SolidWorks® 2011

Hands-on Test Drive

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

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

Document Number: MKHOTBK1109

Table of Contents

Using This Book

As the title implies, this book is a hands-on experience. You will be using SolidWorks® very quickly. You will be experiencing many of the major capabilities of SolidWorks very quickly. You will be learning by doing. Once you start up SolidWorks, you will be working with models, assemblies, and drawings for the rest of the book that you will create. This book is just a guide.

Before you Begin

SolidWorks files that have been included for your use are loaded to the SolidWorks Test Drive folder on your system. This book is written with the assumption that you will also save your files in the same folder.

About the Author

David Planchard is the founder of D&M Education LLC. Before starting D&M Education, he spent over 28 years in industry and academia holding various engineering, marketing, and teaching positions and degrees. He holds five U.S. patents and one International patent on equipment design. He has published and authored numerous papers on Machine Design, Product Design, Mechanics of Materials, and Solid Modeling. David holds a BSME, MSM and a CSWA certification. David is a SolidWorks Solution Partner and has co-authored over 35 SolidWorks publications in the past ten years.

Hands-on Test Drive

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

- The ease of use of a Microsoft® Windows® application
- The power of 2D sketching and 3D modeling
- Ability to quickly sketch ideas, experiment with features and dimensions, and produce 3D models and detailed 2D drawings
- SolidWorks® Sustainability: SustainabilityXpress and Sustainability
- SolidWorks® Intelligent Feature Technology (SWIFT™)
- Assembly modeling and associative drawings with automatic generation of dimensions
- Communication with SolidWorks® eDrawings®, SolidWorks DXF PropertyManager, and 3D ContentCentral
- Model visualization with SolidWorks PhotoView 360
- Analysis with SolidWorks® SimulationXpress using Optimization

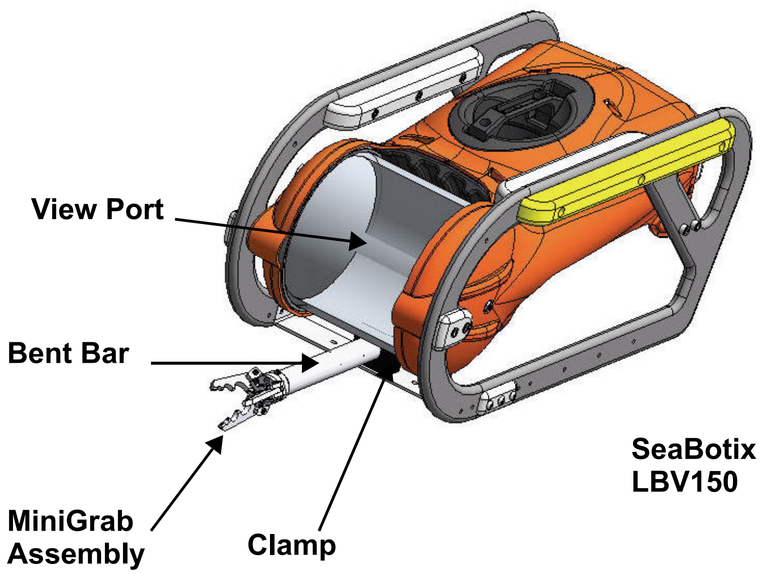
Hands-on Test Drive

The SeaBotix LBV150

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

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

The company selected SolidWorks® mechanical design software for the Little Benthic Vehicle project because of its ease of use, ability to model organic shapes and surfaces, eDrawings communication capabilities, and seamless integration with SolidWorks® Simulation analysis software. By deploying SolidWorks on the Little Benthic Vehicle project, SeaBotix reduced its design cycle by 50 percent, minimized tooling modification costs, effectively communicated design information among several locations, and introduced the first full-production, lightweight submersible remotely operated vehicle to a mass market.



Once you have had a chance to experience firsthand the ease of using SolidWorks solid modeling software, you will create an assembly using the parts you've built plus some parts we've built ahead of time.

You will then make a drawing of one of the components, complete with dimensions. If a printer is available, you can print out a hard copy of the drawing.

Parts, Assemblies, and Drawings

Parts are single three-dimensional (3D) objects. Parts are the basic building blocks of 3D modeling. Parts can be included as components in assemblies and represented in drawings. The SeaBotix LBV150 that we are designing has hundreds of parts. The parts that we will address include: Clamp, Mount, and Bent Bar.

Assemblies are logical collections of components. These components can be parts or other assemblies. All of the parts of the SeaBotix LBV150 are combined into a single assembly. The parts are combined according to the way they are designed to function. They are placed in relation to each other and these relationships can be captured so that you can communicate your design intent to others. For example, the Clamp part will be placed into an assembly in relation to the Bent Bar. When these parts are manufactured, this is how they would go together, or be assembled.

An assembly within an assembly is called a subassembly. The SeaBotix LBV150 contains the MiniGrab subassembly. Particularly in large projects, different subassemblies will be designed by different people, even different companies.

Drawings are 2D representations of 3D parts or assemblies. Drawings are required for manufacturing, quality assurance, supply chain management, and other functions.

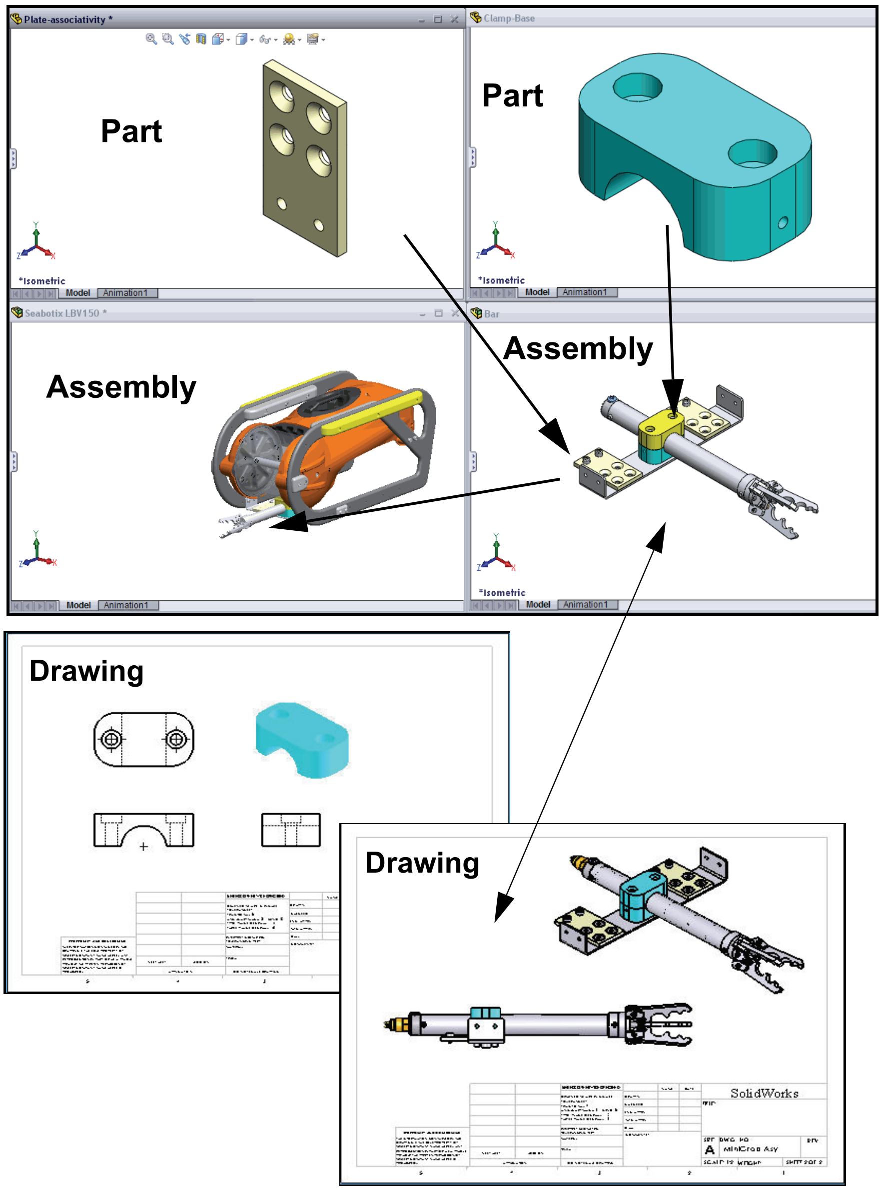
In SolidWorks, parts, assemblies, and drawings are associative. This means that changes in one location are reflected in all of the other locations where they need to be reflected. Changes that you make to an assembly are reflected in the associative drawings of that assembly. Changes that you make to a part are reflected in the associative assembly.

Typically, you design each part, combine the parts into assemblies, and generate drawings in order to manufacture the parts and assemblies.

DFMXpress for SolidWorks® is an easy-to-use Design for Manufacturability (DFM) tool for designers and manufacturing engineers that is seamlessly integrated into the SolidWorks environment. It facilitates upstream manufacturability validation and identification of areas of a design that are difficult, expensive or impossible to manufacture.

Relationship of Parts, Assemblies, and Drawings

The following illustration displays the relationship among parts, assemblies, and drawings in SolidWorks.



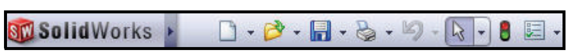
SolidWorks User Interface (UI)

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

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

Menu Bar Toolbar

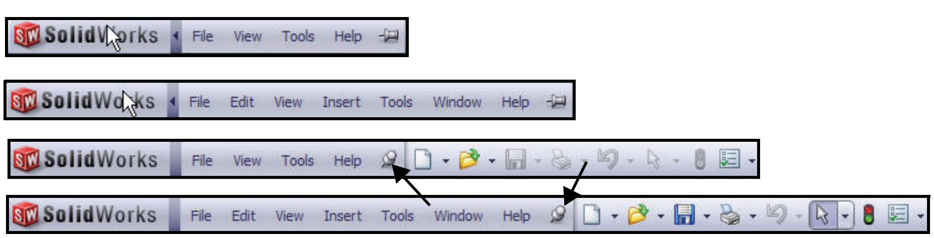
The Menu Bar toolbar contains a set of the most frequently used tool buttons from the Standard toolbar. The available tools are: **New** - Creates a new document, **Open** - Opens an existing document, **Save** - Saves an active document, **Print** - Prints an active document, **Undo** - Reverses the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



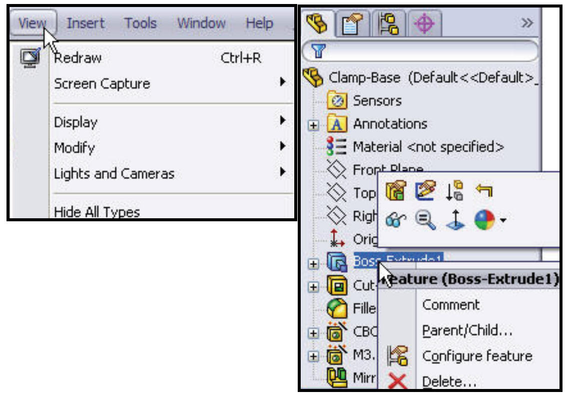
Menu Bar Menu

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

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



Drop-down / Pop-up Context Toolbar



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

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

Note: Context toolbars are available for most commonly used selections.

Keyboard Shortcuts

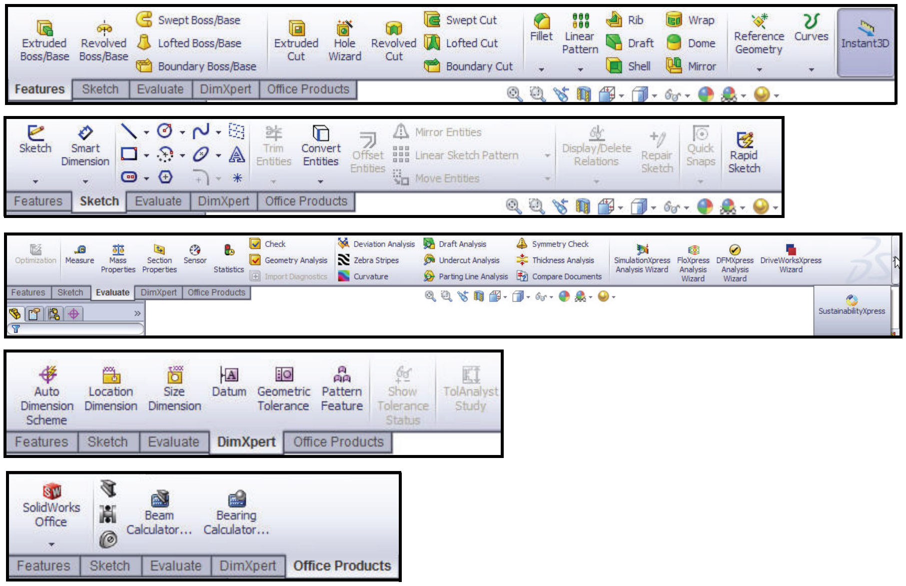
Some menu items indicate a keyboard shortcut like this: Cut Ctrl+X. SolidWorks conforms to standard Windows conventions for shortcuts such as **Ctrl+O** for **File, Open**; **Ctrl+S** for **File, Save**; **Ctrl+X** for **Cut**; **Ctrl+C** for **Copy**; and so on. In addition, you can customize SolidWorks by creating your own shortcuts.

CommandManager

The CommandManager is a context-sensitive toolbar that automatically updates based on the toolbar you want to access. By default, it has toolbars embedded in it based on your active document type. When you click a tab below the CommandManager, it updates to display that toolbar. Example, of you click the Sketch tab, the Sketch toolbar is displayed. The default tabs for a part document are: **Features**, **Sketch**, **Evaluate**, **DimXpert** and **Office Products**.

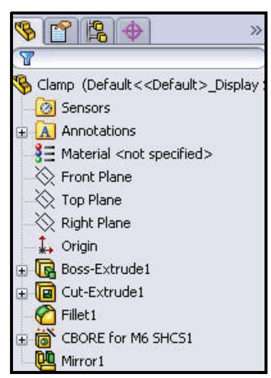
Below is an illustrated CommandManager for a default Part document.

Note: SolidWorks® Toolbox is an Add-In.



Note: If you have SolidWorks, SolidWorks Professional, or SolidWorks Premium, the Office Products tab appears on the CommandManager.

FeatureManager



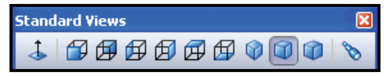
The FeatureManager® design tree is a unique part of the SolidWorks software that employs patented SolidWorks technology to visually display all of the features in a part, assembly, or drawing.

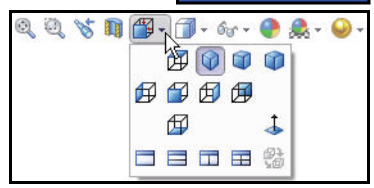
As features are created, they are added to the FeatureManager. As a result, the FeatureManager represents the chronological sequence of modeling operations. The FeatureManager also allows access to editing the features and objects that it contains.

The Part FeatureManager consist of four default tabs:

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

Heads-up View Toolbar





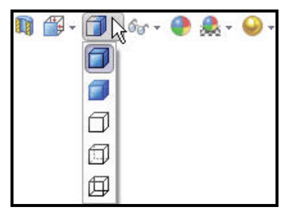
For an active drawing document

SolidWorks provides the user with numerous view options from the Standard Views, View, and Heads-up View toolbar.

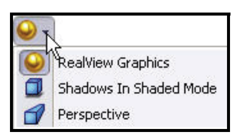
The Heads-up View toolbar is a transparent toolbar that is displayed in the Graphics area when a document is active

The following views are available:

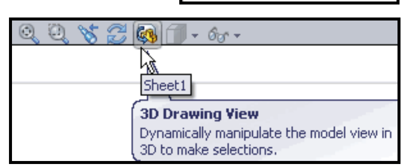
- **Zoom to Fit** . Zooms the model to fit the Graphics area.
- **Zoom to Area** . Zooms to the areas you select with a bounding box.
- **Previous View** . Displays the previous view.
- **Section View** . Displays the cutaway of a part or assembly using one or more cross section planes.



- **View Orientation** . Select a view orientation or the number of viewports from the drop-down menu.
- **Display Style** . Select the style for the active view from the drop-down menu.
- **Hide/Show Items** . Select items to hide or show in the Graphics area.
- **Edit Appearance** . Edits the Appearance of the selected entity.
- **Apply Scene** . Applies a scene to an active part or assembly document.



- **View Setting** . Select the following setting from the drop-down menu: *RealView Graphics*, *Shadows in Shaded Mode*, and *Perspective*.



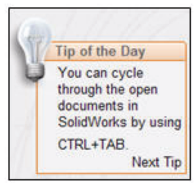
- **Rotate view** . Rotates a drawing view.
- **3D Drawing View** . Dynamically manipulates the drawing view to make a selection.

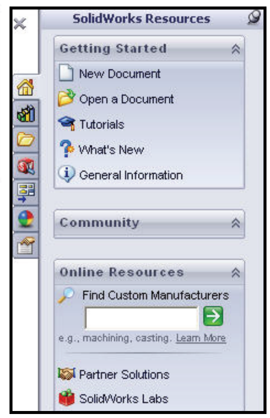
Task Pane

The Task Pane is displayed when a SolidWorks session starts. The Task Pane contains the following default tabs: **SolidWorks Resources**, **Design Library**, **File Explorer**, **SolidWorks Search**, **View Palette**, **Appearances/Scenes**, and **Custom Properties**.

Note: The Document Recover tab is only displayed in the Task Pane if your systems terminates unexpectedly with an active document and if auto-recovery is enabled in Systems Options.

SolidWorks Resources

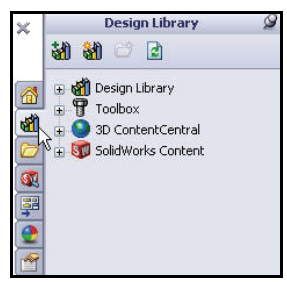




The basic SolidWorks Resources tab displays the following default selections: *Getting Started*, *Community*, *Online Resources*, and *Tip of the Day*.

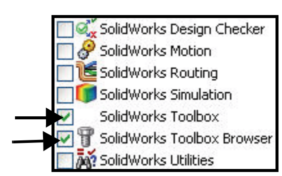
Other user interfaces are available: *Machine Design*, *Mold Design*, or *Consumer Products Design* during the initial software installation selection.

Design Library

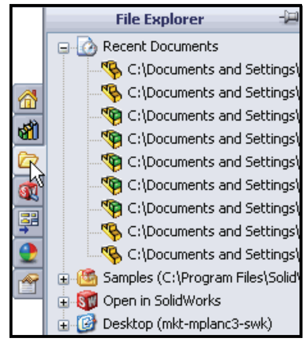


The Design Library contains reusable parts, assemblies, and other elements, including library features. The Design Library tab contains four default selections. Each default selection contains additional subcategories. The default selections are: *Design Library, Toolbox, 3D ContentCentral*, and *SolidWorks Content*.

Note: Click **Tools**, **Add-Ins..**, **SolidWorks Toolbox** and **SolidWorks Toolbox Browser** to activate the SolidWorks Toolbox. You can also add folders to the Design Library.

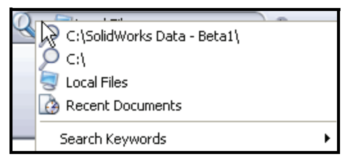


File Explorer



File Explorer duplicates Windows Explorer from your local computer and displays the recent documents and directories: open in SolidWorks and Desktop.

Search

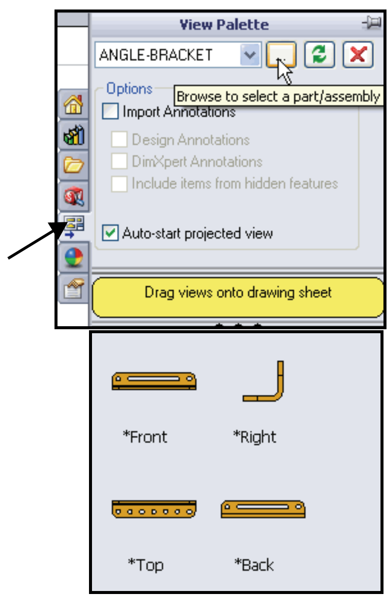


SolidWorks Search is installed with Microsoft Windows Search and indexes the resources once before searching begins, either after installation or when you initiate the first search.

The SolidWorks Search box is displayed in the upper right corner of the SolidWorks Graphics area. Enter the text or key words to search. Click the drop-down arrow to view the last 10 recent searches.

The Search tool in the Task Pane searches the following default locations: *All locations, Local Files, Design Library, SolidWorks Toolbox, and 3D ContentCentral.*

View Palette

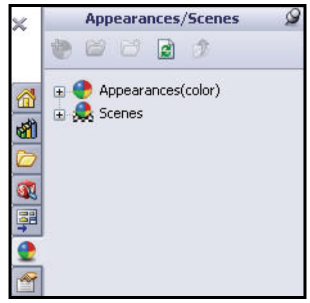


The View Palette tab located in the Task Pane provides the ability to insert drawing views of an active document, or click the Browse button to locate the desired document.

Click and drag the view from the View Palette into an active drawing sheet to create a drawing view.

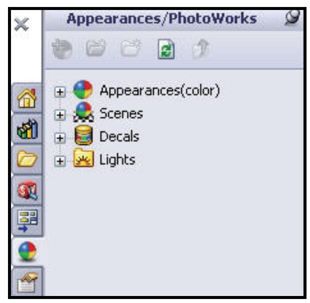
Note: There are over 200 enhancements in SolidWorks 2010. Over 90% of these enhancements where requested directly by customers.

Appearances/Scenes



Appearances/Scenes tab provides a simplified way to display models in a photo-realistic setting using a library of appearances and scenes.

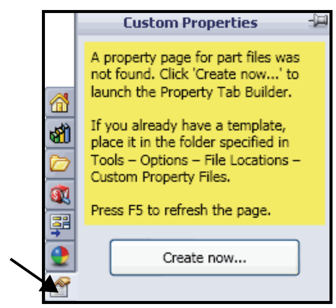
On RealView compatible systems, you can select Appearances and Scenes to display your model in the Graphics area. Drag and drop a selected appearance onto the model or FeatureManager. View the results in the Graphics area.



Note: PhotoWorks needs to be active to apply the scenes tool.

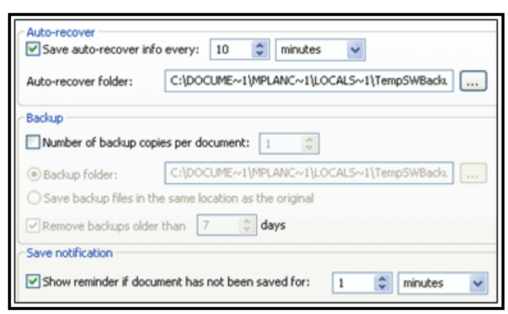
Note: Appearances/PhotoWorks graphics is only available with supported graphics cards. For the latest information on graphics cards that support Appearances/PhotoWorks display, visit: www.solidworks.com/pages/services/videocardtesting.html.

Custom Properties



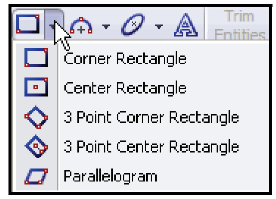
The Custom Properties tab located in the Task Pane. Custom Properties provides the ability to enter custom and configuration specific properties directly into SolidWorks files.

Document Recovery



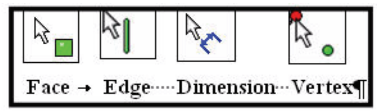
If Auto recovery is initiated in the System Options section and the system terminates unexpectedly with an active document, the saved information files are available on the Task Pane Document Recovery tab the next time you start a SolidWorks session.

Consolidated Flyout Tool Buttons

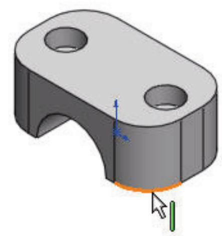


Similar commands are grouped into consolidated flyout buttons on the toolbar and in the CommandManager. Example: Variations of the Rectangle Sketch tool are grouped together into a single button with a flyout control.

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.

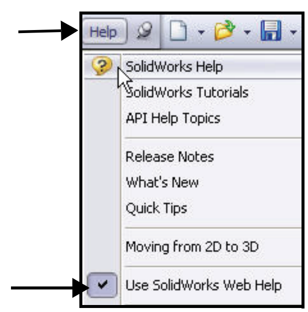


Mouse Buttons

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.

Getting Help

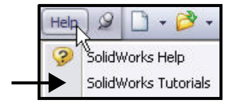


SolidWorks has a comprehensive Home Help Page function that is design to assist the new and experience user. It provides information on What's New, SolidWorks Glossary, New Release notes, and more.

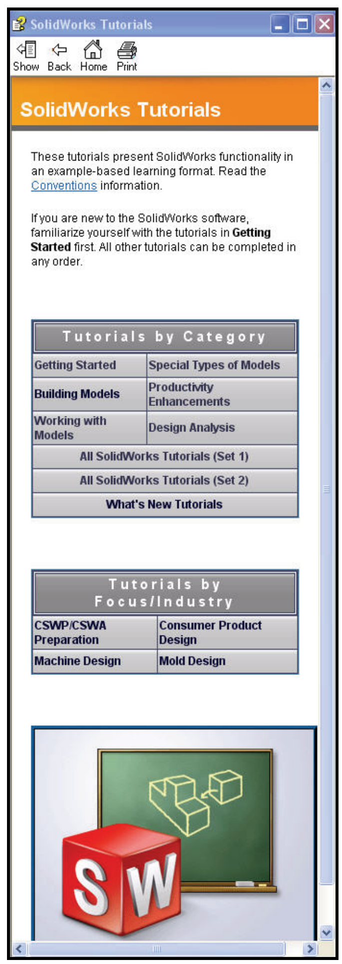
Click **Help**, **SolidWorks Help**  from the Menu Bar menu to view the comprehensive SolidWorks Home Help Page screen.

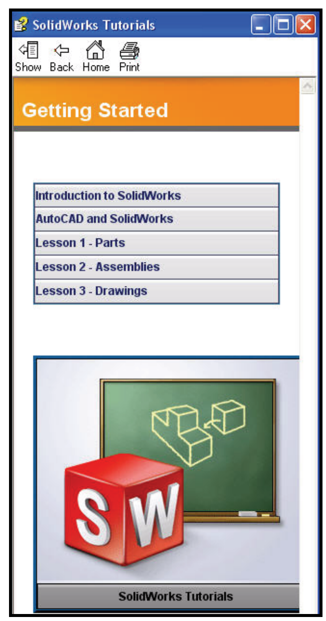
SolidWorks Tutorials

The SolidWorks Tutorials provide step-by-step lessons with sample files covering SolidWorks terminology, concepts, functions, features, What's New and many Add-Ins. Work or view the 30 minute lesson tutorials to learn and strengthen your skills.

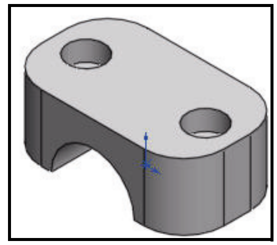


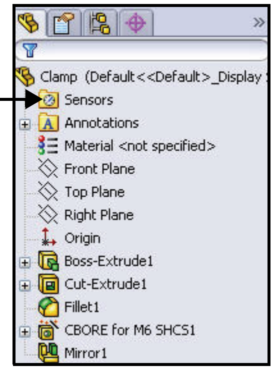
Click **Help**, **SolidWorks Tutorials** from the Menu Bar menu, or click the SolidWorks **Resources** tab from the Task Pane and click **Tutorials**. View the available tutorials. The tutorials are displayed by category.





Let’s Build Something!





The first part that you will build is the Clamp for the SeaBotix LBV150. To begin building the part, you need to open a new SolidWorks document. As you learned earlier, SolidWorks uses three kinds of documents: parts, assemblies, and drawings. Since the Clamp is a part, you will open a new part document.

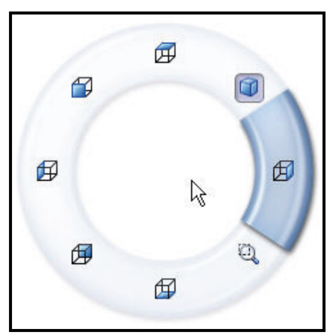
The new part document will be created based on a template. A template forms the foundation of a new document. The template can include user-defined parameters, annotations, or geometry.

Templates allow you to define your only specific parameters, and then open new documents with those customized settings already set. In this way, you define parameters only once and the new documents are created with the customized settings. This can save you a lot of time.

You can also create multiple templates for each document type. Today, you are going to use a provided part template since you have not yet customized any templates of your own.

Note: The Sensors tool located in the FeatureManager monitors selected properties in a part or assembly and alerts you when they deviate from the specified limit. There are four sensor types: *Mass properties, Measurement, Interference Detection*, and *Simulation data*.

Note: New in 2010 is the mouse gesture wheel. Right-click and drag in the Graphics area to display the wheel. Use the mouse gesture wheel as a shortcut to execute a command, similar to a keyboard shortcut. The user can customize the wheel for sketching, part, assemblies, and drawings.



Starting SolidWorks and Opening a New Part

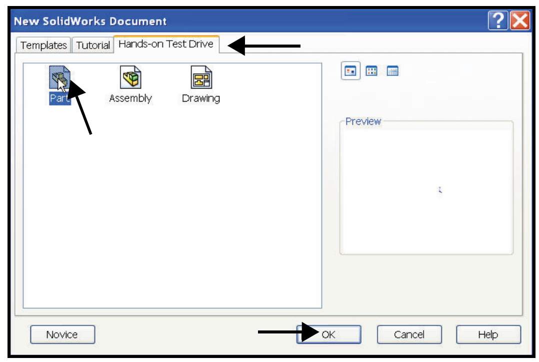
1 Start the SolidWorks application.

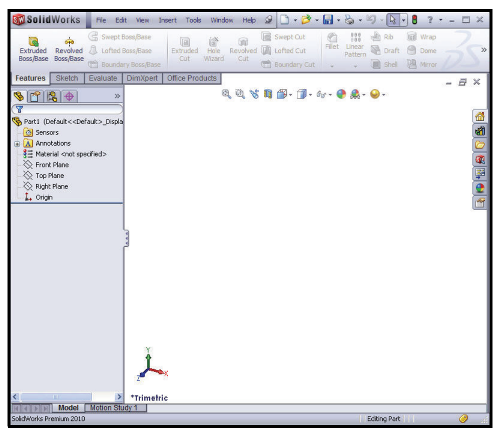
- Click the **Start** menu from the Windows interface.
- Click **All Programs**, **SolidWorks 2010**, **SolidWorks 2010**.

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

2 Open a new part.

- Click **New** from the Menu Bar toolbar. The New SolidWorks Document dialog box is displayed.
- Click the **Part** template from the Hands-on Test Drive tab.
- Click **OK** from the New SolidWorks Document dialog box. A new Part document window is displayed.

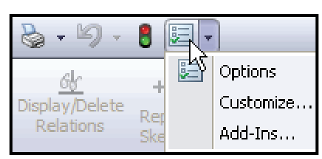


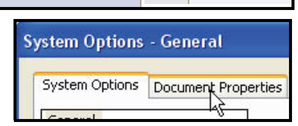


Note: Templates are part, assembly, and drawing documents that include user-defined parameters and are the basis for a new document. The Hands-on Test Drive tab was created for this class.

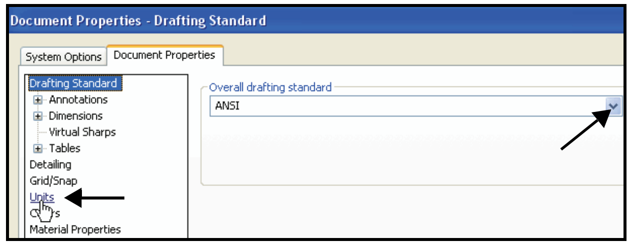
3 Set Document Properties.

- Click **Options**, **Document Properties** tab from the Menu Bar toolbar. The Document Properties - Drafting Standard dialog box is displayed. ANSI is the default Overall drafting standard used in this book.
- Select **ANSI** for Overall drafting standard from the drop-down menu.



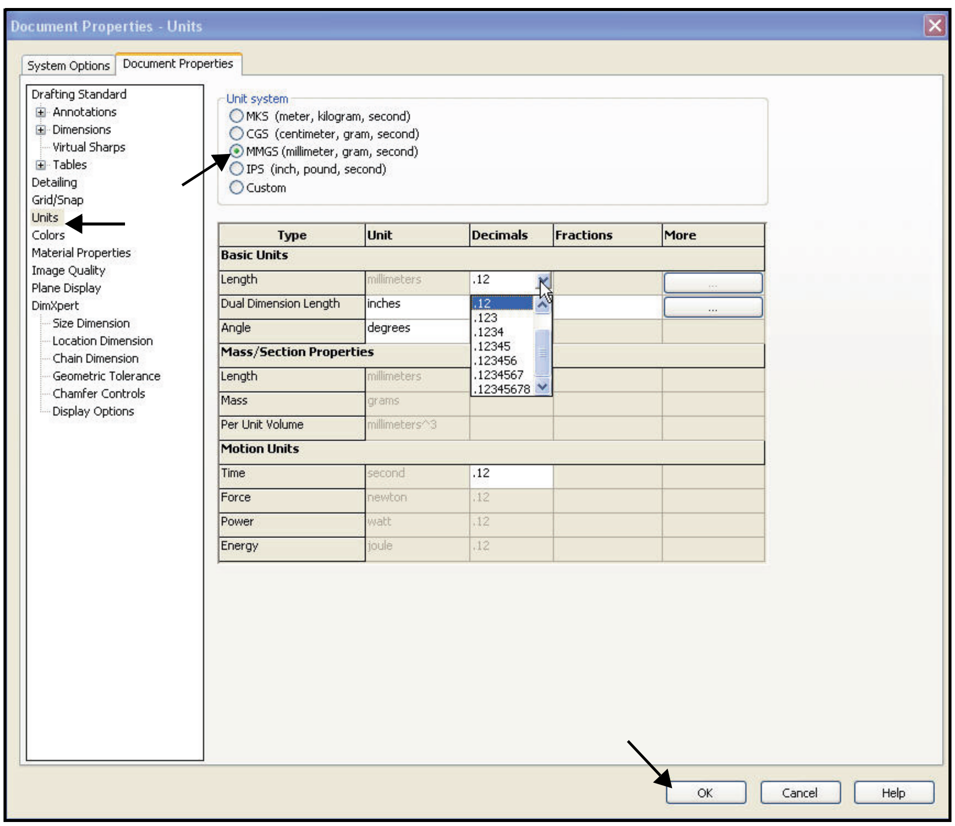


- Click the **Units** folder.
- Select **MMGS**, (millimeter, gram, second).
- Click inside the **Length Decimals** box.
- Select **.12** decimal places from the drop-down menu for Basic Units as illustrated.



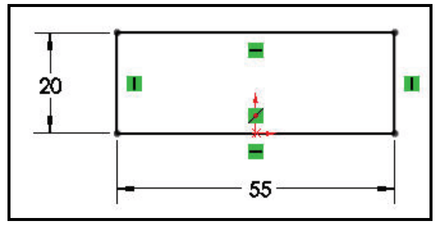
4 Accept the default settings.

- Click **OK** from the Document Properties - Units dialog box. Return to the SolidWorks Graphics area.

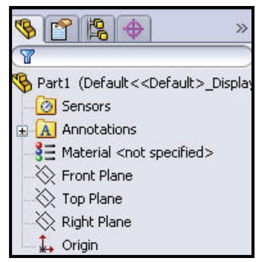


Begin Sketching

Solid models are built from features. Initially, features are based on 2D sketches. The sketch is the basis for a 3D model.



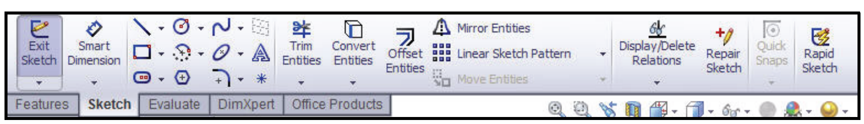
Create a 2D sketch that you will later make into a 3D solid. For the Clamp part, create a 2D sketch and insert dimensions as illustrated.



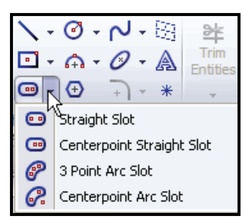
Sketches are flat or planar. You need a plane on which to sketch. A SolidWorks part contains three default sketch planes. They represent the Front, Top, and Right Planes. For the Clamp, use the Front Plane as the Sketch plane.

While sketching, you can select from a number of available sketch tools from the Sketch toolbar such as a Line sketch tool, Corner Rectangle Sketch tool, Circle sketch tool, Arc Sketch tool etc.

Create a sketch by clicking where you want the sketch to start. Move the cursor to where you want it to end, then click again.



Note: The Consolidated Slot Sketch toolbar provides the ability to sketch a Straight Slot using two end points, a Centerpoint Straight Slot from the center point, a 3 Point Arc Slot using three points along the arc, or a Centerpoint Arc Slot using the center point of the arc and the two end points.

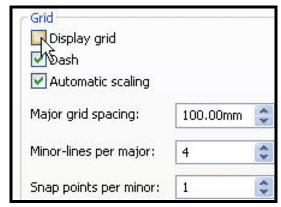


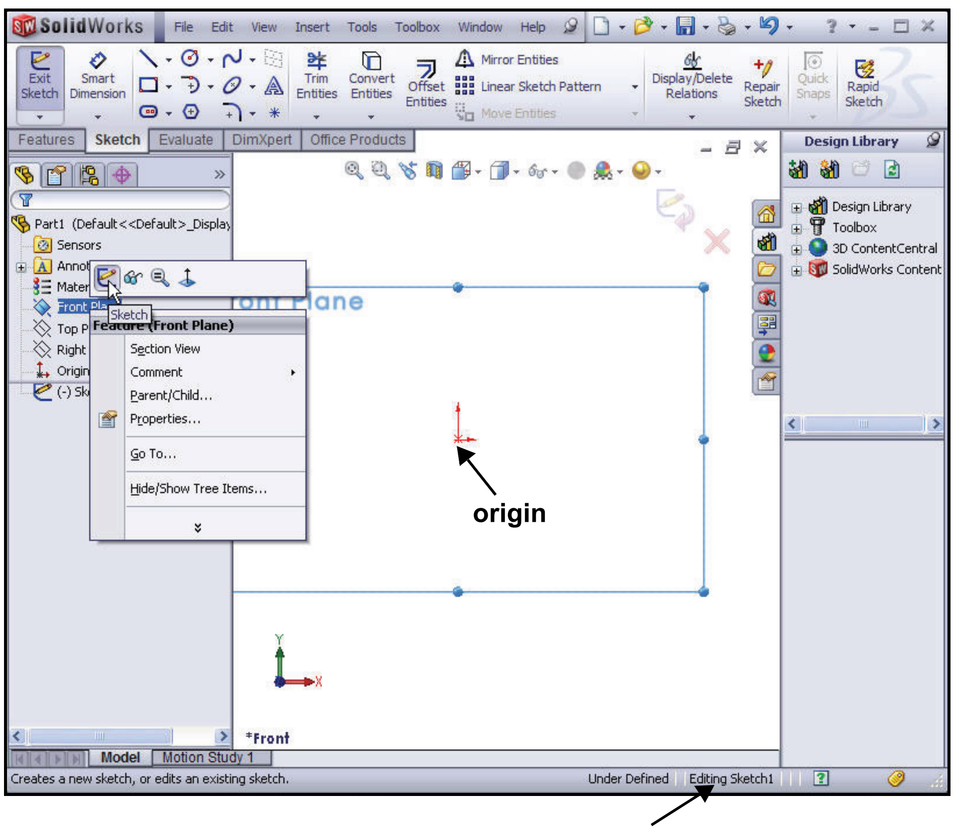
Selecting the Sketch Plane

1 Select the Front Plane.

- Right-click **Front Plane** from the FeatureManager.
- Click the **Sketch** tool from the Context toolbar as illustrated. The Front Plane is displayed in the Graphics area in the Front view. You are in the Sketch mode.

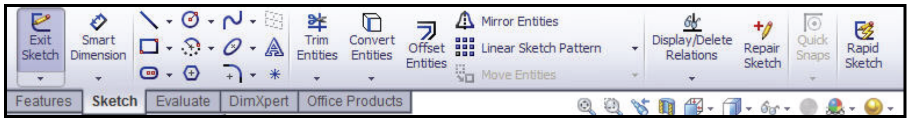
Note: The default grid is displayed if Grid is active.





Indications that you are in Sketch Mode

The Sketch toolbar is active.

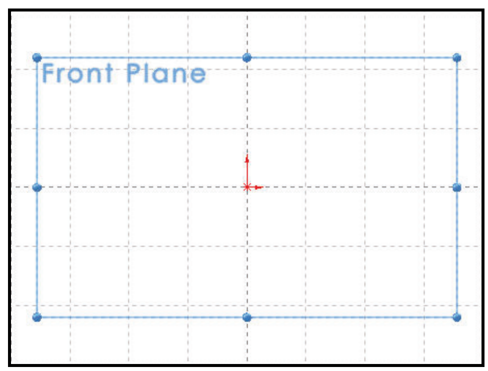


The title bar of the window tells you that you are in a sketch.

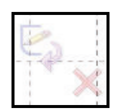
The status bar at the bottom of the window informs you that you are editing a

sketch. 

The grid display is setup in Options, Document Properties, Grid/Snap. You can turn the Grid on or off.

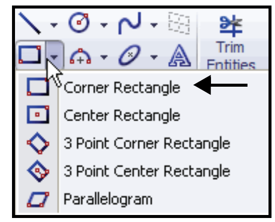


When numerous SolidWorks commands are active, a symbol or a set of symbols are displayed in the upper right corner of the Graphics area. This area is called the Confirmation Corner.



When you activate or open a sketch, the Confirmation Corner box displays two symbols. The first symbol, is the sketch tool icon. The second symbol, is a large red X. These two symbols supply a visual reminder that you are in an active sketch.

Sketching the Rectangle



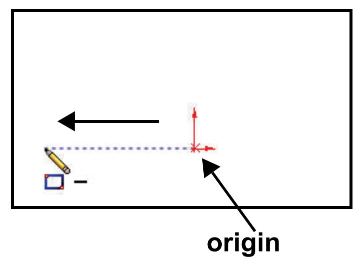
1 Sketch a rectangle.

- Click the **Corner Rectangle** tool from the Consolidated drop-down Sketch toolbar. The Rectangle PropertyManager is displayed. The Corner Rectangle tool is displayed on the mouse icon.

Note: Similar commands are grouped into Consolidated flyout buttons on the toolbar and in the CommandManager.

2 Start directly to the left of the origin.

- The Corner Rectangle-Horizontal symbol provides sketch feedback. This symbol informs you when you are directly to the left of the origin. Click a **position** directly to the left of the origin as illustrated.

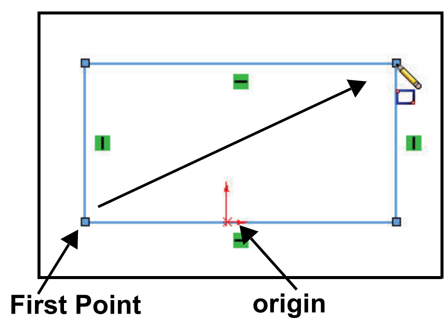


3 Complete the rectangle.

- Move **diagonally up** and to the right of the origin.
- Click a **position** to complete the rectangle as illustrated.

4 Zoom to fit.

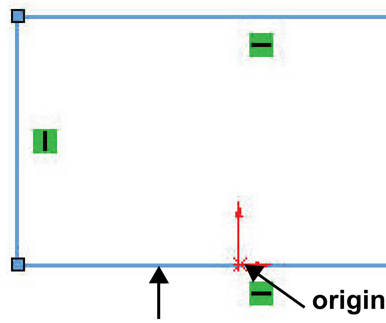
- Press the **f** key on the keyboard to return to the full Graphics area.



Note: When you sketch, click the mouse button, move the mouse pointer to the next location, and click the mouse button again.

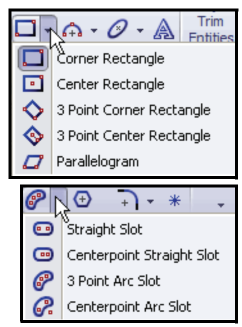
Adding Geometric Relations

The rectangle is sketched on the correct sketch plane, but it is not in the correct location. Instead of placing the rectangle at some point in space, relate it to the origin.



Note: When a sketch is active, a sketch origin is displayed in red and represents the (0,0,0) coordinate of the sketch.

SolidWorks provides the ability to add relations that link model elements together in a meaningful way. Build symmetry into the sketch. Apply a Midpoint Relation between the origin and the bottom horizontal line of the sketch.

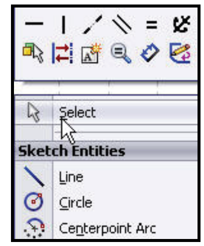


Note: SolidWorks provides various Sketch tools, (Center Rectangle, Centerpoint Straight Slot, etc.) that automatically inserts the needed geometric relations.

Relate the Origin to the Bottom Horizontal Line

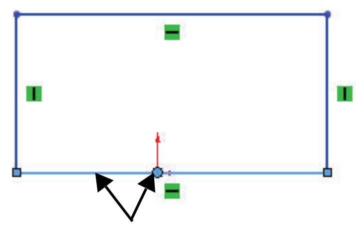
1 Select Entities.

- Right-click **Select** in the Graphics area to deselect the Corner Rectangle Sketch tool.



2 Add a Geometric relation.

- Click the **origin**. The origin is highlighted in the FeatureManager.
- Hold the **Ctrl** key down.
- Click the **bottom horizontal** line. The Properties PropertyManager is displayed.
- Release the **Ctrl** key.
- Click **Midpoint** from the Add Relations box. Midpoint0 is added to the Existing Relations box.



Selected Entities

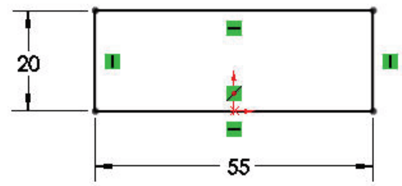
Note: SolidWorks provides the ability to either select available geometric relations from the Pop-up Context toolbar in the Graphics area or from the Properties PropertyManager.

3 View the results.

- Click **OK** from the Properties PropertyManager. The bottom horizontal line of the rectangle is symmetric with the origin. If the bottom horizontal line increases or decreases in size, the line will remain symmetric about the origin.

Defining the size

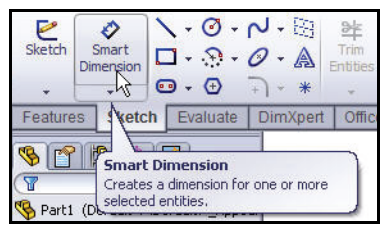
Now that you have the basic shape, we need to define the size. You do this by using dimensions. In SolidWorks, dimensions are not just static numbers that tell you the size of something. Instead, dimensions are used to change the size and shape of the model.



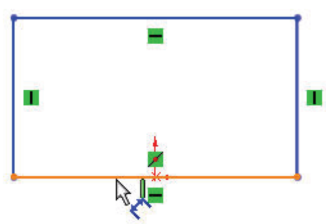
Dimension sketch entities and other objects with the Smart Dimension tool from the Sketch toolbar. The type of dimension is determined by the items which you select.

Example:

- If you select a line, the system creates a linear dimension.
- If you select a circle, the system creates a diameter dimension.
- If you select two parallel lines, the system creates a linear dimension between the two lines.

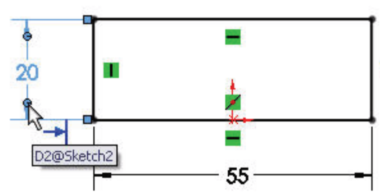


Some systems require the user to learn different commands for each type of dimension. Not so with SolidWorks. You will dimension the height and width of the rectangle using the Smart Dimension tool from the Sketch toolbar.



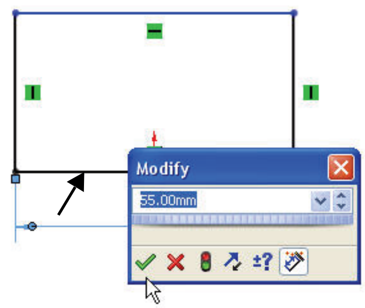
The Smart Dimension tool provides an icon feedback symbol.

