SolidWorks[®] 2011

SolidWorks Simulation Hands-on Test Drive

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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

Hands on Test Drive

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.

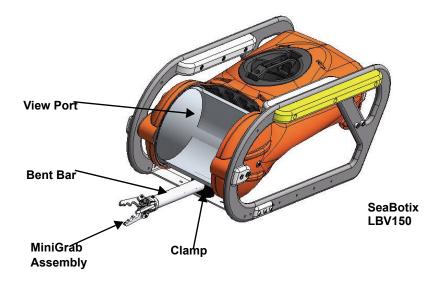
The SeaBotix LBV150

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, lowcost, 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

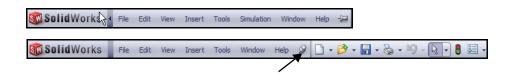
The Menu Bar toolbar contains a set of the most frequently used tool buttons. The available tools are: New 🗋 - Creates a new document, Open 📄 - Opens an existing document, Save 🖃 - Saves an active document, Print 🗟 - Prints an active document, Undo 🗐 - Reverses the last action, Select 🔄 - Selects sketch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

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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 *s* option displays both the Menu Bar toolbar and the Menu Bar menu.



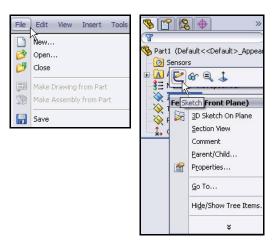
User Interface

Hands on Test Drive

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 rightclick) 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: Redraw Ctrl+R 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 S, PropertyManager

ConfigurationManager [8], and DimXpertManager

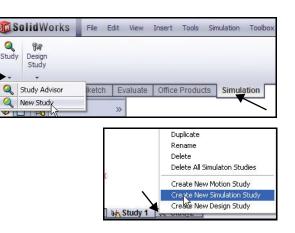


SolidWorks Simulation CommandManager Tab

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.

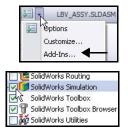
Note: Create a New Study using

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.

Hands on Test Drive

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.

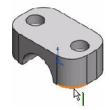
Getting SolidWorks Help

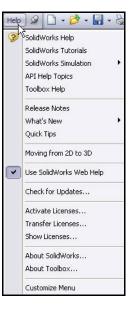
SolidWorks has a comprehensive Home help Page function that is design to assist the new and experience user. It provides information on What's New, SolidWorks Glossary, New Release notes, and more.

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.







Hands on Test Drive

SolidWorks Simulation

Getting SolidWorks Simulation Help.

Click Study Advisor,

Study Advisor from the Simulation tab in the

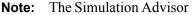


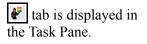
CommandManager with

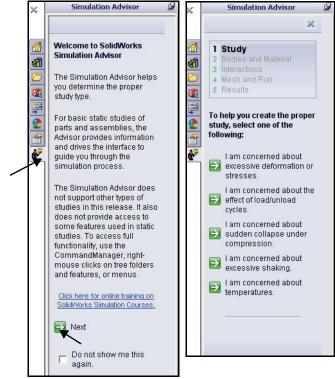
an active study to obtain the Simulation Advisor.

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.







Hands on Test Drive

SolidWorks Tutorials and SolidWorks Simulation Tutorials

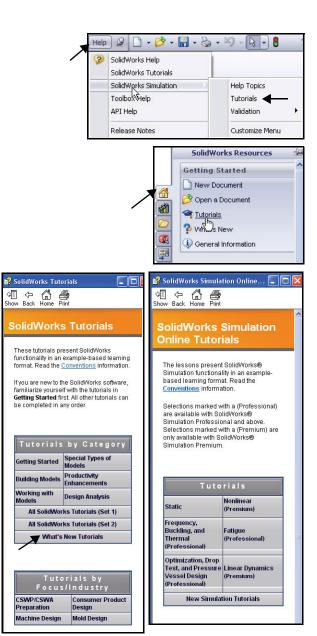
The SolidWorks Tutorials provide step-by-step lessons with sample files covering SolidWorks terminology, concepts, functions, features, and many Add-Ins. Work or view the lesson tutorials to learn and strengthen your skills.

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

Note: You can also access the SolidWorks Tutorials, click the SolidWorks **Resources**

tab from the Task Pane and click **Tutorials**. View the available tutorials.

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



SolidWorks Simulation

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

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



SolidWorks and SolidWorks Simulation

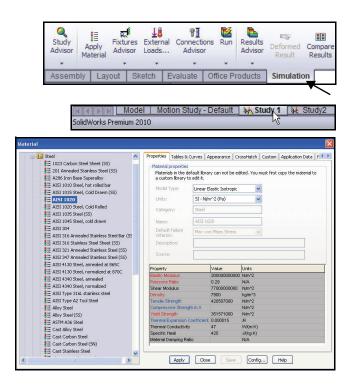
SolidWorks Simulation allows you to test a design and run multiple analysis iterations without ever leaving SolidWorks.

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.



SolidWorks Simulation

Analyze the Housing

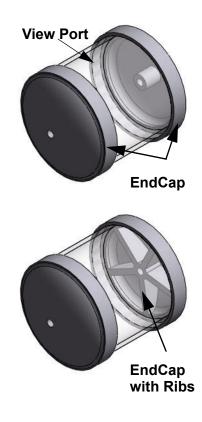
For your first analysis, explore the design validation of the Housing components in the SeaBotix LBV150 assembly using SolidWorks Simulation.

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 sideby-side for a final design comparison.



SolidWorks Simulation

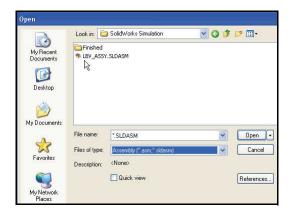
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Starting a SolidWorks Session

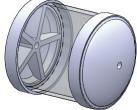
- 1 Start a SolidWorks Session.
 - Click the **Start** menu.
 - Click All Programs, SolidWorks 2010, SolidWorks 2010.
- **Note:** You can quickly start a SolidWorks 2010 session by double-clicking the left mouse button on the desktop shortcut, if there is a shortcut icon on the system desktop.

2 Open the SeaBotix LBV150 Assembly.

- Click **Open** if 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.



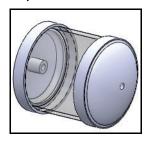




SolidWorks Simulation

- 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.





SeaBotix LBV150 *

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eformation

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.



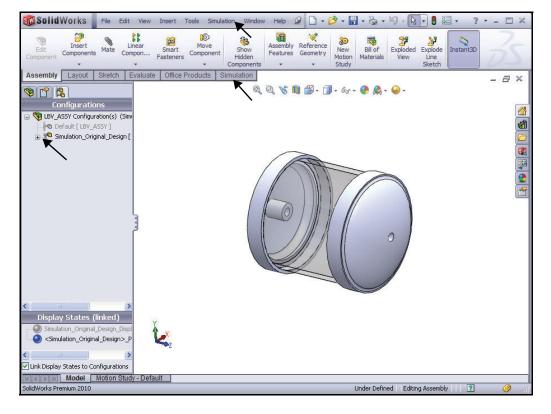
差 🔸

📃 Options

Customize..

SolidWorks Simulation

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



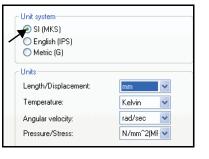
5 Set Default Options in SolidWorks Simulation.

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



Analyze the Housing

- 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.

Units	Display color charts					
Load/Fixture	✓ Display Polici on Bito					
Mesh	Position					
Results	Predefined positions					
Plot						
Color Chart	O User defined					
Derault Plots derault Plots derault Plots	Horizontal from left: 80 🔯 %					
Plot1	Vertical from top: 20 💭 %					
Plot2	Width					
Vict3	◯ Wide					
Q Thermal Study Results	Normal					
	🔘 Thin					
Qo Top Test Study Results	Number format					
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E Eatigue Study Results	No. of decimal places: 3 🤤					
Plot1						
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Get Monlinear Study Results						
	Color options					
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Plot3						
User information	No of chart colors: 12 🛟 🗌 Flip					
Report	User defined					
Study Report	5					
	Specify color for values above yield for vonMises plot					

SolidWorks Simulation

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.



SolidWorks Simulation

Creating a Static Analysis Study

- 1 Create a Static Analysis Study.
 - Click Simulation tab in the CommandManager.

ST So	lidWor	ks	File E	Edit Vie	w In:	sert 1	Tools	Simulation	Toolbox
Study Advisor	I≣ Apply Material	Fixture Adviso		and the second	nection dvisor	s Run	Adviso		II Compare Results
St	l udy Adviso	or	ketch	Evalu	ate C		roducts	Simulati	ion
	ew Study						_		

- 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.



Create a Static Analysis Study

SolidWorks Simulation

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.

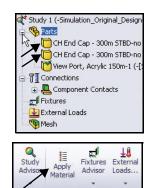
Define and apply material to the two EndCaps in the Housing assembly in the next section.

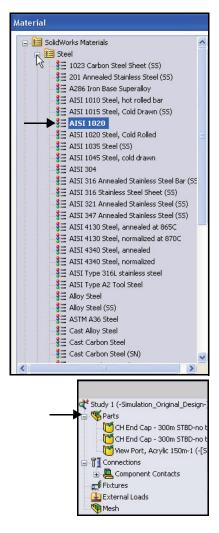
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📲 A286 Iron Base Superalloy		om library to e					
₿Ξ AISI 1010 Steel, hot rolled bar	Model Type: Linear Ela		lastic Isotropic				
AISI 1015 Steel, Cold Drawn (SS)	moder	Type:					
	Units:	Units; SI - N/m Category: Steel		n^2 (Pa)	~		
IIII 1020 Steel, Cold Rolled	Catego						
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AISI 304	Defaul	t failure	Max you	n Mises Stress			
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→ 📑 AISI 316 Stainless Steel Sheet (SS)	Descrip	ption:					
→ 📑 AISI 321 Annealed Stainless Steel (SS)			}				
♣∃ AISI 347 Annealed Stainless Steel (SS)	Source	1					
→ 📑 AISI 4130 Steel, annealed at 865C	Durante	2		Value	Units		
AISI 4130 Steel, normalized at 870C	Property Elastic Mo	- dulum	_	20000000000	1 1001100		
→ 📲 AISI 4340 Steel, annealed	Poissons			0.29	N/A		
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→ 📲 AISI Type 316L stainless steel	Density			7900	kg/m^3 N/m^2		
→ 📑 AISI Type A2 Tool Steel	Tensile St	trength		420507000			
E Alloy Steel	Compress	sive Strength	in X		N/m^2		
E Alloy Steel (SS)	Yield Stre	A Dealer and the state of the second second second second		351571000	N/m^2		
STM A36 Steel	Contraction of the second	Expansion Co	efficient		K		
Steel	100000000000000000000000000000000000000	Thermal Conductivity		47	WW(m·K)		
E Cast Carbon Steel		Specific Heat Material Damping Ratio		420	J/(kg·K) N/A		
E Cast Carbon Steel (SN)	iwaterial D	amping Natio			INNA		
E Cast Stainless Steel	1						

SolidWorks Simulation

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 <u>s</u> on the Parts folder indicates that material is assigned to the parts.





Assigning Materials in SolidWorks Simulation

SolidWorks Simulation

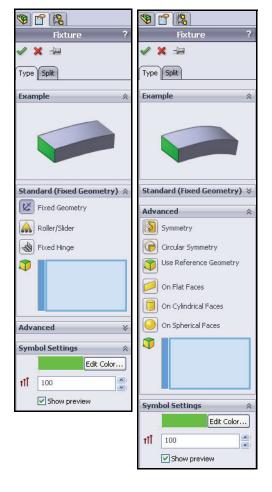
Applying Fixtures

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.

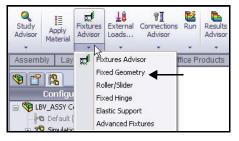
In this section, address an On cylindrical face fixture.



SolidWorks Simulation

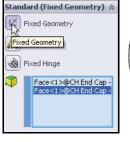
Applying a Fixture

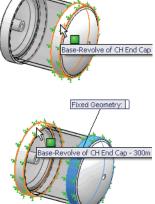
- 1 Apply a Fixture.
 - Click the Fixtures Advisor dropdown arrow from the Simulation tab in the CommandManager.
 - Click Fixed Geometry. The Fixture PropertyManager is displayed. The Fixed Geometry option is selected by default. Fix the model to simulate how the two EndCaps are mounted to the Housing.

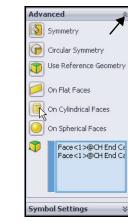


2 Select the Faces to be Fixed.

- 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.

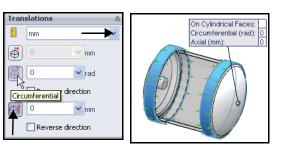
Applying Fixtures

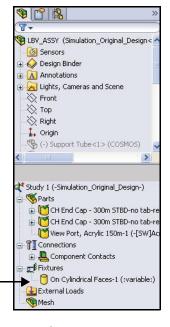
SolidWorks Simulation

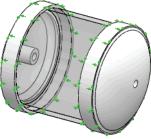
- 4 Select Units and Displacement Components.
 - Select **mm** from the Unit drop-down menu.
 - Click the Circumferential
 box.
 - Click the Axial *box*. View the results in the Graphics area.
- 5 Apply the Fixture.
 - Click **OK** ✓ from the Fixture

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.





