SolidWorks[®] 2013

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

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SolidWorks Simulation

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.

Introduction

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:

- SeaBotix LBV150 assembly
- Housing assembly
- MiniGrab assembly
- EndCap part
- 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.

Hands on Test Drive

Notes:

SolidWorks Simulation

Notes:

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 2013 (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, **File Properties** - Shows the summary information of the active document, **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 *solution* option displays both the Menu Bar toolbar and the Menu Bar menu.

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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:

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

FeatureManager Design Tree

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

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

FeatureManager (%), PropertyManager (1), ConfigurationManager (%), DimXpertManager (1) and DisplayManager (1).



Ctrl+R

User Interface

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 the Motion 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: Check Use SolidWorks Web Help for internet access.



Hands on Test Drive

SolidWorks Simulation

Getting SolidWorks Simulation Help

Click Study Advisor,

Study Advisor from the Simulation tab in the

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Study Advisor	Apply Material	Fixtures Advisor	External Loads	Connectio Advisor	ons Run	Results Advisor	Deformed Result	Compare Results		Plo
-				*		1				
🔍 St	udy Advis	or	te D	imXpert	Office P	roducts	Simulati	on		
	ew Study					-/	•			

CommandManager with an active study to obtain the Simulation Advisor.

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

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

Note: The Simulation

Advisor ***** tab is displayed in the Task Pane.



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 Examples to view whats new in SolidWorks 2013.



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 Simulation

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, ConfigurationManager 😫

tab, and the DisplayManager (a) 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 and SolidWorks Simulation



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.



Starting a SolidWorks Session

- 1 Start a SolidWorks Session.
 - Click the **Start** menu.
 - Click All Programs, SolidWorks 2013, SolidWorks 2013.
- **Note:** You can quickly start a SolidWorks 2013 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** A 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.

🛱 Open	Simulation Tes SolidWorks Simul	ation
Coganize New foldeworks Southers Coganize New folder Costap Downloads Downloads Recent Places Downloads Downloads Consents Music Pictures Vices	Simulation Fe., • SolidWorks Simulation Documents library SolidWorks Simulation Name	ation → 1+3 Sector Social/Views Simulation → III → III → III → III → III → Arrange by: Folder → Date ms 60/2/20 10/13/2
Computer Computer Configurat Driptay St	Metalved Resolved Code: Resolved	Do not lead hidden components Ute Spedepak Quick Riter: Specific III Asembly ("arror,"sdarm) v





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 * S Dr Customize... Add-Ins...



- 4 Activate SolidWorks Simulation.
 - Click the Options arrow drop-down arrow as illustrated from the Menu bar toolbar.
 - Click Add-Ins. The Add-Ins dialog box is displayed.
 - Check the SolidWorks Simulation box.
 - Click **OK** from the Add-Ins dialog box.
- **Note:** Displayed Add-Ins may vary per system setup.

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



5 Set Default Options in SolidWorks Simulation.

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



SolidWorks Simulation

- 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/mm²(MPa) for Pressure/Stress.
- 6 Set Results Folder.
 - Click the Results folder.
 - Click the Automatic box in the Default solver section.
- 7 Set Number Format.
 - Click the Color Chart folder.
 - Click **Floating** for Number format. View your options.
 - Click **OK** from the Default Options Plot Color Chart dialog box.

Plot1 User defined	ystem Options Default Options Units Units Mesh Results Plot Solor Chart Default Plots Plot2 Plot2 Plot2 Plot1 Plot2 Plot3 Plot1 Plot2 Plot3 Plot1 Plot2 Plot2 Plot3 Plot2 Plot2 Plot3 Plot2 Plot	 ✓ Display color charts ✓ Display plot details Position ● Predefined positions ● User defined Horizontal from left: 80 2 2 2 2 2 2 ✓ Vertical from top: 20 2 2 2 2 ✓ Vidth ● Vide ● Normal ● Trin Number format ● Scientific ● Floating ● Vide ● Normal ● Trin Number format ● Scientific ● Floating ● Vide ● Loss repearator (.) Color options ● Default ● Flip User defined
User information	User information	5 -





Unit system

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.

Note: The 2D Simplification option creates a 2D simplification study to simplify certain 3D models by simulating them in 2D and saving analysis time. Available analysis types include plane stress, plane strain, extruded and axisymmetric.

SolidWorks Simulation



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Message

Name

Туре

Study

Study stresses, displacements, strains and

factor of safety for components with linear material

Study 1

Static

SolidWorks Simulation

Creating a Static Analysis Study

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

os so	LIDWOR	KS	File	Edit	View	Inse	rt T	iools S	imulation	Window
Study Advisor	₿≣ Apply Material	हूर्ण Fixture Adviso	s Ext r Loa	ternal ads	Connec Advis	tions	Run	Results Advisor	Deformed Result	II Comp Resu
St St	udy Advis ew Study	sor		n E	valuate	Of	ice P	roducts	Simulat	ion

- 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. View the various available Study types.
- Click the **Static ≤** button for Type.



- Click OK from the Study PropertyManager. Study 1 (-Simulation_Original_Design-) is displayed. View the default folders.
- 3 Return to the FeatureManager.
 - Click the FeatureManager 🛐 tab.
- **Note:** A green check mark **on** a Study folder indicates that material is assigned.
- Note: If needed, return to the FeatureManager.



Assigning Materials in SolidWorks Simulation

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

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

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



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

Works Materials	 Properties 	Tables & Curves	Appearance (CrossHatch	Custom	Application
Steel	Material	properties				
1023 Carbon Steel Sheet (SS)	Materi	als in the default lib	ary can not be	edited. You r	nust first	copy the mat
201 Annealed Stainless Steel (SS)	a cust	om library to edit it.				
A286 Iron Base Superalloy	Model	Tuper	Elastic Testroniu	9		
AISI 1010 Steel, hot rolled bar	Moder	Linear	Elasue Isou opio			
AISI 1015 Steel, Cold Drawn (SS)	Units:	SI - N	/m^2 (Pa)		·	
151,1020	Categ	Steel			1	
ISI 1920 Steel, Cold Rolled	E	ory.				
ISI 1035 Steel (SS)	Name	AISI	1020			
ISI 1045 Steel, cold drawn	Defau	It failure	on Misson Strong			
SI 304	criterio	on:	on Mises Stress			
I 316 Annealed Stainless Steel Bar (SS	Descri	ption:				
SI 316 Stainless Steel Sheet (SS)	Counce					
I 321 Annealed Stainless Steel (SS)	Source					
SI 347 Annealed Stainless Steel (SS)	Sustai	nability: Defin	ed			
ISI 4130 Steel, annealed at 865C						
ISI 4130 Steel, normalized at 870C	Property	5.0	Value	Units		
ISI 4340 Steel, annealed	Elastic M	lodulus	2e+011	N/m^2		
SI 4340 Steel, normalized	Poissons	s Ratio	0.29	N/A		
SI Type 316L stainless steel	Shear M	odulus	7.7e+010	N/m^2		
I Type A2 Tool Steel	Density		7900	kg/m^3		
oy Steel	Tensile 9	Strength	420507000	N/m^2		
y Steel (SS)	Compres	ssive Strength in X		N/m^2		
TM A36 Steel	Yield Str	ength	351571000	N/m^2		
ast Alloy Steel	Thermal	Expansion Coefficie	1.5e-005	/K		
ast Carbon Steel	Specific	Heat	47	U(ka·K)		
Cast Stainless Steel	Material	Damping Ratio	720	N/A		
Chrome Stainless Steel			/			

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** II 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.







