FUNDAMENTALS OF 3D DESIGN AND SIMULATION

SOLIDWORKS EDUCATION EDITION 2022-2023

ENG

This is a preview of the Fundamentals of 3D Design and Simulation.

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SOLIDWORKS®
Education Edition

Fundamentals of 3D Design and Simulation

To be used with SOLIDWORKS Education Edition 2021-2022 or 2022-2023

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Introduction

To the Teacher

The *SOLIDWORKS Education Edition - Fundamentals of 3D Design and Simulation* manual is designed to assist you in teaching SOLIDWORKS and SOLIDWORKS Simulation in an academic setting. This guide offers a competency-based approach to teaching 3D design concepts, analysis and techniques.

Qualified schools on subscription have access to the eBook at no cost to students. Contact your SOLIDWORKS Value Added Reseller to obtain access.

SOLIDWORKS Tutorials

The *SOLIDWORKS Education Edition - Fundamentals of 3D Design and Simulation* manual also supplements the SOLIDWORKS Tutorials.

Accessing the SOLIDWORKS Tutorials

To start the SOLIDWORKS Tutorials, click **Help, SOLIDWORKS Tutorials**. The SOLIDWORKS window is resized and a second window appears next to it with a list of the available tutorials. There are over 40 lessons in the SOLIDWORKS Tutorials. As you move the pointer over the links, an illustration of the tutorial will appear at the bottom of the window. Click the desired link to start that tutorial.

TIP: When you use SOLIDWORKS Simulation to perform analysis, click **Help**, **SOLIDWORKS Simulation**, **Tutorials** to access over 50 lessons and over 80 verification problems. Click **Tools**, **Add-ins** to activate SOLIDWORKS Simulation, SOLIDWORKS Motion, and SOLIDWORKS Flow Simulation.

Conventions

Set your screen resolution to 1280x1024 for optimal viewing of the tutorials.

The following icons appear in the tutorials:

Next Moves to the next screen in the tutorial.

Represents a note or tip. It is not a link; the information is below the icon. Notes and tips provide time-saving steps and helpful hints.

You can click most buttons that appear in the lessons to flash the corresponding SOLIDWORKS button.

Open File or **Set this option** automatically opens the file or sets the option.

A closer look at... links to more information about a topic. Although not required to complete the tutorial, it offers more detail on the subject.

Why did I... links to more information about a procedure, and the reasons for the method given. This information is not required to complete the tutorial.

Show me... demonstrates with a video.

Printing the SOLIDWORKS Tutorials

If you like, you can print the SOLIDWORKS Tutorials by following this procedure:

1. On the tutorial navigation toolbar, click **Show**.

   This displays the table of contents for the SOLIDWORKS Tutorials.

2. Right-click the book representing the lesson you wish to print and select **Print...** from the shortcut menu.

   The **Print Topics** dialog box appears.

3. Select **Print the selected heading and all subtopics**, and click **OK**.
4. Repeat this process for each lesson that you want to print.

My SOLIDWORKS

My.SolidWorks.com is a community website to share, connect, and learn everything about SOLIDWORKS. My SOLIDWORKS learning contains additional video lessons and individual learning paths for your students.

Certification Exams

The Certified SOLIDWORKS Associate(CSWA) - Academic program provides free certification exams for you or your students in a proctored setting. Achieving CSWA proves the fundamentals of engineering design competency. Employers verify students job ready credentials through our online virtual tester. Schools that provide two or more courses in SOLIDWORKS-based instruction can also apply to be a Certified SOLIDWORKS Professional(CSWP) - Academic Provider.

More information and to apply can be found at `www.solidworks.com/cswa-academic`.

Training Files

A complete set of the various files used throughout the course can be downloaded from the following website:
`www.solidworks.com/EDU_Fundamentals3DDesignSim`

The files are organized by lesson number. The `Case Study` folder within each lesson contains the files you need when presenting the lessons. The `Exercises` folder contains any files that are required for doing the laboratory exercises.

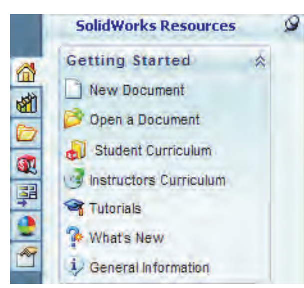
Educator Resources link

The **Instructors Curriculum** link on the **SOLIDWORKS Resources** tab of the Task Pane includes substantial supporting materials to aid in your course presentation. Accessing this page requires a login account for the SOLIDWORKS Customer Portal. These supporting materials afford you flexibility in scope, depth, and presentation.