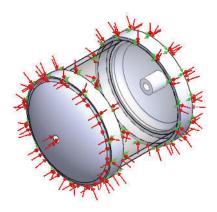


SolidWorks Simulation

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.

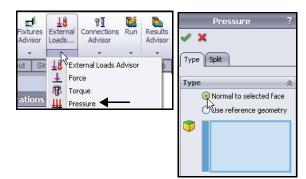


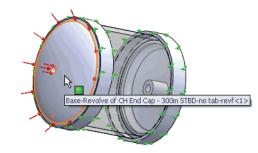


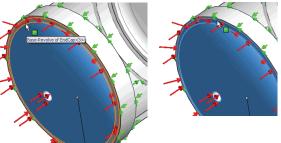
SolidWorks Simulation

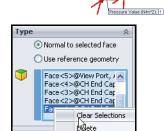
Applying a Pressure Load

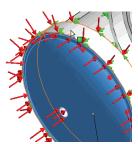
- 1 Apply a Pressure Load.
 - Click the External Loads drop-down arrow from the Simulation tab in the CommandManager.
 - Click Pressure . The Pressure PropertyManager is displayed. The Type tab is selected by default.
 - Click the Normal to selected face box.
- 2 Select the Faces to Apply the Load.
 - 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.







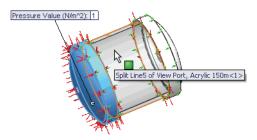




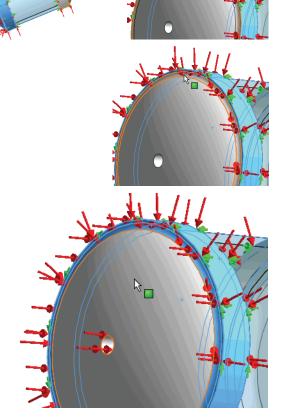
SolidWorks Simulation

- 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.

Pressure Value (N/m^2): 1



- **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.



Applying Loads

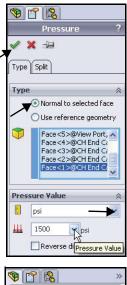
5 Set the Pressure Value.

- 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 <u>u</u> 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.



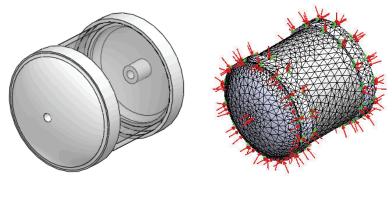


Creating a Mesh and Running the Analysis

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.



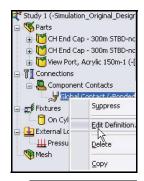
Before Meshing

After Meshing

SolidWorks Simulation

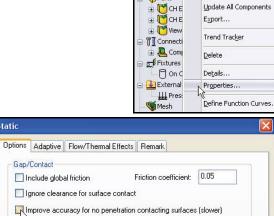
Creating a Compatible Mesh

- 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 I** 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.



Study 1 (-S

🤏 Parts

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Static

SolidWorks Simulation

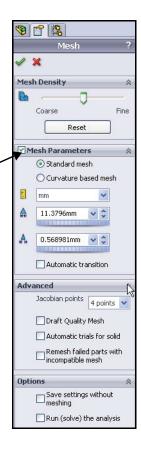
Creating a Mesh

- 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.





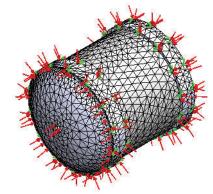
Creating a Mesh and Running the Analysis

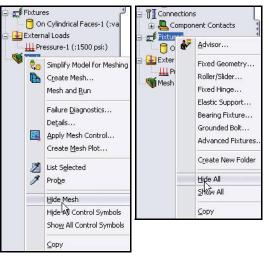
SolidWorks Simulation

- 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 **s** 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 K from the Simulation tab in the CommandManager. Three default plots are created.





SolidWorks Simulation

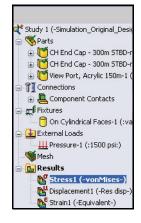
Viewing the Results

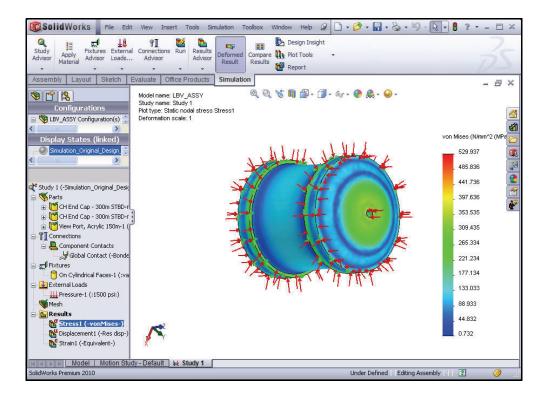
After a successful run of a Static analysis, SolidWorks Simulation creates three default plots: Stress, Displacement, and Strain.

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.



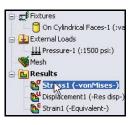


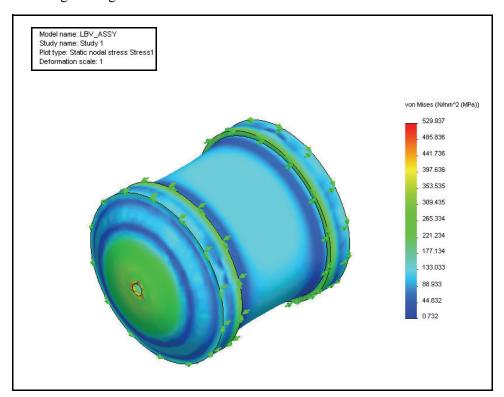
SolidWorks Simulation

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.

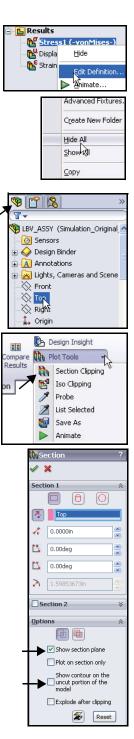






SolidWorks Simulation

- **Note:** To view the stress plot in a different unit system, rightclick 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.



Viewing the Results

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SolidWorks Simulation

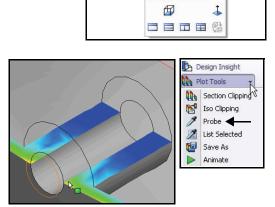
- Click **OK** ✓ from the Section PropertyManager.
- Rotate the model as illustrated with the middle mouse button to view the results.
- **Note:** Deformation is magnified for improved visibility. The deformation can be displayed at any scale.
- Note: Use the Zoom to Area (1) tool located in the Heads-up 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 dropdown arrow from the Simulation tab in the CommandManager.
- Click Probe An Interview Click Probe
 Results PropertyManager is displayed.



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Zoom to Area

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Zooms to the area you select with a bounding box.

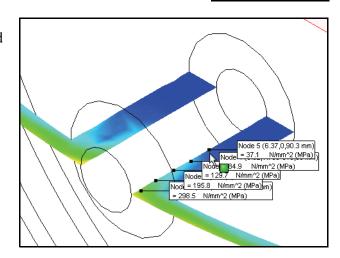
SolidWorks Simulation

Plot

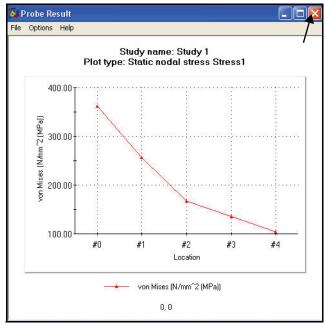
Report Options

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- 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.



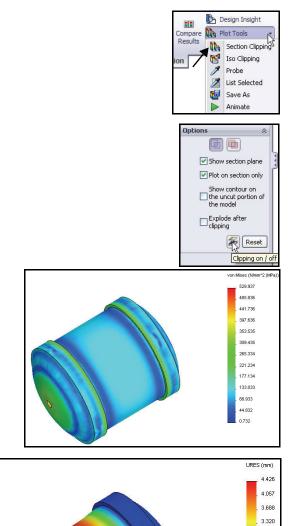
- 7 Review the Plot.
 - Review the plot. This is an excellent way to examine the variation in stress across the geometry of your part.
- 8 Close the Probe Results dialog box.
 - Close the Probe Results dialog box.
- 9 Close the Probe Result PropertyManager.
 - Click **OK** from the Probe Result PropertyManager.



SolidWorks Simulation

10 Deactivate the Section Plot.

- Click the Plot Tools dropdown 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.

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2.582 2.214 1.845 1.476 1.108 0.739 0.370 0.002

SolidWorks Simulation

13 Animate the Displacement Plot.

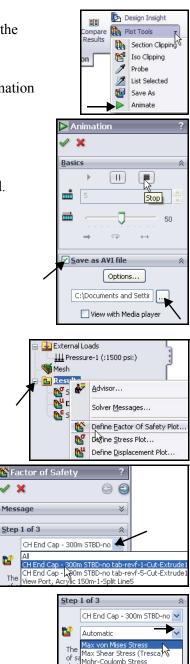
- Click the **Plot Tools** drop-down arrow from the Simulation tab in the CommandManager.
- Click Animate ▶. The Animation PropertyManager is displayed. View the animation in the Graphics area.
- 14 Stop the Animation.
 - Click Stop
- 15 Save the Animation.
 - Check the Save as AVI file box as illustrated.
 - 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.



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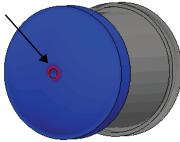
Max Normal Stress mate Automatic stress used for ductie n Mohr-Coulomb used for brittle

Viewing the Results

SolidWorks Simulation

- Click Next Stock 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.

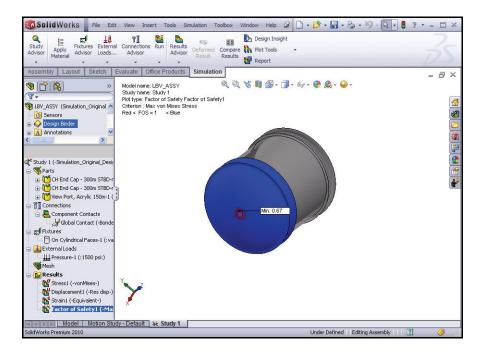


Viewing the Results

SolidWorks Simulation

- 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.





SolidWorks Simulation

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 selfviewing, small, and hence convenient to send via email.

Save in: 🔁 SeaBotix LBV150-Study 1 💽 🖓 P 🖽	Save As		? 🗙
	Save in:	🕒 SeaBotix LBV150-Study 1 🔹 🔾 🗘 🕫 🕬	•
File name: SeaBotix LBV150-Study 1-Stress-Plot1 (-vonMises-) Save			
Save as type: eDrawings Files (*.analysis.easm) Cancel	Save as ty	vpe: eDrawings Files (*.analysis.easm)	Cancel

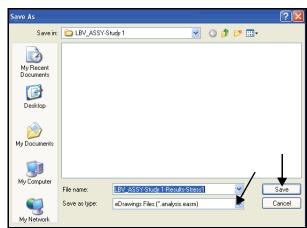
SolidWorks Simulation

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.

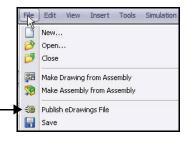


■ Click Save.



2 Publish a SolidWorks eDrawing.

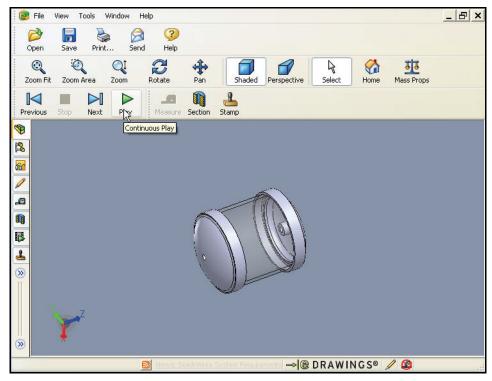
Click File, Publish eDrawings File from the Menu bar menu. The Save Configurations to eDrawings file dialog box is displayed.



SolidWorks Simulation

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

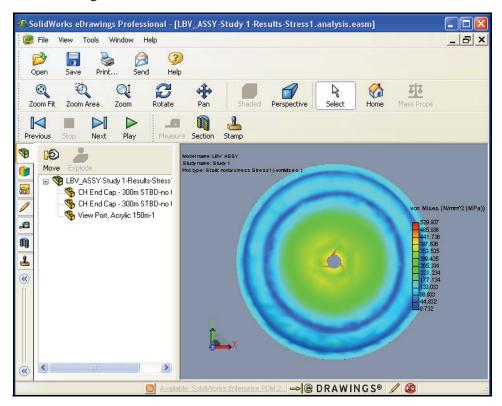




SolidWorks Simulation

- 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.





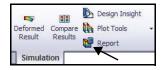
Creating a SolidWorks eDrawings File

SolidWorks Simulation

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.

