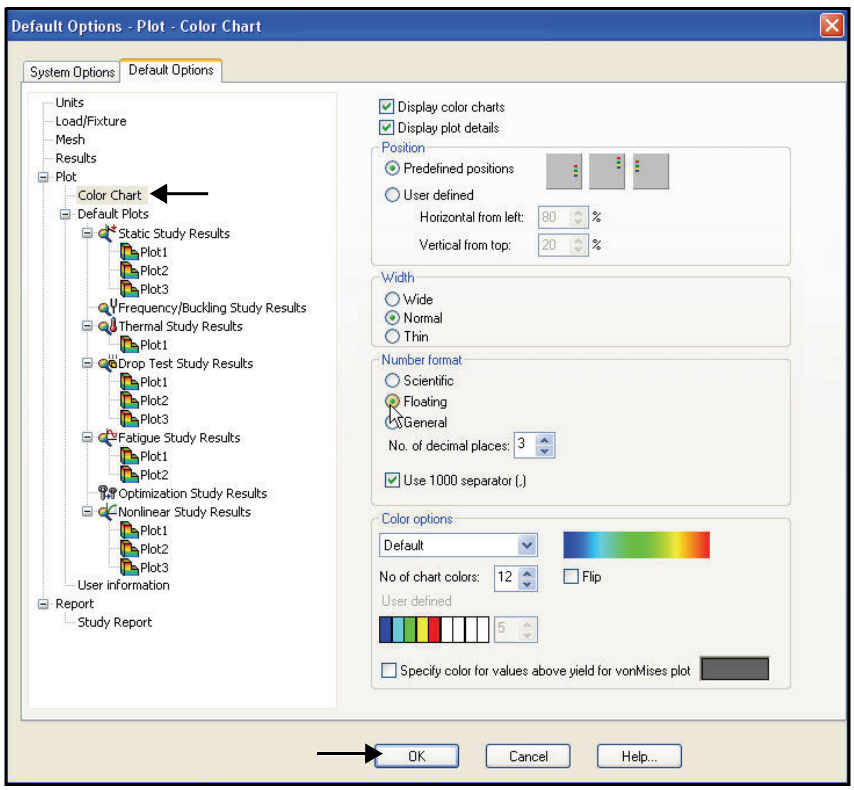


- Click the **Defaults Options** tab. View the Default Options - Unit dialog box.
- Click the **Units** folder.
- Click the **SI (MKS)** Unit system box.
- Select **mm** for Length/Displacement.
- Select **Kelvin** for Temperature.
- Select **rad/sec** for Angular velocity.
- Select **N/mm^2(MPa)** for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

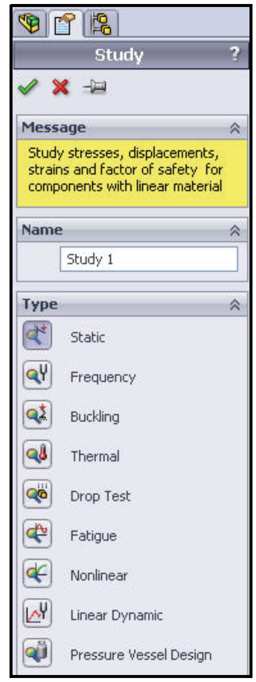


6 Set Number format.

- 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.

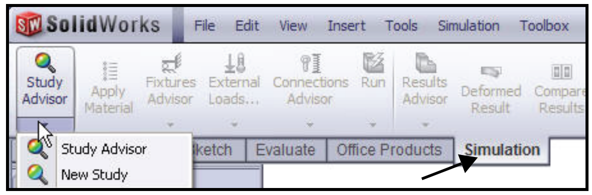
The first default Study name is Study 1.

SolidWorks Simulation offers six different results options. They are:

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

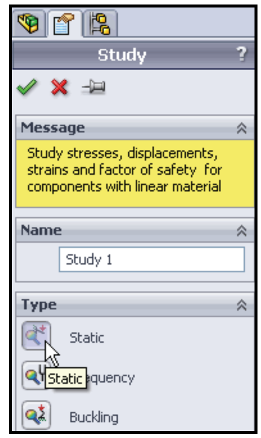
Static studies can help you avoid failure due to high stresses. A factor of safety less than one indicates likely material failure. Large factors of safety in a continuous region indicate that you can probably remove some material from this region.

Creating a Static Analysis Study



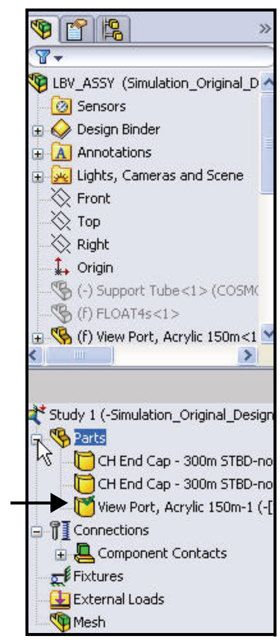
1 Create a Static Analysis Study.

- 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.

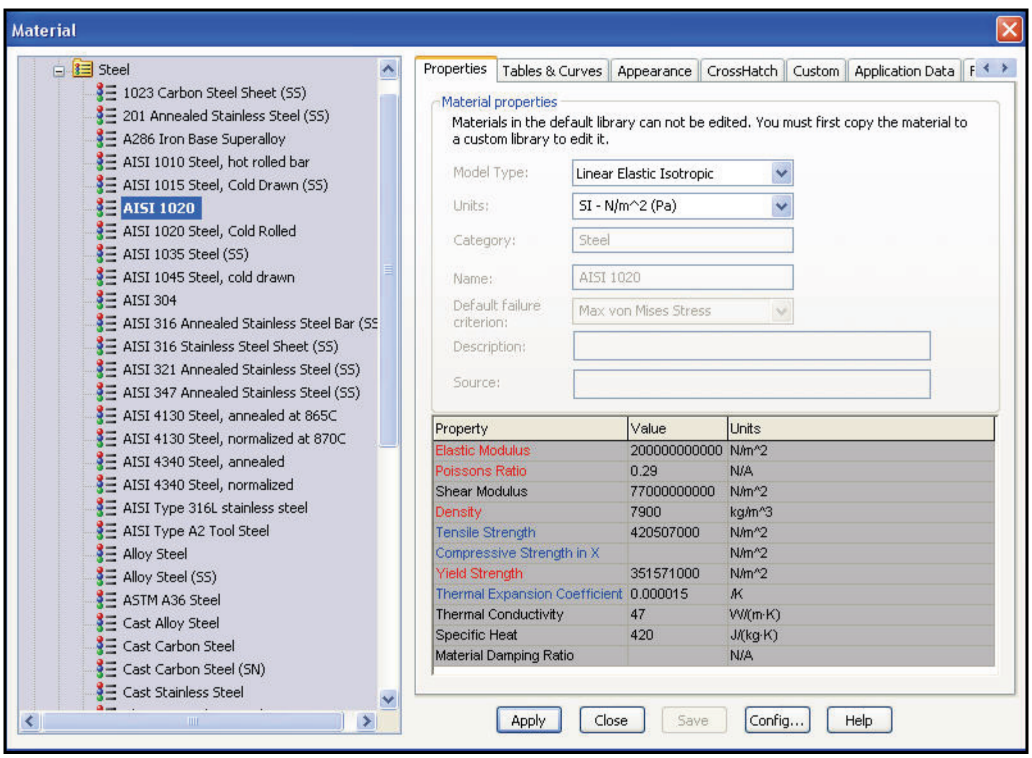
Assigning Materials in SolidWorks Simulation

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

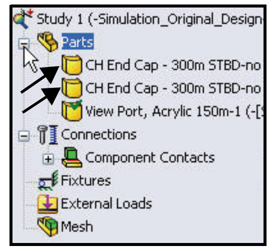
The Properties tab in the Material dialog box allows you to define a material source, material model, and material properties. You can define constant or temperature-dependent properties.

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

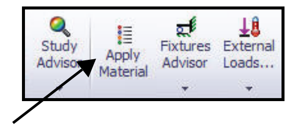
Define and apply material to the two EndCaps in the Housing assembly in the next section.



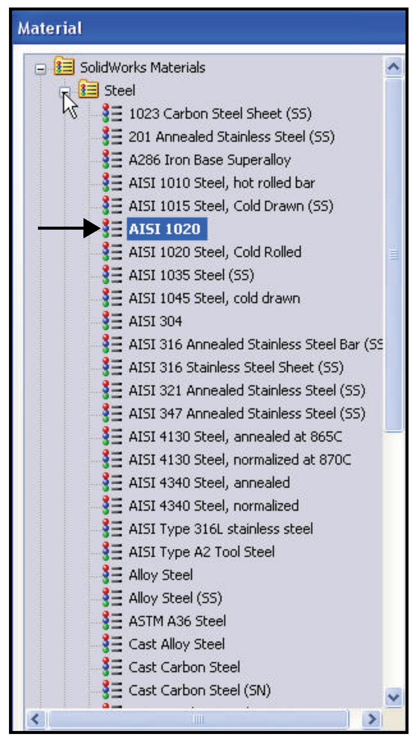
Selecting parts and Applying Material in SolidWorks Simulation



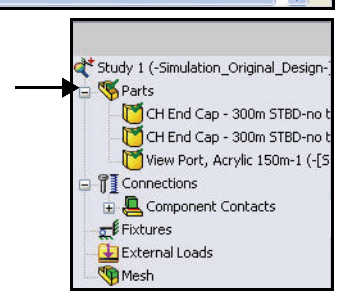
1 **Select the two EndCaps.**
- Expand the **Parts** folder.
- Click the first **CH EndCap** part.
- Hold the **Ctrl** key down.
- Click the second **CH EndCap** part.
- Release the **Ctrl** key.
- Click **Apply Material** from the Simulation tab in the CommandManager. The Material dialog box is displayed.



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.



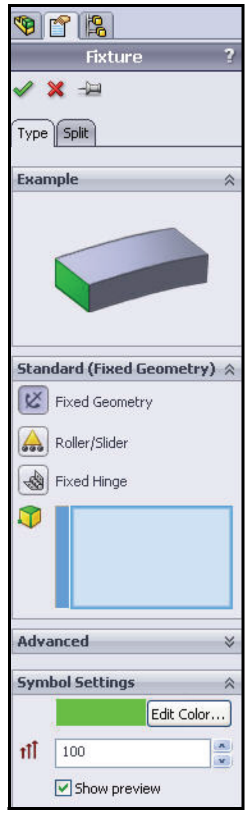
Applying Fixtures

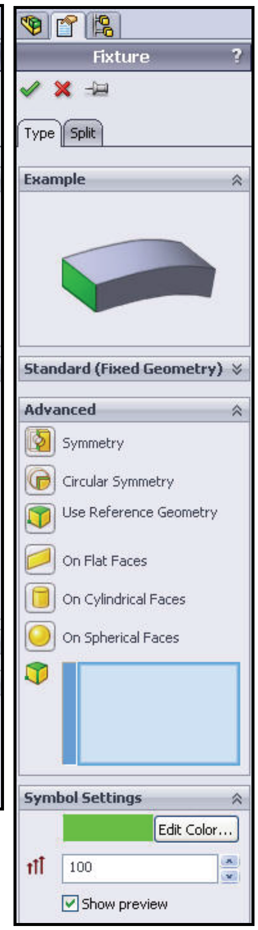
A component that is not fixed will travel indefinitely in the direction of the applied load as a rigid body. Fixtures and loads define the environment of the model.

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.

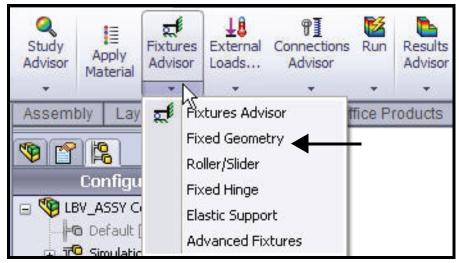




Applying a Fixture

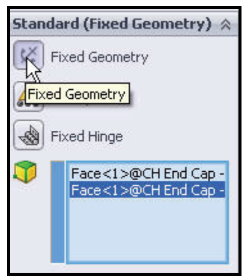
1 Apply a Fixture.

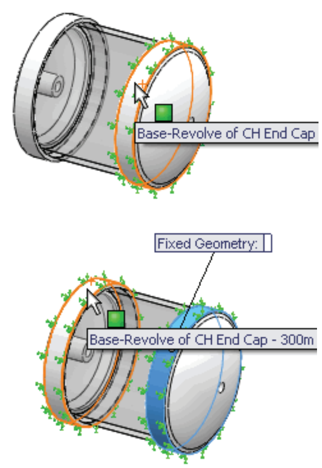
- Click the **Fixtures Advisor** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Fixed Geometry**. The Fixture PropertyManager is displayed. The Fixed Geometry option is selected by default. Fix the model to simulate how the two EndCaps are mounted to the Housing.



2 Select the Faces to be Fixed.

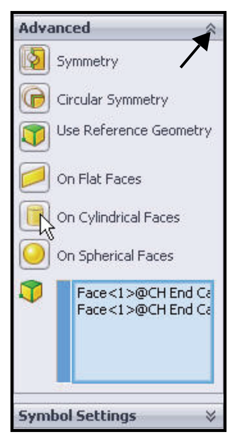
- Click the cylindrical face of the **right EndCap** as illustrated. Face<1> is displayed in the Standard (Fixed Geometry) box.
- Click the cylindrical face of the **left EndCap** as illustrated.





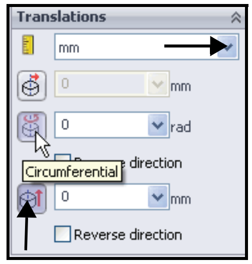
3 Set Fixture Type.

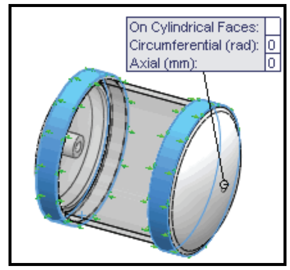
- **Expand** the Advanced dialog box.
- Click the **On Cylindrical Faces** box. The Translations dialog box is displayed.



4 Select Units and Displacement Components.

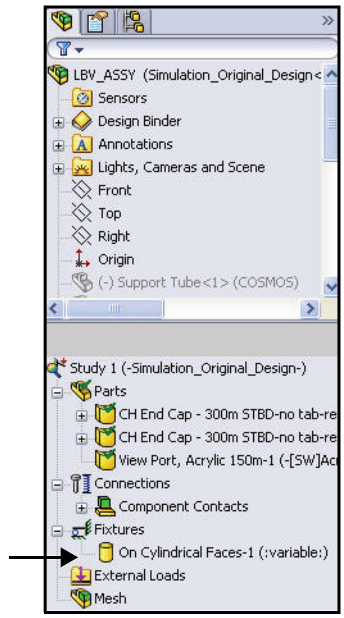
- Select **mm** from the Unit drop-down menu.
- Click the **Circumferential** box.
- Click the **Axial** box. View the results in the Graphics area.



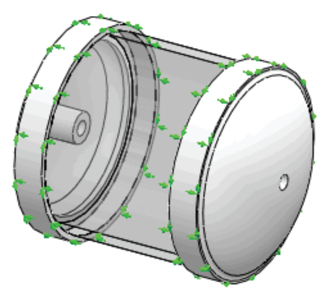


5 Apply the Fixture.

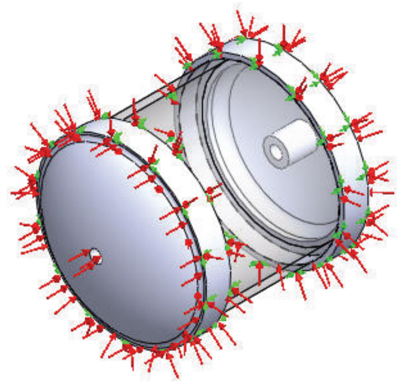
- Click **OK** from the Fixture PropertyManager. An icon named On Cylindrical Faces-1 is displayed in the Fixtures folder.



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



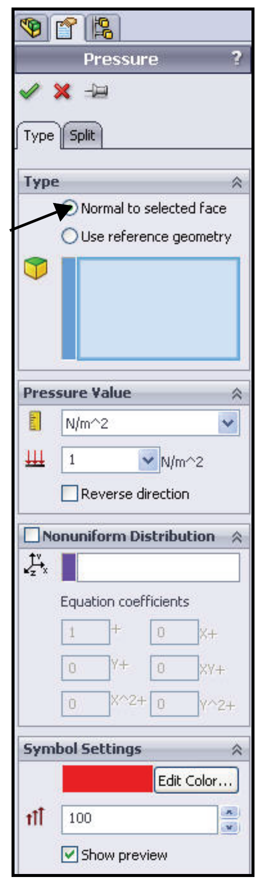
Applying Loads



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.

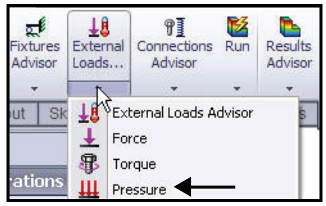
- You will apply a Pressure load to the Housing. The Pressure load will simulate approximately 3,400 feet of seawater.

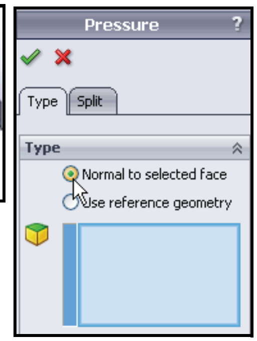
Note: You will use English (IPS) units in this section. Each 33.3 ft. of seawater is approximately equivalent to 1 ATM or 14.7 PSI.



- 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

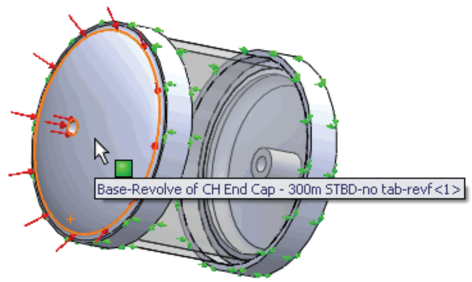




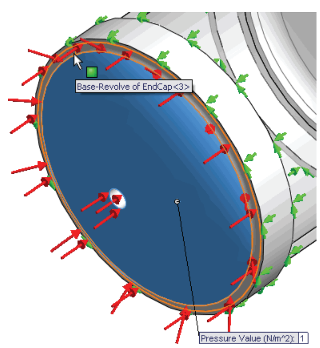
1 Apply a Pressure Load.

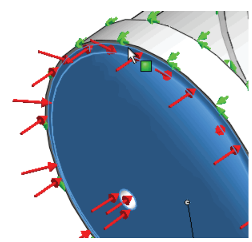
- Click the **External Loads** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Pressure** . The Pressure PropertyManager is displayed. The Type tab is selected by default.
- Click the **Normal to selected face** box.

2 Select the Faces to Apply the Load.



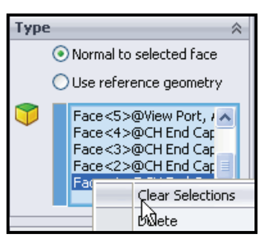
- **Rotate** the model with the middle mouse button as illustrated.
- Click the **front EndCap** as illustrated. Face<1> is displayed in the Faces for Pressure box.

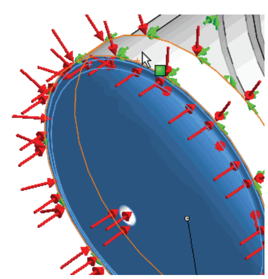




- Zoom in on the **front EndCap** as illustrated.
- Click the other **three faces** of the front EndCap. Face<2>, Face<3>, and Face<4> are displayed in the Faces for Pressure box.

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.

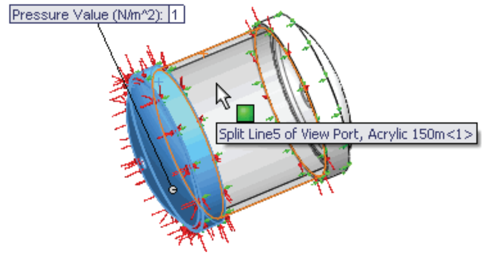




Note: Face ID's in list may vary.

3 Select the View Port Face.

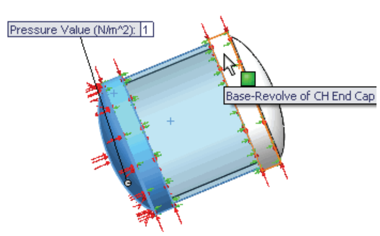
- Press the **f** key to fit the model to the Graphics area.
- **Rotate** the model with the middle mouse button as illustrated.
- Click the **View Port** face. Face<5> is displayed in the Faces for Pressure box. Note the icon feedback symbol for a face and displayed feature information.

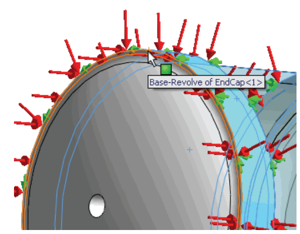


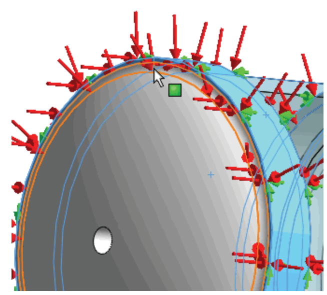
Note: Do not select an inside face.

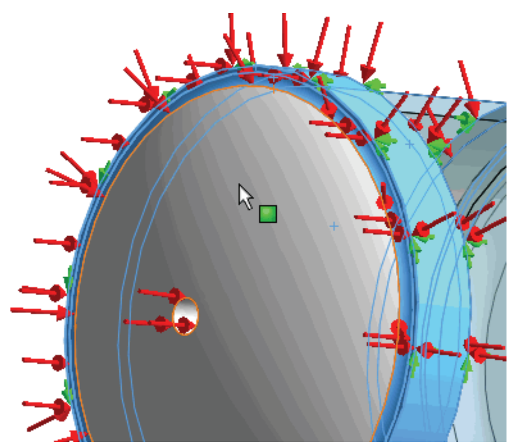
4 Select the Faces to Apply the Load.

- 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.



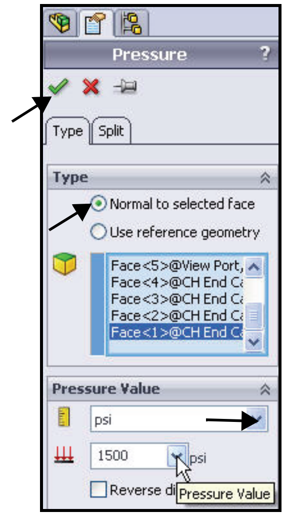






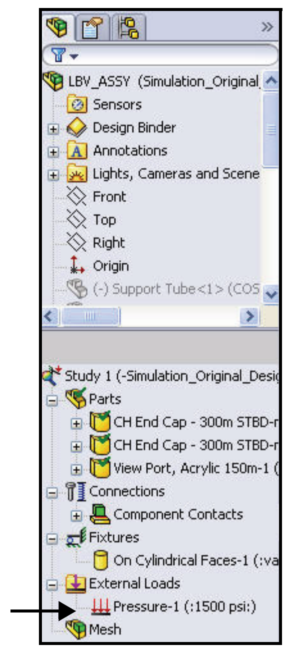
5 Set the Pressure Value.

- 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.

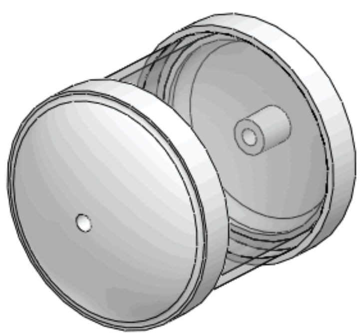
Note: If you change the units after typing a value, SolidWorks Simulation converts the value to the new units.

Creating a Mesh and Running the Analysis

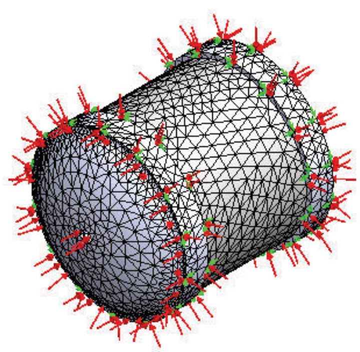
Creating a Mesh is a very crucial step in design analysis. Meshing is basically splitting the geometry into small, simply shaped pieces called finite elements. The automatic mesher in SolidWorks Simulation generates a mesh based on a global element size, tolerance, and local mesh control specifications. 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

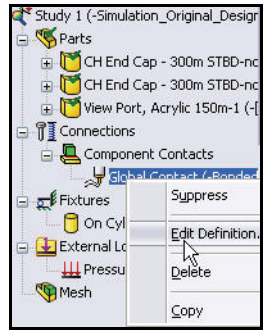


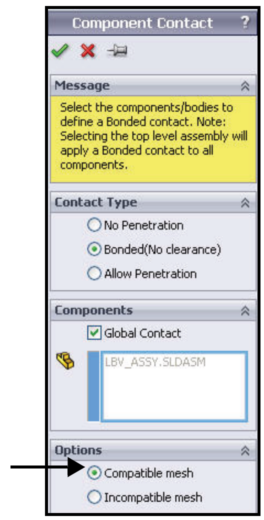
After Meshing

Creating a Compatible Mesh

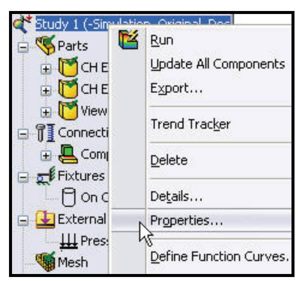
1 Create a Compatible Mesh

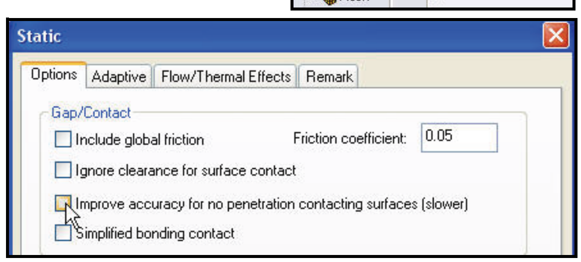
- Expand **Component Contact** from the Study tree.
- Right-click **Global Contact (-Bonded-)**.
- Click **Edit Definition**. The Component Contact PropertyManager is displayed.
- Click **Compatible mesh** from the Options box. Accept the default settings.
- Click **OK** from the Component Contact PropertyManager.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

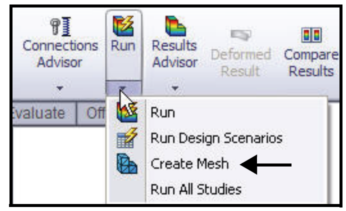




Creating a Mesh

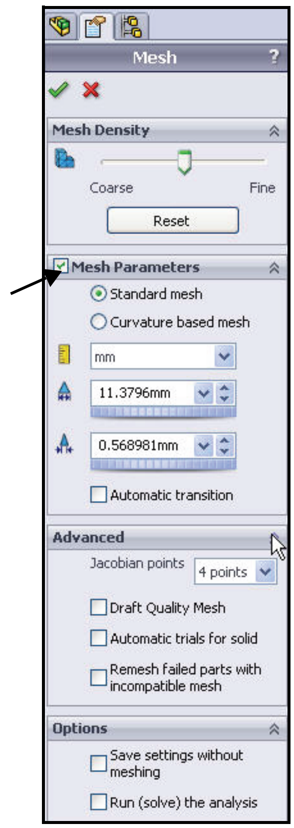
1 Create a Mesh.

- 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.



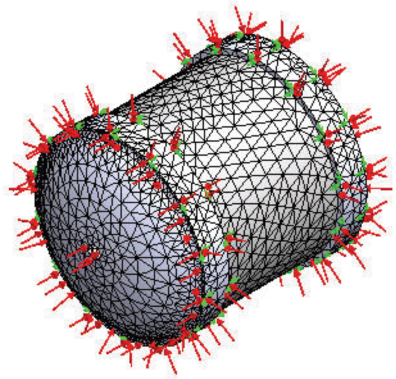
2 Review the Meshing Options.

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



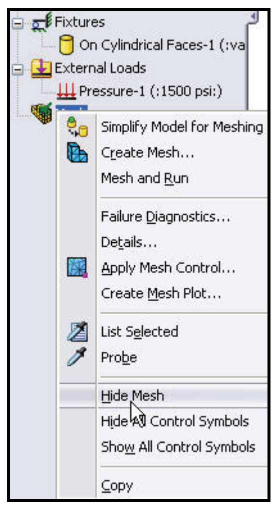
3 Start the Mesh Process.

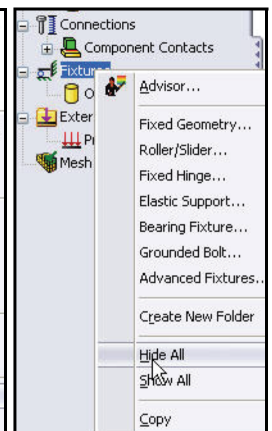
- Click **OK** from the Mesh PropertyManager. Meshing starts and the Mesh Progress window appears. After meshing is completed, SolidWorks Simulation displays the meshed model. A green check mark is applied next to the Mesh folder in the Study.



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

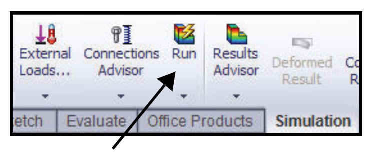
Note: Right-click **Fixtures**. Click **Hide All/Show All** to toggle the visibility of the loads and fixtures.

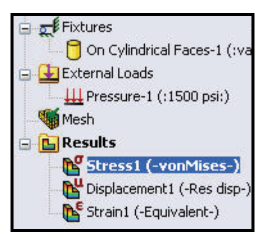




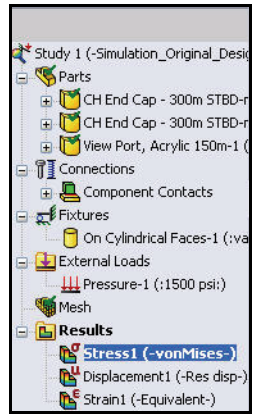
4 Run the Analysis.

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





Viewing the Results

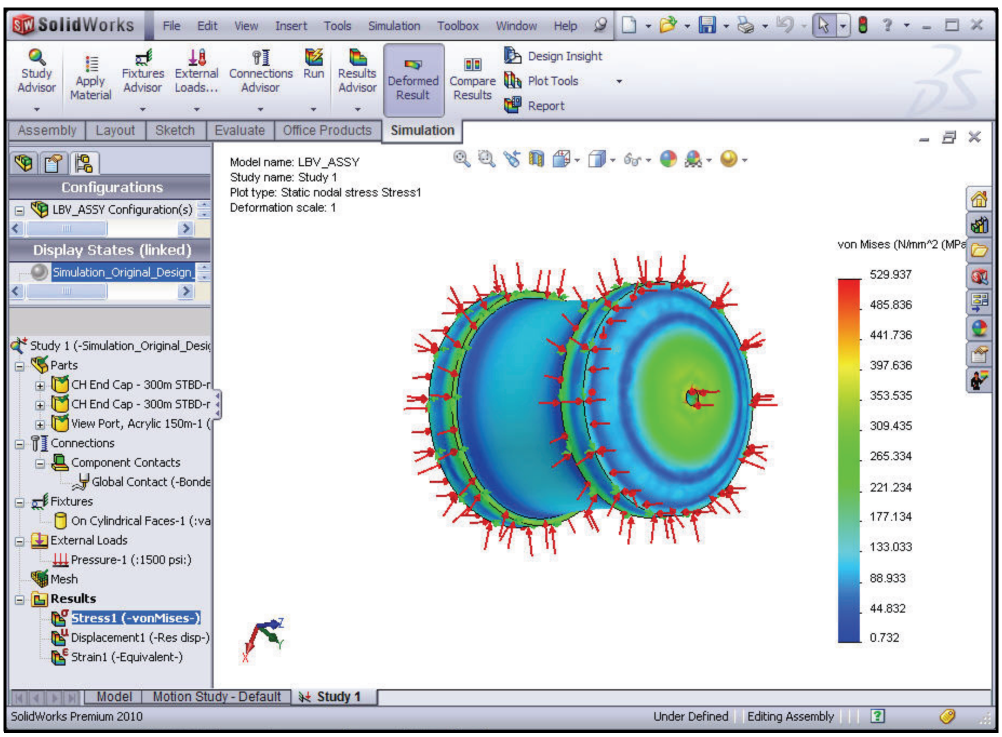


After a successful run of a Static analysis, SolidWorks Simulation creates three default plots: Stress, Displacement, and Strain.

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?

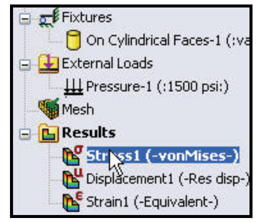
Note: Results may vary depending on the mesh speed.



View the Results

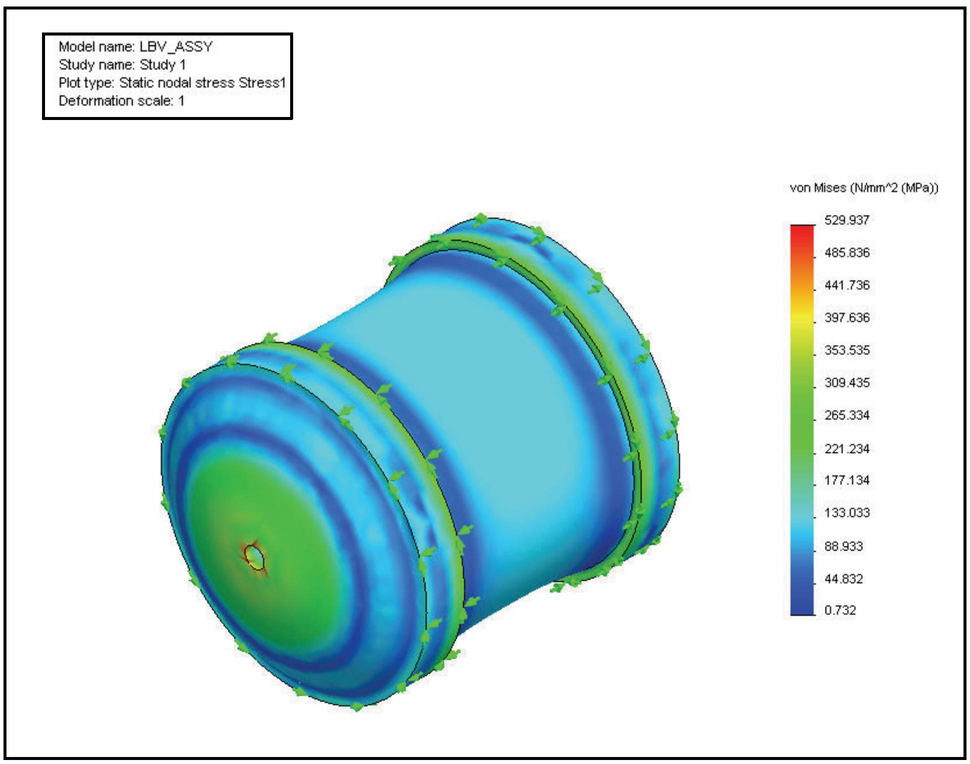
1 **Hide the External Loads.**
- Right-click the **External Loads** folder.
- Click **Hide All**.

2 **View the von Mises Stresses.**

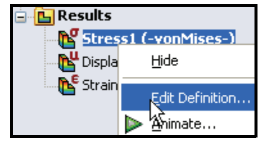


- Double-click **Stress1 (-von Mises-)**. The Stress Plot PropertyManager is displayed. Plot units if needed can be modified from the PropertyManager.
- Click **OK** from the Stress Plot PropertyManager.

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.

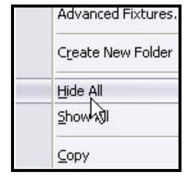


Note: To view the stress plot in a different unit system, right-click the active plot icon. Click **Edit Definition**. Set **units**. Click **OK** from the Stress Plot PropertyManager.



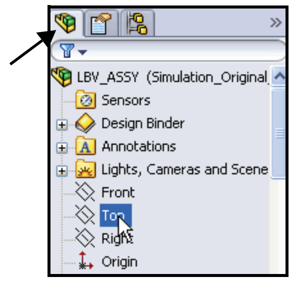
3 Hide the Fixtures.

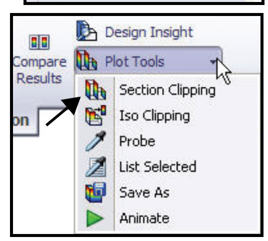
- Right-click the **Fixtures** folder.
- Click **Hide All**.



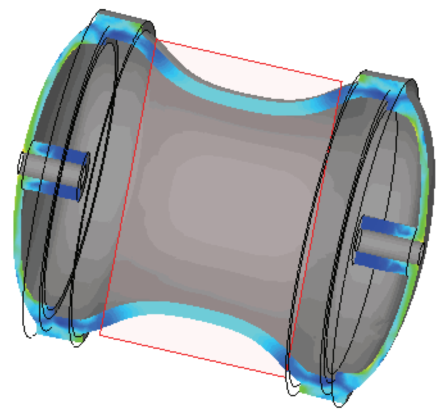
4 Display a Section View using the Top Plane.

- 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.

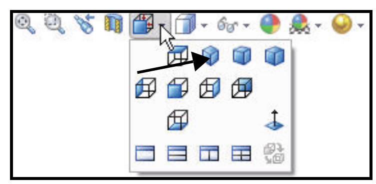


Note: Deformation is magnified for improved visibility. The deformation can be displayed at any scale.

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

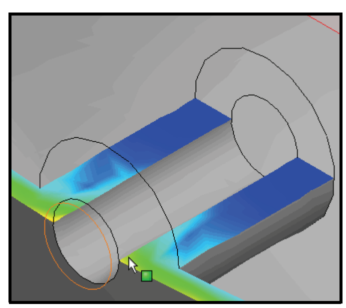
5 Display an Isometric view.

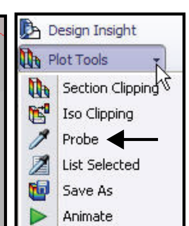
Click **Isometric** view from the Heads-up View toolbar.