1. Start SOLIDWORKS.

   Using the **Start** menu, start the SOLIDWORKS application.

2. SOLIDWORKS Content.



   Click **SOLIDWORKS Resources** to open the SOLIDWORKS Resources Task Pane.

   Click on the **Instructors Curriculum** link which will take you to the SOLIDWORKS Customer Portal web page.

Prerequisites

Students attending this course are expected to have the following:

- Mechanical design experience.
- Experience with the Windows® operating system.
- Completed the online tutorials that are integrated in the SOLIDWORKS software. You can access the online tutorials by clicking **Help, Online Tutorial**.

Course Design Philosophy

This course is designed around a process- or task-based approach to training. A process-based training course emphasizes the processes and procedures you follow to complete a particular task. By utilizing case studies to illustrate these processes, you learn the necessary commands, options and menus in the context of completing a task.

A Note About Dimensions

The drawings and dimensions given in the lab exercises are not intended to reflect any particular drafting standard. In fact, sometimes dimensions are given in a fashion that would never be considered acceptable in industry. The reason for this is the labs are designed to encourage you to apply the information covered in class and to employ and reinforce certain techniques in modeling. As a result, the drawings and dimensions in the exercises are done in a way that complements this objective.

Conventions Used in this Book

This manual uses the following typographic conventions:

Convention	Meaning
Bold Sans Serif	SOLIDWORKS commands and options appear in this style. For example, **Features > Extruded Cut** means click the **Extruded Cut** icon on the **Features** tab of the CommandManager.
`Typewriter`	Feature names and file names appear in this style. For example, `Sketch1`.
17 Do this step	Double lines precede and follow sections of the procedures. This provides separation between the steps of the procedure and large blocks of explanatory text. The steps themselves are numbered in sans serif bold.

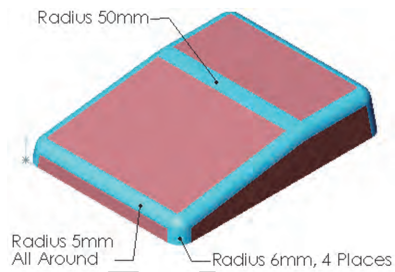
Windows

The screen shots in this manual were made using the SOLIDWORKS software running a mixture of Windows® 7 and Windows 10. You may notice slight differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

Use of Color

The SOLIDWORKS user interface makes extensive use of color to highlight selected geometry and to provide you with visual feedback. This greatly increases the intuitiveness and ease of use of the SOLIDWORKS software. To take maximum advantage of this, the training manuals are printed in full color.

Also, in many cases, we have used additional color in the illustrations to communicate concepts, identify features, and otherwise convey important information. For example, we might show the result of a filleting operation with the fillets in a different color even though, by default, the SOLIDWORKS software would not display the results in that way.



Graphics and Graphics Cards

The SOLIDWORKS software sets a new standard with best-in-class graphics. The combination of a highly reflective material and the realism of **RealView Graphics** is an effective tool for evaluating the quality of advanced part models and surfaces.

RealView Graphics is hardware (graphics card) support of advanced shading in real time. For example, if you rotate a part, it retains its rendered appearance throughout the rotation.

Color Schemes

Out of the box, the SOLIDWORKS software provides several predefined color schemes that control, among other things, the colors used for highlighted items, selected items, sketch relation symbols, and shaded previews of features.

We have not used the same color scheme for every case study and exercise because some colors are more visible and clear than others when used with different colored parts.

In addition, we have changed the viewport background to plain white so that the illustrations reproduce better on white paper.

As a result, because the color settings on your computer may be different than the ones used by the authors of this book, the images you see on your screen may not exactly match those in the book.

User Interface Appearance

Throughout the development of the software, there have been some cosmetic User Interface changes, intended to improve visibility, that do not affect the function of the software. As a policy, dialog images in the manuals which exhibit no functional change from the previous version are not replaced. As such, you may see a mixture of current and “old” UI dialogs and color schemes.

Lesson 1
SOLIDWORKS Basics and the User Interface

Upon successful completion of this lesson, you will be able to:

- Describe the key characteristics of a feature-based, parametric solid modeler.
- Distinguish between sketched and applied features.
- Identify the principal components of the SOLIDWORKS user interface.
- Explain how different dimensioning methodologies convey different design intents.