Stress analysis of LBV_ASSY	
Author: John Smith Company: XYZ	
Company, ATE	
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.	
Contents	
 <u>Cover Page</u> <u>Description</u> <u>Assumptions</u> <u>Model Information</u> <u>Study Properties</u> <u>Units</u> <u>Material Properties</u> Loads and Restraints 	
9. Connector Definitions	
10. Contact	~
🗟 Done 😔 😵 My Computer	

HTML Format

SolidWorks Simulation

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Generating a Static Study Report

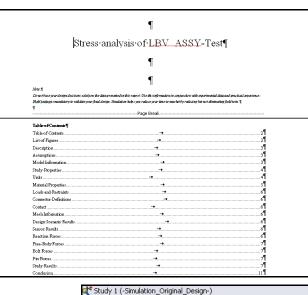
- 1 Generate a Static Study Report.
 - Click Report Provide the Simulation tab in the CommandManager.
 - Select Contemporary for Report Style.
 - Check the **Author** box.
 - Enter a value for **Author**.
 - Check the **Company** box.
 - Enter a value for Company.
 - Scroll through the included sections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

Report format s	nt report forr ettings	nat: Defau	lic		
Repo	t style: 🛛	Contemporary	у		~
Available	sections:	>> De As: Mo Stu	Included sect ver Page scription sumptions idel Information udy Properties its		Move up Move dow Add Remove
Section proper	ties				
Name:	Cover Page	9			
Comments:	Test				
Logo:					Browse
Author:	John Smith	-			
Company:	XYZ	-	_		
ocument settir Report path:	-	ents and Sett	:ings\mplanchari	d∖My Docu	Browse.
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Generating a Report

SolidWorks Simulation

- 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.
- 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.





SolidWorks Simulation

Analysis 2 - Static Study 2

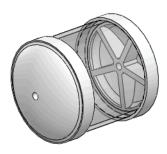
In Study 1, the reports showed critical areas where the factor of safety was less than one.

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.



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🕁 🧐 (f) View Port, Acrylic 150m<1> ->	(C					
😑 🧐 (f) CH End Cap - 300m STBD-no tal	o-r					
😨 🛐 Mates in LBV_ASSY						
- 🙆 Sensors						
🕀 🔝 Annotations						
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	_					
(-) ML30 Feature (CirPattern1)						
(-) ML30: Parent/Child						
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- (-) Aft FI						

SolidWorks Simulation

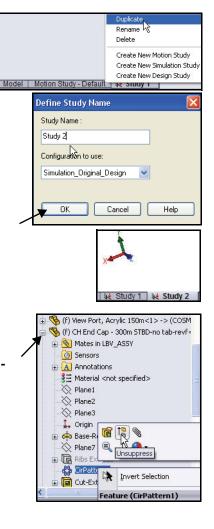
Creating Analysis 2 - Static Study 2

- 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 tabrevf.
- Right-click CirPattern1.
- Click Unsuppress 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.



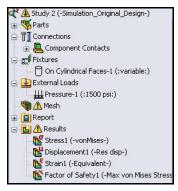




SolidWorks Simulation

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.



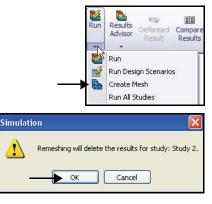
5 Create a Compatible Mesh

- 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.



SolidWorks Simulation

- 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.



1

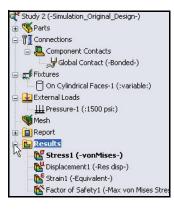
Start the Mesh Process. 7

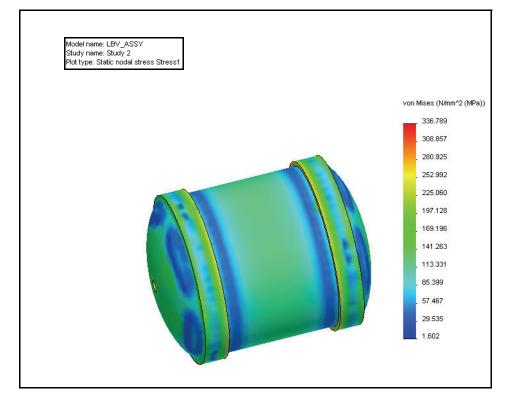
- Check the Mesh Parameters box. View your options.
- Check the **Run (solve) the analysis** box.
- Click **OK** s from the Mesh PropertyManager. Meshing starts and the Mesh Progress window appears. View the results in the Graphics area.



SolidWorks Simulation

- 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.





SolidWorks Simulation

- 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.

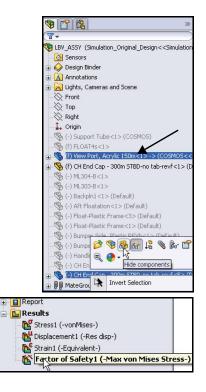
Note: The minimum FOS is now 1.02.



🖳 🛃 External Loads

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.

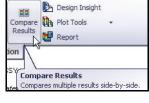


SolidWorks Simulation

Rotate the model and view the results.



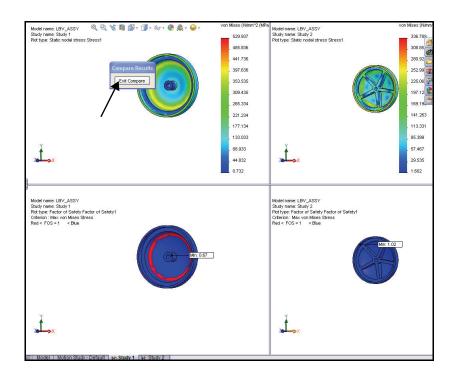
 Click Compare Results from the Simulation tab in the CommandManager. The Compare Results PropertyManager is displayed. Both Study 1 and Study 2 are checked.



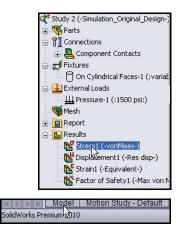
- Click the Manually select results to view box.
- Un-check the **Displacement1** and **Strain1** box under Study 1.
- Check the Stress1 and Factor of Safety1 box under Study 1.
- Check the Stress1 and Factor of Safety1 box under Study 2.
- Click OK from the Compare Results
 PropertyManager. View the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



SolidWorks Simulation



- 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.



udy 1 🗽 Study 2

SolidWorks Simulation

- 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 (a) from the Context toolbar. The components are displayed in the Graphics area.

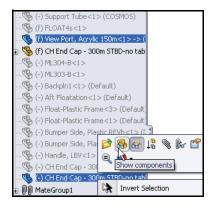
12 Return to Study 1.

- 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.

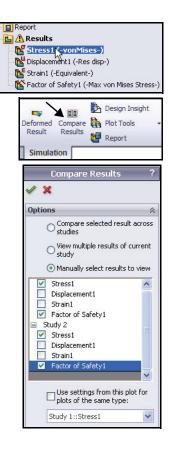
Model Mo

SolidWorks Premium 2010

- 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.



udy - Default 🛛 😽



Analysis 2 - Static Study 2

- tel name: LBV_ASSY dy name: Study 1 type: Static nodal stress Stress1 Q Q 🖏 🦋 🛐 📲 - 🗇 - 66 - 🥐 急 - 🤪 Model name: LBV_ASSY Study name: Study 2 Plot type: Static nodal stress Stress1 529.937 485.836 441.736 397.636 Exit Compare 353.535 309.435 265.334 221.234 141.263 177.134 113.331 133.033 85.399 88.933 57.467 44.832 29.535 0.732 1.602 name: LBV_ASSY name: Study 1 pe: Factor of Safety Factor of Safety1 pn: Max: von Mises Stress FOS = 1 < Blue Model name: LBV_ASSY Study name: Study 2 Plot type: Factor of Safety Factor of Safety1 Orderrion : Max von Mises Stress Red = FOS = 1 = < Blue 1.02 Min: 0.67 1 Model Motion Study - Default 🦗 Study 1 🙀 Study 2 Under De
- 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.



SolidWorks Simulation Conclusion

SolidWorks Simulation Professional

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 with Ribs

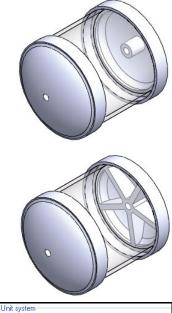


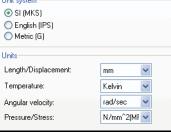
SolidWorks Simulation Professional

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.





Trend Tracker Analysis

SolidWorks Simulation Professional

SolidWorks Simulation

- 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.





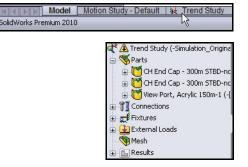
2 If required, activate SolidWorks Simulation.

- 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.

Run Simulation Advisor from CommandManager (You need to restart SolidWorks for the change to take effect)

SolidWorks Simulation Professional

- 3 Vlew the Trend Study.
 - Click the Trend_Study tab as illustrated. The Trend_Study is displayed.



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Connections Run Advisor

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🗄 📲 Connections

➡ ➡ Fixtures
 ➡ External Loads
 ♥ Mesh

E Results

Results

Advisor

Trend Study (-Simulation_Original_D

CH End Cap - 300m STBD-no
 CH End Cap - 300m STBD-no
 CH End Cap - 300m STBD-no

🗄 ॉ View Port, Acrylic 150m-1 (-

🔁 Stress1 (-vonMises-)

Strain1 (-Equivalent-)

🔁 Displacement1 (-Res disp-)

1

Simulation

4 Perform an Analysis on the Study.

- Click Run Manager. 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.



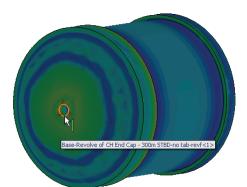
SolidWorks Simulation Professional

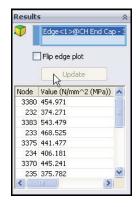
SolidWorks Simulation

📴 Design Insight

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

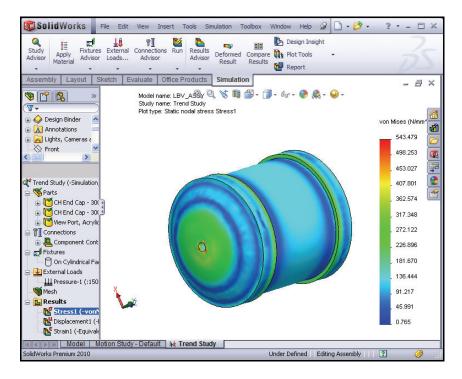






SolidWorks Simulation Professional

- 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.



Trend Tracker Analysis

Kan Run

Trend Stud

🤏 Parts

SolidWorks Simulation Professional

- 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.

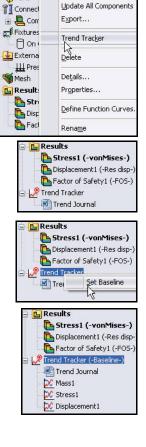
See how Trend Tracker allows you to perform design changes without creating multiple studies or configurations.

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.







SolidWorks Simulation Professional

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Sensor Type

Value : 0 N/m^2
Data Quantity

Mr Stress

Properties

Factor o

Alert

-

N/m^2

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Simulation Data

VON: von Mises Stress

Max over Selected Entitie 🗸

Clear Selections Delete

Customize Menu

LBV ASSY.SLDASM

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- Select Simulation Data for Sensor type from the drop-down menu.
- Select N/m² 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.

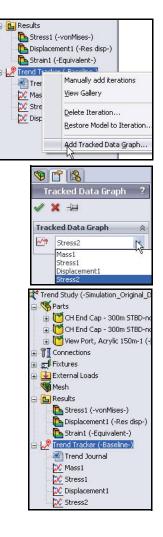
Click the Trend Study tab at the bottom of the Graphics area.



SolidWorks Simulation

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.

SolidWorks Simulation Professional

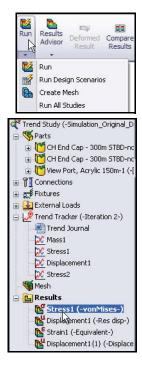
- Expand the first CH End Cap 300m
 STBD from the FeatureManager as illustrated.
- Right-click CirPattern1.
- Click Unsuppress from the Context toolbar. The Housing with the ribbed End caps is displayed in the Graphics area.

16 Return to the Trend Study.

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



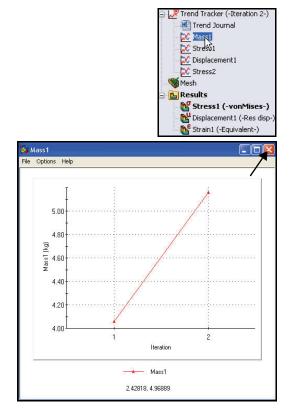




17 Run an Analysis.

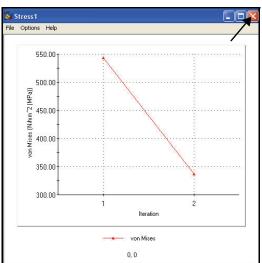
- Click Run Manager. Once the analysis is completed, the plots under the Trend Tracker folder are updated.
- View the Stress1 (-vonMises-) plot.

- 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.



SolidWorks Simulation

SolidWorks Simulation Professional

🛃 External Loads

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.