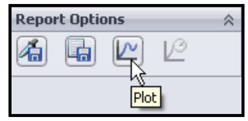
6 Probe the Model.

- 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.

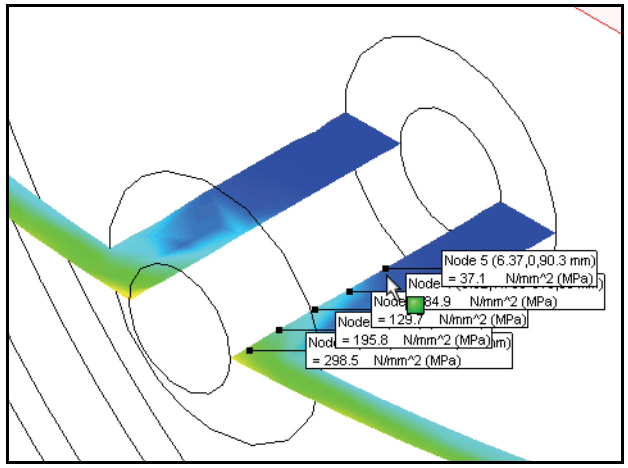




- Click **five points** from front to back as illustrated.
- Click the **Plot** button from the Report Options box. View the results.



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



7 Review the Plot.

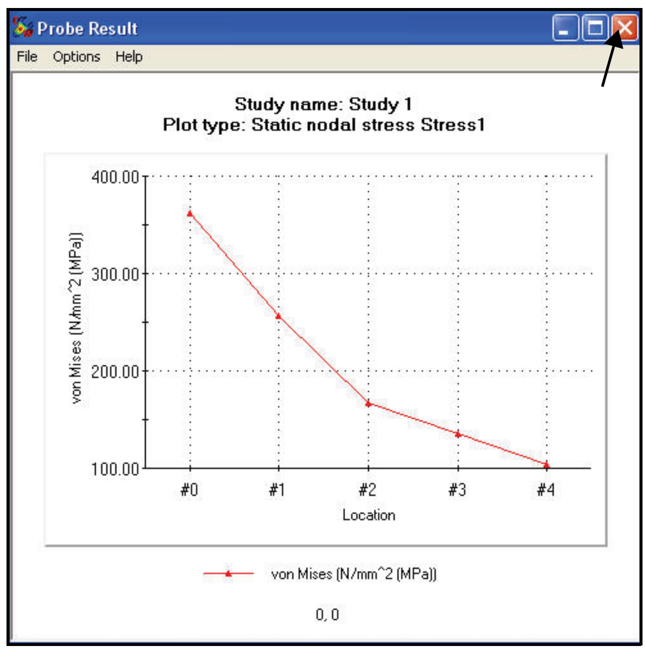
- Review the plot. This is an excellent way to examine the variation in stress across the geometry of your part.

8 Close the Probe Results dialog box.

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

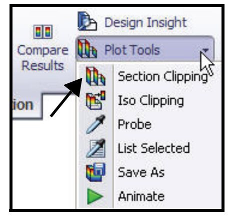
9 Close the Probe Result PropertyManager.

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

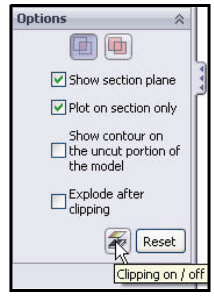


10 Deactivate the Section Plot.

- Click the **Plot Tools** drop-down arrow from the Simulation tab in the CommandManager.



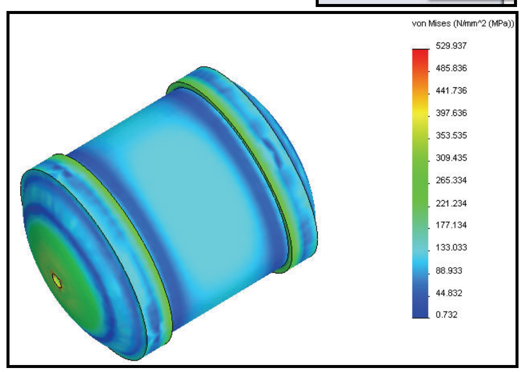
- Click the **Section Clipping** tool. The Section PropertyManager is displayed.
- Click the **Clipping on/off** button from the Options box as illustrated.



- Click **OK** from the Section PropertyManager.

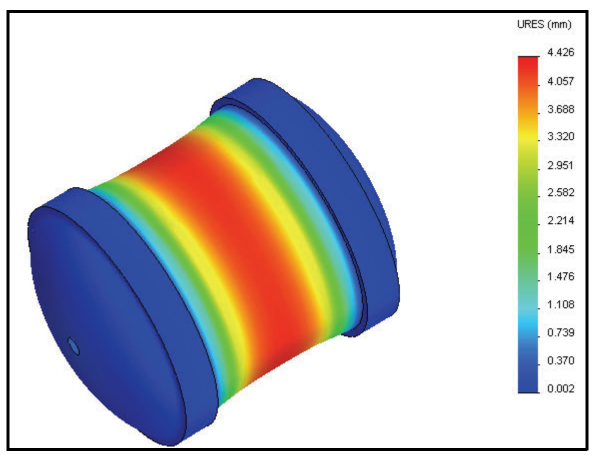
11 Fit the model to the Graphics area.

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

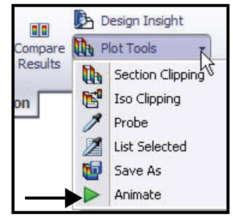


12 View the Displacement Plot.

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

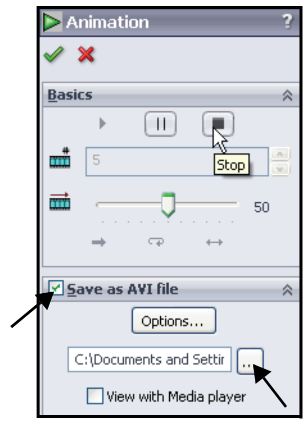


13 Animate the Displacement Plot.



- 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.



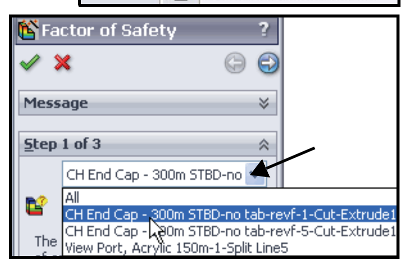
- Click **Stop** .

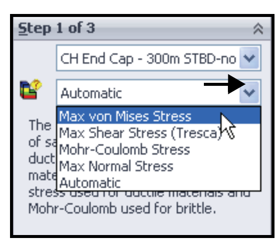
15 Save the Animation.

- 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.

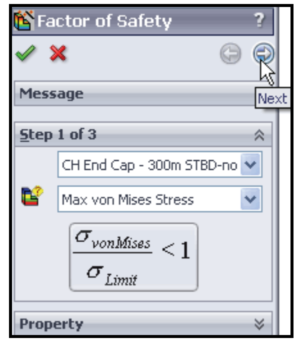
16 Calculate the Factor of Safety.

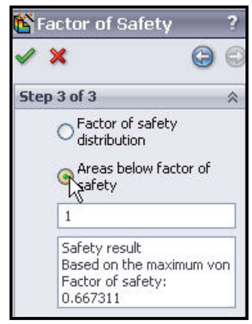
- Right-click the **Results** folder.
- Click the **Define Factor Of Safety Plot** tool. The Factor of Safety PropertyManager is displayed.
- Select the first **CH End Cap** component as illustrated from the drop-down menu.
- Select **Max von Mises Stress** from the drop-down menu as Criterion. Note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

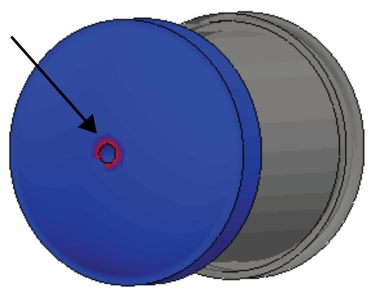




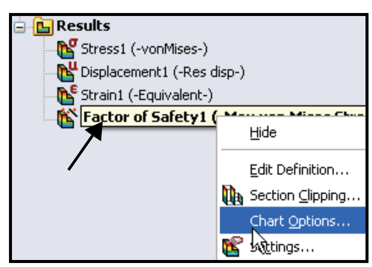
- Click **Next** to continue to step 2. Accept the defaults.
- Click **Next** to continue to step 3.
- Click the **Areas below factor of safety** box.
- Click **OK** from the Factor of Safety PropertyManager. View the model in the Graphics area.
- **Rotate** the model with the middle mouse button. The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



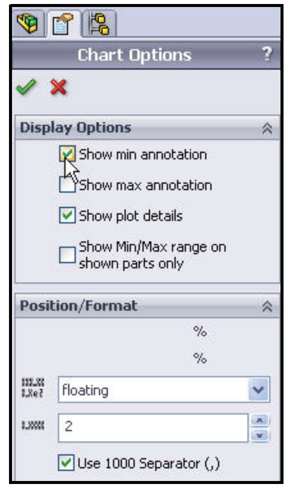




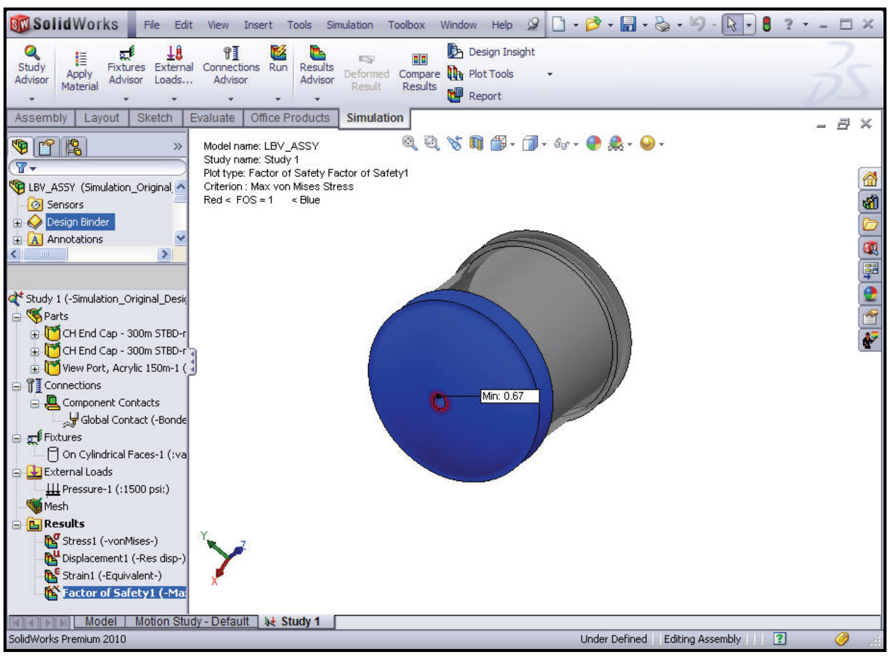
- Right-click **Factor of Safety1** as illustrated from the Results folder.
- Click **Chart Options**. The Chart Options PropertyManager is displayed.



- 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.

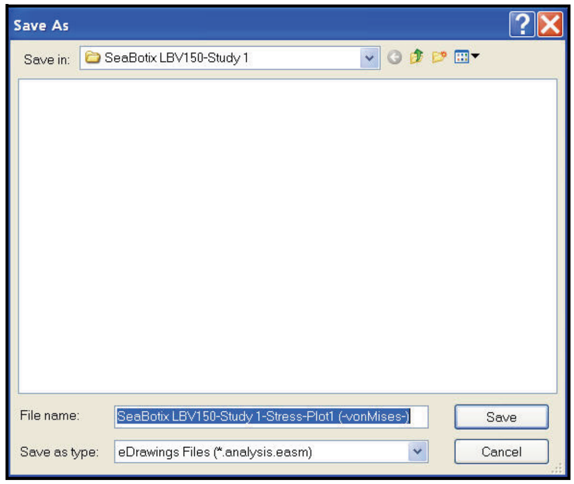


Note: The minimum FOS is 0.67. You did not meet the design goal, which is to obtain a FOS greater than one. In the next study, add structural ribs to the EndCap to obtain the design goal.

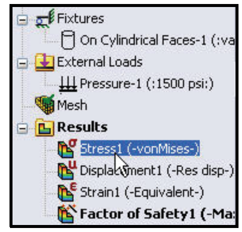


Creating a SolidWorks eDrawings File

You can save result plots in the SolidWorks eDrawings® format. The SolidWorks eDrawings application provides a facility for you to animate and view your analysis results. You can rotate and zoom SolidWorks eDrawings using the eDrawings viewer. The eDrawings files are self-viewing, small, and hence convenient to send via email.

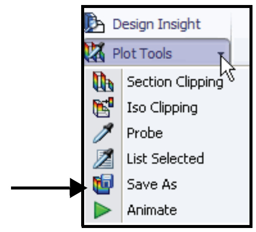


Creating a SolidWorks eDrawings file

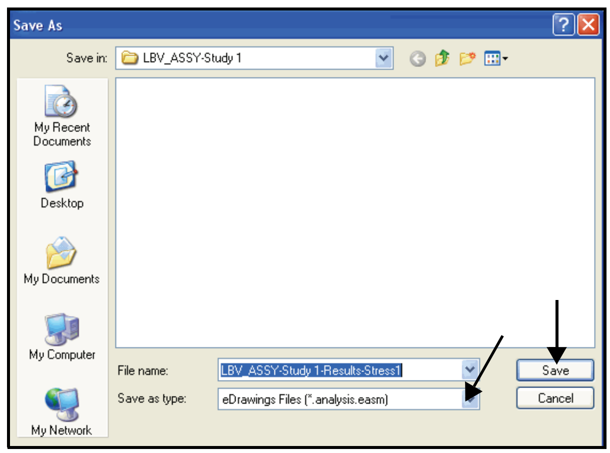


1 **Create a SolidWorks eDrawings file.**

- Double-click **Stress1 (-von Mises-)** from the Results folder.
- Click the **Plot Tools** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Save As** . The Save As box is displayed.
- Select **eDrawings Files** for Save as type. Accept the default name and location.

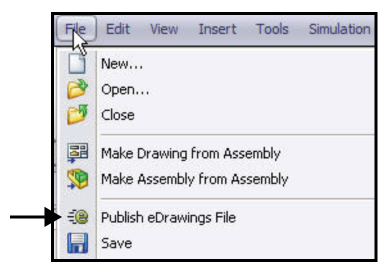


- Click **Save**.

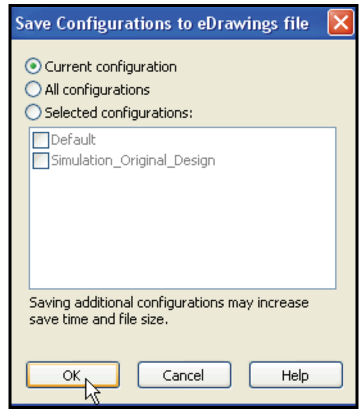


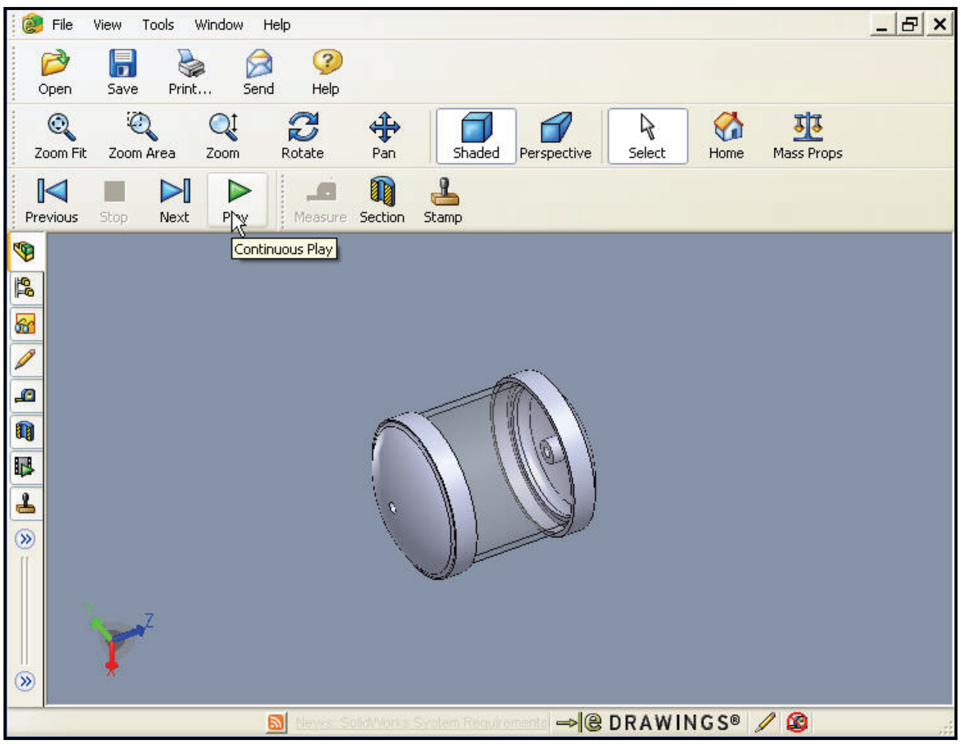
2 **Publish a SolidWorks eDrawing.**

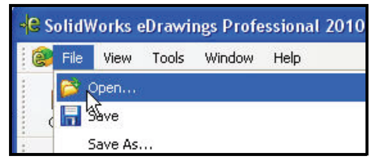
- Click **File**, **Publish eDrawings File** from the Menu bar menu. The Save Configurations to eDrawings file dialog box is displayed.

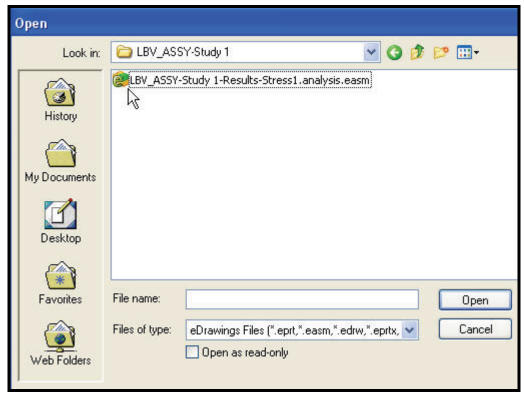


- Accept the default settings. Click **OK** from the dialog box. View the eDrawing.
- Click **Play** . View the eDrawing.
- Click **Stop** .



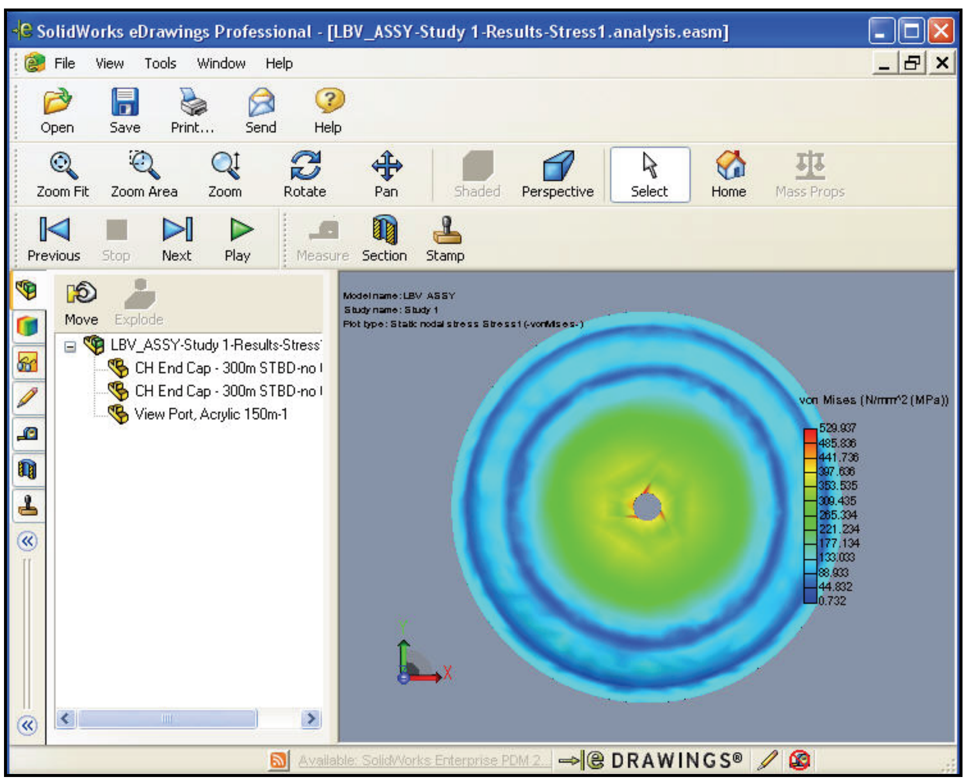




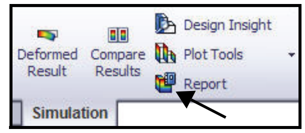


3 **View the Stress1 (-von Mises-) Plot**.

- Click **File, Open** from the Main menu in eDrawings.
- Double-click the **LBV-ASSY-Study 1** in the saved study folder. View the eDrawing for the von Mises Plot.
- Click **Play** . View the eDrawing.
- Click **Stop** .
- **Close** the eDrawing and return to SolidWorks Simulation.
- Click **No**. Do not save the eDrawing.

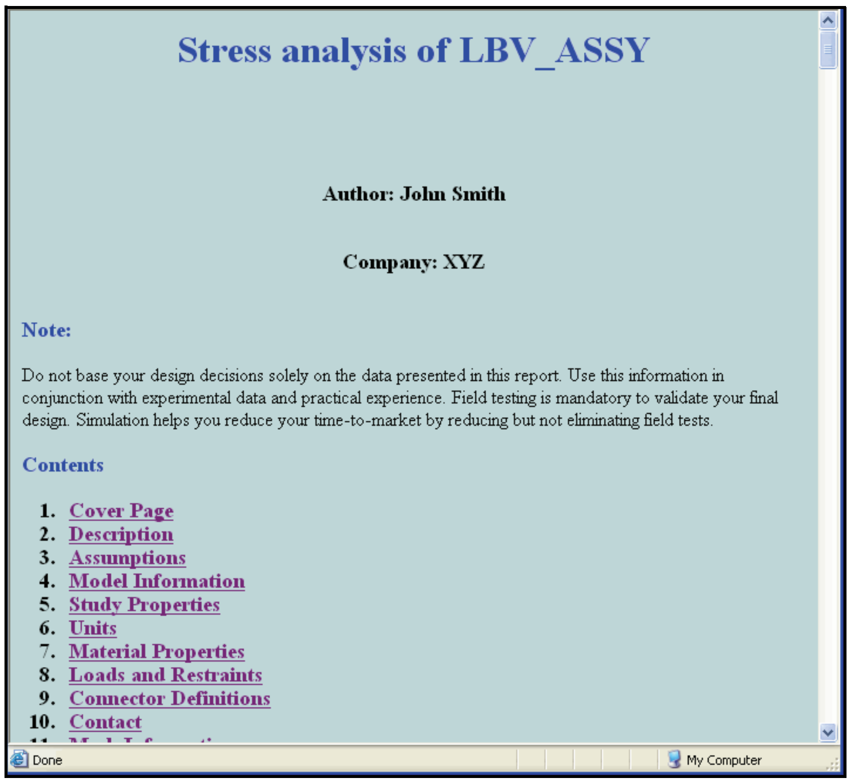


Generating a Report



The Report utility generates an Internet-ready or Microsoft® Word document convenient for review by colleagues and supervisors. The report describes all aspects of the analysis including material properties, applied restraints and loads, and the results.

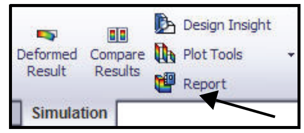
SolidWorks Simulation generates reports in HTML format and Microsoft Word format.

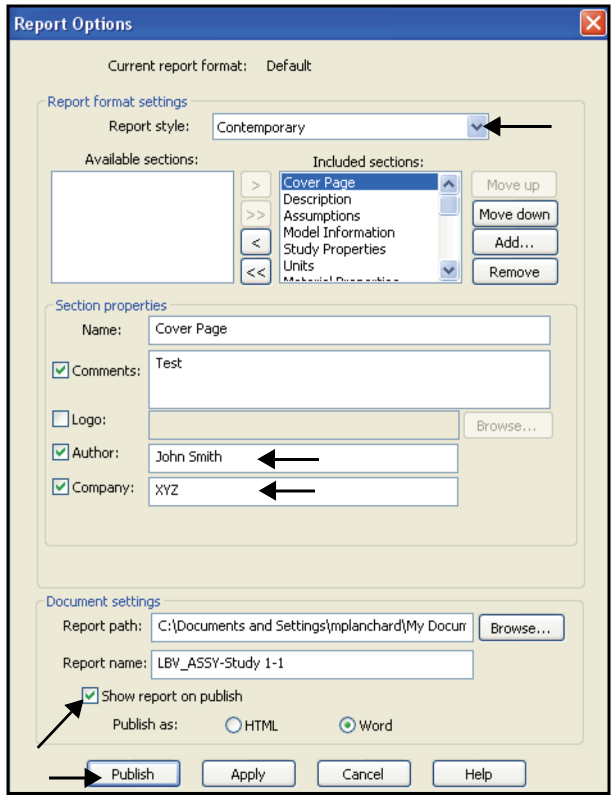


HTML Format

Generating a Static Study Report

1 **Generate a Static Study Report.**

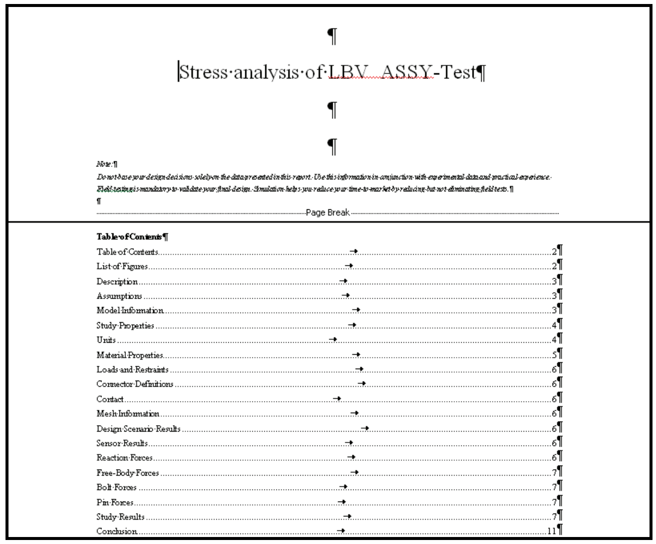




- Click **Report** from the Simulation tab in the CommandManager.
- Select **Contemporary** for Report Style.
- Check the **Author** box.
- Enter a value for **Author**.
- Check the **Company** box.
- Enter a value for **Company**.
- Scroll through the **included sections** list. View your options.
- Check the **Show report on publish** box. Accept the default settings.

2 View the Result.

- Click the **Publish** button. Microsoft Word opens and the report is displayed. Review the contents of the report. Note Result plots are included.



Stress analysis of LBV_ASSY-Test

Note:

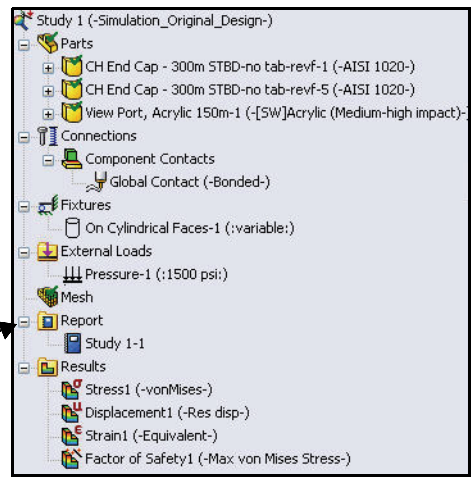
Do not base your design decisions solely on the data presented in this report. Use this information in conjunction with experimental data and practical experience. Field testing is mandatory to validate your final design. Simulation helps you reduce your time-to-market by reducing but not eliminating field tests.

Page Break

Table of Contents

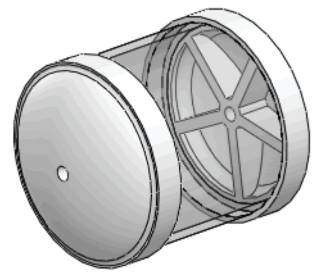
3 Close the Report.

- **Close** the report by exiting Microsoft Word and return to SolidWorks Simulation. The Report folder is displayed.



Note: Reports can be fully customized to your requirements.

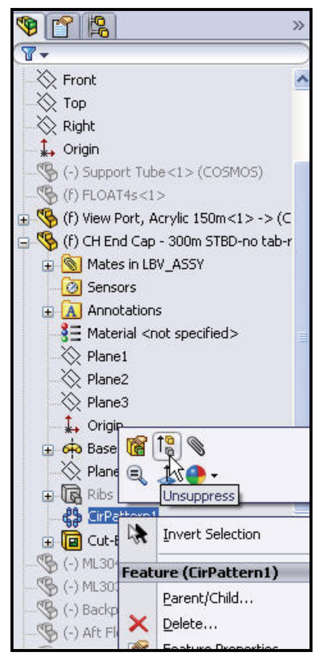
Analysis 2 - Static Study 2



In Study 1, the reports showed critical areas where the factor of safety was less than one.

As a designer, you must decide how you can increase the factor of safety.

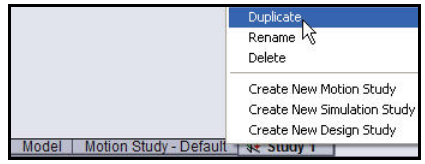
- Do you change the material?
- Do you modify the existing model?
- Should you re-evaluate the restraints and loads?



In this section you will:

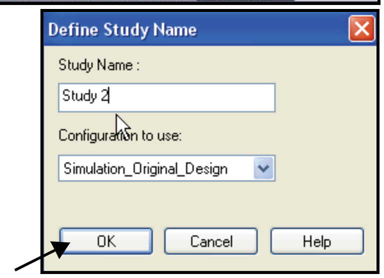
- Modify the EndCap in the Housing assembly. Add ribs to the EndCaps to increase the structural integrity of the Housing. (Due to limited time today, you will simply Unsuppress the ribs from the SolidWorks EndCap FeatureManager.)
- Copy information from Study 1 to Study 2.
- Mesh and Run the new analysis.
- View the results of Study 2.
- Compare Stress and FOS Plots between Study 2 and Study 1.

Creating Analysis 2 - Static Study 2

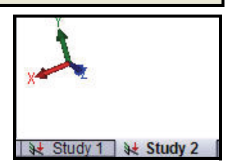


1 Create Study 2.

- Right-click the **Study 1** tab in the bottom section of the Graphic area as illustrated.
- Click **Duplicate**. The Define Study Name dialog box is displayed.
- Enter **Study 2** for Study Name.
- Click **OK** from the Define Study Name dialog box. Study 2 is displayed.

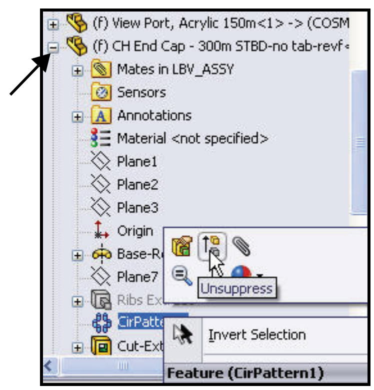


Note: Study 2 is a copy of Study 1.



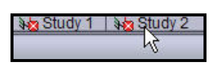
2 Modify the EndCap Part.

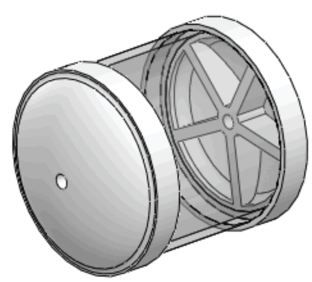
- Click the **Model** tab at the bottom of the Graphics area.
- Expand **CH EndCap - 300m STBD-no tab-revf**.
- Right-click **CirPattern1**.
- Click **Unsuppress**  from the Context toolbar. The Housing with the ribbed EndCaps is displayed in the Graphics area. Both instances of this part are updated.
- Rotate the **model** with the middle mouse button to view the unsuppressed ribs.



3 Return to Study 2.

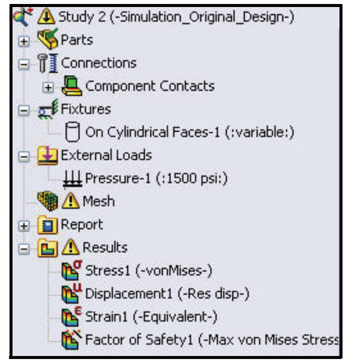
- Click the **Study 2** tab at the bottom of the Graphics area.





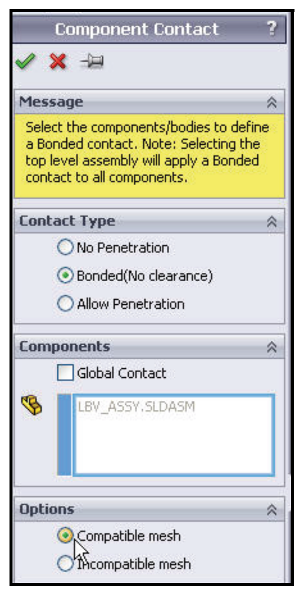
4 Review Study 2.

- Review Study 2. The material and Load/ Fixture information from Study 1 is copied to Study 2. Since the geometry changed, Mesh the model and Run the analysis again.



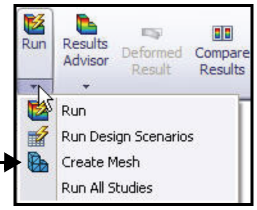
5 Create a Compatible Mesh

- Expand **Component Contact** from the Study 2 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.

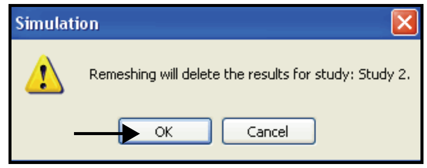


6 Mesh the Model.

- Click the **Run** drop-down arrow from the Simulation tab in the CommandManager.

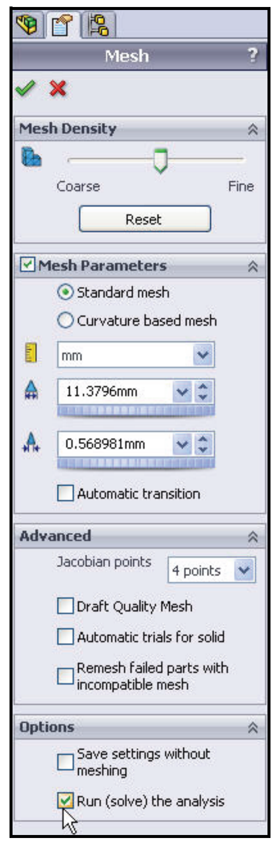


- Click **Create Mesh** .
- Click **OK** to the message, "Remeshing will delete the results for study: Study 2." The Mesh PropertyManager is displayed suggesting Global Size and Tolerance value.



7 Start the Mesh Process.

- Check the **Mesh Parameters** box. View your options.
- Check the **Run (solve) the analysis** box.
- Click **OK** from the Mesh PropertyManager. Meshing starts and the Mesh Progress window appears. View the results in the Graphics area.

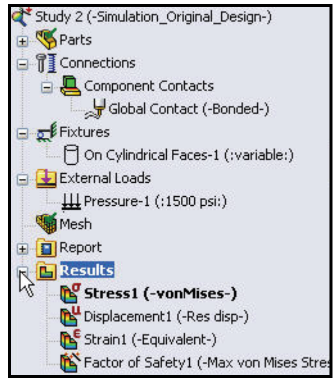


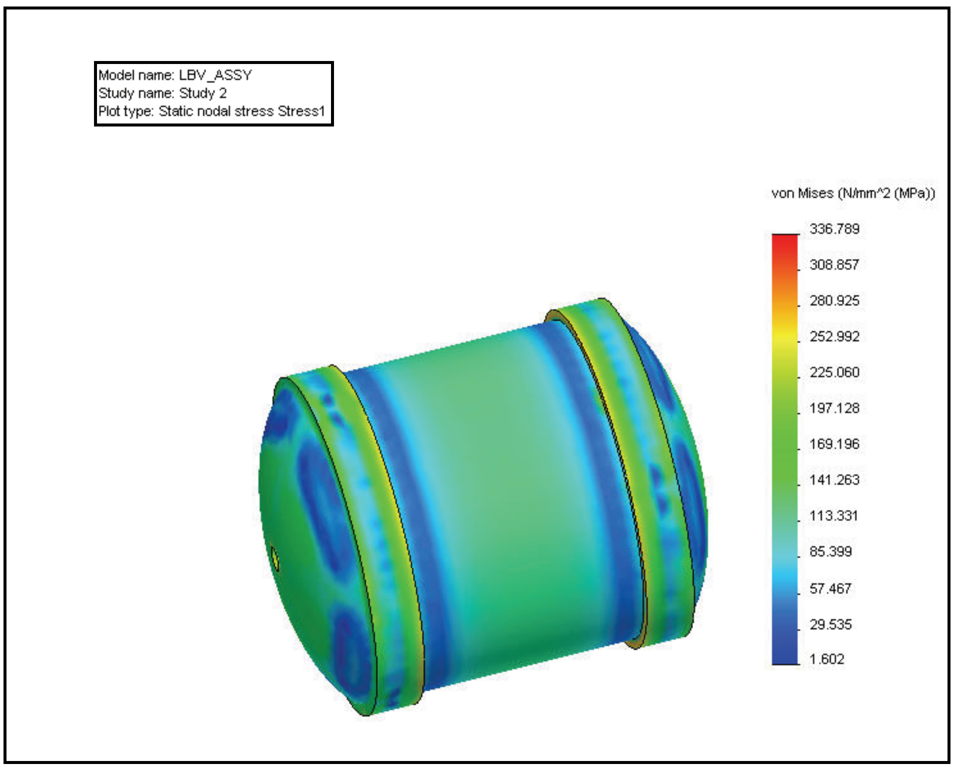
8 View the Results Folder.