What is the SOLIDWORKS Software?

SOLIDWORKS mechanical design automation software is a *feature-based, parametric solid modeling* design tool which takes advantage of the easy to learn Windows graphical user interface. You can create *fully associative* 3D solid models with or without *constraints* while utilizing automatic or user defined relations to capture *design intent*.

The italicized terms in the previous paragraph mean:

- **Feature-based**
  Just as an assembly is made up of a number of individual piece parts, a SOLIDWORKS model is also made up of individual constituent elements. These elements are called features.

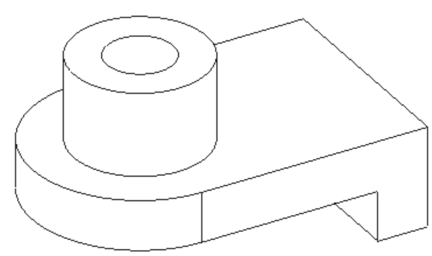
  When you create a model using the SOLIDWORKS software, you work with intelligent, easy to understand geometric features such as bosses, cuts, holes, ribs, fillets, chamfers, and drafts. As the features are created they are applied directly to the work piece.

  Features can be classified as either sketched or applied.

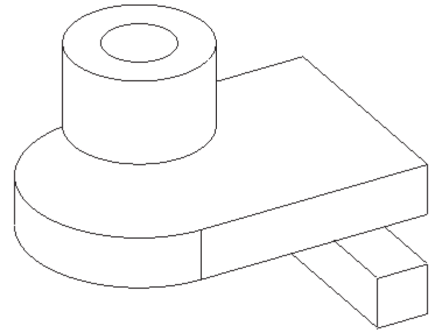
  - Sketched Features: Based upon a 2D sketch. Generally that sketch is transformed into a solid by extrusion, rotation, sweeping or lofting.
  - Applied Features: Created directly on the solid model. Fillets and chamfers are examples of this type of feature.

  The SOLIDWORKS software graphically shows you the feature-based structure of your model in a special window called the FeatureManager® design tree. The FeatureManager design tree not only shows you the sequence in which the features were created, it gives you easy access to all the underlying associated information. You will learn more about the FeatureManager design tree throughout this course.

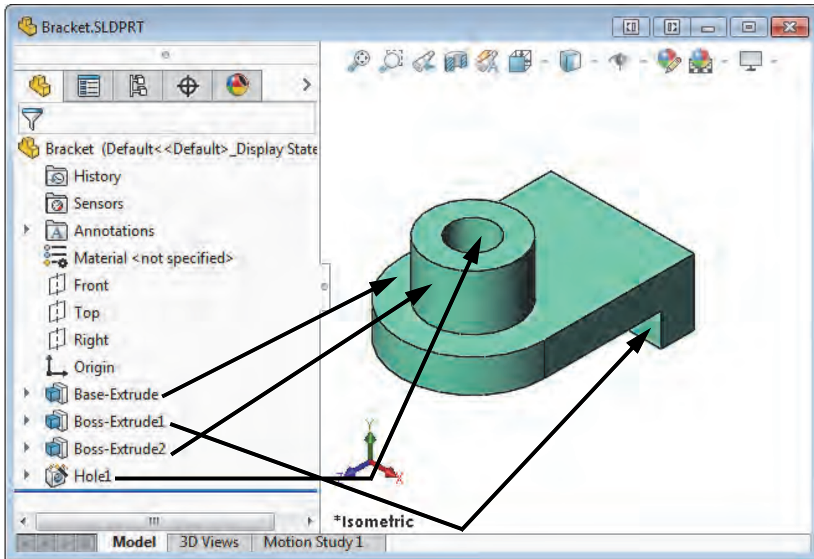
  To illustrate the concept of feature-based modeling, consider the part shown at the right:



  This part can be visualized as a collection of several different features – some of which add material, like the cylindrical boss, and some which remove material, like the blind hole.