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.







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



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.
- Apply the Fixture. 5
 - Click **OK** ✓ from the Fixture PropertyManager. An icon 📴 named On Cylindrical Faces-1 is displayed in the Fixtures folder.

Translations

0 **1**

0

0

1 mm

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.







Clear Selections

billete

Applying Loads
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.

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 🕮 named Pressure-1 in the External Loads folder as illustrated.

Fit the model to the Graphics area. 7

- Press the **f** key. View the model in the Graphics area.
- **Note:** If you change 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

- Create a Compatible Mesh. 1
 - Expand the **Component Contact** folder from the Study tree.
 - Right-click Global Contact (-Bonded-).
 - Click Edit Definition. The Component Contact PropertyManager is displayed. View your options.
 - 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.



Creating a Mesh and Running the Analysis

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 h. 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.
- Click the **Standard mesh** box.
- Expand the Advanced box. View the available advanced options for additional control.

Connections Advisor	Run	Results Advisor Result
valuate Off		Run Run Design Scenarios Create Mesh Run All Studies



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 <u>s</u> is applied next to the Mesh folder in the Study.

- Note: Right-click Mesh. Click Hide 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 Minimum 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 mesh type, and system setup.





SolidWorks Simulation

View the Results

- 1 Hide all symbols.
 - Click the drop-down menu from the
 Hide/ Show items for tool in the
 Heads-up toolbar as illustrated.
 - Click the View Simulation Symbol icon to hide all symbols.
- 2 View the von Mises Stresses.
 - Double-click Stress1 (-von Mises-). The Stress Plot PropertyManager is displayed. Plot units if needed can be modified from the PropertyManager.
 - Click **OK** from the Stress Plot PropertyManager.



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



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

3 Display a Section View using the Top Plane.

- Click the SolidWorks FeatureManager 1 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.









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 tool located in the Heads-up View toolbar to Zoom in on a section of the model.

4 Display an Isometric view.

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



5 Probe the Model.

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





Some to Area
 Zooms to the area you select with a
 bounding box.
 Some to Area
 Zooms to the area you select with a
 bounding box.
 Some to Area
 Some to Area

SolidWorks Simulation

Report Options

Annotations

a G 🕠

- 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.



6 Review the Plot.

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



Viewing the Results

- 9 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.
- 10 Fit the model to the Graphics area.
 - Press the f key. View the results in the Graphics area.





- 11 View the Displacement Plot.
 - Double-click
 Displacement1 (-Res disp-) in the Results folder. View the plot.



SolidWorks Simulation

12 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.

13 Stop the Animation.

Click Stop

14 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.

15 Calculate the Factor of Safety.

- Right-click the Results folder.
- Click the **Define Factor Of**

Safety Plot Stool. The Factor of Safety PropertyManager is displayed.

- Select the **Selected bodies** button.
- Click the first CH End Cap component in the graphics area.
- Select Max von Mises Stress from the drop-down menu as Criterion. Note your options for Criterion.



- Click Next Stock to continue to step 2. Accept the defaults.
- Click Next Stochastic 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

- 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 approximately 0.66. 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.

🎕 😫 🤗	
Chart Options	?
✓ ×	
Display Options	~
Show min annotation	
Show max annotation	
Show plot details	
Show Min/Max range of shown parts only	n
Position/Format	*



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 self-viewing, small, and hence convenient to send via email.

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Favorites	Name	Date modified Typ	e Size		
Desktop		No items match yo	ur search.		
bownloads					
E Recent Places					
1 through					
Documents					
a) Music					
Pictures					
🔣 Videos					
🗳 Homegroup					
Computer					
Local Disk (C:)					
File name:	_ASSY-Study 1-Results-Stress1.an	alysis			
Save as type: eDra	wings Files (*.analysis.easm)				



Creating a SolidWorks eDrawings File

SolidWorks Simulation

🛃 External Loads

Creating a SolidWorks eDrawings file

- Create a SolidWorks eDrawings file. 1
 - 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** ¹ . The Save As box is displayed.
 - Select eDrawings Files for Save as type. Accept the default name and location.
- Mesh Result Options 🕒 Results 💕 Stress1 (-vonMises-) 階 Displacement1 (-Res di: New Strain1 (-Equivalent-) 🌃 Factor of Safety1 (-Ma> 눩 Design Insight Plot Tools Section Clipping 6 Iso Clipping Þ Probe Z List Selected 6 Save As

<u> H</u> Pressure-1 (:1500 psi:)

Click Save.



2 Publish a SolidWorks eDrawing.

Click File, Publish to eDrawings

> from the Menu bar menu. The Save Configurations to eDrawings file dialog box is displayed.



- Accept the default settings. Click OK from the dialog box. If needed click Yes to the dialog box. View the eDrawing.
- Click **Play >**. View the eDrawing.
- Click Stop





- 3 View the Stress1 (-von Mises-) Plot.
 - Click File, Open from the Main menu in eDrawings.
 - Double-click the LBV-ASSY-Study 1 in the saved study folder. View the eDrawing for the von Mises Plot.
 - Click **Play >**. View the eDrawing.
 - Click Stop ■
 - Close the eDrawing and return to SolidWorks Simulation.
 - Click No. Do not save the eDrawing.





SolidWorks Simulation

Generating a Report

The Report utility generates an Internetready or Microsoft[®] Word document convenient for review by colleagues and

Deformed Result Results	1 Incluse Image for Report
Simulation 🛛 🍭 💥 🚺 🛱 - (Report Generates a Word report for the current analysis study.

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.



MS Word Format

SolidWorks Simulation

Generating a Static Study Report

- 1 Generate a Static Study Report.
 - Click Report Provide the Simulation tab in the CommandManager. If needed, click OK to the dialog box.
 - Enter Description: HOTD Report.
 - Check the **Designer** box.
 - Enter Name.
 - Check the **Company** box.
 - Enter Name.
 - Check the Show report on publish box. Accept the default settings.
 - Click the **Apply** button.
 - Click the Publish button.

Curren	nt report format: Static Study Format
Report se	ctions: Section properties
Description	Description:
Model Informa Study Properti Units Units Connector De Contact Inform Mesh Informat Sensor Details Header Informa	tion es E sties trues finitions nation tion tion
Designer:	John Smith
Company:	ABC
URL:	
Logo:	
Address:	
Phone:	Fax:
Report publish o	options
	C:\Users\h\Documents\Hands on Test Drive Simulation 2013\Soli
Report path:	
Report path: Document nam	e: LBV_ASSY-Study 1-1

Report

Design Insight

SolidWorks Simulation

- 2 View the Result.
 - 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.