Note: Click the arrow control point to modify the dimension arrow direction as illustrated.



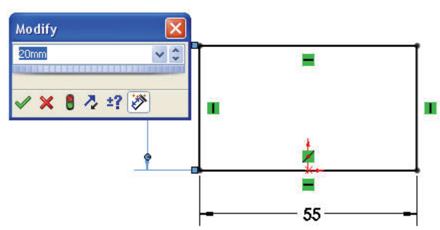
Dimensioning the Sketch

1 Dimension the sketch.



- Click the **Smart Dimension** tool from the Sketch toolbar.
- Click the **bottom horizontal** line.
- Click a **position** below the horizontal line. A dimension appears with the Modify dialog box displaying the current dimension value. Depending on how you sketched the profile, your value may be different from the one shown in the illustration.
- Enter **55**mm in the Modify dialog box.
- Click the **green checkmark** button. This saves the value and closes the Modify dialog box. The dimension forces the width of the rectangle to be 55mm.

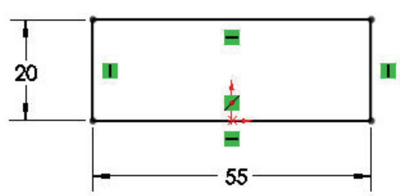
Note: The Smart Dimension tool automatically provides you with the correct dimension units.



2 Zoom to fit.

- Press the **f** key on the keyboard to return to the full Graphics area.

3 Add a vertical dimension.

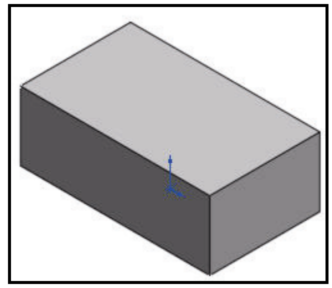


- Click the **left vertical** line.
- Click a **position** to the left of the vertical line. The Modify dialog box is displayed.
- Enter **20**mm in the Modify dialog box.
- Click the **green checkmark** button. The dimensioning of the profile is complete. The profile is now fully defined in size, shape, and position. The sketch geometry is displayed in black. The color black indicates that the profile is fully defined.

4 Turn off the Smart Dimension tool.

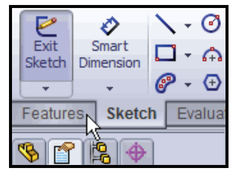
- Press the **Esc** key to deactivate the Smart Dimension tool from the Sketch toolbar.

Extruding a Solid



One way to make a solid feature is to extrude it. An Extruded Boss/Base feature adds material to the model.

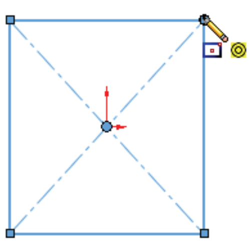
Extruding builds the solid normal to the sketch plane for some specified distance. An extruded boss is a profile that has been projected for a specified distance.



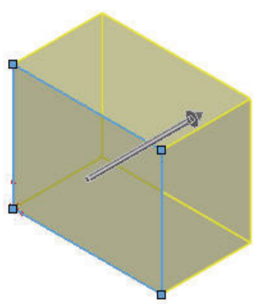
There are numerous options for extruding a sketch such as draft angles, end conditions, and depth. These options allow you to create smart, manufacturable models. However, most of these options are beyond the scope of this quick introduction, so we will just keep it simple.

To Create an Extruded Feature:

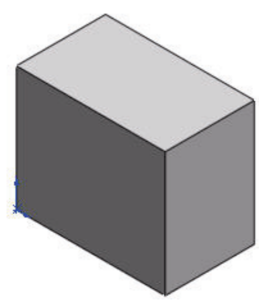
1. Select a sketch plane.
2. Sketch a 2D profile.
3. Extrude the sketch perpendicular to the sketch plane.



Sketch the 2D profile



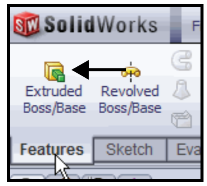
Extrude the sketch



Results

When you use a sketch profile from existing geometry to create an Extruded Boss/ Base feature, the extrusion becomes part of the model that it was extruded from. They are now one part.

Creating an Extruded feature

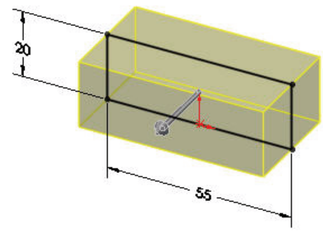


1 Extrude the feature.

- Click the **Features** tab from the CommandManager. The Features toolbar is displayed.
- Click the **Extruded Boss/Base** tool from the Features toolbar. The Boss-Extrude PropertyManager is displayed.

2 Set the End Condition and Depth.

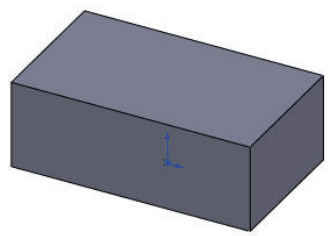
- Select **Mid Plane** for End Condition in Direction 1.
- Enter **32**mm for Depth. You can use the up and down arrow buttons next to the Depth box to change the value by 10mms (default in system options) at a time.



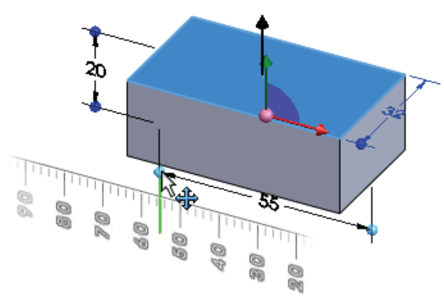
Note: The default Extruded direction is towards the front. The default End Condition is Blind. The Mid Plane option is utilized to incorporate symmetry into the design intent.

3 Accept the values and view the completed feature.

- Click **OK** from the Boss-Extrude PropertyManager. The Boss-Extrude1 feature is created.

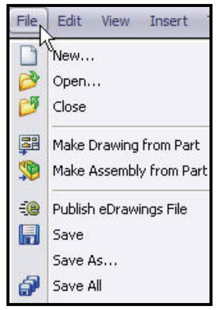


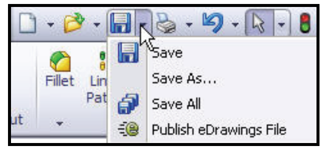
Tip: Instant3D provides the ability to click and drag geometry and the dimension manipulator points to resize or create features directly from the Graphics area. Use the on-screen ruler to measure your modifications. This is addressed later in the book.



Saving Your Work

The work that you perform in SolidWorks is contained in part, assembly, and drawing files. You can save part, assembly, and drawing files to the hard drive of your system. In general, you will have folders on your system that organize your files, usually by project.





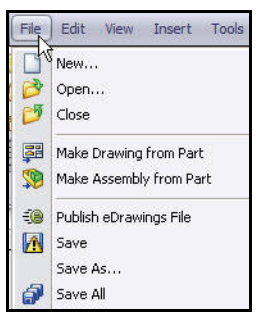
Note: This book is written with the assumption that your files are in and saved to the SolidWorks Test Drive folder.

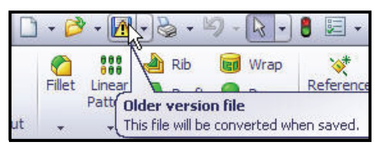
You can save your files as often as you wish. However, there are really only two situations that require you to save your work:

- After you have done something you want to keep.
- Before you try something that you are not sure will work.

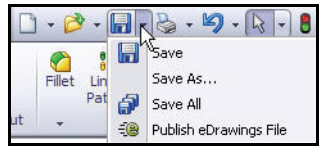
Saving protects work that you have already done. If you try a technique and accidently get results that you didn't anticipate, you can return to your saved file.

Note: Until a file is converted to the current version of SolidWorks and saved, a warning icon is displayed on the Save tool as illustrated.





Saving the Clamp Part

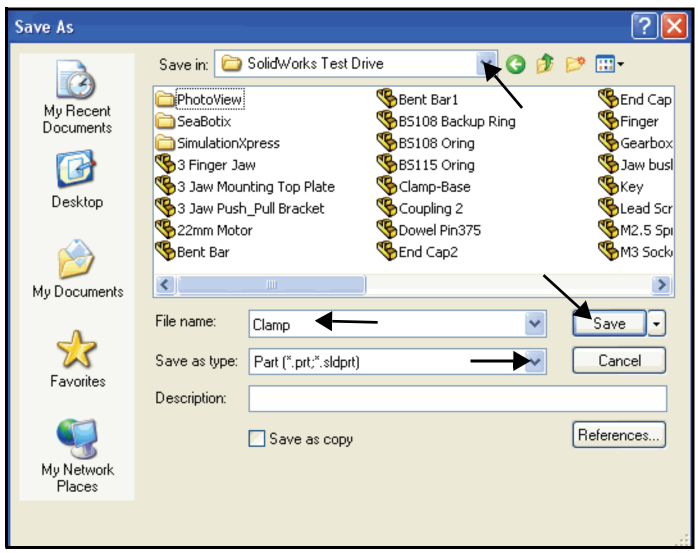


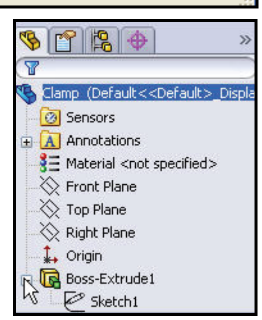
1 Save the Clamp.

- Click **File**, **Save** or click **Save** from the Menu Bar toolbar.
- Specify the Save in: folder, **SolidWorks Test Drive**.
- Select **Part** for Save as type.
- Enter **Clamp** for File name.
- Click **Save**. The extension *.sldprt is added automatically to the part name.

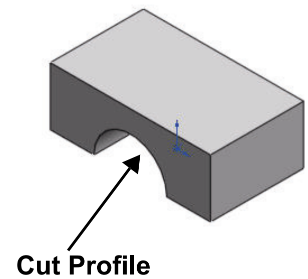
Note: The Clamp FeatureManager is displayed. Sketch1 is fully defined!

.





Creating the Cut Profile



You have the basic shape of the Clamp. The next step is to make the hole that will be used to attach the Clamp to the MiniGrab assembly.

The hole is an Extruded Cut feature. Earlier you extruded a profile to make the shape of the Clamp. To create an Extruded Cut feature, you need a profile. The profile is extruded into an existing solid to remove material.

Rather than sketch on a reference plane, you will sketch directly on a planar face of the model. In general, most sketches are created on planar faces of existing geometry. In this way, you can build on what you have done without having to create the same geometry over again.

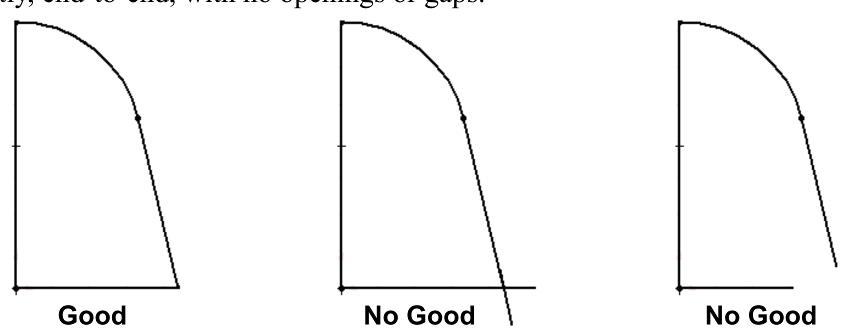
To create the Extruded Cut feature, you need to create a profile to extrude. Once again, you will do this by sketching.

The Status of a Sketch

You may notice that some of the sketch lines are black and some are blue. SolidWorks uses these colors to indicate which lines are fully defined and which lines are underdefined. While SolidWorks has no problems using underdefined sketches, for our purposes today, we will use fully dimensioned and fully defined sketches.

Neatness Counts

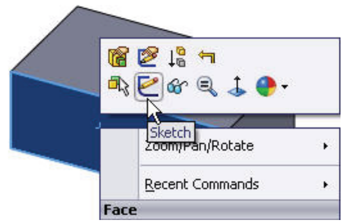
As is true in any CAD system, neatness counts. Your sketch lines should connect neatly, end-to-end, with no openings or gaps.

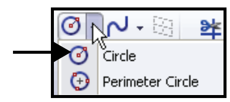


Sketching the Circle

1 Select the face and open a sketch.

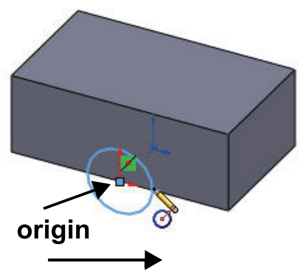
- Right-click the **front face** of the Clamp. The front face is the Sketch plane and is highlighted in the Graphics area.
- Click the **Sketch** tool from the Context toolbar. The Sketch toolbar is displayed.





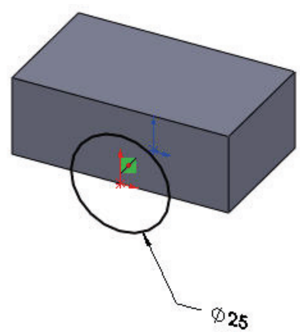
2 Sketch a circle.

- Click the **Circle** tool from the Consolidated Sketch toolbar. The Circle PropertyManager is displayed.
- Click the **origin**.
- Click a **position** to the right of the origin as illustrated.



3 Dimension the circle.

- Click the **Smart Dimension** tool from the Sketch toolbar.
- Click the **circumference** of the circle.
- Click a **position** diagonally to the lower right of the origin.
- Enter **25**mm for the diameter of the circle.



4 Accept the value.

- Click the **green checkmark** button from the Modify dialog box.
- Click **OK** from the Dimension PropertyManager.

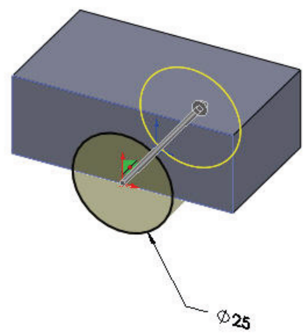
5 Zoom to fit.

- Press the **f** key on the keyboard to return to the full Graphics area.

Tip: Press the **z** key to Zoom out.

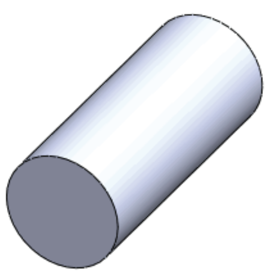
Making the Cut

The next step is to make a hole in the Clamp. You will do this by sketching and extruding a circle as a cut that removes material, rather than as a Boss/Base feature, that adds material. Extruding a cut is very similar to extruding a Boss/ Base feature except that you are projecting the sketch profile into the existing model. To create the cut, apply the Extruded Cut tool and specify the Through All End Condition option in Direction 1.

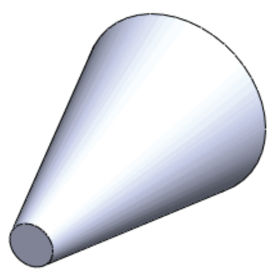


Draft Angle

An Extruded Cut feature that has no draft angle is one in which the sides of the cut are parallel. A draft angle projects the cut along an increasing angle so that one end of the cut is wider than the other.

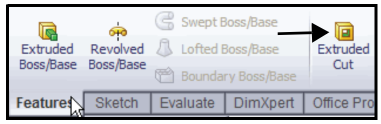


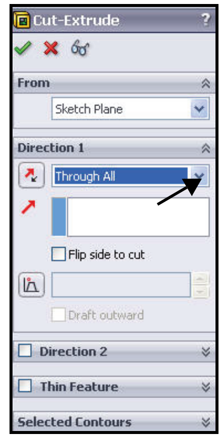
No Draft



Draft

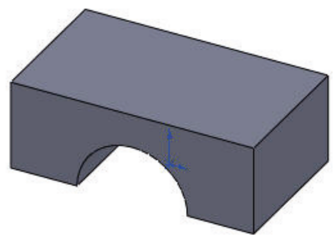
Creating an Extruded Cut Feature





1 **Create an Extruded Cut feature.**
- Click the **Features** tab from the CommandManager.
- Click the **Extruded Cut** tool from the Features toolbar. The Cut-Extrude PropertyManager is displayed.

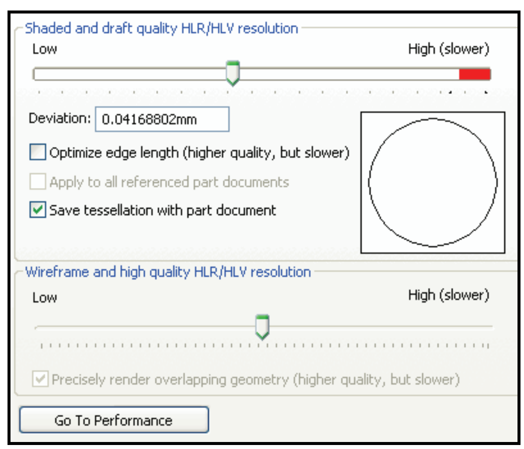
2 **Set the End Condition for Direction 1.**
- Select **Through All** for the End Condition in Direction 1. The direction arrow points towards the back.



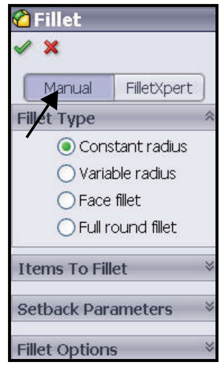
3 **Accept the default values and view the results.**
- Click **OK** from the Cut-Extrude PropertyManager. The Cut-Extrude1 feature is displayed in the FeatureManager.

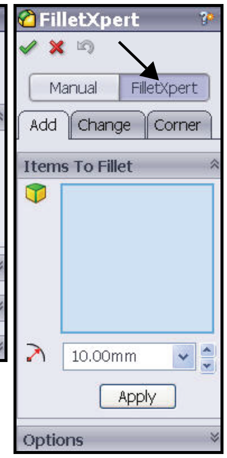
Note: The display of geometry on your system may appear somewhat different from the illustrations. The lines may appear rougher. This is called tesselation.

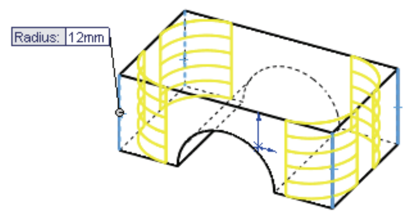
Tesselation or line display is related to the performance of the computer. Higher quality graphics or higher system settings will improve model appearance.



Inserting a Fillet Feature







The Fillet feature rounds sharp edges and faces. Utilize the Fillet feature to round the sharp vertical edges of the Clamp.

The Boss-Extrude1 and the Cut-Extrude1 feature requires a sketch plane and a sketch. The Fillet feature requires an edge or face with a specified radius.

In general, it is best to follow these few rules when inserting a fillet:

1. Add larger fillets before smaller ones. When several fillets converge at a vertex, create the larger fillets first.
2. Add drafts before fillets. If you are creating a molded or cast part with many filleted edges and drafted surfaces, in most cases you should add the draft features before the fillets.
3. Save cosmetic fillets for last. Try to add cosmetic fillets after most other geometry is in place. If you add them earlier, it will take longer to rebuild the part.

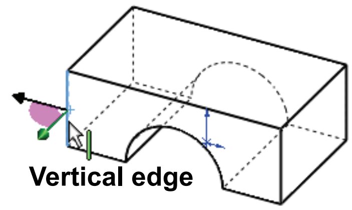
Fillet / FilletXpert PropertyManager

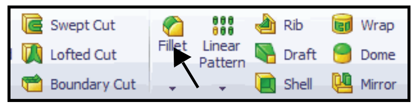
The FilletXpert PropertyManager is displayed when you click the FilletXpert tab in the Fillet PropertyManager. The FilletXpert manages, organizes, and reorders constant radius fillets for you so you can concentrate on your design intent. Use the Add tab to create new constant radius fillets. Use the Change tab to modify existing fillets. Use the Corner tab to create and manage fillet corner features where exactly three filleted edges meet at a single vertex. The PropertyManager remembers your last used state. Two PropertyManager tabs are available:

1. **Manual**. Use this tab to maintain control at the feature level.
2. **FilletXpert**. Use this tab when you want the SolidWorks software to manage the structure of the underlying features for a constant radius fillet. The FilletXpert tab automatically invokes the FeatureXpert and will reorder fillets when required.

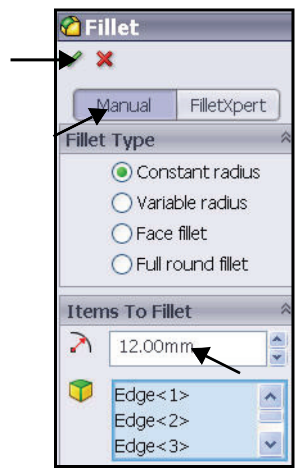
Inserting a Fillet Feature with the Manual Tab Option

1 Select four edges.





- Click the **Hidden Lines Visible** tool from the Heads-up View toolbar.
- Click the **vertical edge** of the Clamp part as illustrated. Note the mouse icon feedback symbol for an edge.
- Hold the **Ctrl** key down.
- Click the other **three vertical edges**. The four vertical edges are highlighted, displayed in blue.
- Release the **Ctrl** key.

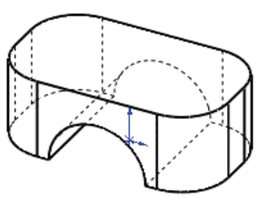


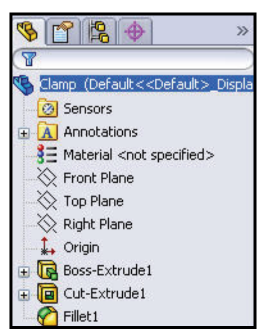
- Click the **Fillet** tool from the Features toolbar. The Fillet PropertyManager is displayed. The selected entities are displayed in the Items To Fillet box.
- Click the **Manual** tab from the Fillet PropertyManager.

2 Set the fillet radius.

- Enter **12**mm for Radius. Constant radius is selected by default.

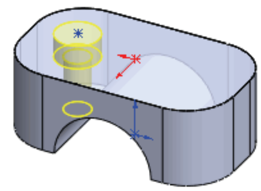
3 Accept the values and view the results.

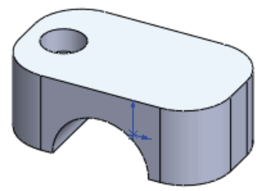




- Click **OK** from the Fillet PropertyManager. The Fillet1 feature is displayed in the FeatureManager design tree. View the results in the Graphics area.

Using the Hole Wizard Feature

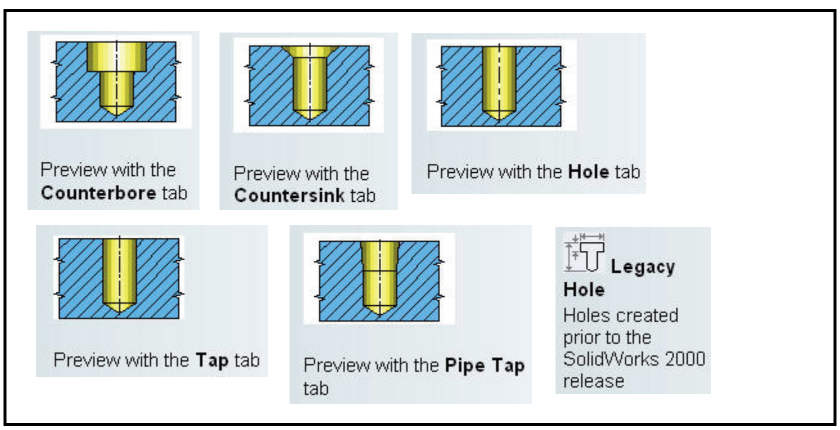




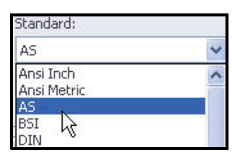
When you create a hole using the Hole Wizard, the type and size of the hole, based on the description, appears in the Hole Wizard FeatureManager. You can create holes on a plane with the Hole Wizard as well as holes on planar and non-planar faces. Holes on a plane allow you to create holes at an angle to the feature.

Functionality includes tabs for the following hole types:

- Counterbore
- Countersink
- Hole
- Tap
- Pipe Tap
- Legacy

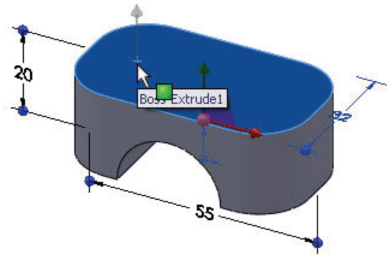


Note: New in SolidWorks 2010 is the Australian Standard option in the Hole Wizard.



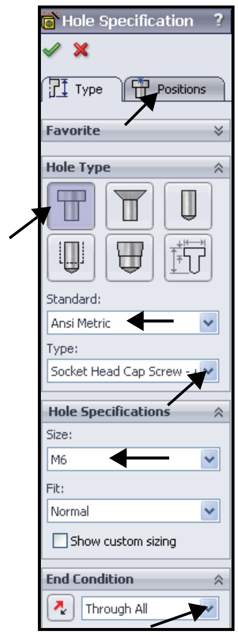
Inserting the Hole Wizard Feature

1 Insert the Hole Wizard feature.



- Click the **Shaded With Edges** tool from the Heads-up View toolbar.
- Click the **upper left top** face of the Clamp part as illustrated. Boss-Extrude1 is highlighted in the FeatureManager.

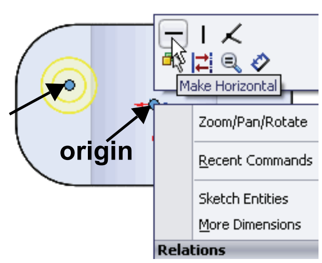
2 Set hole specification.



- Click the **Hole Wizard** tool from the Features toolbar. The Hole Specification PropertyManager is displayed. The Type tab is selected by default.
- Click the **Counterbore** button.
- Select **Ansi Metric** for Standard.
- Select **Socket Head Cap Screw** for Type.
- Select **M6** for Size.
- Select **Through All** for End Condition.

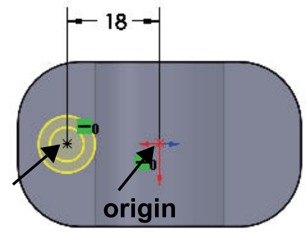
3 Add a horizontal relation.

- Click the **Positions** tab. The Hole Position PropertyManager is displayed. The Point Sketch tool is active.
- Right-click **Select** in the Graphics area to deselect the Point Sketch tool.
- Click the **Top** view tool from the Heads-up View toolbar.
- Press the **f** key to fit the model to the Graphics area.
- Click the **blue centerpoint** of the Counterbore hole. The location of your blue centerpoint may vary per the illustration. The location is based on your initial left top face selection.
- Hold the **Ctrl** key down.
- Click the **origin**. The Properties PropertyManager is displayed.
- Release the **Ctrl** key.
- Right-click **Make Horizontal** from the Context toolbar in the Graphics area.
- Click **OK** from the Properties PropertyManager. The Hole Position PropertyManager is displayed.

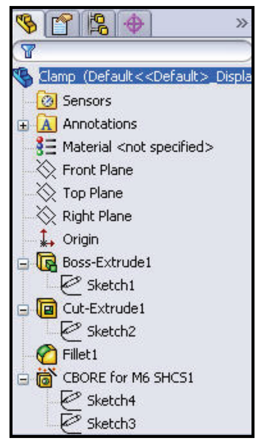


4 Add a dimension.

- Click the **Smart Dimension** tool from the Sketch toolbar.
- Click the **centerpoint** of the Counterbore hole.
- Click the **origin**.
- Click a **position** above the profile.
- Enter **18**mm in the Modify dialog box.

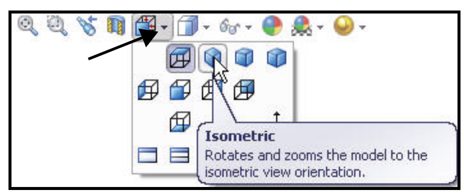


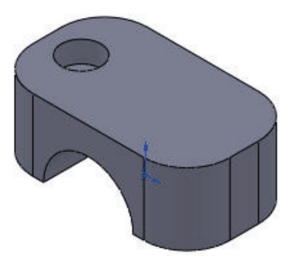
- Click the **green checkmark** button from the Modify dialog box.
- Click **OK** from the Dimension PropertyManager.
- Click **OK** from the Hole Position PropertyManager. CBORE for M6 SHCS1 is displayed in the FeatureManager.



5 Display an Isometric view

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

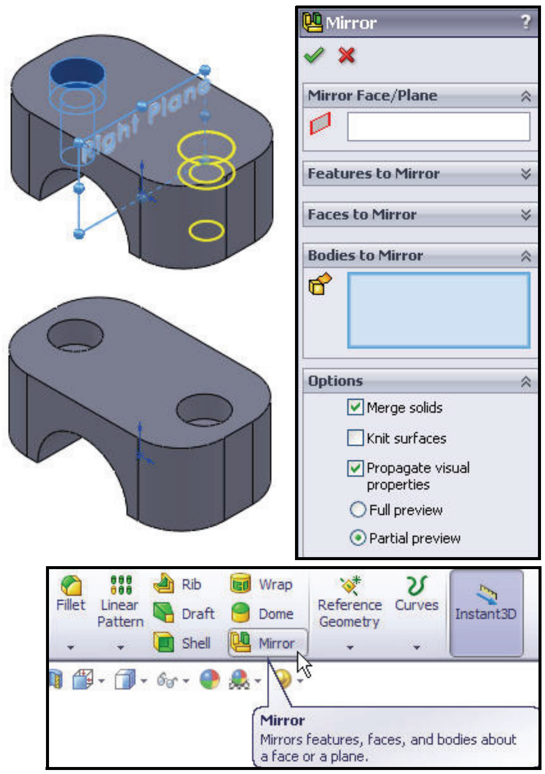




Using the Mirror Feature

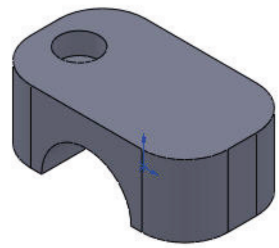
The Mirror feature creates a copy of a feature, or multiple features mirrored about a face or a plane. You can select the feature or you can select the faces that comprise the feature.

Note: If you modify the original feature, the mirrored copy is updated to reflect your changes.

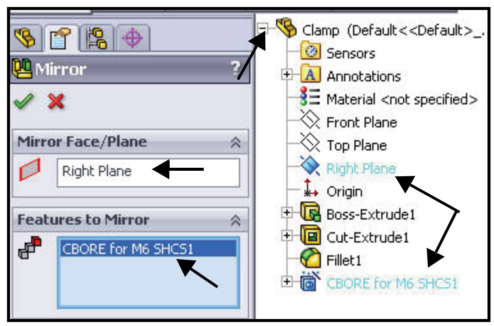


Inserting the Mirror Feature

1 Mirror the counterbore hole.



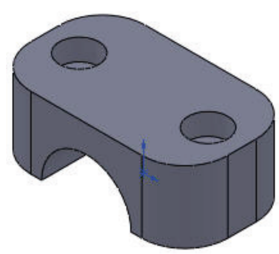
- Click the **Mirror** tool from the Features toolbar. The Mirror PropertyManager is displayed.
- **Expand** Clamp from the flyout FeatureManager in the Graphics area.
- Click **Right Plane** from the flyout FeatureManager. Right Plane is displayed in the Mirror Face/Plane box.



Note: CBORE for M6 SHCS1 is selected and is displayed in the Features to Mirror box.

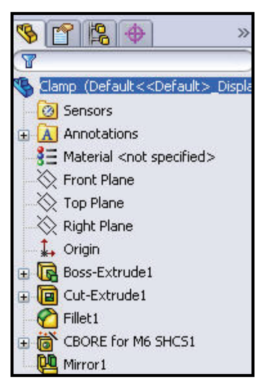
2 Accept the values and view the results.

- Click **OK** from the Mirror PropertyManager. Mirror1 is displayed in the FeatureManager design tree and in the Graphics area.



3 Save the Clamp.

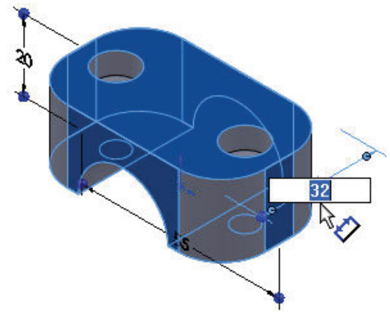
- Click **Save** from the Menu Bar toolbar. View the FeatureManager.



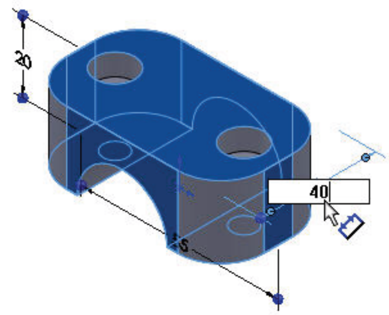
Modifying Dimensions

You can apply dimension values to selected configurations as follows:

- In a part, you can control the dimensions in sketches and in the feature definitions.
- In an assembly, you can control dimensions that belong to assembly features. This includes mates, assemblies, features, cuts and holes, and component patterns.

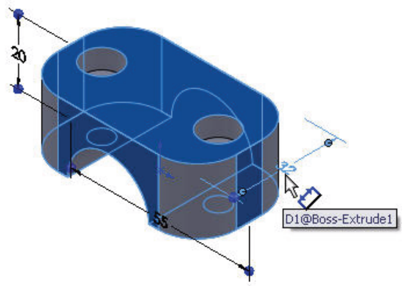


You can change a dimension in a sketch, part, assembly, or drawing in the Modify dialog box.



Note: Instant3D is enabled in this book. To toggle Instant3D mode, click the **Instant3D**  tool from the Features toolbar.

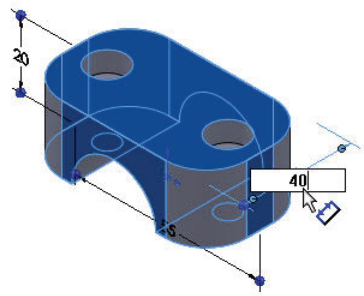
Modifying Dimensions



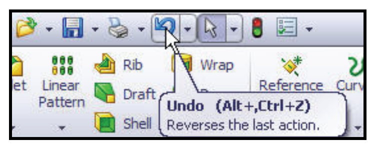
1. **View the Boss-Extrude1 dimensions.**
   - Click **Boss-Extrude1** from the FeatureManager.
   - Click the **32**mm dimension in the Graphics area.

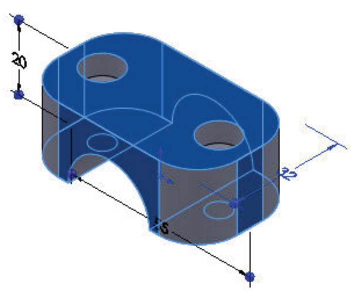
Note: For improved visibility, click and drag dimensions off of the model.

2. **Modify the width dimension of the Boss-Extrude1 feature.**
   - Enter **40**mm. View the updated model in the Graphics area.



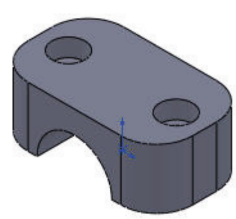
3. **Return to the original dimension.**
   - Click the **Undo** tool from the Menu Bar toolbar to return to the original dimension.
   - Click **Boss-Extrude1** from the FeatureManager to view the original dimensions.





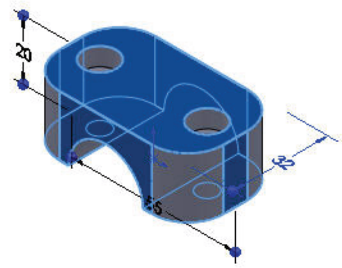
4. **Save the Clamp to the SolidWorks Test Drive folder.**
   - Click **inside** the Graphics area.
   - Click **Save** from the Menu Bar toolbar.

Tip: Click a feature in the Graphics area or in the FeatureManager to display the dimensions.



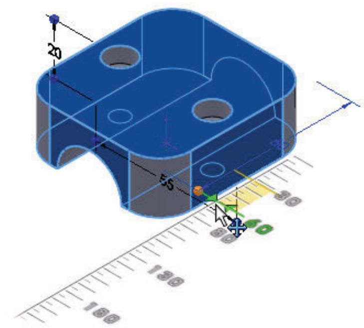
Instant3D

Instant3D provides the ability to quickly create and modify the model geometry, (either a sketch or feature) using drag handles and the Instant3D on-screen ruler.



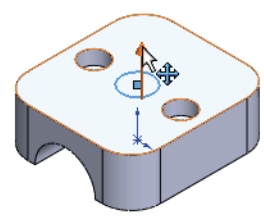
Select a feature from the Graphics area or from the FeatureManager. The drag handles are displayed. Drag geometry and dimension manipulators to resize.

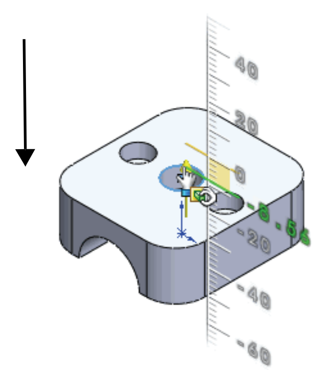
Use the on-screen ruler and click a dimension to precisely measure model modifications.



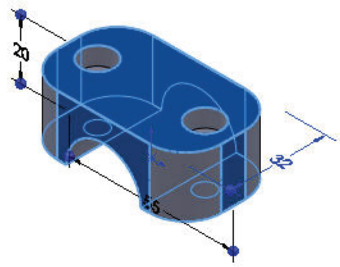
Note: Instant3D is enabled by default. To toggle Instant3D mode, click the **Instant3D** tool from the Features toolbar.

Note: If you create a sketch on an existing feature's face, you can create either a Extruded Boss/Base or Extruded Cut feature depending on the direction pull of the illustrated arrow.





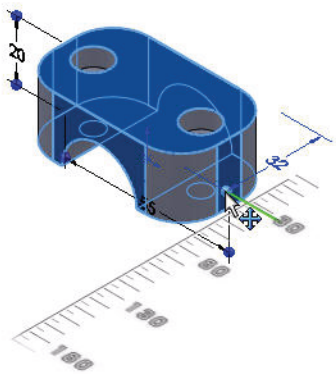
Utilizing Instant3D



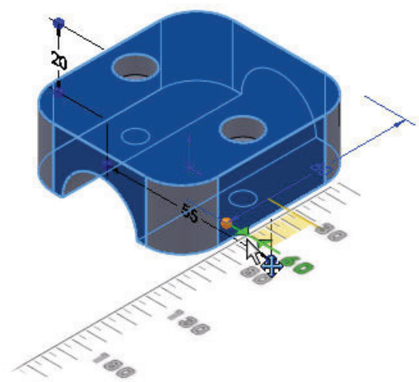
1 **Display the on-screen ruler and the manipulator points.**
   - Click **Boss-Extrude1** from the FeatureManager. View the manipulators points.

2 **Zoom out on the model.**
   - Press the **z** key from the keyboard to zoom out on the model.

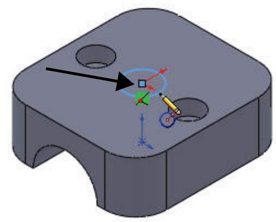
3 **Modify the Boss-Extrude1 feature.**
   - Click and drag the illustrated **manipulator point** of the 32 dimension to the front. View the on-screen ruler. The on-screen ruler displays the model feature dimension.
   - Click a **dimension** on the on-screen ruler. View the results in the Graphics area.



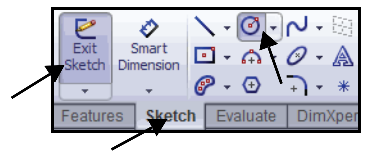
Note: Your dimension may vary from the illustration. Do not worry. You will not save this part.



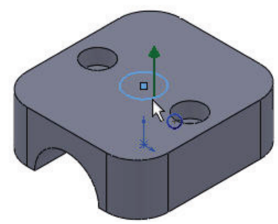
4 Create an Extruded Cut feature.



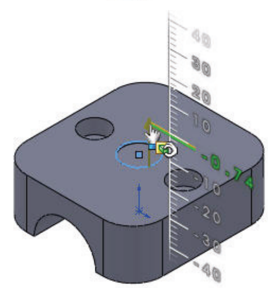
- Click the **top face** of the Clamp as illustrated between the two holes.
- Click the **Sketch** tab from the CommandManager.
- Click the **Circle** tool from the Sketch toolbar. The Circle PropertyManager is displayed.
- Click the **point** on the top face.
- Click a **position** to the right of the point as illustrated.



- Click the **Exit Sketch** tool from the Sketch toolbar.
- Click the **circumference** of the sketch as illustrated. An direction arrow is displayed in the upward direction.

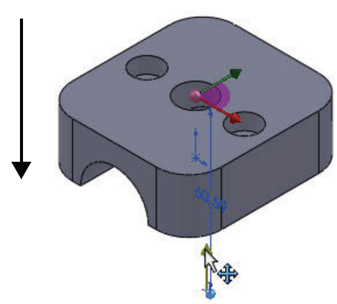


- Click an drag the **arrow downward**. The on-screen ruler is displayed.
- Click a **dimension** on the on-screen ruler. You just created an Extruded Cut feature with the Instant3D tool.



- **Rotate** the model with the middle button to view the Extruded Cut feature.

Note: Dimensions may vary for the illustration.

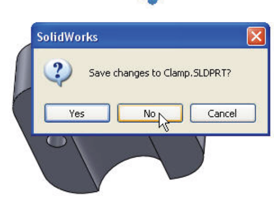


5 Close the model.

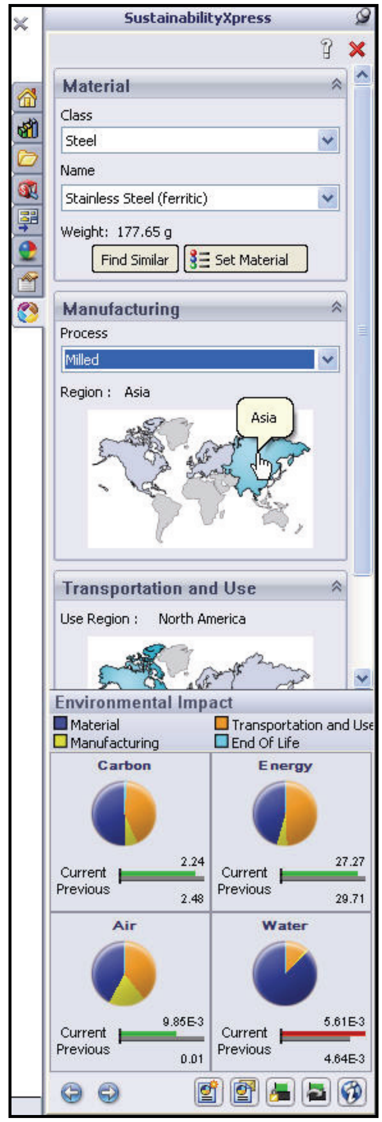
- Click **File**, **Close** from the Menu Bar menu.

6 Do not save the model.

- Click **No** to save changes to Clamp? The Clamp part is used later in this book.



Applying Sustainability



Every license of SolidWorks 2010, contains a copy of SolidWorks SustainabilityXpress. This is great news, because it is a tool that gives designers insight into the environmental impact of their designs.

The environmental impact is calculated in four key areas: *Carbon Footprint*, *Energy Consumption, Air Acidification* and *Water Eutrophication*.

Material and Manufacturing Process region and Transportation and Usage region are used as input variables.

Two SolidWorks Sustainability products are available:

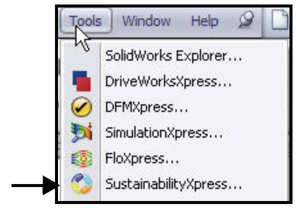
- **SolidWorks SustainabilityXpress**. Handles part documents and is included in the core software.
- **SolidWorks Sustainability**. Handles parts and assemblies, available as a separate product. Other functionality includes configuration support, expanded reporting, and expanded environmental impact options.

SolidWorks Sustainability provides real-time feedback on key impact factors in the Environmental Impact Dashboard, which updates dynamically with any changes. You can generate customize reports to share the results.

Applying SolidWorks Sustainability

1 Apply SolidWorks Sustainability. Identify the Environment impact of a part.

- **Open** the Clamp part.
- Click **Tools**, **SustainabilityXpress** from the Main menu.



View the Sustainability Palette located in the Task Pane area. View your options and categories.

2 Select Material Class.

- Select **Steel** from the drop-down menu.

3 Select Material Name.

- Select **Stainless Steel (ferritic)** from the drop-down menu.

4 Select Manufacturing Process.

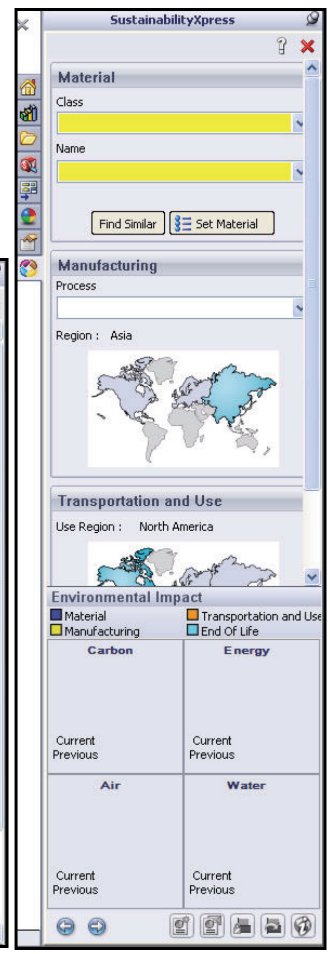
- Select **Milled** from the drop-down menu.

5 Select the Manufacturing Process Region.

- Accept the default setting: **Asia**.

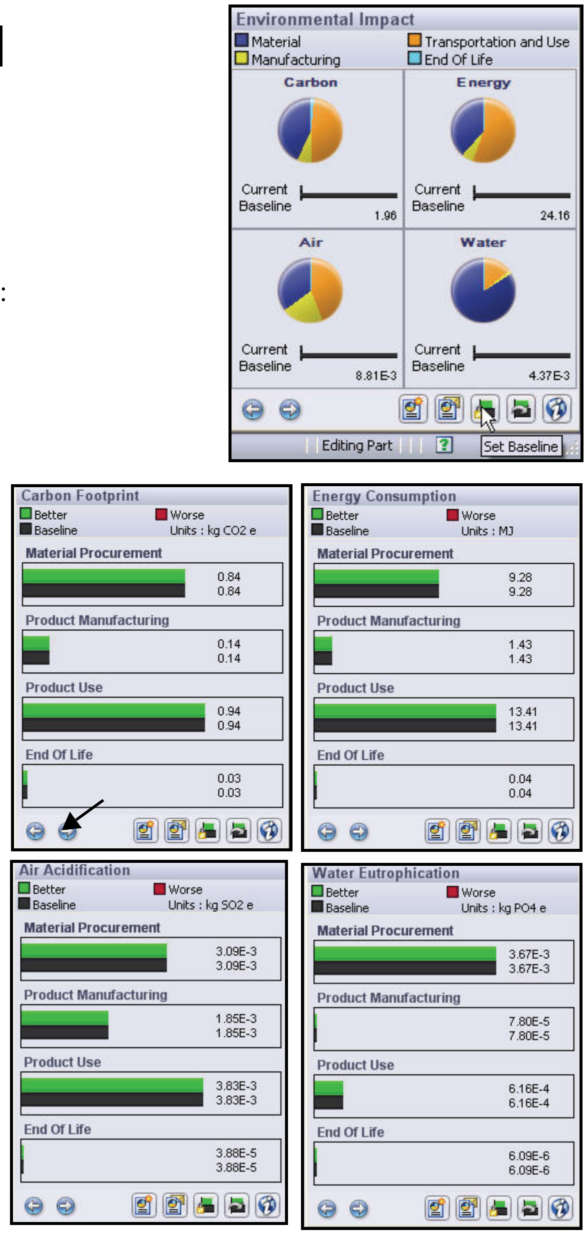
6 Select the Part Transportation and Usage Region.

- Accept the default setting: **North America**.