If we were to map the individual features to their corresponding listing in the FeatureManager design tree, it would look like this:



- **Parametric**
  The dimensions and relations used to create a feature are captured and stored in the model. This not only enables you to capture your design intent, it also enables you to quickly and easily make changes to the model.
  - Driving Dimensions: These are the dimensions used when creating a feature. They include the dimensions associated with the sketch geometry, as well as those associated with the feature itself. A simple example of this would be a feature like a cylindrical boss. The diameter of the boss is controlled by the diameter of the sketched circle. The height of the boss is controlled by the depth to which that circle was extruded when the feature was made.
  - Relations: These include such information as parallelism, tangency, and concentricity. Historically, this type of information has been communicated on drawings via feature control symbols. By capturing this in the sketch, SOLIDWORKS enables you to fully capture your design intent up front, in the model.
- **Solid Modeling**
  A solid model is the most complete type of geometric model used in CAD systems. It contains all the wire frame and surface geometry necessary to fully describe the edges and faces of the model. In addition to the geometric information, it has the information called topology that relates the geometry together. An example of topology would be which faces (surfaces) meet at which edge (curve). This intelligence makes operations such a filleting as easy as selecting an edge and specifying a radius.

- **Fully Associative**
  A SOLIDWORKS model is fully associative to the drawings and assemblies that reference it. Changes to the model are automatically reflected in the associated drawings and assemblies. Likewise, you can make changes in the context of the drawing or assembly and know that those changes will be reflected back in the model.
- **Constraints**
  Geometric relationships such as parallel, perpendicular, horizontal, vertical, concentric, and coincident are just some of the constraints supported in SOLIDWORKS. In addition, equations can be used to establish mathematical relationships among parameters. By using constraints and equations, you can guarantee that design concepts such as through holes or equal radii are captured and maintained.
- **Design Intent**
  The final italicized term is design intent. This subject is worthy of its own section, as follows.

Design Intent

In order to use a parametric modeler like SOLIDWORKS efficiently, you must consider the design intent before modeling. Design intent is your plan as to how the model should behave when it is changed. The way in which the model is created governs how it will be changed. Several factors contribute to how you capture design intent:

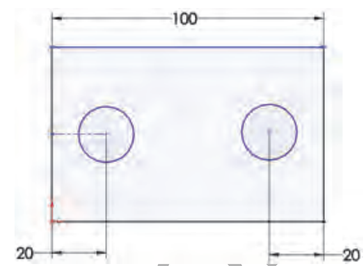
- **Automatic (sketch) Relations**
  Based on how geometry is sketched, these relations can provide common geometric relationships between objects such as parallel, perpendicular, horizontal, and vertical.
- **Equations**
  Used to relate dimensions algebraically, they provide an external way to force changes.
- **Added Relations**
  Added to the model as it is created, relations provide another way to connect related geometry. Some common relations are concentric, tangent, coincident, and collinear.
- **Dimensioning**
  Consider your design intent when applying dimensions to a sketch. What are the dimensions that should drive the design? What values are known? Which are important for the production of the model? The way dimensions are applied to the model will determine how the geometry will change if modifications are made.

Consider the design intent in the following examples.

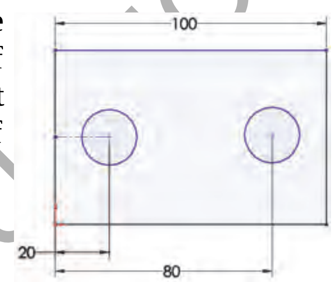
Examples of Design Intent

The design intent of each sketch below is slightly different. How will the geometry be affected if the overall plate width, **100mm**, is changed?

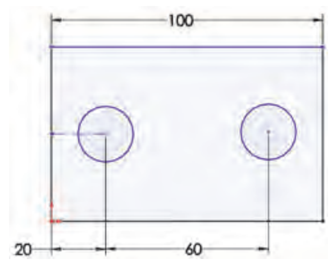
A sketch dimensioned like this will keep the holes **20mm** from each end regardless of the width of the plate.



Baseline dimensions like this will keep the holes positioned relative to the left edge of the plate. The positions of the holes are not affected by changes in the overall width of the plate.

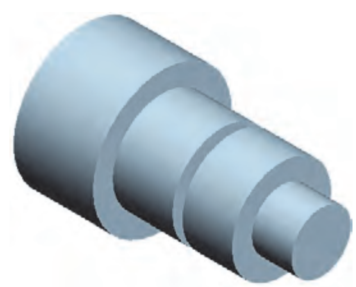


Dimensioning from the edge and from center to center will maintain the distance between the hole centers and allow it to be changed that way.



