SolidWorks® 2011

Hands-on Test Drive

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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

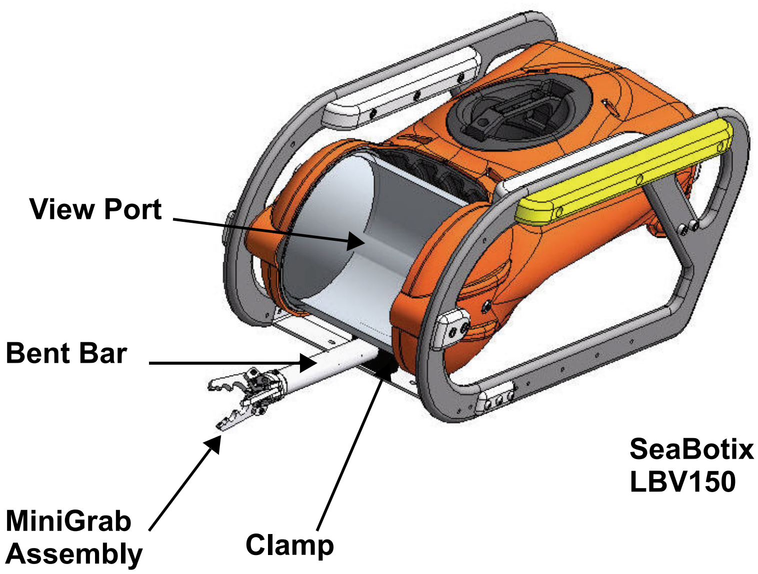
Hands-on Test Drive

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.

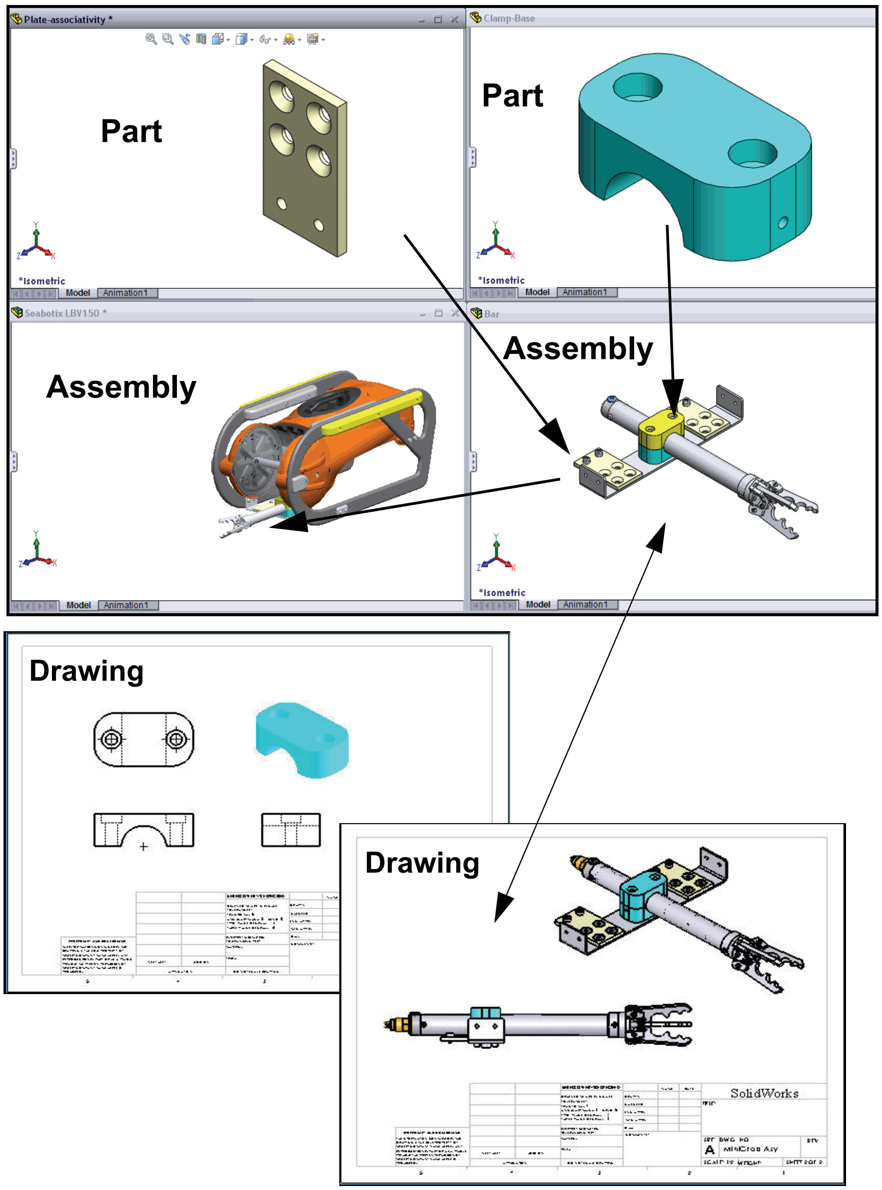
In SolidWorks, parts, assemblies, and drawings are associative. This means that changes in one location are reflected in all of the other locations where they need to be reflected. Changes that you make to an assembly are reflected in the associative drawings of that assembly. Changes that you make to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



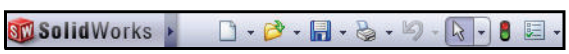
SolidWorks User Interface (UI)