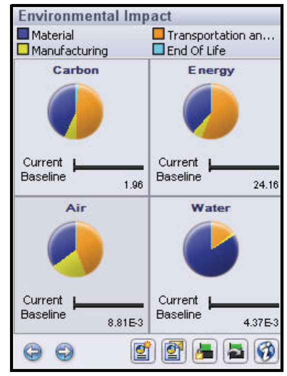
7 **Set the Baseline.**

- Click the **Set Baseline** tool from the bottom of the Environmental Impact screen. The Environmental Impact of this part is displayed. The Environmental Impact is calculated in four key areas: *Carbon Footprint*, *Energy Consumption, Air Acidification* and *Water Eutrophication*.
- Click **inside** the Carbon box to display a Baseline bar chart of the Carbon Footprint.
- Click the **right arrow** to move to the next (Energy Consumption) impact screen to display a Baseline bar chart.
- Click the **right arrow** to move to the next (Air Acidification) impact screen to display a Baseline bar chart.
- Click the **right arrow** to move to the next (Water Eutrophication) impact screen to display a Baseline bar chart.



8 Return to the Baseline Screen.

- Click the **right arrow** to move back to the original impact screen. In the next section, compare the baseline design to a different design so you can determine if you can make a meaningful design change. Let's compare the present material (Stainless Steel (ferritic)) to Nylon 6/10.



9 Select a new Material Class.

- Select **Plastics** from the drop-down menu.

10 Select a new Material Name.

- Select **Nylon 6/10** from the drop-down menu.

11 Select a new Manufacturing Process.

- Select **Injection Molded** from the drop-down menu.

12 Select the Manufacturing Process Region.

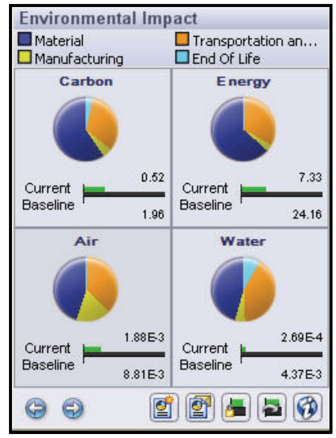
- Accept the default setting: **Asia**, same as the baseline study.

13 Select the Part Transportation and Usage Region.

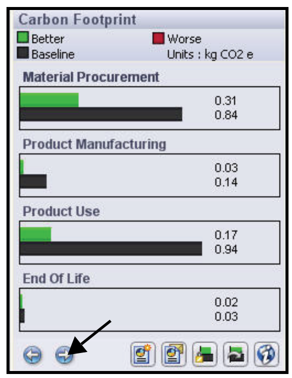
- Accept the default setting: **North America**, same as the baseline study.

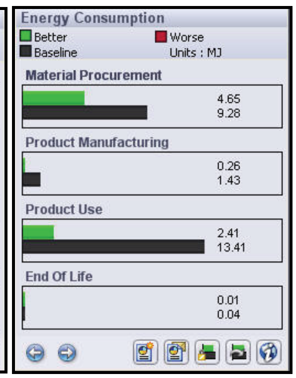
14 View the results.

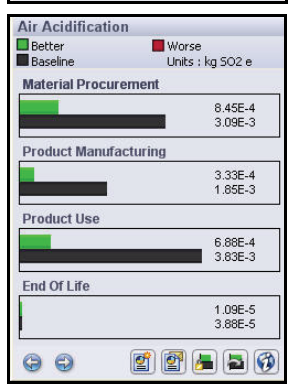
- Changing the material from Stainless Steel (ferritic) to Nylon 6/10 and the manufacturing process from milled to injection molded had a positive environmental impact in all categories, but a further material change may provide a better result.

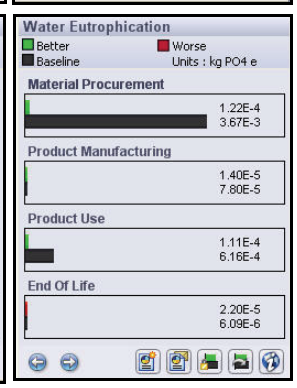


- Click **inside** the Carbon box to display the difference between the baseline and new material.
- Click the **right arrow** to move to the next (Energy Consumption) impact screen to display the difference between the baseline and new material.
- Click the **right arrow** to move to the next (Air Acidification) impact screen to display the difference between the baseline and new material.
- Click the **right arrow** to move to the next (Water Eutrophication) impact screen to display the difference between the baseline and new material.



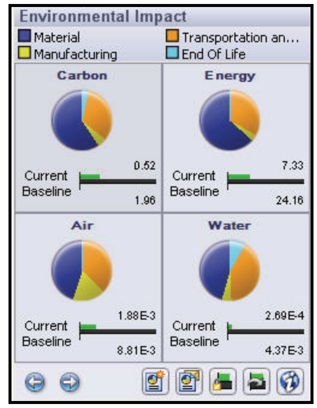






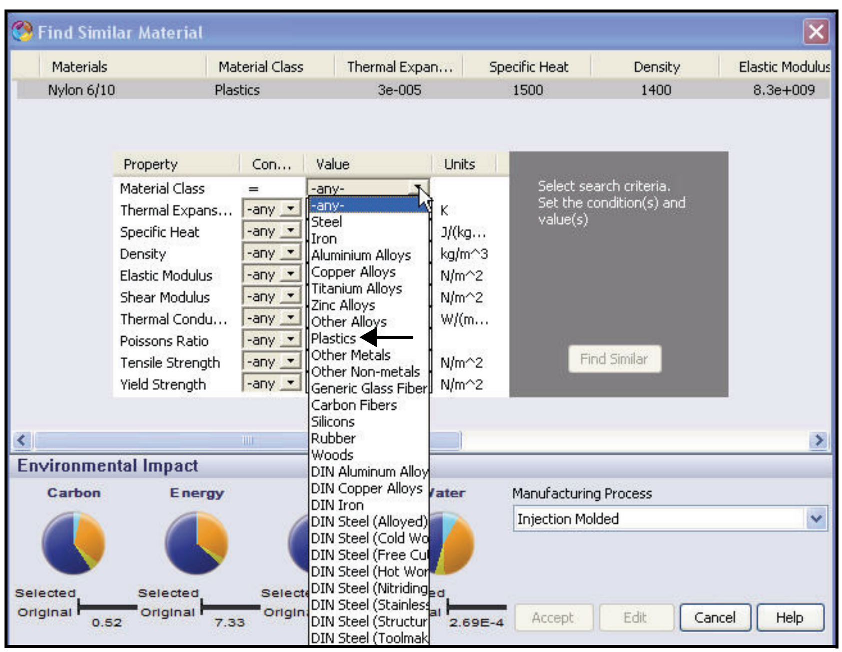
15 Return to the Baseline Screen.

- Click the **right arrow** to move back to the original impact screen. In the next section, find a similar material to Nylon 6/10 to see if you can lower the Environmental Impact of the part.

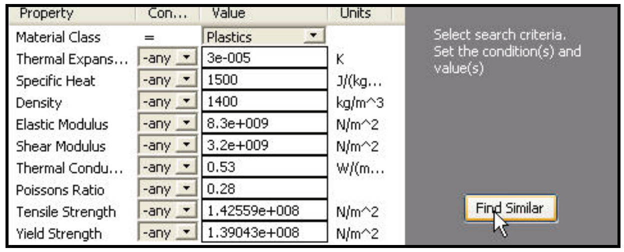


16 Find a similar material and compare the Environmental Impact to Nylon 6/10.

- Click the **Find Similar** button as illustrated. The Find Similar Material dialog box is displayed. View your options.
- Click the **Value (-any-)** drop-down arrow.
- Select **Plastics**. At this time, you can perform a general search or customize your search on physical properties of the material.



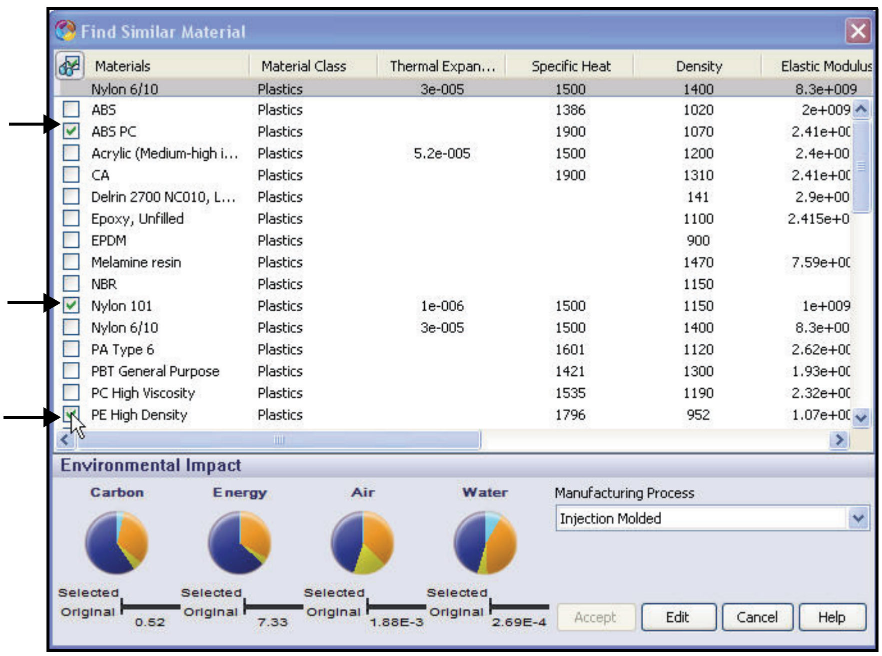
Note: View the available database for materials and material properties.



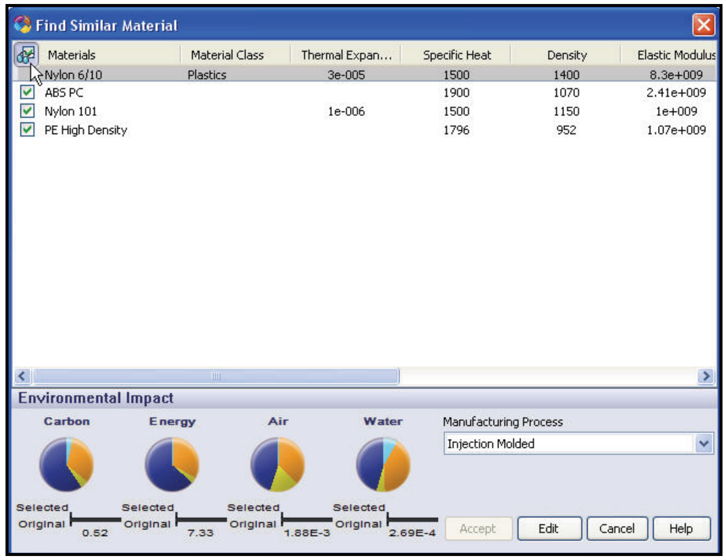
- Click the **Find Similar** button as illustrated. SolidWorks provides a full list of comparable materials that you can further refine.

17 Select a Similar Material from the provided list to compare to Nylon 6/10.

- Check the **ABS PC** material box.
- Check the **Nylon 101** material box.
- Check the **PE High Density** material box.

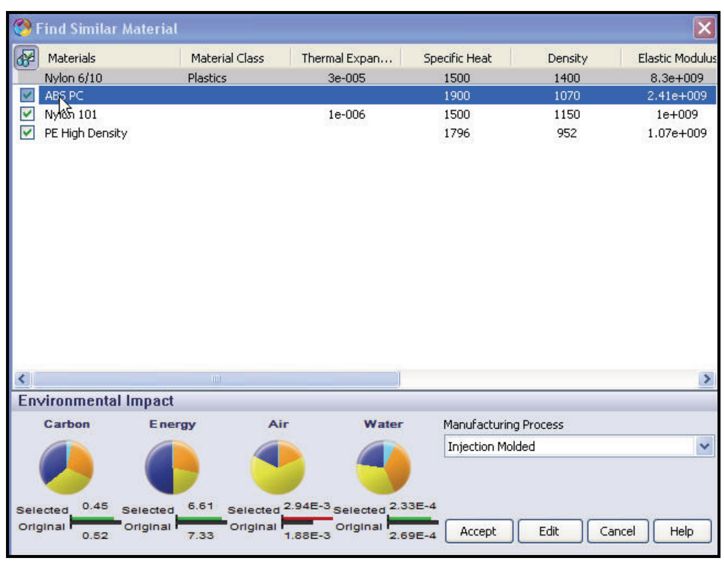


- Click **inside** the top left box (Show Selected Only) as illustrated. The selected materials are displayed.

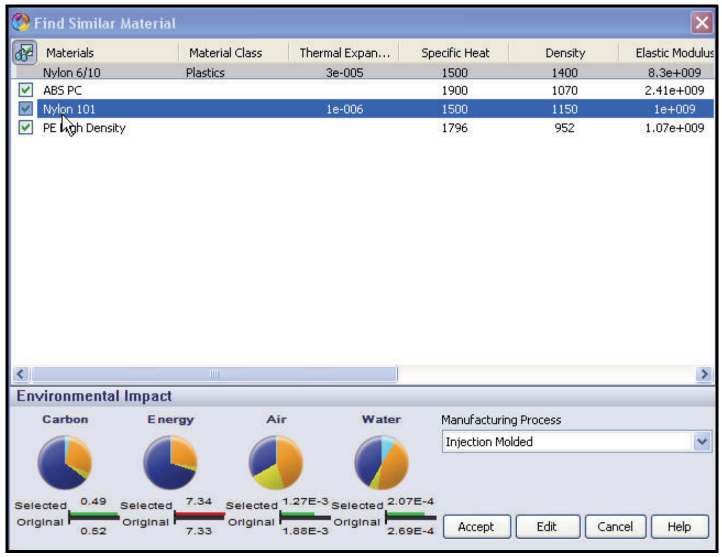


18 View the Environment Impact for the alternative materials.

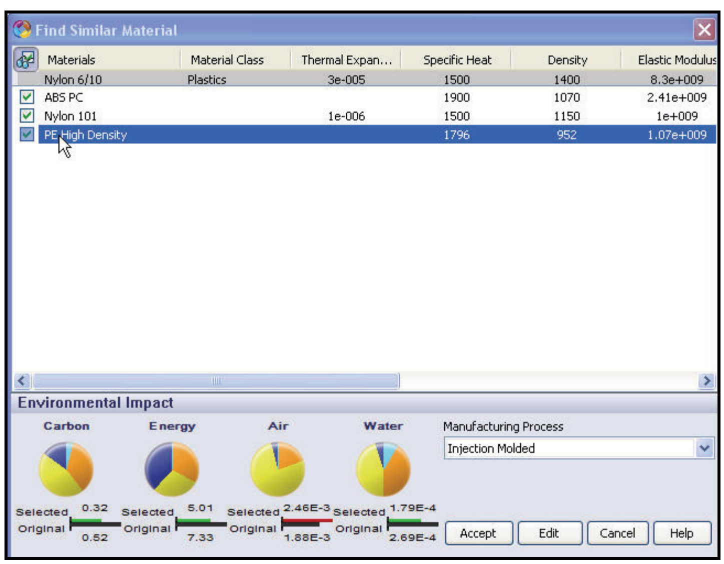
- Click the **ABS PC** material row as illustrated. View the results in the box. The ABS PC material is lower in Carbon Footprint, Energy Consumption, and Water Eutrophication, but higher in Air Acidification.



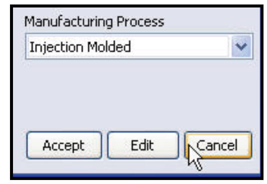
- Click on the **Nylon 101** material row. View the results in the box. The Nylon 101 material is lower in Carbon Footprint, Air Acidification and Water Eutrophication but higher in Energy Consumption.



- Click on the **PE High Density** material row. View the results in the box. The PE High Density material is lower in Carbon Footprint, Energy Consumption and Water Eutrophication, but is higher in Air Acidification. You decide to stay with Nylon 6/10.

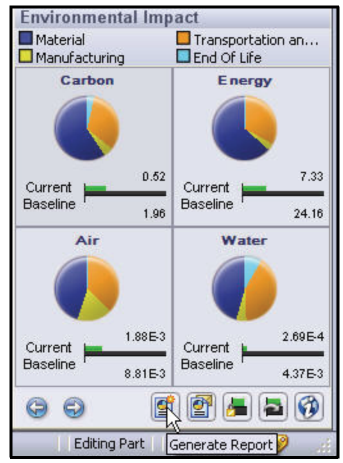


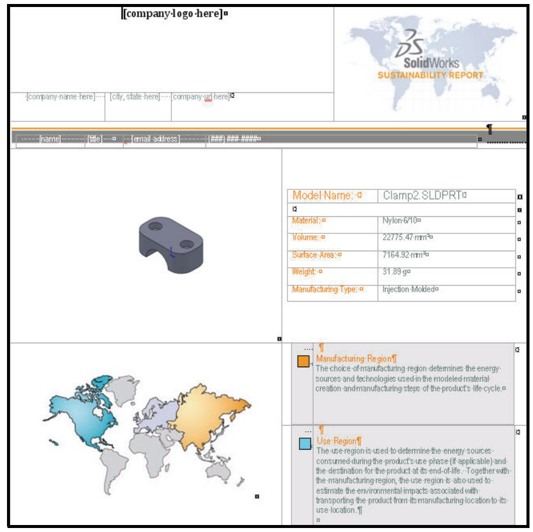
■ Click the **Cancel** button from the Find Similar Material dialog box.



19 Run a Report.

■ Click the **Generate Report** button as illustrated. SolidWorks provides the ability to communicate this report information throughout your organization. SustainabilityXpress generates a report that will compare designs (materials, regions, manufacturing process, etc.) and explain each category (*Carbon Footprint*, *Energy Consumption, Air Acidification* and *Water Eutrophication*) of Environmental Impact and show how each design compares within each category.





20 Close the Report.

- Review the Generated report. **Close** the report.

21 Close the Part.

- Close the **Clamp** part.

Notes

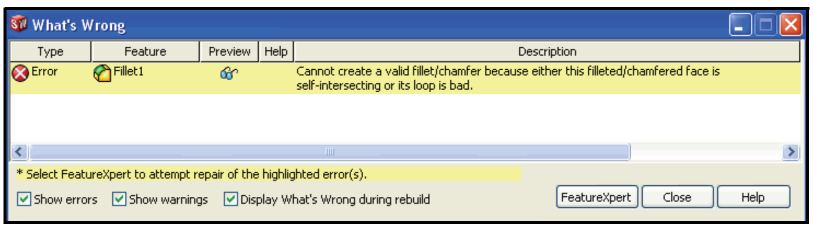
SWIFT™ Technology

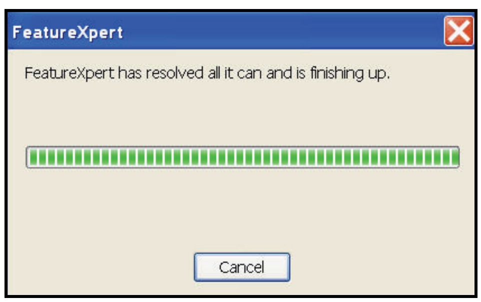
The FeatureXpert is powered by SolidWorks Intelligent Feature Technology (SWIFT™). The FeatureXpert manages fillet and draft features.

The FeatureXpert can change the feature order in the FeatureManager design tree or adjust the tangent properties so a part can successfully rebuild. To a lesser extent, the FeatureXpert can also repair reference planes that have lost their references.

The FeatureXpert supports the following features:

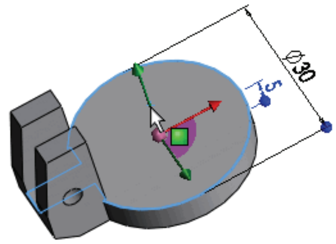
- Constant radius fillets
- Neutral plan drafts
- Reference planes





Utilizing the FeatureXpert

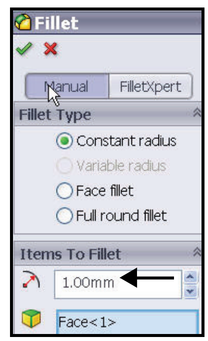
1 Open the Mount2 part.



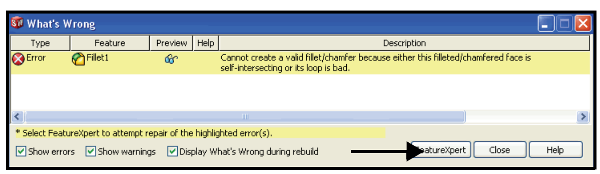
- Click the **Open** tool from the Menu Bar toolbar.
- Double-click **Mount2** from the SolidWorks Test Drive folder. The Mount2 FeatureManager is displayed.

2 Apply a fillet.

- Click the **top circular face** as illustrated. Boss-Extrude1 is highlighted in the FeatureManager.
- Click the **Features** tab in the CommandManager.
- Click the **Fillet** tool from the Features toolbar. The Fillet PropertyManager is displayed.
- Click the **Manual** tab.
- Enter **1**mm for Radius in the Items To Fillet Radius box.

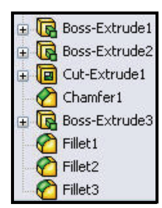


- Click **OK** from the Fillet PropertyManager. The What's Wrong dialog box is displayed. The part failed to create a fillet.



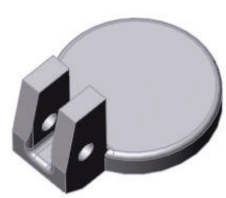
3 Apply the FeatureXpert.

- Click the **FeatureXpert** button from the What's Wrong dialog box. The FeatureXpert dialog box is displayed. The FeatureXpert creates the required fillet features and places them in order in the FeatureManager. View the FeatureManager.



4 Close the Part.

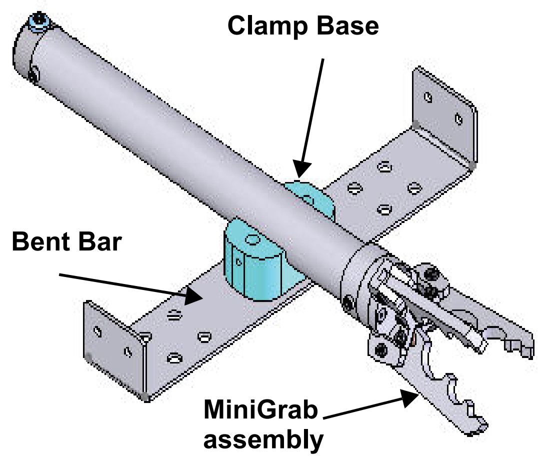
- Click **File**, **Close** from the Menu Bar menu.
- Click **No** to save the part.



Building an Assembly

You have built a component of the SeaBotix LBV150. It is time to put the Clamp component into an assembly.

You will build the Bar assembly in this section. The Bar assembly is a subassembly of the SeaBotix LBV150. In addition to the part you built, you will include a prebuilt Bent Bar part, Clamp Base part, and MiniGrab assembly as illustrated.

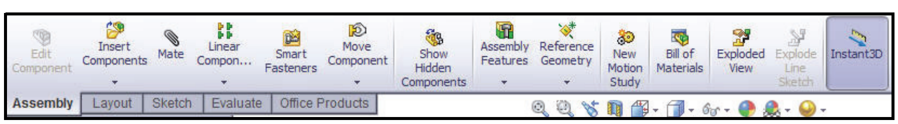


Assembly Basics

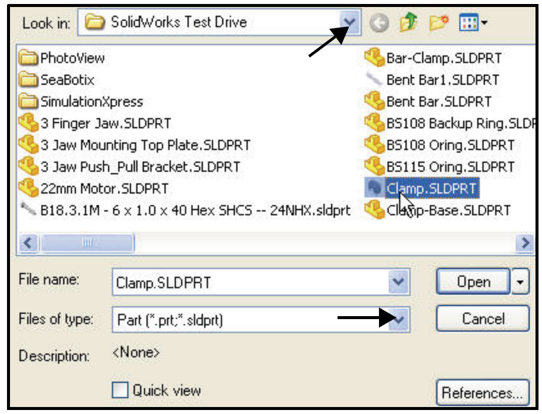
- An assembly contains two or more parts.
- In an assembly, parts are referred to as components.
- Mates are relationships that align and fit components together in an assembly.
- Components and their assemblies are directly related through file linking.
- Changes in the components affect the assembly.
- Changes in the assembly affect the components.

Assembly Tab

The Assembly tab in the CommandManager provides access to the following Assembly tools:

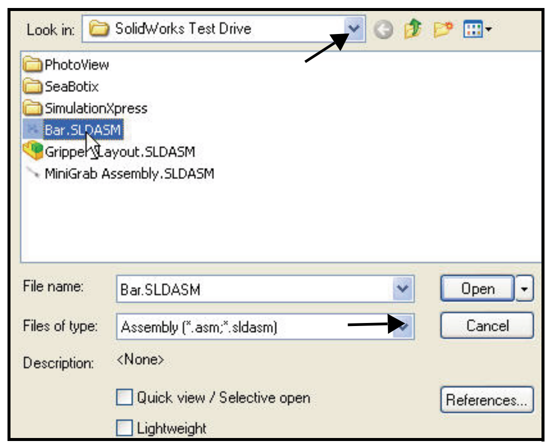


Open the Clamp Part and the Bar Assembly



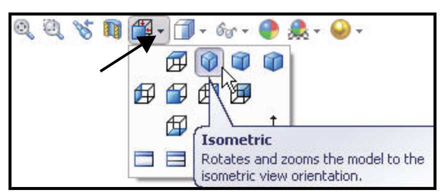
1 **Open the Clamp Part.**

- Click the **Open** tool from the Menu Bar toolbar.
- Browse to the **SolidWorks Test Drive** folder.
- Select **Part** for Files of type.
- Double-click **Clamp**. The Clamp part is displayed in the Graphics area.



2 **Open an Assembly.**

- Click the **Open** tool from the Menu Bar toolbar.
- Browse to the **SolidWorks Test Drive** folder.
- Select **Assembly** for Files of type.
- Double-click **Bar**. The Bar assembly is displayed in the Graphics area.



3 **Change the view orientation.**

- If the assembly is not in an Isometric view, click the **Isometric** view tool from the Heads-up View toolbar or right-click and drag in the Graphics window. Select the Isometric view tool from the default display wheel.

Move Component and Dynamic Collision Detection

There may be times when you have to move a component. You may wish to move components in order to access them more easily, perform dynamic collision detection, create an alternate drawing view, and so forth.

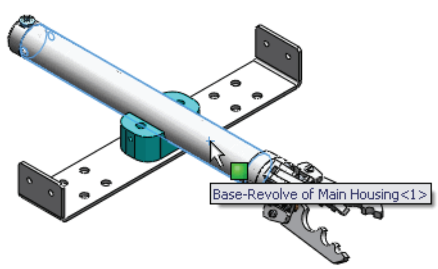
A component can only move as it is allowed to. There may be geometric relations or mate relations that prevent a component from moving in every direction. For example, the MiniGrab in the Bar assembly slides along the Clamp Base.

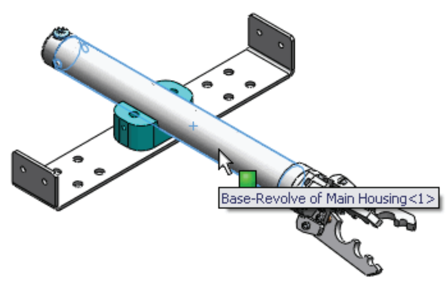
In a moment, you will SmartMate the Clamp component to the Bar assembly. In order to perform this operation, you will use Move Component to position the MiniGrab assembly.

One of the powerful benefits of working with precise solid models is that the system can detect interference conditions between components in an assembly. Combining interference detection with dynamic assembly motion provides dynamic collision detection, another SolidWorks first.

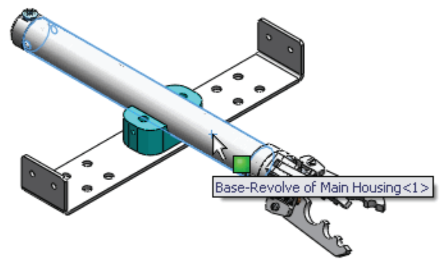
The component will only move within the available degrees of freedom. This simulates realistic behavior. To minimize the workload on your machine, select only the items that you are concerned about and test those.

Note: The first feature that is placed in an assembly is fixed and will not move.





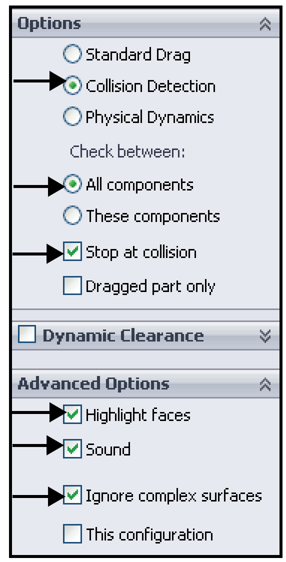
Moving a Component and using Dynamic Collision Detection



1 Move the component.

- Click and drag the **Main Housing** until it is positioned in the middle of the Clamp Base as illustrated.

2 Collision detection.

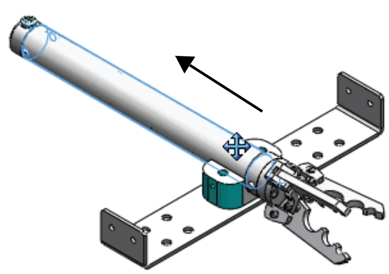


- Click the **Move Component** tool from the Assembly toolbar. The Move Component PropertyManager is displayed.
- Check **Collision Detection**, **All components**, and **Stop at collision** in the Options box.
- Check **Highlight faces**, **Sound**, and **Ignore complex surfaces** in the Advanced Options box.

Note: The expression, “click and drag” refers to the following sequence of actions: Position the cursor on an entity. Click and hold the left mouse button down. Drag the cursor to where you want the entity to go. Release the left mouse button.

3 Move the Main Housing assembly.

- Click and drag the **Main Housing** towards the Clamp-Base. The Housing stops moving when it interferes with the Clamp-Base.



4 Position the Main Housing assembly.

- Click and drag the **Main Housing** to the original location.

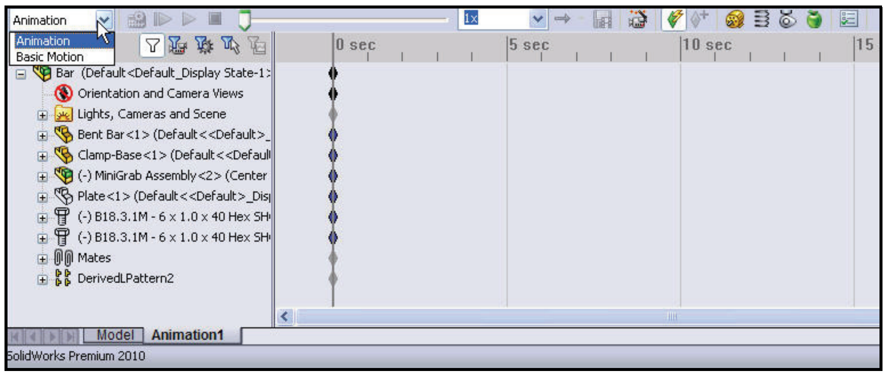
5 Complete the move.

- Click **OK** from the Move Component PropertyManager.

Creating a Motion Study

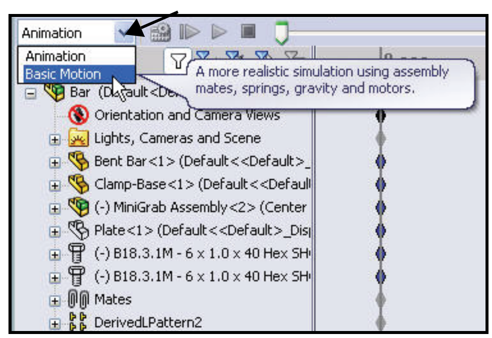
Motions Studies are graphical simulations of motion for an assembly. Access MotionManager from the Motion Study tab. The Motion Study tab is located in the bottom left corner of the Graphics area.

Incorporate visual properties such as lighting and camera perspective. Click the Motion Study tab to view the MotionManager. Click the Model tab to return to the FeatureManager design tree.



Note: Older assemblies created before 2008, the **Animation1** tab is displayed.

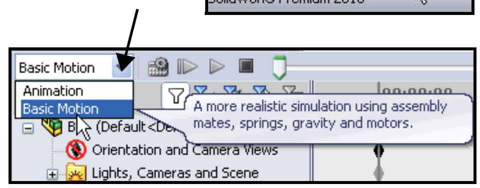
The MotionManager display a timeline-based interface, and provides the following selections from the drop-down menu as illustrated:



- **Animation**: Apply Animation to animate the motion of an assembly. Use the Animation option to create animations for motion that do NOT require accounting for mass or gravity.
- **Basic Motion**: Apply Basic Motion for approximating the effects of motors, springs, collisions, and gravity on assemblies. Basic Motion takes mass into account in calculating motion.

Create a Motion Study

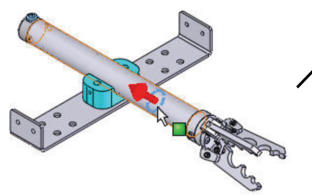
1 **Invoke Motion Study.**
- Click the **Animation1** tab in the lower left corner of the Graphics area.
- Select **Basic Motion** from the drop-down box.

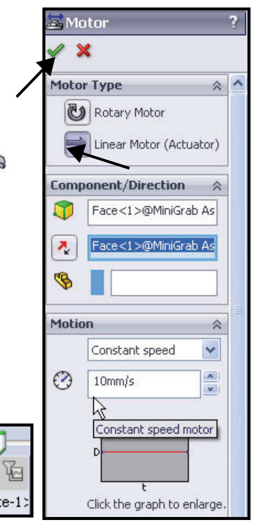


- Click the **Motor** tool from the MotionManager. The Motor PropertyManager is displayed.

- Click **Linear Motor** from the Motor Type box.

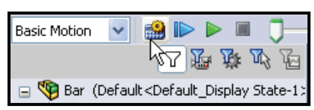
2 **Select the Main Housing and set the speed.**
- Click the **cylindrical face** of the Main Housing of the Bar assembly in the Graphics area. Face<1>@MiniGrab is displayed in the Component/ Direction box. The direction arrow points to the left.
- Enter **10**mm/s in the Motion box.





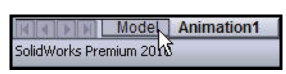
- Click **OK** from the Motor PropertyManager.

3 **Calculate the study.**



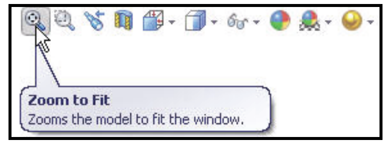
- Click the **Calculate** tool from the MotionManager. The Main Housing moves in the direction of the arrow in the Graphics area.
- Click the **Play from Start** tool from the MotionManager. View the model in the Graphics area.

4 **Return to the main SolidWorks Graphics area.**
- Click the **Model** tab at the bottom of the Graphics area.



Note: Click the **Save** tool in the MotionManager to save the Basic Motion study to an avi file.

Zooming In

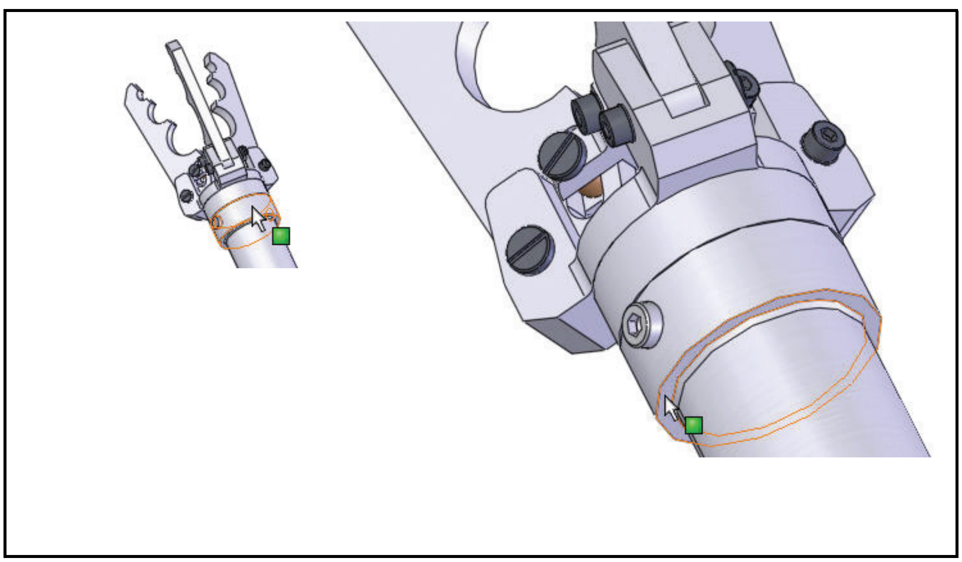


Various modeling operations require you to view and select details of a model, no matter how small they may be. SolidWorks has many different view manipulation tools that allow you to perform this function. **Zoom to Fit** and **Zoom to Area**, are a few examples of these tools.

In a moment, you will use Standard Mates and SmartMates to position the MiniGrab assembly and to attach the Clamp component.

In order to perform the Mates correctly, you have to be able to view the correct edges and faces of the model. Before you create the Mates, you will apply the **Zoom to Area** tool to focus on the details of the models that you need to view.

The following illustrations display the Bar assembly before and after the **Zoom to Area** tool is applied to focus on the details of the model.

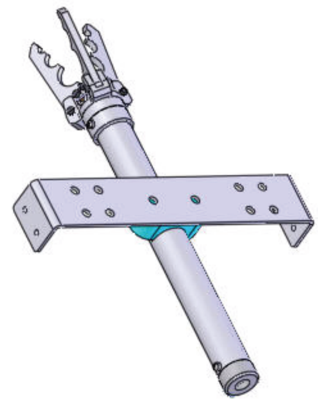


Note: The Magnifying glass tool (g) key provides the ability to inspect a model and make selections without changing the overall view.

Zoom to Detail Areas

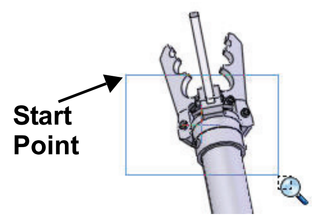
1 Orient the view.

- Rotate the view. Click the **middle mouse button** and slowly move diagonally to the upper left corner.
- Release the **mouse button**.
- **Position** the bar assembly as illustrated.



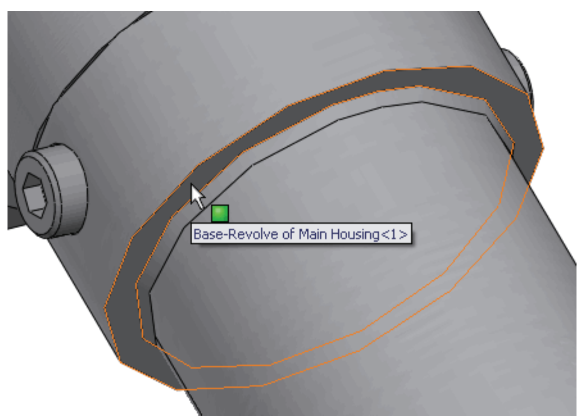
2 Zoom in on the Main Housing.

- Click the **Zoom to Area** tool from the Heads-up View toolbar. Feedback is presented by a symbol attached to the cursor.
- **Box-select** the Main Housing as illustrated.
- Click the **Zoom to Area** tool from the Heads-up View toolbar to deactivate.



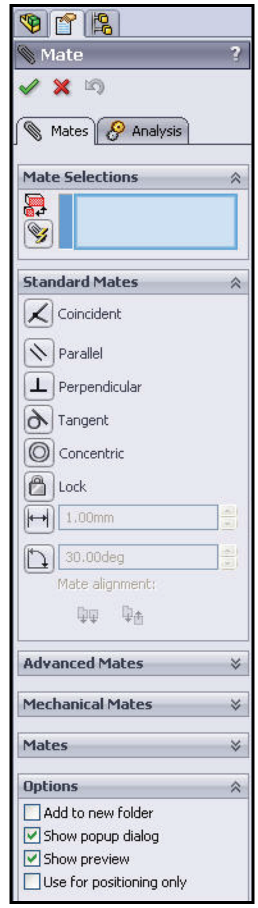
3 Select the face.

- Click the **back circular planar face** of the Main Housing. The selected face is displayed in blue. Note the face icon display on your mouse pointer. This face is required to create a Distance Mate in the next step.



Standard Mates

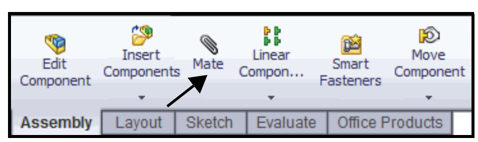
Mates create geometric relationships between assembly components. As you add mates, you define the allowable directions of linear or rotational motion of the components. You can move a component within its degrees of freedom, visualizing the assembly's behavior.



Mate PropertyManager

The components in the Bar assembly utilize Standard Mates and SmartMates. The Mate PropertyManager displays Standard Mates, Advanced Mates, and Mechanical Mates. The components will be assembled with Coincident, Concentric, and Distance mates.

- A Coincident mate forces two planar faces to become coplanar. The faces can move along one another, but cannot be pulled apart.
- A Concentric mate forces two cylindrical faces to become concentric. The faces can move along the common axis, but cannot be moved away from this axis.
- A Distance mate places the selected entities with a specified distance between them. A distance dimension is required.



Inserting a Distance Mate

1 Insert a Distance mate.

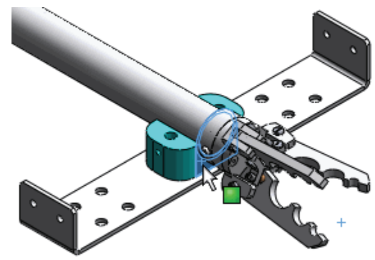
- Click the **Mate** tool from the Assembly toolbar. The back face of the Main Housing is displayed in the Mate Selections box.

2 Change the view orientation.

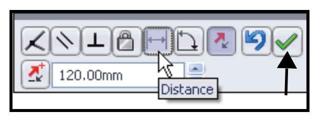
- Click the **Isometric** view tool from the Heads-up View toolbar.

3 Select the mating component.

- Click the **front planar face** of the Clamp-Base as illustrated. SolidWorks automatically selects a Coincident mate. Face<2> is displayed in the Mate Selections box. The part requires a Distance mate.



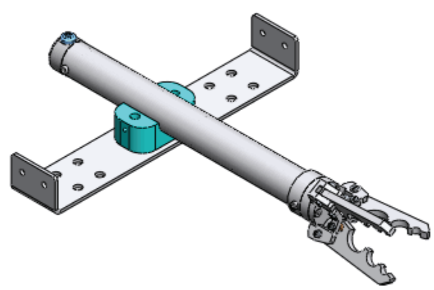
4 Set the Distance Mate value.



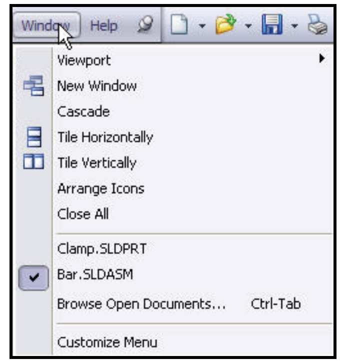
- Click the **Distance** mate tool from the Mate dialog box.
- Enter **120**mm for Distance.
- Click the **green checkmark** button from the pop-up mate dialog box.

5 View the results.

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



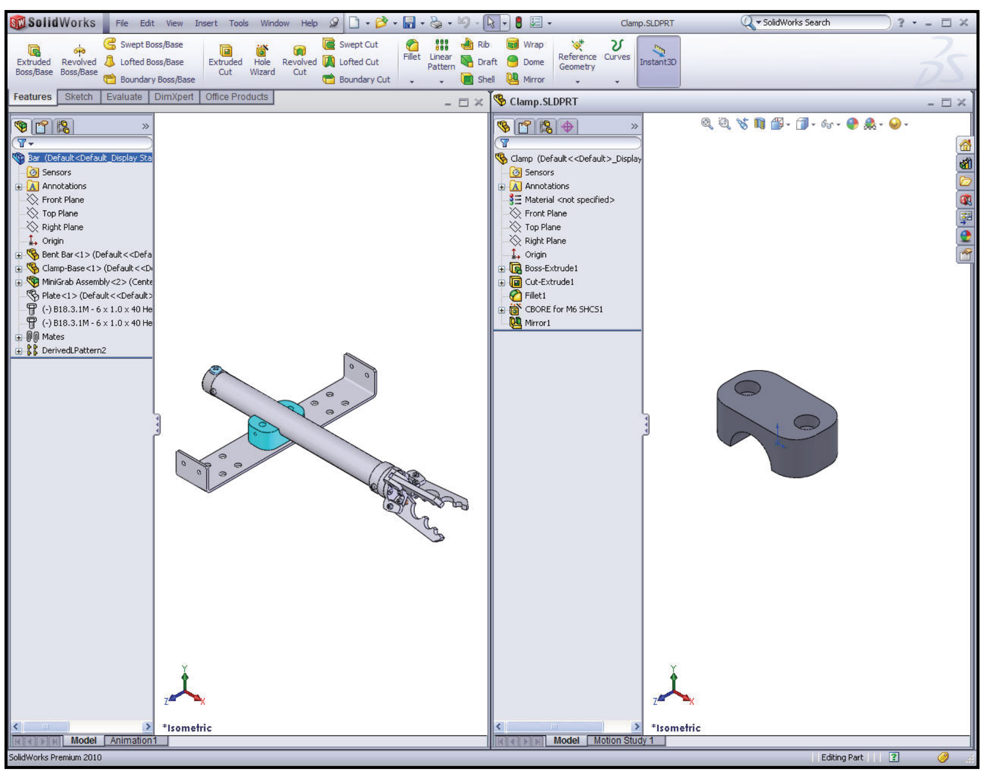
Customizing SolidWorks



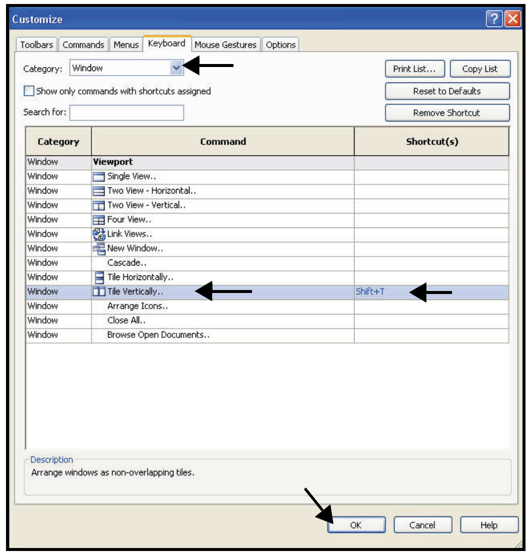
SolidWorks is a native Windows application, so it can take advantage of standard Windows benefits such as keyboard shortcuts. The interface for managing keyboard shortcuts allows you to redefine, (add, delete, or change) shortcuts for all commands. You can assign multiple shortcuts to commands.

Throughout the rest of this book there will be several times when you will need to tile the windows. To speed up this process, let's take a few seconds to create a customized keyboard shortcut for the command.

You have two SolidWorks documents open, the Clamp part and the Bar assembly. There are times when you want to view more than one window at a time. That means you have to resize the windows to fit the screen. The Tile feature does this quickly.



Creating a Keyboard Shortcut



1 Create a keyboard shortcut.

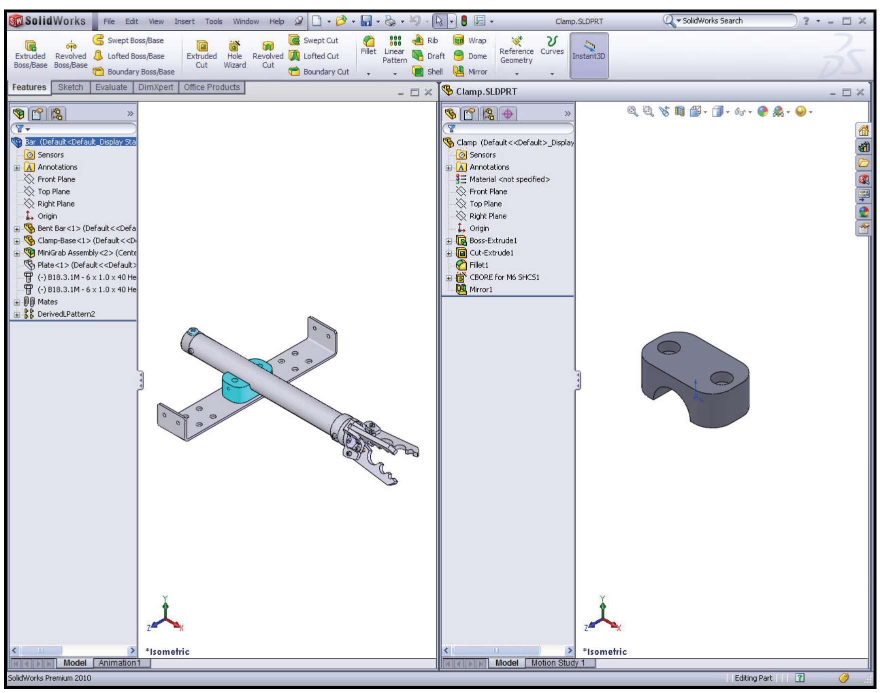
- Click **Tools**, **Customize...** from the Menu Bar menu. The Customize dialog box is displayed.
- Click the **Keyboard** tab.
- Select **Window** from the Category drop-down menu.
- Click **Tile Vertically** from the Command column.