Image: Trend Journal Image: Trend Journal

Prend Tracker (-Iteration 2-)

21 Save and Close the Model.

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

	1	Trend Journa	l¶	
¶.				
File Name: 0	LBV_ASSY.SLDASMo			
Study name:©	Trend-Study0			
Description:¤	α			
1				
Baseline¤	a			
Time Completed: 🗆	Friday, October 02, 2009 7:40:12	•AMo		
Tracked Data:¤	a			
1				
	Sourceo	Турео	Actual Valueo	Normalized Valu
	Mass10	Model·Max [©]	4.05904•(kg)¤	1000
	Stress1 (VON: von Mises Stress)	Model·Max [©]	543.479 (N/mm^2 (MPa))	1000
	Displacement 1 (URES: Resultant Displacement)	Model·Max [©]	4.43455•(mm)¤	1000
	Stress2·(VON: von Mises Stress)	Max-over-Selected-Entities	5.43479e+008·(N/m^2)¤	1000
		1		
Iteration 2¤	a			
Time Completed: ©	Friday, October 02, 2009 7:51:41	·AMO		
Tracked Data:0	a			
ſ				
	Sourceo	Турес	Actual Valueo	Normalized Valu
	Mass 10	Model·Max¤	5.16175•(kg)¤	127¤
	Stress1 (VON: von Mises Stress)	Model·Max [©]	337.151•(N/mm^2•(MPa))	620
	Displacement1 (URES: Resultant Displacement)	Model·Max¤	4.42488•(mm)¤	99a
	Stress2·(VON: von Mises Stress)©	Max·over·Selected Entities	2.5485e+008·(N/m^2)¤	460

SolidWorks Simulation Professional

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

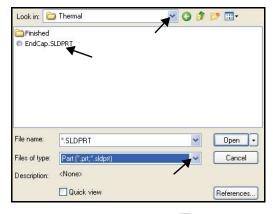


Thermal Analysis

SolidWorks Simulation Professional

Create the Thermal Analysis Study

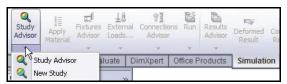
- 1 Open the EndCap Part.
 - Click **Open** if from the Menu bar toolbar.
 - Double-click EndCap from the SeaBotix\SolidWorks Simulation Professional\Thermal folder.
- **Note:** Files of type is Part. The EndCap is displayed in the Graphics area.



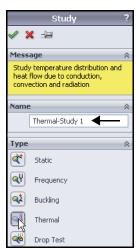


2 Create a Thermal Study.

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



- Click the Study Advisor drop-down arrow from the Simulation tab.
- Click New Study . The Study Property Manager is displayed.
- Enter **Thermal-Study 1** for the name of the Study.
- Click **Thermal 4** for Type.
- 3 Display the Study.
 - Click **OK** ✓ from the Study PropertyManager.



📢 Thermal-Study 1 (-Default-) Connections

M

Run

🔞 Thermal Loads 🎨 Mesh

SolidWorks Simulation Professional

Applying the EndCap Material.

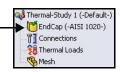
- 1 Apply the Material of the EndCap.
 - Click EndCap from Thermal-Study 1 (-Default-).
 - Click **Apply Material** I from the Simulation tab in the CommandManager. The Material dialog box is displayed. View your options.



- Click **AISI 1020** from the Steel folder.
- Click **Apply**.
- Click **Close** from the Material dialog box.

SolidWorks Materials	^	Properties	Tables & Curves	Appearance C	rossHatch	Custom	Application Date
🖃 🔚 Steel		Material	properties				
1023 Carbon Steel Sheet (SS)				ibrary can not be e	dited. You	must first	copy the materia
📲 201 Annealed Stainless Steel (SS)		a custo	om library to edit i	t.			
🚦 A286 Iron Base Superalloy		Model	Type:	ar Elastic Isotropic			
AISI 1010 Steel, hot rolled bar		11000		· · · · ·) B1		
AISI 1015 Steel, Cold Drawn (SS)		Units	SI -	N/m^2 (Pa)	~		
AI5I 1020		Cateo	ory: Ste	el		7	
AIS 1020 Steel, Cold Rolled						4	
AISI 1035 Steel (SS)		Name:	AIS	I 1020			
🚦 AISI 1045 Steel, cold drawn		Defaul	It failure	von Mises Stress			
ISI 304		criterio	on:	(1011)1205 2(1055			
AISI 316 Annealed Stainless Steel Bar (S	55	Descrij	ption:				
AISI 316 Stainless Steel Sheet (SS)		Source					
AISI 321 Annealed Stainless Steel (SS)		Jource					1
AISI 347 Annealed Stainless Steel (SS)		Property	(Value	Units		
AISI 4130 Steel, annealed at 865C		Elastic Mo	odulus	20000000000	- A 265 March		
- 🚼 AISI 4130 Steel, normalized at 870C		Poissons		0.29	N/A		
📲 AISI 4340 Steel, annealed		Shear Mo	dulus	7700000000	N/m^2		
🚦 AISI 4340 Steel, normalized		Density		7900	kg/m^3		
AISI Type 316L stainless steel		Tensile S		420507000	N/m^2		
📲 AISI Type A2 Tool Steel			sive Strength in X		N/m^2		
📲 Alloy Steel		Yield Stre		351571000	N/m^2		
Alloy Steel (SS)			Expansion Coeffic		AK IO		
ASTM A36 Steel		Specific H	Conductivity	47 420	W/(m·K) J/(kq·K)		
			Damping Ratio	420	N/A		
§∃ Cast Carbon Steel		material	Samping round				
SN)	-						

Note: A green check mark **S** 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 optiona 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 optiona thermostat for transient studies.

SolidWorks Simulation

Applying a Thermal Load

- 1 Apply a Thermal Load.
 - Click the Thermal Loads dropdown 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.



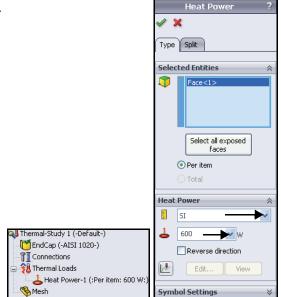


3 Enter Heat Power.

- Select SI from the Units dropdown 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.



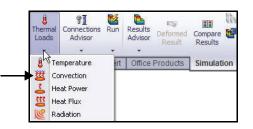
Thermal Loads and Boundary Conditions

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.

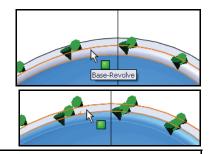
- 2 Select the Exposed Faces.
 - Rotate the **EndCap** with the middle mouse button as illustrated.
 - Click the outside face of the EndCap. Face<1> is displayed in the Faces for Convection box.

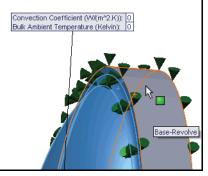




3 Select the other three exposed outside Faces.

- 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.





SolidWorks Simulation Professional

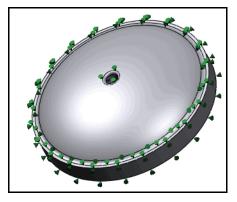
SolidWorks Simulation

- 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.







Thermal Loads and Boundary Conditions

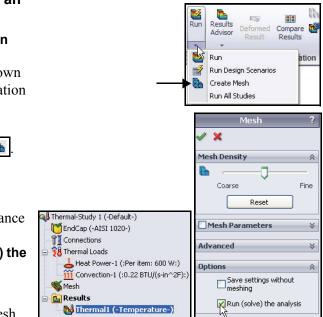
SolidWorks Simulation Professional

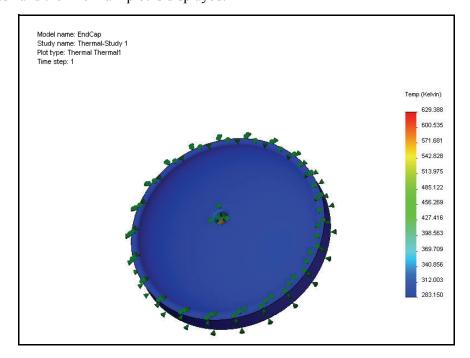
Creating a Mesh and running an Analysis

- 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.

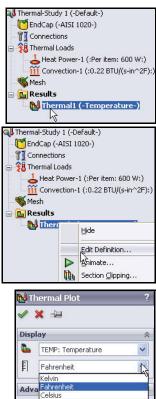
Click **OK** from the Mesh PropertyManager. You created a mesh and the Thermal1 plot is displayed.



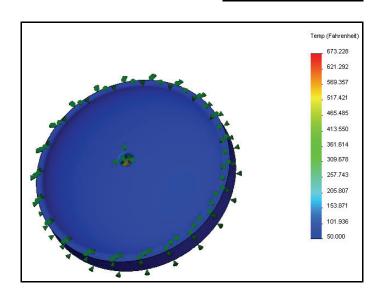


SolidWorks Simulation

- 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 dropdown 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.



Thermal Loads and Boundary Conditions

Property

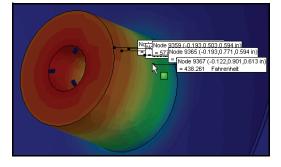
SolidWorks Simulation

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

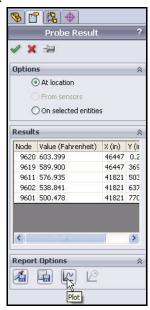
illustrated. The Probe box lists

Plot Tools 94 1 Section Clipp ompare Results 6 Iso Clipping Probe Simula List Selected 1 6 Save As Animate



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.

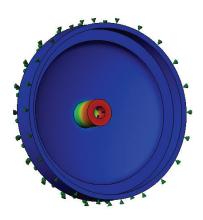


SolidWorks Simulation

Modify the Design

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

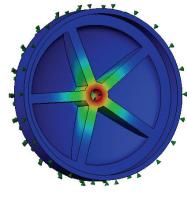
In this section, redesign the EndCap to use ribs. The ribs will help to dissipate the heat generated by the camera and search lights within the EndCap to the surrounding seawater.



Without Ribs

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

SolidWorks Simulation Professional

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.

S Mesh

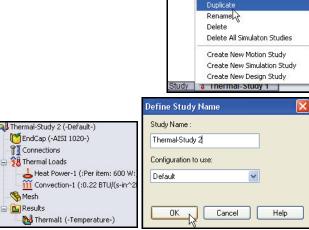
E Results

Add Ribs to the EndCap Part. 2

- Click the **Model** tab at the bottom of the Graphics area.
- Right-click **CirPattern1** from the FeatureManager.
- Click **Unsuppress** [18] 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.



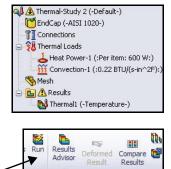


Model Motion Study 1



SolidWorks Simulation Professional

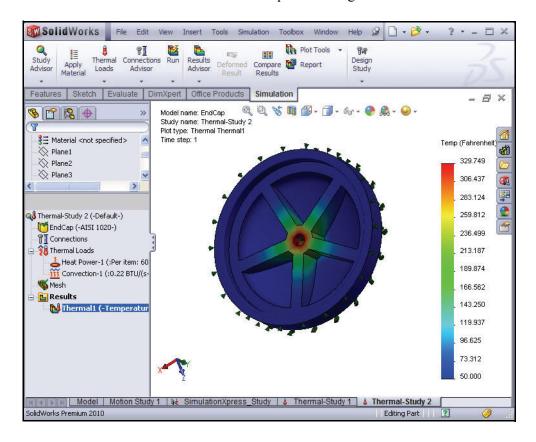
- 4 Review Thermal-Study 2.
 - Review Thermal-Study 2. The Thermal information was copied from the first study to the second study.



Office Products

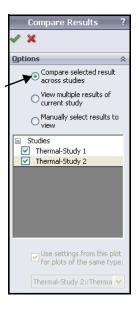
Simulation

- 5 Analysis the Model.
 - Click Run if 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.

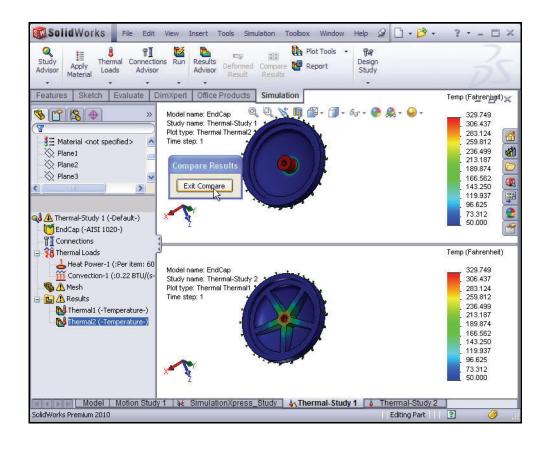


SolidWorks Simulation Professional

- 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.



SolidWorks Simulation



- 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.



SolidWorks Simulation Professional

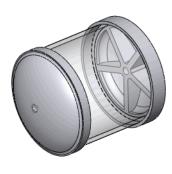
Drop Test Analysis

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.





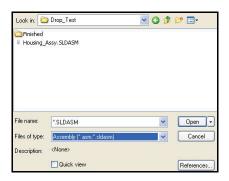




SolidWorks Simulation Professional

Creating a Drop Test Study

- 1 Open the Housing Assembly.
 - Click **Open** *integrable* 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.





Drop Test Analysis

SolidWorks Simulation Professional

Searts 8

8 Set

Res.