	Simulation of
	LBV_ASSY
	Date: Monday, October 08, 2012 Designer: John Smith Study name: Study 1 Analysis type: Statio
	Table of Contents
	Description 1
	Assumptions
	Model Information
	Study Properties
	Units
	Material Properties
Description	Loads and Fixtures
HOTD Report	Connector Definitions
	Contact Information
	Mesh Information
	Sensor Details
	Resultant Forces
	Beams
	Study Results 11
	Conclusion 14



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 in class, 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.





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 to from the Context toolbar. The Housing with the ribbed EndCaps is displayed in the Graphics area. Both instances of this part are updated.
- Rotate the model with the middle mouse button to view the unsuppressed ribs.

3 Return to Study 2.

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



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

▼ A Study 2 (-Simulation_Origin ● Study 2 (-Simulation_Origin ● Study 2 (-Simulation_Origin ● Connections ● Component Contacts ● E Component Contacts ● Fixtures ● On Cylindrical Faces-1 (● External Loads ● Mesh

Result Options

5 Mesh the Model.

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

Rur	Results Advisor Result Result
	Run Run Design Scenarios Create Mesh Run All Studies
Simulation	X
A Remeshing will delete the re	sults for study: Study 2.
->	OK Cancel

SolidWorks Simulation

Mesh

Reset

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Fine

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Mesh Density

Coarse

mm

Advanced

Options

Mesh Parameters

Standard mesh
 Curvature based mesh

▲ 11.37962568 ★ ★
0.56898128m ★ ★
■ Automatic transition

Jacobian points 4 points
Draft Quality Mesh
Automatic trials for solid
Remesh failed parts with
incompatible mesh

Save settings without meshing Run (solve) the analysis

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

77- Study 2 (-Simulation_Original_I Parts Connections 🗄 🖶 Component Contacts Fixtures 📋 On Cylindrical Faces-1 (: 🛃 External Loads H Pressure-1 (:1500 psi:) Mesh Result Options 🗄 🔳 Report 📲 Stress1 (-vonMises-) 📲 Displacement1 (-Res dis National (-Equivalent-) 🕂 Factor of Safety1 (-Max

- 7 View the Results Folder.
 - Expand the **Results** folder.

SolidWorks Simulation

- 8 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. View the results in the Graphics area.
- **Note:** Results may vary depending on mesh type, system setup and system options.





Analysis 2 - Static Study 2

- 9 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 1.12.

SolidWorks Simulation



10 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 Factor of Safety PropertyManager. The two components are hidden in the Graphics area. View the single CH End Cap.





Rotate the model and view the results.





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







- Click the **Exit Compare** button in the Compare Results dialog box.
- Click the Study 2 tab if needed. 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.



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 from the Context toolbar. The components are displayed in the Graphics area.

11 Return to Study 1.

- Click the Study 1 tab at the bottom of the Graphics area. Study 1 is displayed.
- Double-click Stress1 (-vonMises-) from the Results folder. View the Graphics area.
- Click **OK** from the Stress Plot 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.



Report	
Results	
- Contraction Stress St	
Displacement1 (-Res disp-)	
Strain1 (-Equivalent-)	
- 🌇 Factor of Safety1 (-Max von Mises	Stress-)
	a surgers
📑 🔪 💵 🦺 Design In	sight
Deformed Compare 🚺 Plot Tools	-
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Simulation	
Comnare Results	2
compare restarts	20
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plots of the same type:	cror
Shudu LuShocci	
DUUY 1::DUESS1	

SolidWorks Simulation

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



12 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.

Notes:

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 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.



Unit system	
SI (MKS)	
🔘 English (IPS)	
🔘 Metric (G)	
Units	
Length/Displacement:	mm 👻
Temperature:	Kelvin 👻
Angular velocity:	rad/sec 🔹
Pressure/Stress:	N/mm^2(MF ▼

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.



SolidWorks Simulation Professional

- 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.



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



Documents library

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SolidWorks Simulation Professional

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



- 4 Perform an Analysis on the Study.
 - Click **Run** if from the Simulation tab in the CommandManager. The analysis runs and three default plots are created.

5 View the Von Mises Stress on the EndCap.

- The plot is displayed in the Graphics area. Double-click Stress1 (-vonMises-). The Stress Plot PropertyManager is displayed. View your available options.
- Click OK from the Stress Plot PropertyManager.
- 6 Fit the model to the Graphics area.
 - Press the **f** key.
- **Tip:** To Zoom out, press the **z** key.
 - 7 Hide all symbols.
 - Click the drop-down menu from the Hide/ Show
 items for tool in the Heads-up toolbar as illustrated.
 - Click the View Simulation Symbol icon to hide all symbols.





Trend Tracker Analysis

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

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

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

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





SolidWorks Simulation

Design Insight



SolidWorks Simulation

8 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.



SolidWorks Simulation Professional

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



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- Right-click the **Trend Tracker** folder.
- Click Set Baseline.
- **Expand** the Trend Tracker folder. View the created graph icons.
- Mesh Trend Tracker Trend Journal Stress1 Result Options

Set Baseline

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.

SolidWorks Simulation

11 Add Sensors.

- Click the Model tab at the bottom of the Graphics area.
- Right-click the Sensors folder from the Assembly FeatureManager.
- Click Add Sensor. The Sensor PropertyManager is displayed.
- Select Simulation Data for Sensor type from the drop-down menu.
- Select N/mm²(MPa) 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.

12 Return to Trend Study.

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









Trend Tracker Analysis

SolidWorks Simulation Professional

- 13 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.



14 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 Premium 2013 x64 Edition

SolidWorks Simulation

- 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.

15 Return to the Trend Study.

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









16 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.

SolidWorks Simulation Professional

17 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.





18 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.



Trend Tracker Analysis

SolidWorks Simulation

19 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.

20 Save and Close the Model.

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



11	rend Jour	nal	
File Name: Study name: Description:	LBV_ASSY Trend Study		
Baseline Time Completed: Wednesday, Tracked Data:	September 26, 2012	2 9:53:35 AM	
Source	Туре	Actual Value	Normalized Value
Massl	Model Max	4.05904 (kg)	100
Stressl (VON: von Mises Stress)	Model Max	530.556 (N/mm^2 (MPa))	100
Displacement1 (URES: Resultant Displacement)	Model Max	4.42245 (mm)	100
Stress2 (VON: von Mises Stress)	Max over Selected Entities	530.556 (N/mm^2 (MPa))	100
Iteration 2 Time Completed: Wednesday,	September 26, 2012	10:02:21 AM	Normalized
Source	Туре	Actual Value	Value
Source Massl	Type Model Max	Actual Value 5.16175 (kg)	Value 127
Source Massl Stressl (VON: von Mises Stress)	Type Model Max Model Max	Actual Value 5.16175 (kg) 354.27 (N/mm*2 (MPa))	Value 127 66
Source Massi Stressi (VON: von Mises Stress) Displacement (URES: Resultant Displacement)	Type Model Max Model Max Model Max	Actual Value 5.16175 (kg) 354.27 (N/mm²2 (MPa)) 4.42189 (mm)	Value 127 66 99

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

Create the Thermal Analysis Study

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

open				
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Downloads Recent Places	Name			
Libraries	🌯 EndCap			
Documents	2			
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Configurati	ons: Default 👻		References	ł
19 ANSI			Quick Filter:	9 28 12
File nar	ne: EndCap	•	Part (*.prt;*.sldprt) Open	Cancel



2 Create a Thermal Study.

 Click the Simulation tab in the CommandManager.