2 Assign the keyboard shortcut.

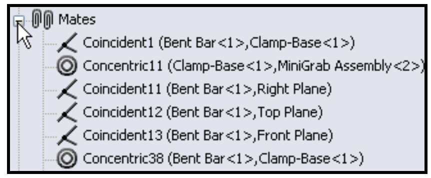
- Click in the **column area** under Shortcut(s).
- Press **<Shift + t>**.
- Click **OK** to close the Customize dialog box.

3 Vertical Tile the windows.

- Using the newly defined shortcut, press **<Shift + t>** to resize the windows so you can view both the Clamp part and the Bar assembly.



Creating a SmartMate



You can create some types of mating relationships automatically. Mates created with these methods are referred to as SmartMates.

To position the components in an assembly, use Mating Relationships. Mating Relationships are similar to geometric relations such as Midpoint, that you used in the sketch of the rectangle. The  folder stores all the mating relationships that position the components with respect to each other.

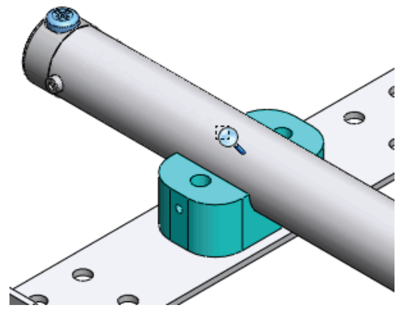
You can create SmartMates in various ways:

1. You can create mates when you insert a part into an assembly, by dragging the part in specific ways from an open part window. The entity that you use to drag determines the types of mates that are added.
2. You can specify a mate reference in the part document. A mate reference identifies the entity to use when you insert the part from a file.
3. You can create mates by inferring potential mate partners when you move a part within the assembly.

SolidWorks SmartMates will automatically insert mates between the Clamp and the MiniGrab assembly and the Clamp and the Clamp-Base. It will add:

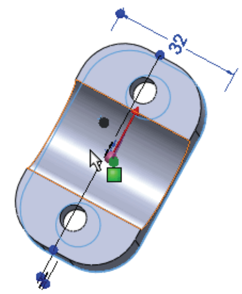
- *Concentric mate* — between the cylindrical cut of the Clamp and the cylindrical face of the Main Housing.
- *Concentric mate* — between the Cbore cylindrical face of the Clamp and the Cbore cylindrical face of the Clamp-Base.
- *Coincident mate* — between the bottom left face of the Clamp and the top left face of the Clamp-Base.

Inserting the Clamp Component



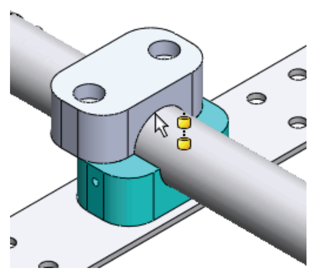
1 Zoom in.

- Click inside the **Bar** Graphics area.
- Click the **Zoom to Area** tool from the Heads-up View toolbar.
- **Zoom in** on the top face of the Clamp-Base and the Main Housing.
- Click the **Zoom to Area** tool to deactivate.



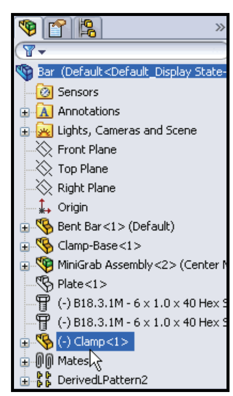
2 Rotate the Clamp and zoom.

- Click the **Clamp** in the Graphics area.
- Rotate the **Clamp** view to display the bottom cylindrical face.
- If needed, click the **Zoom to Area** tool from the Heads-up View toolbar.
- **Zoom in** on the bottom cut of the Clamp.
- Click the **Zoom to Area** tool to deactivate.



3 Insert the first SmartMate.

- Click and drag the **Clamp** by the bottom face into the Bar assembly Graphics area.
- When you see the Concentric feedback icon, release the **Clamp** on the cylindrical face of the Main Housing.
- Click the **green checkmark** button.



4 View the Bar FeatureManager.

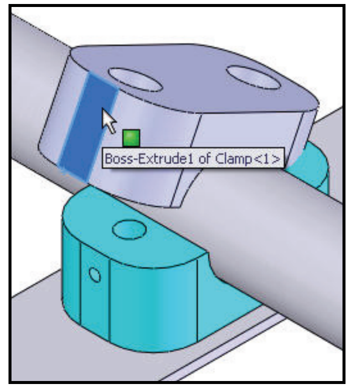
The Clamp component is displayed in the Bar assembly FeatureManager.

5 Close the Clamp Component window.

- **Close** the window's border.
- If required, click **No** to save changes.

6 Maximize the Bar assembly window.

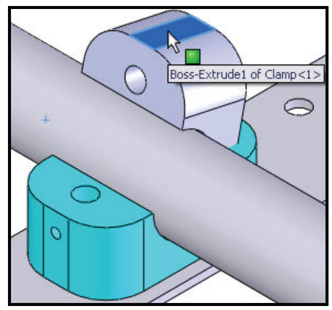
- Click the **Maximize** button in the window's border.



7 Rotate the Clamp component.

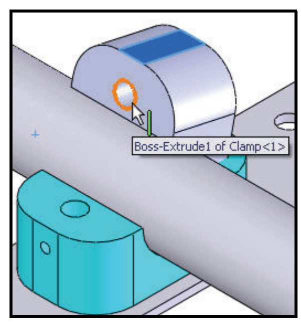
- Press the **z** key on the keyboard two times to zoom out, to view the Clamp on the Main Housing.
- Click and drag the **left planar face** of the Clamp. The Clamp is free to move and rotate along the Main Housing.

- Rotate the **Clamp** to view the bottom left hole.



8 Select the hole.

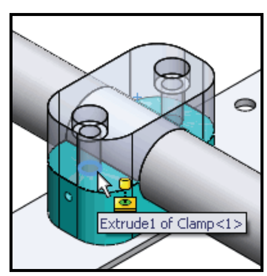
- Click the **bottom circle edge** of the hole as illustrated.
- Hold the **Alt** key down.



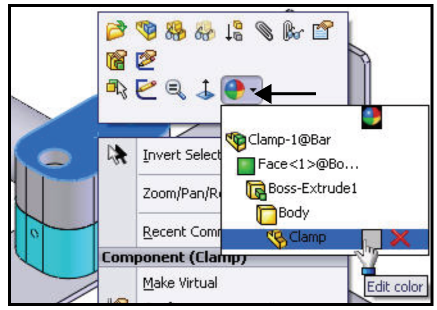
Note: You need to pick the edge of the circle, not the face. Look for the icon feedback.

9 Insert the second SmartMate.

- Drag and drop the **circular edge of the hole** to the top left circular edge of the Clamp-Base. The mouse pointer displays the Concentric/ Coincident feedback icon.
- Release the **Alt** key.
- Release the **mouse** button. The Clamp component is fully mated to the Clamp-Base component. The Clamp component is fixed to the Clamp-Base.



Note: Selecting two circular edges for the second SmartMate creates a Concentric and Coincident mate in a single step.



10 Display an Isometric view.

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

11 Modify the Clamp appearance.

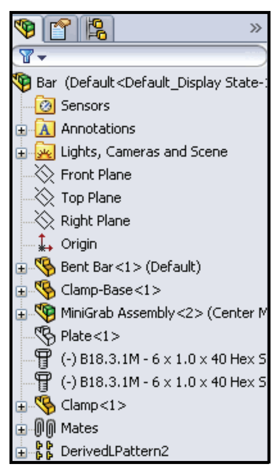
- Right-click the **top face** of the Clamp in the Graphics area.
- Click the **Appearances** tool drop-down arrow.
- Click the **Clamp** box as illustrated. The Color PropertyManager is displayed.
- Click a **yellow color swatch** from the Color box.

12 View the results.

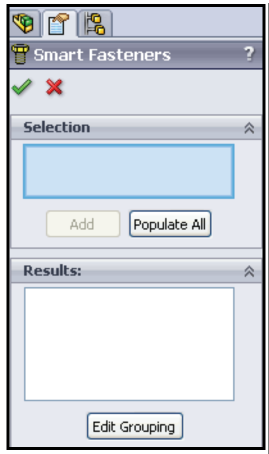
- Click **OK** from the Color PropertyManager. The color is applied to the Clamp.

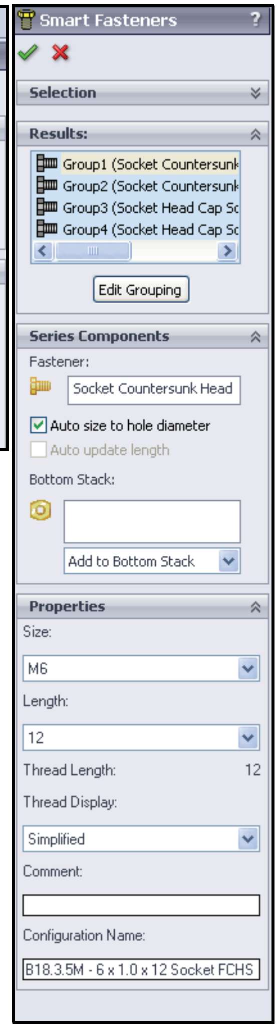
13 Save the Bar assembly.

- Click **Save** from the Menu Bar toolbar.



Using Smart Fasteners





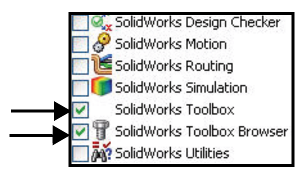
The SeaBotix LBV150 contains hundreds of fasteners. Smart Fasteners adds bolts and screws to selected holes in the assembly. Smart Fasteners uses the SolidWorks Toolbox library of standard hardware.

If you select a component, Smart Fasteners finds the available holes in the component. If you select a face, Smart Fasteners finds the available holes that pass through the surface.

You can also add associated standard nuts and washers to a series of Smart Fasteners.

The Bar assembly contains two hidden Plate components. The Plates require Smart Fasteners.

SolidWorks Toolbox uses SolidWorks Smart Part Technology to automatically select the appropriate fasteners and assemble them in proper sequence. With SolidWorks Toolbox and Smart Fasteners you gain access to an affordable, time saving library of standard parts, a virtual Machinery's Handbook.

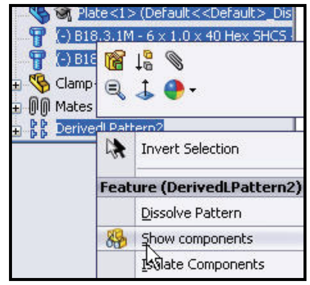


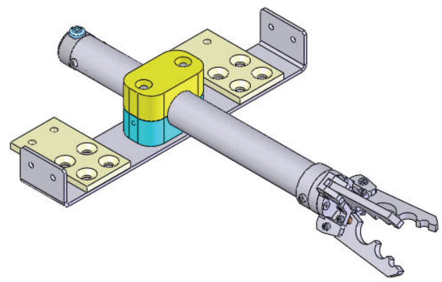
Note: Activate the **SolidWorks Toolbox** and **SolidWorks Toolbox Browse** from the Add-Ins dialog box for this next section.

Using Smart Fasteners

1 Select hidden components.

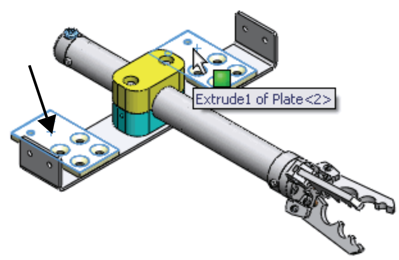
- Click **Plate<1>** from the FeatureManager.
- Hold the **Ctrl** key down.
- Click **(-)B18.3.1M-6 x 1.0 x 40** from the FeatureManager.
- Click the second **(-)B18.3.1M-6 x 1.0 x 40** from the FeatureManager.
- Click **DerivedLPattern2** from the FeatureManager.
- Release the **Ctrl** key.
- **Right-click** in the FeatureManager.
- Click the **Show components** tool. View the two Plate components and center Hex SHCS components. The Plate components contain two different size holes.

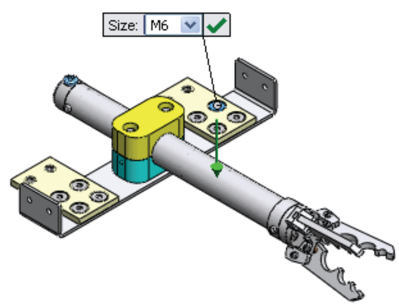




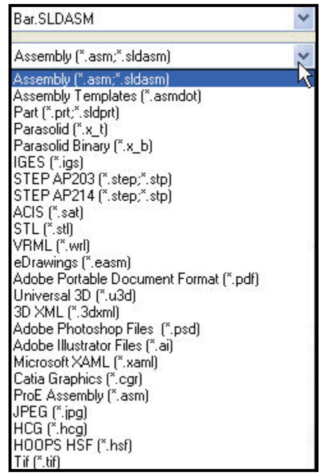
2 Insert Smart Fasteners.

- Click **inside** the Graphics area to deselect the components.
- Click the **Smart Fasteners** tool from the Assembly toolbar.
- Click **OK** to continue. The Smart Fasteners PropertyManager is displayed.
- Click the **top face** of the front Plate.
- Click the **top face** of the back Plate. The selected entities are displayed in the Selection box.
- Click the **Add** button from the Selection box. SolidWorks selects the required fasteners and mates them to the Plate components.
- Click **OK** from the Smart Fasteners PropertyManager. View the results.





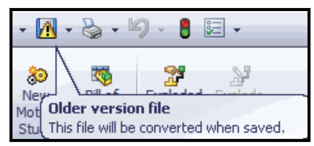
Saving an Assembly

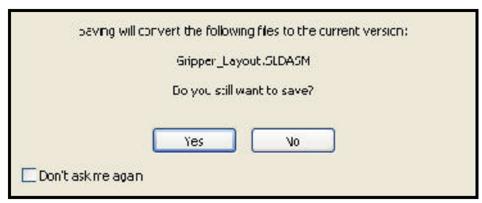


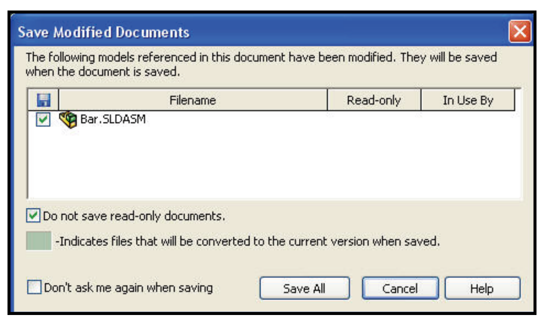
Assemblies are made up of multiple parts. Because SolidWorks is associative, changes to the component parts are reflected in the assembly and changes to the assembly are reflected in the component parts. So, when you save the assembly, you will probably get one or more messages telling you that:

- The assembly hasn't been rebuilt recently.
- Saving the assembly will save any referenced files.

These messages are just reminders from SolidWorks which ensure that your files stay up-to-date and in sync.

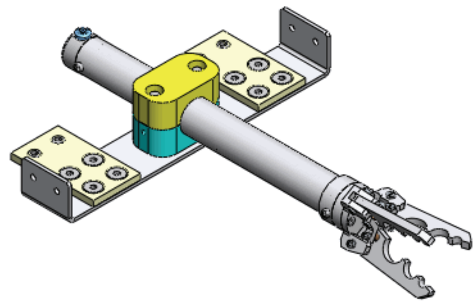






Saving the Assembly

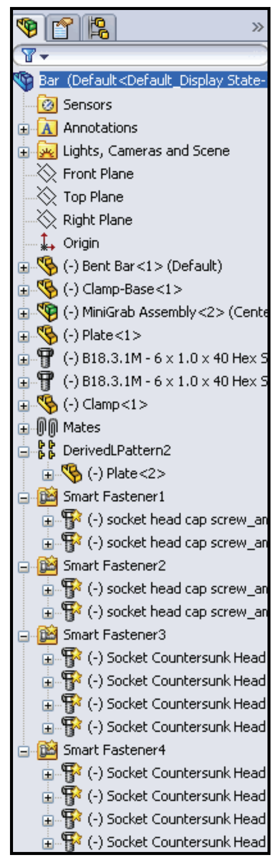
1 Save your work.



- Click **Save** from the Menu Bar toolbar. The extension *.sldasm is added automatically to an assembly name.
- If you get a message telling you that the assembly has not been rebuilt or that files need to be saved, click **Yes**.

2 View the FeatureManager.

- **View** the Bar FeatureManager. SmartFastener1, SmartFastener2, SmartFasterner3, and SmartFastener4 are added to the FeatureManager.



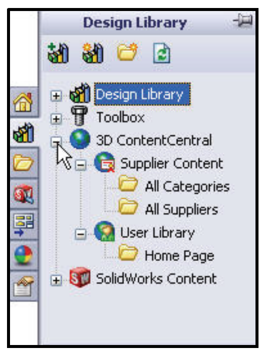
Note: SolidWorks provides the ability to create a Bill of Materials for an assembly without creating a drawing.

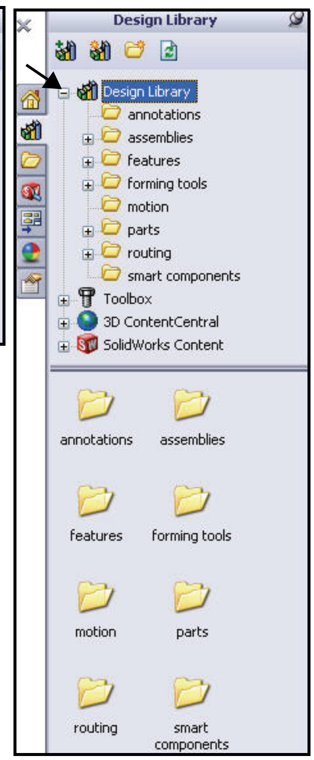
SolidWorks Search Tools

Microsoft Windows Search is installed with SolidWorks and indexes the resources once before searching begins, either after installation, or when you initiate the first search. Subsequent indexing of added files is fast and transparent.

The SolidWorks Search box 

is displayed in the upper right corner of the SolidWorks Graphics window. Enter the text or key words to search. Click the drop-down arrow to view the last 10 searches.





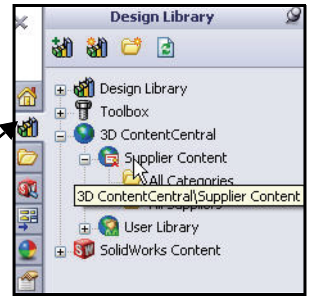
3D ContentCentral® provides access to 3D models from component suppliers and individuals in all major CAD formats.

Access 3D ContentCentral using the Task Pane and selecting the Design Library tab. 3D ContentCentral provides two options:

- *Supplier Content* - Links to supplier Web sites with certified 3D models.
- *User Library* - Links to models from individuals using 3D PartStream.Net®.

Note: In the next section, you require internet access and an account.

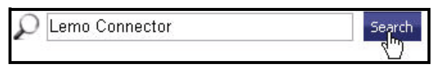
Searching for a Component using 3D ContentCentral



1 Locate a Lemo Connector.

- Click the **Design Library** tool from the Task Pane.
- **Expand** 3D ContentCentral.
- Click **Supplier Content**.

- Double-click the **All Categories** folder.
- Click the **Click here for all categories** hyperlink.
- Enter **Lemo Connector** in the Search box.
- Click **Search**.

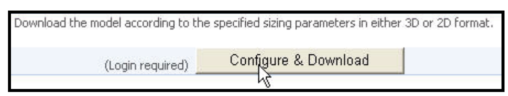


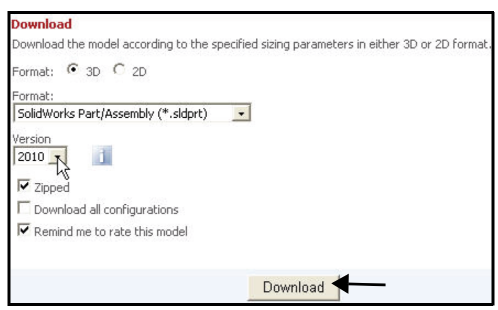
2 Review the Lemo Connectors.

- Click the **Lemo Connector (HEN_OF_305)** icon. View the part.

3 Download the model.

- Click the **Configure & Download** button.
- **Log into** the site.
- Select **SolidWorks Part/ Assembly** for download format.
- Select Version **2010**.
- Click the **Download** button.
- Click **Window, Tile Horizontally** from the Menu Bar menu.
- Click and drag the **Lemo Connector (HEN_OF_305)** icon into the Bar assembly Graphics area. The Browse For Folder dialog box is displayed.

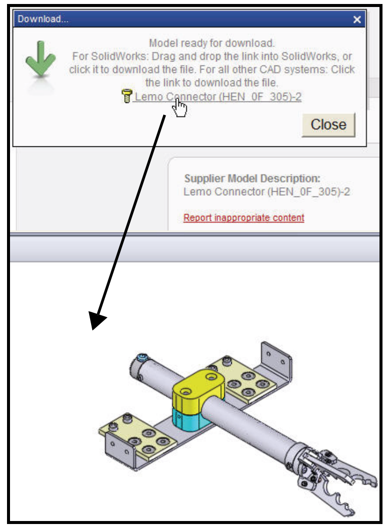




- Select the **SolidWorks Test Drive** Folder.
- Click **OK** from the Browse For Folder dialog box. Accept the model default name. The FeatureManager is displayed.
- Click **Window**, **Tile Horizontally** from the Menu Bar menu. Three windows are displayed.

4 Close the 3D ContentCentral window.

- **Close** the window's border.



5 Insert the Lemo Connector into the Bar assembly Graphics window.

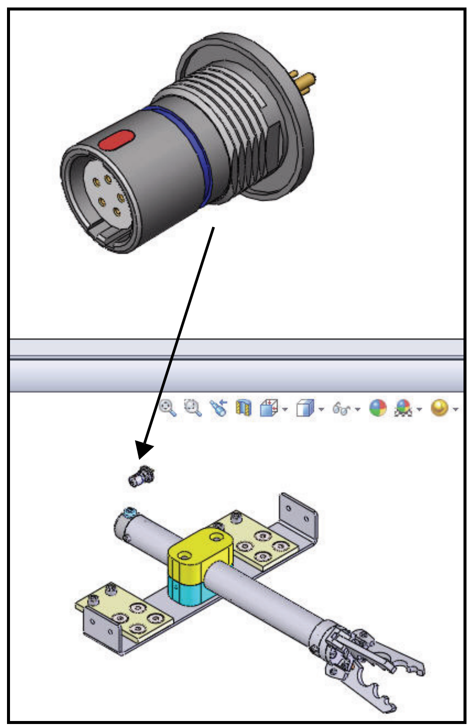
- Click and drag the **Lemo connector** from the Graphics area into the Bar assembly Graphics window as illustrated.

6 Close the Lemo Connector FeatureManager.

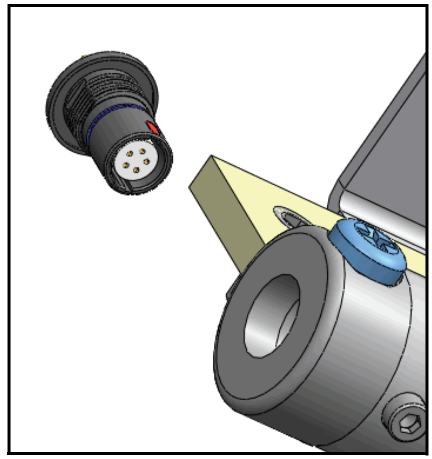
- Click **File**, **Close** from the Menu Bar menu.

7 Maximize the Bar assembly window.

- **Close** the Maximize button in the window's border.

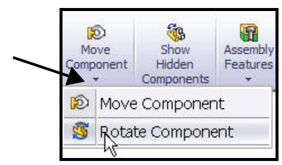


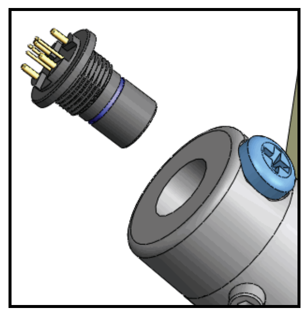
8 Zoom in on the MiniGrab and Lemo Connector.



- Click the **Zoom to Area** tool from the Heads-up View toolbar.
- **Zoom in** on the Lemo Connector and the back of the MiniGrab assembly as illustrated.
- Click the **Zoom to Area** tool to deactivate.

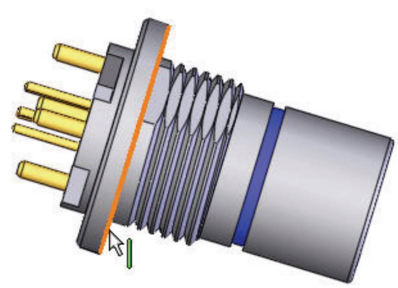
9 Rotate the Lemo connector.





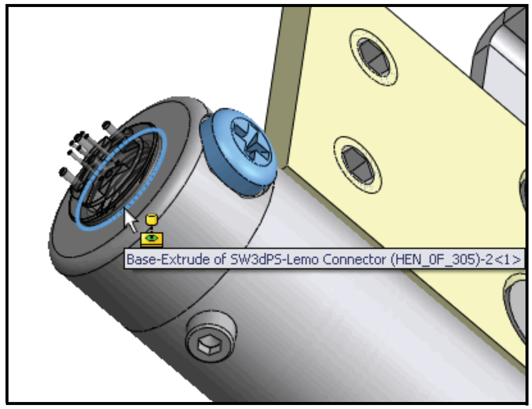
- Click the **Lemo connector** in the Graphics area.
- Click the **Rotate Component** tool from the Assembly toolbar. The Rotate Component PropertyManager is displayed.
- Rotate the **connector** parallel to the Bar.
- Click and drag the **Lemo Connector** near the back of the MiniGrab assembly as illustrated.
- Click **OK** from the Rotate Component PropertyManager.

10 Select the front circular edge of the Lemo connector.



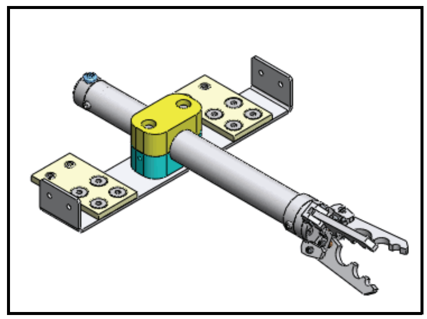
- Click the **Zoom to Area** tool from the Heads-up View toolbar.
- **Zoom in** on the front circular edge of the Lemo Connector.
- Click the **Zoom to Area** tool to deactivate.
- Click the **front circular edge** of the Lemo Connector. Feedback is presented by a symbol attached to the cursor indicating that you are selecting an edge.

11 Insert the SmartMates.



- Hold the **Alt** key down.
- Drag and drop the **circular edge** of the Connector onto the back middle circular edge of the MiniGrab. The Concentric/Coincident feedback  icon is displayed.
- Release the **Alt** key.
- Release the **mouse** button.

Note: It is critical to obtain the correct Concentric/Coincident feedback icon display.

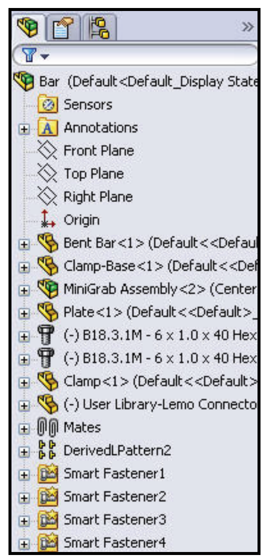


12 Display an Isometric view.

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

13 Save the Bar assembly.

- Click **Save**  from the Menu Bar toolbar. View the results.



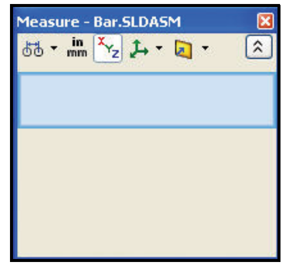
Notes

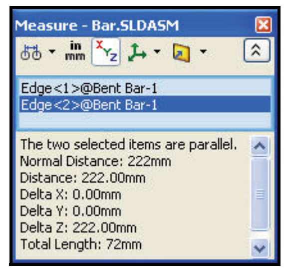
Measure Tool

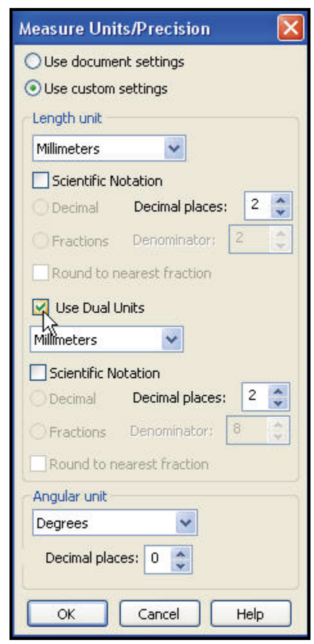
The Measures tool provides the ability to measure distance, angle, radius, and size of and between lines, points, surfaces, and planes in sketches, 3D models, assemblies, or drawings. When you select a vertex or sketch point, the x, y, and z coordinates are displayed.

When the Measure tool is not active, commonly-used measurements for selected entities appear in the status bar.

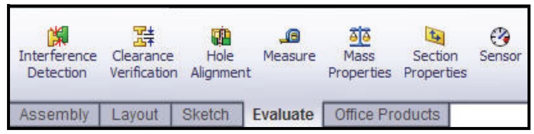
The Measure tool provides the ability to display dual dimension units.





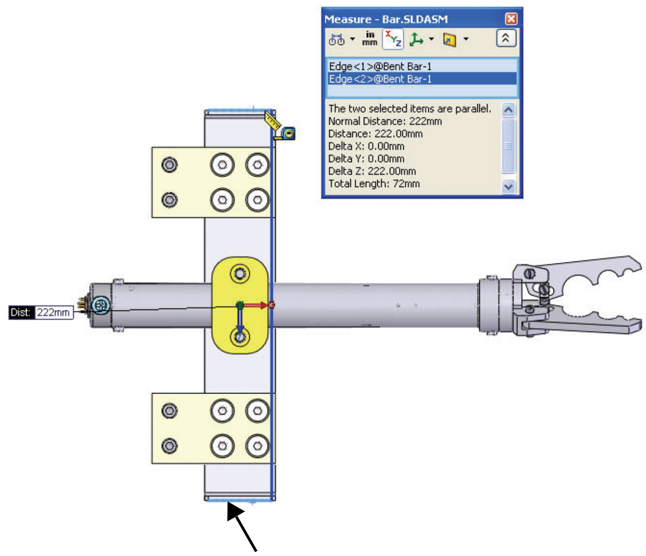


Applying the Measure tool



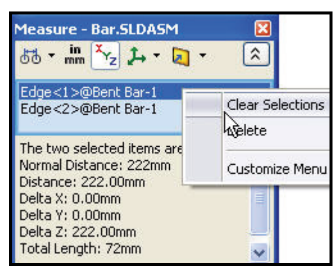
1 **Display a Top view.**
- Click the **Top** view  tool from the Heads-up View toolbar.

2 **Apply the Measure tool.**
- Click the **Measure**  tool from the Evaluate tab. The Measure - Bar dialog box is displayed.

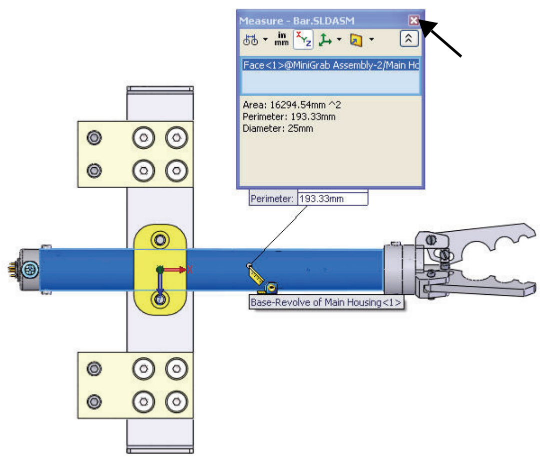


3 **Measure the overall width of the Bar Assembly.**
- Click the top edge of the **Bent Bar** as illustrated.
- Click the bottom edge of the **Bent Bar** as illustrated. The Measure - Bar dialog box displays the results.

4 **Measure the diameter of the Bar assembly.**
- Right-click inside the **Selection** dialog box as illustrated.
- Click **Clear Selections**.



- Click the **cylindrical face** of the Bar assembly. View the Area, Diameter, and Perimeter results.

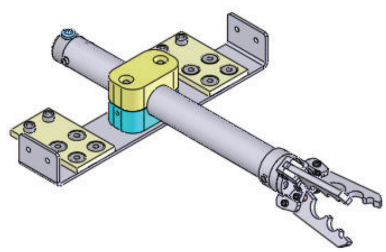


5 Close the Measure dialog box.

- **Close** the Measure - Bar dialog box.

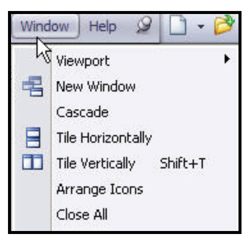
6 Display an Isometric view.

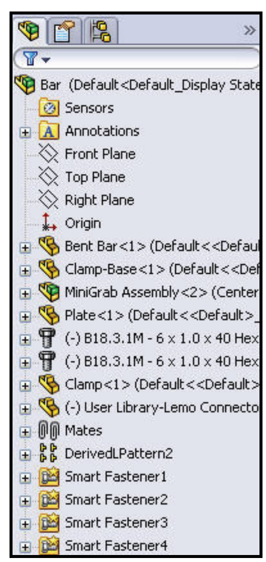
- Click the **Isometric** view tool from the Heads-up View toolbar.



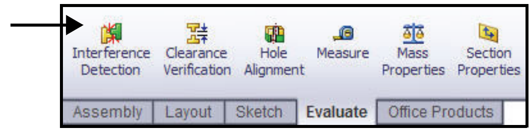
7 Close all documents.

- Click **Window**, **Close All** from the Menu Bar menu. The Bar assembly is complete.





Interference Detection

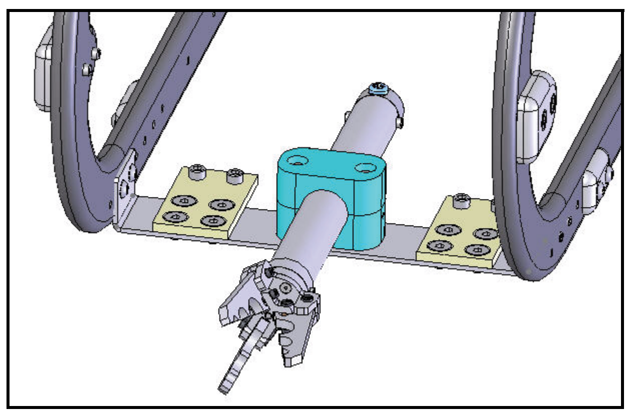


In a complex assembly, it can be difficult to visually determine whether components interfere with each other. With Interference Detection you can:

- Determine the interference between components.
- Display the true volume of interference as a shaded volume.
- Change the display settings of the interfering and non-interfering components.
- Select to ignore interferences such as press fit and threaded fasteners.

Note: The CommandManager tabs are document dependent.

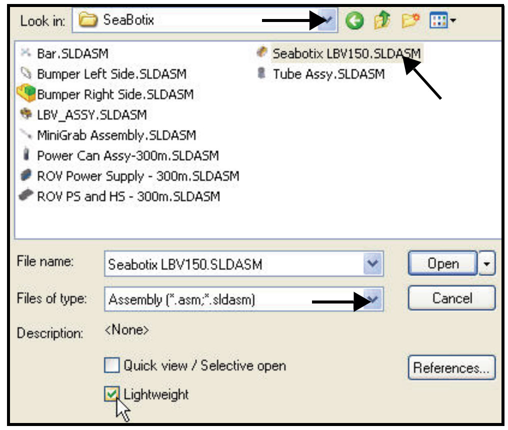
The SeaBotix LBV150 contains hundreds of components. When working with assemblies, you can improve performance significantly by utilizing lightweight components. Lightweight components are visible, however, only a subset of model data is loaded into memory. The remaining model data is loaded on an as needed basis.



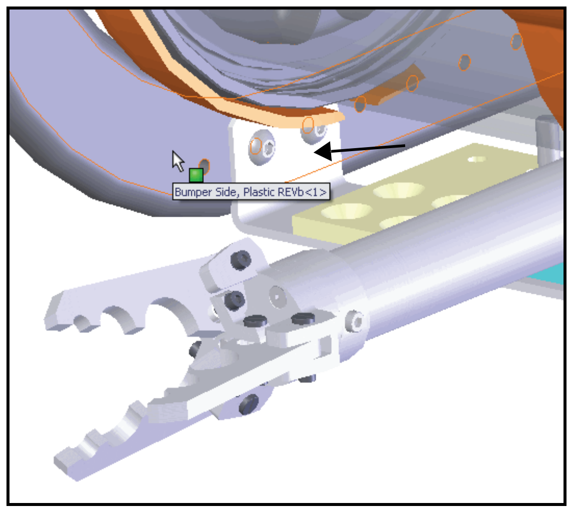
Open the SeaBotix LBV150 assembly in the lightweight state. Explore Interference Detection between the Bent Bar and the Bumper. As you select the lightweight components, SolidWorks automatically loads the model data into memory to calculate the interference volume.

Note: The Clearance Verification  tool provides the ability to check the clearance between selected components in assemblies. The software checks the minimum distance between the components and reports clearances that fail to meet the minimum acceptable clearance you specify.

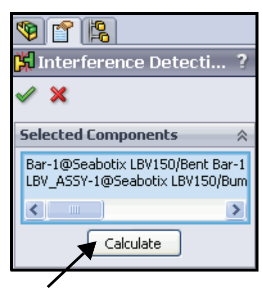
Calculating Interference in an Assembly



1. **Open the SeaBotix LBV150 assembly.**
   - Click **Open** from the Menu Bar toolbar.
   - Select the **SolidWorks Test Drive\SeaBotix** folder.
   - Click the **SeaBotix LBV150** assembly.
   - Check the **Lightweight** box.
   - Click **Open**.

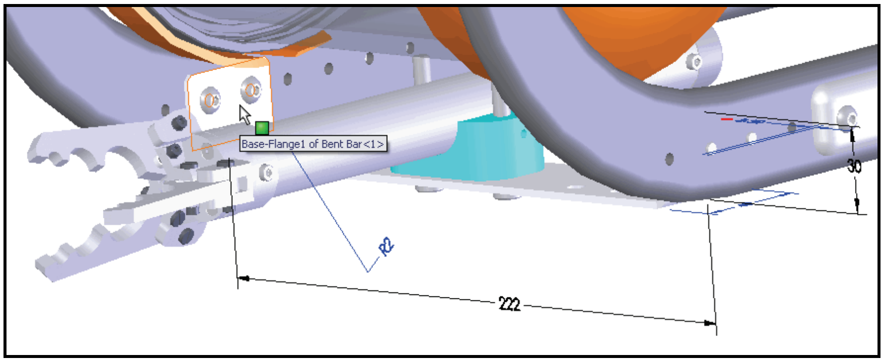


2. **Zoom to area.**
   - Click the **Zoom to Area** tool.
   - **Zoom in** on the left side of the Bent Bar and the Bumper.
   - Click the **Zoom to Area** tool to deactivate.
3. **Select the components for Interference Detection.**
   - Click the **left face** of the Bent Bar.
   - Hold the **Ctrl** key down.
   - Select the **inside face** of the left Bumper.
   - Release the **Ctrl** key.
4. **Calculate the interference.**
   - Click the **Evaluate** tab from the CommandManager.

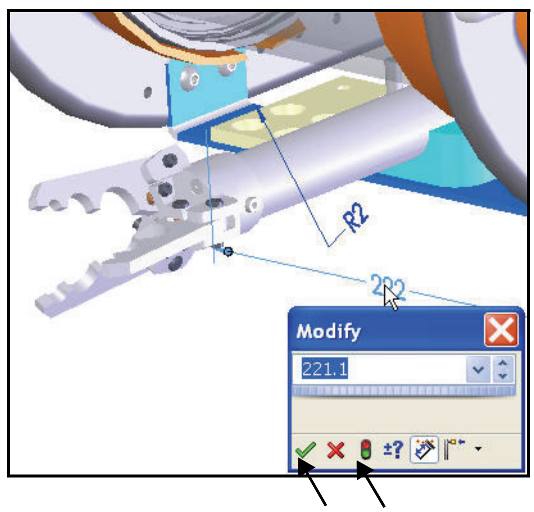


   - Click the **Interference Detection** tool from the Evaluate toolbar. The two selected faces are displayed in the Selected Components box.
   - Click the **Calculate** button. The Results box displays the interference volume. The left face of the Bent Bar interference is displayed in red in the Graphics area.

- Click **OK** from the Interference Detection PropertyManager. The linear distance between the inside faces of the Bumper parts should be 221.1mm.

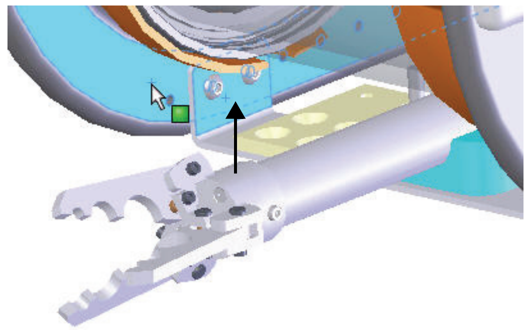


5 Modify the Bent Bar dimension.



- Double-click the left face of the **Bent Bar** in the Graphics area as illustrated. Dimensions are displayed.
- Press the **f** key to fit the model to the Graphics area.
- Double-click the **222** dimensions.
- Enter **221.1**mm.
- Click **Rebuild** from the Modify dialog box.
- Click the **green checkmark** button from the Modify dialog box.
- Click **OK** from the Dimension PropertyManager.

6 **Select the components for the new Interference Detection.**



- Click the **left face** of the Bent Bar.
- Hold the **Ctrl** key down.
- Select the **inside face** of the left Bumper.
- Release the **Ctrl** key.

7 **Calculate the interference.**

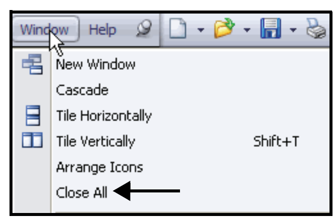
- Click the **Evaluate** tab from the CommandManager.
- Click the **Interference Detection** tool from the Evaluate toolbar. The two selected faces are displayed in the Selected Components box.

- Click the **Calculate** button. The Results box displays No Interferences.
- Click **OK** from the Interference Detection PropertyManager.

8 **Save the SeaBotix LBV150 assembly.**

- Click the **Isometric** view tool from the Heads-up View toolbar.
- Click **Save** from the Menu Bar toolbar.

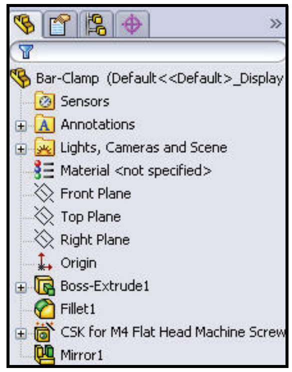
9 **Close all models.**

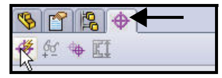


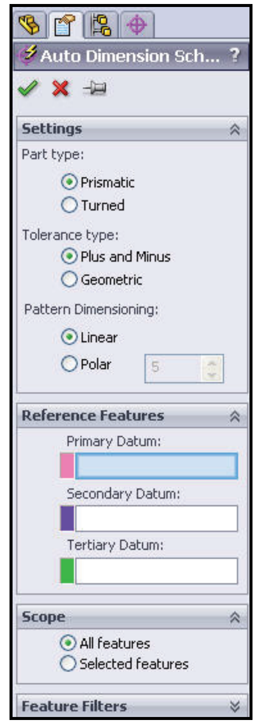
- Click **Windows, Close all** from the Menu Bar menu.

Applying DimXpert to a Part

DimXpert for parts is a set of tools you use to apply dimensions and tolerances to your model according to the requirements of ASME Y14.5 and Y14.41 as well as ISO 1101 and ISO 16792 Standards.







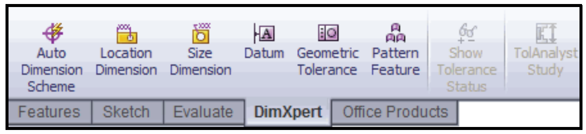
DimXpert tools are accessible from the DimXpertManager tab in the FeatureManager.

The DimXpertManager tab in the FeatureManager and the Auto Dimension Scheme tool provides the ability to insert dimensions and tolerances manually or automatically.

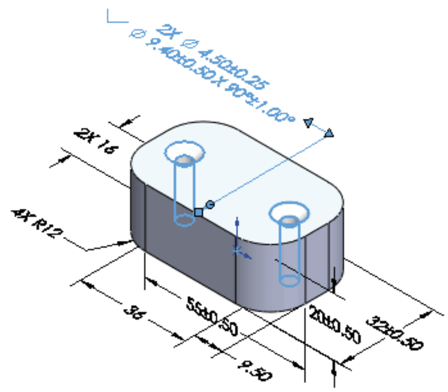
The DimXpertManager provides access to the following selections: **AutoDimension Scheme**, **Show Tolerance Status**, **Copy Scheme**, and **TolAnalyst Study**.

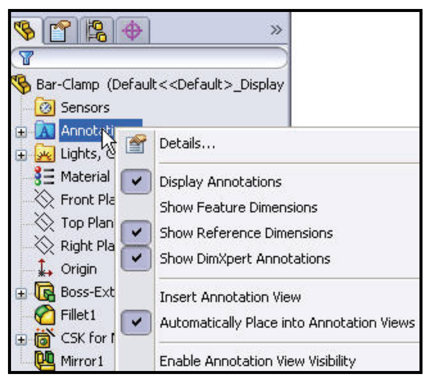
Note: DimXpert dimensions are used in TolAnalyst™. TolAnalyst is a tolerance analysis tool used to study the effects tolerances and assembly methods have on dimensional stack-up between two features of an assembly. The result of each study is a minimum and maximum tolerance stack, a minimum and maximum root sum squared (RSS) tolerance stack, and a list of contributing features and tolerances. TolAnalyst is available in SolidWorks Premium.

Note: Apply the default settings: Prismatic and Plus and Minus from the Settings dialog box.



Note: To hide DimXpert Annotations, right-click the **Annotations** folder from the FeatureManager, and un-check the **Show DimXpert Annotations** box.



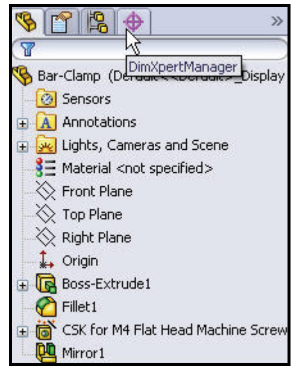


Utilizing DimXpert for a Part

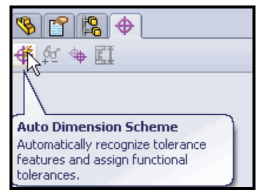
1 Open the Bar-Clamp part.

- Click **Open** from the Menu Bar toolbar.
- Double-click the **Bar-Clamp** part from the SolidWorks Test Drive folder. The FeatureManager is displayed.