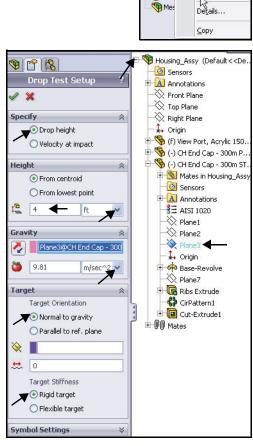
Gonnections

🔆 Droptest Study 1 (-Default-)

🗄 📮 Component Contacts

Define/Edit...

- 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 dropdown 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/sec² 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.







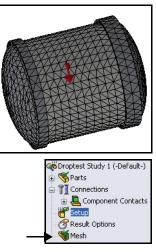
SolidWorks Simulation

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.







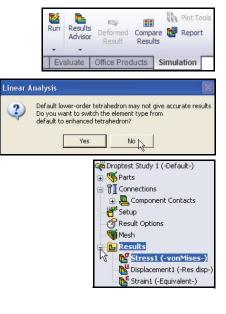
SolidWorks Simulation Professional

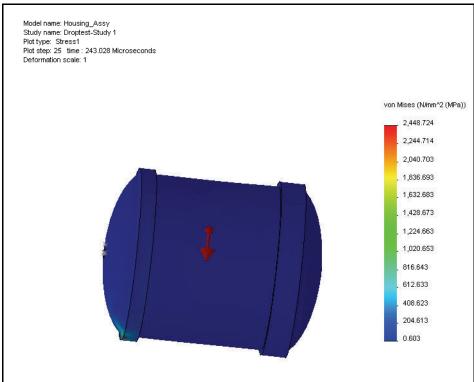
Running the Analysis

- 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.





Running the Analysis

SolidWorks Simulation

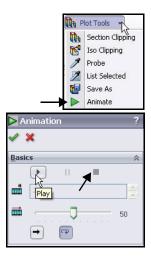
- 3 Set Scale Factor and View the von-Mises Plot.
 - Double-click Stress1 (-vonMises-). The Stress Plot PropertyManager is displayed. View the options.
 - Click OK from the Stress Plot PropertyManager.
 - Right-click Stress1 (-vonMises-).
 - Click Edit Definition. The Stress Plot PropertyManager is displayed.
 - 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.

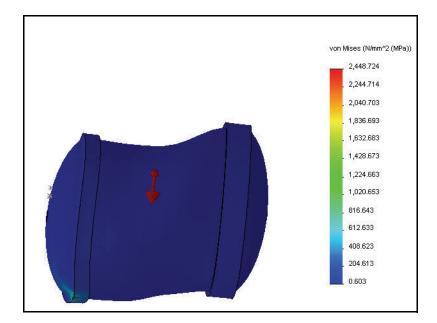


SolidWorks Simulation Professional

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 i** to start the animation. View the animation in the Graphics area.
 - Click **Stop** to stop the animation.

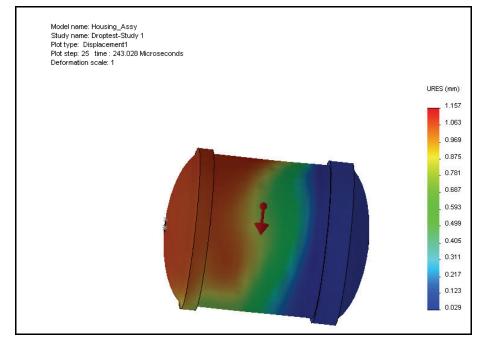




■ Click **OK** from the Animation PropertyManager.

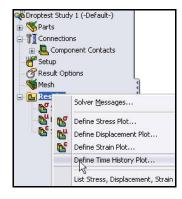
SolidWorks Simulation Professional

- **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.



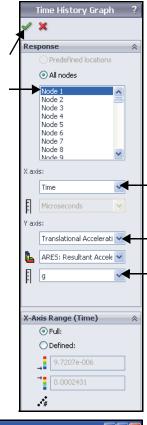
3 Create the Time History Graph.

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

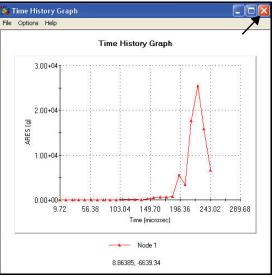


SolidWorks Simulation Professional

- 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.

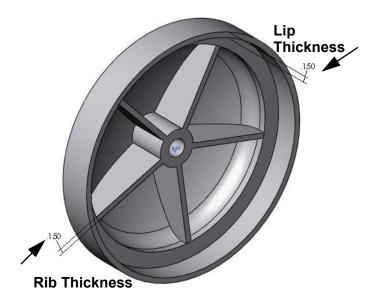


Running the Analysis

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.

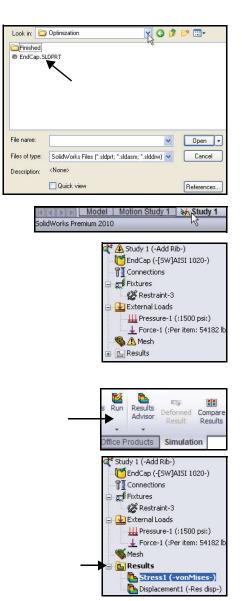


Optimization Analysis

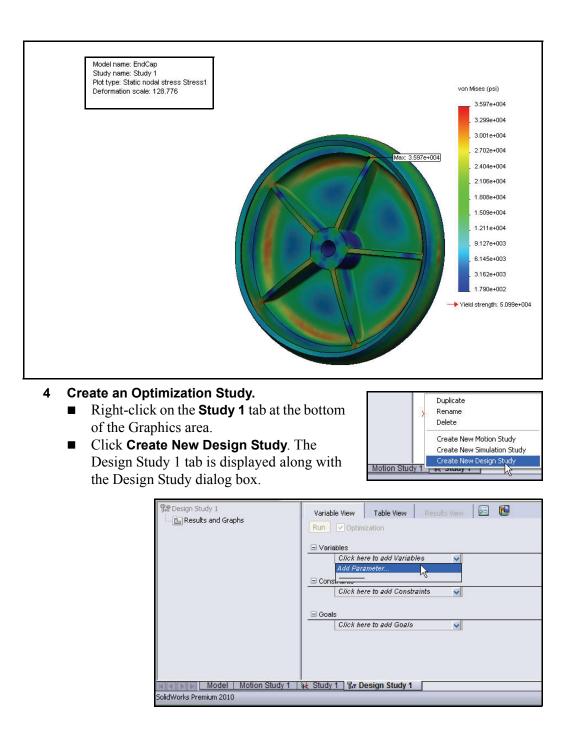
SolidWorks Simulation Professional

Creating an Optimization Analysis

- 1 Open the Part.
 - Click **Open** *i* 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 Might 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.



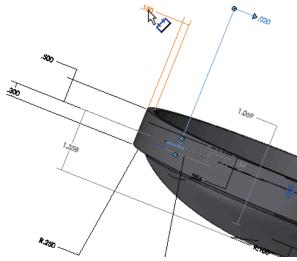
SolidWorks Simulation



- **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.

SolidWorks Simulation Professional



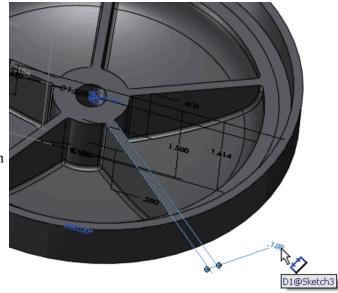


- Click the .150 EndCap Lip thickness dimension as illustrated. The selected dimension is displayed in the Add Parameters dialog box.
- Enter EndCapThickness for Name.
- Click Apply. The information is added to the Parameters dialog box.

Name:	EndCapThickness		
Comment (optional):			
Filter:	Model dimensions	~	
Туре:	Linear Dimension	*	
User defined value:	0.15 in		1
Model dimension:	D8@Sketch2@EndCap.Part		

SolidWorks Simulation Professional

- 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.



Name:	RibThickness			
Comment (optional):				
Filter:	Model dimensions		*	
Туре:	Linear Dimension		~	
User defined value:	0.15	in		18
Model dimension:	D1@Sketch3@EndCa	ap.Part		

Name	Туре	Unit	User defined va	Current value	Express	Comment
EndCa		in	0.15	0.15	D8@Ske	
RibThi	Length/Di	lin	0.15	0.15	D1@Ske	

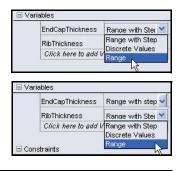
SolidWorks Simulation Professional

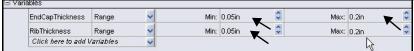
8 Expand the Variables cell in the Design Study.

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

Run Optimization			Total active	scenarios: 9						
EndCapThickness	Range with step 🗸	Min:	0.075in	*	Max:	0.225in	*	Step:	0.075in	
RibThickness	Range with step	Min:	0.075in	\$	Max:	0.225in	\$	Step:	0.075in	
	42									

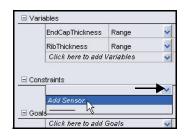
- 9 Set the ranges for the variables in the Design Study.
 - Select Range from the drop-down menu for EndCap Thickness.
 - Select Range from the drop-down menu for Rib Thickness.
 - Enter the illustrated numbers for the EndCap Thickness (Min: & Max:) range and the Rib Thickness (Min: & Max:) range.





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

 Click Add Sensor from the Constraint dropdown menu. The Sensor PropertyManager is displayed.



Optimization Analysis

SolidWorks Simulation Professional

SolidWorks Simulation

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



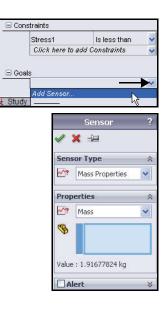
11 Set the conditions

- 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.



SolidWorks Simulation Professional

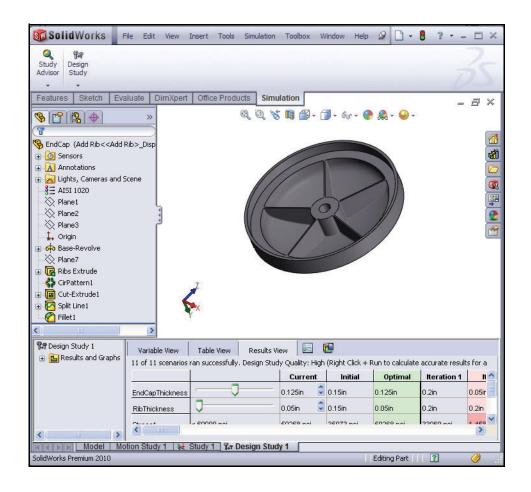
- 13 Set the condition for the Goal.
 - Select Minimize.

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.



Variable	View	Table \	/iew Result	s View	3
Rup [🛛 Optimiz	ation			
NC F	ibThickne	SS	Range	~	
	Click here	to add	Variables	~	
Constr	aints tress1		Is less than	~	
	Click here	to add	Constraints	~	
🖃 Goals					
ħ	lass1		Minimize	~	
	Click here	to add	Goals	~	



Optimization Analysis

SolidWorks Simulation Professional

SolidWorks Simulation

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.

		Current	Initia	Optimal
ndCapThickness	·	0.15in	0.15in	0.125in
RibThickness		0.15in	0.15in	0.05in
Stress1	< 60000 psi	35973 psi	35973 psi	59268 psi
/lass1	Minimize	1.91678 kg	1.91678 kg	1.70618 kg
		Current	Initial	Optirga
EndCapThickness		Current 0.125in	0.15in	Optima 0.125in
	0		A Constant	
EndCapThickness		0.125in	0.15in	0.125in

		Current
EndCapThickness		0.125in ᅌ
RibThickness		0.05in
Stress1	< 60000 psi	59268 psi
Mass1	Minimize	1.70618 kg

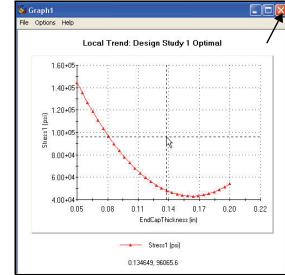
16 View the Trend Results.

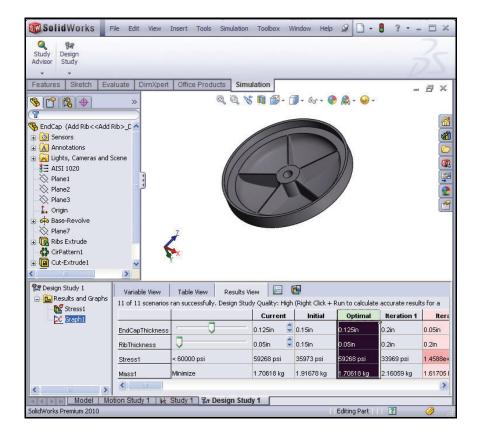
- Right-click the Results and Graphs folder.
- Click Define Local Trend Graph. The Local Trend PropertyManager is displayed. View your options.
- Click **Constraint**. Accept the default settings.
- Click OK from the PropertyManager. View the results in the Graphics area.

👫 Design Study		Va
Stri	Purge Results Define Design History	Graph
	Define Local Trend Gra	aph
- B	Local Trend Graph	2
De	X sign variables (X-Axis)	*
Y-	EndCapThickness	*
	Objective Constraint Stress1 Normalized to initial value	v e
Lo	cal trend at	*
	Optimal	~

SolidWorks Simulation Professional

- 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.