	Study Adviso	or	aluate	DimXpert	Office P	roducts	Simulati	on
T	1 Parenter	*	-	-	-	-		
Stud Advis	iy sor Apply Material	Fixture Advisor	s Exter	nal Connecti	ons Run	Results Advisor	Deformed Result	Co

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

	Study	?
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Mess	age	*
Study heat conv	/ temperature distribution and flow due to conduction, ection and radiation	
Name	2	*
	Thermal-Study 1	
Туре		~
*	Static	
۹Y	Frequency	
٩\$	Buckling	
	Thermal	
	Drop Test	

Thermal Analysis

SolidWorks Simulation Professional

Applying the EndCap Material.

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

				Therr	mal-Sturn nd <u>Cap</u> ionnect hermal 1esh	dy 1 (-De ions Loads	fault-)
is soi	.I D WOR	KS	File Ed	dit Vi	iew I	nsert 1	Fools
Study Advisor	Apoly Matarial	Thermal Loads	P Conne Adv	ctions isor	Run	Results Advisor	Deform Resu
Feature	s i ket	ch Ev	aluate	Dim	Xpert	Office	Produc
§ 🖆	Apply	Materi	al rial to se	lected	Items		

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Steel	Material pro	perties				
1023 Carbon Steel Sheet (SS)	Materials i	in the default librar	y can not be	edited. You i	must first	copy the materi
201 Annealed Stainless Steel (SS)	a custom	ibrary to edit it.				
A286 Iron Base Superalloy	Model Typ	e: Linear F	lastic Isotropic		-	
1010 Steel, hot rolled bar						
1 1015 Steel, Cold Drawn (SS)	Units:	SI - N/m	^2 (Pa)	•	•	
51 1020	Category	Steel			1	
2 1020 Steel, Cold Rolled					_	
151 1035 Steel (55) =	Name:	AISI 10	20			
AISE 204	Default fa	ilure Max vor	Mises Stress	-	-	
316 Annealed Stainless Steel Bar (SS	criterion:					
316 Stainless Steel Sheet (SS)	Descriptio	n:				
321 Annealed Stainless Steel (SS)	Source:					
847 Annealed Stainless Steel (SS)						
130 Steel, annealed at 865C	Sustainab	ility: Defined				
4130 Steel, normalized at 870C			1			
I 4340 Steel, annealed	Property		Value	Units		
4340 Steel, normalized	Elastic Modu	ilus	2e+011	N/m^2		
Type 316L stainless steel	Poisson's Ra	lue	0.29 7.7e±010	N/A	_	
Type A2 Tool Steel	Density	iuo	7900	ko/m^3		
y Steel	Tensile Stre	ngth	420507000	N/m^2		
Steel (SS)	Compressiv	e Strength in X		N/m^2		
100.01	Yield Streng	th	351571000	N/m^2		
A36 Steel	Thermal Exp	ansion Coefficient	1.5e-005	/K		
Alloy Steel	Thermel Con	ductivity	47	W/(m·K)		
M A36 Steel t Alloy Steel t Carbon Steel	mermarcor	200 m m m m m 🖉 🗤				
M A36 Steel it Alloy Steel it Carbon Steel t Stainless Steel	Specific Hea	at Defin	420	J/(kg·K)		

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



Thermal Loads and Boundary Conditions

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

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

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

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

SolidWorks Simulation Professional

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 inside 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.







- 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.



SolidWorks Simulation Professional

Applying Convection

- 1 Apply Convection.
 - Click the Thermal Loads drop-down arrow from the Simulation tab in the CommandManager.
 - Click **Convection**. The Convection

PropertyManager is displayed.

2 Select the Exposed Faces.

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





3 Select the other three exposed outside Faces.

- Click the other three outside faces of the EndCap. Face<2>, Face<3>, and Face<4> are displayed in the Faces for Convection box. Rotate the model to select Face<4>.
- **Note:** Apply the **Zoom to Area** tool from the Heads-up View toolbar to select the correct faces.







- 4 Set Units and Value.
 - Select English (IPS) from the Units drop-down menu.
 - Enter **0.22** in the Convection Coefficient box.
 - Enter **50** in the Bulk Ambient Temperature box.
- **Note:** The inputs simulate seawater conditions at the operating depth of 3,400 feet.

- 5 Apply the Values.
 - Click OK from the Convection PropertyManager. Convection-1 is displayed.
- 6 Fit the model to the Graphics area.
 - Press the f key.
- **Note:** SolidWorks Simulation Professional applies convection to the four selected exposed faces and creates a single entry. Convection symbols appear on the four selected outside faces.

SolidWorks Simulation







SolidWorks Simulation

Creating a Mesh and Run 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 **Standard mesh** box.
 - 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.





Thermal Loads and Boundary Conditions

- 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.



SolidWorks Simulation



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



Thermal Loads and Boundary Conditions

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 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 Professional

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

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

2 Add Ribs to the EndCap Part.

- Click the **Model** tab at the bottom of the Graphics area.
- Right-click **CirPattern1** from the FeatureManager.
- Click **Unsuppress** ¹ from the Context toolbar. The EndCap with Ribs is displayed in the Graphics area.
- Return to Thermal-Study 2. 3
 - Click the **Thermal-Study 2** tab at the bottom of the Graphics area.

	Duplicate
	Rename Delete Delete All Simulaton Studies
	Create New Motion Study Create New Simulation Study Create New Design Study
	Study a mermai-study 1
	Define Study Name
Thermal-Study 2 (-Default-)	Study Name :
The connections	I hermal-Study 2
Heat Power-1 (:Per item: 600 W:	Default
Sesults	Cancel Help

olidWorks Premium 2013 x64 Edition 🧐 😫 🔶 🤭 T % EndCap (Default<<Default>_Di

Model | Motion Study 1







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

5 Analysis the Model.

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



SolidWorks Simulation

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.

Plot Tools	 Report Include Image for Report
on are Results	@ Ø, Ø
	on re Results

- 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 results. The two
 Studies are displayed.





- 7 Return to Study 2.
 - Click the Exit Compare button. View Thermal-Study 1.
- 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** A 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.

C Open			
🕒 🗣 🌽 « SolidWorks Sir	nulation Professional Drop_Test		٥
Organize 👻 New folder		III • 🗔 🔞	Ē
★ Favorites C ■ Desktop D	ocuments library	Arrange by: Folder *	
Downloads N	ame	Date	m
🛱 Libraries	HDusing_Assy	9/27/ 9/27	20
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Mode Cenfigurations Display States	III Resolved Default CDefault>_PhotoWor	Do not load hidden components Use Speedpak References Quick Fitter:	•
File name:	Housing_Assy	Assembly (".asm;".sldasm) Open Cancel	



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.





SolidWorks Simulation

- 4 Setup 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 FeatureManager from the Graphics area.
 - Expand the **second CH EndCap** component as illustrated
 - 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.