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

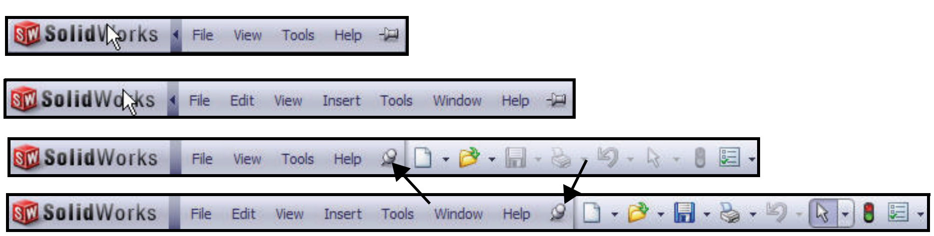
The Menu Bar toolbar contains a set of the most frequently used tool buttons from the Standard toolbar. The available tools are: **New** - Creates a new document, **Open** - Opens an existing document, **Save** - Saves an active document, **Print** - Prints an active document, **Undo** - Reverses the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



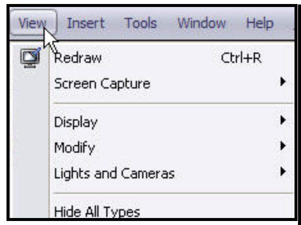
Menu Bar Menu

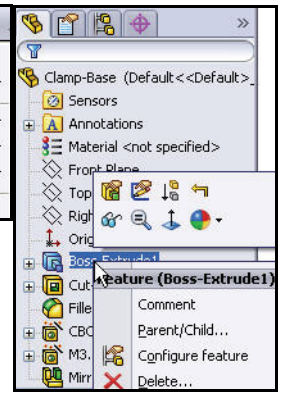
Drag the mouse cursor over the SolidWorks name in the Menu Bar toolbar to display the default Menu Bar menu. SolidWorks provides a context-sensitive menu structure. The menu titles remain the same for all three types of documents: part, assembly, and drawing but the menu items change depending on which 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: 

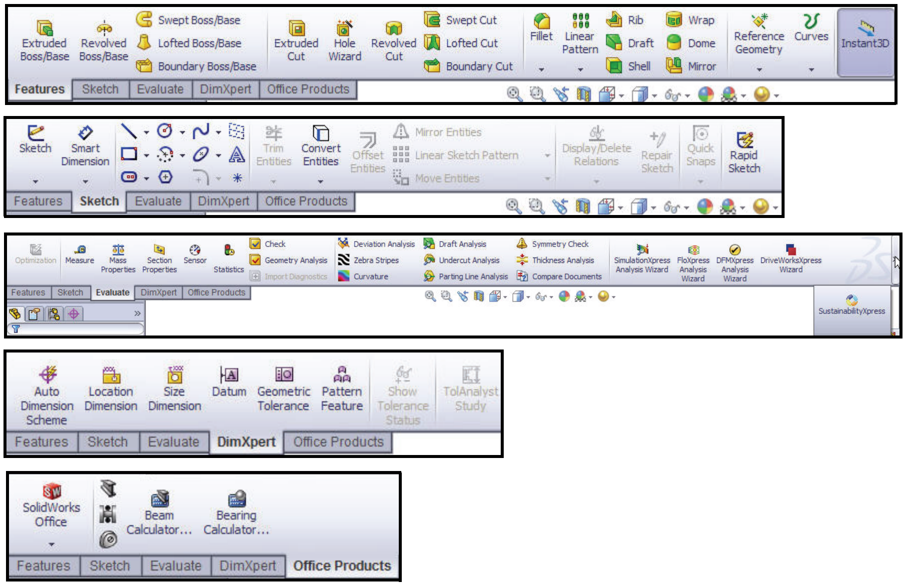
. SolidWorks conforms to standard Windows conventions for shortcuts such as **Ctrl+O** for **File, Open**; **Ctrl+S** for **File, Save**; **Ctrl+X** for **Cut**; **Ctrl+C** for **Copy**; and so on. In addition, you can customize SolidWorks by creating your own shortcuts.

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**.

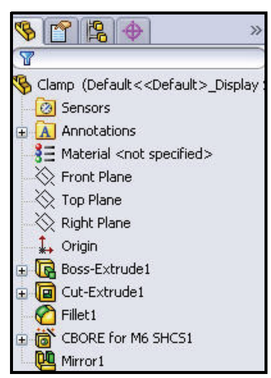
Below is an illustrated CommandManager for a default Part document.

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:

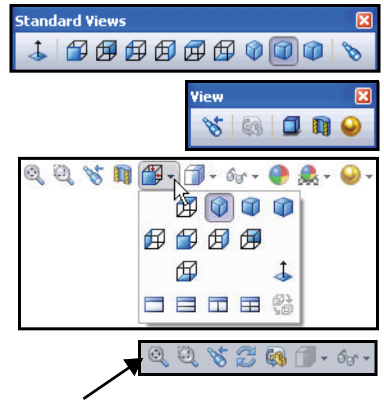
FeatureManager , **PropertyManager** ,
ConfigurationManager , and **DimXpertManager** .

Heads-up View Toolbar

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

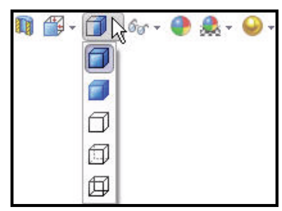
The Heads-up View toolbar is a transparent toolbar that is displayed in the Graphics area when a document is active

The following views are available:

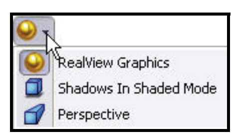


For an active drawing document

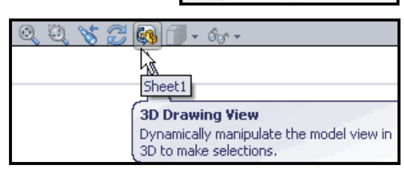
- **Zoom to Fit** . Zooms the model to fit the Graphics area.
- **Zoom to Area** . Zooms to the areas you select with a bounding box.
- **Previous View** . Displays the previous view.
- **Section View** . Displays the cutaway of a part or assembly using one or more cross section planes.
- **View Orientation**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

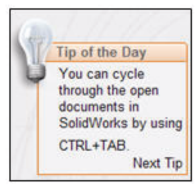


Task Pane

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.