- Expand the **Results** folder.

9 View the von Mises Stresses Plot.

- Double-click **Stress1 (-von Mises-)**. The von Mises stress plot is displayed. View your options.
- Click **OK**  from the Stress Plot PropertyManager.

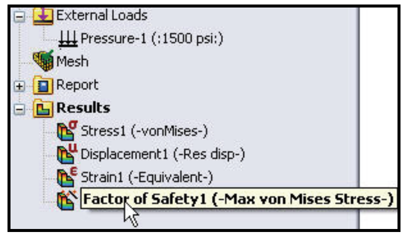


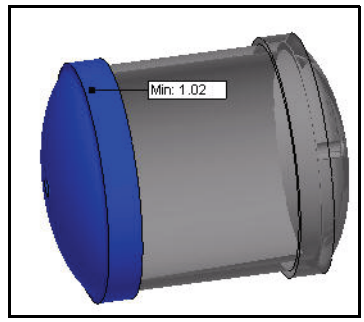


10 View the Factor of Safety.

- Double-click **Factor of Safety1 (-Max von Mises Stress-)**.
- **Rotate** the model to view the blue surface. The blue area displays a FOS above 1.

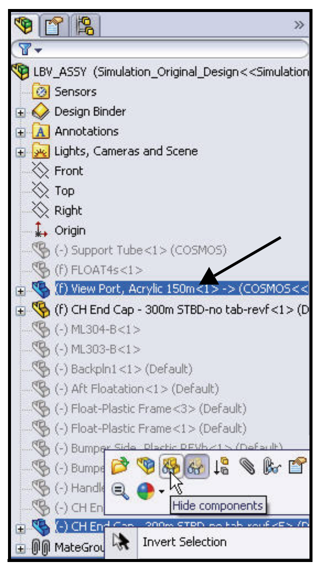
Note: The minimum FOS is now 1.02.

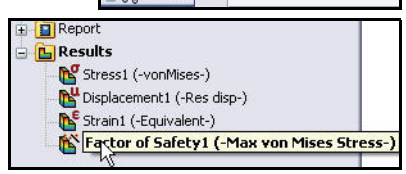




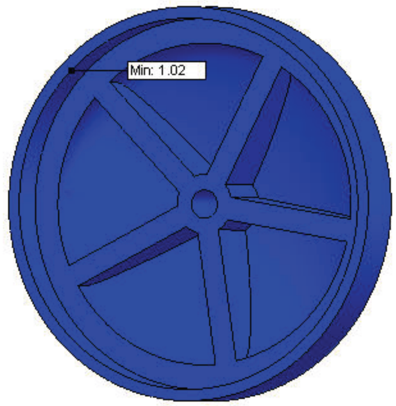
11 Compare Study 2 to Study 1.

- Click **Isometric** view from the Heads-up View toolbar.
- Click **(f) View Port** in the FeatureManager.
- Hold the **Ctrl** key down.
- Click the second **CH End Cap - 300mm** component. Both components are selected.
- Release the **Ctrl** key.
- Right-click **Hide components** from the Context toolbar.
- Double-click **Factor of Safety1 (-Max von Mises Stress-)**.
- Click **OK** from the PropertyManager. The two components are hidden in the Graphics area. View the single CH End Cap.

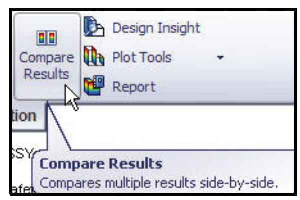




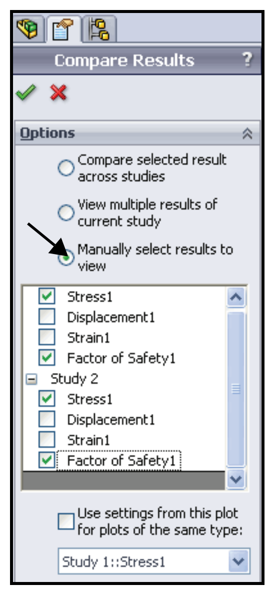
- **Rotate** the model and view the results.

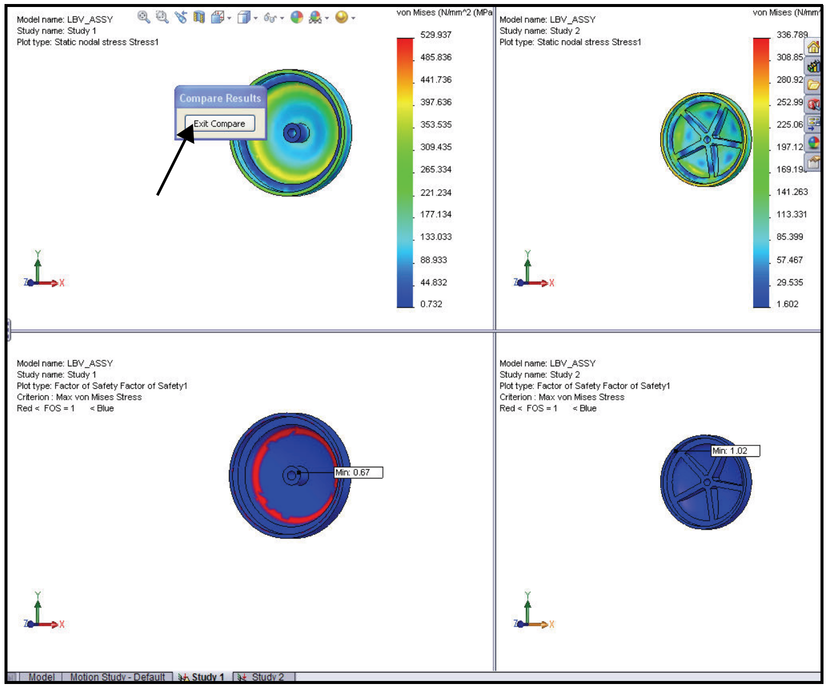


- Click **Compare Results** from the Simulation tab in the CommandManager. The Compare Results PropertyManager is displayed. Both Study 1 and Study 2 are checked.

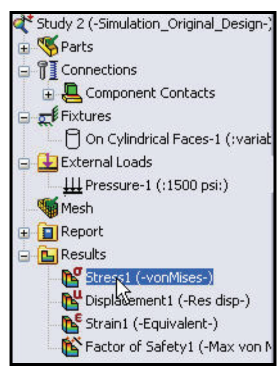


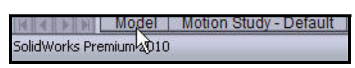
- Click the **Manually select results to view** box.
- Un-check the **Displacement1** and **Strain1** box under Study 1.
- Check the **Stress1** and **Factor of Safety1** box under Study 1.
- Check the **Stress1** and **Factor of Safety1** box under Study 2.
- Click **OK** from the Compare Results PropertyManager. View the Graphics area. The two Studies are displayed.



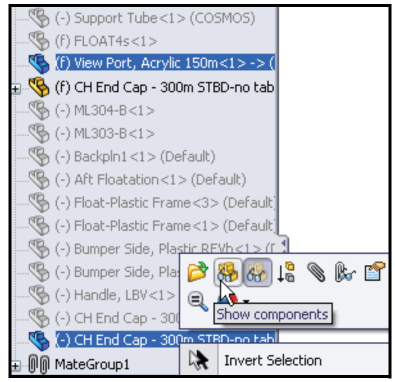


- Click the **Exit Compare** button in the Compare Results dialog box. Study 2 is displayed in the Graphics area.
- Double-click **Stress1 (-vonMises-)** from the Results folder. View the Graphics area.
- Click the **Model** tab at the bottom of the Graphics area to return to SolidWorks and to displayed the Assembly FeatureManager.



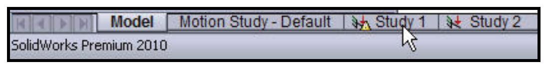


- Click **(f) View Port** in the FeatureManager.
- Hold the **Ctrl** key down.
- Click the second **CH End Cap - 300mm** component. Both components are selected.
- Release the **Ctrl** key.
- Right-click **Show components** from the Context toolbar. The components are displayed in the Graphics area.

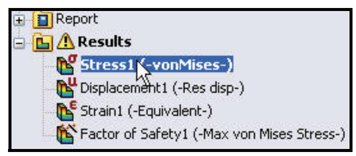


12 Return to Study 1.

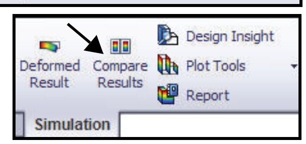
- Click the **Study 1** tab at the bottom of the Graphics area. Study 1 is displayed.



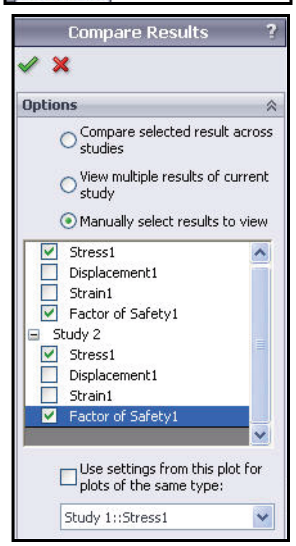
- Double-click **Stress1 (-vonMises-)** from the Results folder. View the Graphics area.



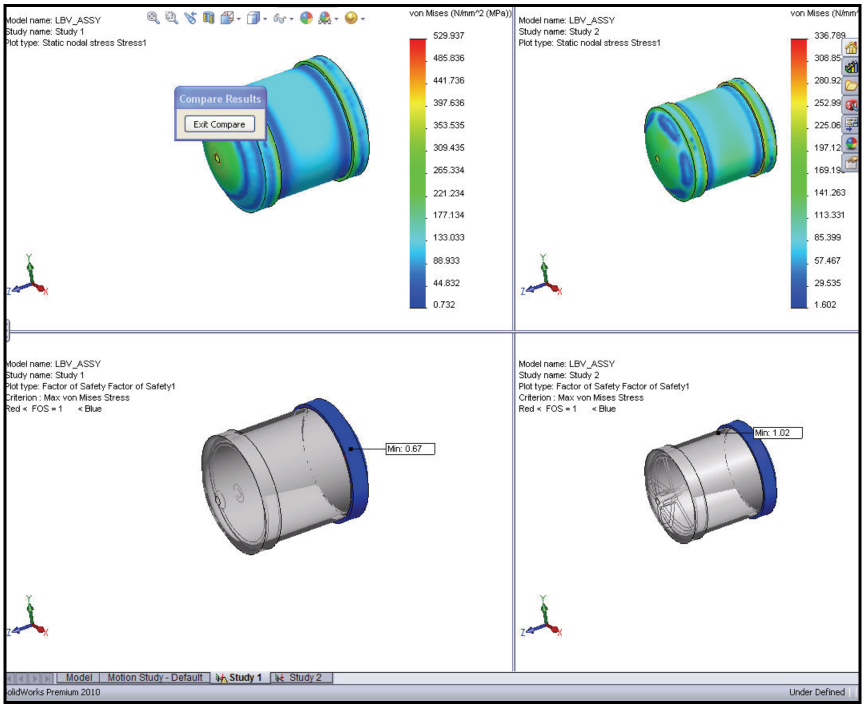
- Click **OK** from the PropertyManager.
- Click **Compare Results** from the Simulation tab in the CommandManager. The Compare Results PropertyManager is displayed.



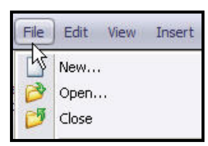
- Click the **Manually select results to view** box.
- Un-check the **Displacement1** and **Strain1** box under Study 1.
- Check the **Stress1** box and **Factor of Safety1** box under Study 2.
- Click **OK** from the Compare Results PropertyManager. View the Graphics area. The two Studies are displayed.



- Click the **Exit Compare** button in the Compare Results dialog box. Study 1 is displayed in the Graphics area.



13 Save and Close the Model.



- Click **Save** .
- Click **File**, **Close** from the Menu bar menu.

Note: Your design goal is complete. The structural ribs in the EndCap provided an FOS greater than one.

SolidWorks Simulation Conclusion

During this short session on using SolidWorks Simulation, you have had a brief exposure to the main concepts of static analysis. Integrated within SolidWorks 3D mechanical design software, SolidWorks Simulation allows you to update all of your design changes automatically and to become immediately productive using familiar SolidWorks functions and commands.

Compare alternative designs easily and quickly. SolidWorks Simulation lets you study different design configurations created with SolidWorks software and choose the optimal design for final production.

Study the interaction between different assembly components. SolidWorks Simulation provides powerful tools for you to study and optimize assemblies.

Simulate real-world operating conditions. SolidWorks Simulation includes several types of loads and restraints as well as part-to-part contact to represent real-life situations. All loads and restraints are associative with the geometry and automatically update with changes in your design.

Automate analysis tasks. SolidWorks Simulation utilizes a number of automation tools to simplify the analysis process and help you to work more efficiently.

Interpret analysis results with powerful and intuitive visualization tools. Once you have completed your analysis, SolidWorks Simulation offers a variety of results visualization tools that allow you to gain valuable insight into the performance of your models.

Collaborate and share analysis results. SolidWorks Simulation makes it easy for you to collaborate and share analysis results effectively with everyone involved in the product development process.

SolidWorks Simulation Professional

When you complete this chapter, you will have experienced the power and capabilities of SolidWorks® Simulation Professional, including:

- The benefits of Thermal analysis, Drop Test, Optimization, and Fatigue analysis.
- The ease of use of SolidWorks® Simulation Professional to explore design iterations using Trend Tracker.
- The steps for performing upfront analysis on your designs.
- The integration between SolidWorks® Simulation Professional and SolidWorks.
- The results of cost savings by avoiding field failures and eliminating the prototype bottleneck.
- The ability to document your analysis findings automatically.
- The method to update your assembly based on the analysis results.

Time: 35 - 40 minutes

SolidWorks Simulation Professional

In the first part of your analysis, you utilized SolidWorks Simulation to perform two static analyses on the Housing. Next, you will use applications available in SolidWorks Simulation Professional to continue your investigation. SolidWorks Simulation Professional combines all of the features of SolidWorks Simulation plus additional software analysis applications. SolidWorks Simulation Professional includes:

- Static analysis of parts and assemblies
- Drop Test simulation
- Frequency and Buckling analysis
- Fatigue analysis
- Optimization performance
- Pressure vessel analysis
- Thermal analysis
- Trend Tracker to document design iterations

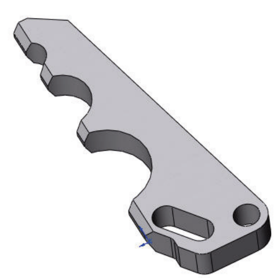
In this second part of your analysis, you will perform the following studies:

- Thermal analysis to determine the heat dissipation from the EndCap surrounded by seawater.
- Drop Test simulation of the Housing from a height of four feet.
- Optimization to find the best combination of EndCap thickness and Rib thickness to minimize the mass.
- Fatigue analysis on the 3 Finger Jaw.

EndCap

EndCap with Ribs

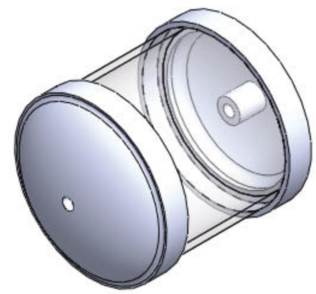
Housing



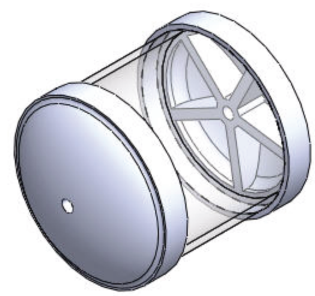
3 Finger Jaw

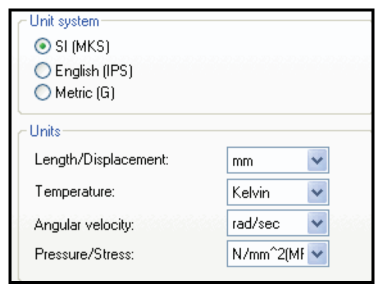
Trend Tracker Analysis

When you complete this chapter, you will have experienced the power and capabilities of the trend analysis feature inside SolidWorks Simulation Professional.

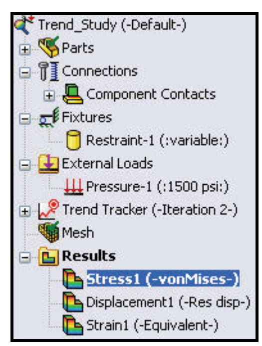


- Trend analysis allows you to track the changes that were made to your designs in a systematic way.
- It helps you to compare the various design changes and understand why and how your changes were better or worse than your previous designs.
- It provides complete and automated documentation of the analysis changes throughout your design cycle.





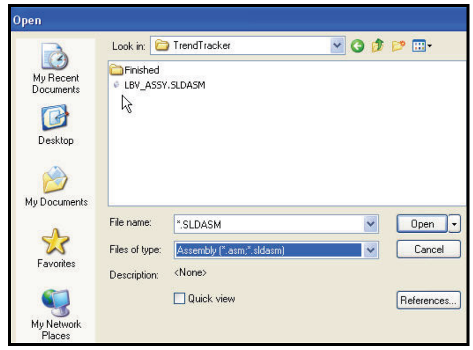
You will start by performing a trend analysis on the housing components of the SeaBotix LBV150 assembly. This is the same assembly that you analyzed before using the static analysis feature inside SolidWorks Simulation.



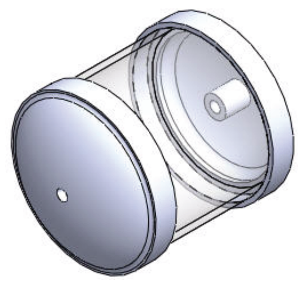
Time: 15 - 20 minutes

1 Open the Housing_Assy Assembly.

- Click **Open** from the Menu bar toolbar.
- Double-click the **LBV_Assy** from the SeaBotix\SolidWorks Simulation Professional\TrendTracker folder. The LBV_Assy is displayed.

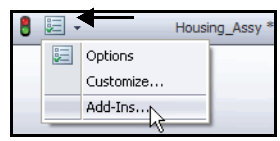


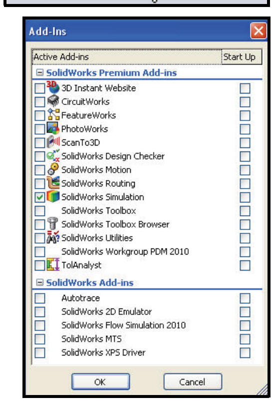
Note: View the Trend_Study tab in the bottom section of the Graphics area if SolidWorks Simulation is active.



2 If required, activate SolidWorks Simulation.

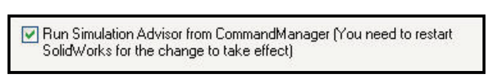
- Click the **Options** drop-down arrow from the Menu bar toolbar.
- Click **Add-Ins**. The Add-Ins dialog box is displayed.
- Check the **SolidWorks Simulation** box.
- Click **OK** from the Add-Ins box.





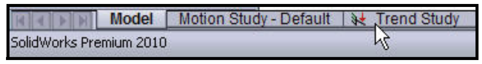
Note: You don't have to activate SolidWorks Simulation if your SolidWorks Simulation is already added in.

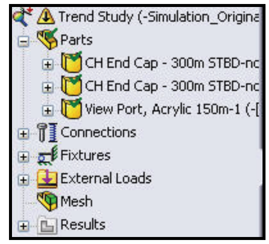
Note: To display the Simulation Advisor CommandManager, check the Run Simulation Advisor box under Simulation System Options.



3 VIew the Trend Study.

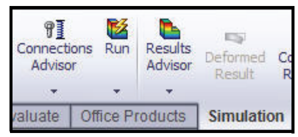
- Click the **Trend_Study** tab as illustrated. The Trend_Study is displayed.





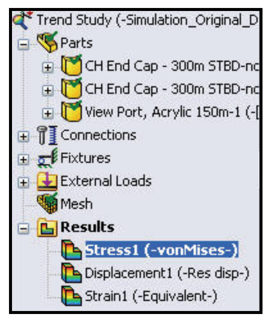
4 Perform an Analysis on the Study.

- Click **Run**  from the Simulation tab in the CommandManager. The analysis runs and three default plots are created.



5 View the Von Mises Stress on the EndCap.

- The plot is displayed in the Graphics area. Double-click **Stress1 (-vonMises-)**. The Stress Plot PropertyManager is displayed. View your available options.
- Click **OK**  from the Stress Plot PropertyManager.



6 Fit the model to the Graphics area.

- Press the **f** key.

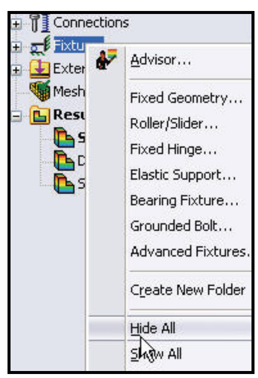
Tip: To Zoom out, press the **z** key.

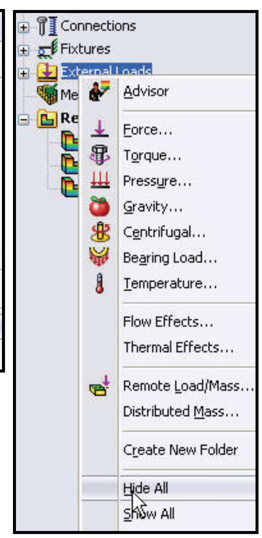
7 Hide the Fixtures in the Graphics area.

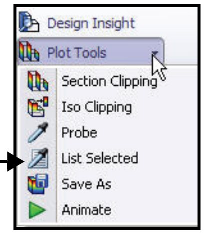
- Right-click the **Fixtures** folder.
- Click **Hide All**.

8 Hide the External Loads.

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

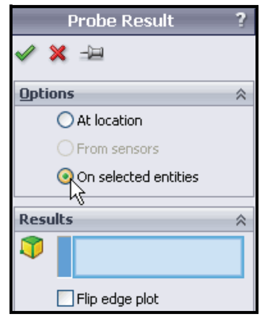




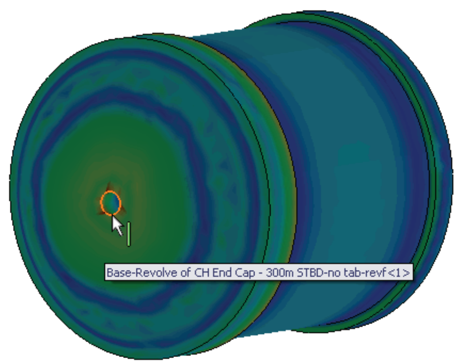


- Click the **Plot Tools** drop-down arrow from the Simulation tab in the CommandManager.
- Click **List Selected** . The Probe Results PropertyManager is displayed.

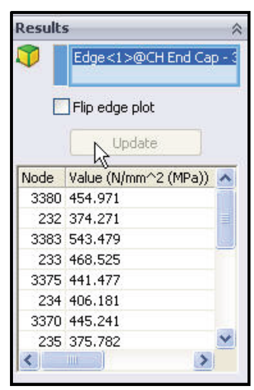
Note: The On selected entities box is selected by default.



- Zoom in on the **front hole** of the EndCap as illustrated.

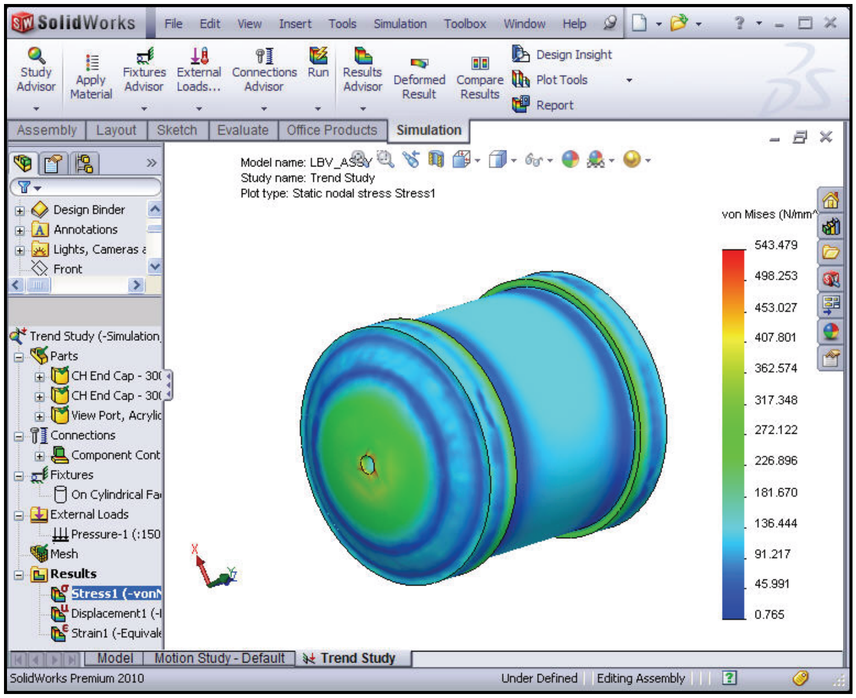


- Click the **edge of the front hole** of the EndCap. Note: The icon feedback symbol of an edge. Edge<1> is displayed in the Results box.
- Click the **Update** button. View the results.
- Click **OK**  from the Probe Result PropertyManager.

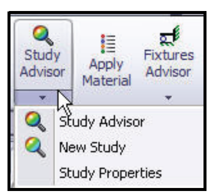


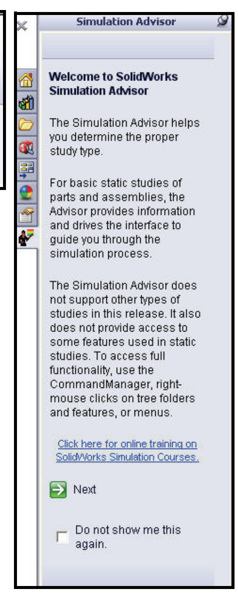
9 Fit the model to the Graphics area.

- Press the **f** key.



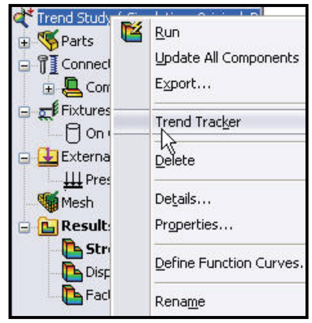
Note: Study Advisor recommends study types and outputs to expect. Study Advisor helps the user to define sensors and creates studies automatically.





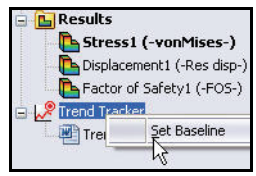
10 Invoke Trend Tracker.

- Right-click **Trend Study (-Simulation_Origin_Design)**.
- Click **Trend Tracker**. The Tend Tracker folder is displayed.

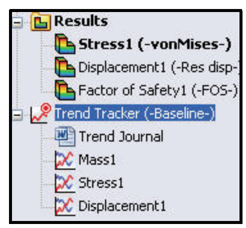


11 Set a Baseline.

- Right-click the **Trend Tracker** folder.
- Click **Set Baseline**. View the created graph icons.



Note: The current stress analysis will be the baseline to which future designs are compared to.



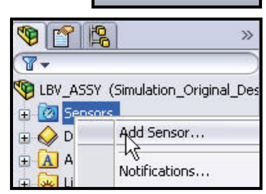
Perform design changes to strengthen the End caps. View how the new designs changes compare with the initial (Baseline) design in terms of: stress, displacement, etc. using the Trend Tracker tool.

See how Trend Tracker allows you to perform design changes without creating multiple studies or configurations.

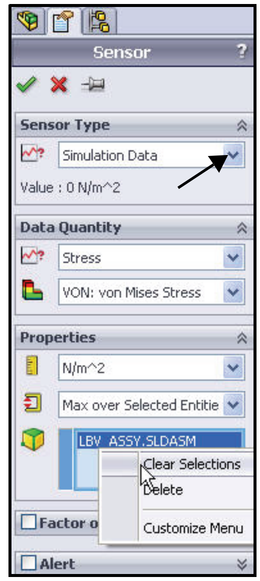
In the next section, define a sensor. You define sensors to monitor result quantities at a set of locations, mass properties of components or bodies, interferences between components for assemblies, and dimensions.

12 Add Sensors.

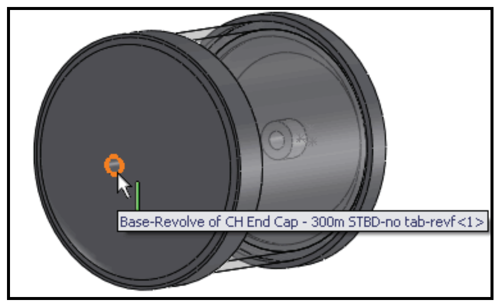
- Click the **Model** tab at the bottom of the Graphics area.
- Right-click the **Sensors** folder from the Assembly FeatureManager.
- Click **Add Sensor.** The Sensor PropertyManager is displayed.



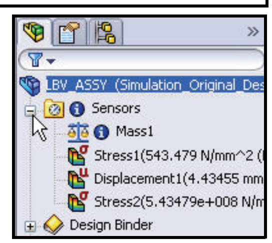
- Select **Simulation Data** for Sensor type from the drop-down menu.
- Select **N/m^2** for Units.
- Select **Max over Selected Entities** for Criterion.
- Right-click **Clear Selections** in the selection box as illustrated.



- Click the **edge of the front hole** of the EndCap as illustrated. Note: The icon feedback symbol of an edge. Edge<1> is displayed in the selection box.

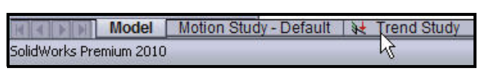


- Click **OK** from the Sensor PropertyManager.
- Expand the **Sensor** folder in the Assembly FeatureManager. View the folders.



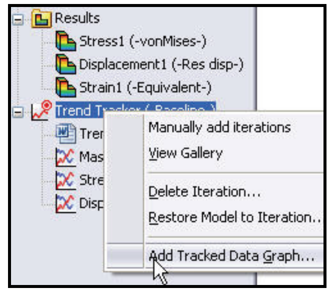
13 Return to Trend Study.

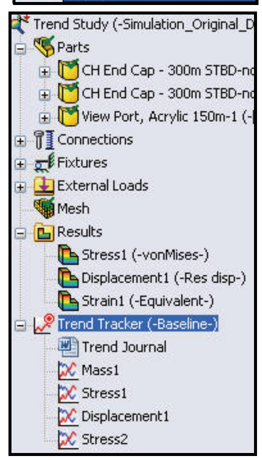
- Click the **Trend Study** tab at the bottom of the Graphics area.



14 Add a second Tracked Data Graph.

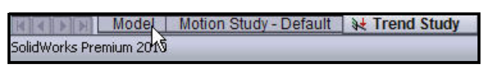
- Right-click the **Trend Tracker (Baseline)** folder.
- Click **Add Tracked Data Graph**. The Tracked Data Graph PropertyManager is displayed.
- Select **Stress2** for Sensor Type from the drop-down menu as illustrated. View your options.
- Click **OK** from the Tracked Data Graph PropertyManager. The Stress2 folder is displayed.



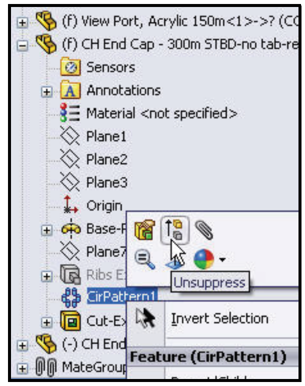


15 Perform a Design Change. Modify the EndCap Part.

- Click the **Model** tab at the bottom of the Graphics area. The Assembly FeatureManager is displayed.

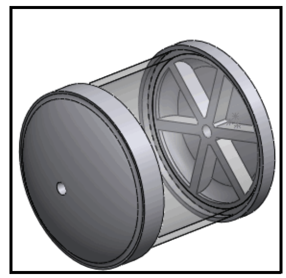


- Expand the first **CH End Cap - 300m STBD** from the FeatureManager as illustrated.
- Right-click **CirPattern1**.
- Click **Unsuppress** from the Context toolbar. The Housing with the ribbed End caps is displayed in the Graphics area.



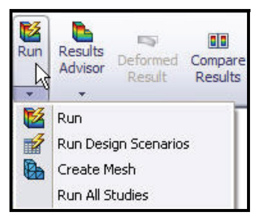
16 Return to the Trend Study.

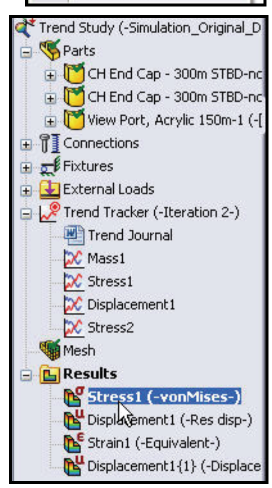
- Click the **Trend Study** tab as the bottom of the Graphics area.



17 Run an Analysis.

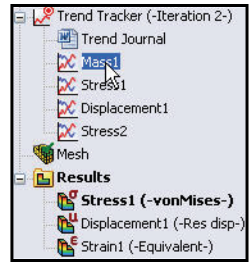
- Click **Run** from the Simulation tab in the CommandManager. Once the analysis is completed, the plots under the Trend Tracker folder are updated.
- **View** the Stress1 (-vonMises-) plot.





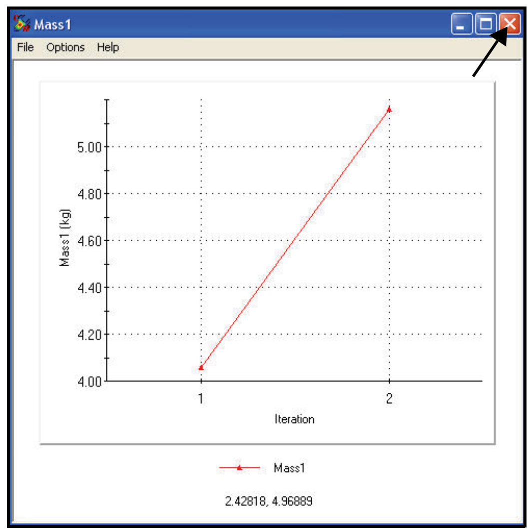
18 Examine the total mass of the EndCap Part.

- Double-click the **Mass1** folded as illustrated. The total mass increase from the first iteration to the second iteration due to the addition of the ribs.



Note: The additional weight is expected to increase the FOS.

- **Close** the graph.

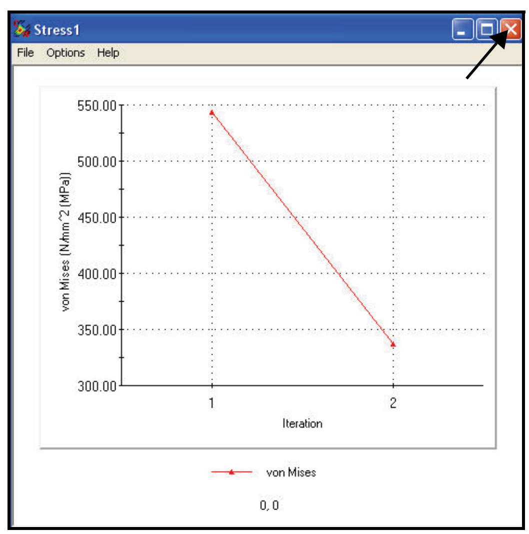


19 Examine the Stress1 graph.

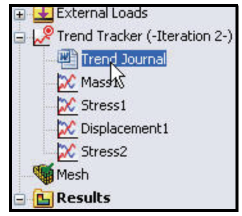
- Double-click the **Stress1** folder. View the results.

Note: The maximum von Mises stress in the hole has decreased due to the addition of the ribs.

- **Close** the graph.



20 Review the Trend Journal.



- Double-click the **Trend Journal** folder. The Trend Journal is displayed. The journal contains all details about the different iterations that were performed on the model.
- **Close** the Trend Journal by closing Microsoft Word.

Using Trend Tracker, you can also roll back your model to an intermediate iteration without having to save any conceptual changes. Trend Tracker is also integrated with Design Scenarios in SolidWorks Simulation Professional to track structural feature changes.

21 Save and Close the Model.

- Click **Save** .
- Click **File**, **Close** from the Menu bar menu.

Trend Journal

File Name:	LBV_ASSY.SLDASM
Study name:	Trend Study
Description:	

Baseline	
Time Completed:	Friday, October 02, 2009 7:40:12 AM
Tracked Data:	

Source	Type	Actual Value	Normalized Value
Mass1	Model Max	4.05904 (kg)	100
Stress1 (VON: von Mises Stress)	Model Max	543.479 (N/mm^2 (MPa))	100
Displacement1 (URES: Resultant Displacement)	Model Max	4.43455 (mm)	100
Stress2 (VON: von Mises Stress)	Max over Selected Entities	5.43479e+008 (N/m^2)	100

Iteration 2	
Time Completed:	Friday, October 02, 2009 7:51:41 AM
Tracked Data:	

Source	Type	Actual Value	Normalized Value
Mass1	Model Max	5.16175 (kg)	127
Stress1 (VON: von Mises Stress)	Model Max	337.151 (N/mm^2 (MPa))	62
Displacement1 (URES: Resultant Displacement)	Model Max	4.42488 (mm)	99
Stress2 (VON: von Mises Stress)	Max over Selected Entities	2.5485e+008 (N/m^2)	46

Thermal Analysis

Design performance can be compromised due to excessive temperatures or heat transfer between components. SolidWorks Simulation Professional allows you to perform thermal analyses with the following parameters:

- Conduction, convection, and radiation
- Steady state and transient with time-dependent loads
- Temperature-dependent materials and loads
- Temperature, heat flux, and heat power
- Thermostats for closed-loop feedback in transient studies
- Thermal contact resistance

You will again perform an analysis on the EndCap of the Housing. The Housing contains the camera and lighting system of the SeaBotix LBV150 assembly. The EndCap analysis will determine the amount of heat lost to the surrounding seawater. You will only address natural convection today. To simplify the model, the camera and lighting system are represented as a concentrated heat source.