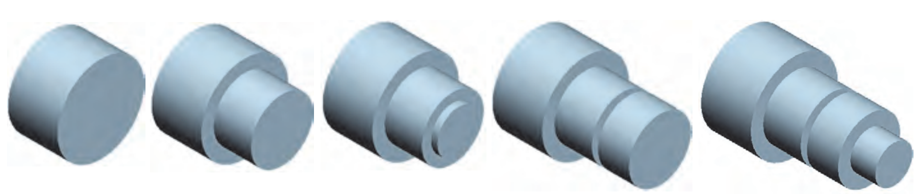
How Features Affect Design Intent

Design intent is affected by more than just how a sketch is dimensioned. The choice of features and the modeling methodology are also important. For example, consider the case of a simple stepped shaft as shown at the right. There are several ways a part like this could be built and each way creates a part that is geometrically identical.



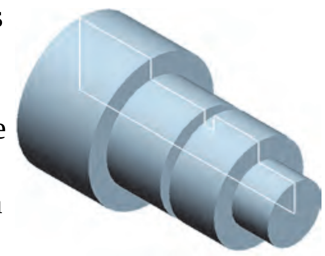
The "Layer Cake" Approach

The layer cake approach builds the part one piece at a time, adding each layer, or feature, onto the previous one, like this:



Changing the thickness of one layer has a ripple effect, changing the position of all the other layers that were created after it.

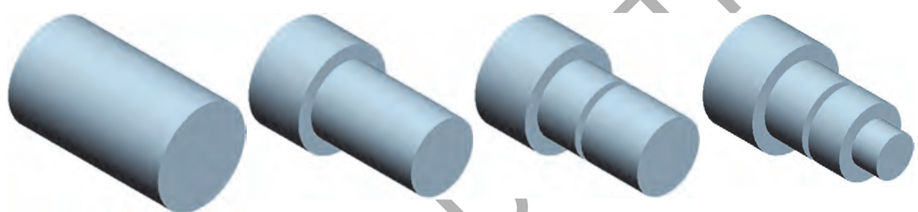
The “Potter’s Wheel” Approach



The potter’s wheel approach builds the part as a single, revolved feature. A single sketch representing the cross section includes all the information and dimensions necessary to make the part as one feature. While this approach may seem very efficient, having all the design information contained within a single feature limits flexibility and can make changes awkward.

The Manufacturing Approach

The manufacturing approach to modeling mimics the way the part would be manufactured. For example, if this stepped shaft was turned on a lathe, you would start with a piece of bar stock and remove material using a series of cuts.



There is not really a right or wrong answer when trying to determine which approach to use. SOLIDWORKS allows for great flexibility and making changes to models is relatively easy. But creating models with design intent in mind will result in well built documents that are easily modifiable and well suited for re-use, making your job easier.

File References

SOLIDWORKS creates files that are compound documents that contain elements from other files. File references are created by linking files rather than duplicating information in multiple files.

Referenced files do not have to be stored with the document that references them. In most practical applications, the referenced documents are stored in multiple locations on the computer or network. SOLIDWORKS provides several tools to determine the references that exist and their location.

Object Linking and Embedding (OLE)

In the Windows environment, information sharing between files can be handled either by linking or embedding the information.

The main differences between linked objects and embedded objects are where the data is stored and how you update the data after you place it in the destination file.

Linked Objects

When an object is linked, information is updated only if the source file is modified. Linked data is stored in the source file. The destination file stores only the location of the source file (an external reference), and it displays a representation of the linked data.