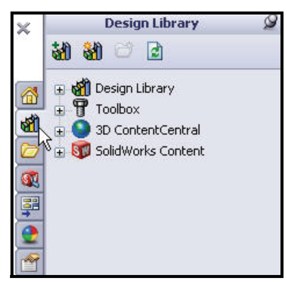
SolidWorks Resources



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

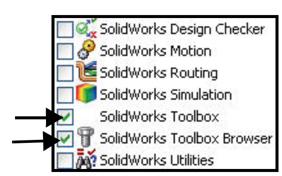
Other user interfaces are available: *Machine Design*, *Mold Design*, or *Consumer Products Design* during the initial software installation selection.

Design Library

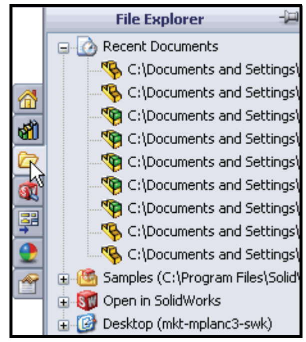


The Design Library contains reusable parts, assemblies, and other elements, including library features. The Design Library tab contains four default selections. Each default selection contains additional subcategories. The default selections are: *Design Library, Toolbox, 3D ContentCentral*, and *SolidWorks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

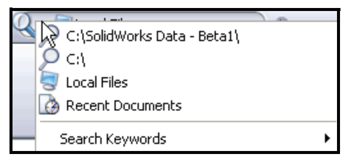


File Explorer



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

Search

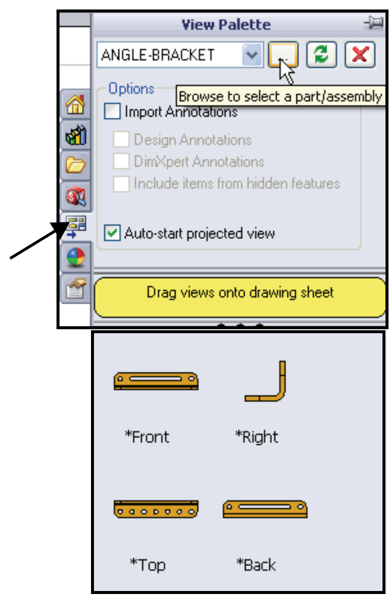


SolidWorks Search is installed with Microsoft Windows Search and indexes the resources once before searching begins, either after installation or when you initiate the first search.

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

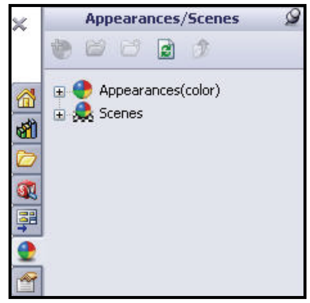


The View Palette tab located in the Task Pane provides the ability to insert drawing views of an active document, or click the Browse button to locate the desired document.

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.

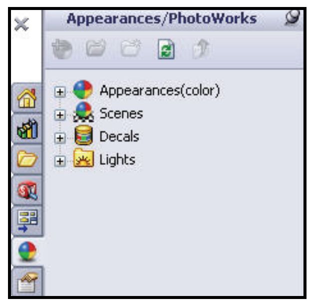
Appearances/Scenes



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

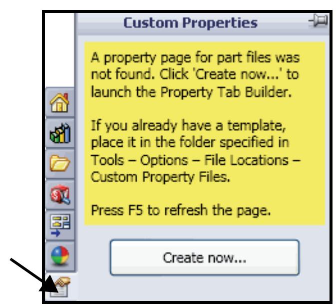
On RealView compatible systems, you can select Appearances and Scenes to display your model in the Graphics area. Drag and drop a selected appearance onto the model or FeatureManager. View the results in the Graphics area.

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



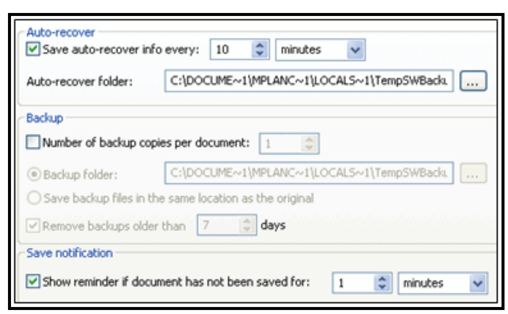
Note: Appearances/PhotoWorks graphics is only available with supported graphics cards. For the latest information on graphics cards that support Appearances/PhotoWorks display, visit: www.solidworks.com/pages/services/videocardtesting.html.

Custom Properties



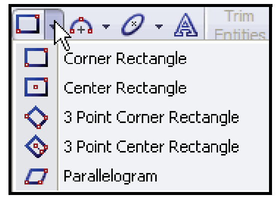
The Custom Properties tab located in the Task Pane. Custom Properties provides the ability to enter custom and configuration specific properties directly into SolidWorks files.

Document Recovery



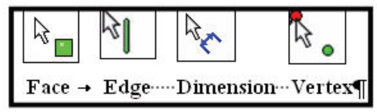
If Auto recovery is initiated in the System Options section and the system terminates unexpectedly with an active document, the saved information files are available on the Task Pane Document Recovery tab the next time you start a SolidWorks session.