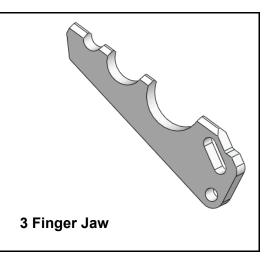


SolidWorks Simulation Professional

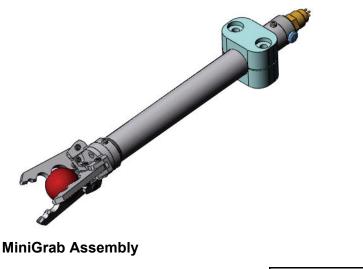
SolidWorks Simulation

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.



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.

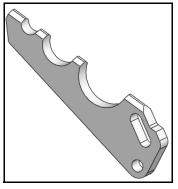




SolidWorks Simulation Professional

Creating a Fatigue Analysis

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



Design Study

4

Study Advisor

New Study

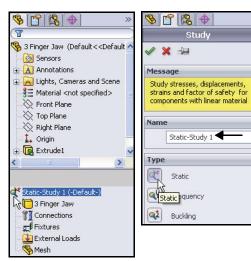
Study

Advisor

0

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.





SolidWorks Simulation Professional

SolidWorks Simulation

Applying Material

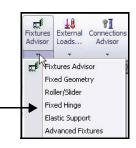
- 1 Apply Material.
 - Click Apply Material from the Simulation tab in the CommandManager. The Material dialog box is displayed.
- Study Advisor Advisor Fixtures Advisor Features Sketch Evaluate DimXpert Office P
- Expand the Aluminum Alloys folder.
- Click 6061-T6(SS) Alloy. View the material properties.
- Click Apply.
- Click **Close**. Material is applied to the part.
- **Note:** A green check mark so on the Parts folder indicates that material is assigned to the parts.

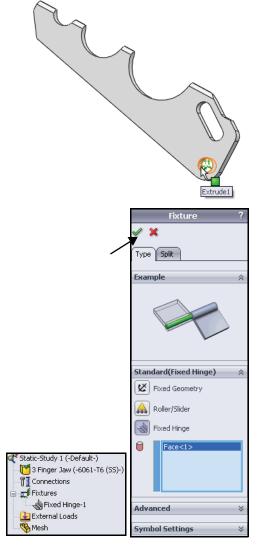
§Ξ 4032-T6	Properties	Tables & Curves	Appearance	CrossHatch	Custom	Application Dat	ta F
5052-H32	Material	properties					
5052-H34		als in the default lit	arary can not b	e edited. You	must first	copy the materia	alto
5052-H36		om library to edit it					
5052-H38	Model	Turnet	ar Elastic Isotro				
5052-H38, Rod (SS)	Model	Lines	ar Elastic Isotro	hir 🛛			
i052-O	Units:	SI - 1	V/m^2 (Pa)	*	1		
052-0, Rod (SS)	Catego	anu Alun	ninium Alloys		1		
086-H32, Rod (SS)	Cotogi	Pilon	in itom Pilloys				
5154-O, Rod (SS)	Name:	606	1-T6 (SS)				
5454-H111	Defaul	t failure	von Mises Stre				
5454-H112	criterio		vor (mises atre:	~ ^			
5454-H32	Descrip	ation:					
5454-H34							
5454-0	Source						
6061 Alloy	Durante		lean	ha			_
5061-O (55)	Property Elastic mo	al de la co	Value	Unit: S7e+010 N/m	57);		_
061-T4 (SS)	Poisson's		0.33	N/A	2		
061-T6 (SS)	Shear mo			13e+010 N/m	2		
06340	Mass den		2700	kg/m			
6063-Name : '6061-T6 (SS)' Description :	Tensile st	rength	31000002	.1 N/m	<u>^2</u>		
Appearance : 'satin finish aluminum'	Compress	sive Strength in X		N/m	<u>^2</u>		
6063 XHatch : 'ANSI38 (Aluminum)'	Yield stre	where the property of the second states are a second states	275000000		^2		
6063-T5		xpansion coefficie		K	10		
6063-T6		onductivity	166.9	VV/(t			
6063-T6, Rod (SS)	Specific h Material D	eat amping Ratio	896	J/(kj N/A	(N)		
6063-T83	Waterial D	amping Natio		N/A			
7050-173510	1						_
7050-T7451 🔍		Apply C	ose Sav	e Confi	a]	Help	
-							
				2	Static_St.	udy 1 (-Defau	(F_)
				-	100 C		
				100	<u> </u>	ger Jaw (-606	1-16
					Conr		
					Fixtu	ires	
					Exte	rnal Loads	
					🕵 Mest		

SolidWorks Simulation Professional

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.





Loads...

-

1

Fixtures Advisor

External Connection

Advisor

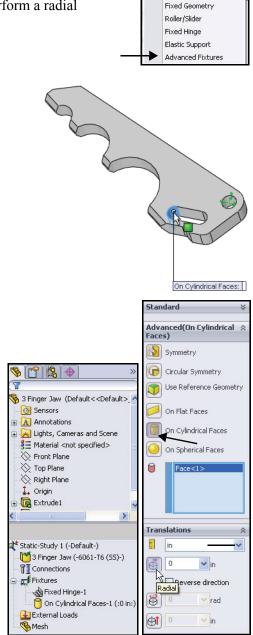
-

SolidWorks Simulation Professional

- 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.

Select the inside Cylindrical Face. 4

- 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.

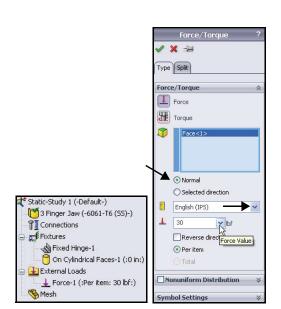
2 Select the contact face.

- 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.



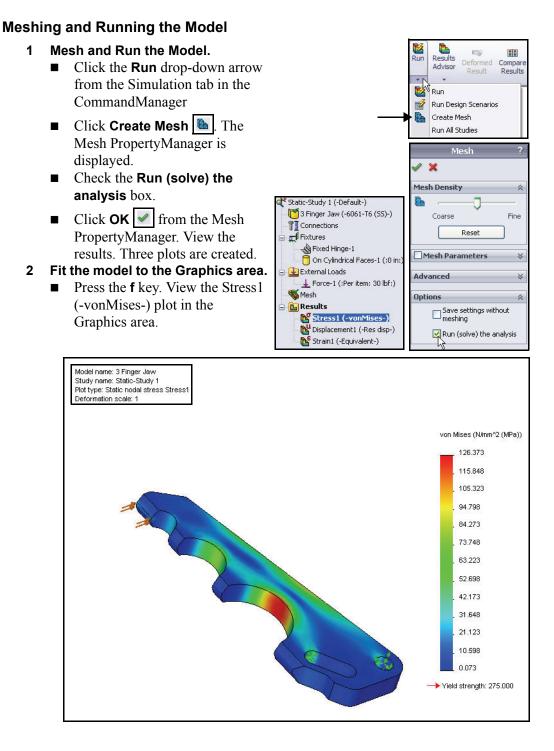
3 Set Units and Value.

- 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.



SolidWorks Simulation Professional

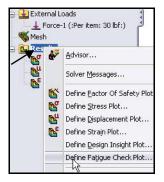
SolidWorks Simulation

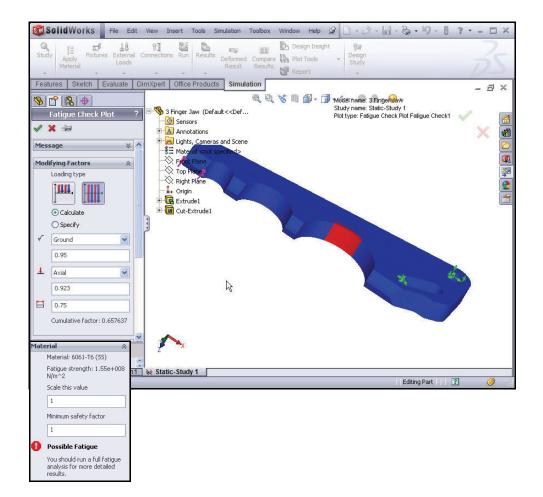


SolidWorks Simulation Professional

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.





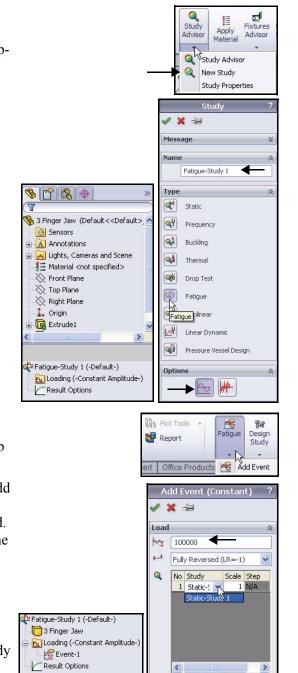
SolidWorks Simulation Professional

Creating a New Fatigue Study.

- 1 Create a New Fatigue Study.
 - Click the Study Advisor dropdown arrow from the Simulation tab in the CommandManager.
 - Click New Study . The Study PropertyManager is displayed.
 - Enter **Fatigue-Study 1** for Name.
 - Check Fatigue der 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 <a>[Main]
 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.



SolidWorks Simulation Professional

SolidWorks Simulation

- 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.

3 Finger	Тыл	
Evi		Apply/Edit Fatigue Data

3003-H14, Rod (S5) 3003-H16 3003-H16 3003-H18 3003-H18, Rod (S5) 3003-0, Rod (S5) 3004-0, Rod (S5) 3004-0, Rod (S5) 303-0 303-0 3004-0, Rod (S5) 305-0. F6 Permanent Mold cast (S5) 4032-16 5052-H32 5052-H34 5052-H34 5052-H34 5052-H35 5052-H36 5052-H37 5052-H38, Rod (S5) 5052-H36 5052-H36 5052-H37 5052-H38, Rod (S5) 5052-00, Rod (S5) 5052-00, Rod (S5) 5052-01, Rod (S5) 5052-02, Rod (S5) 5052-03, Rod (S5) 5052-04, Rod (S5) 5052-05, Rod (S5) 5052-06, Rod (S5) 5052-07, Rod (S5) 5052-07, Rod (S5) 5052-08, Rod (S5) 5052-09, Rod	Material		Σ
395.0-16 Permanent Mold cast (S5) 4032-16 5052-1132 5052-1132 5052-1136 5052-1138, Rod (S5) 5052-1138, Rod (S5) 5052-1138, Rod (S5) 5052-1138, Rod (S5) 5052-00, Rod (S5) 5052-00, Rod (S5) 5052-1131 5052-1132 5052-1138, Rod (S5) 5052-00, Rod (S5) 5054-111 5154-00, Rod (S5)	 3003-H14, Rod (S5) 3003-H16 3003-H18 3003-H18, Rod (S5) 3003-H18, Rod (S5) 3003-O, Rod (S5) 3003-O, Rod (S5) 	Source Interpolate: Log-log O Define: Curve-0(R=-1) O Derive from material Elastic Modulus:	view
Points A B 5052-H34 100 6380H30.21 5052-H38 5052-H38, Rod (S5) 5052-H38, Rod (S5) 3500 5052-H38, Rod (S5) 3500 5052-H38, Rod (S5) 52000 5052-H38, Rod (S5) 55154-0, Rod (S5) 5554-H31 5454-H31 5545-H34 5454-H34	356.0-T6 Permanent Mold cast (55)		^2
\$ \$454+1111 \$ \$454+112 \$ \$454+112 \$ \$454+132 \$ \$454+134 \$ \$454+0 \$ \$454+0	S052-H34 S052-H36 S052-H38 S052-H38, Rod (S5) S052-H38, Rod (S5) S052-O, Rod (S5) S052-O, Rod (S5)	1 100 836380430.21 2 200 490070876.14 3 500 360044006.04 4 1000 290141345.57 5 2000 236501769.08 6 5000 185300355.16	File View Save
6061-0 (55) 6661-74 (55)	\$ \$454-H111 \$ \$454-H112 \$ \$454-H12 \$ \$454-H32 \$ \$454-H34 \$ \$454-H34 \$ \$ \$454-H34 \$ \$ \$454-H34 \$ \$ \$ \$454-H34 \$ \$ \$ \$454-H34 \$ \$ \$ \$454-H34 \$	Source:	

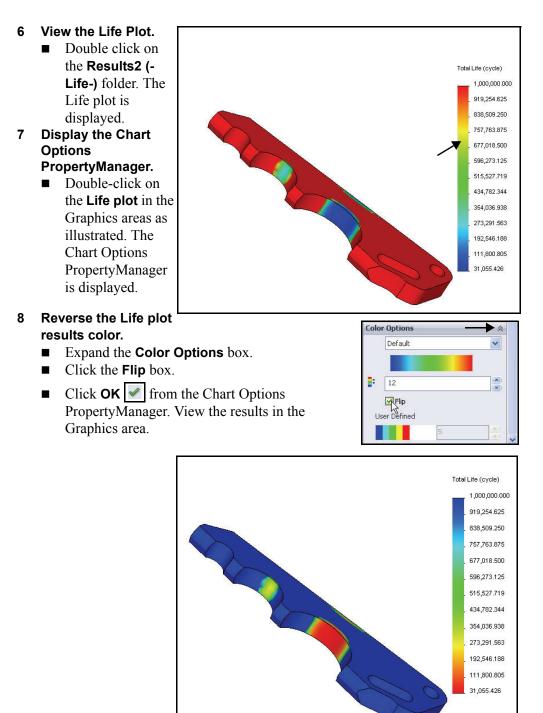
5 Run the Study.