SolidWorks Simulation Professional

Meshing the Model

- 1 Mesh the model.
 - Click the Run drop-down arrow menu from the Simulation tab in the CommandManager
 - Click **Create Mesh b**. The Mesh PropertyManager is displayed.
 - Check the **Standard mesh** box as illustrated.
 - Expand the **Advanced** dialog box.
 - Check the **Draft Quality Mesh** box.
- **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 S 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** Maine E.
 - Click No in the Linear Analysis dialog box to retain your element choice. The analysis runs and the default plots are created.
- **Note:** Run time will take approximately 15 30 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

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.

👋 Droptest Study 1 (-Default-) 🛨 🍕 Parts Gonnections 🗄 📮 Component Contacts 🍯 Mesh 🚰 Setup Result Options 🖃 陆 Results Sivess1 (-vonMises-) No Displacement 1 (-Res disp-💕 Strain1 (-Equivalent-) 🖔 Droptest Study 1 (-Default-) 🗄 🍕 Parts 😑 🗊 Connections Component Contacts Mesh 🖑 Setup Result Options 😑 🚹 Results 💕 Str Hide 🚰 Disp 💕 Stra Edit Definition.. Ansnate ... 🧐 😭 😭 Stress Plot 🖌 🗶 🚽 Display VON: von Mises Stress • Advanced Options * Plot Step → @ 243.009 microsec 25 -Deformed Shape Automatic 22.32212257

SolidWorks Simulation

True scale
 User defined
 1

Property

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.
 - If needed, click **Play >** to start the animation. View the animation in the Graphics area.
 - Click **Stop** to stop the animation.





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

Running the Analysis

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

SolidWorks Simulation

Notes:

1
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.



SolidWorks Simulation Professional

Creating an Optimization Analysis

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

2 View Static Study1.

- A Static study was created for this part. Click the Study 1 tab located at the bottom of the Graphics area as illustrated. Study 1 is displayed.
- 3 Run Study 1.
 - Click **Run** ≦ from the Simulation tab in the CommandManager. View the created plots in the Results folder. The Stress1 (-vonMises-) plot is displayed in the Graphics area.





- of the Graphics area.
- Click Create New Design Study. The Design Study 1 tab is displayed along with the Design Study dialog box.



Processing Study 1	Variable View Table View Results View 🔄 👹 🔗 📊 Run 💟 Optimization
	Click here to add Variables ▼
	Click here to add Constraints
	Click here to add Goals
Model Motion Stud	<
SolidWorks Premium 2013 x64 Edition	n by

SolidWorks Simulation Professional

- 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 dropdown menu. The Parameters dialog box is displayed.
 - 6 Enter the Name.
 - Click **inside** the Name box.
 - Enter EndCap Thickness for name.
 - 7 Enter EndCap Lip thickness.
 - Click the dropdown arrow from the Model Dimension Category.
 - Select Model
 Dimension from the Category column.
 - Click **inside** the Value column box.
 - Rotate the model with the middle mouse button and zoom in on the .150in EndCap Lip thickness dimension.

Name	Category		Value	Units	
EndCap Thickness	Model Dimension	-	0.15	in	-
	Model Dimension	-	0	-	-



■ Click the **.150**

EndCap Lip thickness dimension as illustrated. The selected dimension is displayed in the Value column. Units are displayed in the Units box.

SolidWorks Simulation Professional

 Click Apply. The information is added to the Parameters dialog box.



- 8 Select the Second Design Variable.
- 9 Enter the Name.
 - Click **inside** the Name box.
 - Enter Rib
 Thickness for name.
- 10 Enter Rib Thickness.
 - Click the dropdown arrow from the Model Dimension Category.
 - Select Model
 Dimension from the Category column.
 - Click **inside** the Value column box.
 - Click the .150 EndCap Rip thickness dimension as illustrated.
 - Click **Apply**.
 - Click **OK**.





Optimization Analysis

SolidWorks Simulation Professional

11 Expand the Variables cell in the Design Study.

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

Variables		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
Constraints	Constraints								

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

- Select Range from the drop-down menu for EndCapThickness.
- Select Range from the drop-down menu for RibThickness.
- Enter the illustrated numbers for the EndCapThickness (Min: & Max:) range and the RibThickness (Min: & Max:) range.

🖃 Variables	
EndCapThicknes	ss Range 😽
RibThickness	Rande 📉
Click here to a	dd V Range with Step K Discrete Values
🖃 Constraints	Range

EndCapThickness	Range	~	Min:	0.05in 🚽	÷	Max:	0.2in	
RibThickness	Range	~	Min:	0.05in		Max:	0.2in	~
Click here to add	Variables	~		•			N	

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

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

	EndCapThickness	Range	
	RibThickness	Range	
	Click here to add	Variables	
🗆 Co	nstraints		->

SolidWorks Simulation Professional

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



14 Set the conditions for the Constraint.

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



15 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

- 16 Set the condition for the Goal.
 - Select Minimize.
- 17 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.

🖃 Go	oals		
	Mass1	Minimize	Y
	Click here to add G	Minimize	-
Stuc	dy 1 🖫 Design Stu	Maximize Is exactly	

N	EndCapThickness	Range	~	Min:	0.05in
	RibThickness	Range	~	Min:	0.05in
	Click here to add	Variables	~		
🗆 Con	straints Stress1	Is less than	~	Max:	60000 psi

3 SOLIDWORKS File Edit Vier	w Insert Tools	Simulation Toolbox Wind	ow Help	9 7 - 6	2	· • •	. 1	2.0	1 5%
Study									
Advisor									
-								N=	_
Features Sketch Evaluate DimX	pert Office Product	s Simulation of a	8 🚺 📳	- 🗇 - 6 ₀ -	• 🕐 🧶 - I	<u>-</u>	20	u _ #	\mathbb{X}
🥵 🖀 😫 👄 👋 🚿									
T									
SendCap (Add Rib< <add rib="">_Di</add>									
Sensors									
			A						1
->> Plane1		AV			////	Δ			
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- 🔆 Plane3			I	M		<i>y</i>			•
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						10			
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CirPattern1									
E- Cut-Extrude1	>								
E-M Split Line1	4								
Stan Study 1	Mariable Manu	Table Man Day the Ma		B 🔥 🔲					_
E-B Results and Graphs	11 of 11 scenarios r	able view Results view	Ouslity: High						
	11 of 11 section of the	an addeesardiiy. Dealgiri adday	Quality. High						
			Current	Initial	Optimal	Iteration 1	Iteration 2	Iteration 3	
	EndCap Thickness	-0	0.11715in	0.15in	0.11715in	0.2in	0.05in	0.2in	0.0
	RIb Thickness	-0	0.05832in	0.15in	0.05832in	0.2in	0.2in	0.05in	0.0
	Stress1	< 60000 psi	57985 psi	34004 psi	57985 psi	31149 psi	1.4639e+005 psi	58521 psi	1.4
	Mass1	Minimize	1.68773 kg	1.91678 kg	1.68773 kg	2.16059 kg	1.61705 kg	1.98096 kg	1.4
	•	III]						÷.
Model Motion Study 1	¥ Study 1 1 T⊶ De	sign Study 1							
and the second se									

SolidWorks Simulation Professional

18 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.