Consolidated Flyout Tool Buttons

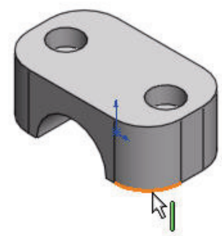


Similar commands are grouped into consolidated flyout buttons on the toolbar and in the CommandManager. Example: Variations of the Rectangle Sketch tool are grouped together into a single button with a flyout control.

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.

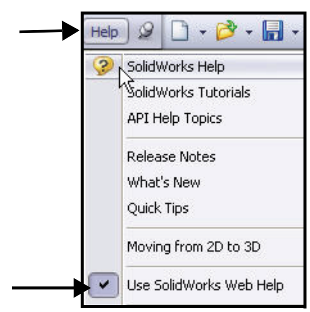


Mouse Buttons

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.

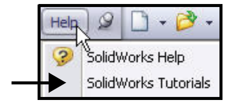
Getting Help



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.

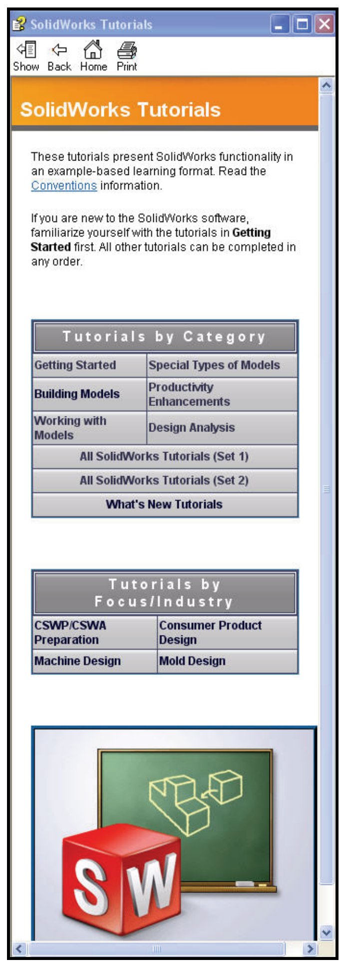
Click **Help**, **SolidWorks Help** from the Menu Bar menu to view the comprehensive SolidWorks Home Help Page screen.

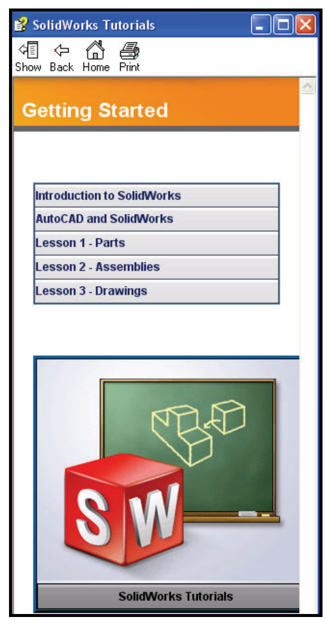
SolidWorks Tutorials



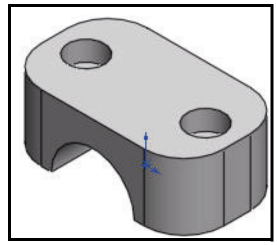
The SolidWorks Tutorials provide step-by-step lessons with sample files covering SolidWorks terminology, concepts, functions, features, What's New and many Add-Ins. Work or view the 30 minute lesson tutorials to learn and strengthen your skills.

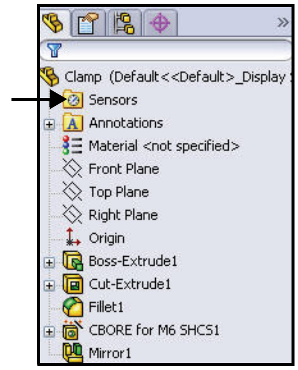
Click **Help**, **SolidWorks Tutorials** from the Menu Bar menu, or click the SolidWorks **Resources** tab from the Task Pane and click **Tutorials**. View the available tutorials. The tutorials are displayed by category.





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.

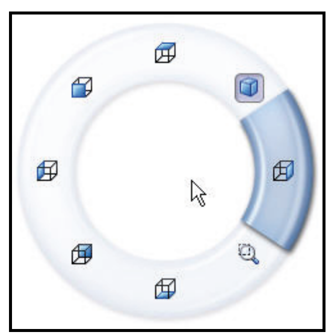
The new part document will be created based on a template. A template forms the foundation of a new document. The template can include user-defined parameters, annotations, or geometry.

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.



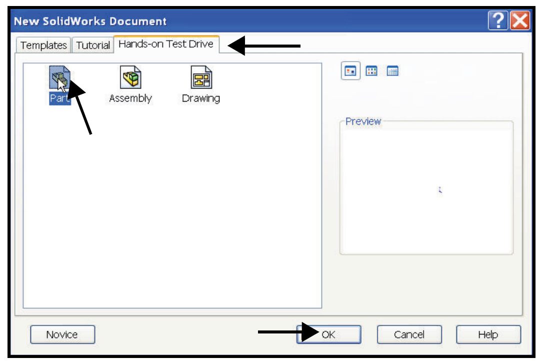
Starting SolidWorks and Opening a New Part

1 Start the SolidWorks application.

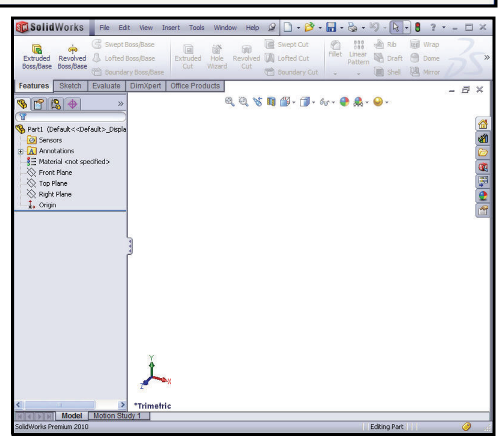
- Click the **Start** menu from the Windows interface.
- Click **All Programs**, **SolidWorks 2010**, **SolidWorks 2010**.