- Click Run Manager. 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.



SolidWorks Simulation Professional

SolidWorks Simulation

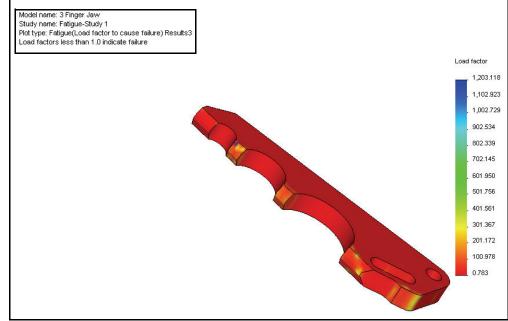


SolidWorks Simulation Professional

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.



SolidWorks Simulation Professional Conclusion

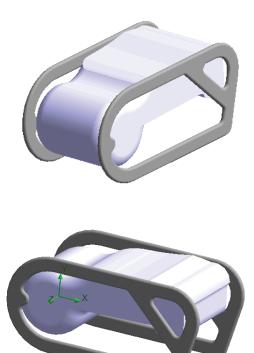
SolidWorks Flow Simulation

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.



SolidWorks Flow Simulation

Starting a SolidWorks Flow Simulation Session

- 1 Open the SeaBotix LBV150 Assembly.
 - Click **Open** if 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.



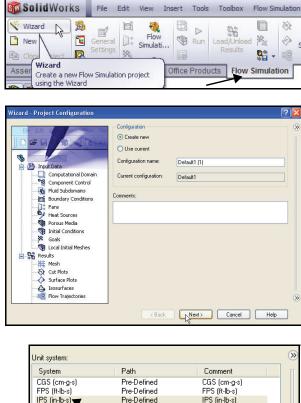


SolidWorks Flow Simulation

SolidWorks Simulation

Flow Simulatio

- Start the SolidWorks Flow 3 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 dropdown menu as illustrated.
 - Click **Next>**. The Wizard -Analysis Type dialog box is displayed.



System	Path		Comm	nent	
CGS (cm·g·s)	Pre-Defi			cm-g-s)	
FPS (ft-lb-s)	Pre-Defi		FPS (f		
IPS (in-lb-s)	Pre-Defi		IPS (ir		
NMM (mm-g-s)	Pre-Defi			(mm-g-s)	
SI (m-kg-s) USA	 Pre-Defin Pre-Defin 		SI (m-ł USA	(g-s)	
USA	Fie-Dell	nea	USA		
Create new	Name:	IPS (ir	Hb-s) (modified)		
Parameter		Units	Decimal Places	1.0 Unit SI =	^
🛛 Main					
Pressure & stress		lbf/in^2	4	0.000145037	
Velocity		Y	0	1.94384449	
Mass		Meter/	econd	2.20462248	
Length		Kilome	ter/hour	39.3700787	
Temperature		Mile/ho	ur	-459.67	
Physical time		Knot 🛛		1	
E Geometrical Character		Foot/s		1	
	ISUC	Inch/s			
E Loads&Motion		Yard/s			×
<			eter/second ter/second	>	
		Foot/m			
< Back	. Ne	Custor		Help	
		000101			_

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

Internal External Exclude internal space Physical Features Value Heat conduction in solids Radiation Time-dependent Gravity Rotation Rotation Reference axis: X Cancel Help	Analysis type	Consider closed cavities
Physical Features Value Heat conduction in solids Radiation Radiation Time-dependent Gravity Rotation Rotation Dependency	◯ Internal	Exclude cavities without flow conditions
Heat conduction in solids Radiation Time-dependent Gravity Rotation	External	Exclude internal space
Radiation Time-dependent Gravity Rotation Reference axis: X Dependency Image: State of the state	Physical Features	Value
Time-dependent	Heat conduction in	solids
Gravity Rotation		
Rotation		
Reference axis: X V Dependency 3	-	
	Rotation	
	Reference axis: X	V Dependency (»
<pre></pre>		
	< Back	Next > Cancel Help

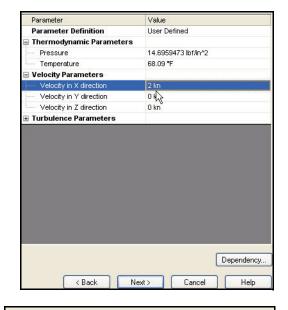
- Expand the **Liquids** folder.
- Click Water.
- Click the Add button. Water is displayed in the Project Fluids box.
- Click Next>. The Wizard -Wall Conditions dialog box is displayed. Accept the default settings.
- Click Next>. The Wizard -Initial and Ambient Conditions dialog box is displayed.

Fluids	Path	^	New
Propane	Pre-Defined		
R123	Pre-Defined		
R134a	Pre-Defined		
R22	Pre-Defined		
RC318	Pre-Defined		
Water	Pre-Defined		
∃ Non-Newtonian Liquids			
+ Compres Water iquids			
± Real Gases			
+ Steam			

Project Fluids	Default Fluid	Remove
Water(Liquids)		
Flow Characteristic	Value	Т
Flow type	Laminar and Turbulent	-
Flow type		

SolidWorks Flow Simulation

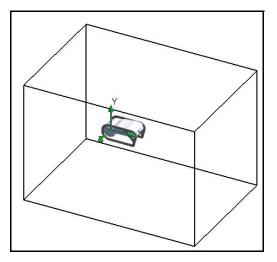
- Double-click inside the Value box of Velocity in X direction as illustrated.
- Enter **2** Kn for Velocity.
- Click Next>. The Wizard -Results and Geometry Resolution dialog box is displayed.
- **Note:** Two knots is the operating speed.
 - 4 Complete the SolidWorks Flow Simulation Wizard.
 - Accept all default settings. Click the **Finish** button.



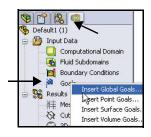
Result re	solution							<u>ا</u> 📎		
1	2	3	4	5	6	7	8			
- Minimum	- · · ·									
Manual specification of the minimum gap size Minimum gap size refers to the feature dimension Minimum gap size:										
							*			
Minimum wall thickness										
Minimum wall thickness refers to the feature dimension										
							*			
Advanced narrow channel refinement I Optimize thin walls resolution										
	< Ba	ack	Finish		Cancel		Help]		

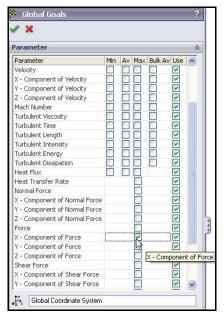
SolidWorks Flow Simulation

- 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 stab.
 - 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.





SolidWorks Flow Simulation

SolidWorks Simulation

3

- 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,

Run

Startup

Mesh

Solve

Use

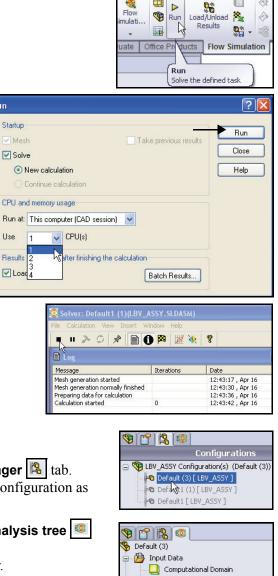
Besults 2

Load 4

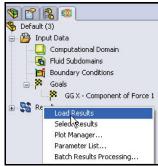
- Click the **Run** button.
- Note: To save classroom time, we will stop the analysis and open the Results folder to review completed results.
 - Stop the Analysis. 8
 - Click **Stop** from the Solver box as illustrated.
 - Click **No** to the question, "Do you want to save the results?"
 - Click File, Close from the Solver Main menu.

Open the Configuration 9 with the Solved Results.

- Click the **ConfigurationManager k** 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.



1



SolidWorks Flow Simulation

🛄 Computational Domain 🚯 Fluid Subdomains 📷 Boundary Conditions

Insert.

🖗 GG X - Component of Force 1

Load Results

My Recent Documents

Look in: 🔁 3

🚞 \$results_tmp article_study

> 🌚 😭 😫 🥯 😽 Default (3) 🚊 **)** Input Data

> > . 8 Goals

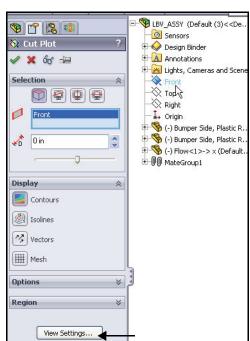
👫 Results 📫 Mesh 💸 Cut 🍟

🔘 3D-F 🞸 Surface Plets 실 Isosurfaces Flow Trajectories

- 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.



Click the **View Settings** button in the Cut Plot PropertyManager. The View Settings dialog box is displayed.



SolidWorks Flow Simulation

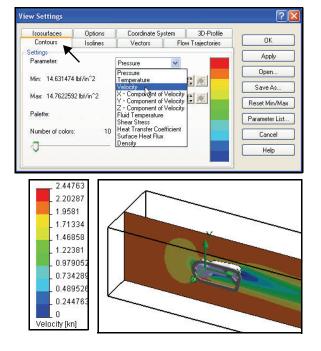
SolidWorks Simulation

- 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.

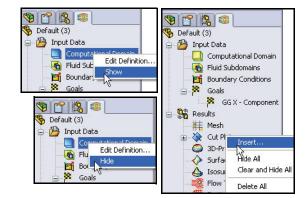
■ Click **OK** from the Cut Plot

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.



SolidWorks Flow Simulation

Sensors

🗄 🤯 Design Binder

🗄 🔝 Annotations

Kront

🗼 Origin

💱 LBV_ASSY (Default (3)<<De..

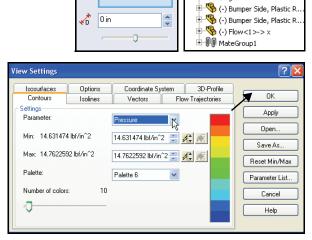
🗄 🚂 Lights, Cameras and Scene

15 Change the Selected Plane.

- Expand **LBV_Assy** from the flyout 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.



🧐 😭 😵

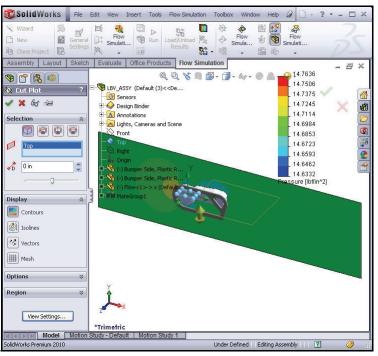
/ 🗙 68 💷

e 🖗 🖶

🐼 Cut Plot 🛛

Selection

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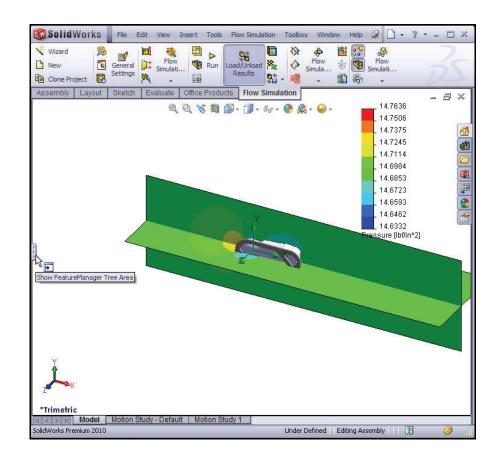


SolidWorks Flow Simulation

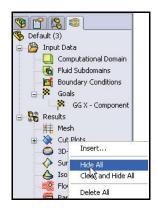
SolidWorks Simulation

- 17 View the Second Section Plot.
 - Click **OK** ✓ from the Cut Plot PropertyManager.

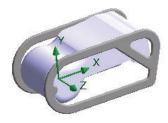
Note: Click the FeatureManager tree tab as illustrated to view the full Graphics area.



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



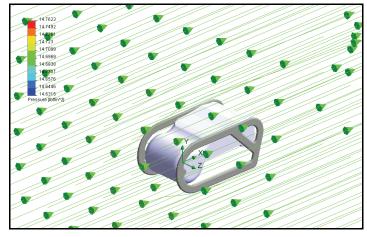


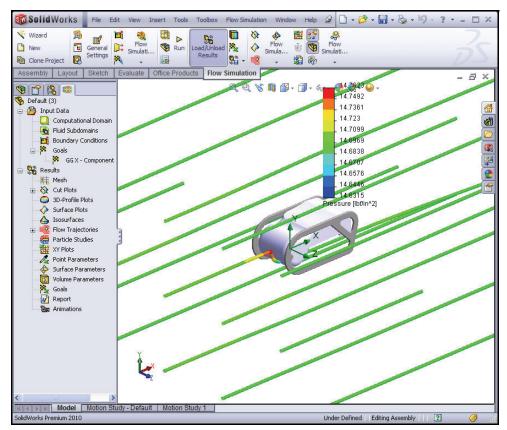


SolidWorks Flow Simulation

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.