Your design goal is to improve the thermal distribution of the EndCap. You will learn if the addition of Ribs, "mass," will help to dissipate the generated heat from the camera and lighting system to the surrounding seawater.

Without Ribs

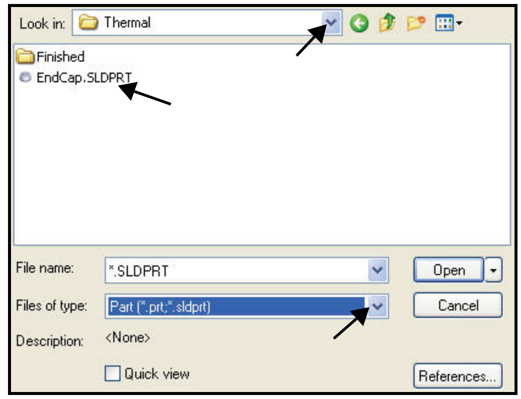
With Ribs

Time: 10 - 15 minutes

Create the Thermal Analysis Study

1 Open the EndCap Part.

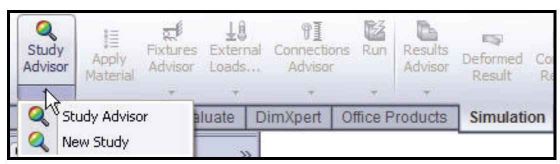
- Click **Open** from the Menu bar toolbar.
- Double-click **EndCap** from the SeaBotix\SolidWorks Simulation Professional\Thermal folder.



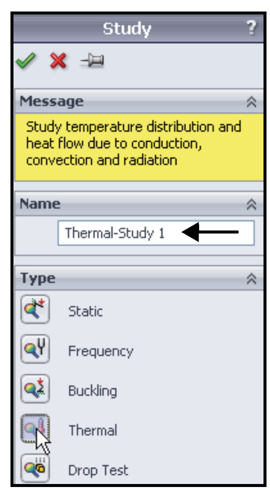
Note: Files of type is Part. The EndCap is displayed in the Graphics area.

2 Create a Thermal Study.

- Click the **Simulation** tab in the CommandManager.



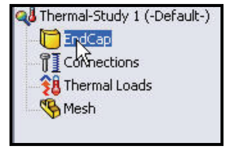
- Click the **Study Advisor** drop-down arrow from the Simulation tab.
- Click **New Study** . The Study PropertyManager is displayed.
- Enter **Thermal-Study 1** for the name of the Study.
- Click **Thermal** for Type.



3 Display the Study.

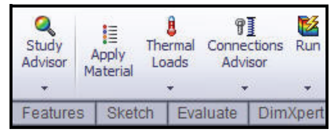
- Click **OK** from the Study PropertyManager.

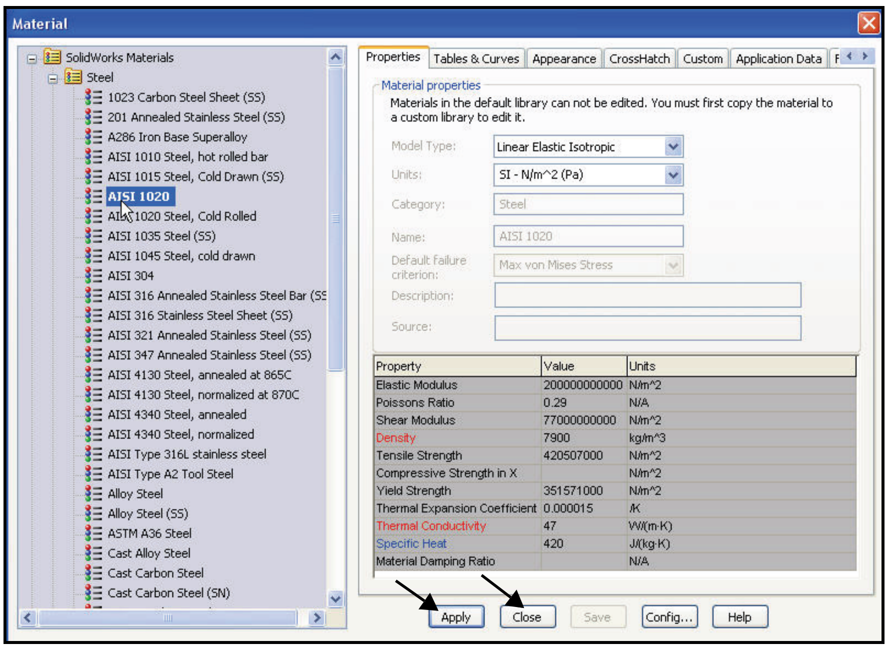
Applying the EndCap Material.



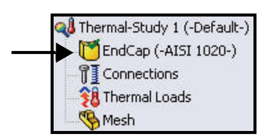
1 Apply the Material of the EndCap.

- Click **EndCap** from Thermal-Study 1 (-Default-).
- Click **Apply Material**  from the Simulation tab in the CommandManager. The Material dialog box is displayed. View your options.
- Click **AISI 1020** from the Steel folder.
- Click **Apply**.
- Click **Close** from the Material dialog box.





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



Thermal Loads and Boundary Conditions

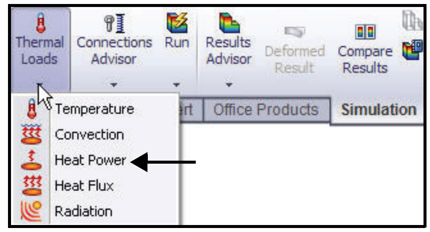
Thermal loads and restraints are only available for thermal studies. For steady state thermal studies with a heat source, a mechanism for heat dissipation must be defined. Otherwise, analysis stops because the temperatures increase without bound. Transient thermal studies run for a relatively short period of time and thus do not require a heat dissipation mechanism.

You will assume natural convection for the EndCap. You will apply a 600 watt power load to the system to simulate the heat load generated from the internal camera and search lights.

The following types of loads and restraints are available for thermal studies:

Load Type	Geometric Entities	Reference Geometry Type	Required Input
Temperature	Vertexes, Edges, Faces and components	N/A	Unit and temperature value.
Convection	Faces	N/A	Film coefficient and bulk temperature in the desired units.
Radiation	Faces	N/A	Unit and value of the surrounding temperature, emissivity, and view factor for surface to ambient radiation.
Heat Flux	Faces and an optional vertex for **thermostat** location for transient studies	N/A	Unit and value of the heat flux (heat power/unit area). Temperature range for optional thermostat for transient studies.
Heat Power	Vertexes, Edges, Faces, Components, and an optional vertex for **thermostat** location for transient studies	N/A	Unit and value of the heat power. The specified value is applied to each selected entity. Temperature range for optional thermostat for transient studies.

Applying a Thermal Load



1 Apply a Thermal Load.

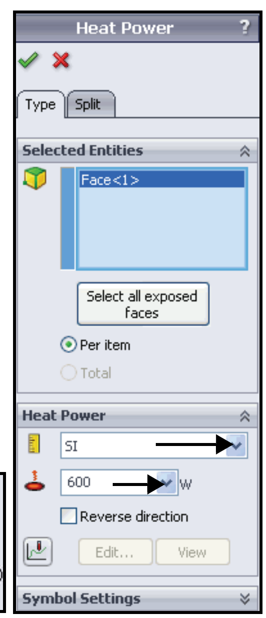
- Click the **Thermal Loads** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Heat Power** . The Heat Power PropertyManager is displayed.

2 Select the Face.

- Zoom in on the **center hole face** of the End Cap.
- Click the **inside center hole face** of the EndCap as illustrated. Face<1> is displayed in the Selected Entities box. Note the icon system feedback symbol for a face.

3 Enter Heat Power.

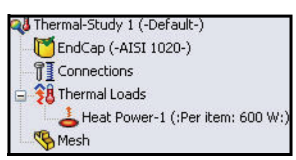
- Select **SI** from the Units drop-down menu.
- Enter **600** watts in the Heat Power box.



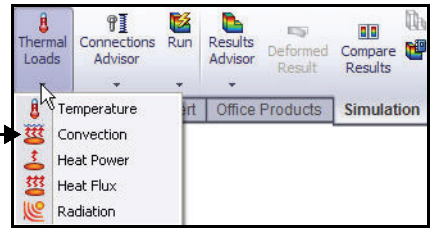
Note: 600 watts is an estimate for the total amount of power generated by the camera and the internal search lights of the assembly.

4 Apply the Values.

Click **OK** from the Heat Power PropertyManager. Heat Power-1 is displayed.



Applying Convection

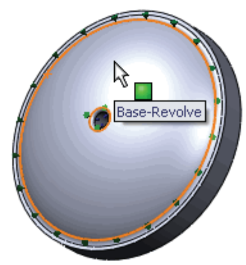


1 Apply Convection.

- Click the **Thermal Loads** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Convection** . The Convection PropertyManager is displayed.

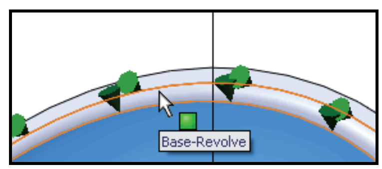
2 Select the Exposed Faces.

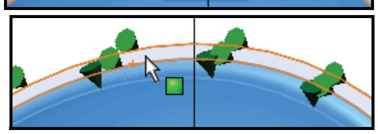
- Rotate the **EndCap** with the middle mouse button as illustrated.
- Click the **outside face** of the EndCap. Face<1> is displayed in the Faces for Convection box.



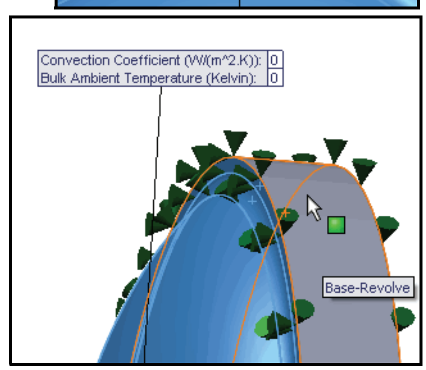
3 Select the other three exposed outside Faces.

- Click the other **three outside faces** of the EndCap. Face<2>, Face<3>, and Face<4> are displayed in the Faces for Convection box. Rotate the model to select Face<4>.





Note: Apply the **Zoom to Area** tool from the Heads-up View toolbar to select the correct faces.



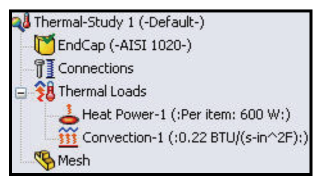
4 Set Units and Value.

- Select **English (IPS)** from the Units drop-down menu.
- Enter **0.22** in the Convection Coefficient box.
- Enter **50** in the Bulk Ambient Temperature box.

Note: The inputs simulate seawater conditions at the operating depth of 3,400 feet.

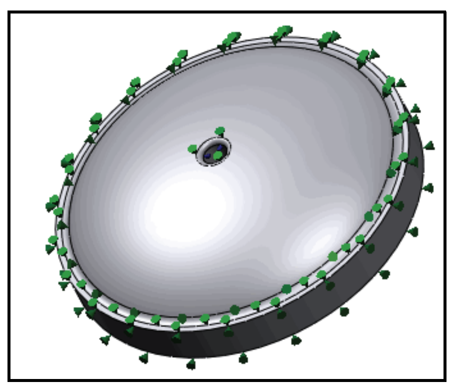
5 Apply the Values.

- Click **OK**  from the Convection PropertyManager. Convection-1 is displayed.



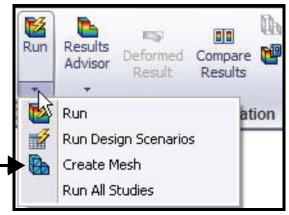
6 Fit the model to the Graphics area.

- Press the **f** key.

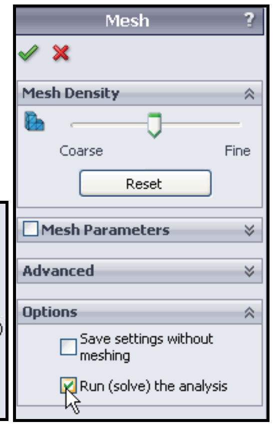


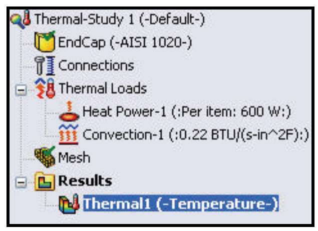
Note: SolidWorks Simulation Professional applies convection to the four selected exposed faces and creates a single entry. Convection symbols appear on the four selected outside faces.

Creating a Mesh and running an Analysis



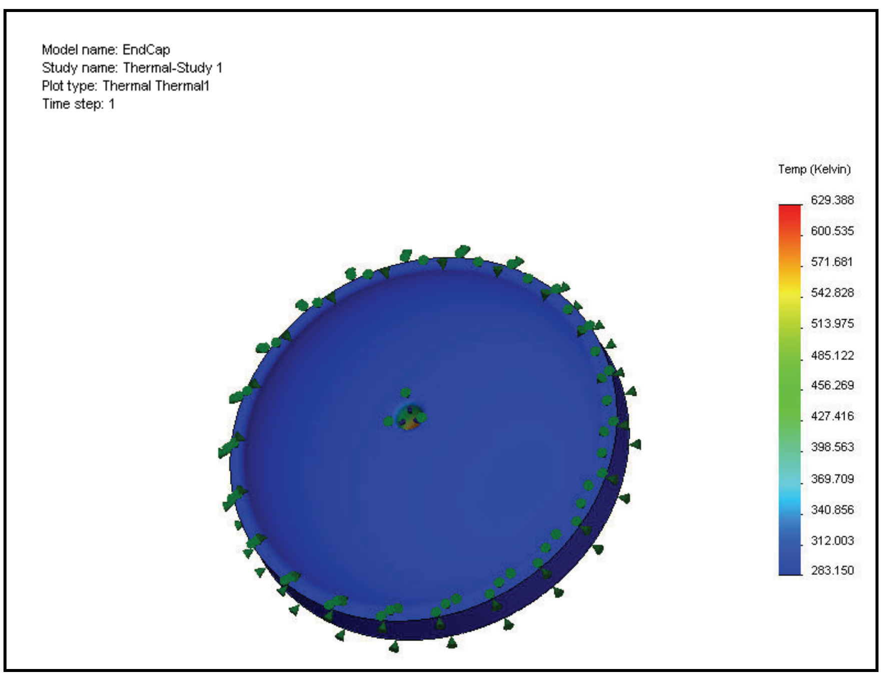
1 **Create a Mesh and run an Analysis.**
   - Click the **Run** drop-down arrow from the Simulation tab in the CommandManager.
   - Click **Create Mesh** . The Mesh PropertyManager is displayed suggesting Global Size and Tolerance value.
   - Check the **Run (solve) the analysis** box.





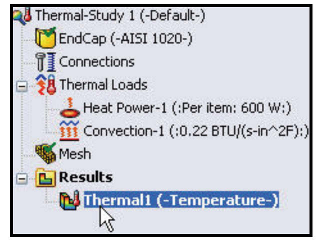
2 **Start the Mesh Process.**

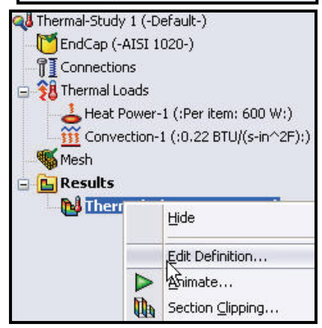
Click **OK** from the Mesh PropertyManager. You created a mesh and the Thermal1 plot is displayed.



3 View the Thermal Plot.

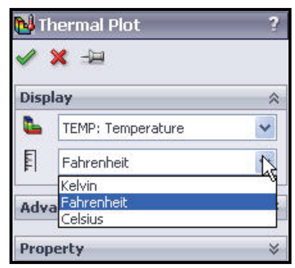
- Double-click **Thermal1 (-Temperature-)**. The Thermal Plot PropertyManager is displayed. View the options.
- Click **OK** from the Thermal Plot PropertyManager.
- Right-click **Thermal1 (-Temperature-).**
- Click **Edit definition**. The Thermal Plot PropertyManager is displayed.



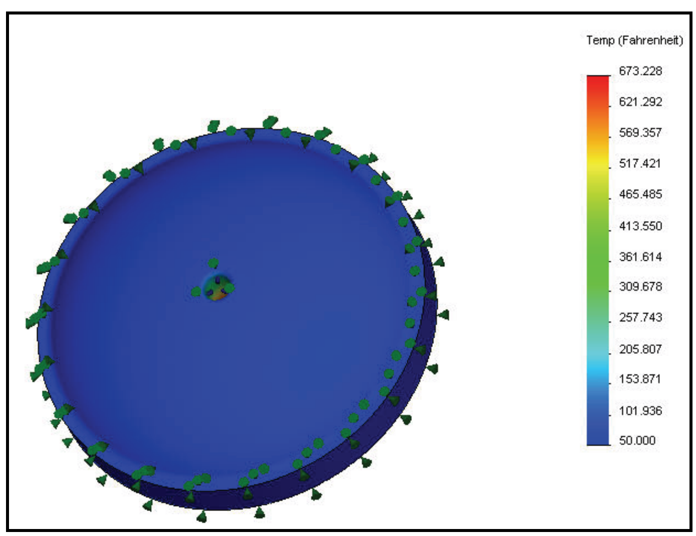


4 Modify Temperature units.

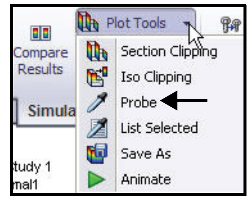
- Select **Fahrenheit** from the Temperature drop-down menu.
- Click **OK** from the Thermal Plot PropertyManager. The Thermal Plot is displayed in Fahrenheit.
- **Rotate** the model with the middle mouse button to view the temperature profile.



Note: Note that the maximum temperature is approximately 673°F.

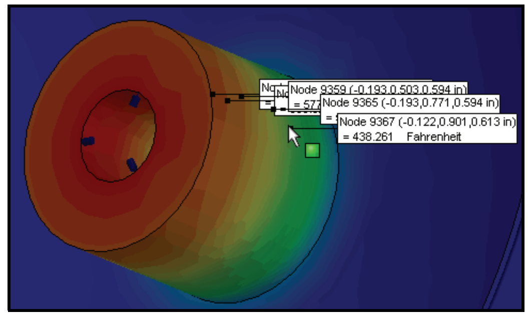


Applying the Probe tool

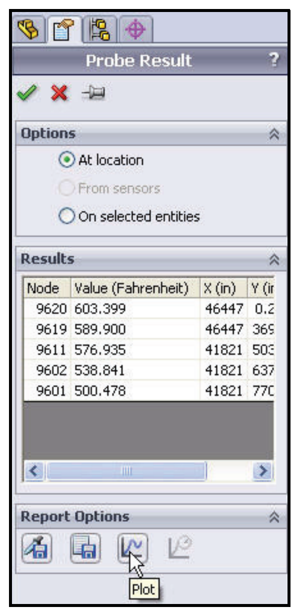


1 Apply the Probe tool.

- Click the **Plot Tools** drop-down arrow menu from the Simulation tab in the CommandManager.
- Click **Probe** . The Probe PropertyManager is displayed. The Probe tool provides the ability to list the temperature at a specific location in the model.
- Zoom in on the **inside face** as illustrated.
- Click five **points** as illustrated from top to bottom as illustrated. The Probe box lists the temperature and the X, Y, and Z coordinates of the selected vertices in the global coordinate system.



Note: Results will vary depending on your selected position of the EndCap.



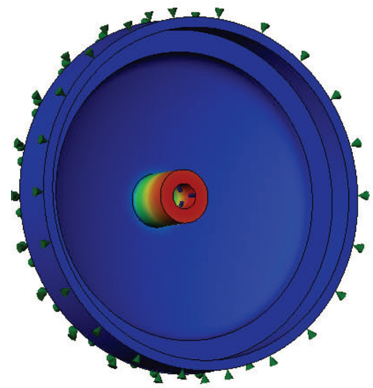
2 View and close the Probe Plot.

- Click **Plot** . The Probe Result window appears with a graph of temperatures at the selected vertices versus node numbers at the vertices. View the plot.
- **Close** the plot.
- Click **OK** in the Probe Result PropertyManager.

3 Fit the model to the Graphics window.

- Press the **f** key.

Modify the Design



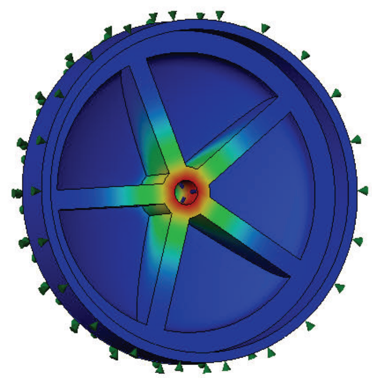
Without Ribs

In the first study, temperatures reaching approximately 673°F on the center hole of the EndCap were calculated using the supplied Load information.

In this section, redesign the EndCap to use ribs. The ribs will help to dissipate the heat generated by the camera and search lights within the EndCap to the surrounding seawater.

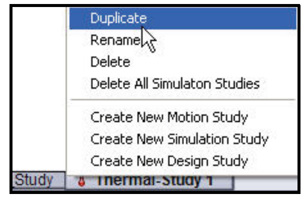
You will:

- Unsuppress the rib feature in the EndCap part.
- Copy and paste the material and Load/Restraint information from the first study to the second study.
- Mesh and Run the second analysis.
- View the results of the second study.
- Compare the first study to the second study.



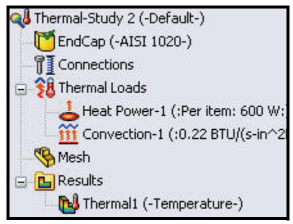
With Ribs

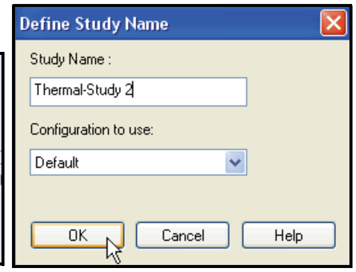
Create the Second Analysis



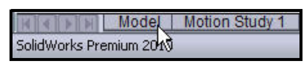
1 Create Thermal-Study 2.

- Right-click the **Thermal-Study 1** tab at the bottom of the Graphics area as illustrated.
- Click **Duplicate**. The Define Study Name dialog box is displayed.
- Enter **Thermal-Study 2** for new Study name.
- Click **OK** from the Define Study Name dialog box. Thermal-Study 2 is displayed.

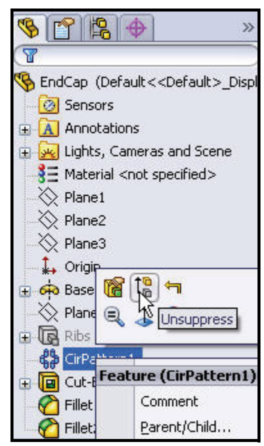




2 Add Ribs to the EndCap Part.



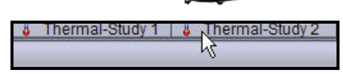
- Click the **Model** tab at the bottom of the Graphics area.
- Right-click **CirPattern1** from the FeatureManager.



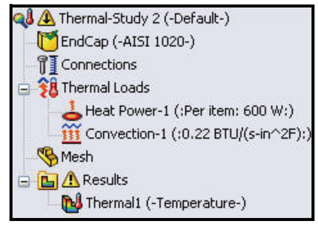
- Click **Unsuppress**  from the Context toolbar. The EndCap with Ribs is displayed in the Graphics area.

3 Return to Thermal-Study 2.

- Click the **Thermal-Study 2** tab at the bottom of the Graphics area.

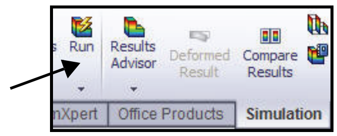


4 Review Thermal-Study 2.



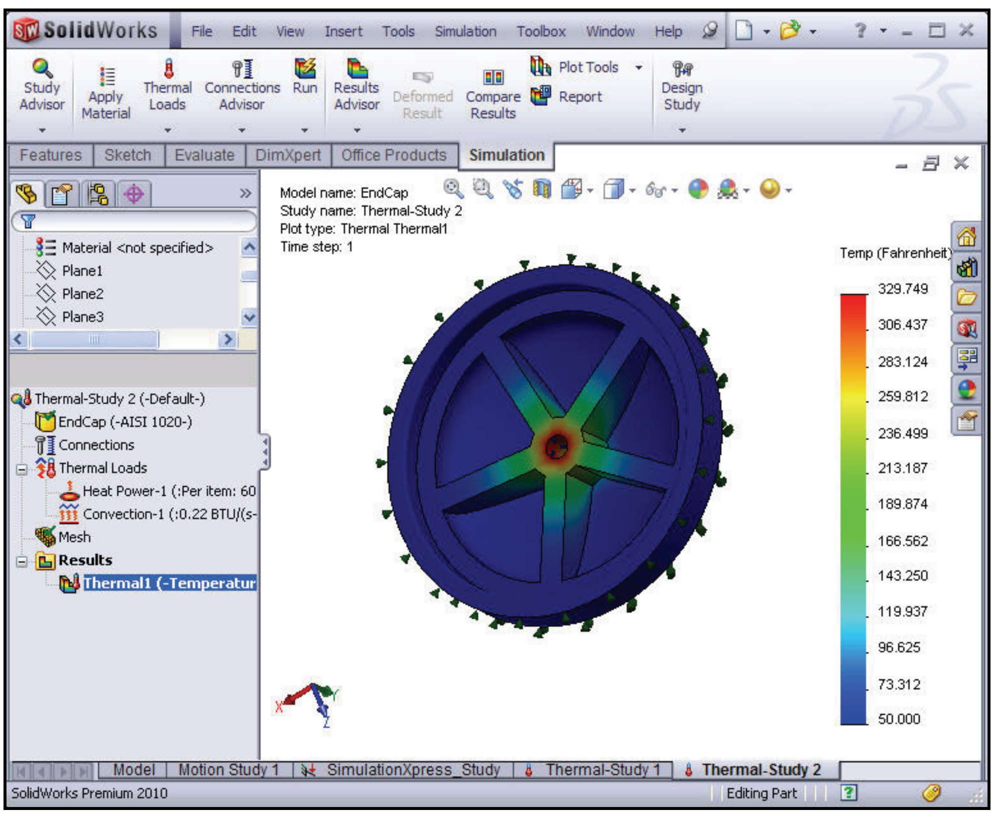
- Review Thermal-Study 2. The Thermal information was copied from the first study to the second study.

5 Analysis the Model.

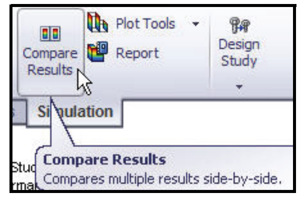


- Click **Run** from the Simulation tab in the CommandManager. Thermal1 (-Temperature-) is displayed. View the plot in the Graphics area.

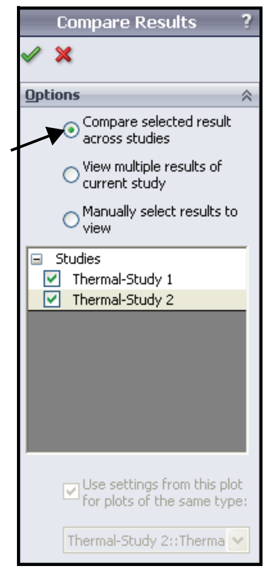
Note: The addition of the ribs resulted in a temperature range between 50 and 329 °F.

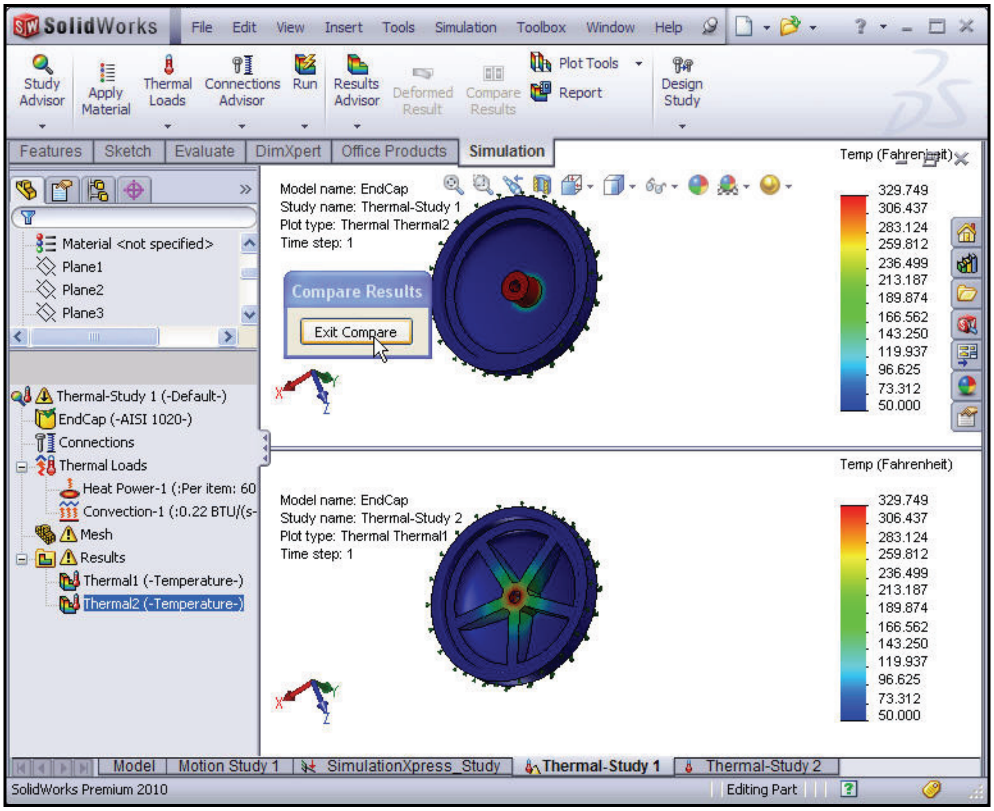


6 Compare Study 2 to Study 1.



- Click **Compare Results** from the Simulation CommandManager. The Compare Results PropertyManager is displayed. Both Study 1 and Study 2 are checked.
- Click the **Compare selected results across studies** box. Note: The Use settings from this plot for plots of the same type box is selected.
- Click **OK** from the Compare Results PropertyManager. View the Graphics area. The two Studies are displayed.



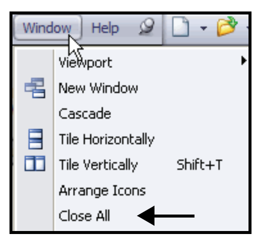


7 **Return to Study 2.**

- Click the **Exit Compare** button. View Thermal-Study 2.

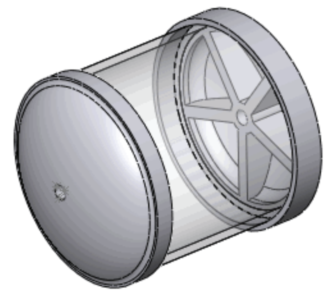
8 **Save and Close the Model.**

- Click **Save** .
- Click **Window**, **Close All** from the Menu bar menu.



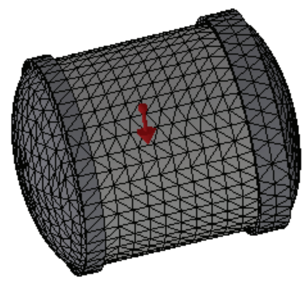
Note: You improved the thermal dissipation of the Endcap by adding the Ribs. The Ribs added mass, which in turned provided a better thermal load path to the entire part.

Drop Test Analysis



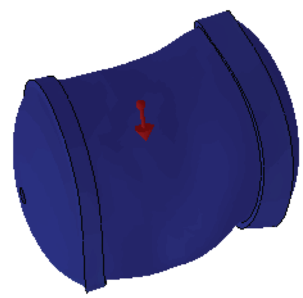
A Drop Test study evaluates the effect of the impact of a part or an assembly with a rigid or flexible planar surface. Dropping an object on the floor is a typical application and hence the name. The program calculates impact and gravity loads automatically. No other loads or restraints are allowed. The program solves a dynamic problem as a function of time.

Will your Design Fail?



The study does not answer this question automatically. It can predict the separation of components due to impact. You will use the results to assess the possibility of such an event occurring. You will use maximum stresses to predict material failure and contact forces to predict separation of components.

Perform a Drop Test analysis on the Housing component.

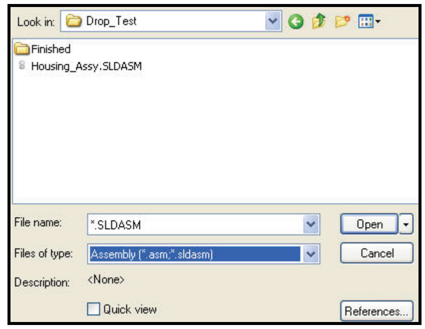


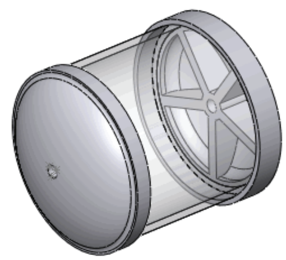
Time: 20 - 25 minutes

Creating a Drop Test Study

1 Open the Housing Assembly.

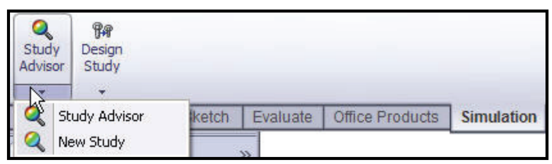
- Click **Open** from the Menu bar toolbar.
- Double-click the **Housing_Assy** assembly from the SeaBotix\SolidWorks Simulation Professional\Drop_Test folder. The Housing is displayed in the Graphics area.





2 Create a Drop Test Study.

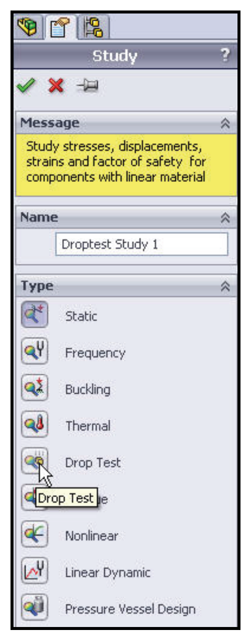
- Click the **Study Advisor** drop-down arrow from the Simulation tab in the CommandManager.

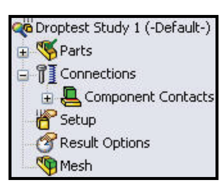


- Click **New Study**. The Study PropertyManager is displayed.
- Enter **Droptest Study 1** for Study Name.
- Click the **Drop Test** button for Type.

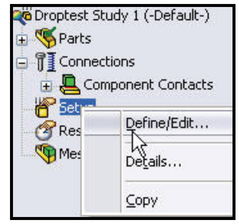
3 Display the Study.

Click **OK** from the Study PropertyManager. Droptest Study 1 (-Default-) is displayed.

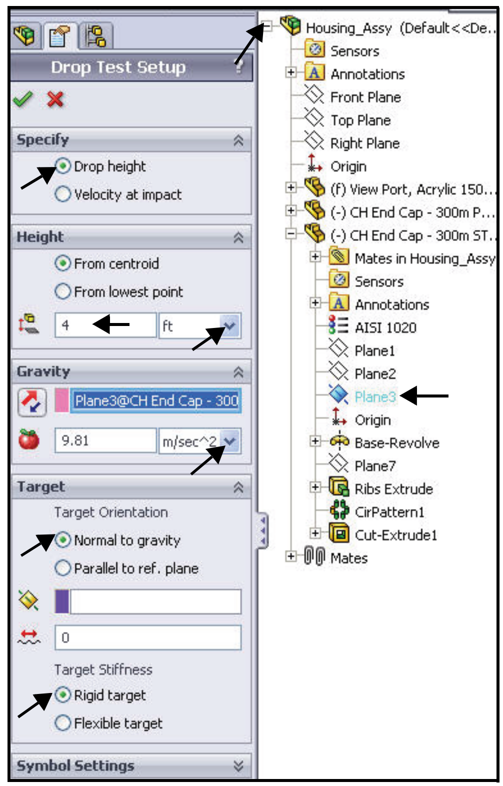




4 Set up the Drop Test Study.

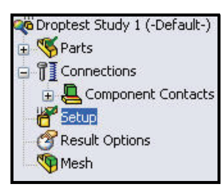


- Right-click the **Setup** folder as illustrated.
- Click **Define/Edit**. The Drop Test Setup PropertyManager is displayed.

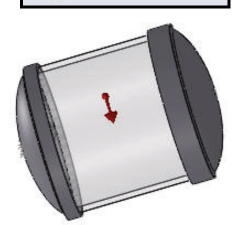


- Check the **Drop height** box.
- Select **ft** for units from the drop-down menu.
- Enter **4** in the Drop height from centroid box.
- Click inside the **Gravity** box.
- Expand the **Housing_Assy** flyout from the Graphics area.
- Expand the **second CH EndCap** component as illustrated
- Click inside the **Gravity** plane selection set.
- Click **Plane3** from the flyout FeatureManager. Note: Under the second CH EndCap component. Plane 3 is displayed in the Gravity box.
- Select **m/sec^2** for the Gravity magnitude units.
- Click the **Normal to gravity** box.
- Click the **Rigid target** box for Target Stiffness.

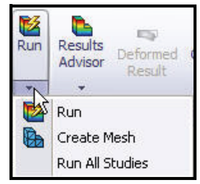
5 Display the Study.



- Click **OK** ✓ from the Drop Test Setup PropertyManager. Setup is displayed with a check mark.
- **Rotate** the model with the middle mouse button. View the direction arrow pointing downwards.