2 Apply the DimXpert tool.



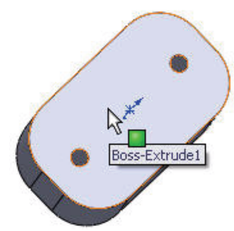
- Click the **DimXpertManager** tab in the FeatureManager. The DimXpertManager is displayed.



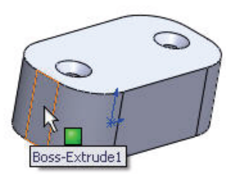
- Click the **Auto Dimension Scheme** tool from the DimXpertManager. The Auto Dimension PropertyManager is displayed.

Note: Prismatic, Plus and Minus, and Linear are selected by default.

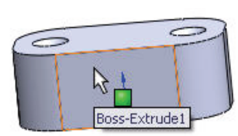
3 Select the Reference Planes.



- Use the middle mouse button and **rotate** the model to view the bottom face.
- Click the **bottom face** of the model as illustrated. Plane1 is displayed in the Primary Datum box.
- Click **inside** the Secondary Datum box.

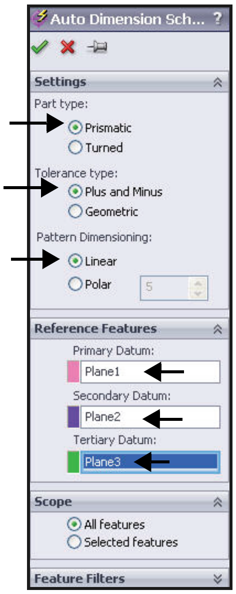


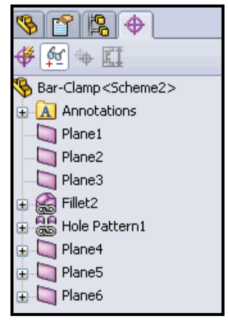
- Use the middle mouse button and **rotate** the model to view the left face.
- Click the **left face** of the model as illustrated. Plane2 is displayed in the Secondary Datum box.
- Click **inside** the Tertiary Datum box.



- Use the middle mouse button and **rotate** the model to view the front face.
- Click the **front face** of the model as illustrated. Plane3 is displayed in the Tertiary Datum box. The three planes are selected and displayed in the Reference Features box.

■ Click **OK** from the Auto Dimension PropertyManager. View the results. All faces are displayed in green in the Graphics area.





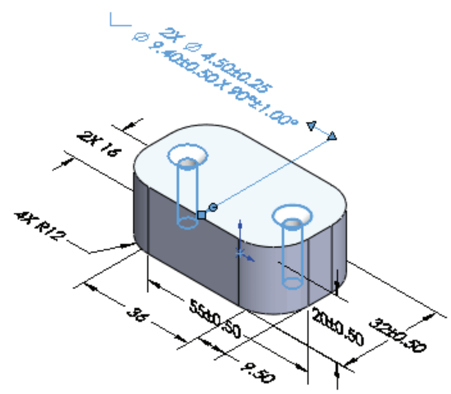
Note: DimXpert uses topology recognition. The benefit of topology recognition is that it recognizes manufacturing features that model recognition cannot, such as slots, notches, and pockets. Only topology recognition is used for features on imported bodies. Topology features update if you make geometry changes, but new instances are not added to pattern features.

4 Display an Isometric view.

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

■ Press the **z** key to fit the model to the Graphics area.

■ Click and drag the **dimensions** off the model to improve visibility.

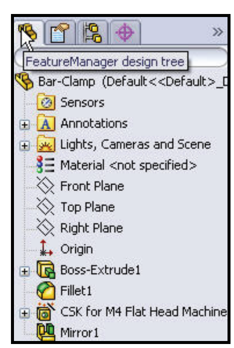


5 Return to the FeatureManager.

■ Click the **FeatureManager** tab.

6 Save the model.

■ Click **Save** from the Menu Bar toolbar.



Notes

What About Drawings?

SolidWorks allows you to easily create drawings from parts or assemblies. These drawings are fully associative with the parts and assemblies they reference. If you change a dimension on the finished drawing, that change propagates back to the model. Likewise, if you change the model, the drawing updates automatically.

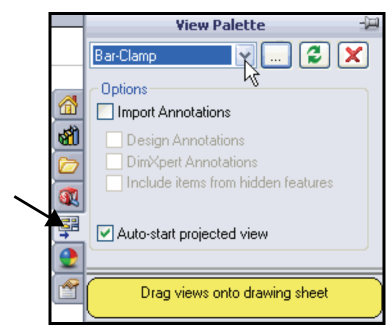
Drawings communicate three things about the objects they represent:

- **Shape** – Views communicate the shape of an object.
- **Size** – Dimensions communicate the size of an object.
- **Other information** – Notes communicate nongraphic information about manufacturing processes such as drill, ream, bore, paint, plate, grind, heat treat, remove burrs, and so forth.

Creating a Drawing – General Procedure

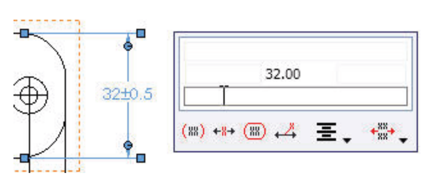
1. Open a new drawing of the desired size.
2. Open the part or assembly you wish to detail.
3. Add views — usually three standard views plus any specialized views such as detail, auxiliary, or section views.
4. Insert the dimensions and arrange the dimensions on the drawing.
5. Add additional sheets, views, and/or notes if required.

You will create a multi-sheet drawing that contains parts, assemblies and a Bill of Materials. Begin with the default drawing template. The first sheet contains an Isometric view of the Bar assembly.

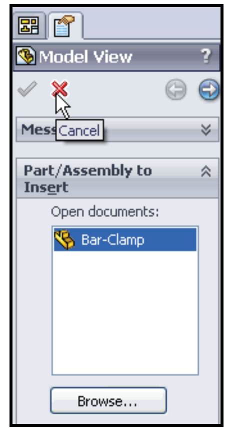


Note: The **View Palette**  tab located in the Task Pane provides the ability to click and drag drawing views into an active sheet. The View Palette contains images of standard views, annotation views, section views, and flat pattern (sheet metal parts) of the selected model.

Note: SolidWorks 2010 provides a new dimension palette to reduce mouse travel, dimension editing, and to contain the most common fields to be edited.



Creating a New Drawing



1 Create a new drawing.

- Click **New** from the Menu Bar toolbar.
- Double-click the **Drawing** template from the Hands-on Test Drive tab.
- The Sheet Format/Size dialog box is displayed. A (ANSI) Landscape is selected by default. Click **OK**.
- The Model View PropertyManager is displayed. Click **Cancel** from the Model View PropertyManager.

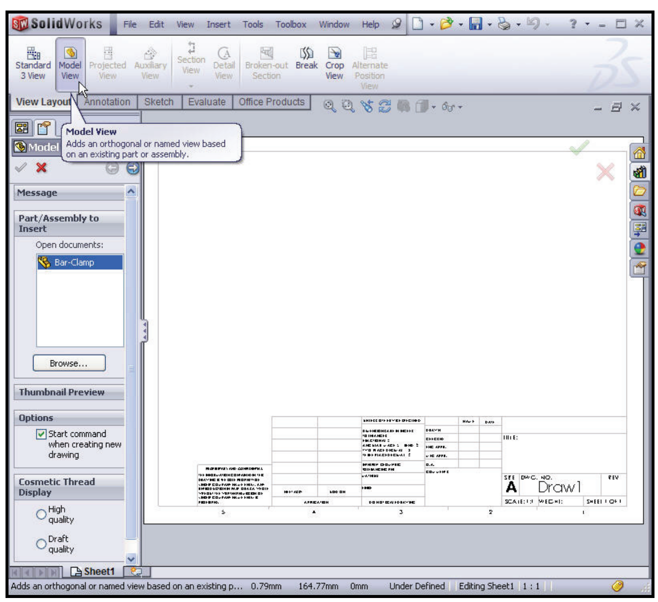
Note: The Model View PropertyManager is displayed if the Start command when creating new drawing is checked from the Options box.

2 View the sheet properties.

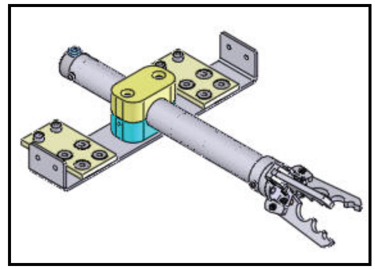
- Right-click **Properties** in the Graphics area. Third angle projection, A (ANSI) Landscape, with a 1:1 scale is displayed.
- Click **OK** from the Sheet Properties dialog box.

3 View the drawing.

- Click the **View Layout** tab. The View Layout toolbar is displayed.
- Click the **Model View** tool. The Model View PropertyManager is displayed.



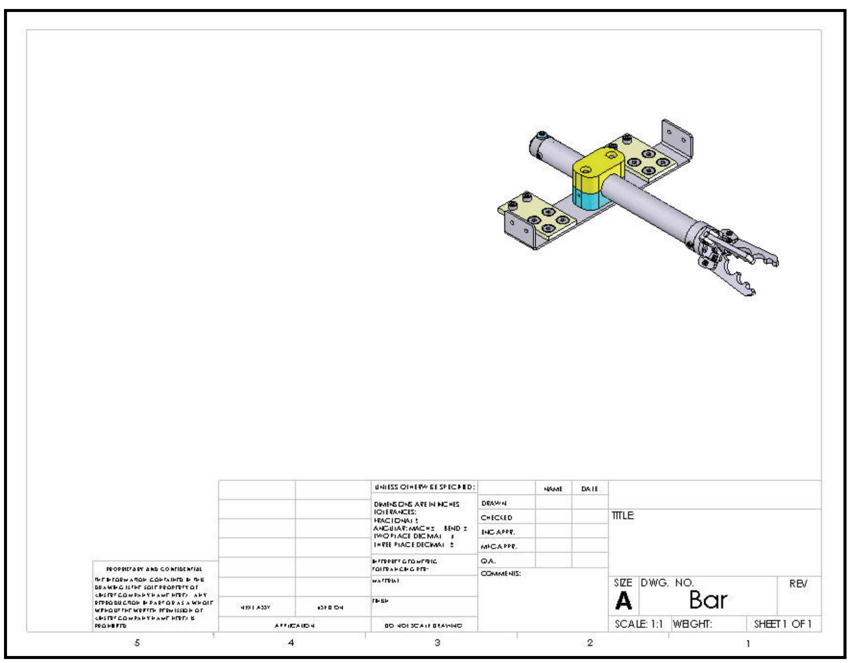
Adding an Isometric View



Pictorial views such as an Isometric view make drawings easier to read and visualize. Because SolidWorks models are true three-dimensional solids, creating pictorial views is as simple as the click of a mouse. Also, since a SolidWorks drawing can reference several different files, parts, or assemblies, you can add detail views of various components on the same sheet or additional sheets in the same drawing.

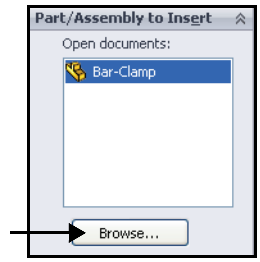
Isometric View

- Displays the part with height, width, and depth equally foreshortened.
- Pictorial rather than Orthographic.
- Shows all three dimensions: height, width, and depth.
- Easier to visualize than Orthographic views.

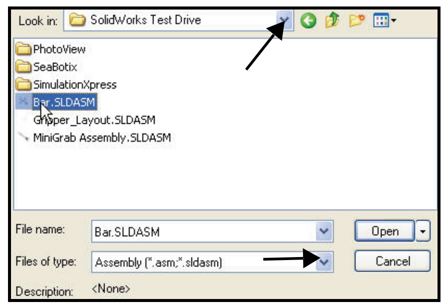


Inserting a Drawing View

1 Open an assembly.
- Click **Browse** from the Model View PropertyManager.



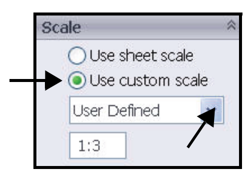
2 Select the Bar assembly.
- Set Files of type: **Assembly**.
- Double-click **Bar**.



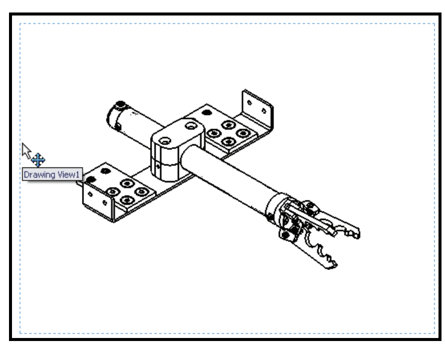
3 Select an Isometric view.
- Single view is selected by default. Click ***Isometric** for Standard views.

4 Set the scale.
- Click **Use custom scale**.
- Select **User Defined**.
- Enter **1:3**.

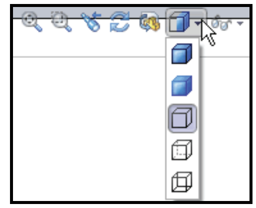
5 Position the view.
- Click in the **upper right corner** of the sheet to position the Isometric view.
- Click **OK** from the Drawing View1 PropertyManager.



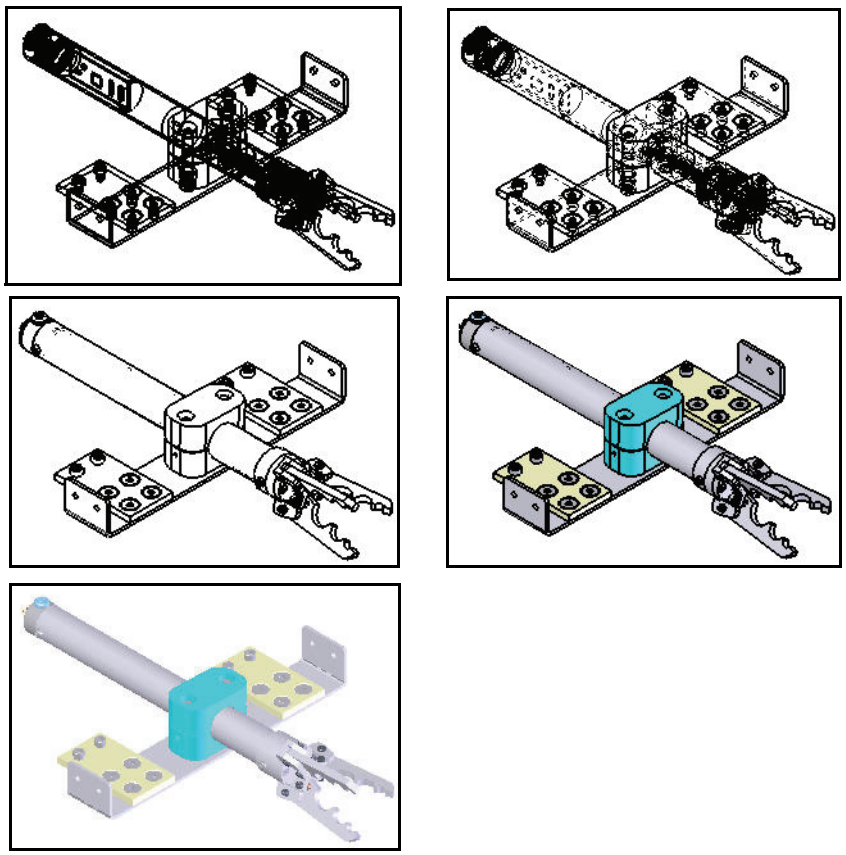
Note: If you do not like the view position, click and drag the view boundary to the correct location.



Drawing View Display



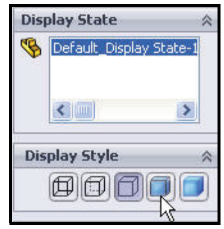
Views can be displayed in several ways: **Wireframe**, **Hidden Lines Visible**, **Hidden Lines Removed**, **Shaded With Edges**, and **Shaded**. The following illustration shows the ways in which models can be displayed in a drawing.

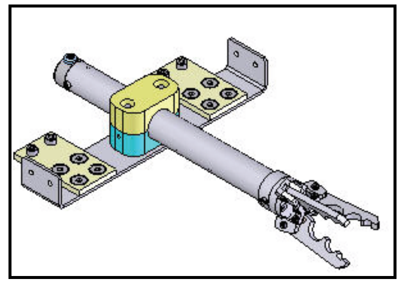


Changing the Drawing View Display

1 **Select the view display.**

- Click inside the **Isometric view** boundary of the Bar. The Drawing View1 PropertyManager is displayed.
- Click **Shaded With Edges** from the Display Style box.





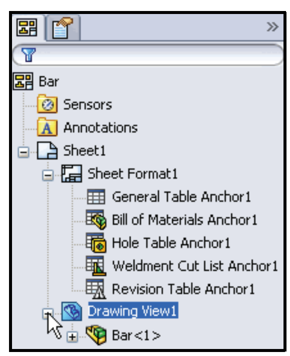
Tip: You can either select the drawing view boundary or select inside the view.

2 **Deactivate the origins.**

- If needed, click **View**, de-select **Origins** from the Menu Bar menu.

3 **Accept the defaults.**

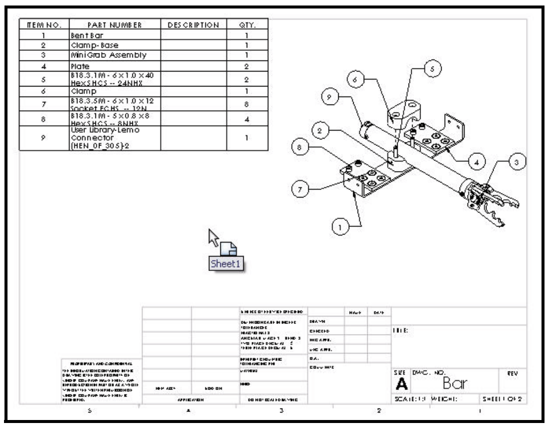
- Click **OK** from the Drawing View1 PropertyManager. View the Bar Drawing FeatureManager.



Note: Later, display Tangent Edges Removed in the Isometric drawing view.

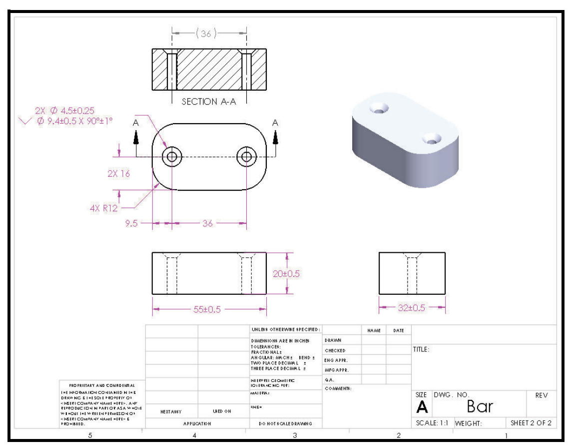
Adding a Sheet to the Drawing

A SolidWorks drawing can contain multiple sheets. Each sheet can contain multiple drawing views. A new sheet is added to display different types of views. Use the drawing sheet tabs at the bottom of the screen to quickly switch between sheets and make selections.

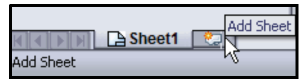


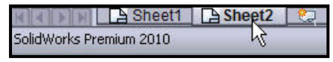
Sheet1

Sheet2



Adding a New Sheet to a Drawing



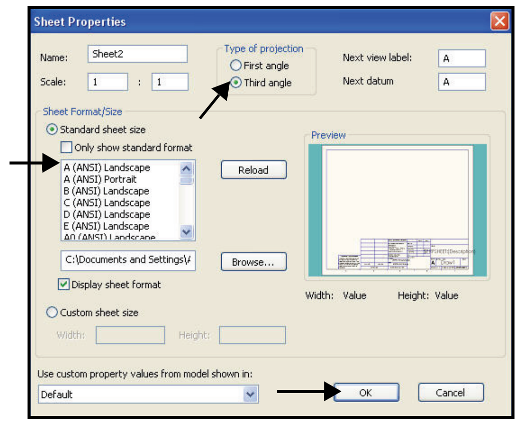


1 Add a new Sheet.

- Click the **Add Sheet** tab in the bottom corner of the Graphics area. Sheet2 is displayed. You can add as many sheets to a drawing as you need.

2 View the sheet default properties.

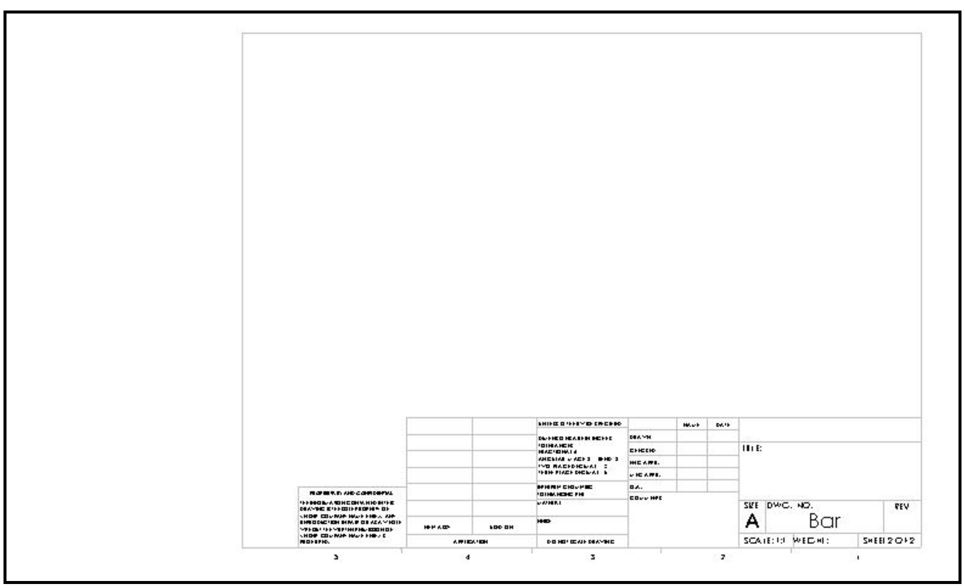
- Right-click **Properties** in the Graphics area. The default name is Sheet2. A (ANSI) Landscape, Third angle projection is selected.



3 Accept the defaults.

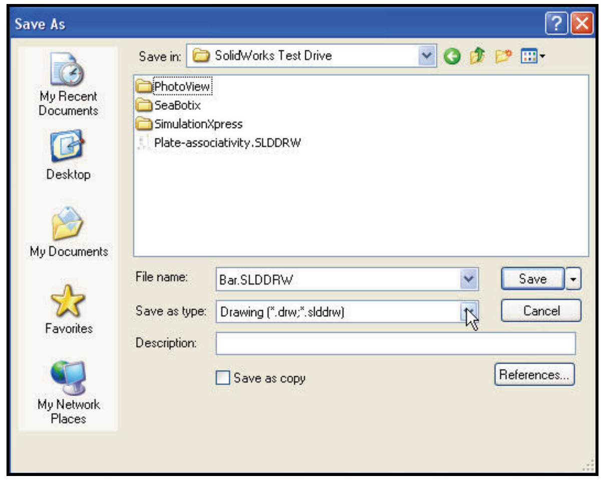
- Click **OK** from the Sheet Properties dialog box. A new drawing sheet is created within the same drawing. Sheet2 is active. The names of the two sheets appear in the named tabs at the bottom of the Graphics area.

Note: By default, SolidWorks selects the same sheet format to keep the title block information consistent.

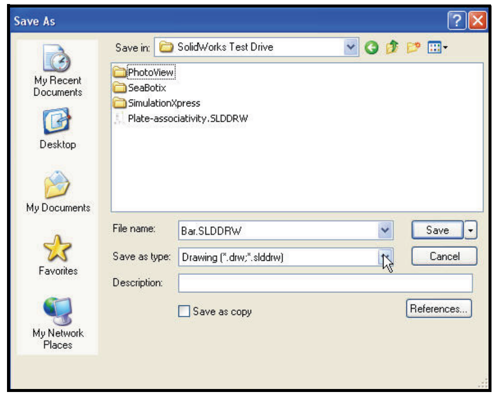


Saving the Drawing

Drawing files have the .slddrw extension. A new drawing takes the name of the first model inserted. The name appears in the title bar. When you save the drawing, the name of the model appears in the Save As dialog box as the default file name, with the default extension .slddrw. You can edit the name before saving the drawing.



Saving the Drawing

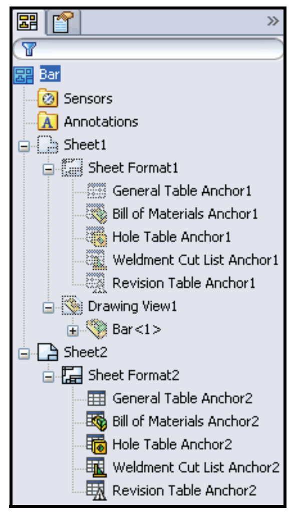


1 Save the drawing.

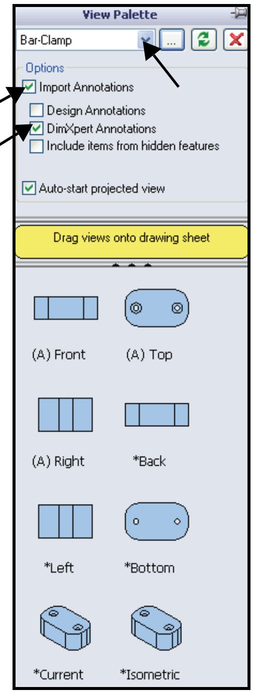
- Click **Save** from the Menu Bar toolbar. Bar is the default drawing name.
- Click **Save** from the Save As dialog box.

2 View the updated FeatureManager.

- View the updated Drawing FeatureManager.



Inserting Standard Views with Annotations

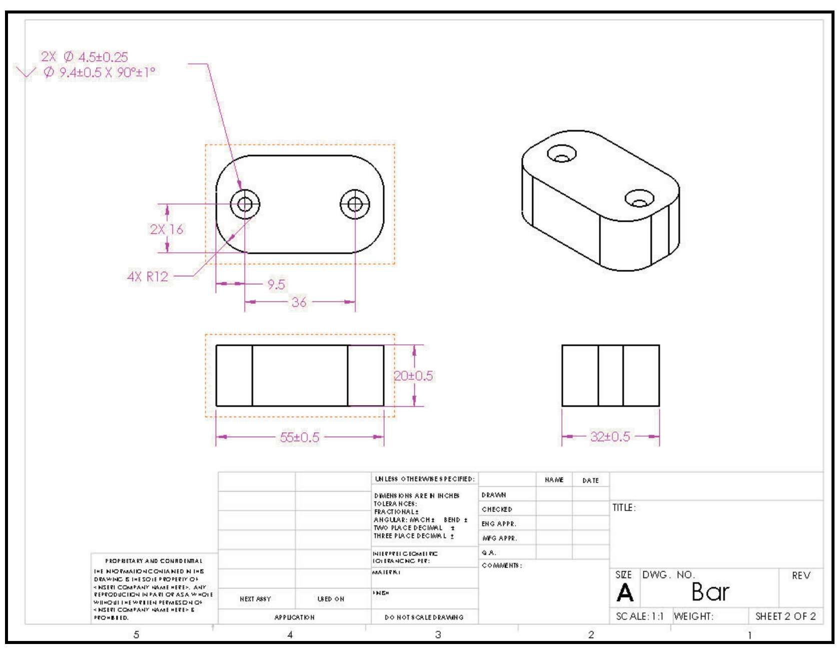


In SolidWorks, creating standard Orthographic views is as easy as click and drag. Click the View Palette tab from the Task Pane.

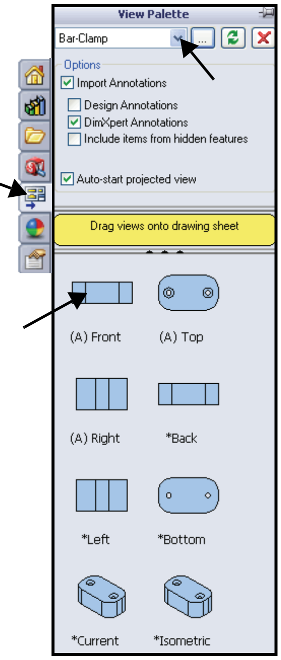
Insert four standard views; Front, Top, Right, and Isometric of the Bar-Clamp part.

Import dimensions and tolerances you created using DimXpert for the Bar-Clamp.

Note: At this time, the Bar-Clamp part is open. Drawing dimensions in drawing views may vary.



Inserting Four Standard Views with Annotations



1 Select the Bar-Clamp part.

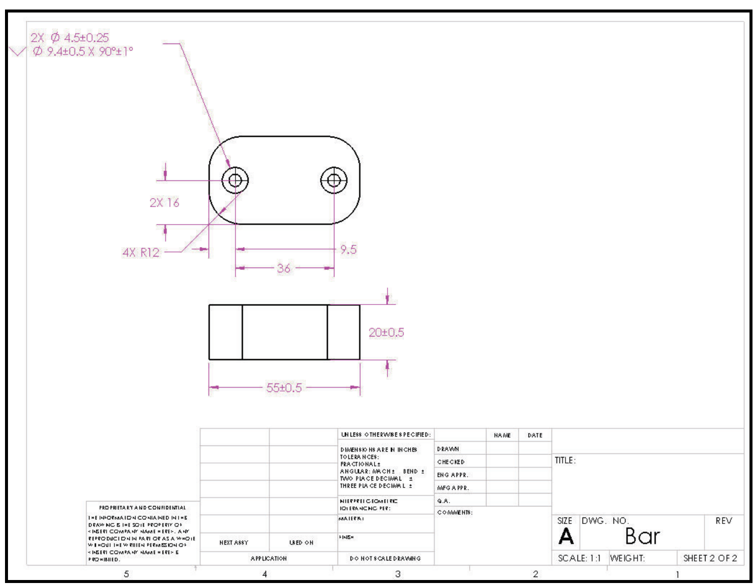
- Click the **View Palette** tab from the Task Pane.
- Select **Bar-Clamp** from the drop-down menu in the View Palette.

2 Insert four views with imported Annotations from DimXpert.

- Check the **Import Annotations** box.
- Check the **DimXpert Annotations** box.
- Click and drag the **Front view** from the View Palette onto the drawing sheet in the lower left corner as illustrated.

3 Create the Top view.

- Click a **position** directly above the Front view. The Top view is created.

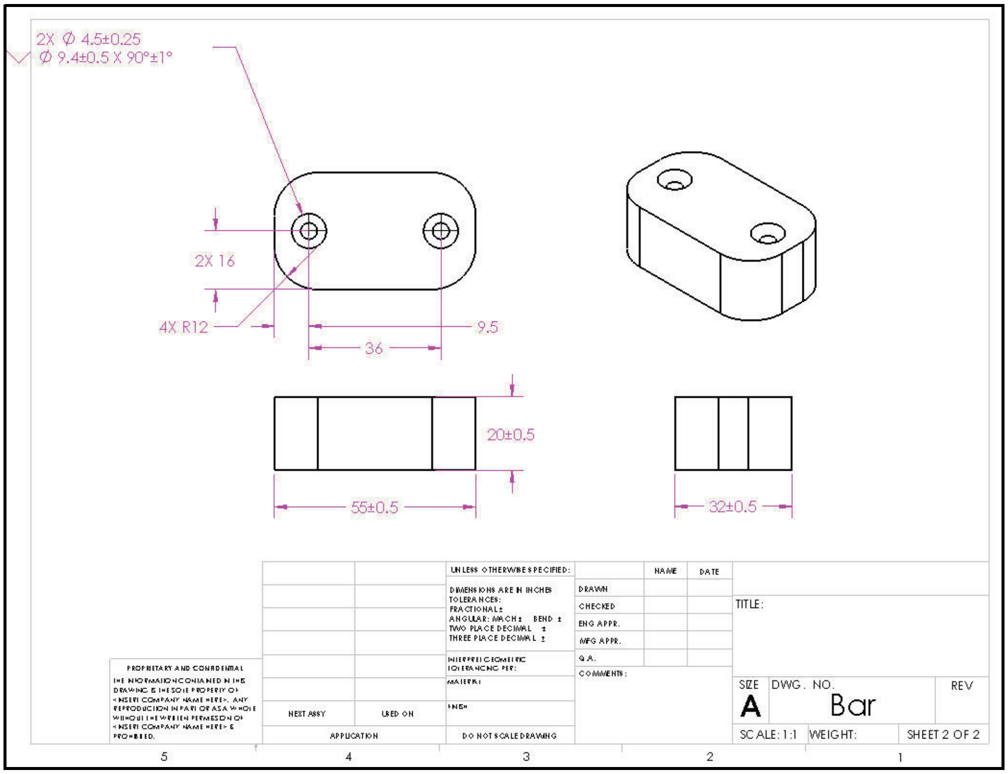


4 Create the Right view.

- Click a **position** directly to the right of the Front view.

5 Create the Isometric view.

- Click a diagonally **position** as illustrated.



6 View the results.

- Click **OK** from the Projected View PropertyManager. The four views are displayed on Sheet2. Click and drag the views, address the arrow heads, and relocate the annotations in the next section.

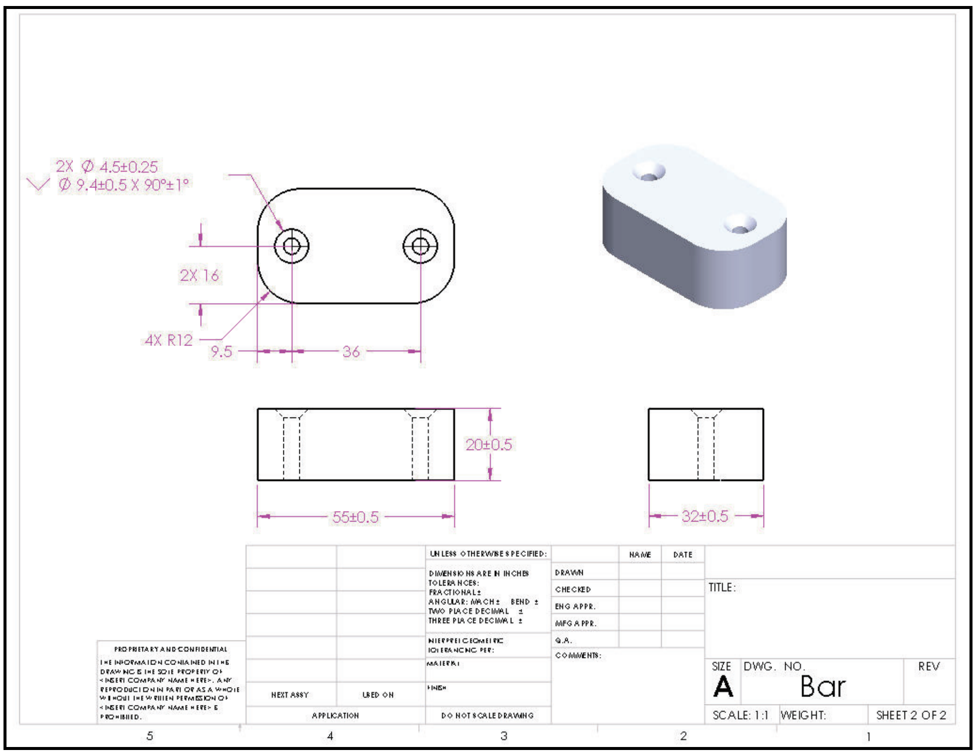
Fine-tuning the Drawing Views

There are four views on Sheet2 in the Bar drawing.

Before you add anything else to this drawing, move the drawing views and relocate the view annotations.

The mouse pointer provides feedback in both the Drawing Sheet and the Drawing View mode.

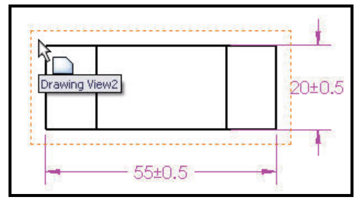
Note: View the mouse pointer for feedback to select Sheet, View, Component, and Edge properties in a drawing.



Manipulating Drawing Views and Annotations

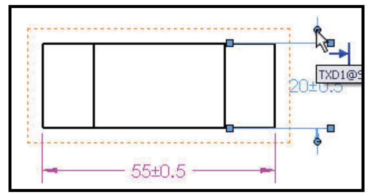
1 Reposition the Front view.

- Click and drag the **Front view boundary**. The alignment of the Top and Right views is fixed in relation to the Front view. The Top view can be moved vertically, and the Right view can be moved horizontally.



2 Move the dimensions off the view.

- Click and drag each **dimension** off the model.



3 Modify the dimension arrow direction.

- **Zoom in** on the dimension arrow.
- Click the **dimension**.
- Click the **blue dot** on the diameter dimension to toggle the arrow as illustrated.

4 Reposition the Top view.

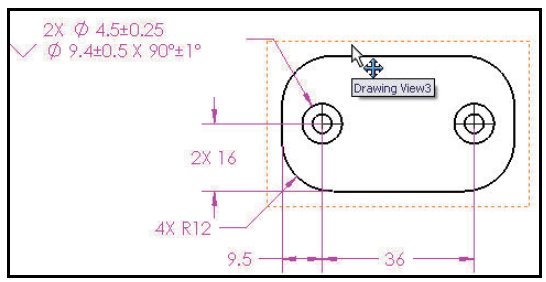
- Click and drag the **Top View boundary**.

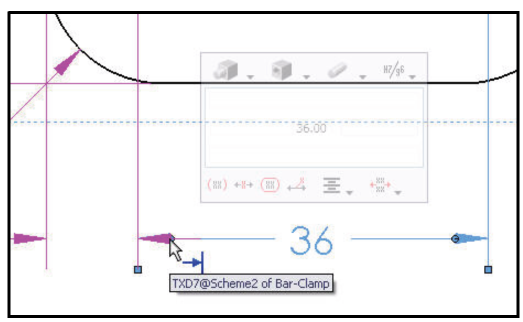
5 Move the dimensions off the view.

- Click and drag each **dimension** off the model.

6 Modify the dimension arrow direction.

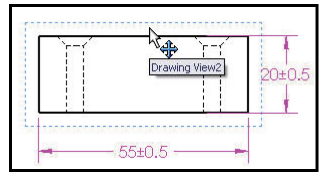
- If required, click the **blue dot** on the diameter dimension to toggle the arrow as illustrated.





7 Display Hidden Lines Visible.

- Click inside the **Front view** boundary.

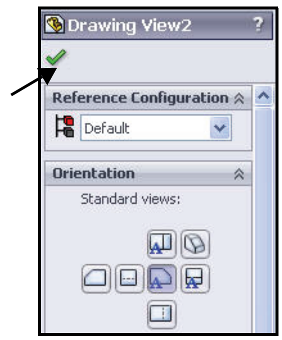


Note: The Drawing View mode icon.

- Click **Hidden Lines Visible** from the Display Style box. The Front, Top, and Isometric views are displayed in Hidden Lines Visible mode.

- Right-click inside the **Front view** boundary.
- Click **Tangent Edge**, **Tangent Edges Removed**.

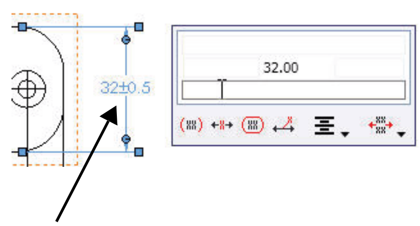
- Click **OK** from the Drawing View2 PropertyManager.
- **Perform** the same above procedure for the Right view.

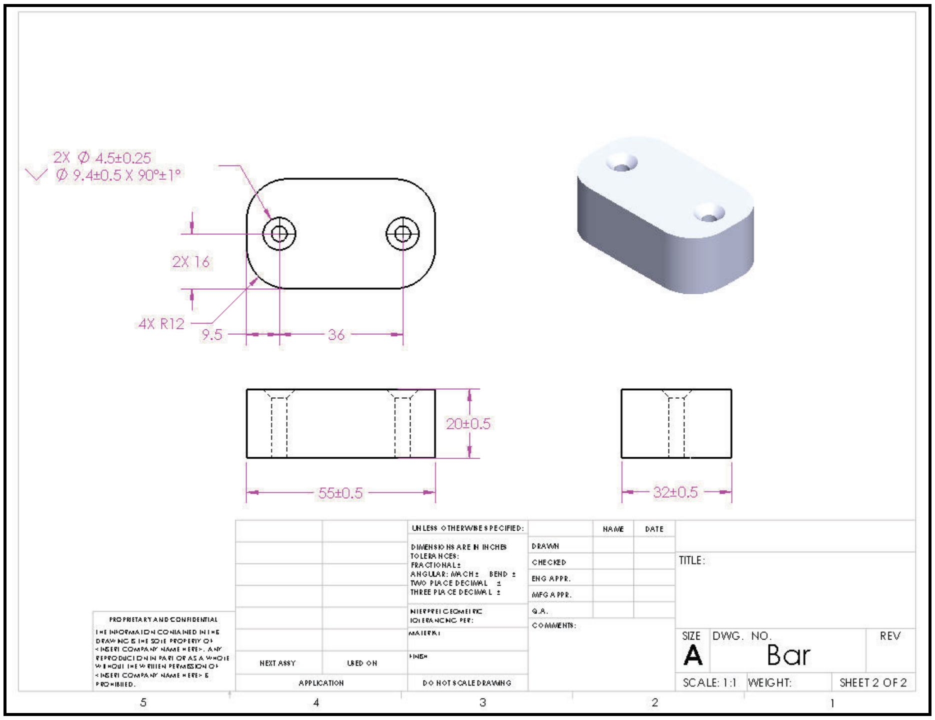


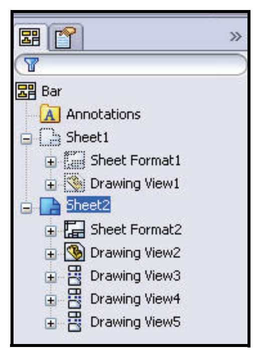
8 Display a Shaded Isometric view.

- Click inside the **Isometric view boundary**.
- Click **Shaded** from the Display Style box.
- Click **inside** Sheet2. View the FeatureManager.

Note: New in 2010 is the ability to edit a dimension in the pop-up dimension palette. This reduces mouse travel and simplifies dimension editing from the Dimension PropertyManager, and contains the most common fields to be edited. Recent edits and predefined standards can be previewed and recalled instantly.

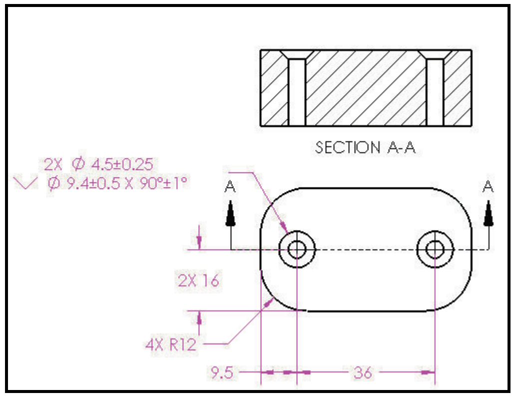



2X Ø 4.5±0.25
⌵ Ø 9.4±0.5 X 90°±1°
2X 16
4X R12
9.5
36
20±0.5
55±0.5
32±0.5
UNLESS OTHERWISE SPECIFIED:
NAME
DATE
DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL±
ANGULAR: MACH± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±
DRAWN
CHECKED
ENG APPR.
MFG APPR.
Q.A.
COMMENTS:
TITLE:
INTERPRET GEOMETRIC TOLERANCING PER:
MATERIAL
FINISH
NEXT ASSY
USED ON
APPLICATION
DO NOT SCALE DRAWING
PROPRIETARY AND CONFIDENTIAL
SIZE
DWG. NO.
REV
A
Bar
SCALE: 1:1
WEIGHT:
SHEET 2 OF 2
5
4
3
2
1


Bar
Annotations
Sheet1
Sheet Format1
Drawing View1
Sheet2
Sheet Format2
Drawing View2
Drawing View3
Drawing View4
Drawing View5

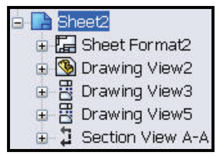
Adding a Section View

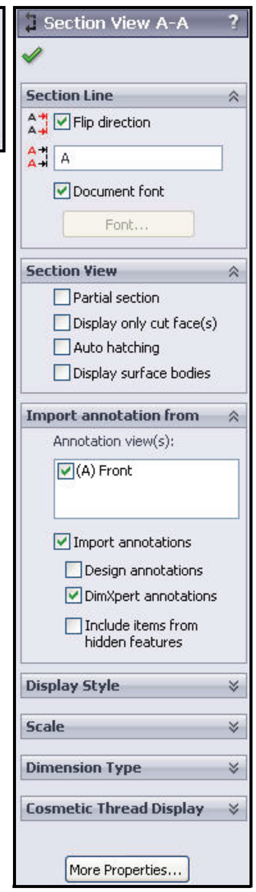
The Section view shows you a slice of a part as if you took a saw and cut it open. Section views are very effective to look at details that would otherwise be very difficult to see.



Section views can be very tedious to create manually in a 2D drafting system. Because SolidWorks models are solids, creating a Section view is nearly automatic. A section view:

- Displays the internal structure of a model.
- Requires a section cutting plane.





Create a Section view in the drawing by cutting the parent view with a section line.

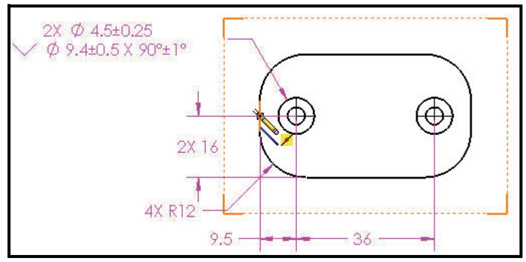
The section view can be a straight cut section or an offset section defined by a stepped section line.

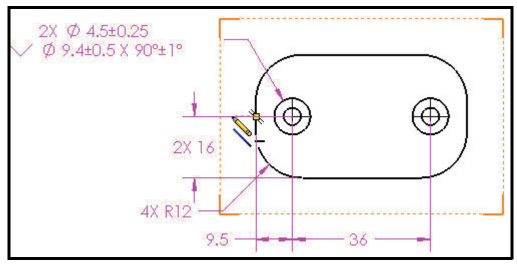
Note: If the section line isn't drawn long enough, the Section view will be incorrect. The section line should run nearly the full width of the view, from border to border.

Adding a Section View

1 Locate the midpoint.

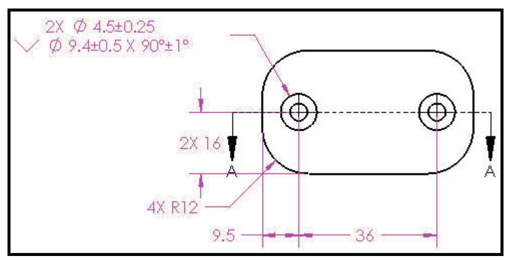
- Click the **Section View** tool from the View Layout toolbar. The Line tool is activated. The Section View PropertyManager is displayed.
- Locate the midpoint of the left vertical edge on the Top view for a reference. Position the **mouse pointer** over the center of the left vertical edge. Do not click the midpoint.





2 Sketch the section line.

- Drag the **mouse pointer** to the left.
- Click a **position** to the left of the profile.
- Click a **position** horizontally to the right of the profile. The section line extends beyond the left and right view boundary.

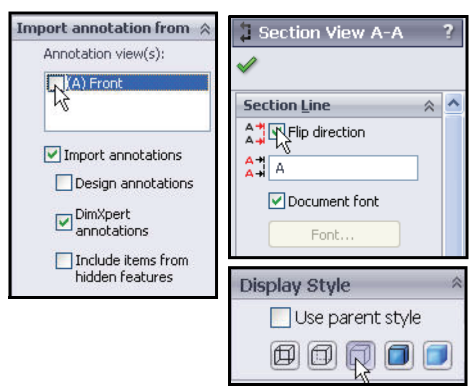


3 Place the Section view.

- Click a **position** above the Top view. The section arrows point downward.
- Check the **Flip direction** in the Section View A-A PropertyManager.
- Un-check the **(A)Front** box as illustrated.