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.



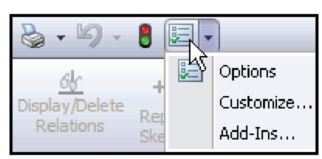
- Click **New** from the Menu Bar toolbar. The New SolidWorks Document dialog box is displayed.
- Click the **Part** template from the Hands-on Test Drive tab.
- Click **OK** from the New SolidWorks Document dialog box. A new Part document window is displayed.

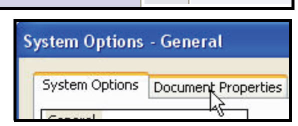


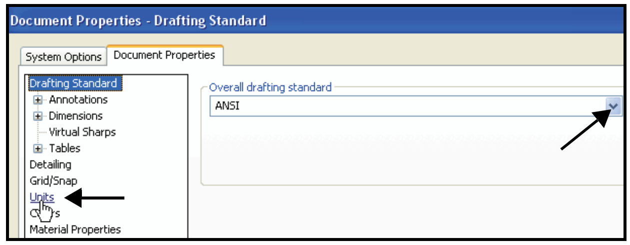
Note: Templates are part, assembly, and drawing documents that include user-defined parameters and are the basis for a new document. The Hands-on Test Drive tab was created for this class.

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.



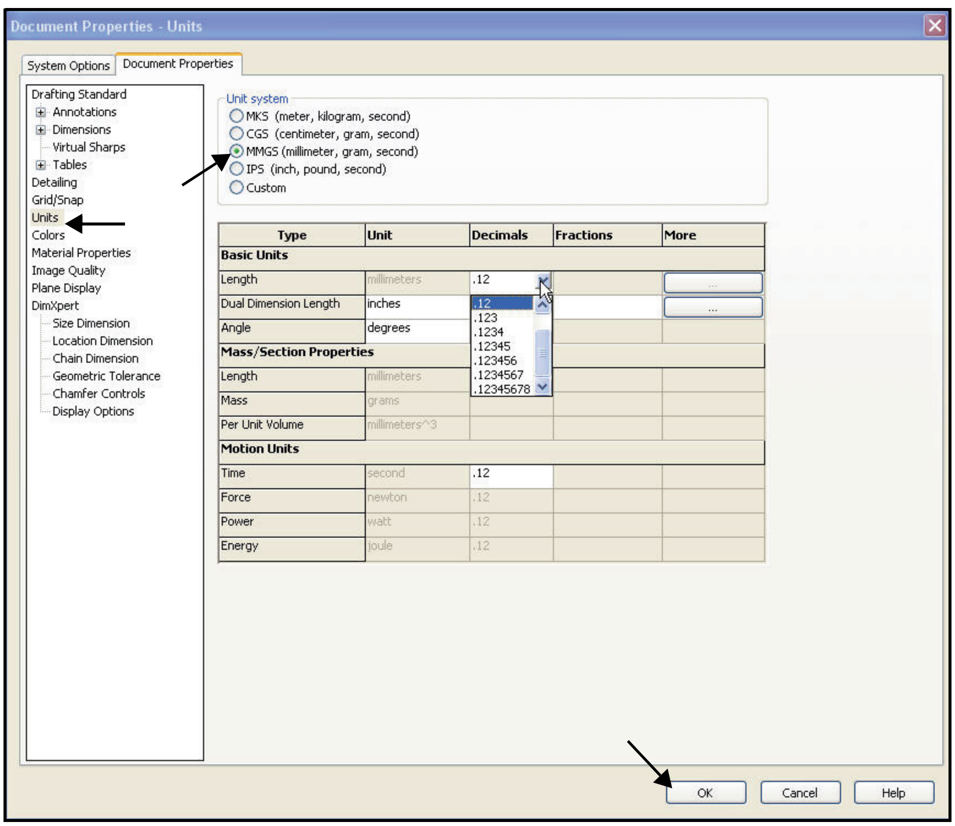




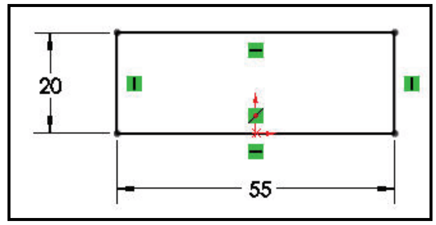
- Click the **Units** folder.
- Select **MMGS**, (millimeter, gram, second).
- Click inside the **Length Decimals** box.
- Select **.12** decimal places from the drop-down menu for Basic Units as illustrated.

4 Accept the default settings.

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

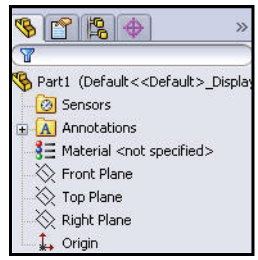


Begin Sketching



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

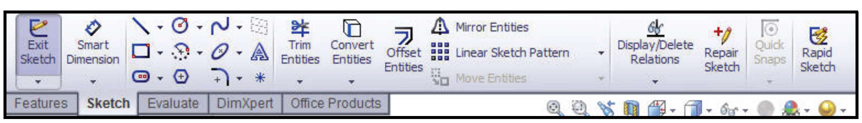
Create a 2D sketch that you will later make into a 3D solid. For the Clamp part, create a 2D sketch and insert dimensions as illustrated.