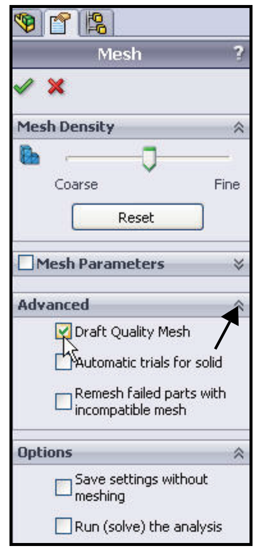


Meshing the Model



1 Mesh the model.

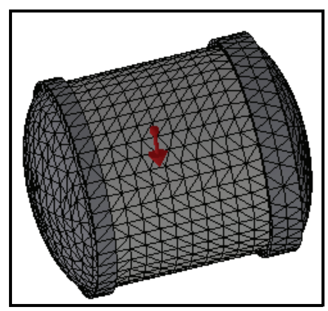
- Click the **Run** drop-down arrow menu from the Simulation tab in the CommandManager
- Click **Create Mesh** . The Mesh PropertyManager is displayed.
- Expand the **Advanced** dialog box.
- Check the **Draft Quality Mesh** box as illustrated.

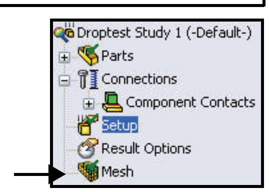


Note: A coarse Mesh Factor will result in a faster mesh time. Actual results will vary depending on Mesh Factor.

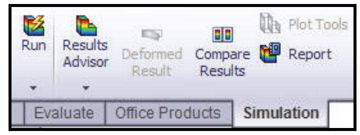
2 Start the Mesh and Analysis Process.

- Click **OK** from the Mesh PropertyManager. Meshing starts and the Mesh Progress window appears. After meshing is completed, a checkmark is displayed next to the Mesh folder.



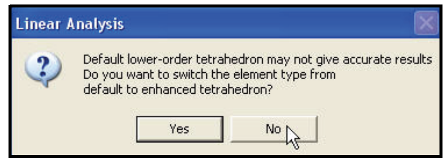


Running the Analysis

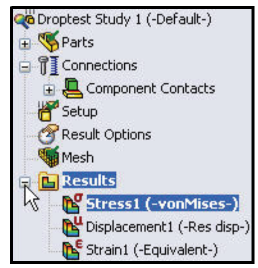


1 Run the Analysis.

- Click **Run** . The Run PropertyManager is displayed. The analysis runs and the default plots are created.
- Click **No** in the Linear Analysis dialog box to retain your element choice.

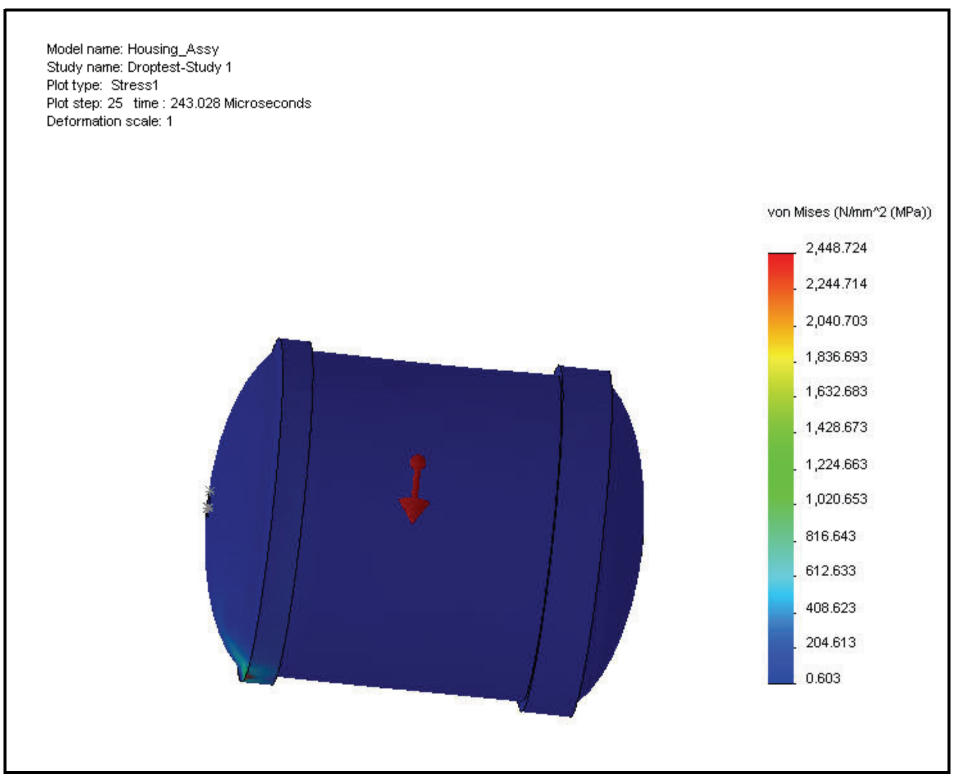


Note: Run time will take approximately 15 seconds.



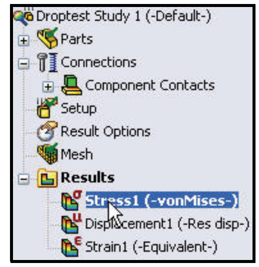
2 Review the Results Folder.

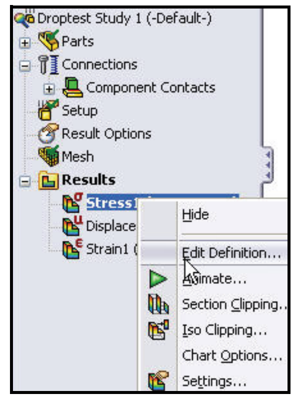
- Expand the **Results** folder. The Results folder contains three plots: Stress, Displacement, and Strain. View the Stress1 (-vonMises-) plot in the Graphics area.

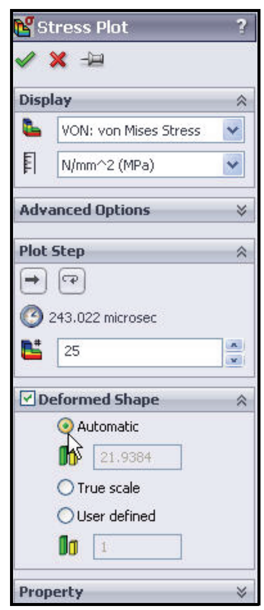


3 Set Scale Factor and View the von-Mises Plot.

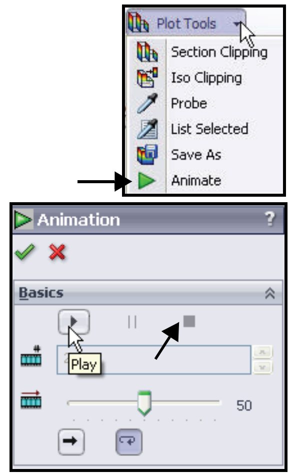
- Double-click **Stress1 (-vonMises-)**. The Stress Plot PropertyManager is displayed. View the options.
- Click **OK** from the Stress Plot PropertyManager.
- Right-click **Stress1 (-vonMises-)**.
- Click **Edit Definition**. The Stress Plot PropertyManager is displayed.
- Click **Automatic** in the Deformed Shape box. Accept the default values.
- Click **OK** from the Stress Plot PropertyManager. View the plot in the Graphics area.





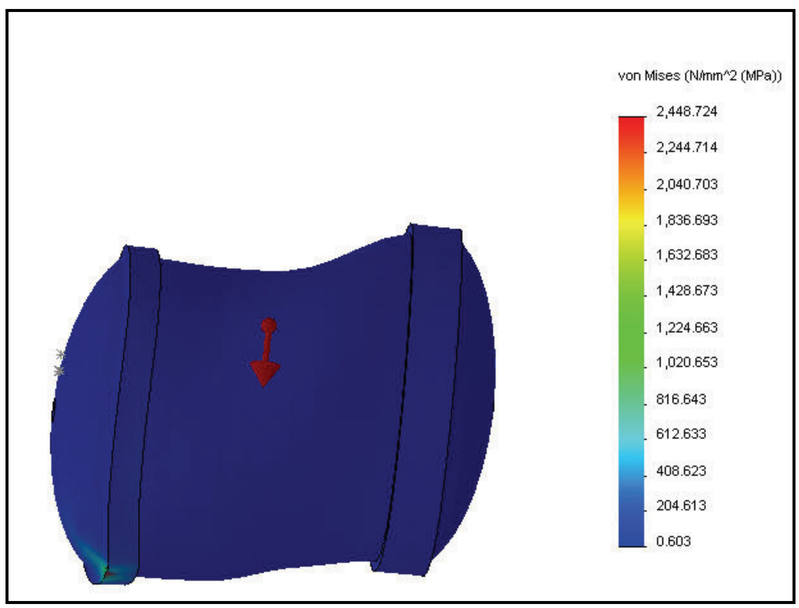


Animating the Plot



1 Animate the Plot.

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

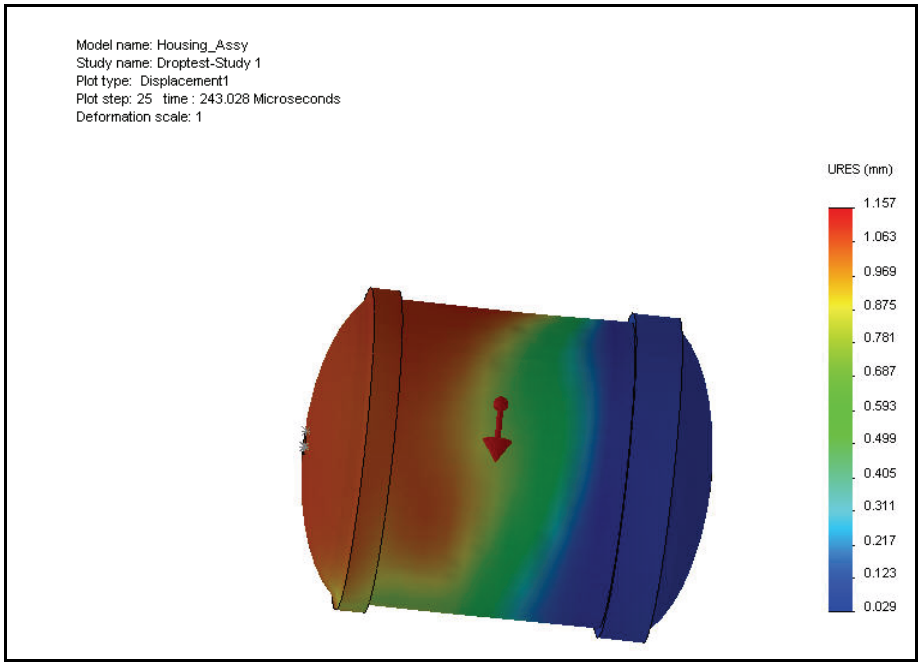


- Click **OK** from the Animation PropertyManager.

Note: You can save the animation of the plot in an AVI file format.

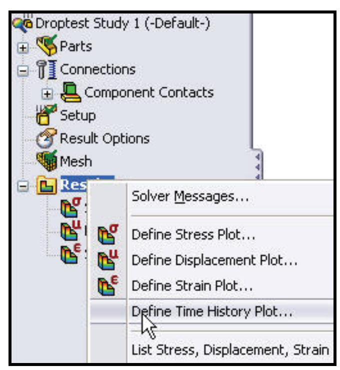
2 View the Displacement Plot.

- Double-click **Displacement1 (-Res disp-)**. View the plot in the Graphics window.

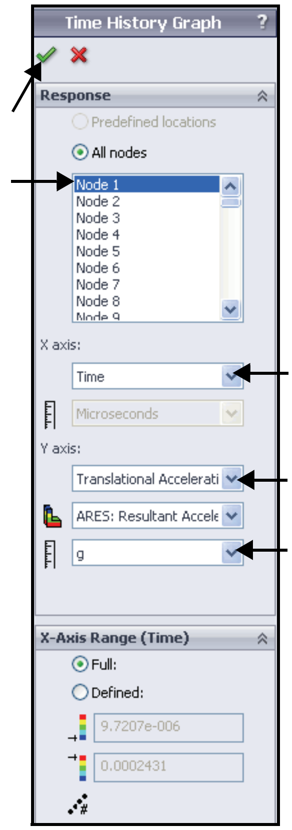


3 Create the Time History Graph.

- Right-click the **Results** folder.
- Click **Define Time History Plot**. The Time History Graph PropertyManager is displayed.



- Click **Node 1** as illustrated.
- Select **Time** for X-Axis from the drop-down menu.
- Select **Translational Acceleration** for Y-Axis from the drop-down menu.
- Select **g** for Units from the drop-down menu.

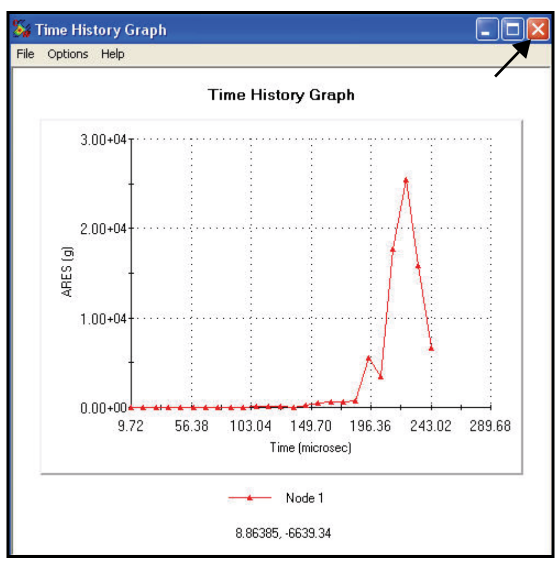


4 View the Time History Graph.

- Click **OK** from the Time History Graph PropertyManager. View the Graph.
- **Close** the Time History Graph.

5 Save and Close the Model.

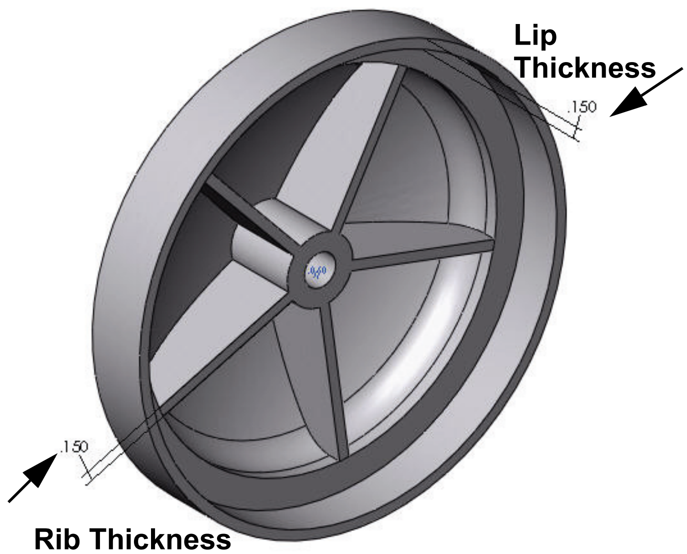
- Click **Save**.
- Click **Window**, **Close All** from the Menu bar menu.



Optimization Analysis

The Optimization analysis enables designers to meet functional design specifications without wasting materials and overdesigning. Seemingly insignificant amounts of weight cut from dozens of components can add significant cost reductions in production, shipping, and packaging. You can also test designs with alternate lighter or lower-cost materials in SolidWorks Simulation.

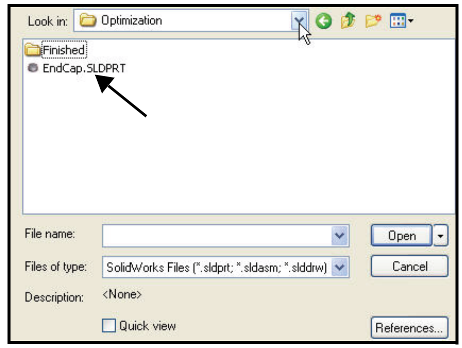
Perform an Optimization analysis today on the EndCap. The goal of the analysis is to minimize the mass of the EndCap. Optimize the EndCap Lip thickness and the Rib thickness in the analysis.



The Factor of Safety is greater than one.

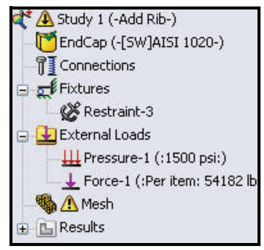
Time: 15 - 20 minutes

Creating an Optimization Analysis



1 Open the Part.

- Click **Open**  from the Menu bar toolbar.
- Double-click **EndCap** from the SeaBotix\SolidWorks Simulation Professional\Optimization folder. The EndCap (Add Rib) configuration is displayed in the Graphics area.

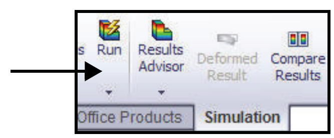


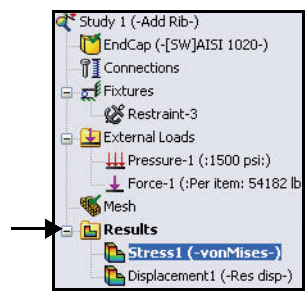
2 View Static Study1.

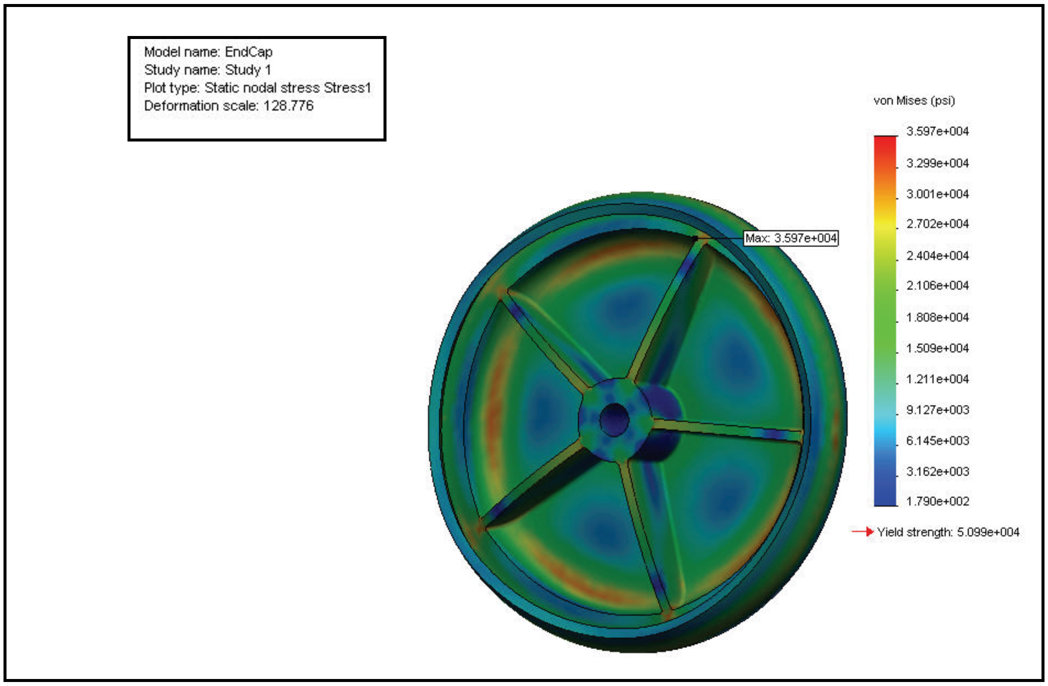
- A static study was created for this part. Click the **Study 1** tab located at the bottom of the Graphics area as illustrated. Study 1 is displayed.

3 Run Study 1.

- Click **Run**  from the Simulation tab in the CommandManager. View the created plots in the Results folder. The Stress1 (-vonMises-) plot is displayed in the Graphics area.

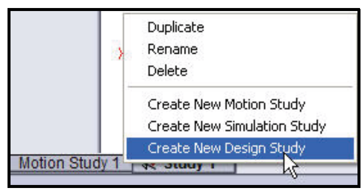


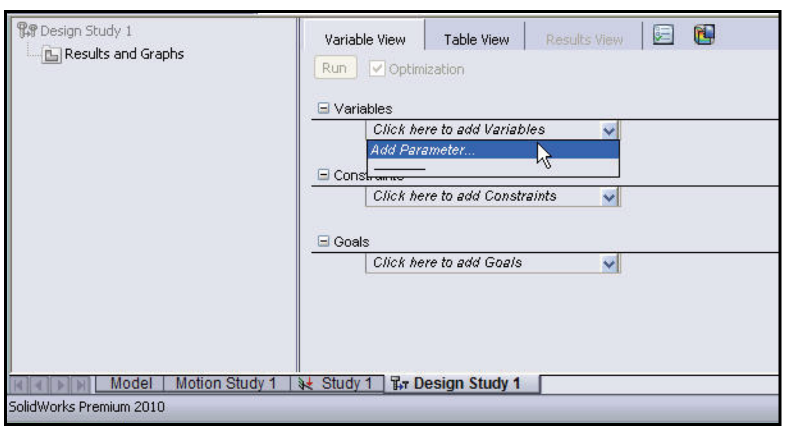


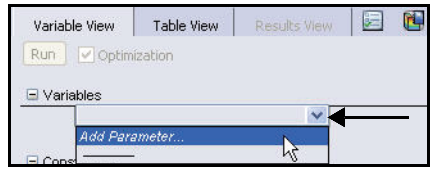


4 **Create an Optimization Study.**

- Right-click on the **Study 1** tab at the bottom of the Graphics area.
- Click **Create New Design Study**. The Design Study 1 tab is displayed along with the Design Study dialog box.







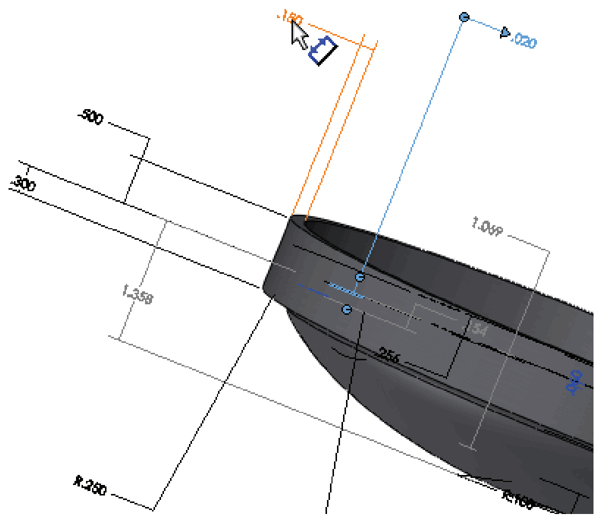
Note: You can also click Simulation, Design Study from the Menu bar menu.

5 Select the First Design Variable (EndCap Thickness) for the Optimization Study.

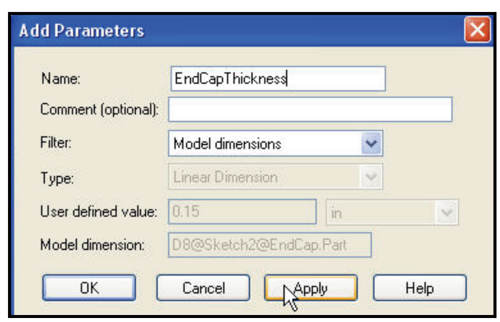
- Click **Add Parameters** from the Variables drop-down menu. The Parameters and Add Parameters dialog box is displayed.

6 Locate the .150 EndCap Lip thickness dimension.

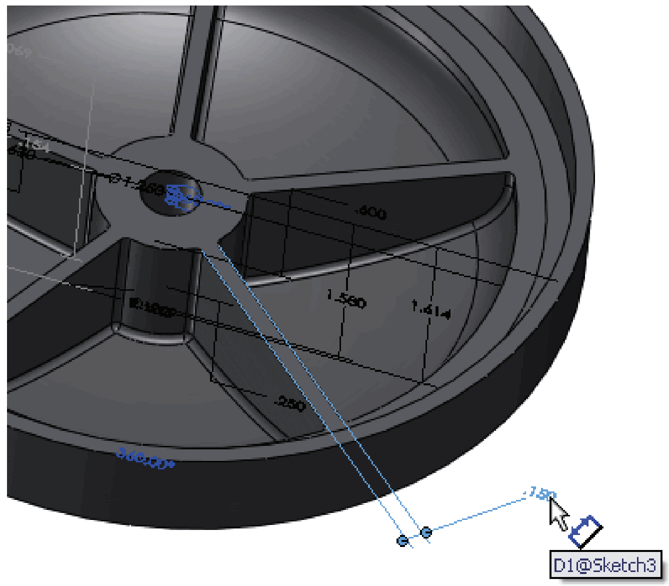
- **Rotate** the model with the middle mouse button and Zoom in on the .150 EndCap Lip thickness dimension.



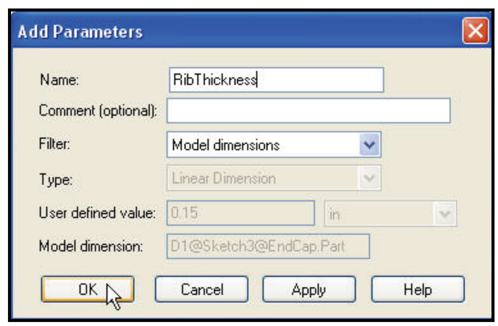
- Click the **.150** EndCap Lip thickness dimension as illustrated. The selected dimension is displayed in the Add Parameters dialog box.
- Enter **EndCapThickness** for Name.
- Click **Apply**. The information is added to the Parameters dialog box.



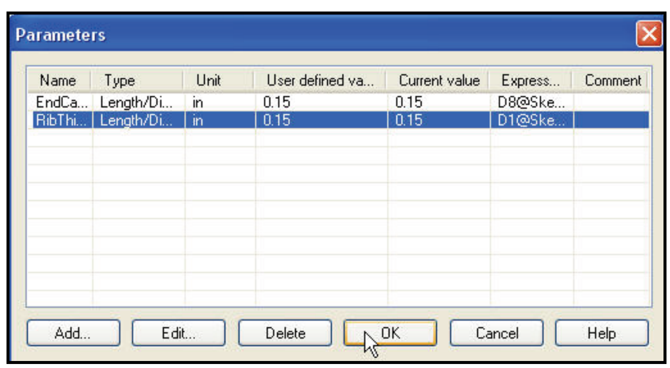
7 Select the Second Design Variable (Rib Thickness) for the Optimization Study.



- Click the **.150** EndCap Rip thickness dimension as illustrated. The selected dimension is displayed in the Add Parameters dialog box.
- Enter **RibThickness** for Name.

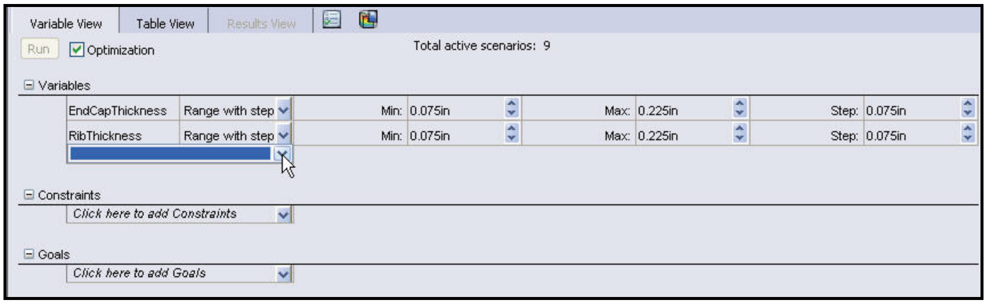


- Click **OK** from the Add Parameters dialog box. The information is added to the Parameters dialog box. View the Parameters dialog box.
- Click **OK** from the Parameters dialog box.



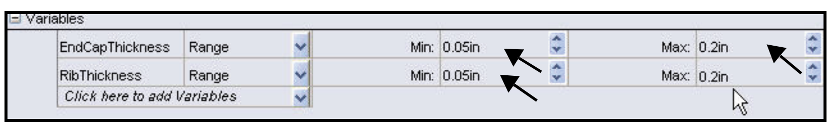
8 Expand the Variables cell in the Design Study.

- Click the **drop-down arrow** in the Variables cell. View the results.



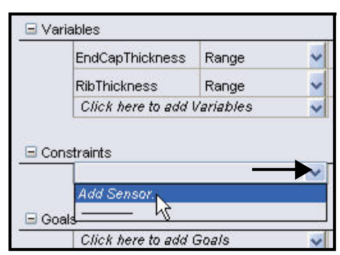
9 Set the ranges for the variables in the Design Study.

- Select **Range** from the drop-down menu for EndCap Thickness.
- Select **Range** from the drop-down menu for Rib Thickness.
- Enter the **illustrated numbers** for the EndCap Thickness (Min: & Max:) range and the Rib Thickness (Min: & Max:) range.



10 Set a Constraint (Sensor to monitor) the study.

- Click **Add Sensor** from the Constraint drop-down menu. The Sensor PropertyManager is displayed.



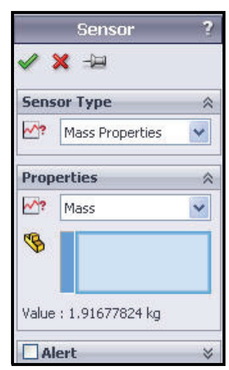
- Select **Simulation Data** for Sensor Type.
- Select **Stress** for Results.
- Select **psi** for Units.
- Click **OK** from the Sensor PropertyManager.

11 Set the conditions for the Constraint.

- Select **Is less than** for Stress.
- Enter **60000** for Max condition.

12 Set a Goal (Sensor to monitor) the study.

- Click **Add Sensor** from the Goals drop-down menu. The Sensor PropertyManager is displayed.
- Accept the default Sensor Type: Mass Properties. Click **OK** from the Sensor PropertyManager.

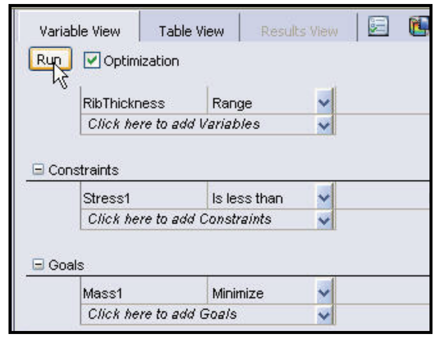


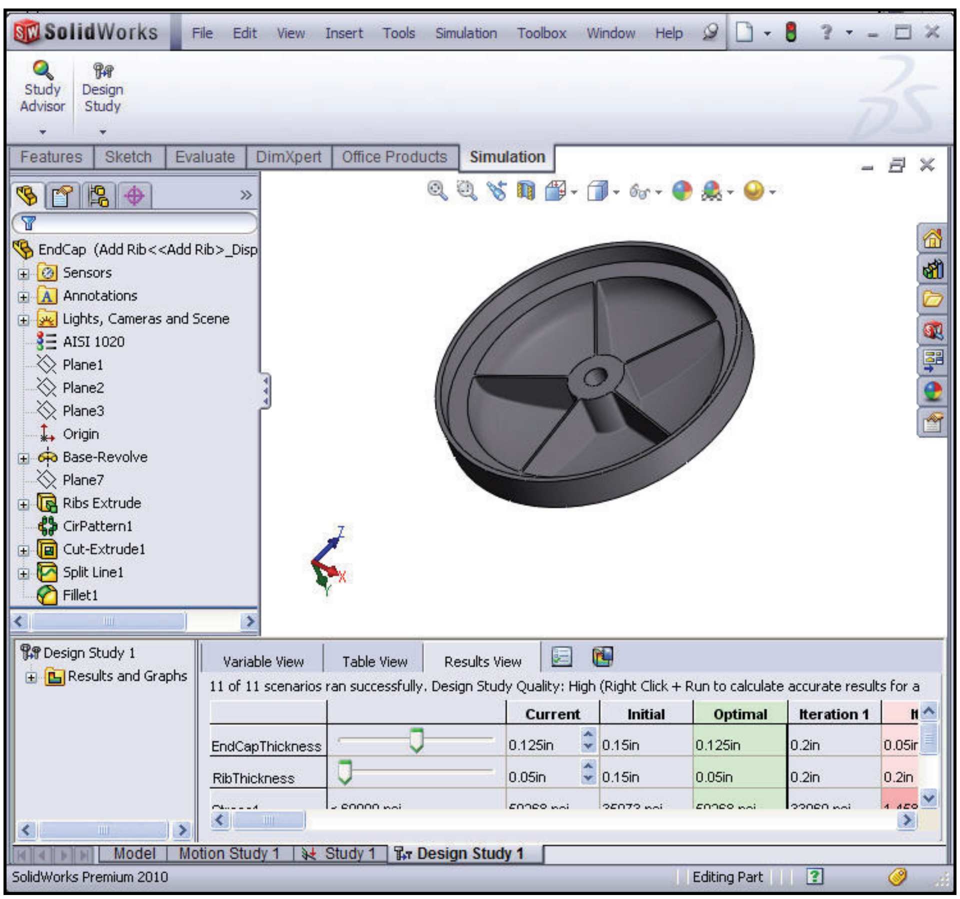
13 Set the condition for the Goal.

- Select **Minimize**.

14 Run the Design Study.

- Click the **Run** button. The results table is displayed and updated as the study is running. This may take a few minutes. View the finished table. You can now interact with the results.

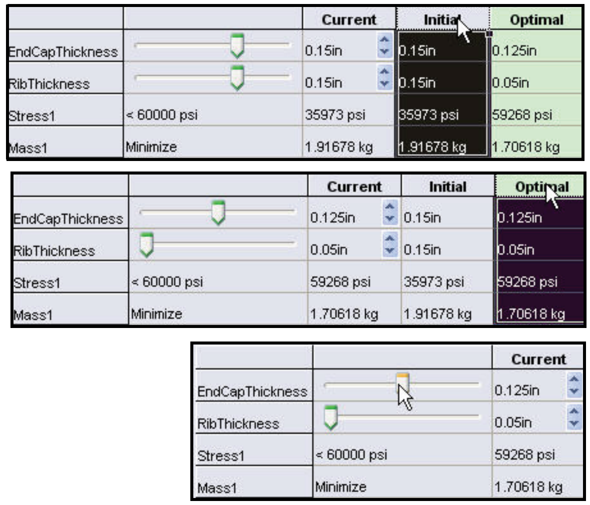




15 Interact with the Results.

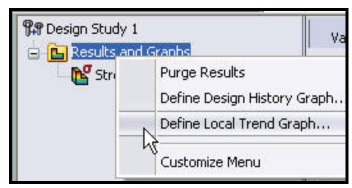
- Click in the **Initial** Column.
- Click in the **Optimal** Column. Compare the two columns.

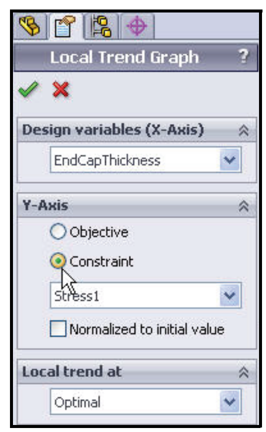
Note: You can look at any of the individual designs by dragging the EndCap Thickness or Rib Thickness slider.



16 View the Trend Results.

- Right-click the **Results and Graphs** folder.
- Click **Define Local Trend Graph**. The Local Trend PropertyManager is displayed. View your options.
- Click **Constraint**. Accept the default settings.
- Click **OK** from the PropertyManager. View the results in the Graphics area.



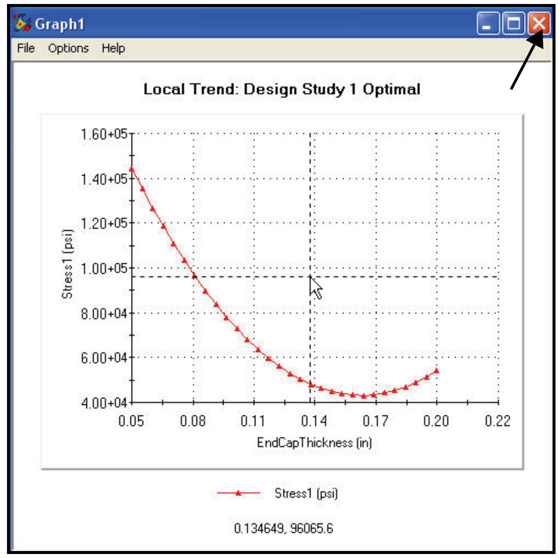


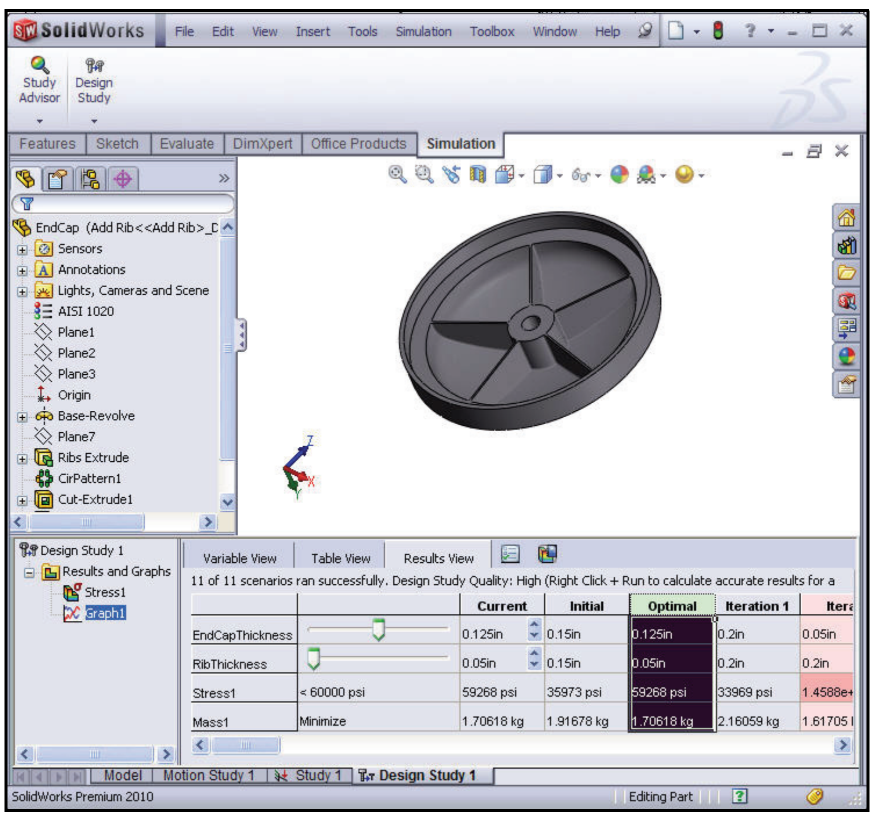
17 Close the Graph1 dialog box.