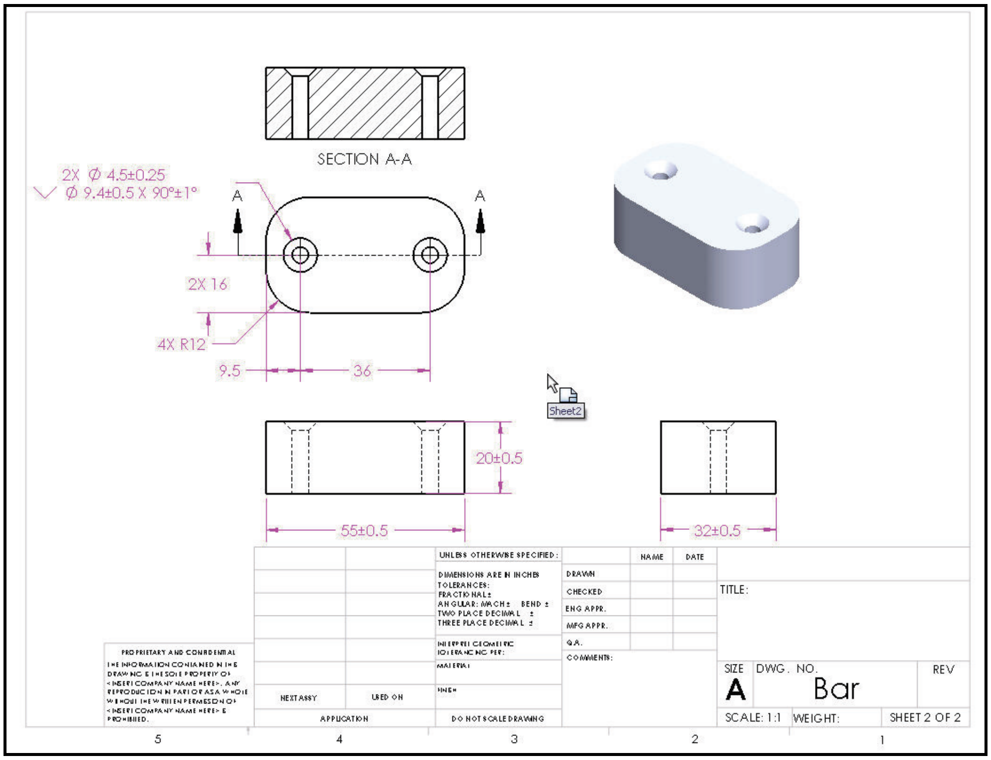
4 Display Hidden Lines Removed.

- Click **Hidden Lines Removed** from the Display Style box.

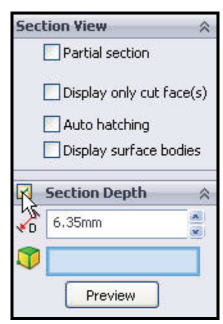


5 View the results.

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

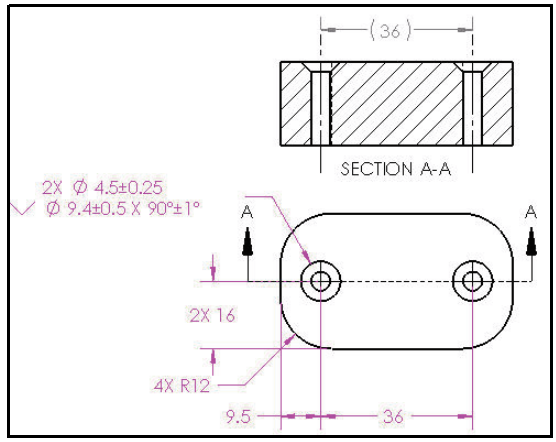


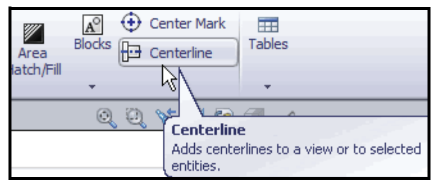
Note: New in 2010 is the ability to set the depth of a section view in parts by specifying how far beyond the section view line you want to see. This is available under Section Depth in the Section View PropertyManager.



Adding an Annotation and Reference Dimension

Insert two centerlines into the Section view. Apply the Centerline tool from the Annotation toolbar.

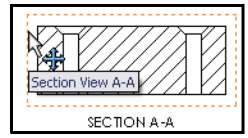




Note: You can insert centerlines into drawing views automatically or manually. SolidWorks avoids duplicate centerlines.

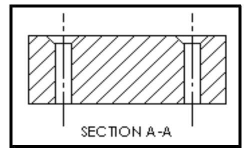
Insert a Reference dimension using the Smart Dimension sketch tool.

Adding an Annotation and Reference Dimension



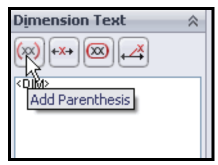
1 Insert Centerlines.

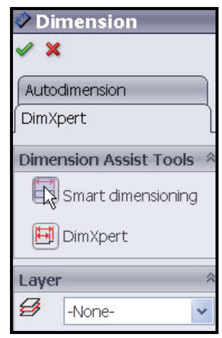
- Click inside the **Section view** boundary as illustrated. The Section View A-A PropertyManager is displayed.
- Click the **Centerline** tool from the Annotation toolbar. Centerlines are displayed in the Section view.



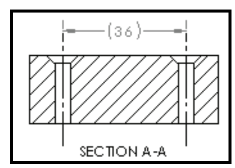
- Click **OK** from the Centerline PropertyManager.

2 Add a reference dimension between the two centerlines.





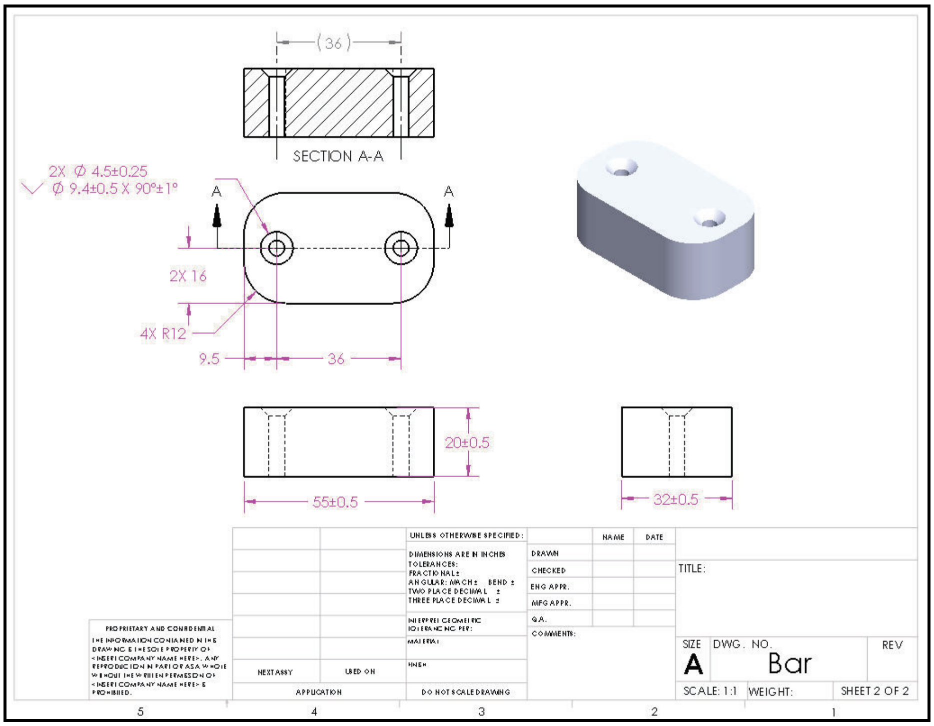
- Click the **Smart Dimension** tool from the Annotation toolbar.
- Click the **Smart dimensioning** box from the Dimension Assist Tools dialog box.
- Click the **left centerline**.
- Click the **right centerline**.
- Click a **position** above the profile to place the reference dimension.
- If needed, click the **Add Parenthesis** box as illustrated.



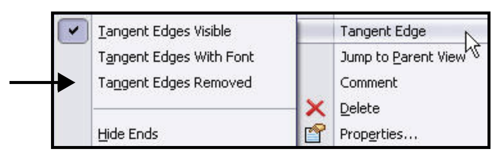
- Click **OK** from the Dimension PropertyManager.

3 View the final drawing.

- **View** Sheet2 of the Bar drawing.

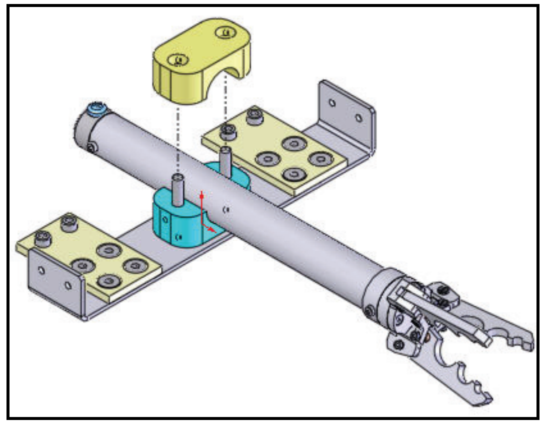


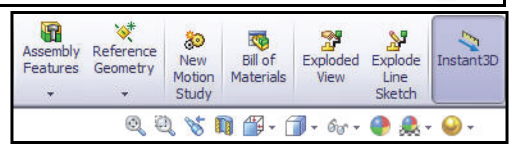
Note: To Display Tangent Edges Removed, right-click in a drawing view, click Tangent Edge, click Tangent Edges Removed.



Exploded Views

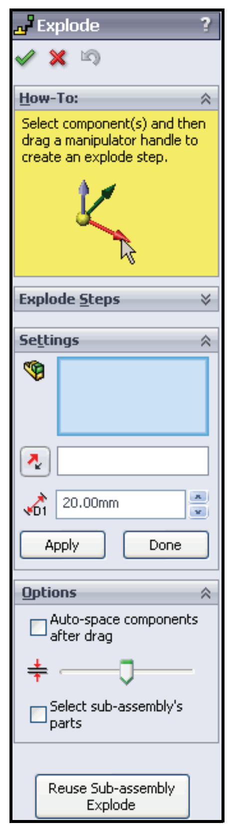
For manufacturing purposes, it is often useful to separate the components of an assembly to visually analyze their relationships. Exploding the view of an assembly allows you to look at it with the components separated. While an assembly is exploded, you cannot add mates to the assembly.



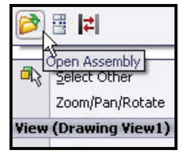


An exploded view consists of one or more explode steps. An exploded view is stored with the assembly configuration with which it is created. Each configuration can have one exploded view.

Today, you will create a single exploded view in the Bar assembly, and then display the exploded view in the Bar drawing.

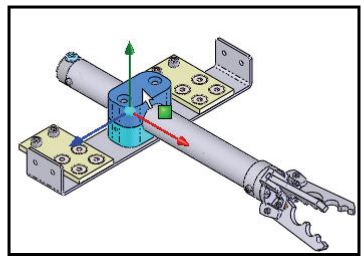


Inserting an Exploded View

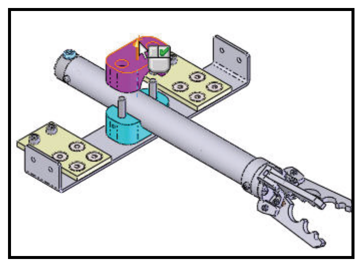


1 **Open the Bar assembly.**
- Click the **Sheet1** tab. Sheet1 is displayed.
- Right-click inside the **Isometric view** boundary.
- Click **Open Assembly**. The Bar FeatureManager is displayed.
- Click the **Isometric** view tool from the Heads-up View toolbar.

2 **Insert an Exploded view.**
- Click the **Exploded View** tool from the Assembly toolbar. The Explode PropertyManager is displayed.

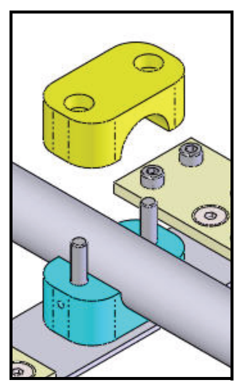


3 **Select the Clamp.**
- Click the **top face** of the Clamp in the Graphics area.
- Drag the **green/orange axis** of the reference triad upward above the Bar assembly. Explode Step1 is created.



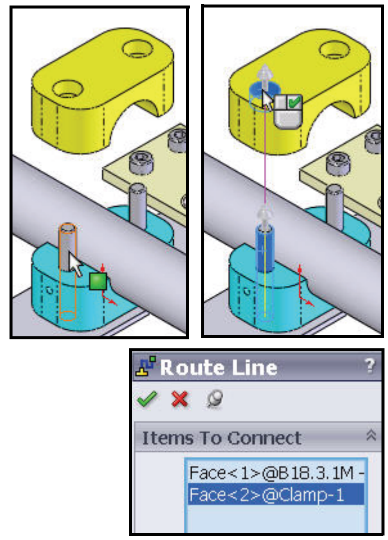
4 **View the results.**
- Click **OK** from the Explode PropertyManager.

5 **Zoom in on the Clamp.**
- Click the **Zoom to Area** tool.
- **Zoom in** on the Clamp.
- Click the **Zoom to Area** tool to deactivate.



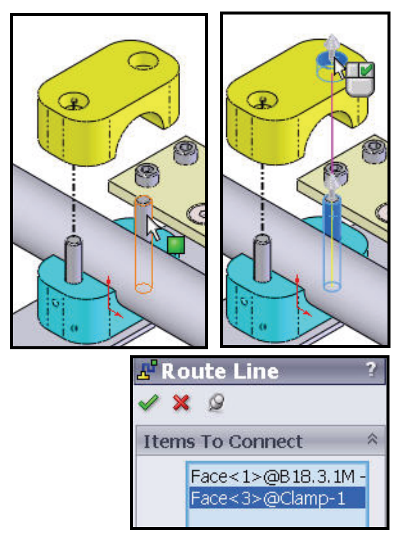
6 Insert the front route line.

- Click the **Explode Line Sketch** tool from the Assembly toolbar. The Route Line PropertyManager is displayed.
- Click the **face** of the front B18.3.1M Hex SHCS as illustrated.
- Click the **face** of the front inside CBORE hole as illustrated. The selected faces are displayed in the Items To Connect box.
- Click **OK** from the Route Line PropertyManager.

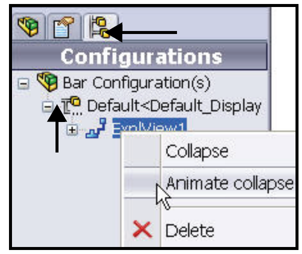


7 Insert the back route line.

- Click the **face** of the back B18.3.1M Hex SHCS as illustrated.
- Click the **face** of the back inside CBORE hole. The selected faces are displayed in the Items To Connect box.
- Click **OK** from the Route Line PropertyManager.
- Click **OK** from the Route Line PropertyManager.



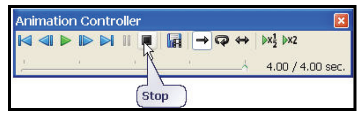
8 Exit and save the model.



- Click **Save** from the Menu Bar toolbar.
- Click **Yes**.

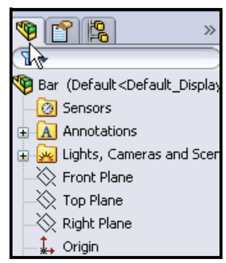
9 Return to a collapse state.

- Click the **ConfigurationManager** tab.
- **Expand** Default.
- Right-click **ExplView1**.
- Click **Animate collapse**. View the animation of the Clamp.
- If required, click **Stop** from the Animation Controller box.



- Click **Close** from the Animation Controller box.

10 Return to the FeatureManager.



- Click the **FeatureManager design tree** tab.

11 Save the Bar assembly.

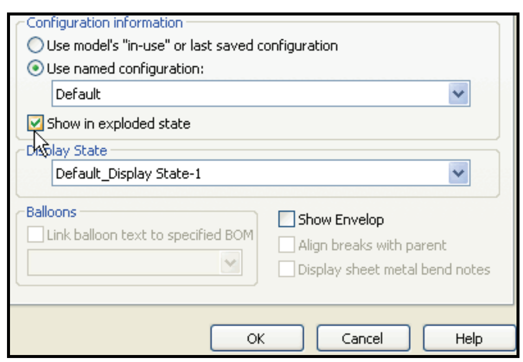
- Click the **Isometric** view tool from the Heads-up View toolbar.
- Click **Save** from the Menu Bar toolbar.

12 Return to the Bar drawing.

- Click **Window**, **Bar - Sheet1** from the Menu Bar toolbar. Sheet1 of the Bar drawing is displayed.

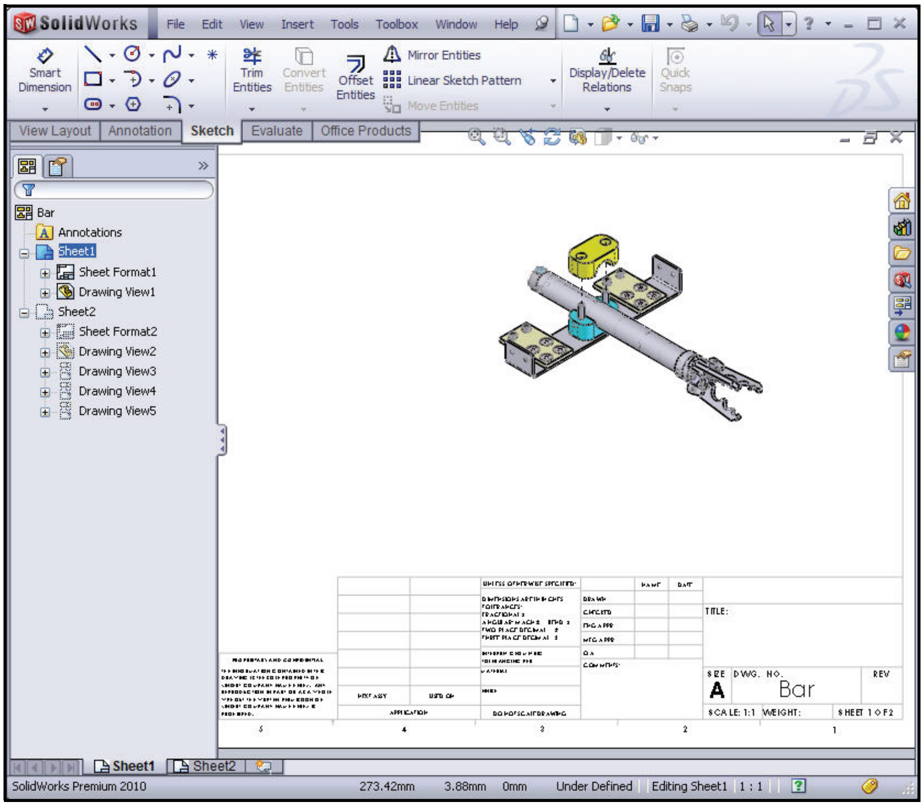
13 Set the Exploded state.

- Right-click inside the **Isometric view** boundary.
- Click **Properties**.
- Check the **Show in exploded state** box.
- Click **OK** from the Drawing View Properties dialog box.
- Click **OK** from the Drawing View 1 PropertyManager.
- Click **Rebuild** from the Menu Bar toolbar.



14 Save the Bar drawing.

- Click **Save** from the Menu Bar toolbar. View the results.

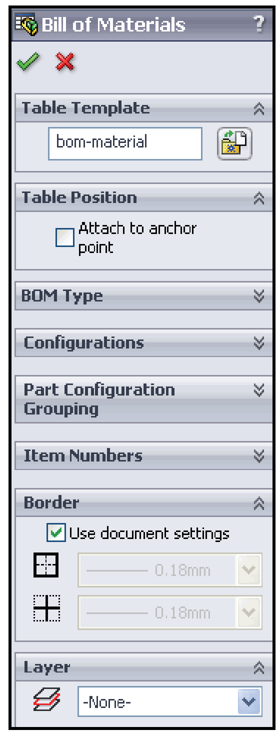


Bill of Materials

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Bent Bar		1
2	Clamp-Base		1
3	MiniGrab Assembly		1
4	Plate		2
5	B18.3.1M - 6 x 1.0 x 40 Hex SHCS -- 24NHX		2
6	Clamp		1
7	B18.3.5M - 6 x 1.0 x 12 Socket FCHS -- 12N		8
8	B18.3.1M - 5 x 0.8 x 8 Hex SHCS -- 8NHX		4
9	SW3dPS-Lemo Connector (HEN_0F_305)-2		1

A drawing can contain a table-based Bill of Materials or an Excel-based Bill of Materials, but not both.

The table-based Bill of Materials is based on SolidWorks tables and includes:



- Templates
- Anchors
- Quantities for configurations
- Whether to keep items that have been deleted from the assembly
- Zero quantity display
- Excluding assembly components
- Following assembly order
- Item number control

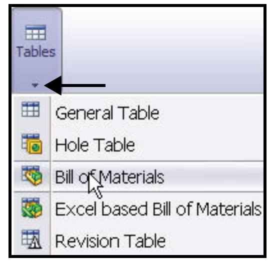
You can specify a starting Item Number, but the increment is always a single integer (1, 2, 3...., 100, 101, 102..., for example).

You can change the text in any cell by double-clicking and editing on the screen, but if you edit data generated by SolidWorks (Item Number, Quantity, and so on), you break the link between the model data and the Bill of Materials.

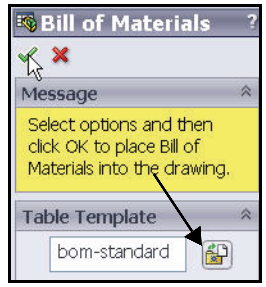
Note: You can now set options for specific assembly components directly from the BOM. You can link drawing BOM's copied from a previously created assembly BOM using the Bill of Materials PropertyManager.

Creating a Bill of Materials

1 Create a Bill of Materials.

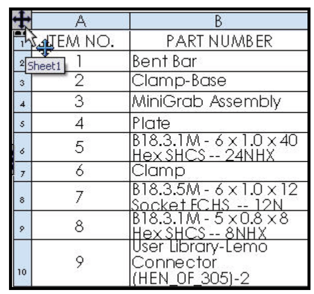


- Click inside the **Isometric view** boundary in Sheet1. The Drawing View1 PropertyManager is displayed.
- Click the **Annotation** tab from the CommandManager.
- Click the **Bill of Materials** tool from the Consolidated Tables drop-down menu. The Bill of Materials PropertyManager is displayed. Accept the default bom-standard for Table Template.
- Click **OK** from the Bill of Materials PropertyManager.



2 Locate the BOM table.

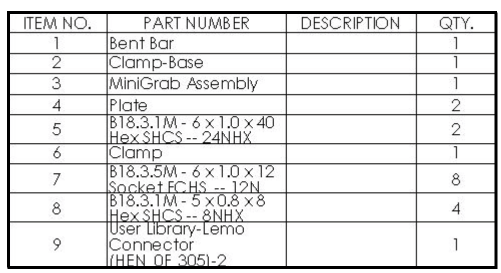
- Click the **left corner** of Sheet1 to position the Bill of Materials. The table creates the following columns: ITEMS NO., PART NUMBER, DESCRIPTION, and QTY.



ITEM NO.	PART NUMBER
1	Bent Bar
2	Clamp-Base
3	MiniGrab Assembly
4	Plate
5	B18.3.1M - 6 x 1.0 x 40 Hex SHCS -- 24NHX
6	Clamp
7	B18.3.5M - 6 x 1.0 x 12 Socket FCHS -- 12N
8	B18.3.1M - 5 x 0.8 x 8 Hex SHCS -- 8NHX
9	User Library-Lemo Connector (HEN_0E_305)-2

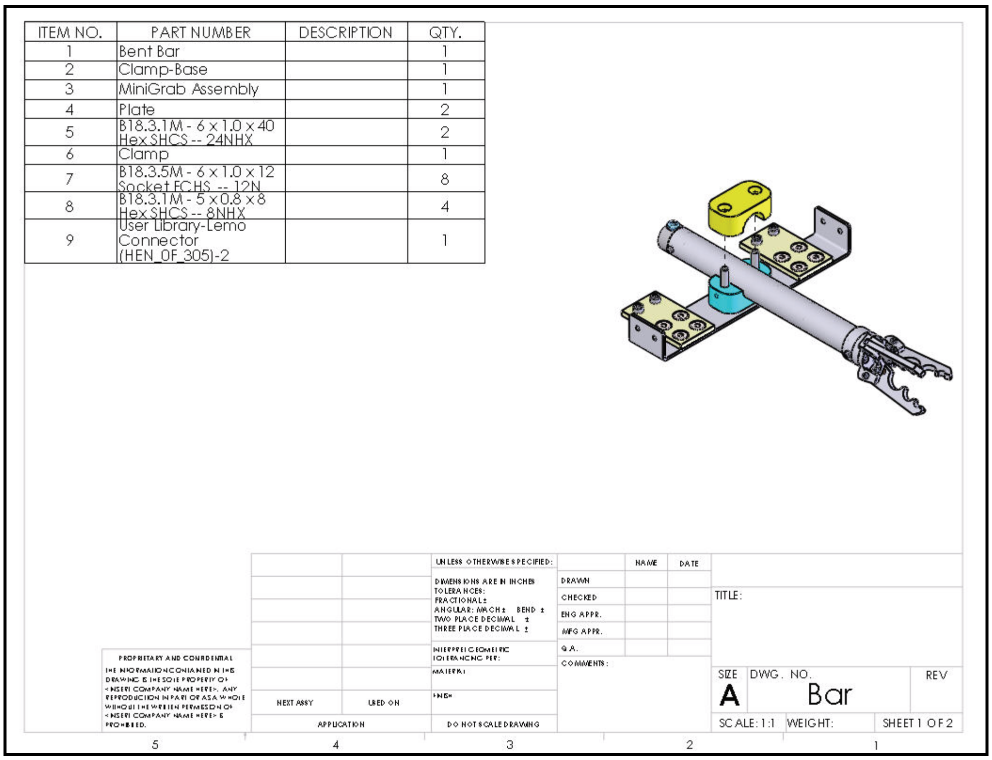
3 Resize the BOM table.

- Resize the BOM table to fit in the upper left corner of the drawing. Click **inside** the Graphics area.
- Click and drag the **vertical lines** of the table to decrease or increase the width of each column.
- Click **inside** the Graphics area to deselect the Bill of Materials.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Bent Bar		1
2	Clamp-Base		1
3	MiniGrab Assembly		1
4	Plate		2
5	B18.3.1M - 6 x 1.0 x 40 Hex SHCS -- 24NHX		2
6	Clamp		1
7	B18.3.5M - 6 x 1.0 x 12 Socket FCHS -- 12N		8
8	B18.3.1M - 5 x 0.8 x 8 Hex SHCS -- 8NHX		4
9	User Library-Lemo Connector (HEN_0E_305)-2		1

Note: By default, SolidWorks utilizes the filename for Part Number in the Bill of Materials. Column headings such as: Part Number, Description, Cost, and Material can be assigned as custom properties in the part. Custom properties are then passed to the drawing.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Bent Bar		1
2	Clamp-Base		1
3	MiniGrab Assembly		1
4	Plate		2
5	B18.3.1M - 6 x 1.0 x 40 Hex SHCS -- 24NHX		2
6	Clamp		1
7	B18.3.5M - 6 x 1.0 x 12 Socket FCHS -- 12N		8
8	B18.3.1M - 5 x 0.8 x 8 Hex SHCS -- 8NHX		4
9	User Library-Lemo Connector (HEN_0F_305)-2		1

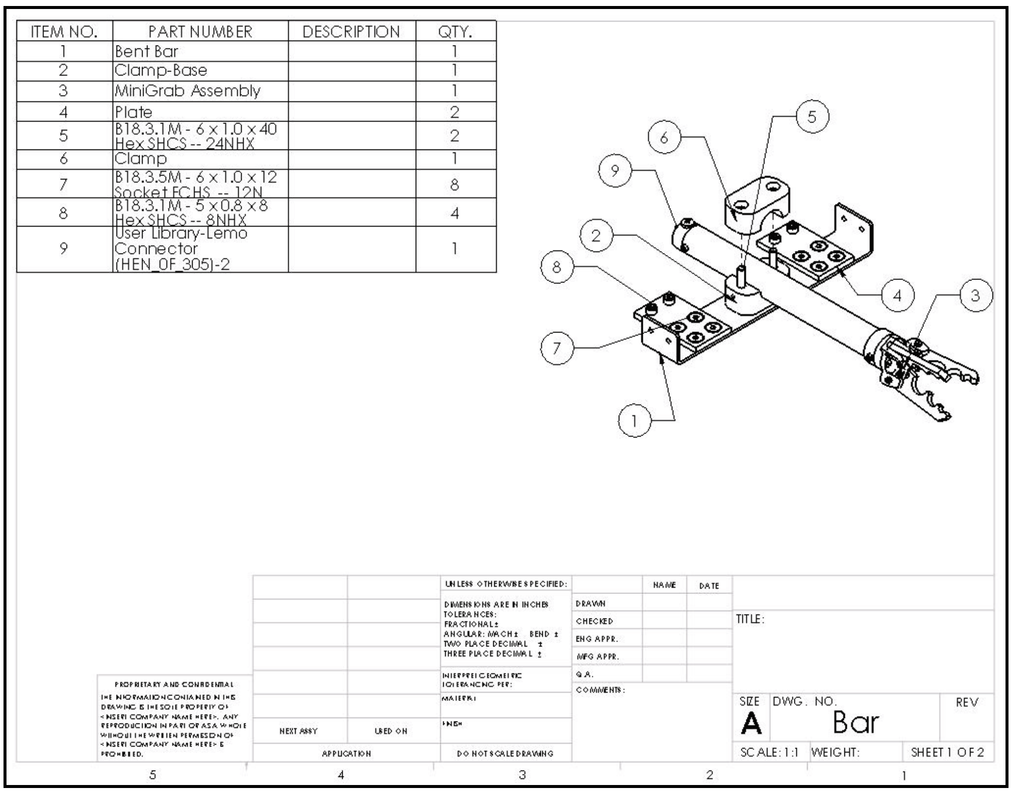
Note: A Bill of Materials can be created in an Assembly document.

Balloons in the BOM

You can create balloons in a drawing document. The balloons label the parts in the assembly and relate them to item numbers of the Bill of Materials (BOM).

However, you do not have to insert a BOM in order to add balloons.

If the drawing has no BOM, the item numbers are the default model values. If there is no BOM on the active sheet, but there is a BOM on another sheet, the numbers from that BOM are used.



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Bent Bar		1
2	Clamp-Base		1
3	MiniGrab Assembly		1
4	Plate		2
5	B18.3.1M - 6 x 1.0 x 40 Hex SHCS -- 24NHX		2
6	Clamp		1
7	B18.3.5M - 6 x 1.0 x 12 Socket FCHS -- 12N		8
8	B18.3.1M - 5 x 0.8 x 8 Hex SHCS -- 8NHX		4
9	User Library-Lemo Connector (HEN_0F_305)-2		1

Adding Balloons to a Drawing

1 Select the view.

- Click inside the **Isometric view** boundary.
- Click **Hidden Lines Removed** from the Display Style box.

2 Add balloons.

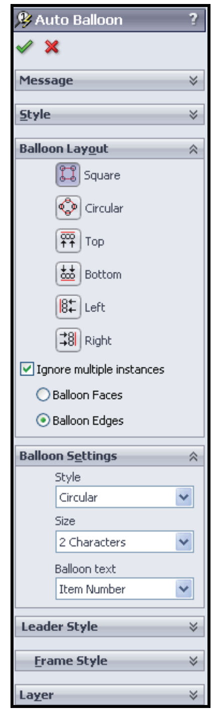
- Click the **AutoBalloon** tool from the Annotation toolbar. The AutoBalloon PropertyManager is displayed. Accept the default options.
- Click **OK** from the Auto Balloon PropertyManager.

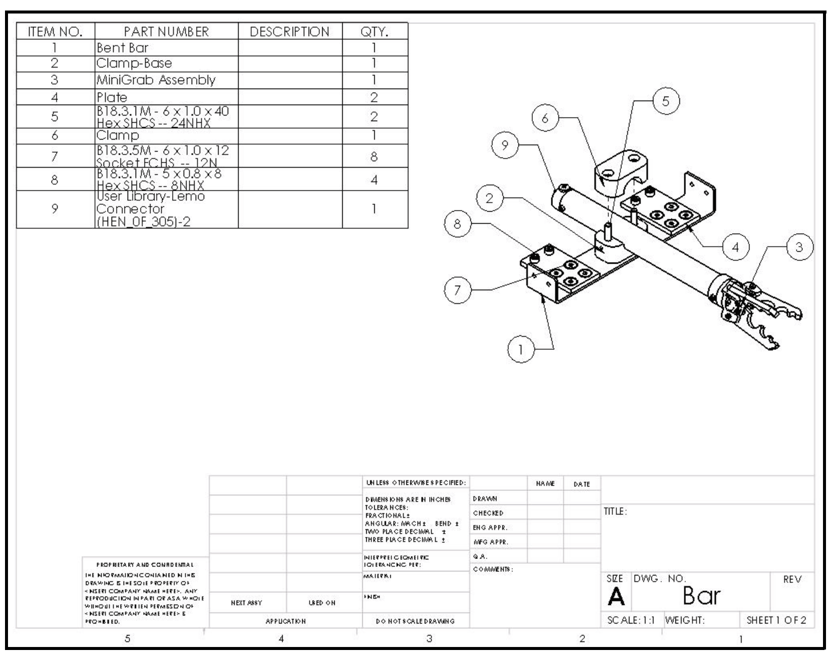
3 Reposition the balloons.

- Click and drag the **balloons** as illustrated on Sheet1.

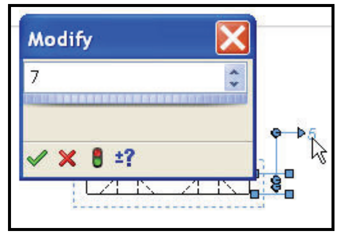
4 Save the Bar drawing.

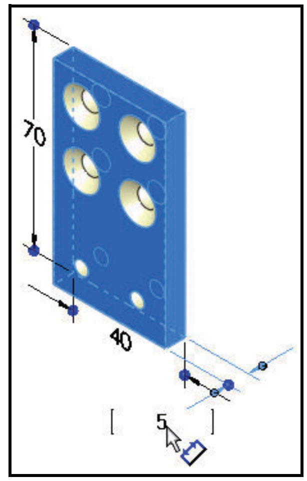
- Click **Rebuild** from the Menu Bar toolbar.
- Click **Save** from the Menu Bar toolbar.





Associativity





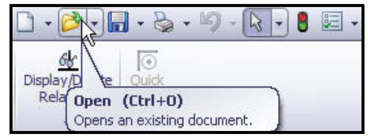
In SolidWorks, parts, assemblies and drawings are associative. If you create a change to a part, the change will propagate to any and all assemblies and drawings that reference that part.

- Changing the values of imported dimensions will change the part.
- You cannot change the values of manually inserted dimensions.
- Changing a dimension on the drawing changes the model.

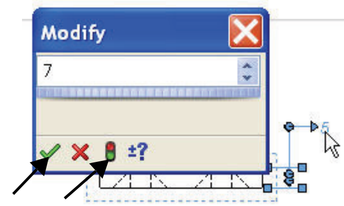
You will first open a drawing and view the dimensions. You will them modify the depth of the Plate-associative part in the drawing. Next view the depth change in the part and then in the associative drawing.

Note: Model dimensions are associative between parts, assemblies, and drawings. Reference dimensions and DimXpert dimensions are one-way associative from the part to the drawing or from the part to the assembly. DimXpert dimensions show up in a different color to help identify them from model dimensions and reference dimensions.

Changing a Model Associativity

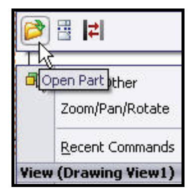


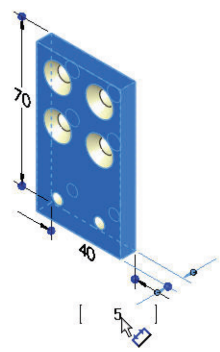
1 **Open the Plate-associativity drawing.**
- Click **Open** from the Menu Bar toolbar.
- **Browse** to the SolidWorks Test Drive folder.
- Select **Drawing** for Files of type.
- Double-click **Plate-associativity**. The Drawing FeatureManager is displayed.



2 **Modify the depth dimension.**
- Double-click the **5** depth dimension in the Top view.
- Enter **7**mm.
- Click **Rebuild** from the Modify dialog box.
- Click the **green check mark** from the Modify dialog box.

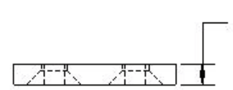
3 **Open the Plate-associativity part.**
- Right-click in the **Front view**.
- Click **Open Part**. The Part FeatureManager is displayed.
- Click **Extrude1** from the FeatureManager.
- Click the **7**mm dimension. The Modify dialog box is displayed.
- Enter **5**mm.



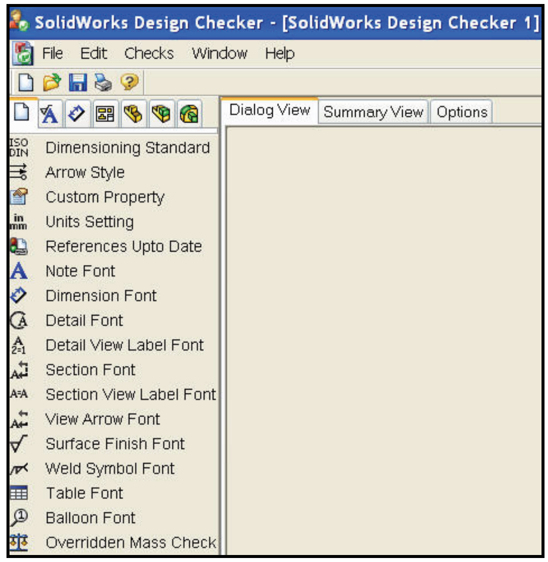


4 **Open the drawing.**
- Click **Window**, **Plate-associativity - Sheet1** from the Menu Bar toolbar. The updated dimension is displayed in the drawing.

5 **Return to Sheet1 of the Bar drawing.**
- Close the drawing. Click **File**, **Close** from the Menu Bar menu.
- Click **Yes to All**.
- Close the part. Click **File**, **Close** from the Menu Bar menu. Sheet1 of the Bar drawing is displayed.



SolidWorks Design Checker



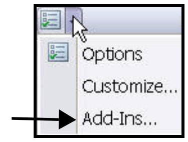
SolidWorks Design Checker is a timesaving tool for ensuring compliance with your organization's design standards. The SolidWorks Design Checker utilizes predefined design criteria and verifies accuracy, completeness, and standard compliance with design elements such as:

- Dimensioning standards
- Annotation and dimension fonts
- Title blocks
- Custom properties
- Layers
- Materials
- Overridden dimensions
- Sketches
- Assembly mates

You can set up and maintain compliance standards efficiently in a single file. For each design check, you can generate HTML-based reports to track results.

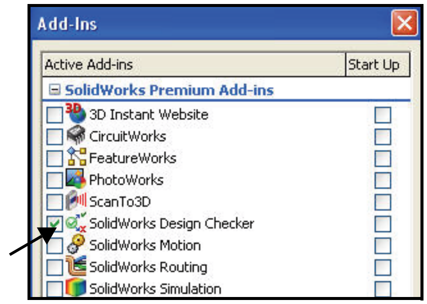
Note: SolidWorks Design Checker is available in SolidWorks Professional and Premium.

Utilizing Design Checker



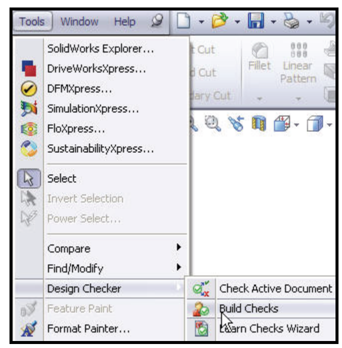
1 Activate Design Checker.

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

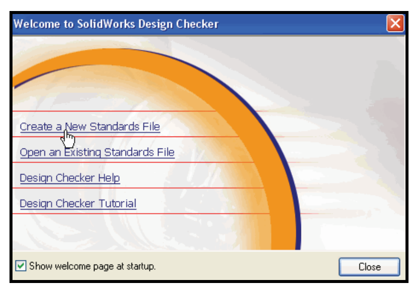


2 Select the Design Checker option.

- Click **Tools**, **Design Checker**, **Build Checks**  from the Menu bar menu. The Welcome to SolidWorks Design Checker dialog box is displayed. Note: If the show welcome page at startup is checked.

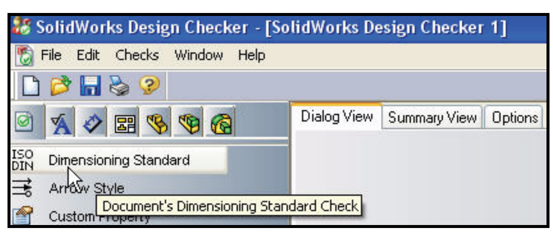


- Click **Create a New Standards File**. The SolidWorks Design Checker Graphics area is displayed.

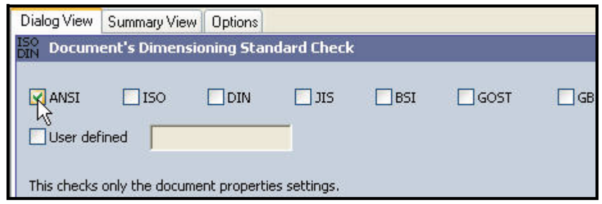


3 Select the Dimensioning Standard Checks.

- Click **Dimensioning Standard**. The Document's Dimensioning Standard Check boxes are displayed.



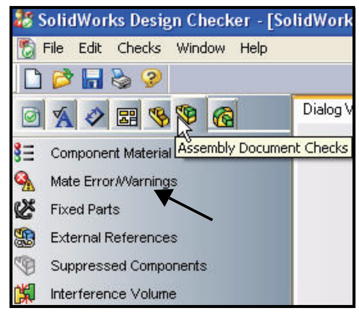
■ Check the **ANSI** box as illustrated.



4 Select the Assembly Mates Checks.

■ Click the **Assembly Documents Checks** tool.

■ Click **Mate Error/Warnings**. The second element to check is displayed in the right window.

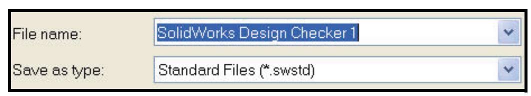


5 Save the Checks document.

■ Click **Save** from the Main menu.

■ Enter **SolidWorks Design Checker 1** for File name in the SolidWorks Test Drive folder.

■ Click **Save**.



6 Exit and return to the Bar drawing.

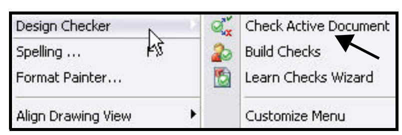
■ Click **File**, **Exit** from the Main menu. Note: The Bar drawing should be open at this time.

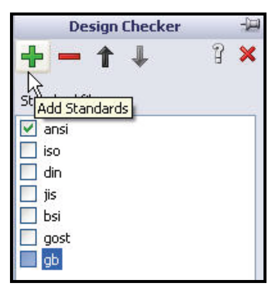
7 Check the Bar drawing.

■ Click **Tools**, **Design Checker**, **Check Active Document** tool. The Design Checker dialog box is displayed in the Task Pane.

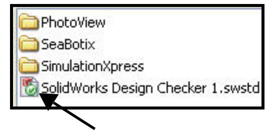
■ **Un-check** all boxes except ansi as illustrated.

■ Click the **Plus sign** in Design Checker as illustrated. Note the Design Checker tab in the Task Pane.

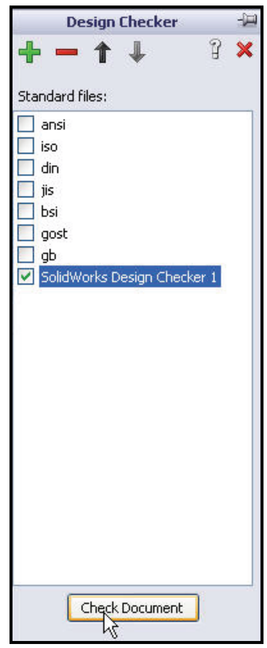




- Double-click **SolidWorks Design Checker 1** from the SolidWorks Test Drive folder. SolidWorks Design Checker 1 is displayed.



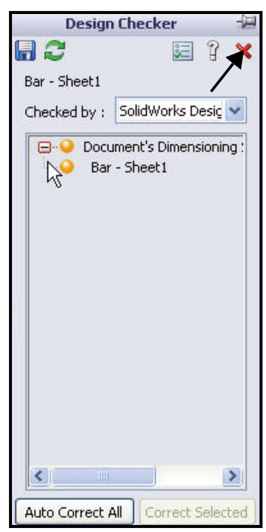
- **Un-check** the ansi box.
- Click the **check** box as illustrated.



- Click the **Check Document** button. View the results. There are no errors or discrepancies.

8 Close Design Checker.

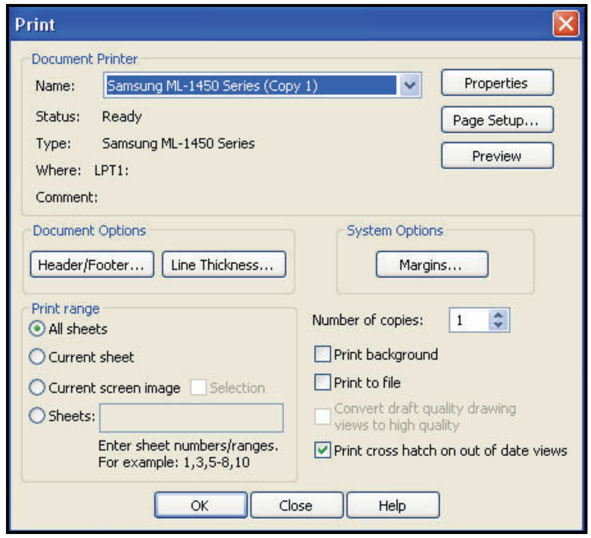
- **Close** the Design Checker dialog box.
- **Resize** the Task Pane to view Sheet1.

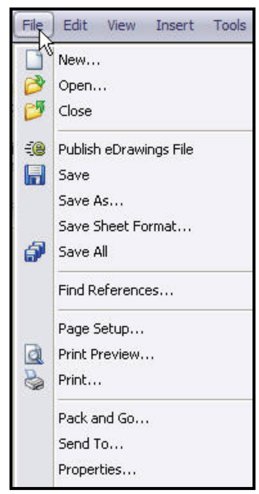


Printing

You can print or plot an entire drawing sheet, or just a selected area of the sheet. You have the option to print in black and white (the default) or in color. You can specify different settings for individual drawing sheets.

The Bar is an A (ANSI) Landscape size drawing. An A-size drawing can be printed on a desktop printer. Paper size is 8.5 x 11 inches.

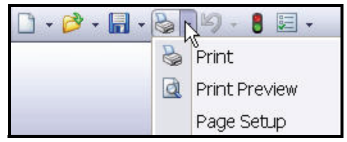


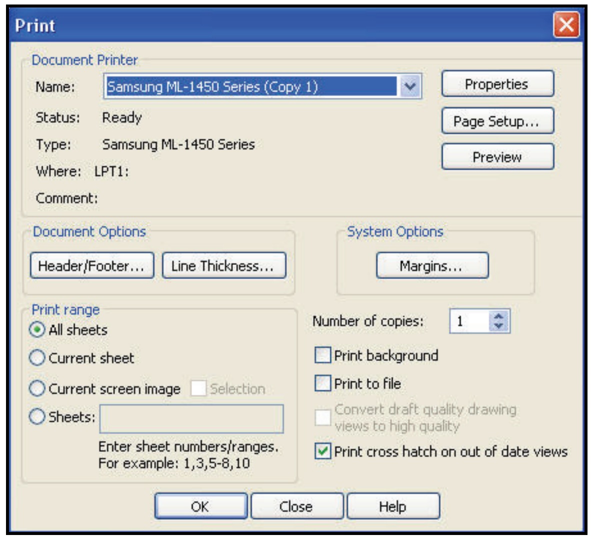


Printing a Drawing

1 Print a Drawing.

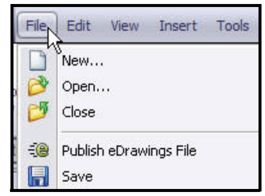
- There are no complicated printing commands. Click **Print** from the Menu Bar toolbar. The drawing will print to the printer associated with the system setup.





Note: As with other operations, you may be asked to rebuild or save the part, assembly, or drawing before you can print.