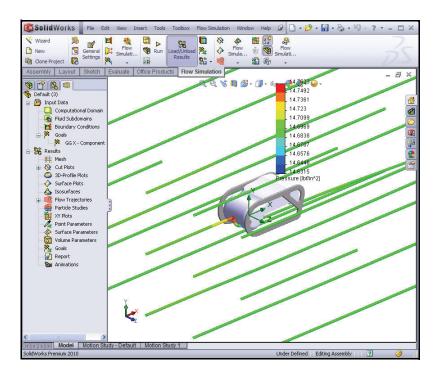


SolidWorks Flow Simulation

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.





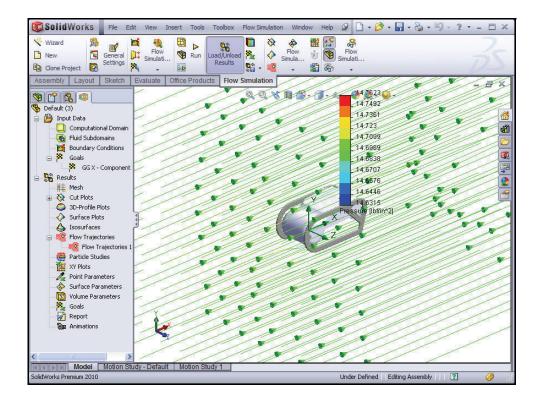




- 2 Edit the Flow Trajectory.
 - Expand the **Flow Trajectories** folder.
 - Right-click Flow Trajectories 1.
 - Click Edit Definition. The Flow Trajectories PropertyManager is displayed.
 - Pin He Flow Trajectories PropertyManager.
 - Enter **100** for the Number of trajectories as illustrated.
 - Click 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.



SolidWorks Flow Simulation

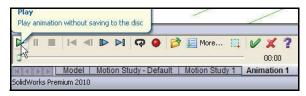


3 Animate the Flow Trajectory study.

- 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.



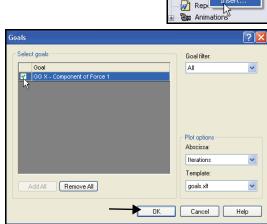


Applying Flow Trajectories

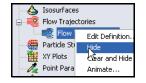
SolidWorks Flow Simulation

SolidWorks Simulation

- 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.



G	palName	✓ f Goal Name	<i>x</i> .									
Ţ	A	В	С	D	E							
1		LBV_ASSY.	SLD	ASM [De	fault (3)]							
2			_									
3		Goal Name	Unit	Value	Averaged Value							
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8	Iterations: 54											
9	Analysis interval: 24											
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11 12 13												
14 4	H → H Summary / X - Component of Force / Plot Data /											
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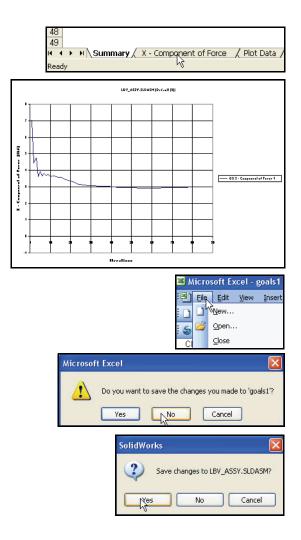
SolidWorks Flow Simulation

SolidWorks Simulation

- 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.

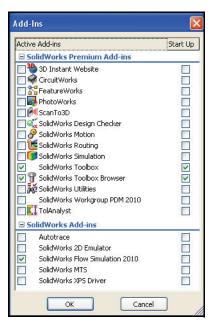


SolidWorks Flow Simulation

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.



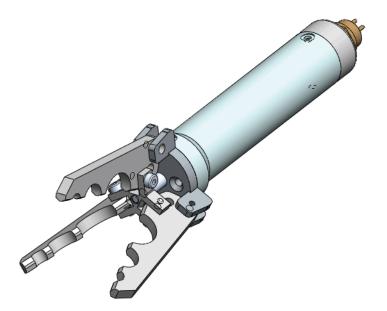
SolidWorks Motion

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.



SolidWorks Motion

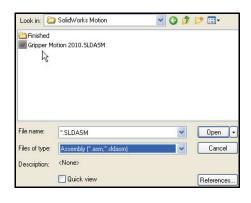
Gripper Motion 2009

Starting a SolidWorks Motion Session

- 1 Open the Gripper Assembly.
 - Click **Open** [2] 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.

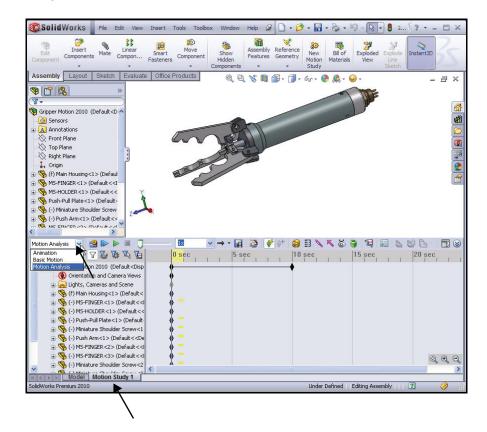


Options.



SolidWorks Motion

- 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.



SolidWorks Motion

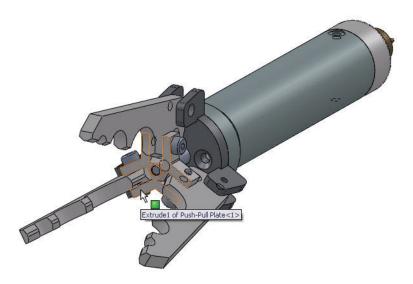
SolidWorks Motion

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.

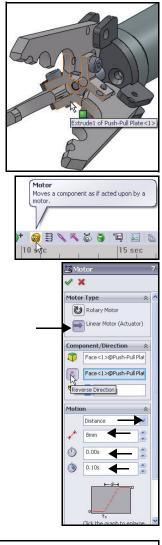


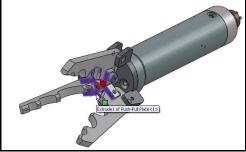


SolidWorks Simulation

Applying Linear Motion

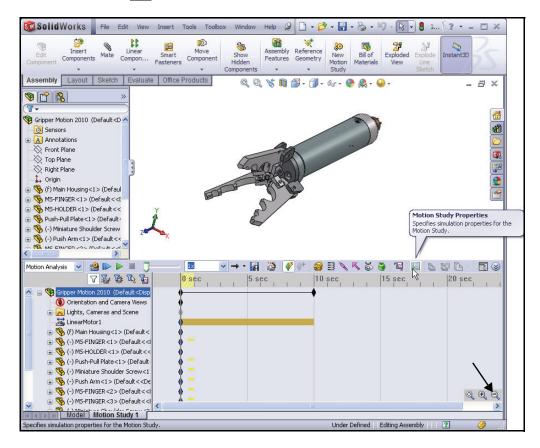
- 4 Apply Linear Motor.
 - Zoom in on the Push-Pull Plate component.
 - Click the Push-Pull Plate component face of the Gripper assembly as illustrated.
- **Note:** View the icon symbol and information feedback.
 - Click the Motor icon from the Motion Manager toolbar. The Motor PropertyManager is displayed.
 - Click the Linear Motor
 (Actuator) box for Motor Type.
 - Click the Reverse Direction button. The direction arrow points inwards.
 - Select **Distance** for the dropdown menu for Motion Type.
 - Enter 8mm 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.





Applying Motion to a Component

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



Applying Motion to a Component

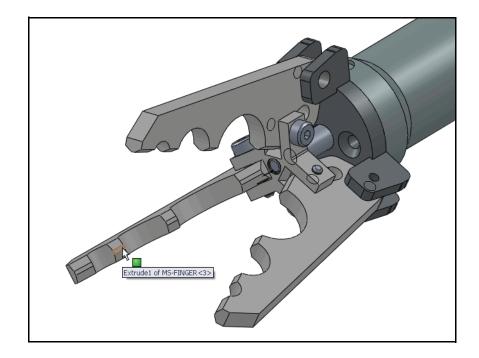
SolidWorks Motion

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.



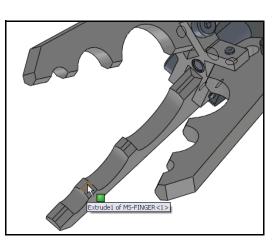
SolidWorks Motion

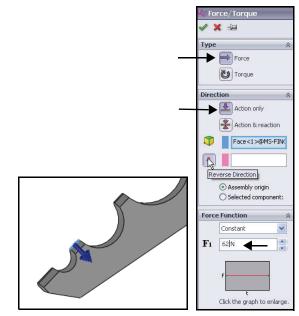
Applying Force to the Gripper Fingers

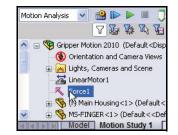
- 5 Select a Contact Face.
 - Rotate the Gripper assembly with the middle mouse button to view the inside faces of a finger as illustrated.
 - Zoom in to selected the first contact face.
- **Note:** Select any of the 3 Gripper fingers.
 - Click the contact finger face as illustrated.

6 Apply the Force.

- Click the Force Sicon 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.





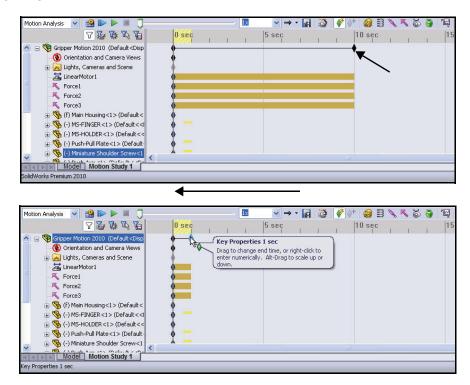


- 7 Apply a Contact Force to the two other Fingers.
 - Repeat Steps 5 and 6 for the other two Gripper fingers. At the end of this step, you should view three Forces and a LinearMotor in the Motion Study FeatureManager as illustrated.

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🔺 😑 🎨 Gripper Motion 2010 (Default <disp< td=""><td></td><td>1</td><td>1</td><td>-</td><td>- 3</td><td>16</td><td>12</td><td>÷</td><td>12</td><td>~</td><td></td><td>1</td></disp<>		1	1	-	- 3	16	12	÷	12	~		1
- 🛞 Orientation and Camera Views												
🕀 🙀 Lights, Cameras and Scene	•											
🛛 😹 LinearMotor1	0											
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🛓 % (f) Main Housing<1> (Default<												
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Model Motion Study 1							100					_
olidWorks Premium 2010												_

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.



SolidWorks Simulation

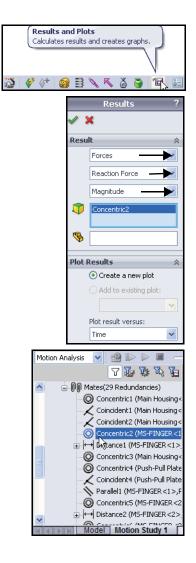
9 Run the SolidWorks Motion Simulation.

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

10 Calculate the Reaction Force at the Finger Hinge.

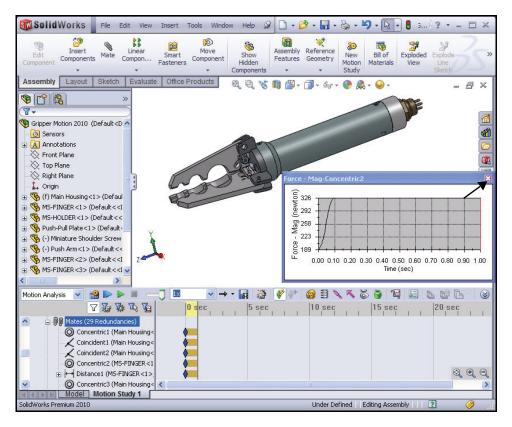
- Click the Results and Plots icon from the Motion Study toolbar. The Results PropertyManager is displayed.
- Select Forces from the Result drop-down menu.
- Select Reaction Force from the Result drop-down menu as a sub-category.
- Select Magnitude from the Result dropdown menu as the Result component.
- 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.





SolidWorks Motion

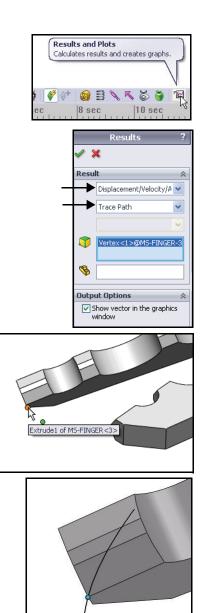
- Click along the **time axis** and view the changes in the Gripper.
- **Close** the Force Mag-Concentric2 plot dialog box.



SolidWorks Simulation

11 Create a Trace Path.

- Click on the Results and Plots I 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.



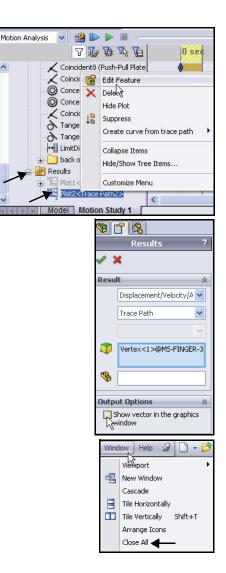
SolidWorks Motion

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** Given a 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 physicsbased 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.

SolidWorks Simulation

SolidWorks Motion Conclusion