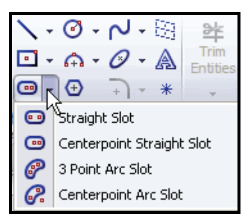
Sketches are flat or planar. You need a plane on which to sketch. A SolidWorks part contains three default sketch planes. They represent the Front, Top, and Right Planes. For the Clamp, use the Front Plane as the Sketch plane.

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.

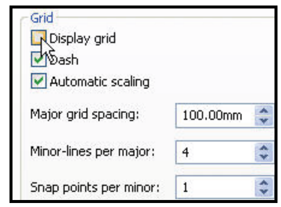


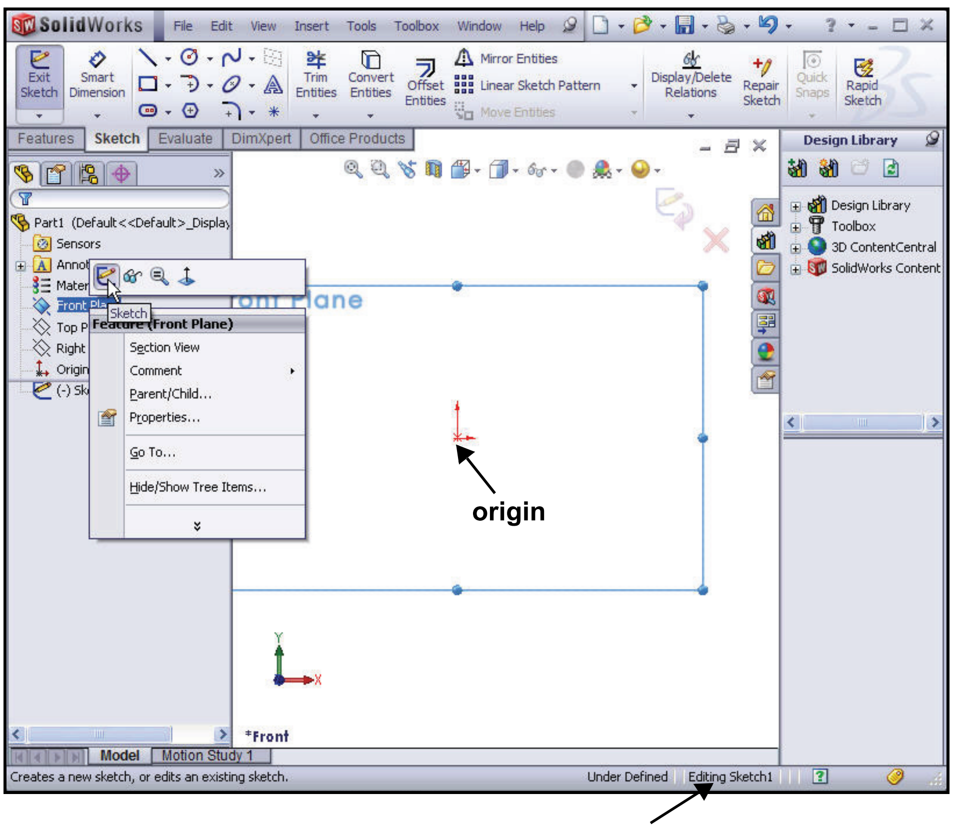
Selecting the Sketch Plane

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.

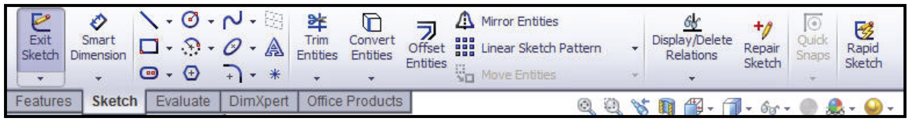
Note: The default grid is displayed if Grid is active.





Indications that you are in Sketch Mode

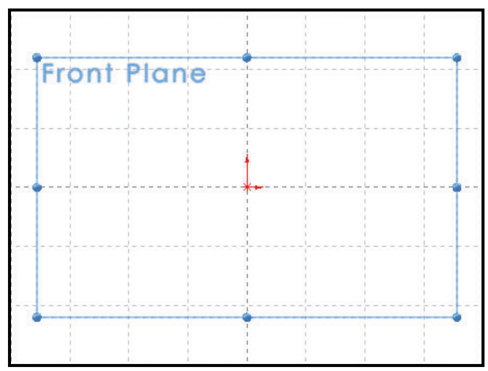
The Sketch toolbar is active.



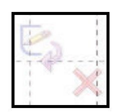
The title bar of the window tells you that you are in a sketch.

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.

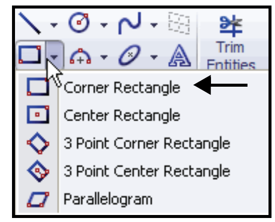


When numerous SolidWorks commands are active, a symbol or a set of symbols are displayed in the upper right corner of the Graphics area. This area is called the Confirmation Corner.



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



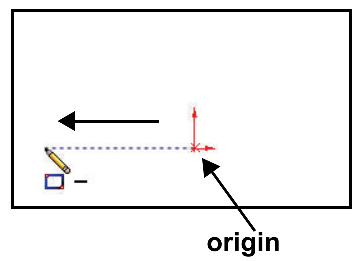
1 Sketch a rectangle.