Sharing Information and Viewing SolidWorks eDrawings Files



For many, 2D drawings still play an important role in communicating information to customers, vendors, and suppliers. Have you ever sent a CAD drawing file to someone only to have them complain, “I can't open this file.” Have you had to communicate with someone who was using a different CAD system than you? Have you ever sent 2D drawings to someone only they didn't have a CAD system at all? It is also inefficient to plot the drawings, roll them up, and drop them in the mail.

SolidWorks eDrawings

SolidWorks eDrawings files are an email-enabled communications tool designed to dramatically improve sharing and interpreting 2D mechanical drawings. eDrawings files are small enough to email, are self-viewing, and significantly easier to understand than 2D drawings.

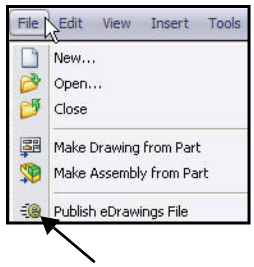
Viewing SolidWorks eDrawings

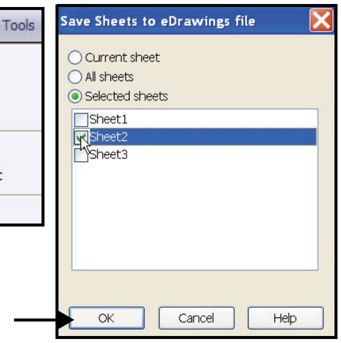
You can view SolidWorks eDrawings files in a very dynamic and interactive way. Unlike static 2D drawings, eDrawings files can be animated and viewed dynamically from all angles. The ability to interact with eDrawings files easily - in an interactive manner - makes it a very effective design collaboration tool. eDrawings Professional gives you the ability to perform markup and annotation of eDrawings files which further enhances design collaboration.

Advantages of SolidWorks eDrawings

- Recipients do not need to have the SolidWorks application to view the file.
- View parts, assemblies, and drawings outside of SolidWorks.
- Files are compact enough to email.
- Creating an eDrawings file is quick and simple.
- eDrawings display decals in SolidWorks parts and assemblies.
- PhotoWorks decals applied to SolidWorks models are visible in eDrawings
- Select the Publish eDrawings File  tool from the eDrawings toolbar or from the Menu bar menu to publish an eDrawings file for a SolidWorks document.
- You can create eDrawings files from other CAD systems too.

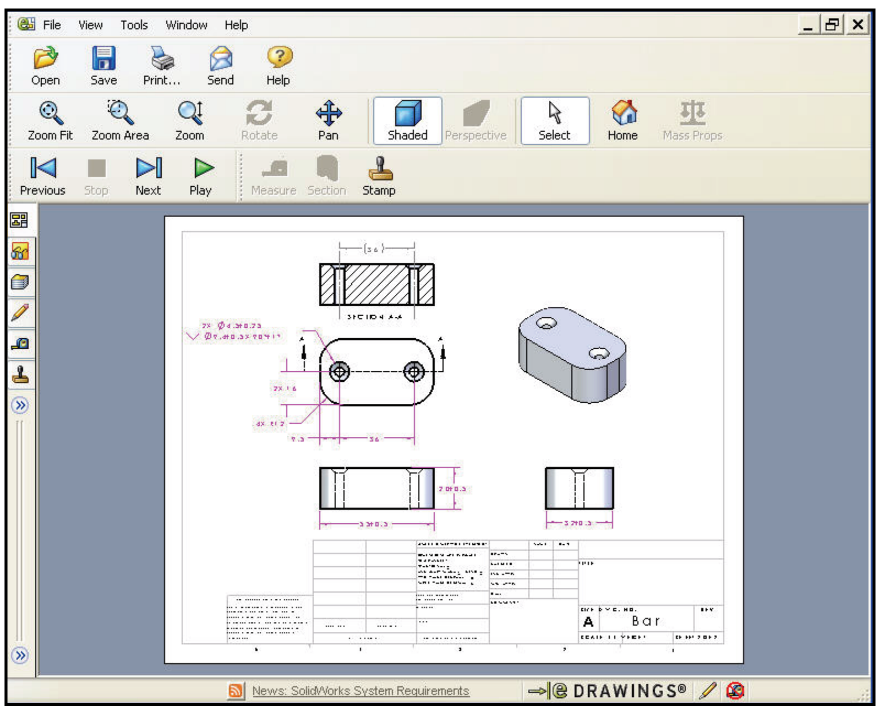
Creating and Viewing an eDrawings File





1 **Publish an eDrawings file.**
   - Click **File**, **Publish eDrawings File** from the Menu Bar menu. The Save Sheets to eDrawings file dialog box is displayed.
   - Click **Selected sheets**.
   - Click **Sheet2**.
   - Click **OK**.

2 **Review the eDrawings Viewer.**
   - **Review** the eDrawings.

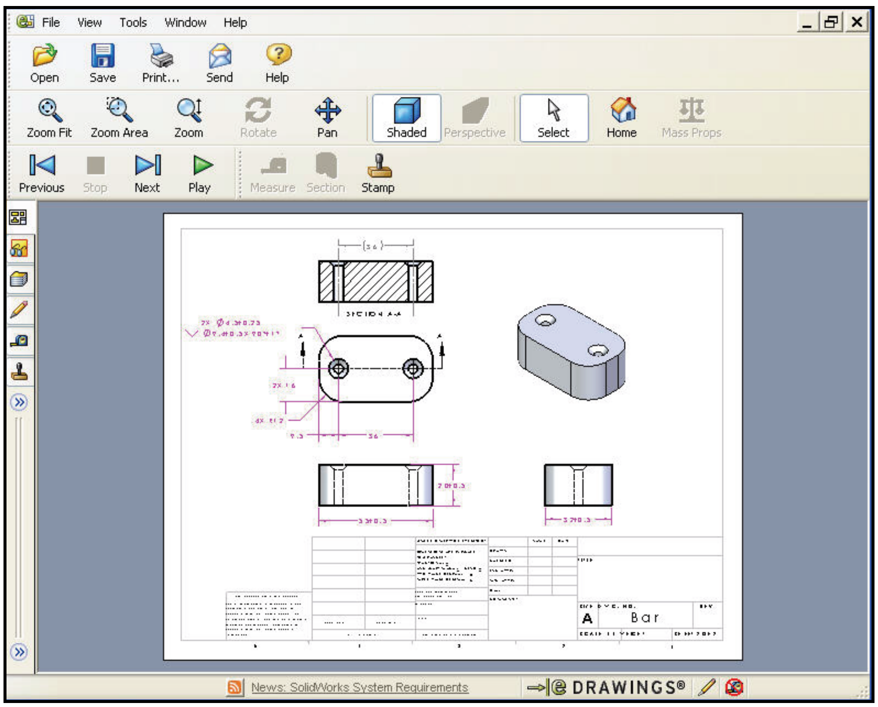


Note: Move Component is one of the features of SolidWorks eDrawings Professional.

Viewing SolidWorks eDrawings Animations

Animation automatically demonstrates how drawing views relate to each other and the physical design. With the click of a button, SolidWorks eDrawings "animate" all views contained in each sheet of your drawing, morphing from one view to another.

The animation continuously shows you the SolidWorks eDrawings file from different views. This dynamic motion is similar to turning a part around in your hand as you inspect it. In a multiple sheet SolidWorks eDrawings file, all sheets are reviewed.



Playing eDrawings Animation

1 Play the animation.

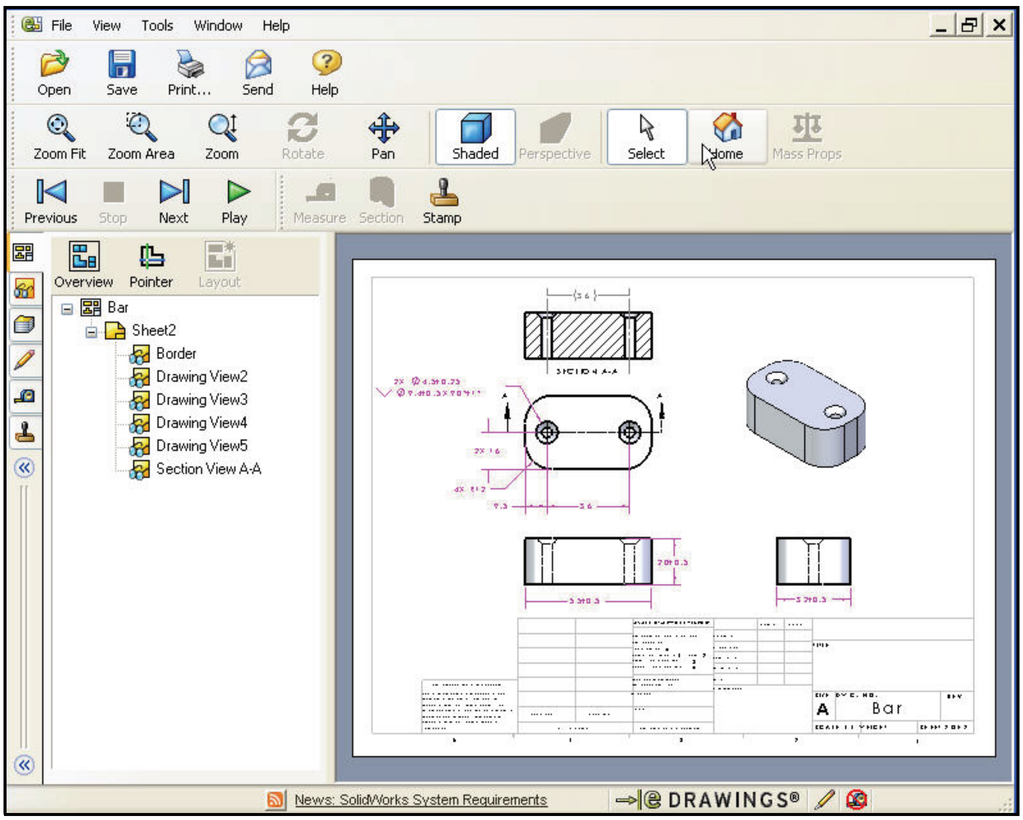
- Click **Play** . The SolidWorks eDrawings file is displayed showing you each view. This display continues until you stop it.

2 Stop the animation.

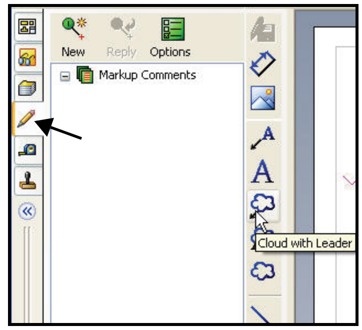
- Click **Stop** .

3 Reset the view.

- Click **Home** , to return the SolidWorks eDrawings file view to its starting position.



Communicating with SolidWorks eDrawings File Markup Tools

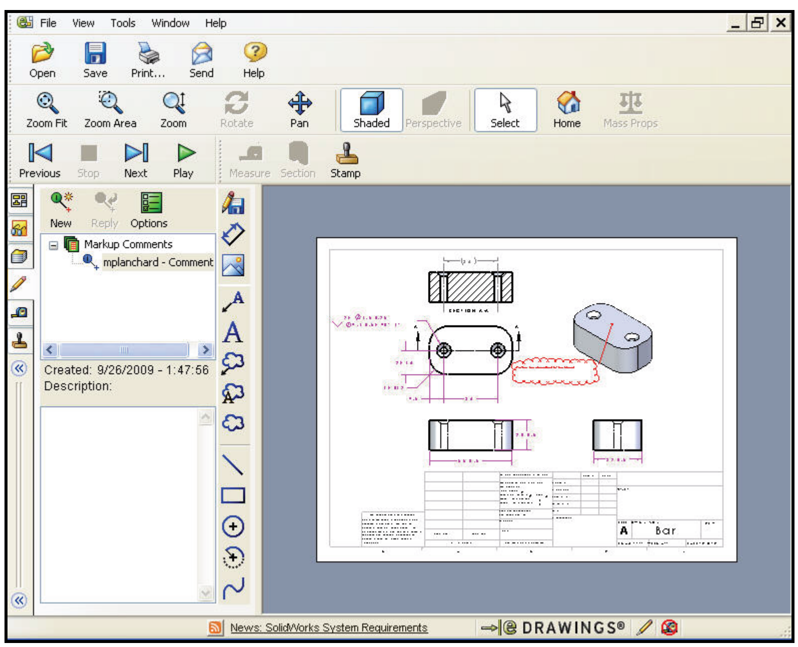


Engineers, designers, vendors, and purchasing agents utilize drawings to communicate. You can create, view, and share your 3D models and 2D drawings with eDrawings.

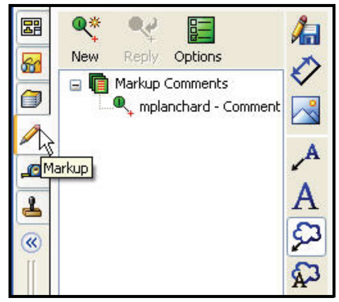
The SolidWorks eDrawings Professional Markup tools provide elements such as geometry, clouds, text, and dimensions. The SolidWorks eDrawings Markup tools support:

- Tablet PC's support markup of SolidWorks eDrawings files using the pen.
- Markup comments displayed as a threaded discussion.
- Addition of long descriptions in the Description box that do not appear on the drawing.
- Display of unread comments in bold in the threaded discussion.
- Ability to edit only your own comments.

Note: Requires SolidWorks Professional or SolidWorks Premium.

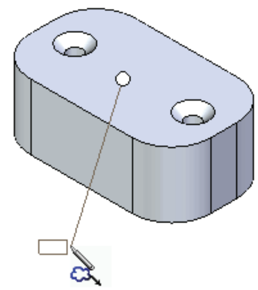


Marking up a SolidWorks eDrawings File

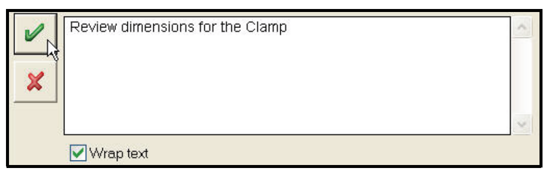


1 Apply the Markup tool.

- Click the **Markup** tool from the left Task Pane toolbar as illustrated. The Markup toolbar is displayed.
- Click the **Cloud with Leader** tool from the Markup toolbar.



- Click the top of the **Clamp** in the Isometric view to locate the arrow of the leader. The Comment text box is displayed.
- Click a **position** below the Clamp to locate the text.

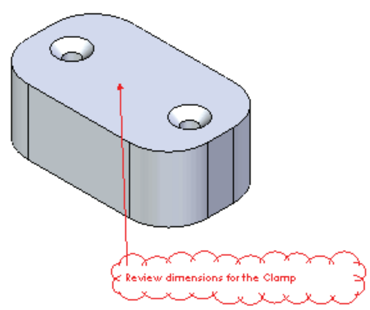


2 Enter a comment.

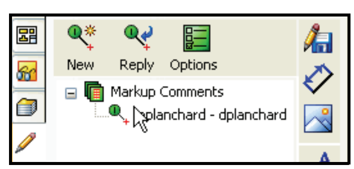
- Enter the text, **Review dimensions for the Clamp**.
- Click the **green checkmark** button.

3 View the comment.

- **View** your comment in the e-Drawing.



Note: In the design process, the eDrawings file would be reviewed by another team member. By clicking on the red cloud you can Reply, Accept, or Reject the comment. The threaded discussion builds in the Markup Comments dialog box.



Saving SolidWorks eDrawings Files

You can save SolidWorks eDrawings files in one of several formats. You can save SolidWorks eDrawings as:

SolidWorks eDrawings — Part, assembly, and drawing SolidWorks eDrawings files that can be viewed by systems that already have SolidWorks eDrawings.

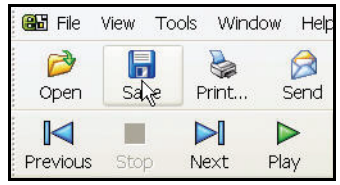
HTML — SolidWorks eDrawings files that are viewed using browsers that support HTML.

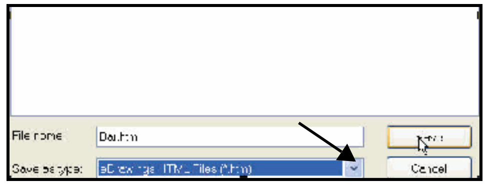
Executable — Self-contained executable file that contains the SolidWorks eDrawings Viewer embedded in the file. These files have .exe as a file name suffix.

Zip — A compressed SolidWorks eDrawings file. These files have .zip as a file name suffix.

Image files — Utilized as pictures. These files have .bmp, .tiff, or .jpg as a file name suffix.

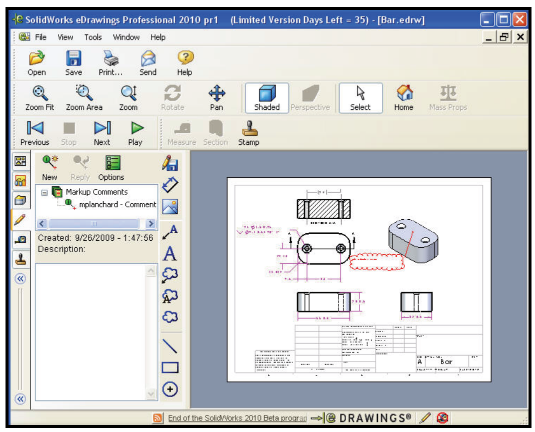
Saving a SolidWorks eDrawings File



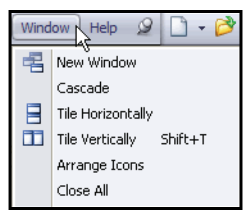


1 **Save the eDrawings file.**
   - Click **Save** from the Main menu.
   - Select the **SolidWorks Test Drive** folder.
   - For Save as type: select **eDrawings HTML Files (*.htm)**. The file can be viewed in a web browser. Accept the default name Bar.
   - Click **Save**.

2 **Close the eDrawings dialog box and return to the Bar drawing Sheet1.**
   - **Close**  the eDrawings dialog box.

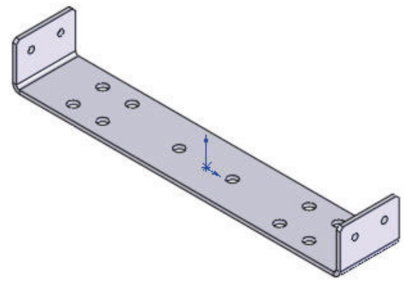


3 **Close all open models.**
   - Click **Window**, **Close All** from the Menu bar menu.



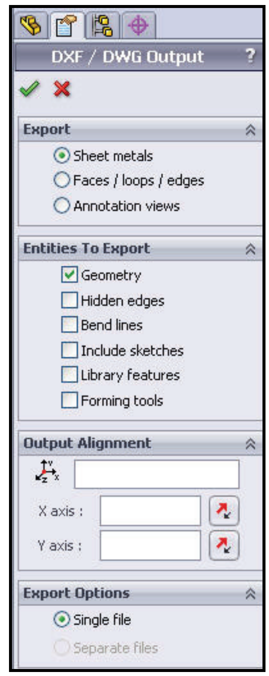
Note: New in SolidWorks 2010 is a print preview option for your eDrawing.

Applying DXF Export to Sheet Metal



The Bent Bar1 part that supports the Clamp and MiniGrab was designed as a sheet metal part. Sheet metal parts contain special properties that represent both a 3D state and a 2D state.

Sheet metal parts are designed in the 3D formed state to be utilized in an assembly and manufactured in the 2D flat state. A sheet metal drawing requires the flat state.



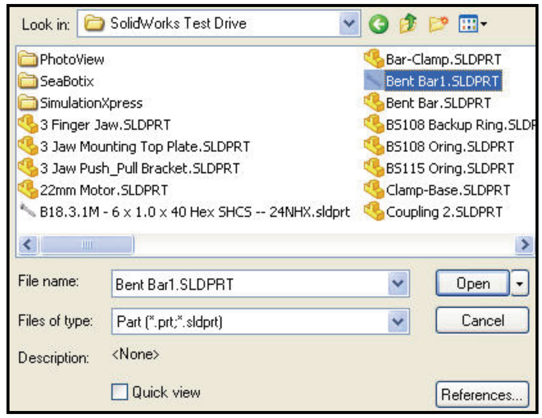
SolidWorks DXF PropertyManager exports sheet metal to a .dxf file. A preview shows what you are exporting and lets you move unwanted entities such as holes, cutouts and bend lines. The scale is 1:1 by default. Also SolidWorks automatically cleans up overlapping entities so a cleaner file is delivered to manufacturing.

Note: This new feature will dramatically speed up creation of .dxf for manufacturing operations such as laser, torch and water jet.

Exporting a Sheet Metal part to DXF

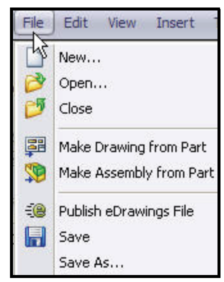
1 Open a Sheet Metal part.

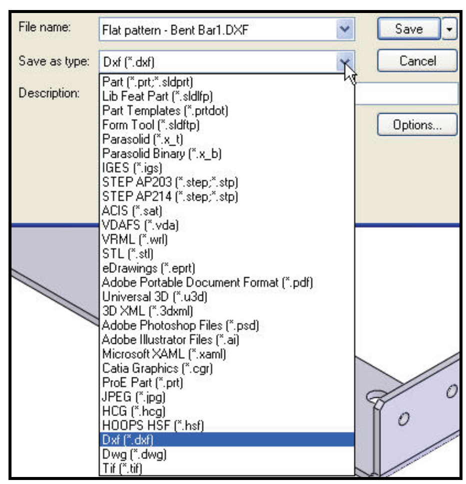
- Click **Bent Bar 1** from the SolidWorks Test Drive folder.

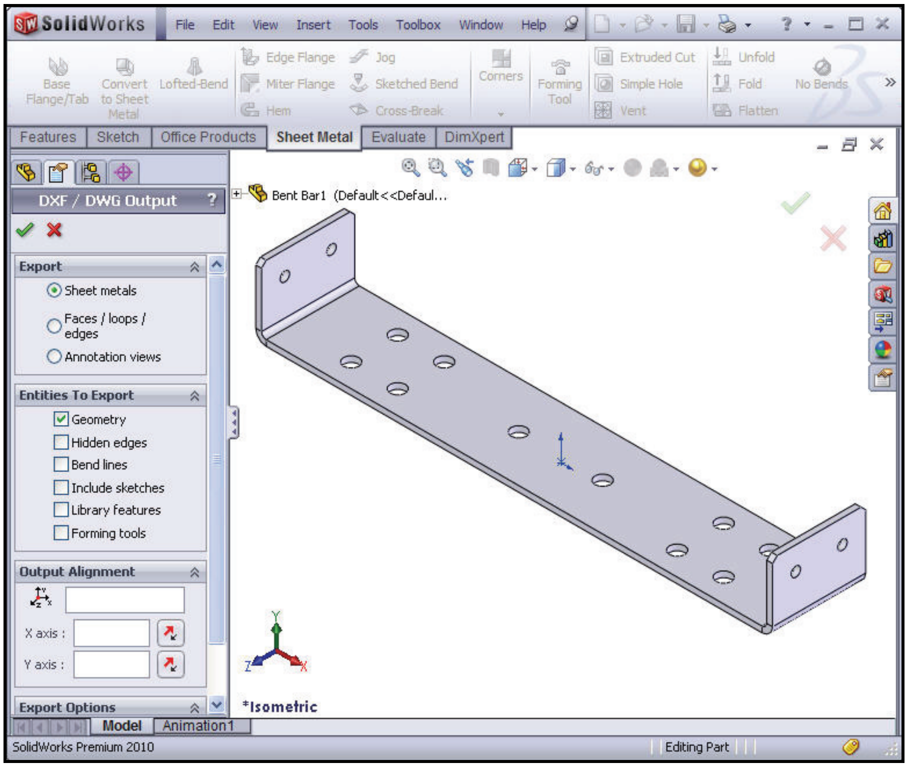


2 Export the Part.

- Click **File**, **Save As** from the Menu bar toolbar.
- Select **Dxf (*.dwf)** for Save as type.
- Click **Save** from the Save As dialog box. View the Output PropertyManager.







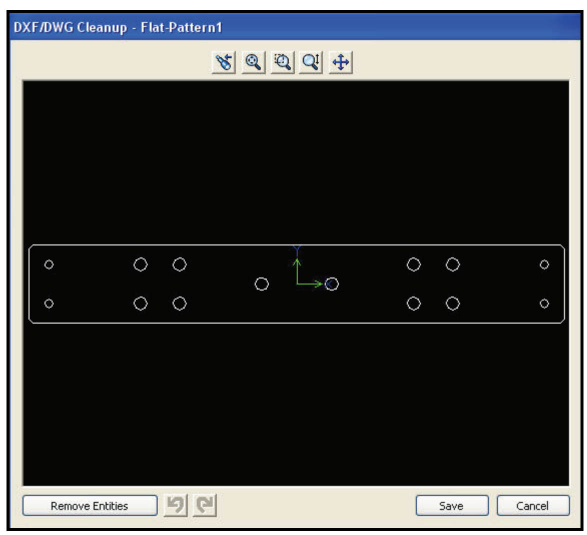
- Accept the default settings. Click **OK** from the Output PropertyManager. View the results.

3 Save the Sheet Metal Part.

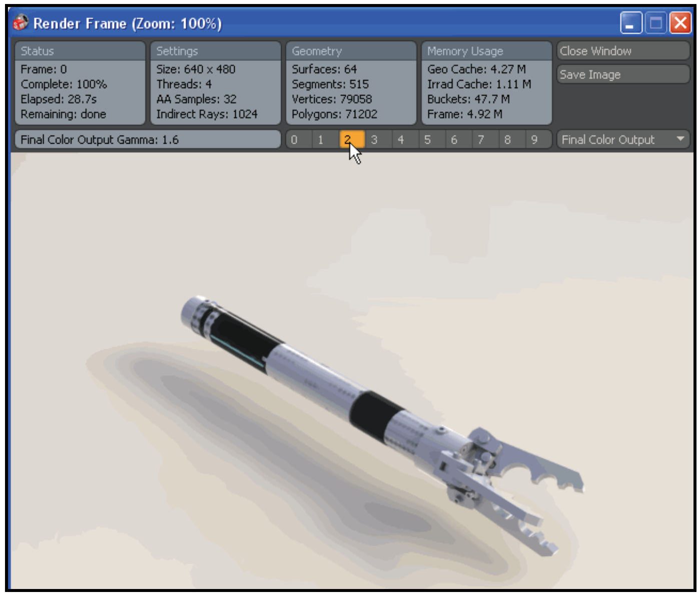
- Click **Save**.

4 Close the Part.

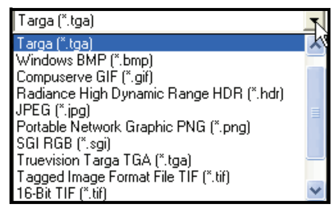
Click **File**, **Close** from the Menu bar menu.



SolidWorks PhotoView 360



SolidWorks PhotoView 360 is a new best-in-class rendering application for creating photorealistic images from 3D CAD models with a simple click and drag approach. Help your customers and colleagues visualize your designs more easily.



Save images to a file type in a variety of formats for printed marketing materials and Web pages.

The target users for SolidWorks PhotoView 360 are designers who require photorealistic images without spending a lot of time with numerous settings.

PhotoView 360 uses High Dynamic Range Imaging (HDRI). HDRI is a set of techniques that allows for a greater dynamic range of exposures.

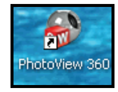
With PhotoView 360, you can begin marketing product concepts early in the development cycle and get to market faster.

Utilize the Bar2 assembly in the SolidWorks Test Drive\PhotoView folder.

Note: SolidWorks PhotoView 360 is required for the next section.

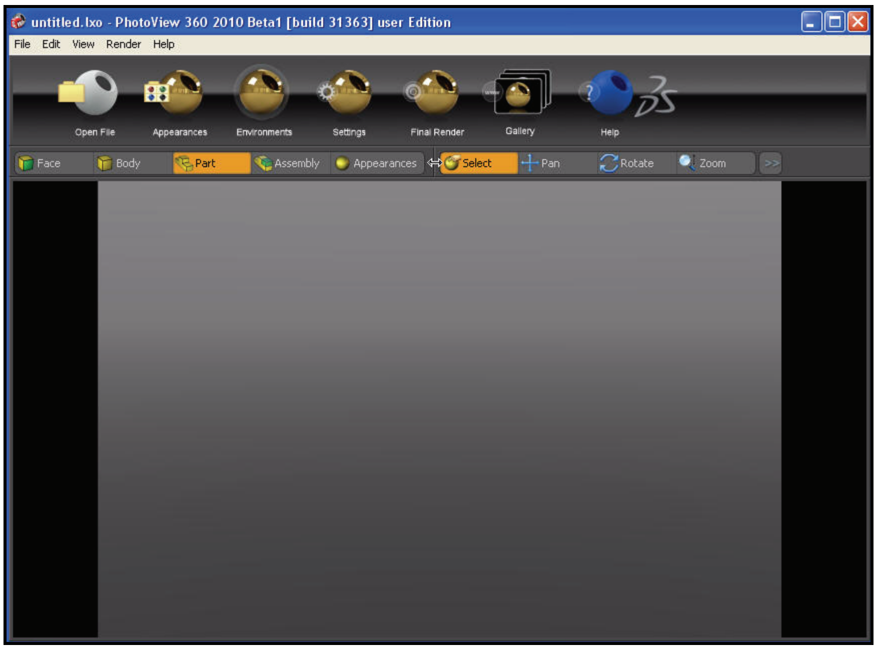
Note: RealView Graphics are carried over to PhotoView 360.

Creating a SolidWorks PhotoView 360 Image



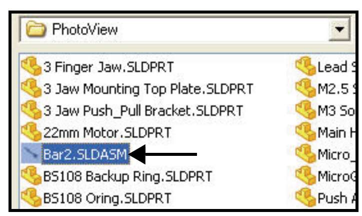
1 Activate SolidWorks PhotoView 360.

- **Minimize** the SolidWorks Graphics area if needed. SolidWorks is not required to run PhotoView 360.
- Click the **PhotoView** icon in the MS Window. The SolidWorks PhotoView 360 dialog box is displayed.



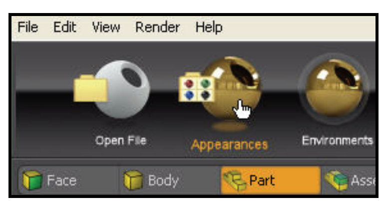
2 Open the Bar2 assembly.

- Click the **Open File** icon from the Main menu.
- Select the **SolidWorks Test Drive\PhotoView** folder.
- Double-click **Bar2**. View the Bar2 assembly in the Graphics area.

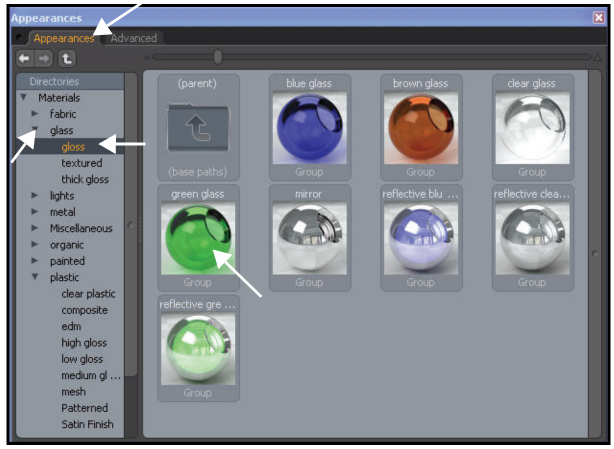


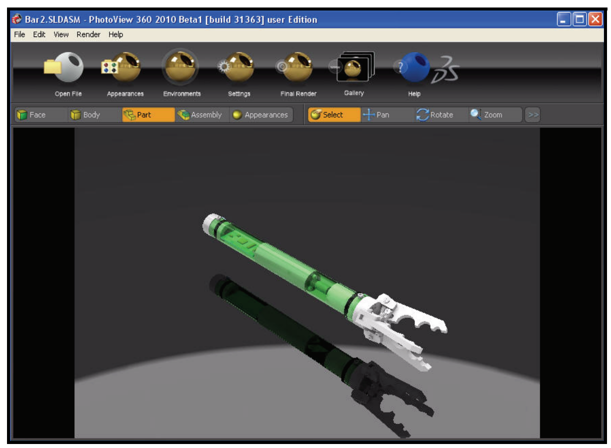
3 Apply Appearance the handle.

- Click the **Appearances** icon from the Main menu. The Appearances dialog box is displayed.



- Click the **Appearances** tab.
- Click the **arrow** next to glass.
- Click **gloss** as illustrated.
- Drag and drop **green glass** onto the handle of the Bar2 assembly. View the results in the Graphics area. PhotoView 360 provides the ability to apply appearance to a body, face, part or an assembly.



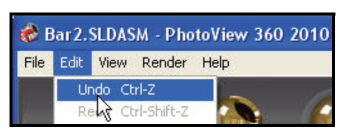


Note: The slider bar in the Appearances dialog box provides the ability to resize the available appearance display.

Note: The Advanced tab provides the ability to customize your selected appearance display.

4 **Undo the Appearance.**

- Click **Edit**, **Undo** from the Main menu. View the results.



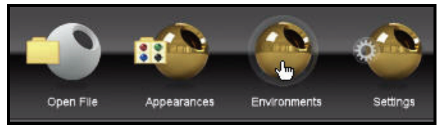
5 **Apply Clear thick glass Material to the Handle.**

- Click **thick gloss** under the glass column as illustrated.
- Drag and drop **clear thick glass** onto the handle of the Bar2 assembly. View the results in the Graphics area.

- **Close** the Appearances dialog box.

6 **Apply an Environment to the model.**

- Click the **Environments** icon from the Main menu. The Environments dialog box is displayed.
- Drag and drop the **abstract studio shadow** environment into the Graphics area.
- **Close** the Environments dialog box. View the results.

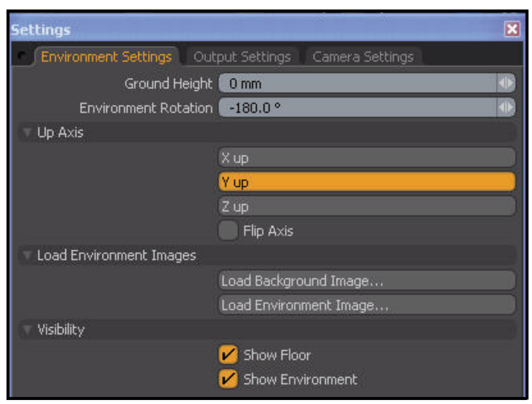


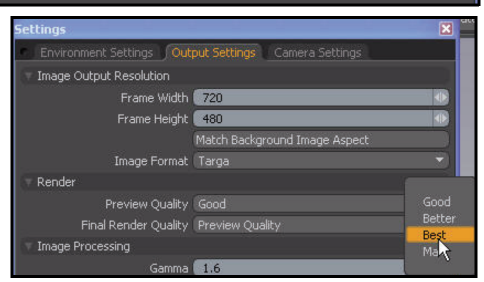
7 **Set Settings properties.**

- Click the **Settings** icon from the Main menu. The Settings dialog box is displayed. View your options. There are three tabs: *Environment Settings*, *Output Settings*, and *Camera Settings*.
- Click the **Output Settings** tab. View your options.

8 **Set Render Preview Quality.**

- Click the **Best** button for Render Preview Quality. View the render results in the Graphics area.
- **Close** the Settings dialog box.



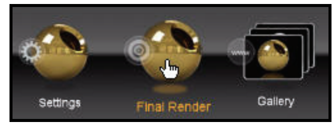


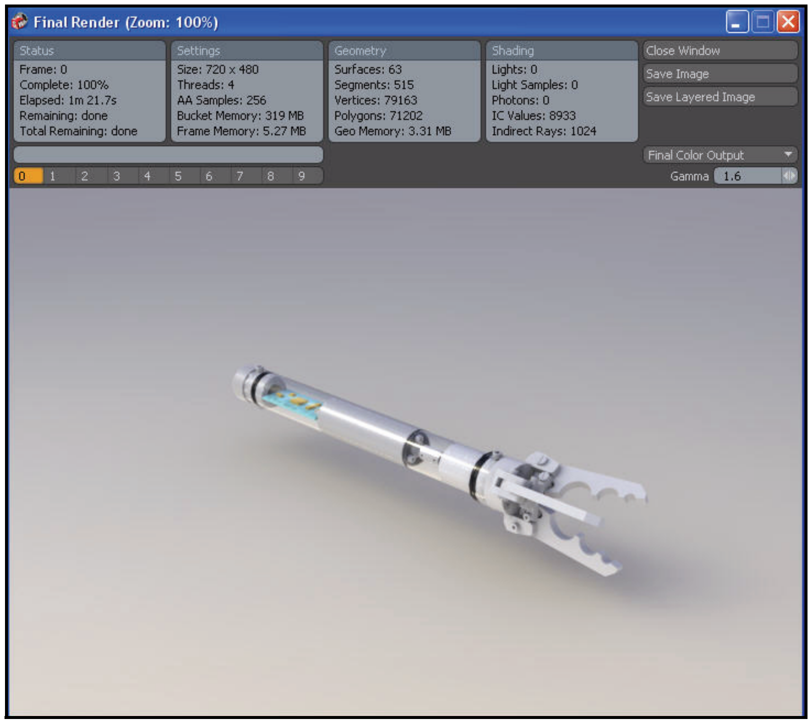
Note: The Render Quality option effects the shadows and transparencies of the model.

Note: In 2010, the Camera tool supports depth-of-field control and a new bloom effect.

9 Render the model.

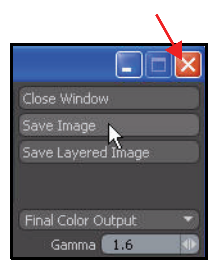
- Click the **Final Render** icon from the Main menu. The Render Frame dialog box is displayed.





Note: PhotoView 360 saves the last ten renders of the model.

Tip: Click the **Save Image** button, and select file format type to save the rendered model.



10 Return to the PhotoView 360 Graphics area.

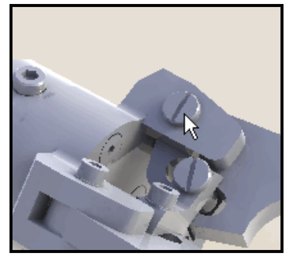
- Click the **Close Window** button from the Render Frame dialog box. View the Graphics area.

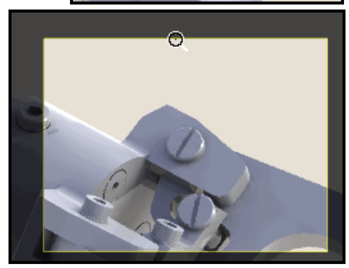
11 Zoom in on a Screw in the Claw.

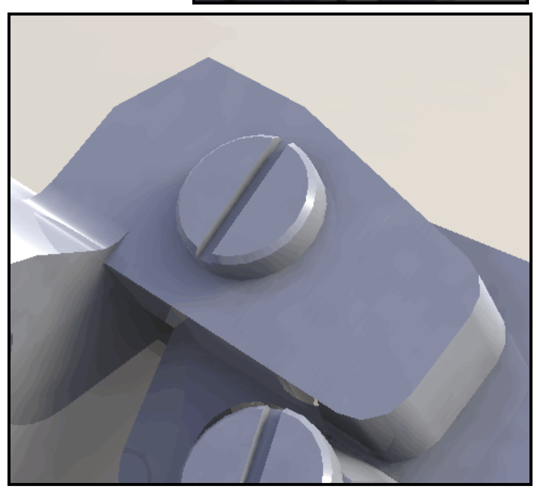
- Rotate the **middle mouse button** to increase the model size in the Graphics area.
- Click the **Zoom Window** button as illustrated.
- Place the **Magnifying glass** icon over the screw on a Claw.
- Click and drag the **Magnifying glass** icon upwards.
- **Release** the mouse button. View the results.

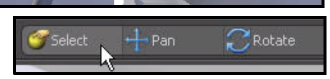
12 Deselect the Box Zoom tool.

- Click the **Select** tool as illustrated.



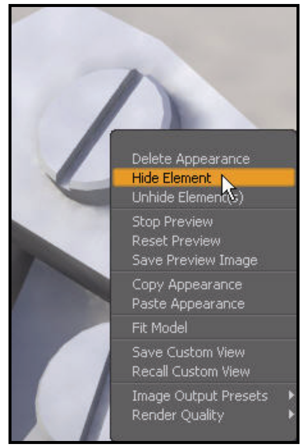






13 Hide a Face.

- Click the **Face** button.
- Click the **face** of the part as illustrated.
- Right-click **Hide Element**. View the results.



14 Display the Hidden Face.

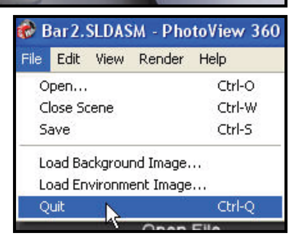
- Right-click **Unhide Elements(s)**. View the results.

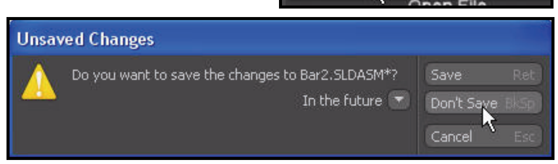
15 Fit the model to the Graphics area.

- Click the **Fit to View** button. View the results.

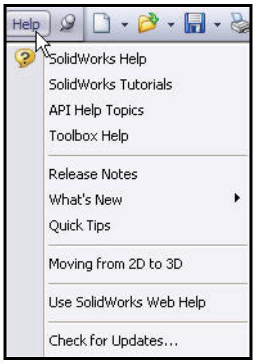
16 Close the model and PhotoView 360 session.

- Click **File**, **Quit** from the Main menu.
- Click the **Don't Save.** You are now finished with this section. You have experienced the power and ease of SolidWorks PhotoView 360.





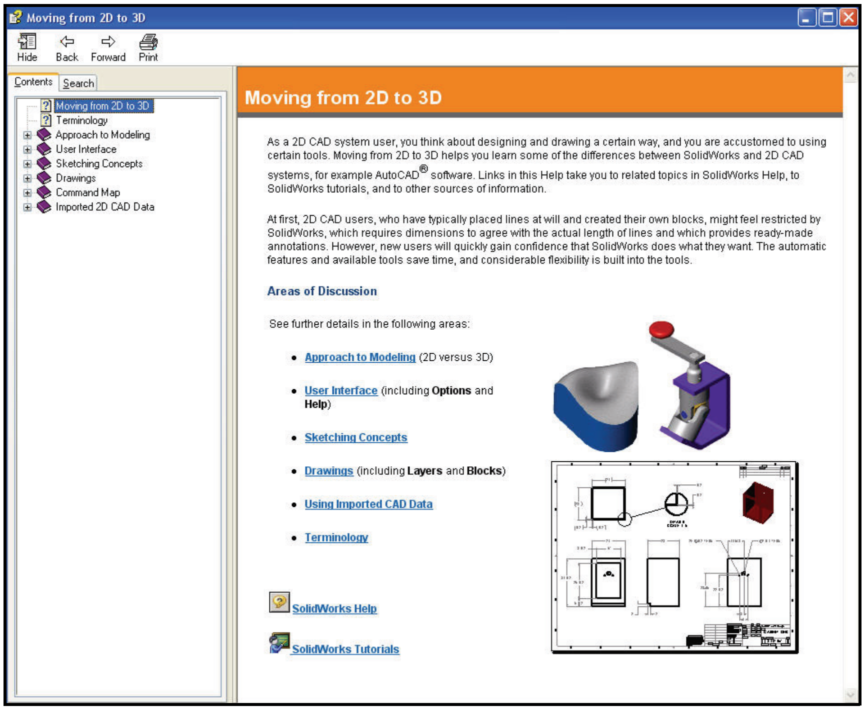
Moving from AutoCAD



SolidWorks provides a section under Help in the Menu bar for users moving from AutoCAD. This section helps you transition from 2D AutoCAD to 3D SolidWorks.

It compares terms and concepts, explains SolidWorks approaches to design, and provides links into SolidWorks help, tutorials, and other resources.

SolidWorks supplies many tools that are not available in AutoCAD. New users will quickly gain confidence that SolidWorks does what they want. The automatic features and available tools save time, and considerable flexibility is built into the available tools.



SolidWorks Hands-on Test Drive Conclusion

During this short hands-on session, you have had a chance to explore some of the capabilities of SolidWorks. Although you have only scratched the surface of what SolidWorks can do, you have seen first hand how:

- Ease of use shortens the learning curve.
- Design communication tools demonstrate more effectively how products will look and perform.
- Powerful modeling capabilities allow you to fully define your product and increase your productivity.
- 100% editability maximizes design considerations.
- Fully associative documents keep parts, assemblies, and drawings up-to-date and in sync.

SolidWorks is the standard in 3D modeling because it is powerful, easy to use, and affordable.

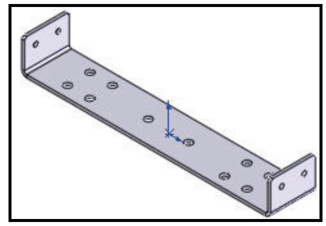
Notes

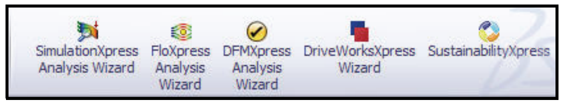
SolidWorks SimulationXpress

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

- The benefits of analysis.
- The ease of use of SolidWorks® SimulationXpress to perform analysis on your design.
- The steps for performing up front analysis on your designs.
- How SolidWorks integrates with analysis.
- Cost savings by reducing unnecessary material.
- The ability to document your analysis findings automatically.
- The knowledge to optimize and update your model based on the analysis results.

Stress Analysis of the Bent Bar Part





In this section, apply SimulationXpress to quickly analyze the Bent Bar part which is used in the Bar assembly. Performing analysis is very quick and easy to do. There are only five steps required:

1. Apply fixtures.
2. Apply loads.
3. Define material.
4. Run the analyze.
5. View the results.
6. Optimize the part, (Optional).

After performing a first-pass analysis on the Bent Bar part and assessing its safety, you will change the design to reduce its weight and re-run the analysis. When you are sure that the final design is safe, you will review the results in the SeaBotix LBV150.

Design Analysis

After building your design in SolidWorks, you may need to answer questions like:

- Will the part break?
- How will it deform?
- Can I use less material without affecting performance?

In the absence of analysis tools, expensive prototype-test design cycles take place to ensure that the product's performance meets customer expectations. Design analysis makes it possible to perform design cycles quickly and inexpensively on computer models instead of testing costly physical prototypes. Design analysis also facilitates studies of many design options and aids in developing optimized designs.

Stress Analysis

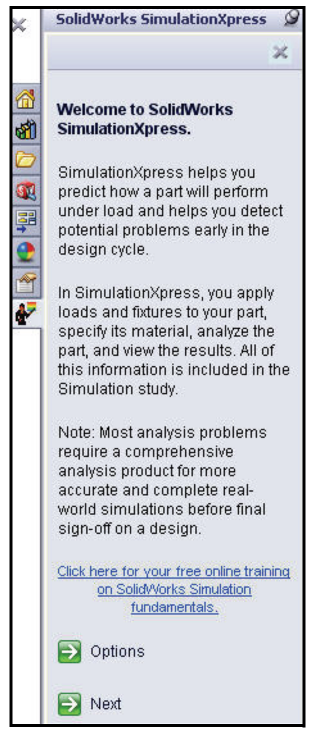
Stress analysis or static analysis is the most common design analysis test. It predicts how the model deforms under loading. It calculates displacements, strains, and stresses throughout the part based on material, restraints, and loads. A material fails when the stress reaches a certain level. Different materials fail at different stress levels. SolidWorks® SimulationXpress uses linear static analysis, based on the Finite Element Method (FEM), to calculate stresses.

Linear static analysis makes the following assumptions to calculate stresses in the part:

- **Linearity Assumption**. Means that the induced response is directly proportional to the applied loads.
- **Elasticity Assumption**. Indicates that the part returns to its original shape if the loads are removed.
- **Static Assumption**. Implies that loads are applied slowly and gradually until they reach their full magnitudes.

User Interface

SolidWorks SimulationXpress Wizard guides you through various steps to define material properties, restraints, loads, to analyze the part, optimize the part, and to view your results. The SolidWorks SimulationXpress Wizard consists of the following components:



Welcome Menu: Allows you to set the default units, specify a folder for saving the analysis results, and to Start a new analysis.