Variable View	Table View	Results View	1	9 📂 📊	
11 of 11 scenarios ra	an successfully.	Design Study Qu	ality: High		
			Current	Inițial	Optima
EndCap Thickness		0.	15in	0.15in	0.11715in
Rlb Thickness		0.	15in	0.15in	0.05832in
Stress1	< 60000 psi	34	4004 psi	34004 psi	57985 ps
	2333-2372 (Salara			1.010701	4 000000
Mass1	Minimize	1.	916/8 kg	1.91678 Kg	1.68/73
Mass1 Variable View 11 of 11 scenarios ra	Minimize Table View an successfully.	Results View Design Study Qu	91678 kg El 🕅 🕅 ality: High	1.916/8 kg	1.687731
Mass1 Variable View	Minimize Table View an successfully.	1. Results View Design Study Qu	91678 kg	1.916/8 kg	0ptim
Mass1 Variable View 11 of 11 scenarios ra EndCap Thickness	Minimize Table View an successfully.	1. Results View Design Study Qu	91678 kg ality: High Current 11715in	1.91678 kg	0.11715in
Mass1 Variable View 11 of 11 scenarios ra EndCap Thickness Rlb Thickness	Minimize Table View an successfully.	1. Results View Design Study Qu 0. 0.	91678 kg ality: High <u>Current</u> 11715in 05832in	Initial 0.15in 0.15in	0.11715in 0.05832in
Mass1 Variable View 11 of 11 scenarios ra EndCap Thickness Rlb Thickness Stress1	Minimize Table View an successfully.	Results View Design Study Qu 0. 0. 57	91678 kg ality: High Current 11715in 05832in 7985 psi	Initial 0.15in 0.15in 34004 psi	Optim 0.11715in 0.05832in 57985 ps

EndCapThickness	
RibThickness	Jk
Stress1	< 60000 psi

19 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.





SolidWorks Simulation Professional

SolidWorks Simulation

- 20 Close the Graph1 dialog box.
 - Click Close.
- 21 Save and Close the Model.
 - Click Save 🔙.
 - Click Window, Close All from the Menu bar menu.

🎇 Graph1 File Options Help Local Trend: Design Study 1 Initial 1.60+05 1.40+05 1.20+05 [isi 1.00+05 Stress 8.00+04 6.00+04 4.00+04 2.00+04 0.05 0.08 0.11 0.14 0.17 0.20 0.22 EndCap Thickness (in) Stress1 (psi) 0.226661, 164667



SolidWorks Simulation Professional

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.





Fatigue Analysis

SolidWorks Simulation Professional

Creating a Fatigue Analysis

- 1 Open the Part.
 - Click Open 2 from the Menu bar toolbar.
 - Double-click 3 Finger Jaw from the SeaBotix\SolidWorks Simulation Professional\Fatigue folder.





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Study

Advisor

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84

Design

Study

Study Advisor

2 Create a Static Analysis Study.

- Click the Study Advisor drop-down arrow from the Simulation tab in the CommandManager.
- Click New Study <a>. 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

Applying Material

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

Material prope Materials in t a custom libr	erties the default librar			
Material prope Materials in t a custom libr	the default librar			
a custom libr	and deradic librar	can not be edited	ou must first	conv the mater
	ary to eutric.	y can not be euted. I	ou must mist	copy the mater
Model Type:	Linear Ela	astic Isotropic	•	
Units:	SI - N/m	^2 (Pa)	•	
Category:	Aluminiu	m Alloys		
Name:	6061-T6	(SS)		
Default failu	re (_	
criterion:	Max von	Mises Stress	~	
Description:				
Source:				
Suctainabilit	Defined			
Sascairiabilic	×			
Property		Value	Unite	_
Flastic modulu		6 90000067e+010	N/m^2	
Poisson's ratio		0.33	N/A	
Shear modulus	3	2.60000013e+010	N/m^2	
Mass density		2700	kg/m^3	
Tensile strengt	th	31000002.1	N/m^2	
Compressive S	Strength in X		N/m^2	
Yield strength		275000000.9	N/m^2	
Thermal expan	Ision coefficient	2.40-005	/K	
Specific heat	Curity	896	J/(ka·K)	
Material Dampi	no Ratio		N/A	
			01.910	
AD	ply Close	Save	onfig	Help
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	Units: Category: Name: Default failu criterion: Description: Source: Sustainabilit Property Elastic modulus Mass densfy Tensile streng Compressive E Yield strength Thermal condu Specific heat Material Damni	Units: SI -N/m Category: Aluminiu Name: 6061-T6 Default failure Max von criterion: Max von Source: Sustainability: Defined Property Elastic modulus Poisson's ratio Shear modulus Mass density Tensile strength Thermal expansion coefficient Thermal expansion coefficient Thermal expansion coefficient Thermal expansion coefficient Thermal expansion coefficient Thermal expansion coefficient	Units: SI - N/m^2 (Pa) Category: Aluminium Alloys Name: 6061-T6 (SS) Default failure Max von Mises Stress criterion: Max von Mises Stress Description: Source: Sustainability: Defined Property Value Elastic modulus 6.90000067e+010 Poisson's ratio 0.33 Shear modulus 2.600000013e+010 Mass density 2700 Tensie strength 31000002.1 Compressive Strength in X Yield strength Yield strength 2.4e-005 Thermal expansion coefficient 896 Material Daminin Ratio Save	Units: SI - N/m^2 (Pa) Category: Aluminium Alloys Name: 6061-76 (SS) Default failure Max von Mises Stress criterion:

SOLIDWORKS File Edit View Insert T Study Advisor Advisor Lise File File Lise File Lise File File

SolidWorks Simulation Professional

SolidWorks Simulation

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.





SolidWorks Simulation Professional

all.

ixtures

Advisor Loads...

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Fixed Geometry Roller/Slider

±8 ¶ External Connection:

Advisor

- 3 Add a Second Fixture.
 - Click the Fixtures Advisor drop-down arrow from the Simulation tab in the CommandManager.
 - Click Advance Fixtures. The Fixture PropertyManager is displayed. Perform a radial support on the right-hand face.
- 4 Select the inside Cylindrical Face.
 - Click the On Cylindrical
 Faces box.
 - Rotate the model to view the side cylindrical face as illustrated.
 - Click the inside face of the slot as illustrated. Face<1> is displayed.
- 5 Select Units and Displacement Components.
 - Select **inch** from the Units drop-down menu.
 - Click the **Radial i** box.
- 6 Apply the Second Fixture.
 - Click OK from the Fixture PropertyManager. On Cylindrical Faces-1 is displayed.





SolidWorks Simulation Professional

SolidWorks Simulation

Applying a Force

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

2 Select the contact face.

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



- 3 Set Units and Value.
 - Select English (IPS) in the Units box.
 - Enter **30**lbf in the Force value box.
- **Note:** 30lbf is the normal force that the MiniGrab assembly can apply in holding an object from the sea floor.
 - 4 Apply the Force.
 - Click OK from the Force/ Torque PropertyManager. Force-1 is displayed.