- 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.

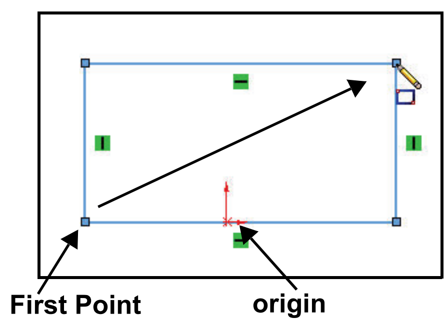


3 Complete the rectangle.

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

4 Zoom to fit.

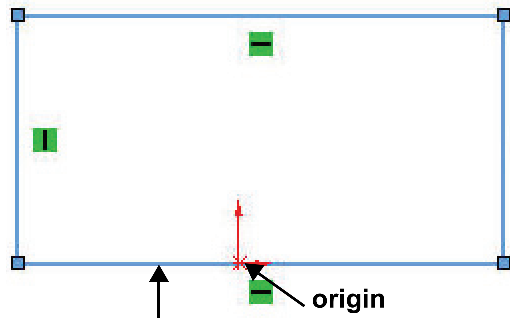
- Press the **f** key on the keyboard to return to the full Graphics area.



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

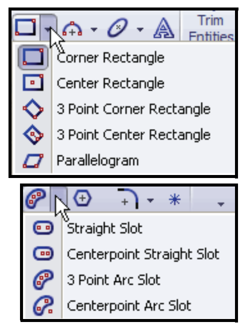
Adding Geometric Relations

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.



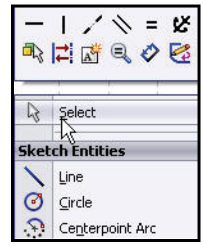
Note: When a sketch is active, a sketch origin is displayed in red and represents the (0,0,0) coordinate of the sketch.

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.



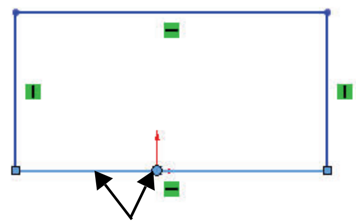
Note: SolidWorks provides various Sketch tools, (Center Rectangle, Centerpoint Straight Slot, etc.) that automatically inserts the needed geometric relations.

Relate the Origin to the Bottom Horizontal Line

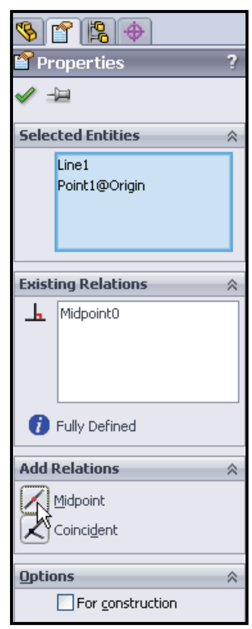


1 **Select Entities.**
- Right-click **Select** in the Graphics area to deselect the Corner Rectangle Sketch tool.

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

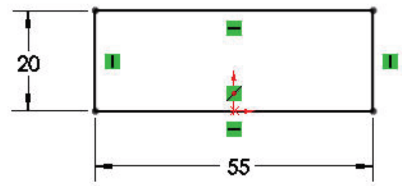


Note: SolidWorks provides the ability to either select available geometric relations from the Pop-up Context toolbar in the Graphics area or from the Properties PropertyManager.

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.

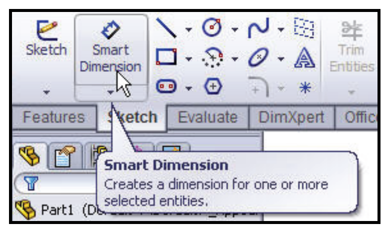
Defining the size

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.



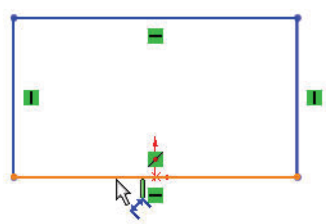
Dimension sketch entities and other objects with the Smart Dimension tool from the Sketch toolbar. The type of dimension is determined by the items which you select.

Example:



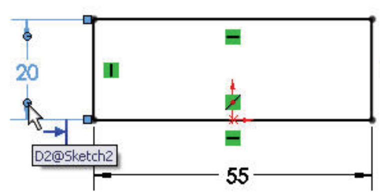
- If you select a line, the system creates a linear dimension.
- If you select a circle, the system creates a diameter dimension.
- If you select two parallel lines, the system creates a linear dimension between the two lines.

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.



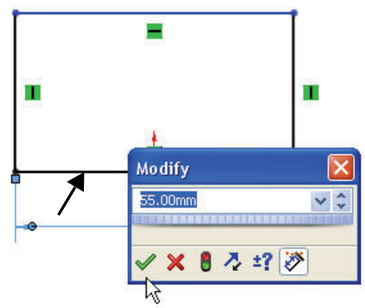
The Smart Dimension tool provides an icon feedback symbol.

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



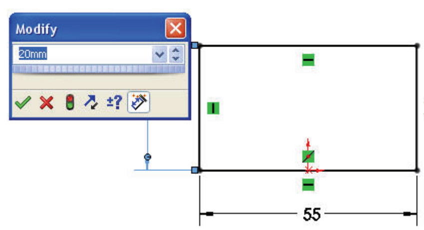
Dimensioning the Sketch

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.

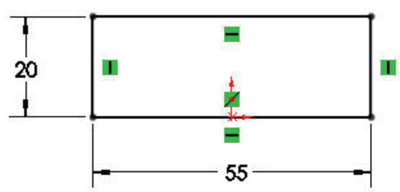
Note: The Smart Dimension tool automatically provides you with the correct dimension units.