Fixture Menu: Apply Fixtures / restraints to faces of the part.

Loads Menu: Apply forces or pressures to faces of the part.

Material Menu: Applies material properties to the part. The material can be assigned from the material library or you can input the material properties.

Run Menu: Runs the analysis. You can select to analyze with the default settings or modify the settings.

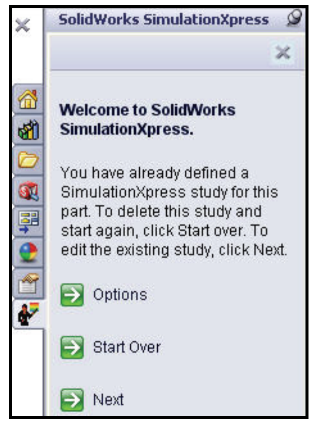
Results Menu: View analysis results in the following ways:

- Show critical areas where the factor of safety is less than a specified value.
- Display the stress distribution in the model with or without annotation for the maximum and minimum stress values.

- Display resultant displacement distribution in the model with or without annotation for the maximum and minimum displacement values.
- Show deformed shape of the model.
- Generate an HTML report.
- Generate eDrawings files for the analysis results.

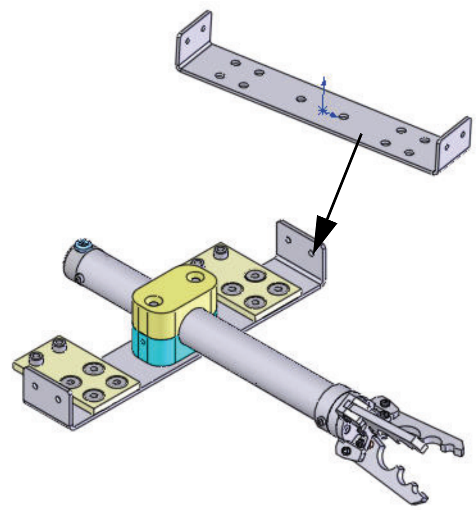
Optimize Menu: Optimize a model dimension based on a specified criterion.

Start Over button: Click this button to delete existing analysis data and results and start a new analysis session.

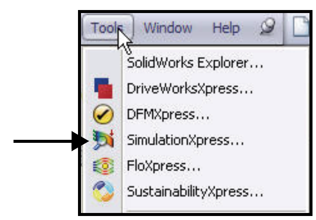


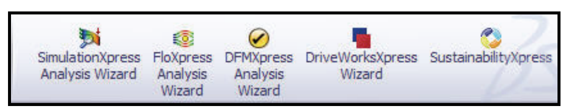
Let's Analyze the Bent Bar Part

You will perform stress analysis on the Bent Bar part. The Bent Bar part is used in the Bar assembly.

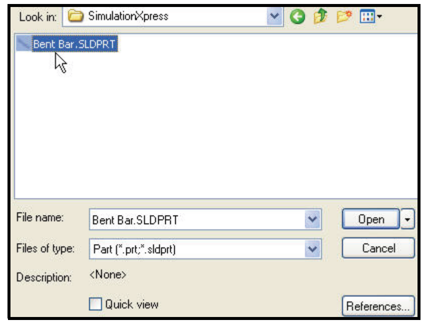


Note: New for 2010, SolidWorks® SimulationXpress provides a new redesigned user interface which eases the new user into the world of simulation. Use of the Task Pane and PropertyManager improves both the workflow and model interaction.





Opening the Bent Bar Part

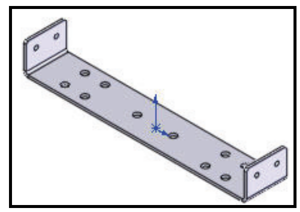


1 Open the Bent Bar part.

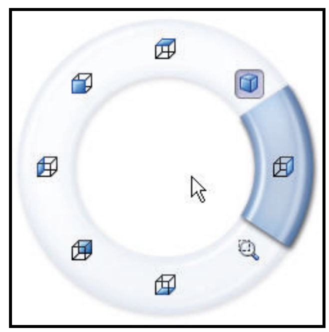
- Click **Open**  from the Menu Bar toolbar.
- **Select** the SolidWorks Test Drive\SimulationXpress folder.
- Set Files of type: to **Part**.
- Double-click **Bent Bar**. The Bent Bar is displayed in the Graphics area.

2 Change the view orientation.

- If the part is not displayed in an Isometric view, click the **Isometric** view  tool from the Heads-up View toolbar.



Note: New in 2010 is the Graphics area display wheel. Right-click with the right mouse button and drag in the Graphics area to display the wheel. The wheel displays eight possible commands. The user can customize the wheel for sketching, part, assembles, and drawings.



Running SolidWorks SimulationXpress and Setting Analysis Options

Once the part is open in SolidWorks, you can launch the SimulationXpress application and start your analysis right away. On the Options dialog box, you can set the default system of units and the destination folder for the analysis results.

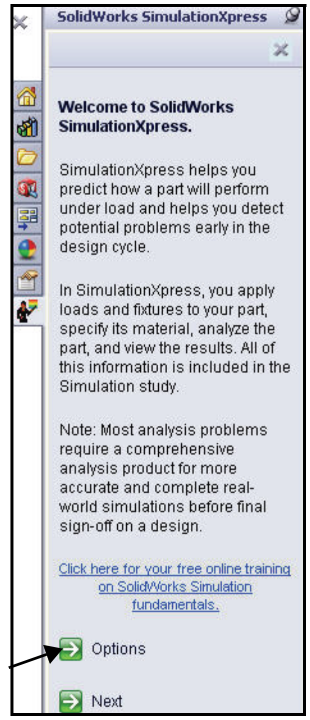
Systems of Units

The following table lists the quantities used by SolidWorks® SimulationXpress and their units in different systems of units:

		SI	English (IPS)	Metric
Loads	Force	N (Newton)	lb (pound)	Kgf
	Pressure	N/m^2	psi (lb/in^2)	Kgf/cm^2
Material Properties	Ex: Elastic modulus	N/m^2	psi (lb/in^2)	Kgf/cm^2
	NUXY: Poisson’s ratio	No units	No units	No units
	SIGYLD: Yield Strength	N/m^2	psi (lb/in^2)	Kgf/cm^2
	DENS: Mass density	Kg/m^3	lb/in^3	Kg/cm^3
Results	Equivalent Stress	N/m^2	psi (lb/in^2)	Kgf/cm^2

Table 1: Systems of units used in SolidWorks SimulationXpress

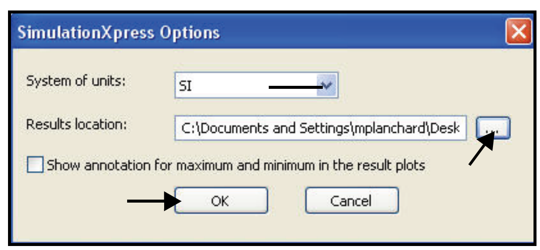
Running SolidWorks SimulationXpress and Setting Analysis Options



1 **Run SolidWorks SimulationXpress.**

- Click the **SimulationXpress Analysis Wizard** tool for the Evaluate tab in the CommandManager. The SolidWorks SimulationXpress application starts with the Welcome screen.

2 **Set document Units.**



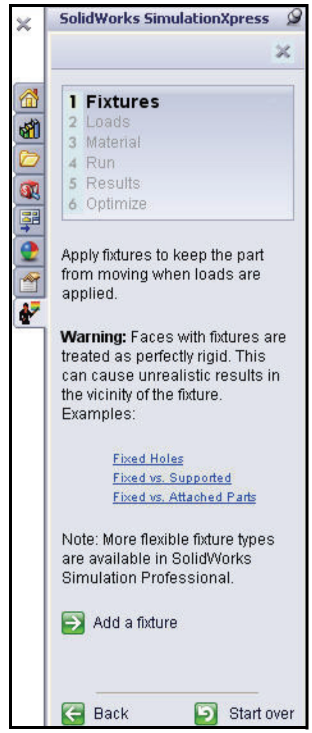
- Click the **Options** button from the Welcome screen.
- Set System of units to **SI, (MMGS)**.
- Set the Results location to the **SolidWorks Test Drive\SimulationXpress** folder.
- Click **OK**.
- Click **Next**.

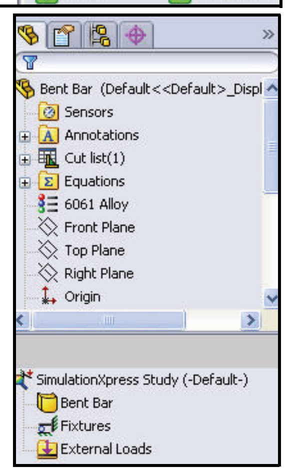
Applying Fixtures

A part that is not fixed will travel indefinitely in the direction of the applied load as a rigid body. In the Fixture section, you define how the Bent Bar part is fastened to the SeaBotix LBV150.

The restrained faces are fixed in space. You must restrain one face of the part to prevent the analysis program from stopping, due to instability caused by rigid body motion.

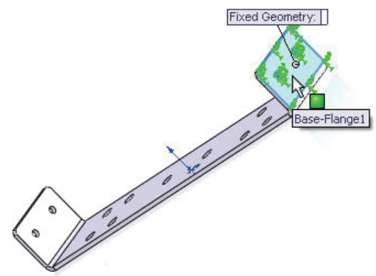
Note: Explore the different Fixed options: *Fixed Holes*, *Fixed vs. Supported* and *Fixed vs. Attached Parts*.



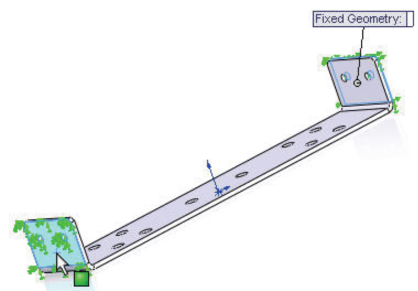


Applying a Fixture

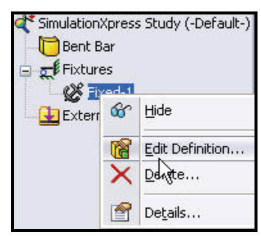
1. **Apply a fixture.**
   - Click **Add a fixture**. The Fixture PropertyManager is displayed. The Fixture section collects information on where the Bent Bar is fixed. You can specify multiple sets of fixtures. Each set can have multiple faces.



2. **Select the fixed faces.**
   - Click the **outside right** and **outside left** planar faces of the Bent Bar as illustrated. Face <1> and Face<2> is displayed in the selection box.
3. **Apply the fixture.**
   - Click **OK** from the Fixture PropertyManager. Fixed-1 is displayed.



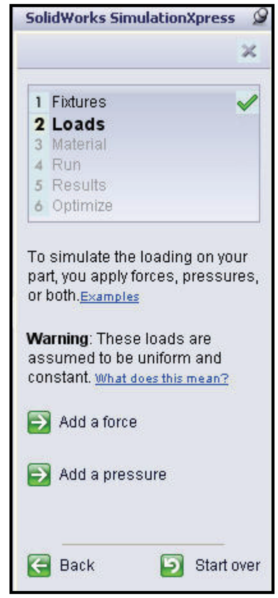
Note: To edit Fixed-1, right-click Fixed-1, click Edit Definition. The Fixture PropertyManager is displayed.

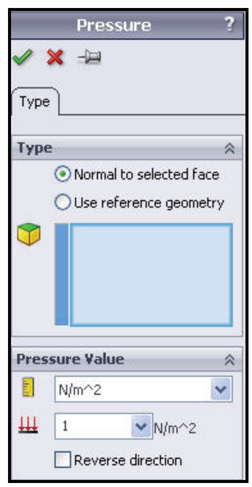


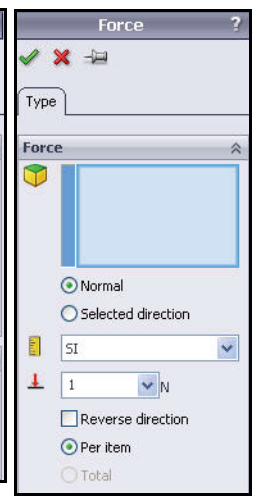
Applying Loads

The Loads option provides the ability to specify the loads acting on the part. A load can either be a force or a pressure.

You can apply multiple loads to a single face or to multiple faces. The direction of a force can be specified with respect to planes or normal to selected faces. The pressure is always applied normal to selected faces.

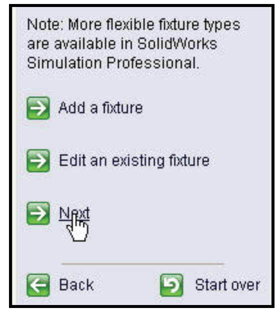




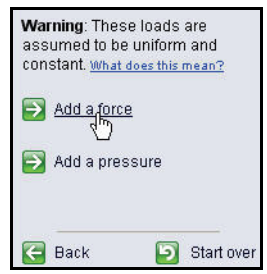


Applying a Load

1 **Apply a load.**
- Click **Next**. Collect information on loads acting on the Bent Bar part. You can specify multiple sets of forces or pressures. Each set can have multiple faces.

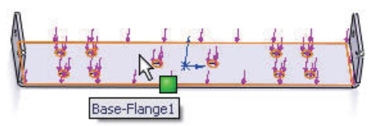


2 **Select a load type.**
- Click **Add a force**. The Force PropertyManager is displayed. Normal is selected by default.

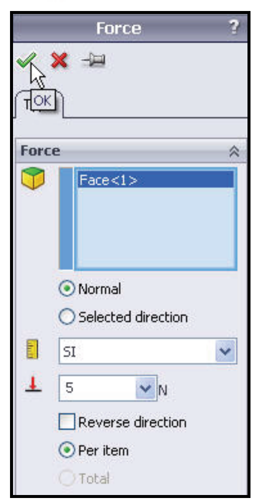


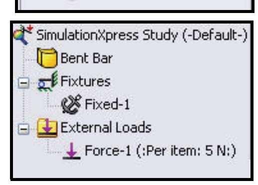
3 **Enter the force.**
- Enter **5N** for Force Value.

4 **Select the face to which the force is applied.**
- Click the **top face** of the Bent Bar as illustrated. Face <1> is displayed in the selection box. The force points downwards.



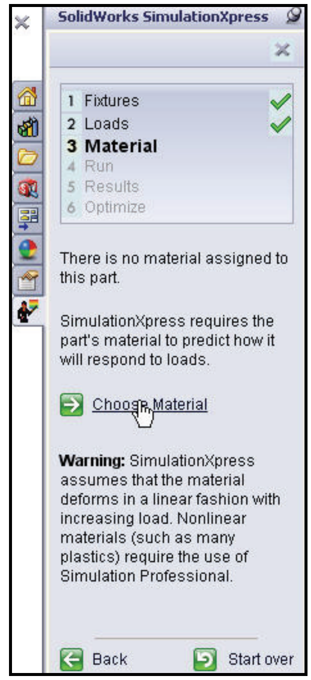
5 **Apply the Load.**
- Click **OK** ✔ from the Force PropertyManager. Force-1 is displayed in the SimulationXpress Study tree.

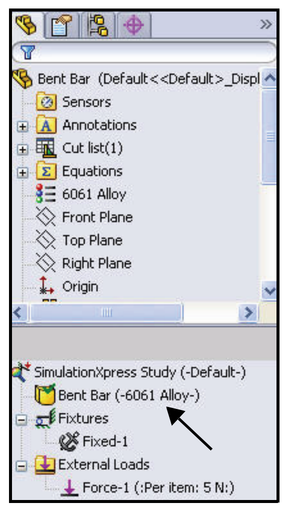




6 **Apply Material in the next section.**
- Click **Next**.

Assigning Material





The response of the part depends on the material it is made of. SolidWorks SimulationXpress must know the elastic properties of the material of your part. You can pick a material from the SolidWorks material library or define your own material properties. SolidWorks SimulationXpress uses the following material properties to perform stress analysis.

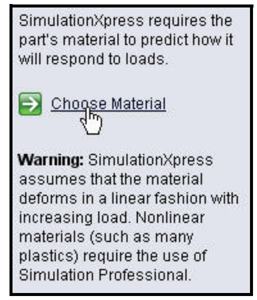
Elastic Modulus (EX). For a linear elastic material, the elastic modulus is the stress required to cause a unit strain in the material. In other words, stress divided by the associated strain. The modulus of elasticity was first introduced by Young and is often called the Young's Modulus.

Poisson's Ratio (NUXY). Extension of the material in the longitudinal direction is accompanied by shrinking in the lateral directions. For example, if a body is subjected to a tensile stress in the X-direction, then Poisson's Ratio NUXY is defined as the ratio of lateral strain in the Y-direction divided by the longitudinal strain in the X-direction. Poisson's ratios are dimensionless quantities. If not defined, the program assumes a default value of 0.

Yield Strength (SIGYLD). SolidWorks SimulationXpress uses this material property to calculate the factor of safety distribution. SolidWorks SimulationXpress assumes that the material starts yielding when the equivalent (von Mises) stress reaches this value.

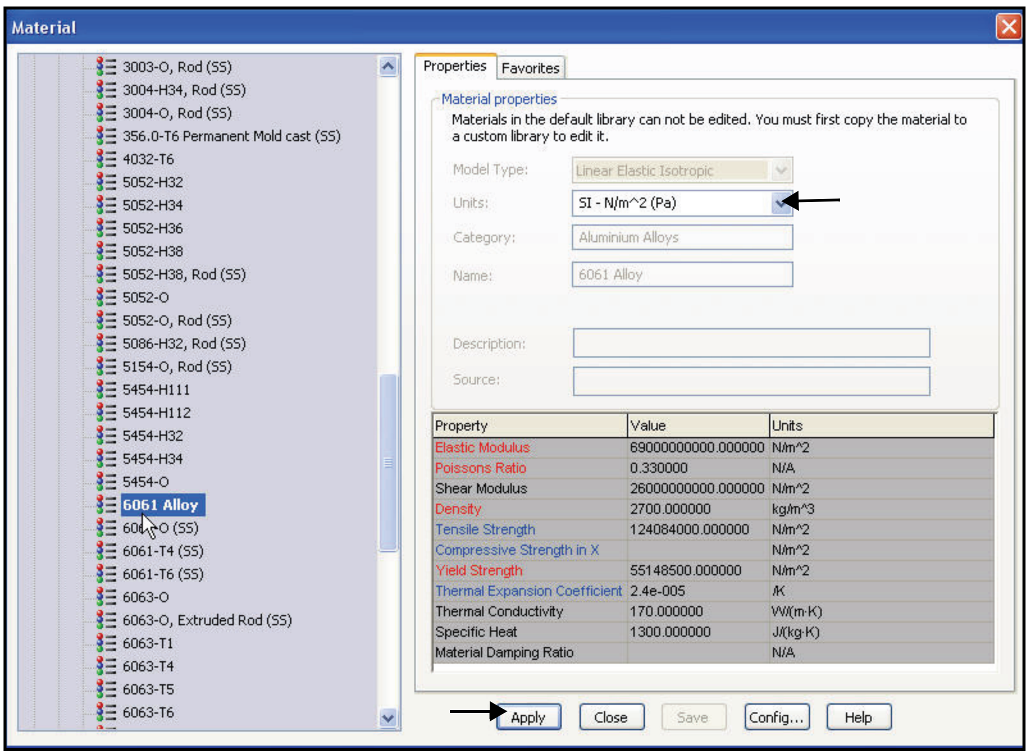
Mass Density (DENS). The density is mass per unit volume. Density units are lb/in^3 in the English system, and kg/m^3 in the SI system. SolidWorks SimulationXpress uses the mass density to include mass properties of the part in the report file.

Viewing the Material of the Bent Bar

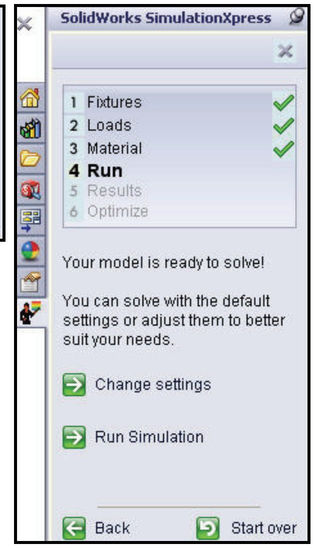


1 **View the Material options.**

- Click **Choose Material**. The Material dialog box is displayed
- **Expand** Aluminum Alloy.
- Click **6061 Alloy**. SI is selected for Units.
- Click **Apply**.
- Click **Close**.

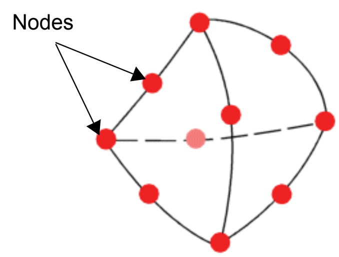


Running the Analysis



The Run option allows you to run the analysis. SimulationXpress prepares the model for analysis and then calculates displacements, strains, and stresses.

The first phase in the analysis is meshing. Meshing is basically splitting the geometry into small, simple-shaped pieces called finite elements.



A 3D Element

Design analysis uses finite elements to calculate the model's response to the applied loads and restraints. SimulationXpress estimates a default element size for the model based on its volume, surface area, and other geometric details. You can instruct SimulationXpress to use the default element size or you can use a different element size.

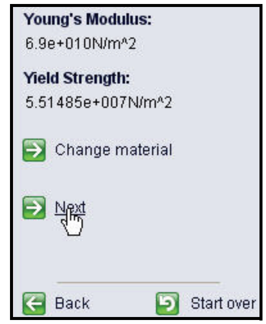
After meshing the model successfully, the second phase starts automatically. SimulationXpress formulates the equations governing the behavior of each element taking into consideration its connectivity to other elements. These equations relate the displacements to known material properties, restraints, and loads. The program then organizes the equations into a large set of simultaneous algebraic equations. The solver finds the displacements in the X, Y, and Z directions at each node.

Using the displacements, the program calculates the strains in various directions. Finally, the program uses mathematical expressions to calculate stresses.

Running the Analysis

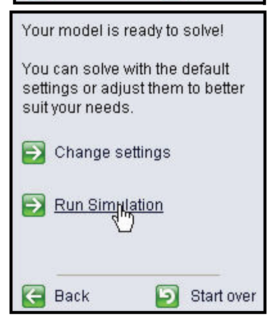
1 Analyze the results.

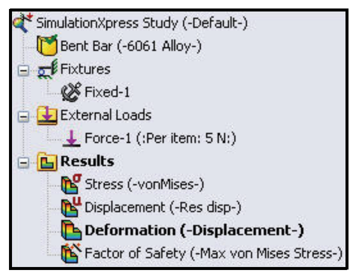
- Click **Next**.

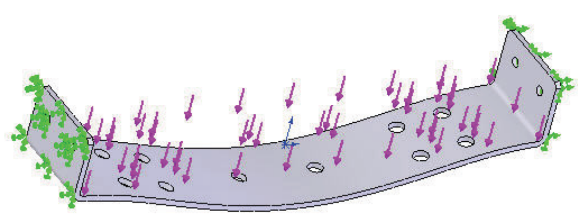


2 Run the analysis.

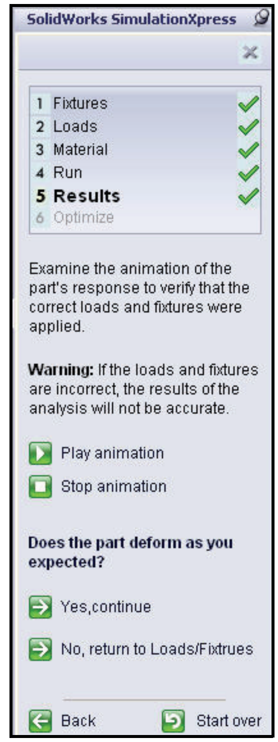
- Click **Run Simulation**. The analysis starts. When the analysis is complete, a green check mark  is displayed in the Run and Results section. View the SimulationXpress Study tree and animation in the Graphics area.







Viewing Results



Viewing results is an essential step in the analysis process. This is the step in which you evaluate how good your design is at withstanding the specified working conditions. This step should lead you to make important decisions about whether to accept the design and move to prototyping, make further improvements on the design, or try additional sets of loads and restraints.

SimulationXpress uses the maximum von Mises stress criterion to calculate the factors of safety. This criterion states that a ductile material starts to yield when the equivalent stress (von Mises stress) reaches the yield strength of the material. The yield strength (SIGYLD) is defined as a material property. SimulationXpress calculates the factor of safety (FOS) at a point by dividing the yield strength by the equivalent stress at that point.

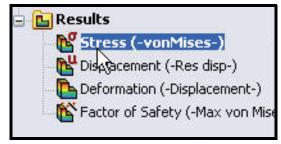
Interpretation of factor of safety values:

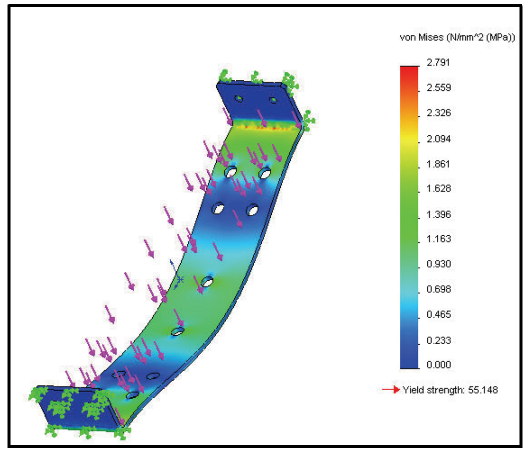
- A factor of safety less than 1.0 at a location indicates that the material at that location has yielded and that the design is not safe.
- A factor of safety of 1.0 at a location indicates that the material at that location has just started to yield.
- A factor of safety greater than 1.0 at a location indicates that the material at that location has not yielded.
- The material at a location will start to yield if you apply new loads equal to the current loads multiplied by the resulting factor of safety.

Viewing the Results

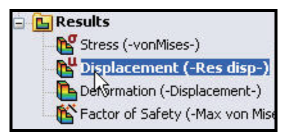
1 View the results.

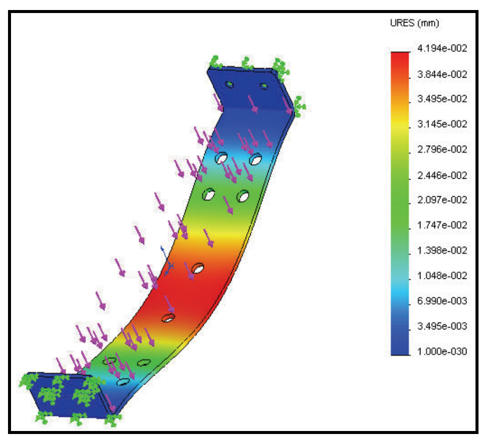
- Click **Stop animation**.
- Double-click the **Stress (-vonMises-)** folder. View the results.



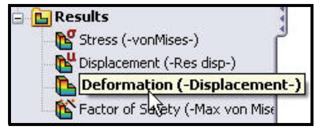


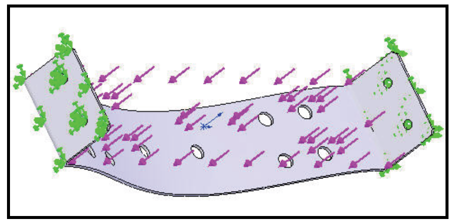
- Double-click the **Displacement** folder. View the results.



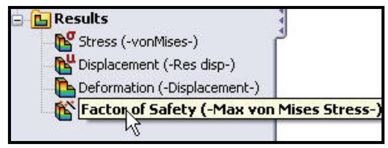


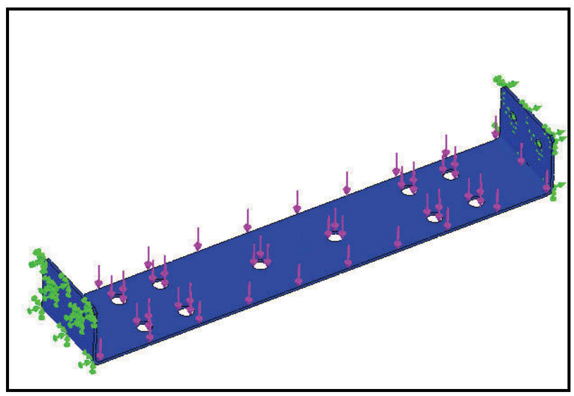
- Double-click the **Deformation** folder. View the results.





- Double-click the **Factor of Safety** folder. View the results.

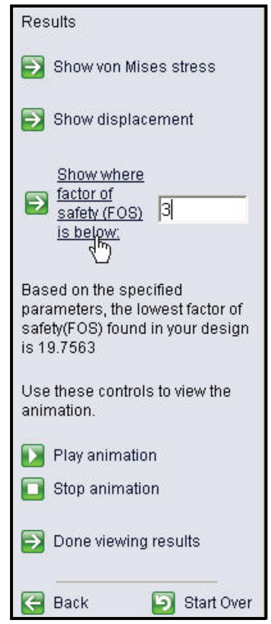




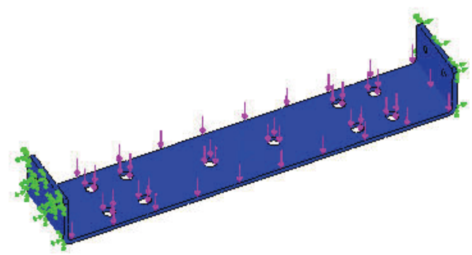
- Click **Yes, continue**.
- The factor of safety of the Bent Bar part is approximately 20. This indicated that the current design is safe or may be overdesigned.

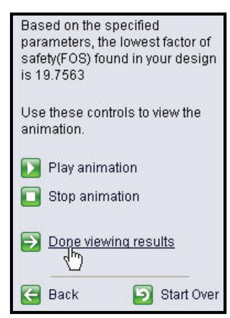
2 Modify the factor of safety.

- Enter **3** in the Show where factor of safety (FOS) is below box.
- Click the **Show where factor of safety (FOS) is below:** button as illustrated. The following plot is displayed. Regions in blue have factors of safety greater than 3 (overdesigned regions). Regions in red have factors of safety less than 3.



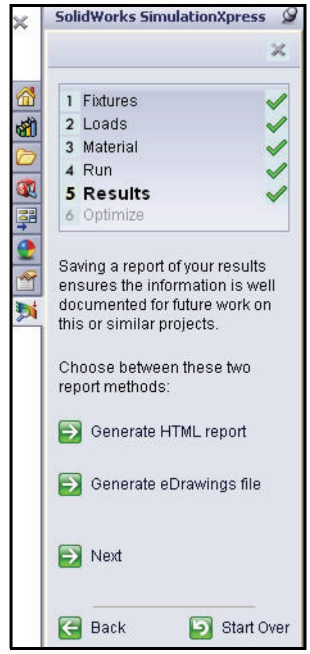
- **Rotate** the model with the middle mouse button to view the area. All areas are displayed in blue.
- Click **Done viewing results**.





Generating the Analysis Report

Documenting analysis data and results is essential. SolidWorks® SimulationXpress helps you document your analysis data and results quickly and systematically by generating an Internet-ready HTML (Hyper Text Markup Language) report that can be viewed by your Internet browser or an eDrawing.



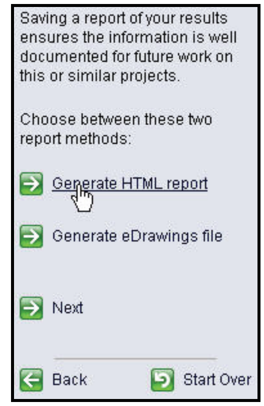
The HTML report is structured to describe all aspects of the analysis including material properties, applied restraints and loads, as well as results of the analysis.

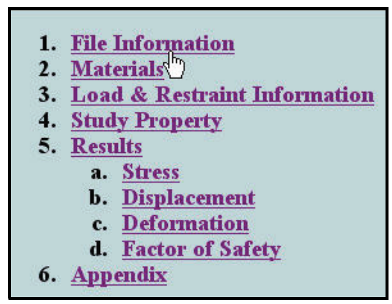
Result plots are automatically included in the report. A printer-friendly version of the report can be generated automatically. SolidWorks® SimulationXpress reports provide an excellent way to painlessly document your work and share your findings with others online or in printed format.

Generating the Analysis Report

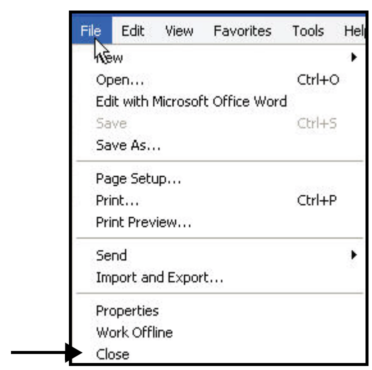
1 **Generate the analysis report.**

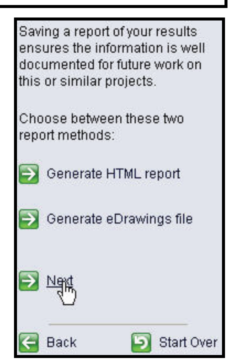
- Click **Generate HTML report**. View the information that is provided. As an exercise, fill out the File Information section.



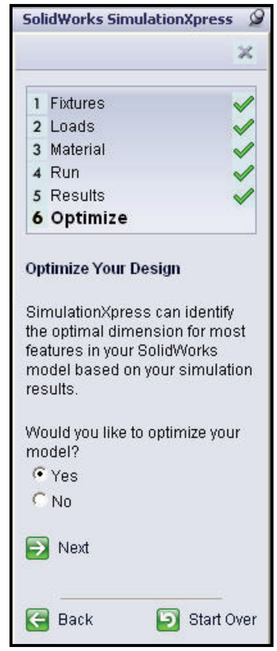


- **Close** the document and return to SolidWorks.
- Click **Next**.





Optimize

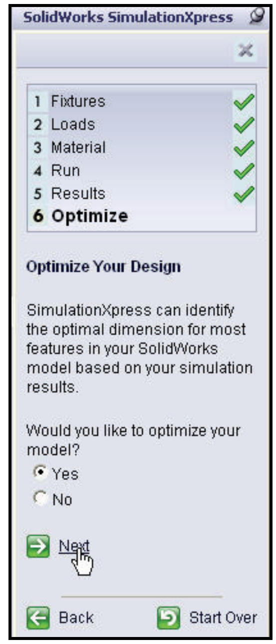


The Optimize option allows you to perform an optimization analysis after completing the stress analysis. The software tries to find the optimal value for one model dimension while satisfying a specified criterion:

- Factor of Safety
- Maximum Stress
- Maximum displacement

You can either input your desired Factor of Safety or allow the SolidWorks® SimulationXpress to calculate Factor of Safety based on the upper and lower limits.

Running the Optimization

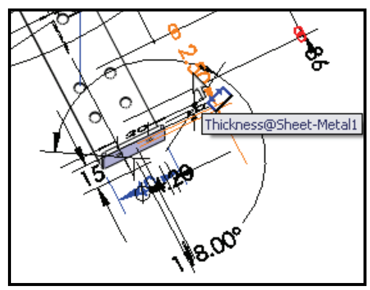


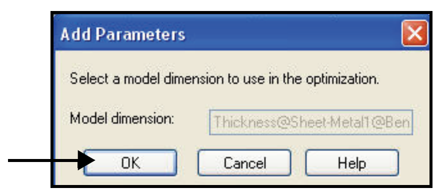
1 Run Optimization.

- Click **Next**. The Optimization table is displayed in the bottom half of the Graphics area.

2 Select an Optimization criterion.

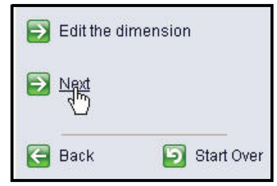
- Select the dimension you want to modify. Click the **2.50mm** dimension thickness of the Sheet metal from the Graphics area. The dimension is displayed in the Model dimension box.
- Click **OK** from the Add Parameters dialog box.

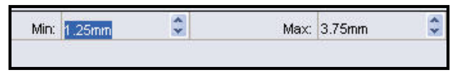




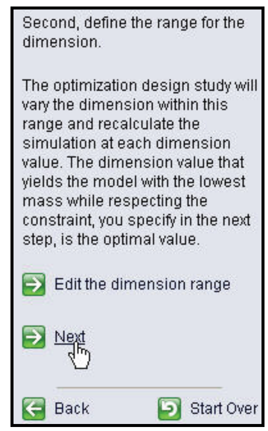
3 Optimize the part.

- Click **Next.** Define the Max. and Min. range for the dimension. The default Max. range is 3.75mm. The default Min. range is 1.25. Accept the default ranges.

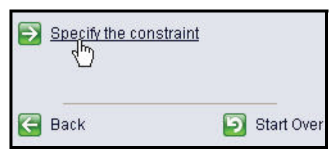


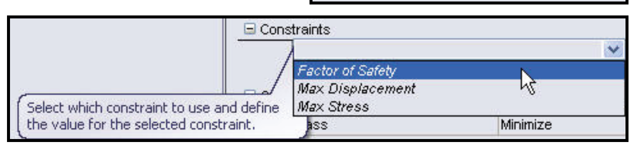


- Click **Next**. Specify the constraint for the study.
- Click **Next**.
- Click **Specify the constraint**. Select the Factor of Safety for the constraint.



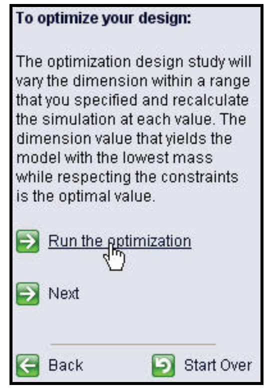
- Click the **drop-down arrow** from the Constraints column.
- Click **Factor of Safety**.
- Select **6** for Min. Factor of Safety from the drop-down menu.
- Click **Next**.

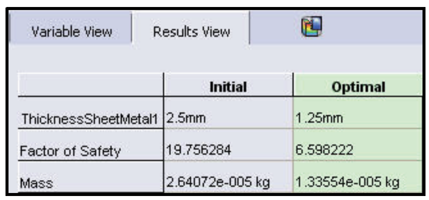




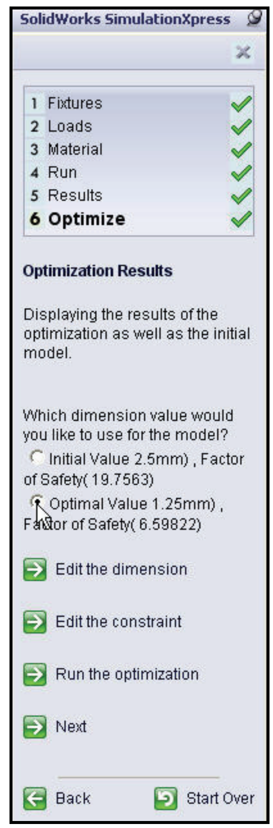
4 Run the Optimization Study.

- Click **Run Optimization.**
- **View** the results. There is approximately 50% material savings with the new design using the lower bounds of 1.25mm thickness for the Sheet metal part. Accept the default options.
- Check the **Optimal Value** box.





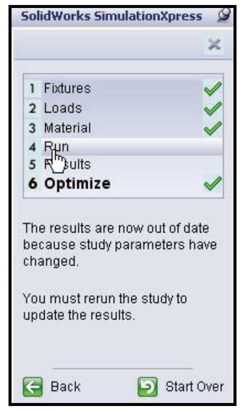
Variable View	Results View	
	Initial	Optimal
ThicknessSheetMetal1	2.5mm	1.25mm
Factor of Safety	19.756284	6.598222
Mass	2.64072e-005 kg	1.33554e-005 kg

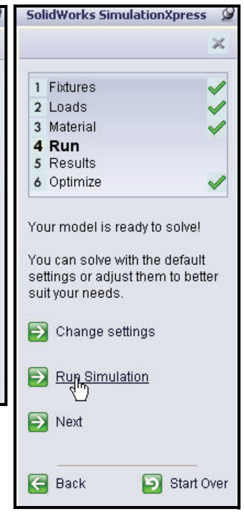


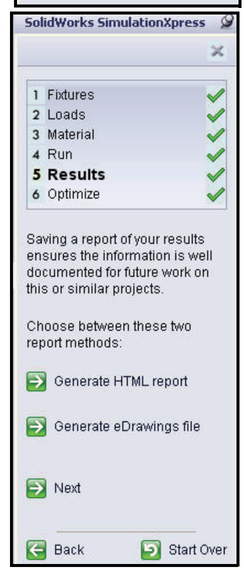
- Click **Next**. The results are not out of date because the study parameters have changed. You must re-run the study to update the plot results.

5 **Re-run the Study.**

- Click **Run** in the Wizard window.
- Click **Run Simulation**. The SolidWorks® SimulationXpress Wizard window is updated.







6 **Save the part.**

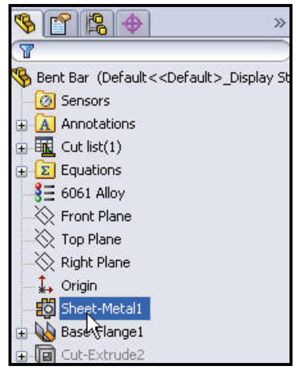
- Click **Save** .

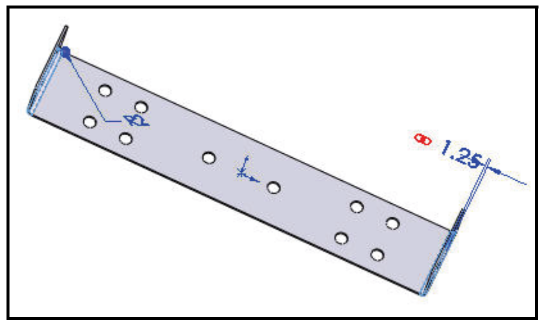
7 **View the updated part.**

- Click **Sheet-Metal1** in the FeatureManager. View the new updated thickness for the part in the Graphics area.

8 **Close SolidWorks SimulationXpress.**

- **Close** the model.





Conclusion

During this short session on using SolidWorks® SimulationXpress, you have had a brief exposure to the main concepts of analysis. The ease of use of SolidWorks® SimulationXpress allows designers to conduct up front analysis on part designs directly in SolidWorks. Results of analyses are used to drive changes in geometry or material selection allowing you to test many options quickly. Analysis therefore helps you optimize your designs, minimize the cost, and reduce the time-to-market.

While SolidWorks® SimulationXpress analyzes parts only, SolidWorks® SimulationXpress is an analysis program that analyzes parts as well as assemblies and performs additional types of analyses such as modal, buckling, and thermal.

Glossary

animate View a model or an eDrawings file in a dynamic manner. Animation simulates motion or displays different views.

assembly An assembly is a document in which parts, features, and other assemblies (subassemblies) are mated together. The parts and subassemblies exist in documents separate from the assembly. For example, in an assembly, a piston can be mated to other parts, such as a connecting rod or cylinder. This new assembly can then be used as a sub-assembly in an assembly of an engine. The extension for a SolidWorks assembly file name is .SLDASM.

axis An axis is a straight line that can be used to create model geometry, features, or patterns. An axis can be made in a number of different ways, including using the intersection of two planes.

boss/base A base is the first solid feature of a part, created by a boss. A boss is a feature that creates the base of a part, or adds material to a part, by extruding, revolving, sweeping, or lofting a sketch, or by thickening a surface.

click-drag As you sketch, if you click and drag the pointer, you are in click-drag mode. When you release the pointer, the sketch entity is complete.

closed profile A closed profile (or closed contour) is a sketch or sketch entity with no exposed endpoints, for example, a circle.

collapse Collapse is the opposite of explode. The collapse action returns an exploded assembly's parts to their normal positions.

component A component is any part or subassembly within an assembly.

Configuration Manager The ConfigurationManager on the left side of the SolidWorks window is a means to create, select, and view the configurations of parts and assemblies.

degrees of freedom Geometry that is not defined by dimensions or relations is free to move. In 2D sketches, there are three degrees of freedom: movement along the X and Y axes, and rotation about the Z axis (the axis normal to the sketch plane). In 3D sketches and in assemblies, there are six degrees of freedom: movement along the X, Y, and Z axes, and rotation about the X, Y, and Z axes.

document A SolidWorks document is a file containing a part, assembly, or drawing.

double-click Click two times with the left mouse button.

drawing A drawing is a 2D representation of a 3D part or assembly. The extension for a SolidWorks drawing file name is .SLDDRW.

drawing sheet A drawing sheet is a page in a drawing document.

eDrawings file Compact representation of a part, assembly, or drawing. eDrawings files are compact enough to email and can be created for a number of CAD file types including SolidWorks.

face A face is a selectable area (planar or otherwise) of a model or surface with boundaries that help define the shape of the model or surface. For example, a rectangular solid has six faces.

feature A feature is an individual shape that, combined with other features, makes up a part or assembly. Some features — such as bosses and cuts — originate as sketches. Other features, such as shells and fillets, modify a feature's geometry. However, not all features have associated geometry. Features are always listed in the FeatureManager design tree.

FeatureManager design tree The FeatureManager design tree on the left side of the SolidWorks window provides an outline view of the active part, assembly, or drawing.

graphics area The graphics area is the area in the SolidWorks window where the part, assembly, or drawing appears.

line A line is a straight sketch entity with two endpoints. A line can be created by projecting an external entity such as an edge, plane, axis, or sketch curve into the sketch.

mate A mate is a geometric relationship, such as coincident, perpendicular, tangent, and so on, between parts in an assembly.

mategroup A mategroup is a collection of mates that are solved together. The order in which the mates appear within the mategroup does not matter.

model A model is the 3D solid geometry in a part or assembly document. If a part or assembly document contains multiple configurations, each configuration is a separate model.

named view A named view is a specific view of a part or assembly (isometric, top, and so on) or a user-defined name for a specific view. Named views from the view orientation list can be inserted into drawings.

open profile An open profile (or open contour) is a sketch or sketch entity with endpoints exposed. For example, a U-shaped profile is open.

overdefined A sketch is overdefined when dimensions or relations are either in conflict or redundant.

part A part is a single 3D object made up of features. A part can become a component in an assembly, and it can be represented in 2D in a drawing. Examples of parts are bolt, pin, plate, and so on. The extension for a SolidWorks part file name is .SLDPRT.

planar An entity is planar if it can lie on one plane. For example, a circle is planar, but a helix is not.

plane Planes are flat construction geometry. Planes can be used for a 2D sketch, section view of a model, a neutral plane in a draft feature, and others.

point A point is a singular location in a sketch, or a projection into a sketch at a single location of an external entity (origin, vertex, axis, or point in an external sketch).

profile A profile is a sketch entity used to create a feature (such as a loft) or a drawing view (such as a detail view). A profile can be open (such as a U shape or open spline) or closed (such as a circle or closed spline).

PropertyManager The PropertyManager is on the left side of the SolidWorks window for dynamic editing of sketch entities and most features.

rebuild The rebuild tool updates (or regenerates) the document with any changes made since the last time the model was rebuilt. Rebuild is typically used after changing a model dimension.

relation A relation is a geometric constraint between sketch entities or between a sketch entity and a plane, axis, edge, or vertex. Relations can be added automatically or manually.

revolve Revolve is a feature tool that creates a base or boss, a revolved cut, or revolved surface by revolving one or more sketched profiles around a centerline.

section A section is another term for profile in sweeps.

section view A section view (or section cut) is:
(1) a part or assembly view cut by a plane, or
(2) a drawing view created by cutting another drawing view with a section line.

sheet format A sheet format typically includes page size and orientation, standard text, borders, title blocks, and so on. Sheet formats can be customized and saved for future use. Each sheet of a drawing document can have a different format.

shell Shell is a feature tool that hollows out a part, leaving open the selected faces and thin walls on the remaining faces. A hollow part is created when no faces are selected to be open.

sketch A 2D sketch is a collection of lines and other 2D objects on a plane or face that forms the basis for a feature such as a base or a boss. A 3D sketch is nonplanar and can be used to guide a sweep or loft, for example.

SmartMates A SmartMate is an assembly mating relation that is created automatically.

subassembly A subassembly is an assembly document that is part of a larger assembly. For example, the steering mechanism of a car is a subassembly of the car.

surface A surface is a zero-thickness planar or 3D entity with edge boundaries. Surfaces are often used to create solid features. Reference surfaces can be used to modify solid features.

underdefined A sketch is underdefined when there are not enough dimensions and relations to prevent entities from moving or changing size.