Meshing and Running the Model

- 1 Mesh and Run the Model.
 - Click the Run drop-down arrow from the Simulation tab in the CommandManager
 - Click Create Mesh h. The Mesh PropertyManager is displayed.
 - Click Standard mesh.
 - Check the **Run (solve) the** analysis box.
 - Click **OK** from the Mesh PropertyManager. View the results. Three plots are created.
- 2 Fit the model to the Graphics area.

Model name: 3 Finger Jaw Study name: Static-Study 1 Plot type: Static nodal stress Stress1

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





SolidWorks Simulation Professional

Performing a Fatigue Check Plot.

- 1 Create a New Fatigue Study.
 - Right-click the **Results** folder.
 - Click **Define Fatigue Check Plot**. The Fatigue Check Plot PropertyManager is displayed.
- 2 View the Fatigue Check Plot.
 - Click the Fully Reversing Load button. View the results in the Graphics area. There is a possible Fatigue issue.
 - Click **Cancel** ★ from the Fatigue Check Plot PropertyManager.





Creating a New Fatigue Study.

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



Fatigue Analysis

SolidWorks Simulation Professional

- 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.

🗬 Fatigue-Stud	y 1 (-Default-)
⊕ Eu Loa	Apply/Edit Fatigue Data

SolidWorks Simulation

		Properties Source Interpol Defin Defin Ba Ba	Tables & Curves Para ate: Log-log e: Semi-log e from matt Linear ased on ASME Austen ased on ASME Carbon	tic Steel curves Steel curves	v CrossHa	
		Table dat	ta			
356.0-T6 Permanent Mold cast (SS)		Stress R	atio (R): -1	Units: N/m^2	•	
4032-T6		Points	A	В	•	
		1	100	636360430.2		El-
5052-H34		2	200	490070676.1	1	File
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5052-H38	E	4	1000	290141345.6		
5052-H38, Rod (SS)		5	2000	236501769.1		Save
5052-O		6	5000	185300355.2		
5052-O, Rod (SS)		17	10000	156042404.3		
		Source:				
5454-H112						
======================================						
6061-T4 (SS)				/		
						-

- 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

- 6 View the Life Plot.
 - Double click on the Results2 (-Life-) folder. The Life plot is displayed.
- 7 Display the Chart Options PropertyManager.
 - Double-click on the Total Life (cycle) plot in the Graphics areas as illustrated. The Chart Options PropertyManager is displayed.



8 Reverse the Life plot results color.

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





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. You are finished with the SolidWorks Simulation Professional section of the HOTD manual.









Fatigue Analysis

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.

Notes:

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 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 Flow Simulation module.
 - Click the Options arrow from the Menu bar toolbar as illustrated.
 - Click Add-Ins. The Add-Ins dialog box is displayed.
 - Check the SolidWorks Flow Simulation 2013 box.
 - Click **OK** from the Add-Ins dialog box. The Flow Simulation tab is displayed in the CommandManager.







SolidWorks Simulation

- 3 Start the SolidWorks Flow Simulation Wizard.
 - Click the Flow
 Simulation tab located in the CommandManager.
 - Click the Wizard tool. The Wizard -Project Configuration box is displayed. Create new is selected by default. Accept the default settings.
 - Click Next>. The Wizard Unit System dialog box is displayed.
 - Click IPS (in-lb-s) for Unit system.
 - Click inside the
 Velocity Unit box.
 - Click Knot from the drop-down menu as illustrated.
 - Click .123 from the drop-down menu from the Decimals in results display column.
 - Click Next>. The Wizard - Analysis Type dialog box is displayed.





SolidWorks Flow Simulation

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



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



Project Fluids	Default Fluid	Bemove
Water (Liquids)		Tremove
Flow Characteristic	Value	
Flow Characteristic Flow type	Value Laminar and Turbulent	

SolidWorks 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.



Minimum gap size

Minimum gap size:

Minimum wall thickness

Minimum wall thickness:

Advanced narrow channel refinement

< Back

Manual specification of the minimum gap size
Minimum gap size refers to the feature dimension

Manual specification of the minimum wall thickness
Minimum wall thickness refers to the feature dimension

inement I Optimize thin walls resolution

SolidWorks 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 🧧 tab.

- Expand the Input Data folder.
- Right-click Goals.
- Click Insert Global Goals. The Global Goals PropertyManager is displayed.
- Scroll down and check the Max box in Force(X).
- Click **OK** ✓ from the Global Goals PropertyManager.



SolidWorks Flow Simulation

- 7 Run the Analysis.
 - Right-click Run from Project 1 as illustrated. The Run dialog box is displayed.
 - Select **1 CPU** from the drop-down menu,
 - Click the **Run** button.
- **Note:** To save classroom time, we will stop the analysis and open the Results folder to review completed results.
 - 8 Stop the Analysis.
 - Wait until the third iteration and click Stop from the Solver box as illustrated.
 - Click No to the question, "Do you want to save the results?"
 - Click **File**, **Close** from the Solver Main menu.
 - 9 Open the Configuration with the Solved Results.
 - Click the **ConfigurationManager** ¹ tab.
 - Double-click the Default (3) configuration as illustrated.
 - Click the Flow Simulation analysis tree 4 tab.
 - Right-click the **Results** folder.
 - Click Load Results. The Load Results dialog box is displayed.



Oefault	(3)				
🗄 🖓 Inp	ut Data				
	Computational Domain				
📷 Boundary Conditions					
🖕 🏁 Goals					
L.	GG X - Component of Force 1				
i≟ ©a Res	Load Results Select Results				

SolidWorks Flow Simulation

SolidWorks Simulation

- Double-click **3.fld** in folder 3.
- 10 Create a Section Plot.
 - Expand the **Results** folder.
 - Right-click the **Cut Plots** folder.
 - Click Insert. The Cut Plots PropertyManager is displayed. Front Plane is selected by default.



- Expand LBV_ASSY from the flyout FeatureManager. View the features. Front Plane is displayed in the Section Plane box.
- Click the Contours button in the Display box.
- Click the drop-down menu in the Contours box to select Velocity as illustrated.



SolidWorks Simulation



12 Move the Section Plot in the Graphics area.

- **Expand** the Cut Plots folder.
- Click Cut Plot1. View the control arrow in the Graphics area.
- Click and drag the control arrow to the left side of the computational domain.
 View the changing results of the Section plot.





SolidWorks Simulation

- Click and drag the control arrow to the right side of the computational domain.
- Move the Section Plot back to its original location in the computational domain.



SolidWorks Flow Simulation

SolidWorks Simulation

- 13 View the moving Computational Domain.
 - Right-click **Cut Plot1**.
 - Click **Play**. View the results in the Graphics area.
- 14 Stop the moving Computational Domain.
 - Right-click Cut Plot1.
 - Click Stop. View the results in the Graphics area.
- **15** Hide the Computational Domain.
 - Right-click the Computational Domain folder.
 - Click Hide.
- 16 Create a Second Cut Plot.
 - Right-click the **Cut Plots** folder.
 - Click **Insert**. Front Plane is selected by default.






SolidWorks Flow Simulation

17 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 or face box.

18 Continue the Second Cut Plot.

- Click the Contours button.
 Select Pressure from the dropdown menu for Parameter.
- Click the Adjust Minimum and Maximum button. View the range.
- Click **OK** ✓ from the Cut Plot PropertyManager. View the Section plots in the Graphics area.