- Click **Close**.

18 Save and Close the Model.

- Click **Save** .
- Click **Window**, **Close All** from the Menu bar menu.

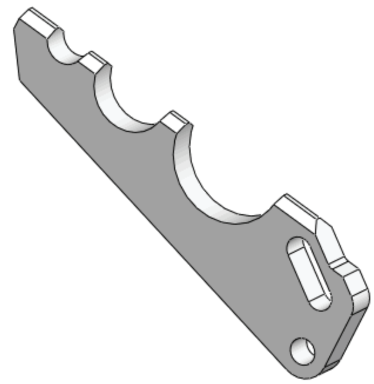




		Current	Initial	Optimal	Iteration 1	Itera
EndCapThickness		0.125in	0.15in	0.125in	0.2in	0.05in
RibThickness		0.05in	0.15in	0.05in	0.2in	0.2in
Stress1	< 60000 psi	59268 psi	35973 psi	59268 psi	33969 psi	1.4588e+
Mass1	Minimize	1.70618 kg	1.91678 kg	1.70618 kg	2.16059 kg	1.61705

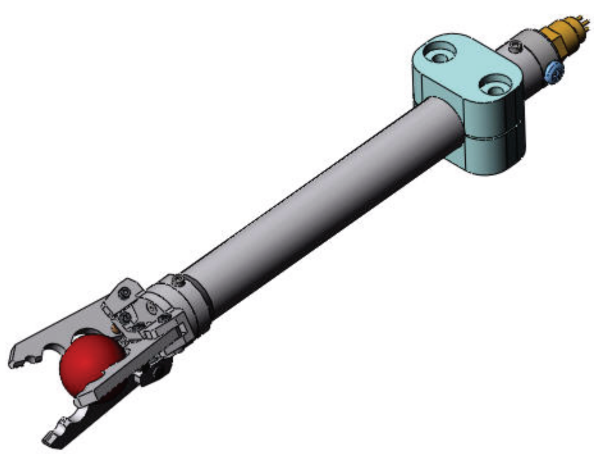
Fatigue Analysis

It is observed that repeated loading and unloading weakens objects over time even when the induced stresses are considerably lower than the allowable stress limits. This phenomenon is known as fatigue. Each cycle of stress fluctuation weakens the object to some extent. After a number of cycles, the object becomes so weak that it fails. Fatigue is a primary cause of the failure in many objects, especially those made of metals.



3 Finger Jaw

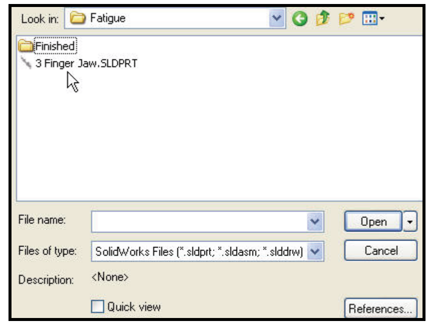
The SeaBotix LBV150 contains an optional MiniGrab assembly. In this study, you will analyze the 3 Finger Jaw part which is attached to the SeaBotix LBV150 to grip and hold objects from the sea floor. Before you create the Fatigue analysis, perform a Static analysis with a force applied to the tips of the 3 Finger Jaw.

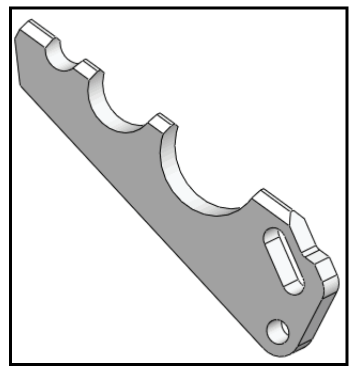


MiniGrab Assembly

Time: 15 - 20 minutes

Creating a Fatigue Analysis

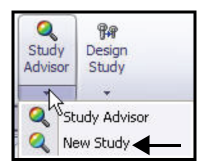




1 Open the Part.

- Click **Open** from the Menu bar toolbar.
- Double-click **3 Finger Jaw** from the SeaBotix\SolidWorks Simulation Professional\Fatigue folder.

2 Create a Static Analysis Study.

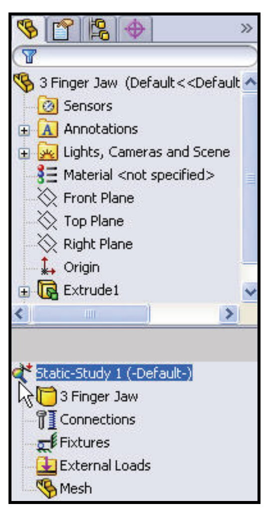


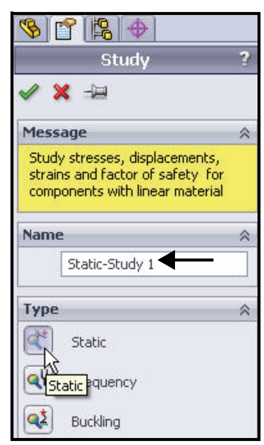
- Click the **Study Advisor** drop-down arrow from the Simulation tab in the CommandManager.
- Click **New Study** . The Study PropertyManager is displayed.
- Enter **Static-Study 1** for name.
- Click **Static** for Type.

3 Display Static-Study 1.

- Click **OK** from the Study PropertyManager.

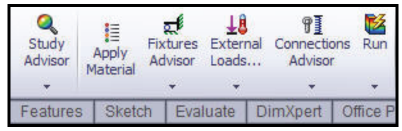
Note: The Static-Study 1 tab is displayed in the bottom corner of the Graphics area.





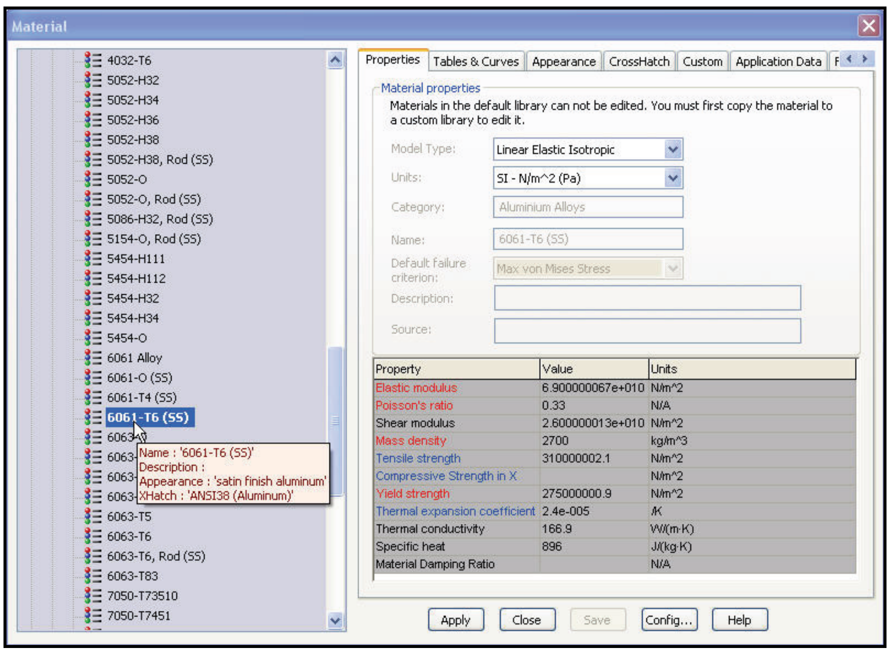
Applying Material

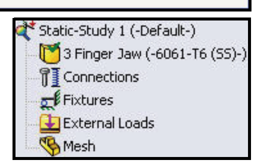
1 Apply Material.



- Click **Apply Material** from the Simulation tab in the CommandManager. The Material dialog box is displayed.
- Expand the **Aluminum Alloys** folder.
- Click **6061-T6(SS) Alloy**. View the material properties.
- Click **Apply**.
- Click **Close**. Material is applied to the part.

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

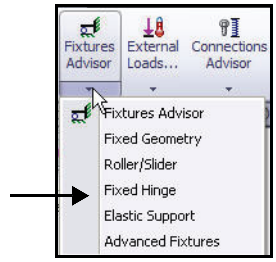




Adding a Fixture

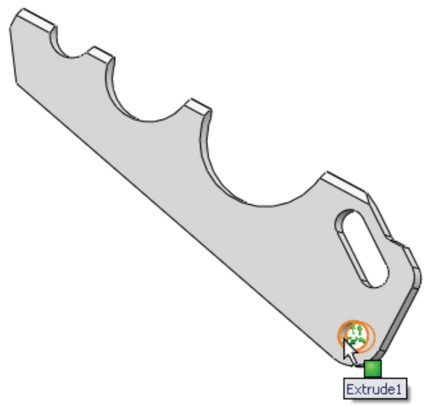
1 Add a Fixture.

- Click the **Fixtures Advisor** drop-down arrow from the Simulation tab in the CommandManager
- Click **Fixed Hinge**. The Fixture PropertyManager is displayed.



2 Select the Cylindrical Face to be Fixed.

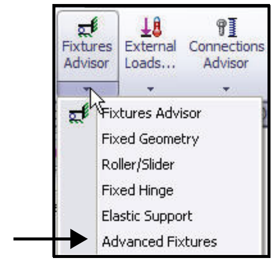
- Click the **inside cylindrical face** of the hole in the 3 Finger Jaw as illustrated. Face<1> is displayed. Note the icon feedback symbol for a face.



- Click **OK** from the Fixture PropertyManager. Fixed Hinge-1 is displayed.

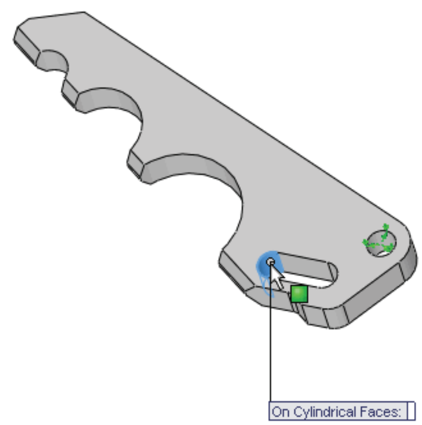
3 Add a Second Fixture.

- Click the **Fixtures Advisor** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Advance Fixtures.** The Fixture PropertyManager is displayed. Perform a radial support on the right-hand face.



4 Select the inside Cylindrical Face.

- Click the **On Cylindrical Faces** box.
- **Rotate** the model to view the side cylindrical face as illustrated.
- Click the **inside face of the slot** as illustrated. Face<1> is displayed.

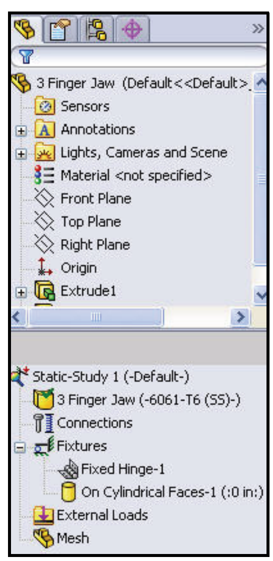


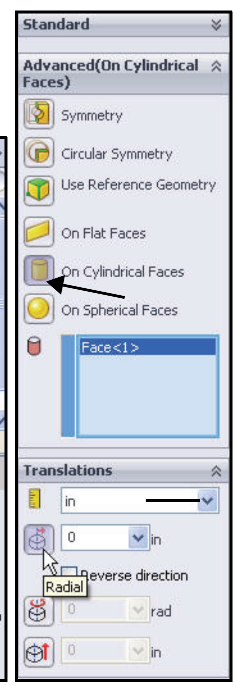
5 Select Units and Displacement Components.

- Select **inch** from the Units drop-down menu.
- Click the **Radial** box.

6 Apply the Second Fixture.

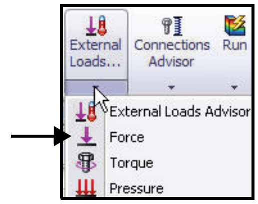
- Click **OK** from the Fixture PropertyManager. On Cylindrical Faces-1 is displayed.





Applying a Force

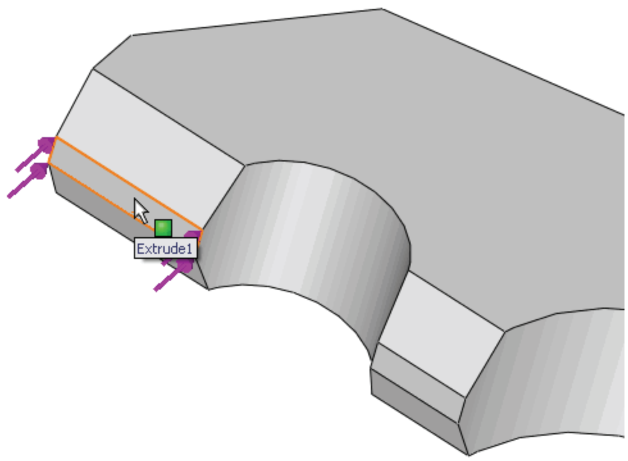
1 Apply a Force.



- Click the **External Loads** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Force** . The Force/Torque PropertyManager is displayed.
- Check the **Normal** box.

2 Select the contact face.

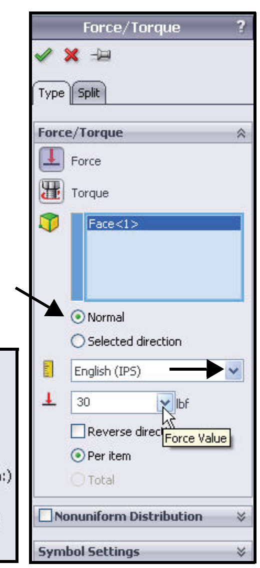
- **Rotate** the model with the middle mouse button to view the top contact face as illustrated.
- Click the **top contact face**. Face<1> is displayed in the Faces for Normal Force box.



3 Set Units and Value.

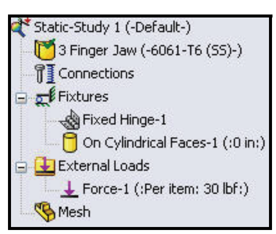
- Select **English (IPS)** in the Units box.
- Enter **30**lbf in the Force value box.

Note: 30lbf is the normal force that the MiniGrab assembly can apply in holding an object from the sea floor.



4 Apply the Force.

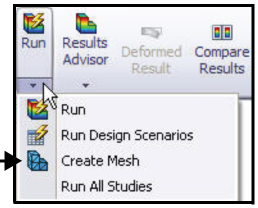
- Click **OK** from the Force/ Torque PropertyManager. Force-1 is displayed.

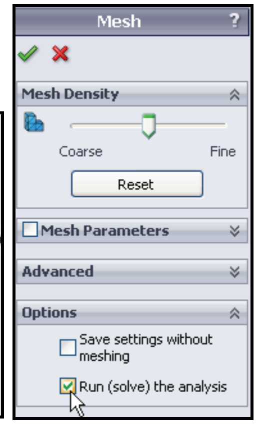


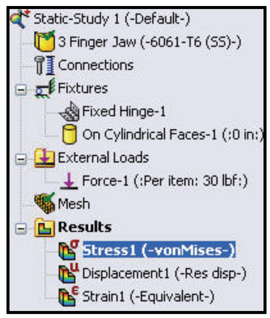
Meshing and Running the Model

1 Mesh and Run the Model.

- Click the **Run** drop-down arrow from the Simulation tab in the CommandManager
- Click **Create Mesh**. The Mesh PropertyManager is displayed.
- Check the **Run (solve) the analysis** box.
- Click **OK** from the Mesh PropertyManager. View the results. Three plots are created.

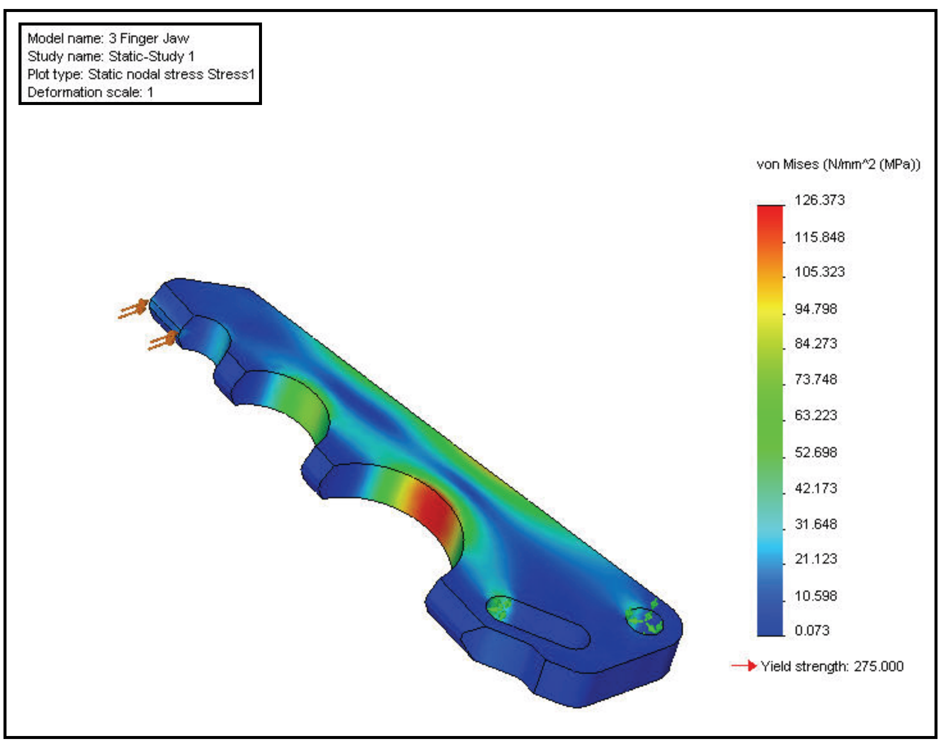




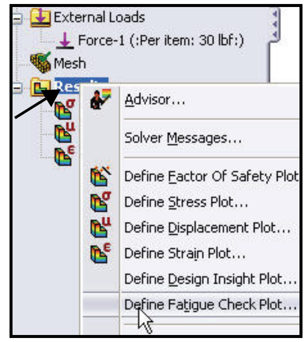


2 Fit the model to the Graphics area.

- Press the **f** key. View the Stress1 (-vonMises-) plot in the Graphics area.

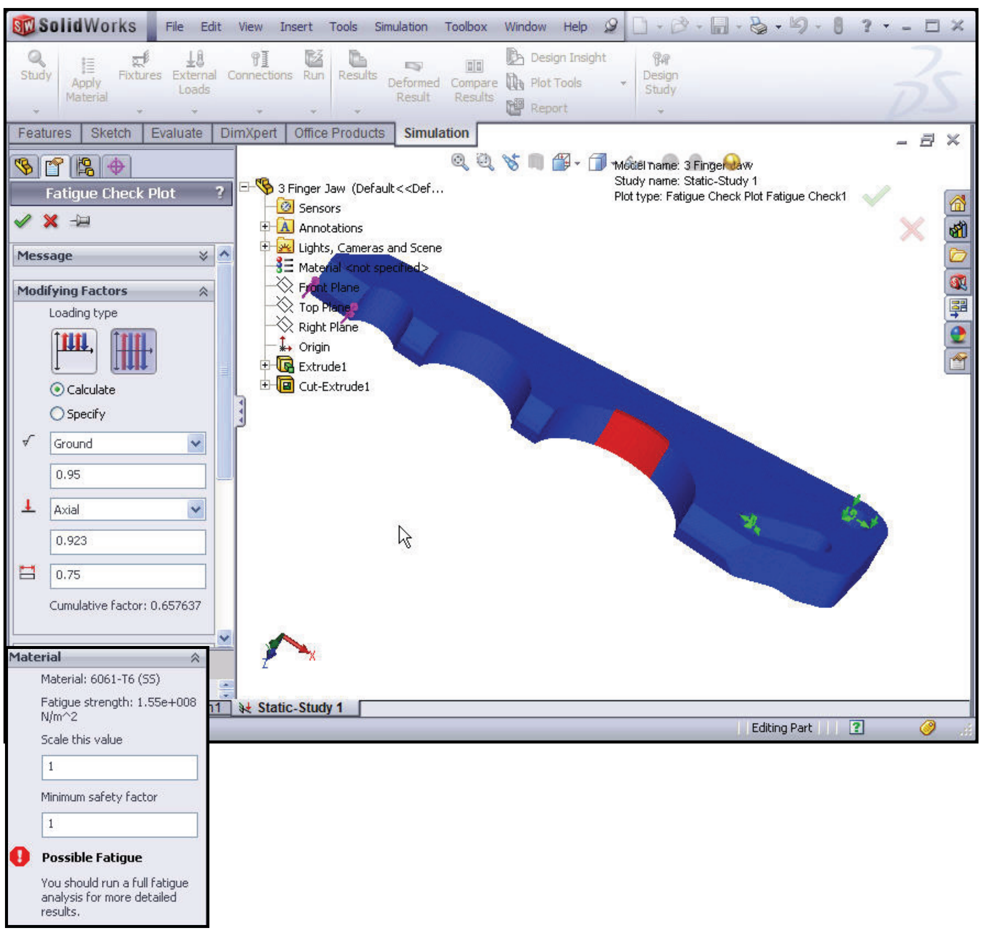


Perform a Fatigue Check Plot.



1 **Create a New Fatigue Study.**
- Right-click the **Results** folder.
- Click **Define Fatigue Check Plot**. The Fatigue Check Plot PropertyManager is displayed.

2 **View the Fatigue Check Plot.**
- Click the **Fully Reversing Load** button. View the results in the Graphics area. There is a possible Fatigue issue.
- Click **Cancel** from the Fatigue Check Plot PropertyManager.



Creating a New Fatigue Study.

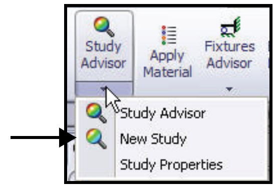
1 Create a New Fatigue Study.

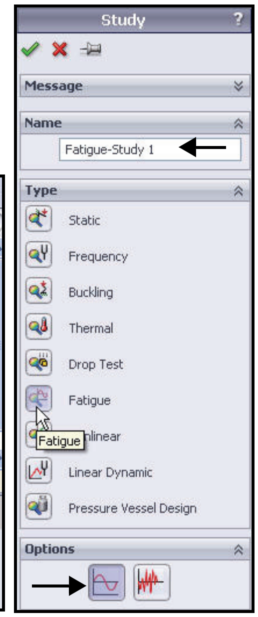
- Click the **Study Advisor** drop-down arrow from the Simulation tab in the CommandManager.
- Click **New Study**. The Study PropertyManager is displayed.
- Enter **Fatigue-Study 1** for Name.
- Check **Fatigue** for Type.

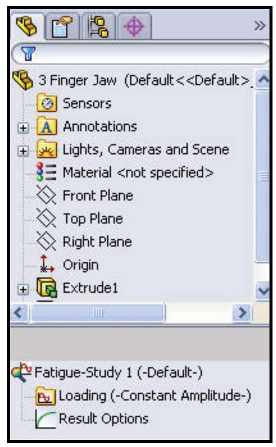
2 Display the Study.

- Click **OK** from the Study PropertyManager. View Fatigue-Study 1 (-Default-).

Note: The Fatigue-Study 1 tab is displayed in the bottom corner of the Graphics area.

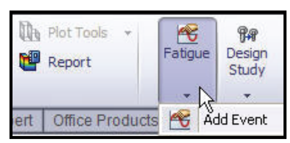


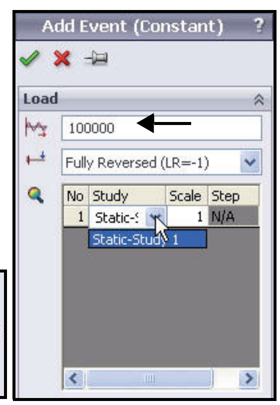


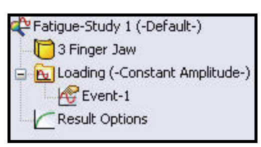


3 View Loading Event.

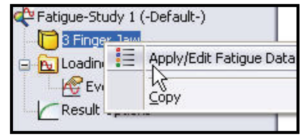
- Click the **Fatigue** drop-down arrow from the Simulation tab in the CommandManager.
- Click **Add Event**. The Add Event (Constant) PropertyManager is displayed.
- Select **Static-Study 1** from the drop-down menu.
- Enter **100000** cycles box.
- Click **OK** from the Add Event (Constant) PropertyManager.
- Click **3Finger Jaw** in the Study tree. View the results.



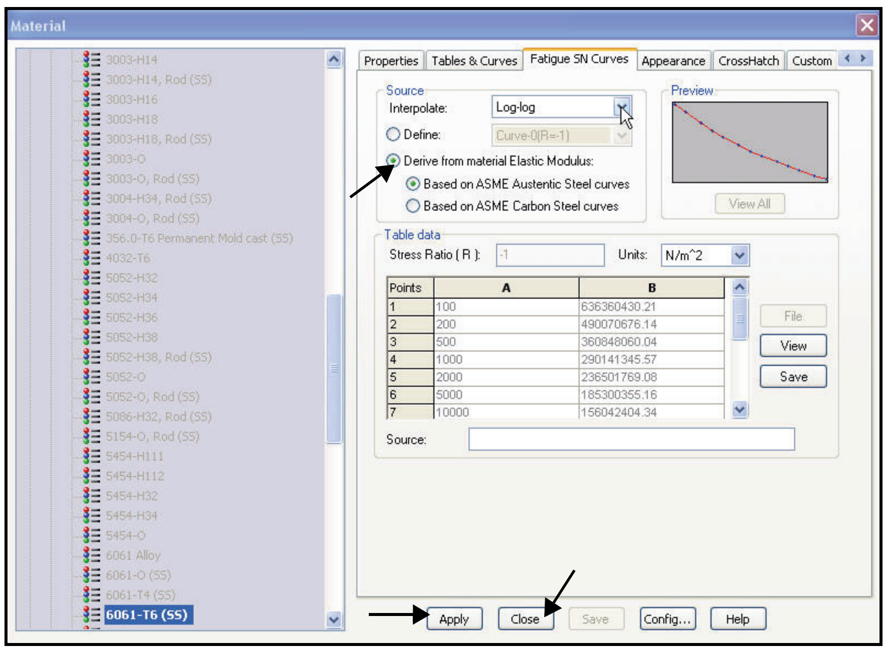




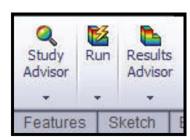
4 Edit the Fatigue data.



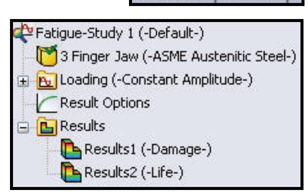
- Right-click **3 Finger Jaw**.
- Click **Apply/Edit Fatigue Data**. The Material dialog box is displayed.
- Check the **Derive from material Elastic Modulus** box.
- Select **Log-log** from the Source area.
- Click **Apply**.
- Click **Close**. View the results.



5 Run the Study.



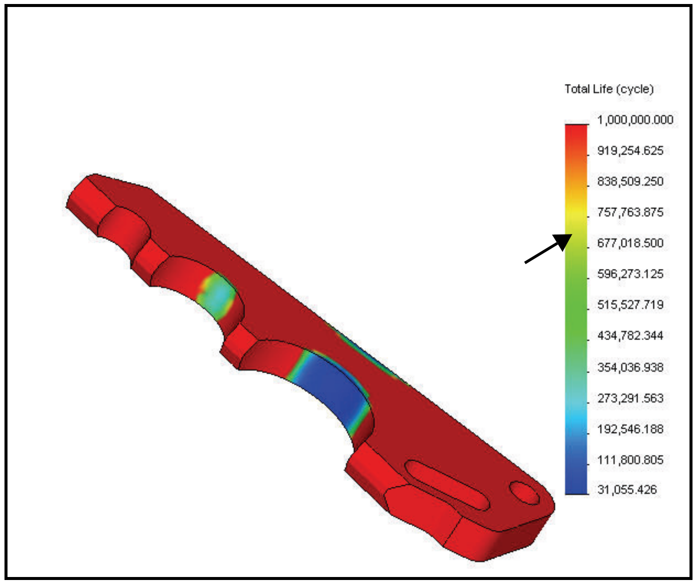
- Click **Run**  from the Simulation CommandManager. View the Results folder.



Note: 100,000 cycles represents approximately 100 cycles/dive x 100 dives/year x 10-year life expectancy of the unit.

6 View the Life Plot.

- Double click on the **Results2 (-Life-)** folder. The Life plot is displayed.

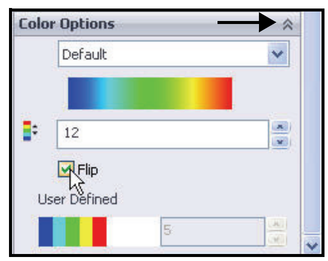


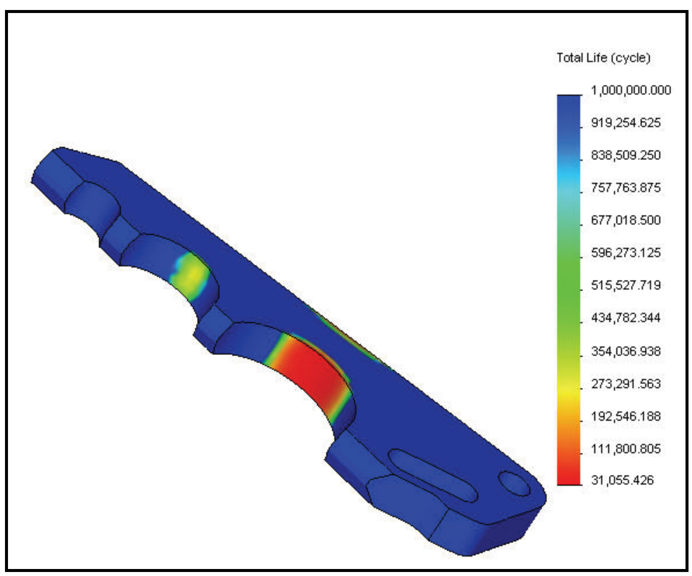
7 Display the Chart Options PropertyManager.

- Double-click on the **Life plot** in the Graphics areas as illustrated. The Chart Options PropertyManager is displayed.

8 Reverse the Life plot results color.

- Expand the **Color Options** box.
- Click the **Flip** box.
- Click **OK** from the Chart Options PropertyManager. View the results in the Graphics area.

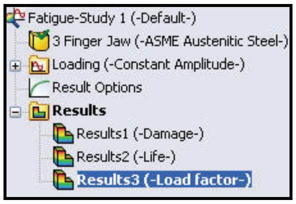


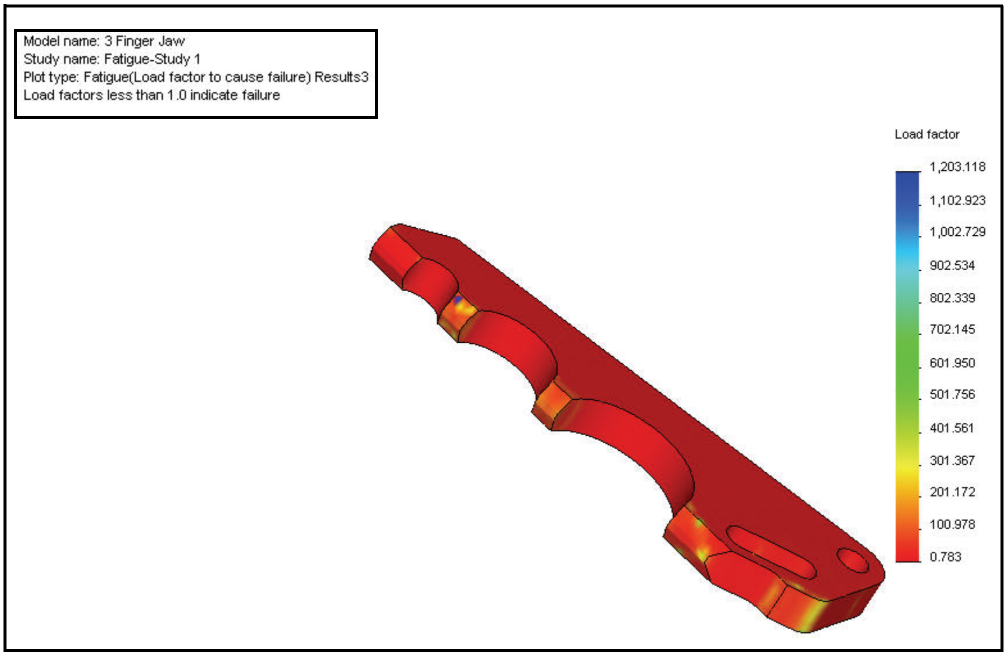


Applying a Load Factor

1 **Apply a Load Factor.**
- Right-click the **Results** folder.
- Click **Define Fatigue Plot**. The Fatigue Plot PropertyManager is displayed.
- Check the **Load Factor** box.
- Click **OK** from the Fatigue Plot PropertyManager. View the Results folder.

2 **Save and Close the Model.**
- Click **Window**, **Close All** from the Menu bar menu.





SolidWorks Simulation Professional Conclusion

In your short time today, you have seen firsthand the functionality of SolidWorks Simulation Professional applications. In addition to the design validation functionality contained in SolidWorks Simulation, SolidWorks Simulation Professional offers expanded analysis capabilities including: Thermal, Frequency, Buckling, Optimization, Fatigue, and Drop Test Simulation.

Understand the effects of temperature changes. Temperature variations encountered by mechanical parts and structures can greatly influence the performance of your designs.

Evaluate natural frequencies or critical buckling loads and their corresponding mode shapes. Often overlooked, inherent vibration modes in structural components or mechanical support systems can shorten the life of your product and cause unexpected failures.

Optimize designs based on your defined criteria. Design optimization automatically determines the optimal design based on your specified criteria.

Simulate virtual drop tests on a variety of surfaces. In the event that your part or assembly might be dropped, find out whether or not it can survive the fall intact.

Study the effects of cyclic loading and fatigue operation conditions. See the effects of fatigue on the overall lifecycle of your part or assembly to find out how long it will last and what design changes can extend its working life.

SolidWorks Flow Simulation

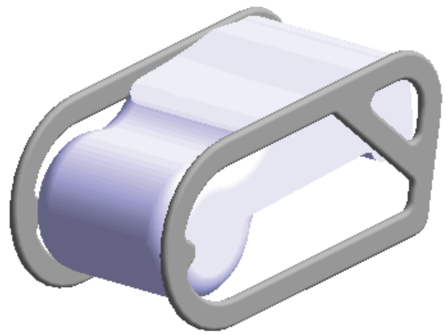
When you complete this chapter, you will have experienced the power and capabilities of SolidWorks Flow Simulation, including:

- The benefits of using fluid-flow analysis.
- The ease of use of SolidWorks Flow Simulation to perform analysis on your design.
- The steps for performing upfront analysis on your designs.
- The integration between SolidWorks Flow Simulation and SolidWorks.
- The results of cost reduction with virtual prototypes to save resources.
- The ability to document your analysis findings automatically.

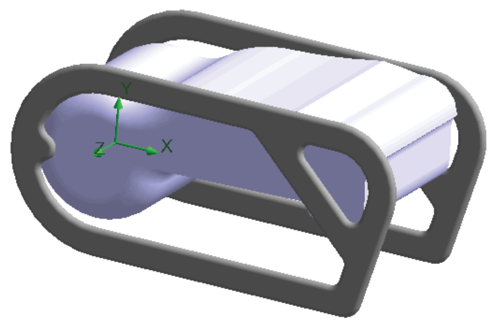
Time: 20 - 25 minutes

SolidWorks Flow Simulation

SolidWorks Flow Simulation is the first easy-to-use fluid-flow simulation and thermal analysis program that is fully embedded inside SolidWorks. You will utilize SolidWorks Flow Simulation to understand, validate, and improve new product ideas during the design phase.

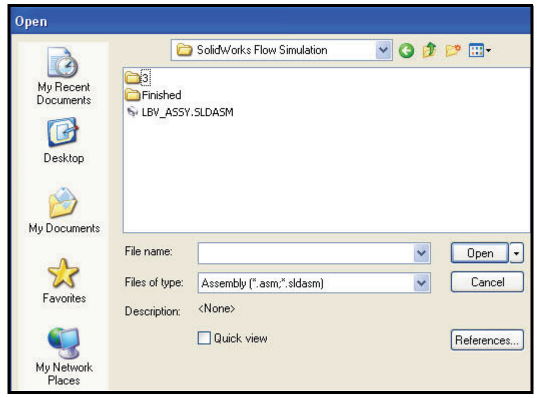


SolidWorks Flow Simulation provides the user insight into parts or assemblies related to fluid flow, heat transfer, and forces on immersed or surrounding solids.



You will use the SolidWorks Flow Simulation Wizard to analyze the drag created by the SeaBotix LBV150 assembly as it moves through seawater. This information is critical to choose the correct size thruster required for the assembly to perform its tasks.

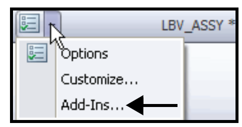
Starting a SolidWorks Flow Simulation Session



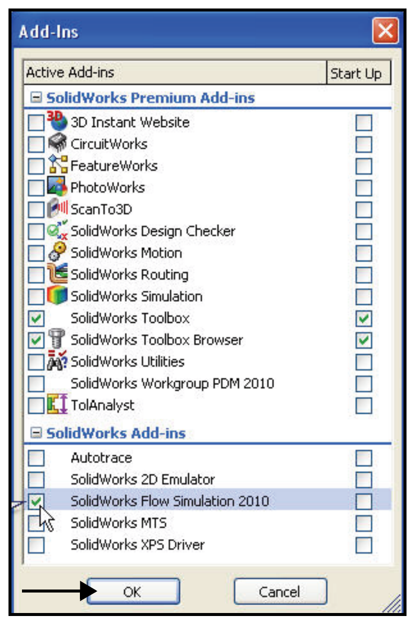
1 Open the SeaBotix LBV150 Assembly.