2 Zoom to fit.

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

3 Add a vertical dimension.

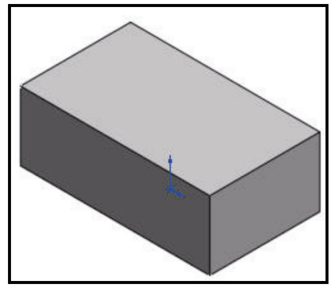


- Click the **left vertical** line.
- Click a **position** to the left of the vertical line. The Modify dialog box is displayed.
- Enter **20**mm in the Modify dialog box.
- Click the **green checkmark** button. The dimensioning of the profile is complete. The profile is now fully defined in size, shape,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

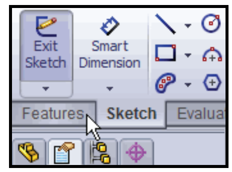
- Press the **Esc** key to deactivate the Smart Dimension tool from the Sketch toolbar.

Extruding a Solid



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

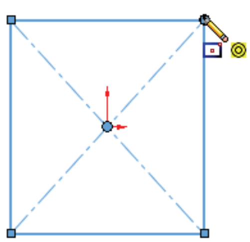
Extruding builds the solid normal to the sketch plane for some specified distance. An extruded boss is a profile that has been projected for a specified distance.



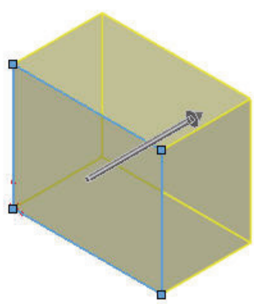
There are numerous options for extruding a sketch such as draft angles, end conditions, and depth. These options allow you to create smart, manufacturable models. However, most of these options are beyond the scope of this quick introduction, so we will just keep it simple.

To Create an Extruded Feature:

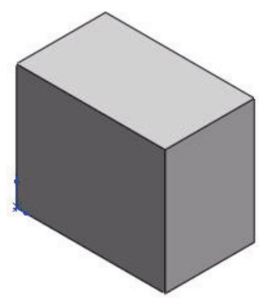
1. Select a sketch plane.
2. Sketch a 2D profile.
3. Extrude the sketch perpendicular to the sketch plane.



Sketch the 2D profile



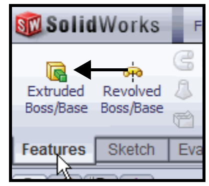
Extrude the sketch



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

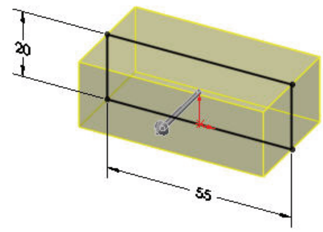


1 Extrude the feature.

- 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.

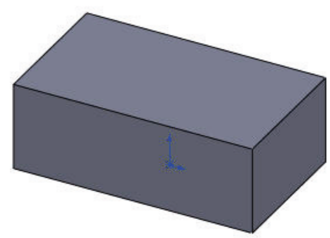
- Select **Mid Plane** for End Condition in Direction 1.
- Enter **32**mm for Depth. You can use the up and down arrow buttons next to the Depth box to change the value by 10mms (default in system options) at a time.



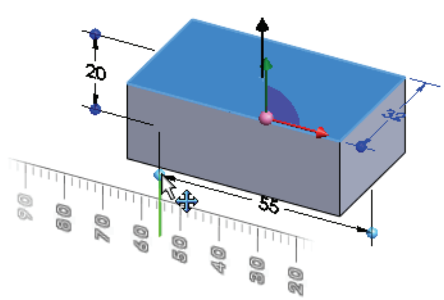
Note: The default Extruded direction is towards the front. The default End Condition is Blind. The Mid Plane option is utilized to incorporate symmetry into the design intent.

3 Accept the values and view the completed feature.

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

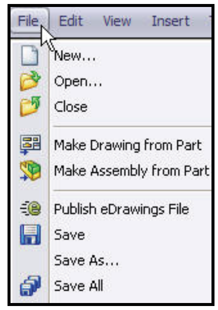


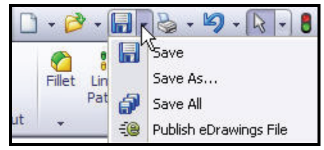
Tip: Instant3D provides the ability to click and drag geometry and the dimension manipulator points to resize or create features directly from the Graphics area. Use the on-screen ruler to measure your modifications. This is addressed later in the book.



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.





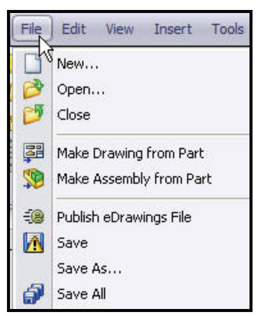
Note: This book is written with the assumption that your files are in and saved to the SolidWorks Test Drive folder.

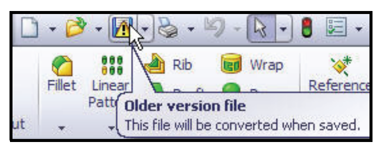
You can save your files as often as you wish. However, there are really only two situations that require you to save your work:

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

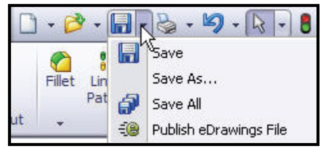
Saving protects work that you have already done. If you try a technique and accidently get results that you didn't anticipate, you can return to your saved file.

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!

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