	-
Image: Selection Image: Selection	BV_ASSY (Default (3) < <d Sensors Construction</d
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Options 😵	
Cure Desire	
Crop Region 🛛 🖇	1

SolidWorks Flow Simulation



SolidWorks Flow Simulation

19 Hide the first Section Plot.

- Right-click **Cut Plot1**.
- Click **Hide**. View the results.

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





SolidWorks Flow Simulation

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

SolidWorks Simulation







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

SolidWorks 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.





SolidWorks Flow Simulation

SolidWorks Simulation

- 2 Edit the Flow Trajectory.
 - Expand the Flow Trajectories folder.
 - Right-click Flow Trajectories 1.
 - Click Edit Definition. The Flow Trajectories PropertyManager is displayed.
 - **Pin** → the Flow Trajectories PropertyManager.
 - Enter **100** for the Number of Points as illustrated.
 - Click Lines with Arrows from the drop-down menu in the Appearance box.
 - **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.

Play		~
Play animation without saving to the disc		R
J Model Motion Study - Default olidWorks Premium 2013 x64 Edition	Motion Study 1	Animation 1

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Particle

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薞 Goal Ploi 🕢 Report

📷 Animatic

Edit I

Edit Definition...

Clear and Hide

Hide

Clone

Play

Delete...

SolidWorks Flow Simulation

- 4 Hide the Flow Trajectory.
 - Right-click the Flow Trajectories 1 folder.
 - Click **Hide**. View the Graphics area.
- 5 Set the Goal Plots.
 - Expand the **Results** folder.
 - Right-click the Goal Plots folder as illustrated.
 - Click Insert. The Goal Plot PropertyManager is displayed.
 - Check the GGX-Component of Force1 box as illustrated.
 - Click OK from the Goal Plot PropertyManager. The Goals dialog box is displayed. View your options.







SolidWorks Flow Simulation

6 View the Excel Plot.

- Click the bottom **Force (X)** tab.
- **View** the plot.





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

- Click Close.
- Select Don't Save when prompted to Save.

8 Save and Close the model.

- Click File, Close from the SolidWorks Main menu.
- Click Save All when prompted to save. You are finished with the SolidWorks Flow Simulation section of the HOTD manual.



Applying Flow Trajectories

SolidWorks Flow Simulation

SolidWorks Simulation

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.

Notes:

SolidWorks Simulation

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

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 Simulation

Starting a SolidWorks Motion Session

- 1 Open the Gripper Assembly.
 - Click **Open** if from the Menu bar menu.
 - Double-click the Gripper Motion 2013 assembly from the SeaBotix\SolidWorks Motion folder.

🛛 Oper 🕌 « SolidWorksSimulati... 🕨 SolidWorks Motion 🕨 Organize - New folder HE - FI 0 **Documents** library Favorites Arrange by: Folder SolidWorks Motion E Desktop Downloads Name E Recent Places Finished Gripper Motion 2013 📜 Libraries S Documents J Music Pictures Videos - -Display States: Mode: Configurations: References.. Ouick Filter: T-g S 28 File name: *.ASM;*.SLDASM -Assembly (*.asm;*.sldasm) Open 💌 Cancel

2 Activate SolidWorks Motion.

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



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



Applying Motion to a Component

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

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





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.
 - Click the Push-Pull Plate component face of the Gripper assembly. The direction arrow points inward.



Applying Motion to a Component

SolidWorks Simulation

5 Display the Function Builder.

Select Segments for Motion Type from the drop-down menu. The Function Builder dialog box is displayed. Use the Function Builder to define motor or force profiles from an imported data set. You can also use the Function Builder to define a motor or force profile from a mathematical expression or from interpolated functions along connected segments. For motor profiles, you can specify time, cycle angle, or any result as the independent variable.

6 Add Rows.

Click Click to add row. A new row is displayed.

Moti	on 🌣	
	Constant Speed	
0	Constant Speed	
0	Distance	
	Oscillating	
	Segments	
	Data Points	
	Expression	
	Servo Motor	
	Load Function from File	
	Delete Functions	Ч



SolidWorks Motion



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

SolidWorks Simulation

- 7 Edit Key Point Time for Motion Study.
 - Right-click the Key Properties icon as illustrated.
 - Click Edit Key Point Time.
 - Enter **.2** for time.
 - Click **Save** from the Edit Time dialog box.
 - Click the **Zoom to Fit** icon in the Motion Study. View the results.



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

- 1 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.
 - 2 Apply the Force.
 - Click the Force icon from the Motion Manager toolbar. The Force/Torque PropertyManager is displayed.
 - Click the Force box for Force Type.
 - Click the Action only box for Direction.
 - Click the Reverse Direction button. The direction arrow points into the finger.
 - Enter 62 N for Constant Value.
 - Click OK from the Force/Torque
 PropertyManager. Force1 is displayed in the
 Motion Study FeatureManager.







Applying Forces

- 3 Apply a Contact Force to the two other Fingers.
 - Repeat Step 1 & 2 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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4 Run the Motion Simulation.

- Click the **Calculate** [■] icon.
- If needed click Yes to the message in the dialog box. View the assembly moving while the analysis is being performed.



5 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.
- Click **OK** ✓ in the Results PropertyManager.



SolidWorks Motion

- Click **No** to the displayed message. View the plot.
- Click along the **time axis** and view the changes in the Gripper.

Note: Do not close Plot1 at this time.



Applying Forces

SolidWorks Motion

6 Edit the LinearMotor1 Feature.

- Right-click LinearMotor1 in the Motion Study FeatureManager.
- Click Edit Feature. The Motor PropertyManager is display.
- Click the Edit button. The Function Builder dialog box is displayed.
- Modify the EndX (.05) time cell as illustrated. Note the update in the plots.
- Click OK from the Function Builder dialog box.
- 7 Return to the SolidWorks Motion Graphics area.



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SolidWorks Simulation

- 8 Re-run the Motion Simulation.
 - Click the Calculate icon. View the results of the new plot in the Graphics area.





Applying Forces

SolidWorks Motion

- 9 Create a Trace Path.
 - Click on the Results and Plots icon from the Motion toolbar. The Result PropertyManager is displayed.
 - Select Displacement/Velocity/ Acceleration from the drop-down menu in the Result box.
 - Select **Trace Path** from the drop-down menu as a sub-category.
 - Click a **point** at the end of a finger as illustrated in the Graphics area. Note the icon feedback symbol.
 - Click OK from the Results PropertyManager.
- **Note:** A Trace Path graphically displays the path that any point on any moving part follows.



10 Zoom in to view the results.

Use the middle mouse wheel to zoom in on the Trace Path.



11 Re-run the Motion Simulation.

 Click Play from Start. View the results in the Graphics area.



SolidWorks Motion

- 12 Disable Playback of view Keys.
 - Right-click Orientation and Camera Views from the Motion Study FeatureManager.
 - Click Disable Playback of View Keys.
 - Click **Play from Start**. View the results in the Graphics area.

13 Rebuild and Save the Assembly.

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

14 Close all models.

Click Window, Close All from the Menu bar menu. You are finished with the SolidWorks Motion section of the HOTD manual.







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