- Click **Open** from the Menu bar toolbar.
- Double-click **LBV_ASSY** from the SeaBotix\SolidWorks Flow Simulation folder. A simplified model opens in the Graphics area.

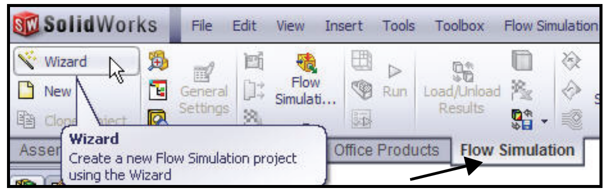
2 Activate SolidWorks Simulation Flow Simulation module.



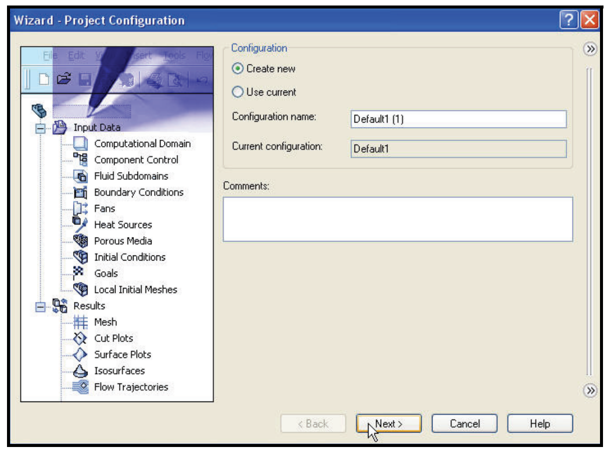
- Click the **Options** drop-down arrow from the Menu bar toolbar as illustrated.
- Click **Add-Ins**. The Add-Ins dialog box is displayed.
- Check the **SolidWorks Flow Simulation 2010** box.
- Click **OK** from the Add-Ins dialog box. The Flow Simulation tab is displayed in the CommandManager.



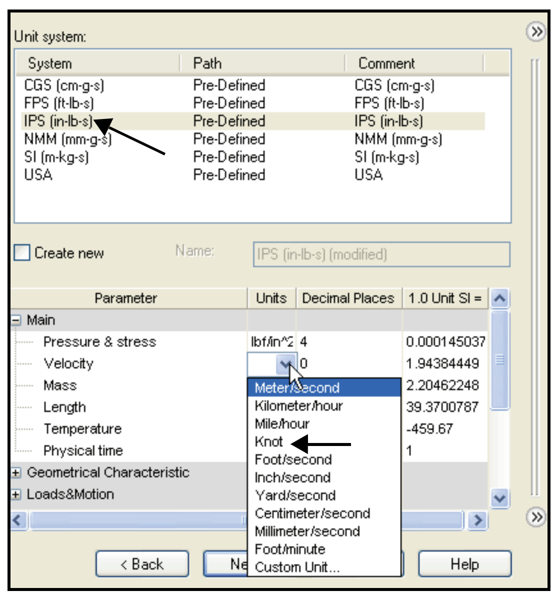
3 Start the SolidWorks Flow Simulation Wizard.



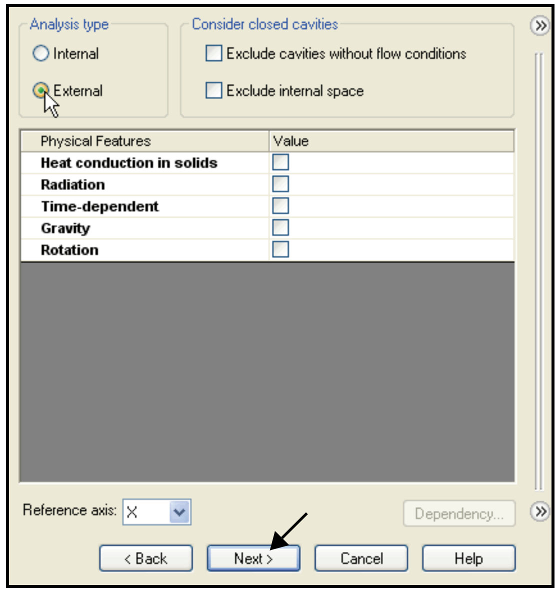
- Click the **Flow Simulation** tab located in the CommandManager.
- Click the **Wizard** tool. The Wizard - Project Configuration box is displayed. Create new is selected by default. Accept the default settings.
- Click **Next>**. The Wizard Unit System dialog box is displayed.



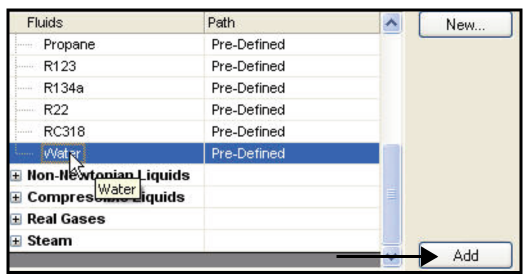
- Click **IPS (in-lb-s)** for Unit system.
- Click inside the **Velocity Unit** box.
- Click **Knot** from the drop-down menu as illustrated.
- Click **Next>**. The Wizard - Analysis Type dialog box is displayed.

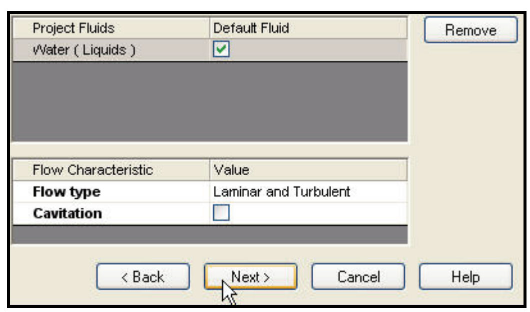


- Click the **External** box for Analysis type.
- Click **Next>**. The Wizard - Default Fluid dialog box is displayed.

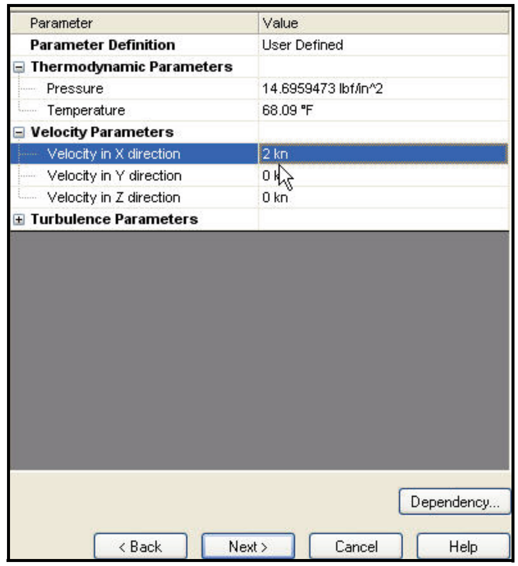


- Expand the **Liquids** folder.
- Click **Water**.
- Click the **Add** button. Water is displayed in the Project Fluids box.
- Click **Next>**. The Wizard - Wall Conditions dialog box is displayed. Accept the default settings.
- Click **Next>**. The Wizard - Initial and Ambient Conditions dialog box is displayed.





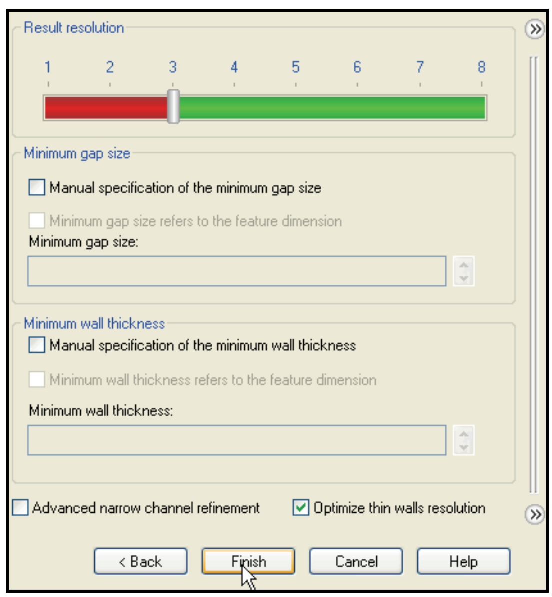
- Double-click inside the Value box of **Velocity in X direction** as illustrated.
- Enter **2** Kn for Velocity.
- Click **Next>**. The Wizard - Results and Geometry Resolution dialog box is displayed.



Note: Two knots is the operating speed.

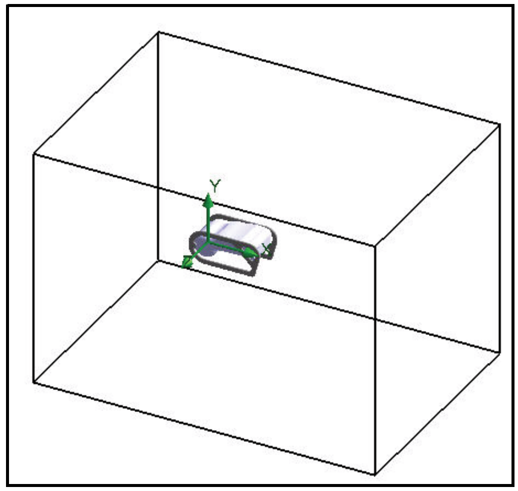
4 Complete the SolidWorks Flow Simulation Wizard.

- Accept all default settings. Click the **Finish** button.



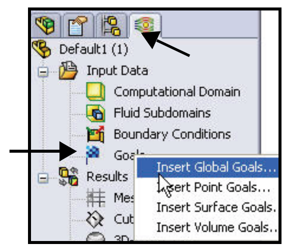
5 View the Simulation.

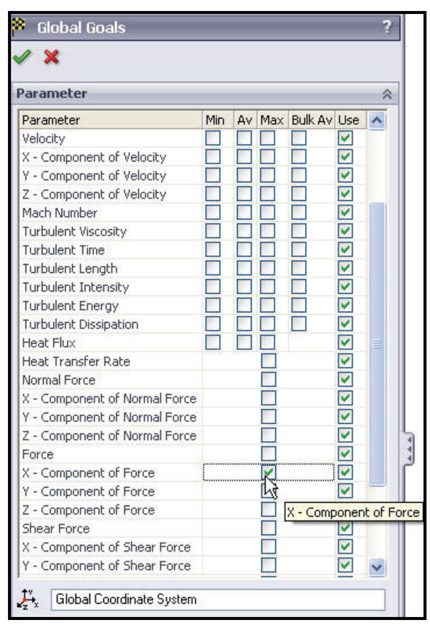
- Press the **z** key three or four times to Zoom out to view the model. The surrounding box simulates the seawater around the assembly.



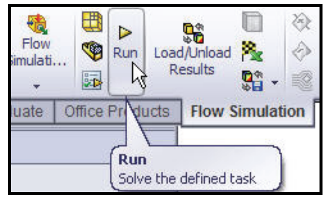
6 Analyze the Drag.

- Click the **Flow Simulation analysis tree** tab.
- Expand the **Input Data** folder.
- Right-click **Goals**.
- Click **Insert Global Goals**. The Global Goals PropertyManager is displayed.
- Scroll down and check the **Max** box in **X-Component of Force**.
- Click **OK** from the Global Goals PropertyManager.

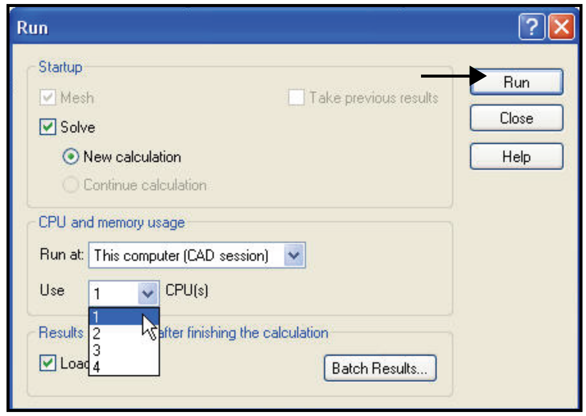




7 Run the Analysis.



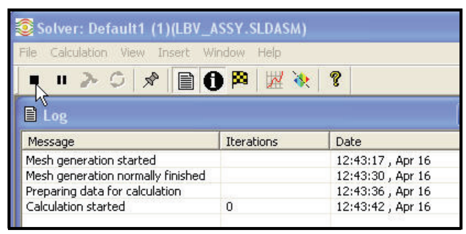
- Click **Run** from the Flow Simulation tab in the CommandManager. The Run dialog box is displayed.
- Select **1 CPU** from the drop-down menu,
- Click the **Run** button.



Note: To save classroom time, we will stop the analysis and open the Results folder to review completed results.

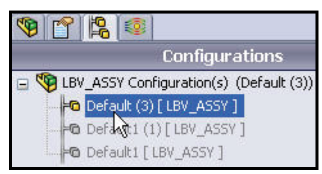
8 Stop the Analysis.

- Click **Stop** from the Solver box as illustrated.
- Click **No** to the question, “Do you want to save the results?”
- Click **File**, **Close** from the Solver Main menu.

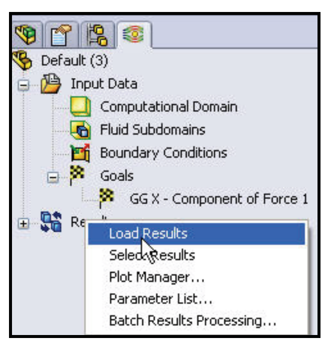


9 Open the Configuration with the Solved Results.

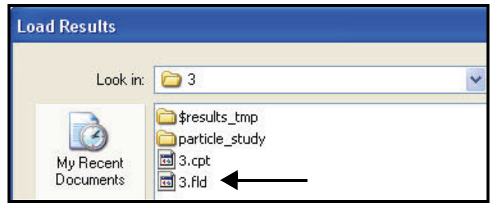
- Click the **ConfigurationManager** tab.
- Double-click the **Default (3)** configuration as illustrated.



- Click the **Flow Simulation analysis tree** tab.
- Right-click the **Results** folder.
- Click **Load Results**. The Load Results dialog box is displayed.

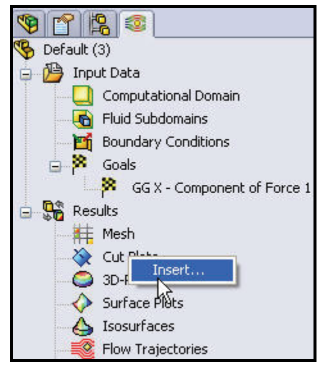


- Double-click **3.fld** in folder 3.

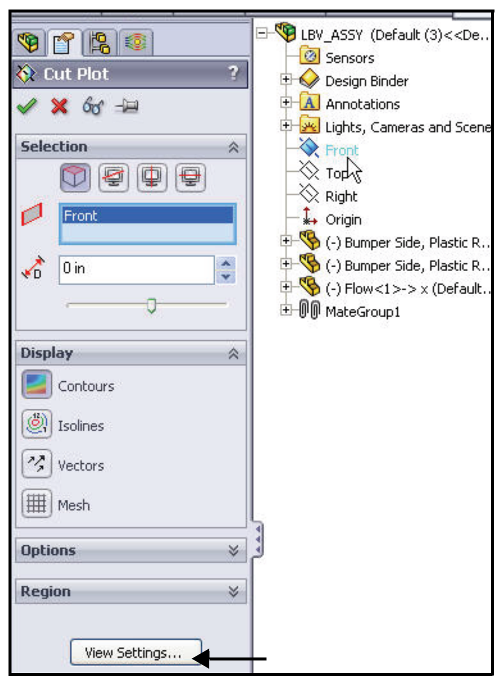


10 Create a Section Plot.

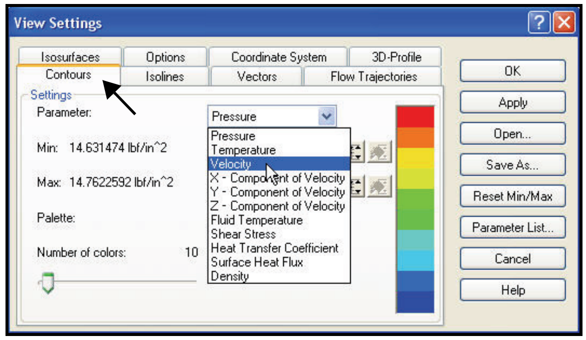
- Expand the **Results** folder.
- Right-click the **Cut Plots** folder.
- Click **Insert**. The Cut Plots PropertyManager is displayed. Front Plane is selected by default.



- Expand **LBV_ASSY** from the fly-out FeatureManager. View the features.
- Click the **View Settings** button in the Cut Plot PropertyManager. The View Settings dialog box is displayed.

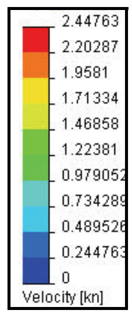


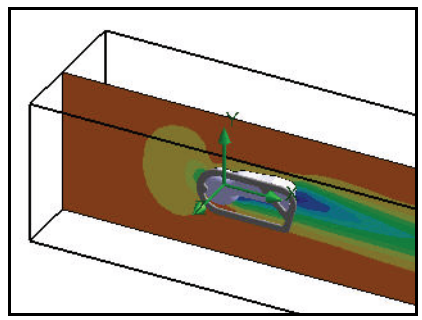
- Click the **Contours** tab.
- Select **Velocity** from the drop-down menu for the Parameter Setting.
- Click **OK** from the View Settings dialog box.



11 View the Section Plot.

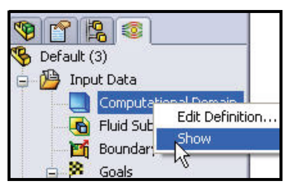
- Click **OK** from the Cut Plot PropertyManager. View the Section plot in the Graphics area.





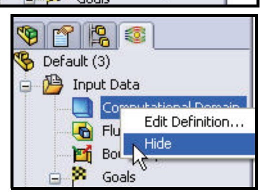
12 View the Computational Domain.

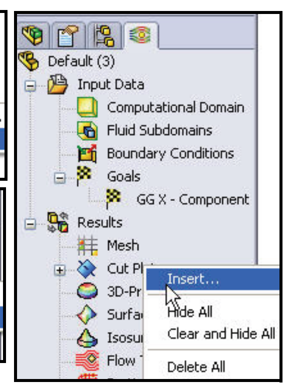
- If needed, right-click the **Computational Domain** folder.
- Click **Show**. View the domain.



13 Hide the Computational Domain.

- Right-click the **Computational Domain** folder.
- Click **Hide**.



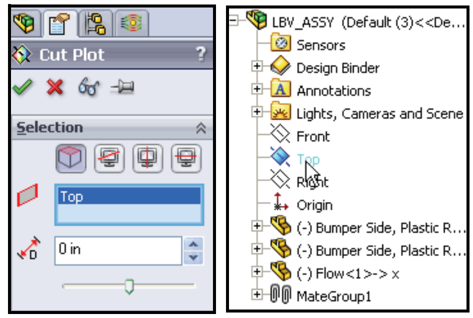


14 Create a Second Cut Plot.

- Right-click the **Cut Plots** folder.
- Click **Insert**. Front Plane is selected by default.

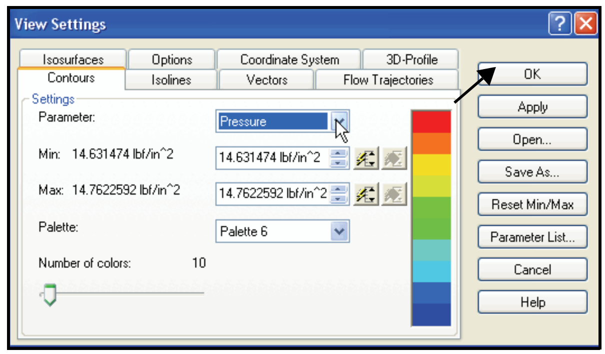
15 Change the Selected Plane.

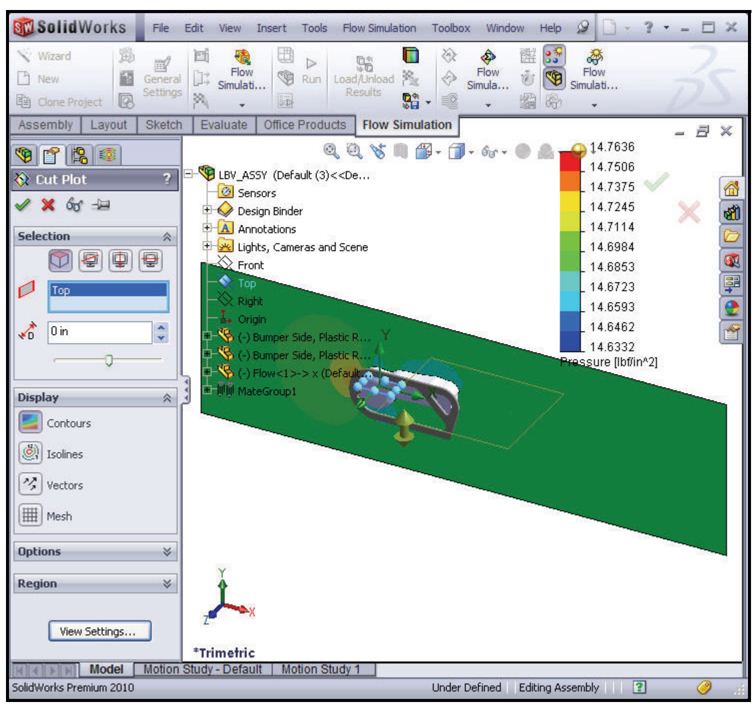
- Expand **LBV_Assy** from the fly-out FeatureManager.
- Click **Top** Plane from the fly-out FeatureManager. Top is displayed in the Selection plane/face box.



16 Continue the Second Cut Plot.

- Click the **View Settings** button.
- Click the **Contours** tab.
- Select **Pressure** from the drop-down menu for Parameter Setting.
- Click **OK** from the View Settings dialog box. View the results in the Graphics area.

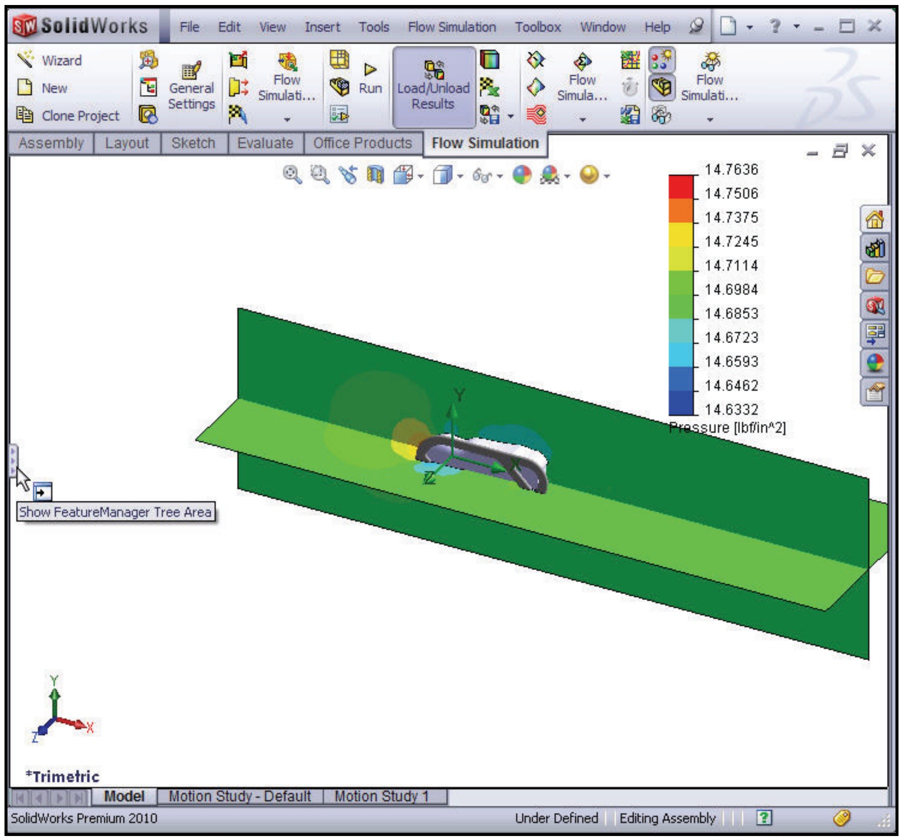




17 View the Second Section Plot.

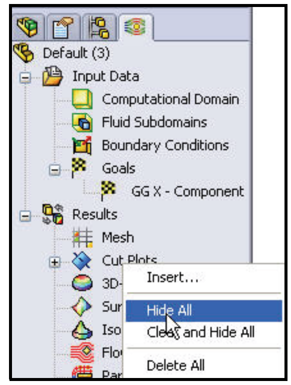
- Click **OK** from the Cut Plot PropertyManager.

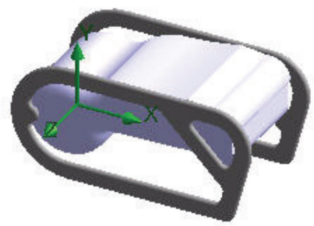
Note: Click the **FeatureManager tree** tab as illustrated to view the full Graphics area.

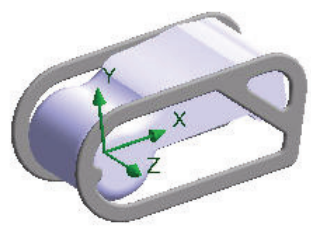


18 Hide the Section Plots.

- Right-click the **Cut Plots** folder.
- Click **Hide All**. View the model in the Graphics area.

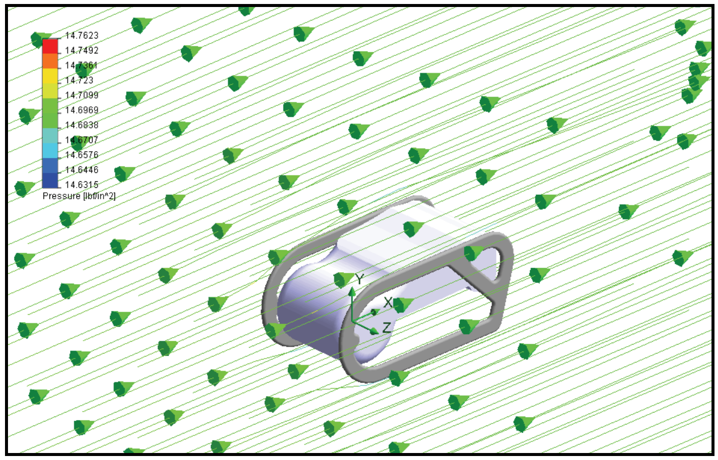


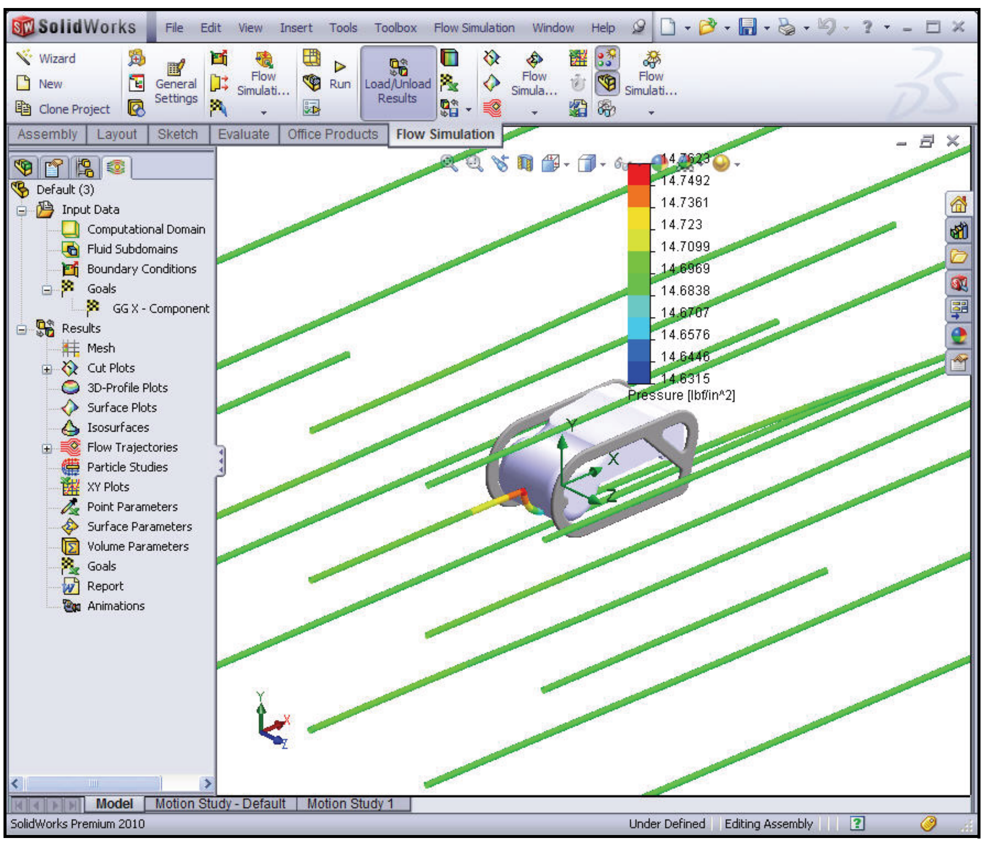




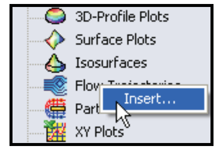
Applying Flow Trajectories

Flow trajectories are displayed as flow streamlines. Flow streamlines are curves where the flow velocity vector is tangent to that curve at any point on the curve.



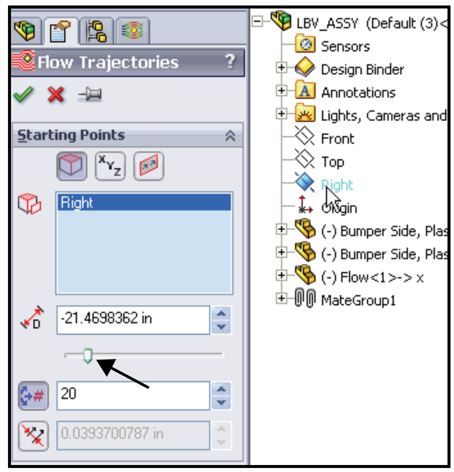


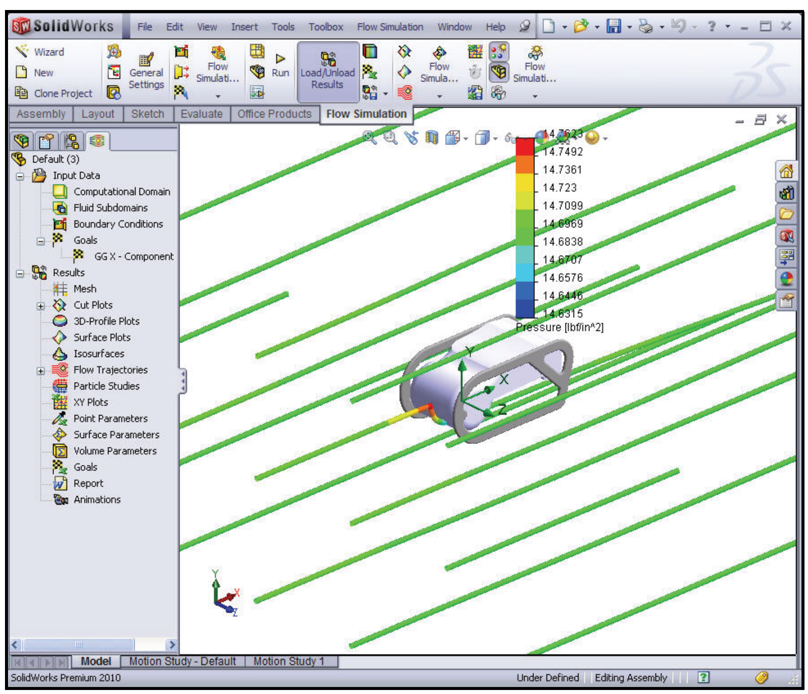
Applying Flow Trajectories



1 **Create a Flow Trajectory.**

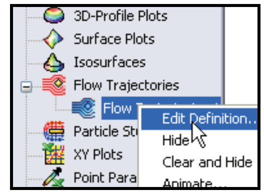
- Right-click the **Flow Trajectories** folder.
- Click **Insert**. The Flow Trajectories PropertyManager is displayed.
- Expand **LBV_Assy** the fly-out FeatureManager.
- Click **Right** Plane. Right is displayed in the Reference box.
- Slide the **Offset slider** as illustrated to approximately -21.
- Click **OK** from the Flow Trajectories PropertyManager. Flow Trajectories 1 is displayed.
- **Zoom-out** and **rotate** the model to view the plot.



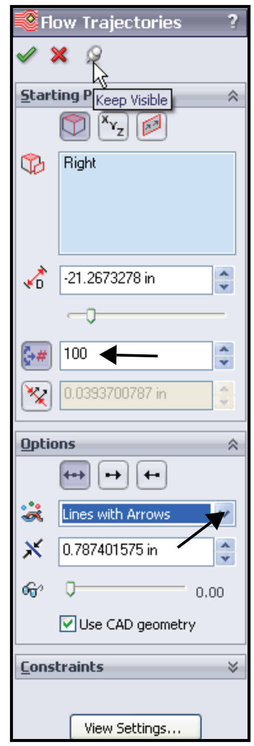


2 Edit the Flow Trajectory.

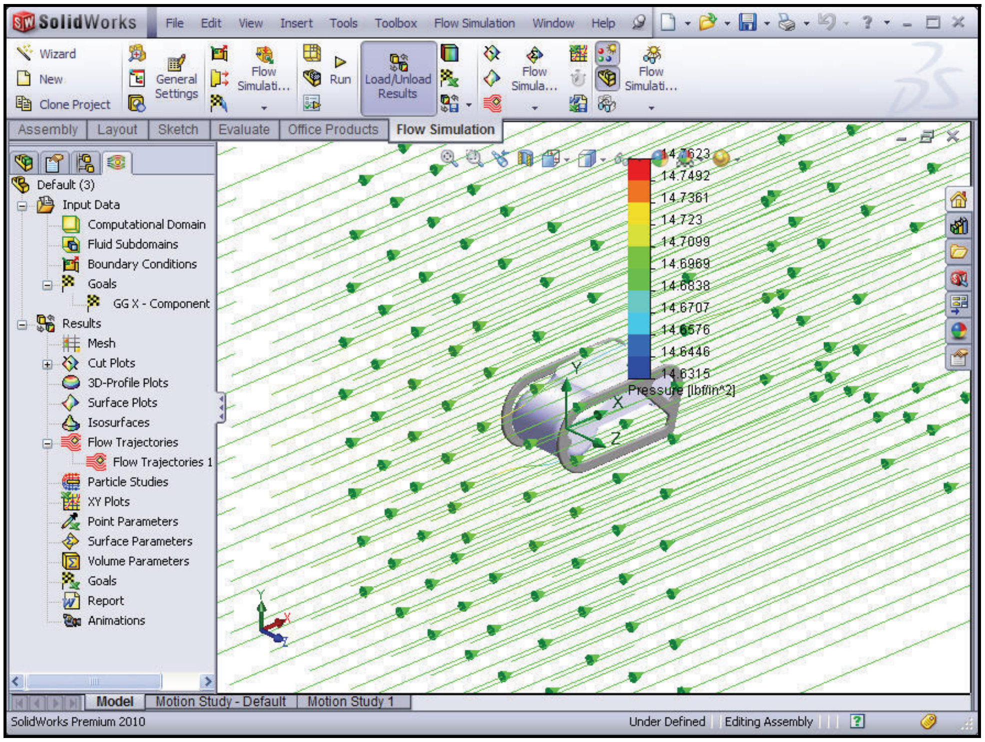
- Expand the **Flow Trajectories** folder.
- Right-click **Flow Trajectories 1**.
- Click **Edit Definition**. The Flow Trajectories PropertyManager is displayed.



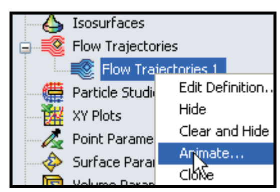
- **Pin** the Flow Trajectories PropertyManager.
- Enter **100** for the Number of trajectories as illustrated.



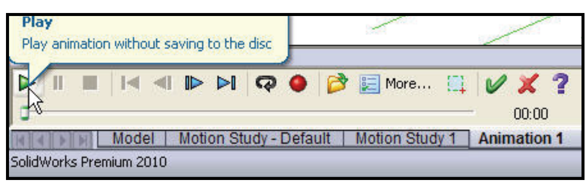
- Click **OK** from the Flow Trajectories PropertyManager. View the model.
- Click **Lines with Arrows** from the drop-down menu in the Options box.
- Click **OK** from the Flow Trajectories PropertyManager. View the model.
- **Un-Pin** the Flow Trajectories PropertyManager.
- Click **OK** from the Flow Trajectories PropertyManager. View the plot. If needed, click the **FeatureManager tree** tab to hide or click and drag the Pressure bar.



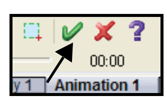
3 Animate the Flow Trajectory study.



- Right-click the **Flow Trajectory 1** folder.
- Click **Animate**. The Animation 1 tab is displayed at the bottom of the Graphics area.

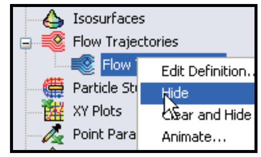


- Click **Play**. View the animation of the model.
- Click **OK** from the Animation toolbar to return to the FeatureManager.



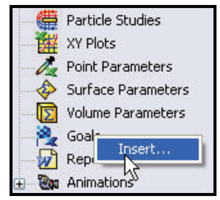
4 Edit the Flow Trajectory.

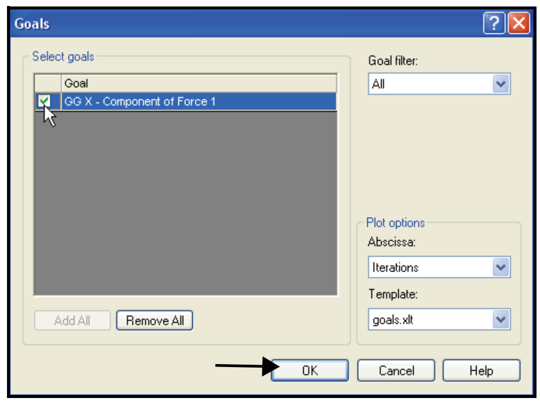
- If needed, right-click the **Flow Trajectories 1** folder.
- Click **Hide**. View the Graphics area.

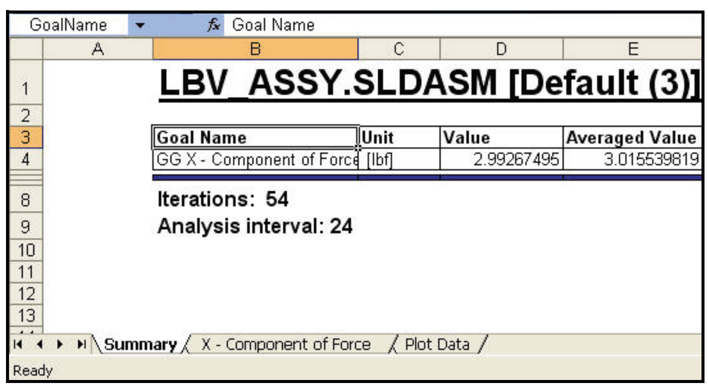


5 Set the Goals.

- Expand the **Results** folder.
- Right-click the **Goals** folder as illustrated.
- Click **Insert**. The Goals dialog box is displayed.
- Check the **GGX-Component of Force1** box as illustrated.
- Click **OK** from the Goals dialog box. The Goals dialog box is displayed. View your options.

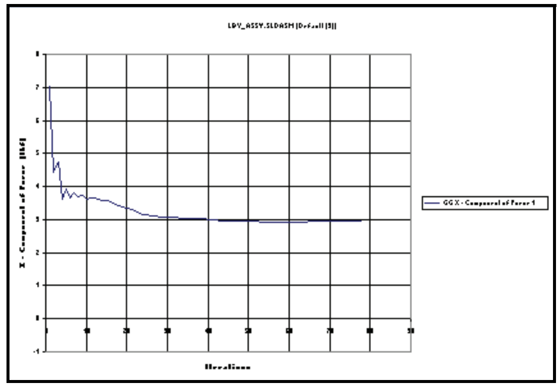






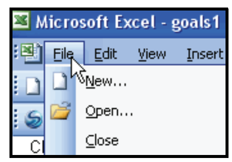
6 View the Excel Plot.

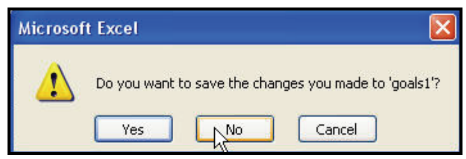
- Click the bottom **X - Component of Force** tab.
- **View** the plot.



7 Close the Excel Plot and return to SolidWorks Flow Simulation.

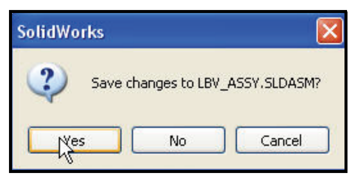
- Click **File**, **Exit** from the Excel menu bar.
- Select **No** when prompted to Save.



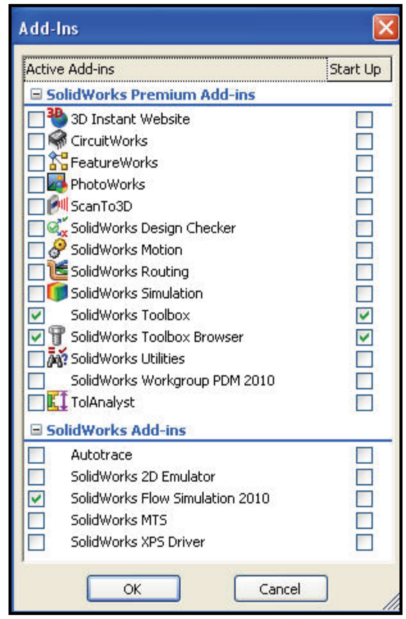


8 Save and Close the model.

- Click **File**, **Close** from the SolidWorks Main menu.
- Click **Yes** when prompted to save.



SolidWorks Flow Simulation



During this short session on using SolidWorks Flow Simulation, you have had a brief exposure to the main concepts of fluid-flow simulation. SolidWorks Flow Simulation gives you insight into parts and assemblies related to fluid flow, heat transfer, and forces on immersed or surrounded solids.

The only fluid-flow simulation product fully integrated with SolidWorks, SolidWorks Flow Simulation is incredibly easy to use; you simply tell the software what you're interested in instead of having to translate analysis design goals into numerical criteria and iteration numbers.

Access physical fluid models for engineering applications. SolidWorks Flow Simulation can analyze a wide range of real fluids such as air, water, juice, ice cream, honey, plastic melts, toothpaste, and blood, which makes it ideal for engineers in nearly every industry.

Simulate real-world operating conditions. SolidWorks Flow Simulation includes several types of boundary conditions to represent real-life situations.

Automate fluid-flow tasks. SolidWorks Flow Simulation utilizes a number of automation tools to simplify the analysis process and help you to work more efficiently.

Interpret results with powerful and intuitive visualization tools. Once you have completed your analysis, SolidWorks Flow Simulation offers a variety of results visualization tools that allow you to gain valuable insight into the performance of your model.

Collaborate and share analysis results. SolidWorks Flow Simulation makes it easy to collaborate and share analysis results effectively with everyone involved in the product development process.

SolidWorks Motion

When you complete this chapter, you will have experienced the power and capabilities of SolidWorks® Motion, including:

- The benefits of using motion analysis.
- The ease of use of SolidWorks® Motion to perform analysis on your design.
- The steps for performing a motion simulation on your designs.
- The integration between SolidWorks Motion and SolidWorks.
- An understanding of the performance aspects and time savings before physical prototyping.

Time: 20 - 25 minutes

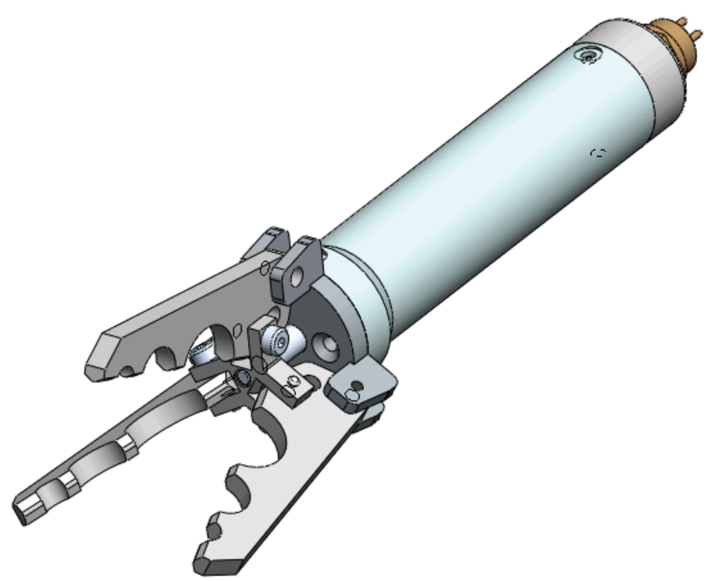
SolidWorks Motion

SolidWorks® Motion is designed for mechanical system simulation and ensures that a mechanism works before it is built.

SolidWorks Motion will:

- Provide confidence that your assembly performs as expected without parts colliding while they move.
- Increase the efficiency of your mechanical design process by providing mechanical system simulation capability within the familiar SolidWorks environment.
- Use a single model, without transferring geometry and other data from application to application.
- Eliminate the expense caused by design changes late in the manufacturing process.
- Speed the design process by reducing costly design change iterations.

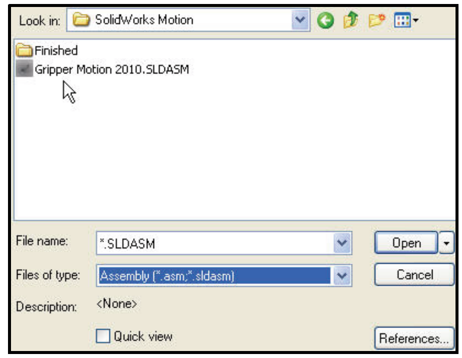
Today, perform an analysis on the Gripper assembly.



Starting a SolidWorks Motion Session

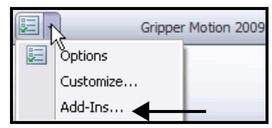
1 Open the Gripper Assembly.

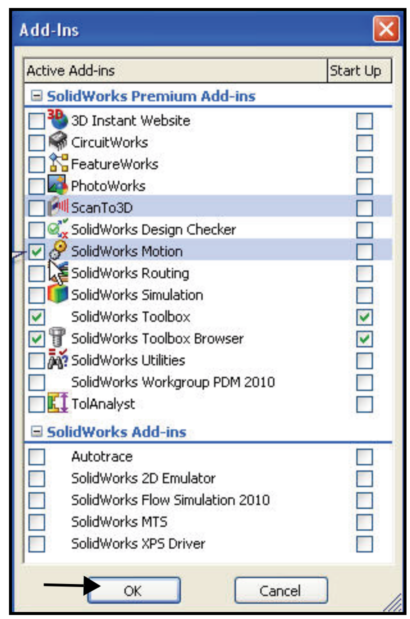
- Click **Open** from the Menu bar menu.
- Double-click the **Gripper Motion 2010** assembly from the SeaBotix\SolidWorks Motion folder.

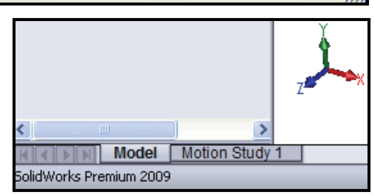


2 Activate SolidWorks Motion.

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

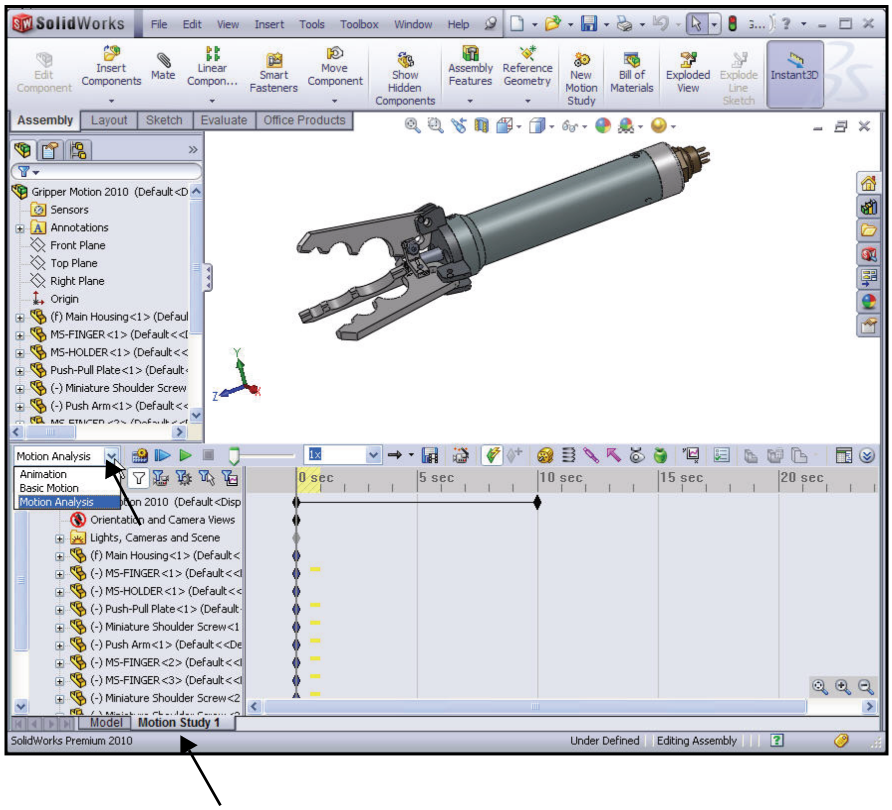




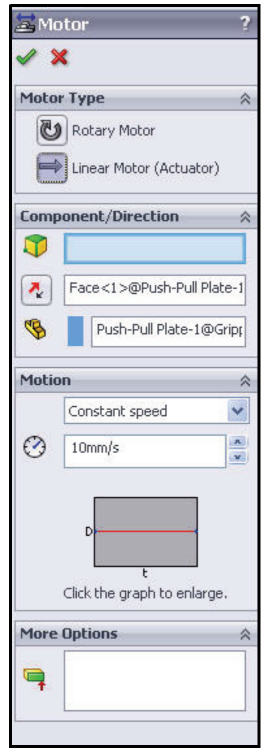


3 Start a SolidWorks Motion Study.

- Click the **Motion Study 1** tab at the bottom of the Graphics area as illustrated.
- Click the **drop-down arrow** from the Motion Study Manager.
- Select **Motion Analysis**. View the available selections from the Motion Study Manager.

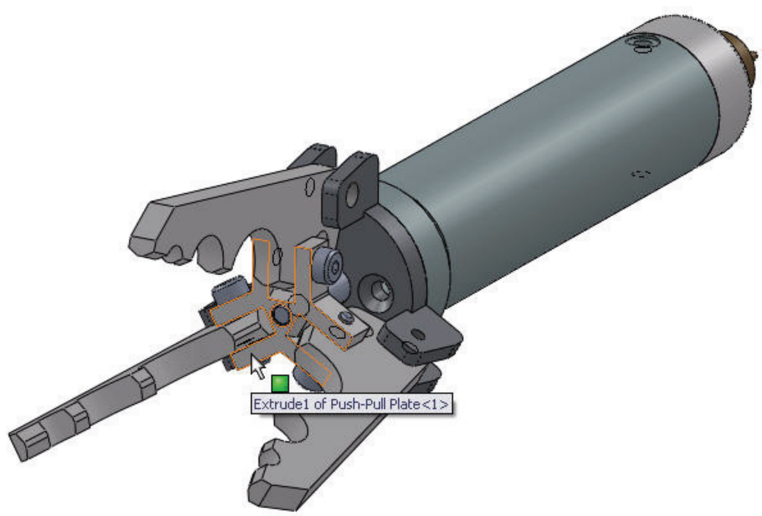


Applying Motion to a Component



A linear motor (actuator) is a device which imparts a translational motion to a component. A linear motor in SolidWorks Motion moves the selected component at a constant speed or variable speed.

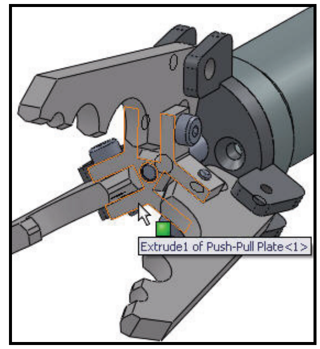
Apply a linear motor to the Push-Pull Plate component in the Gripper assembly. The linear motor will move the Push-Pull Plate component a specified distance in a specified time. This action will cause the fingers of the Gripper assembly to close.



Applying Linear Motion

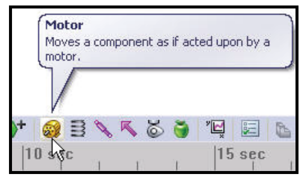
4 Apply Linear Motor.

- Zoom in on the **Push-Pull Plate** component.
- Click the **Push-Pull Plate** component face of the Gripper assembly as illustrated.

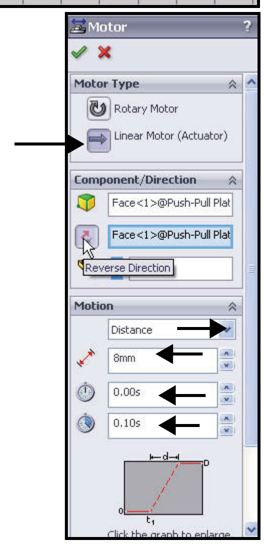


Note: View the icon symbol and information feedback.

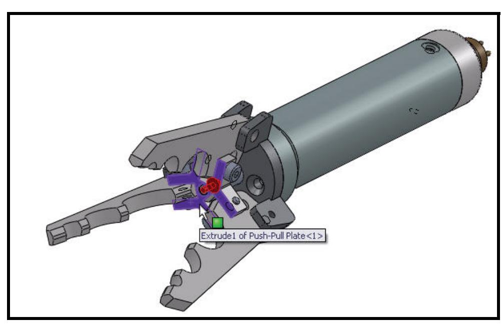
- Click the **Motor** icon from the Motion Manager toolbar. The Motor PropertyManager is displayed.



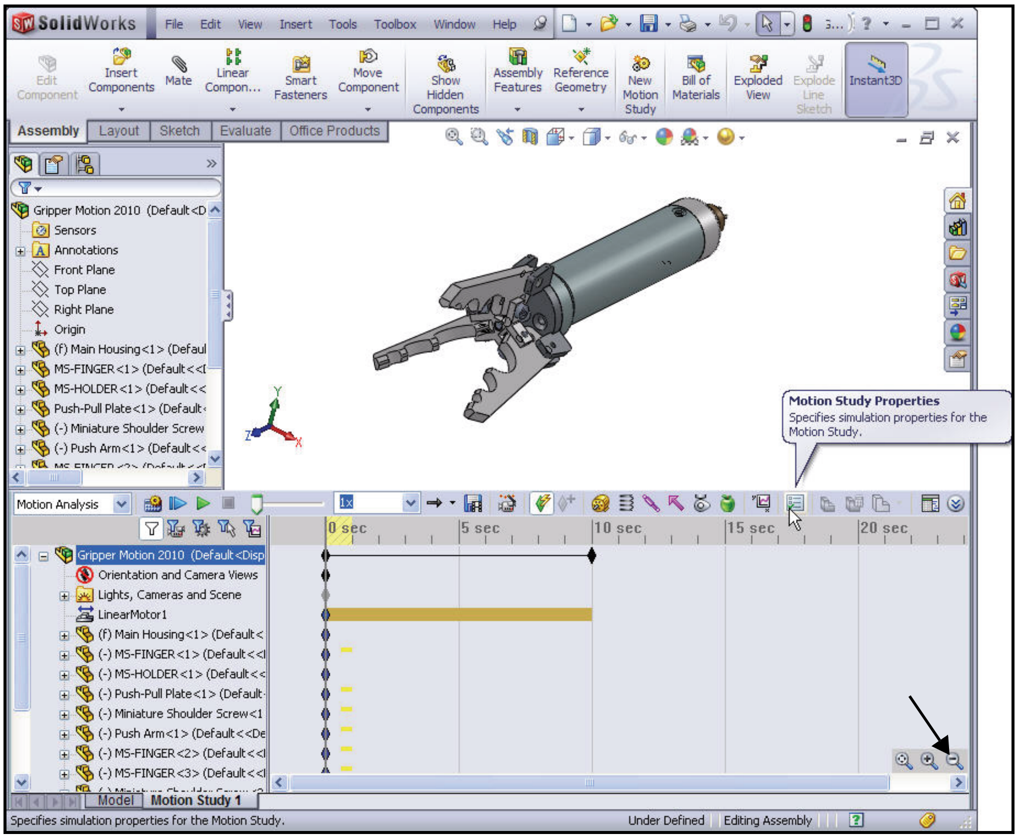
- Click the **Linear Motor (Actuator)** box for Motor Type.
- Click the **Reverse Direction** button. The direction arrow points inwards.
- Select **Distance** for the drop-down menu for Motion Type.
- Enter **8**mm in the Displacement motor box.
- Enter **0** in the Start time box.
- Enter **.1** in the Duration time box.
- Click the **Push-Pull Plate** component face of the Gripper assembly as illustrated for Motor direction. The direction arrow points towards the back.



- Click **OK** from the Motor PropertyManager. LinearMotor1 is displayed in the Motion Study FeatureManager.



- If needed, click the **Zoom Out** tool as illustrated to view the Motion Study time line.
- Click the **Motion Study Properties** tool as illustrated. View your options. Accept the default settings.
- Click **OK** from the Motion Study Properties PropertyManager.

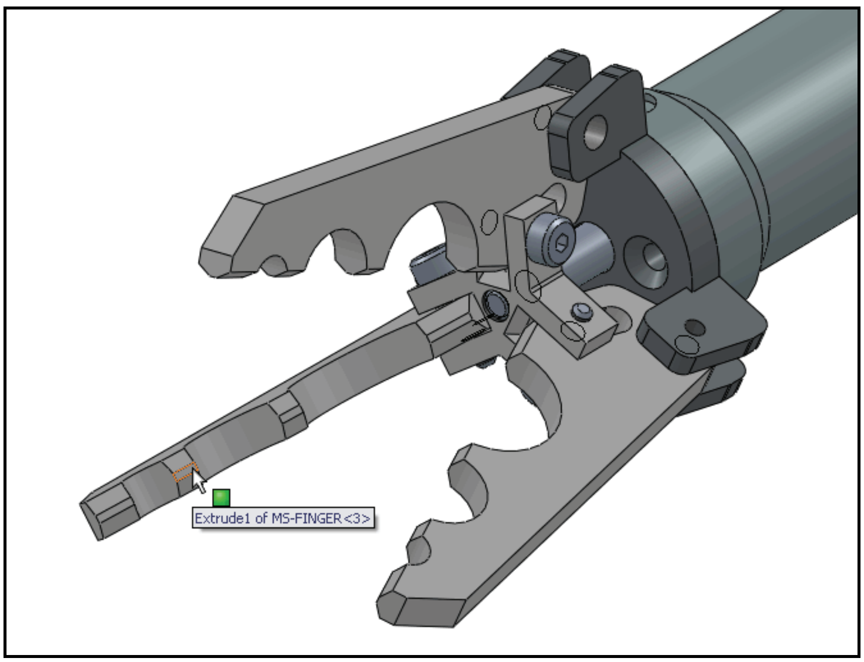


Applying Forces

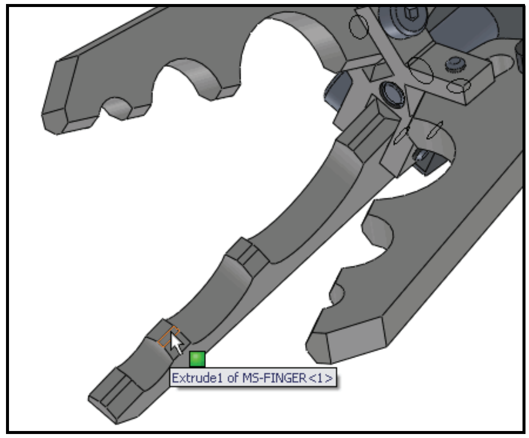
Forces define loads and compliances on parts. Forces may resist motion, such as springs or dampers, or they may induce motion.

The 3 Finger Jaw components experience an applied force. To simulate the loading conditions, you will perform the following tasks:

- Select the middle contact surface from one of the 3 fingers.
- Insert an applied action-only force of 62 N to the selected finger.
- Repeat the process on the other two fingers.
- Create and run a simulation.
- Compute the reaction force at the finger hinge.
- Create a trace path for the tip of one finger.



Applying Force to the Gripper Fingers

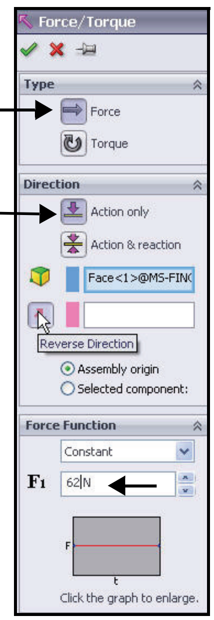


5 Select a Contact Face.

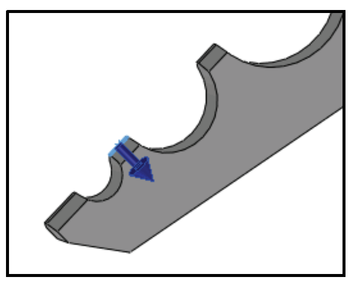
- **Rotate** the Gripper assembly with the middle mouse button to view the inside faces of a finger as illustrated.
- Zoom in to selected the **first contact face**.

Note: Select any of the 3 Gripper fingers.

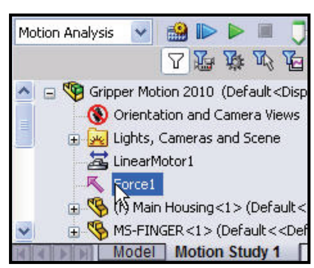
- Click the **contact finger face** as illustrated.



6 Apply the Force.

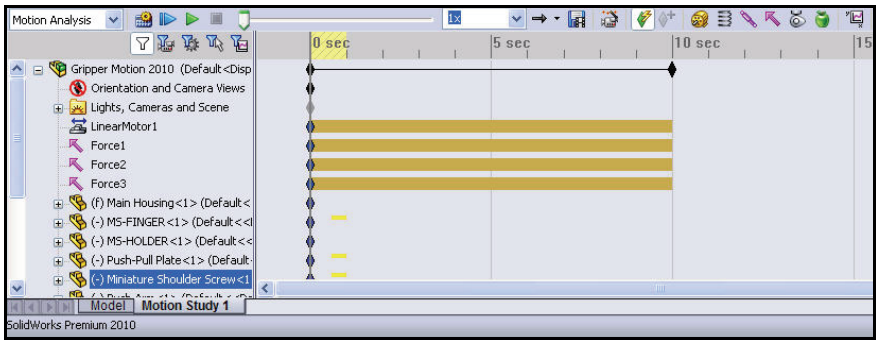


- Click the **Force** icon from the Motion Manager toolbar. The Force/Torque PropertyManager is displayed.
- Click the **Force** box for Force Type.
- Click the **Action only** box for Direction.
- Click the **Reverse Direction** button. The direction arrow points into the finger as illustrated.
- Enter **62 N** for Constant Value.
- Click **OK** from the Force/Torque PropertyManager. Force1 is displayed in the Motion Study FeatureManager.



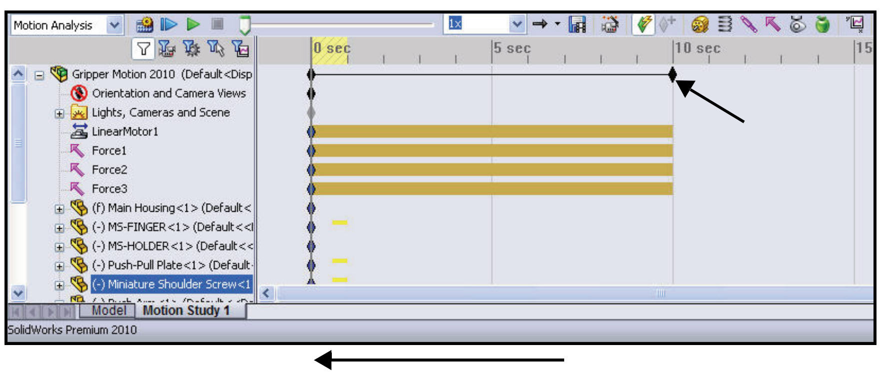
7 Apply a Contact Force to the two other Fingers.

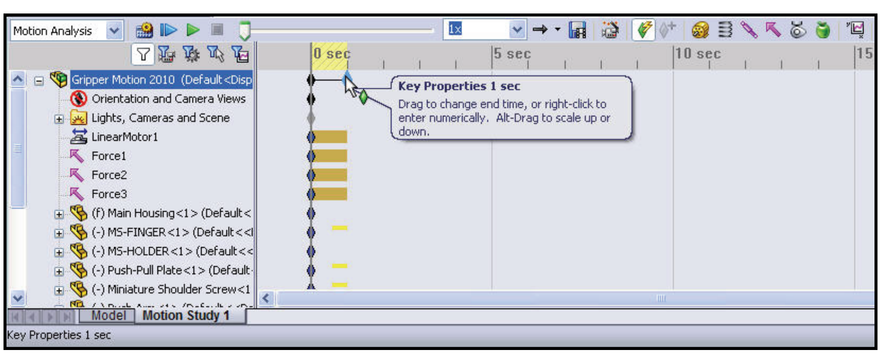
- **Repeat Steps 5 and 6** for the other two Gripper fingers. At the end of this step, you should view three Forces and a LinearMotor in the Motion Study FeatureManager as illustrated.



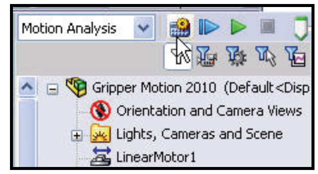
8 Create a SolidWorks Motion Simulation.

- Drag the right-most **Key** on the top timeline, corresponding to **Gripper**, back to 1 second as illustrated. You may need to zoom in on the timeline after getting it close.



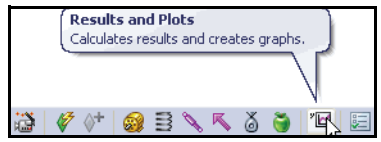


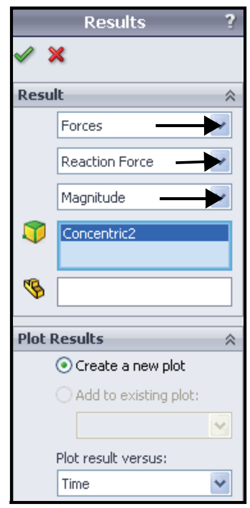
9 Run the SolidWorks Motion Simulation.

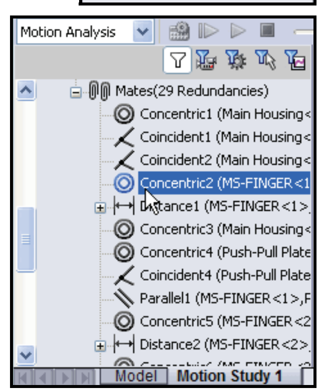


- Click the **Calculate** icon. View the assembly moving while the analysis is being performed.

10 Calculate the Reaction Force at the Finger Hinge.

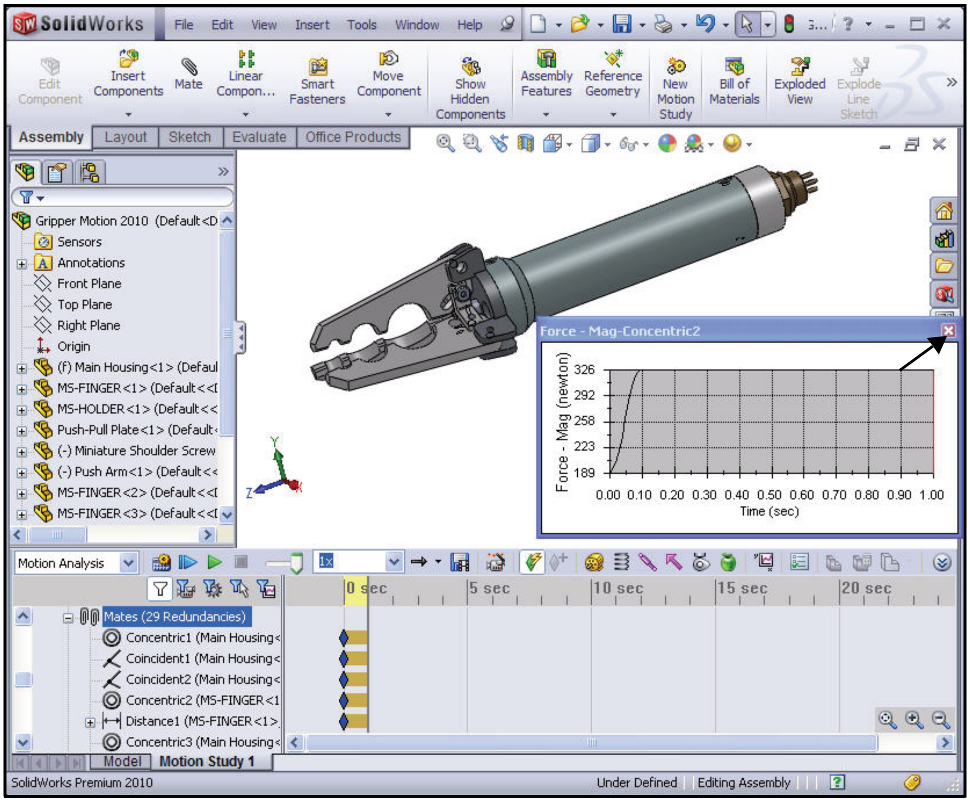






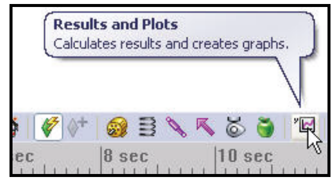
- Click the **Results and Plots** icon from the Motion Study toolbar. The Results PropertyManager is displayed.
- Select **Forces** from the Result drop-down menu.
- Select **Reaction Force** from the Result drop-down menu as a sub-category.
- Select **Magnitude** from the Result drop-down menu as the Result component.
- Expand the **Mates** folder from the Motion Study FeatureManager.
- Click **Concentric 2** from the Mates folder as illustrated.
- Click **OK** in the Results PropertyManager.
- Click **No** to the displayed message. View the plot.

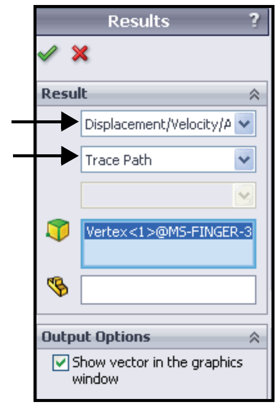
- Click along the **time axis** and view the changes in the Gripper.
- **Close** the Force - Mag-Concentric2 plot dialog box.



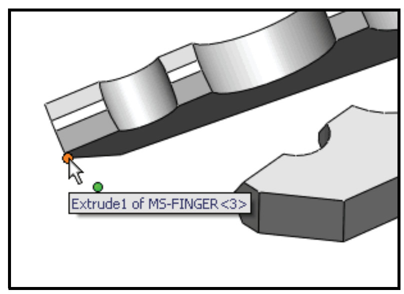
11 Create a Trace Path.

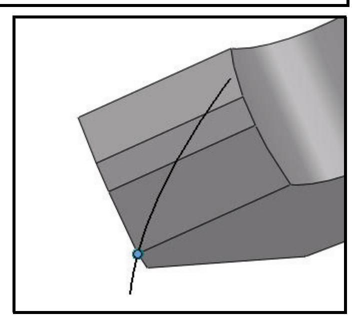
- Click on the **Results and Plots** icon from the Motion toolbar. The Result PropertyManager is displayed.
- Select **Displacement/Velocity/Acceleration** from the drop-down menu in the Result box.
- Select **Trace Path** from the drop-down menu as a sub-category.
- Click a **point** at the end of a finger as illustrated in the Graphics area. Note the icon feedback symbol.
- Click **OK** from the Results PropertyManager.





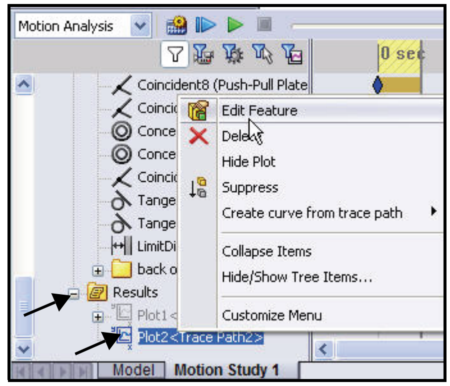
Note: A Trace Path graphically displays the path that any point on any moving part follows.

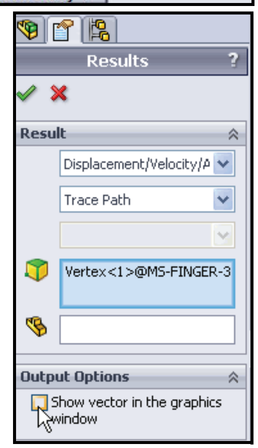




12 Edit a Feature.

- **Scroll down** to the bottom of the Motion Study FeatureManager
- Expand the **Results** folder.
- Right-click **Plot2<TracePath1>.**
- Click **Edit Feature**. The Result PropertyManager is displayed.
- De-select the **Show vector in the graphics window** checkbox. (This is how you can hide a Trace Path without deleting it.)
- Choose **OK** from the Results PropertyManager.



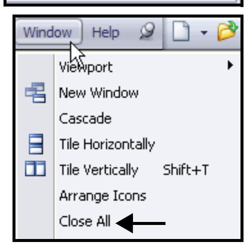


13 Rebuild and Save the Assembly.

- Click **Save** from the Menu bar toolbar.
- Click **OK** to the Rebuild message.

14 Close all models.

- Click **Window, Close All** from the Menu bar menu.



SolidWorks Motion Conclusion

During this short session on SolidWorks Motion, you have seen how physics-based motion simulation can be used to improve the quality and performance of your design. SolidWorks Motion simulates the mechanical operations of motorized assemblies and the physical forces they generate, by determining factors such as power consumption and interference between moving parts. SolidWorks Motion helps you ascertain if your designs will fail, when parts will break, and whether or not they will cause safety hazards.

Leverage the power of SolidWorks. SolidWorks Motion works inside the SolidWorks window and uses existing assembly information to build motion simulation studies.

Transfer loads seamlessly into SolidWorks Simulation to perform stress analysis.
With the seamless transfer of loads from SolidWorks Motion to SolidWorks Simulation, you can visualize stress and displacements on a component as a single time instance or for the entire simulation cycle.

Simulate real-world operating conditions. By combining physics-based motion with assembly information from SolidWorks, SolidWorks Motion can be used in a broad span of industry applications.

Associate physics-based models to engineering conditions. SolidWorks Motion offers several types of joint and force options to represent real-life operating conditions.

Interpret results with powerful and intuitive visualization tools. Once you have completed the motion simulation run, SolidWorks Motion offers a variety of results visualization tools that allow you to gain valuable insight into the performance of your design.

Collaborate and share analysis results. SolidWorks Motion makes it easy to collaborate and share analysis results effectively with everyone involved in the product development process.