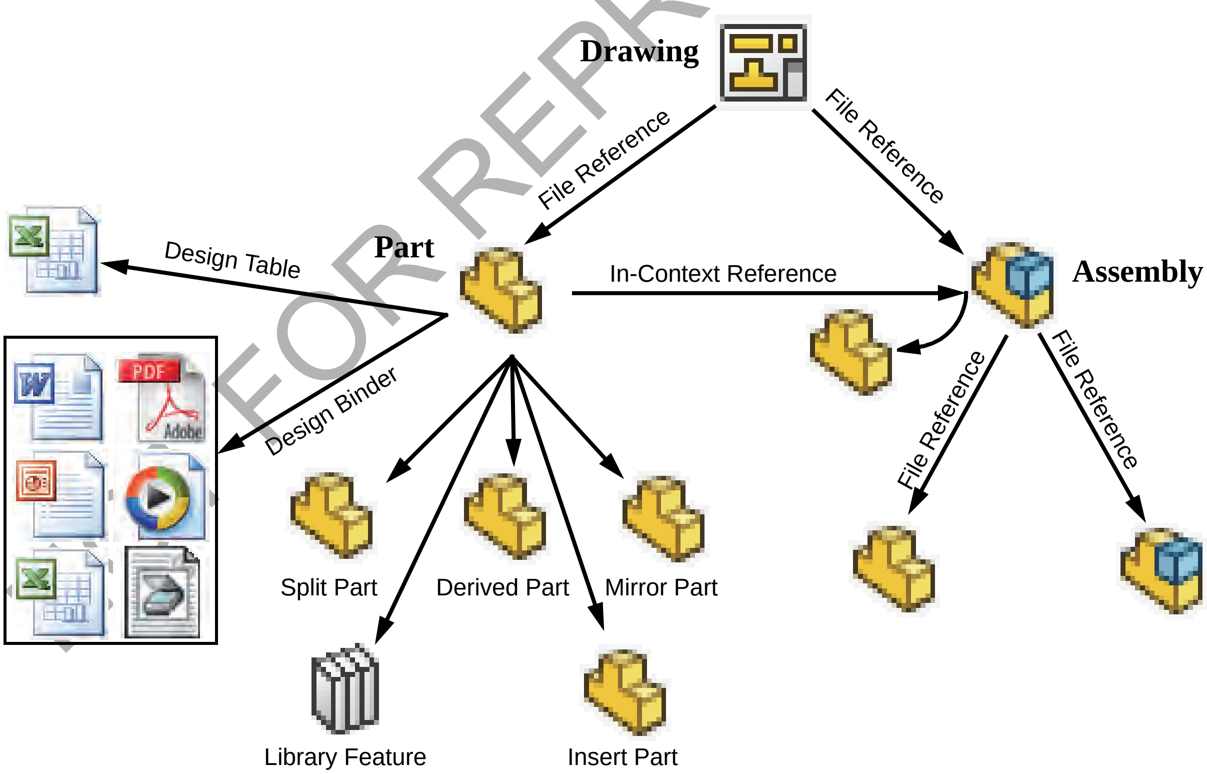
Linking is also useful when you want to include information that is maintained independently, such as data collected by a different department.

Embedded Objects

When you embed an object, information in the destination file doesn't change if you modify the source file. Embedded objects become part of the destination file and, once inserted, are no longer part of the source file.

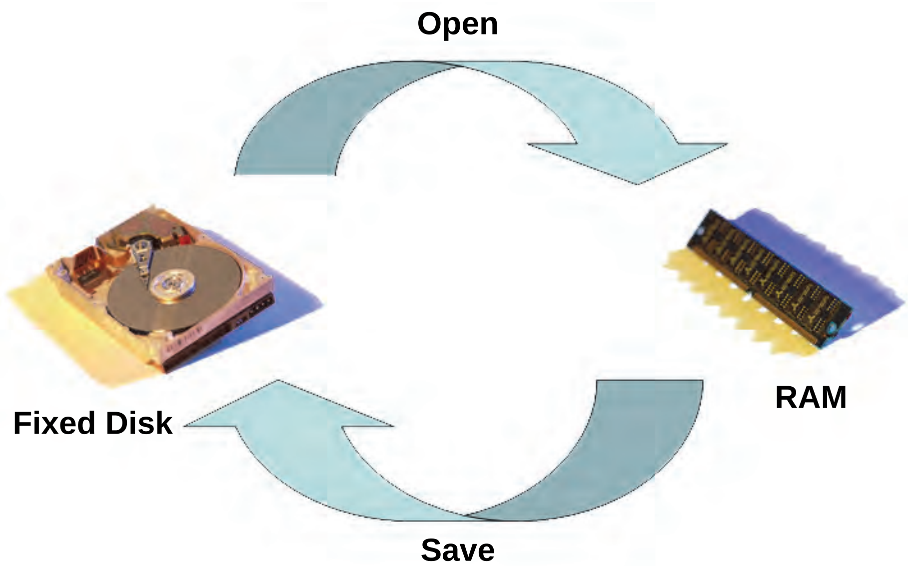
File Reference Example

The many different types of external references created by SOLIDWORKS are shown in the following graphic. Some of the references can be linked or embedded.



Opening Files

SOLIDWORKS is a RAM-resident CAD system. Whenever a file is opened, it is copied from its storage location to the computer's Random Access Memory or RAM. All changes to the file are made to the copy in RAM and only written back to the original files during a **Save** operation.



Computer Memory

To better understand where files are stored and which copy of the file we are working on, it is important to differentiate between the two main types of computer memory.

Random Access Memory

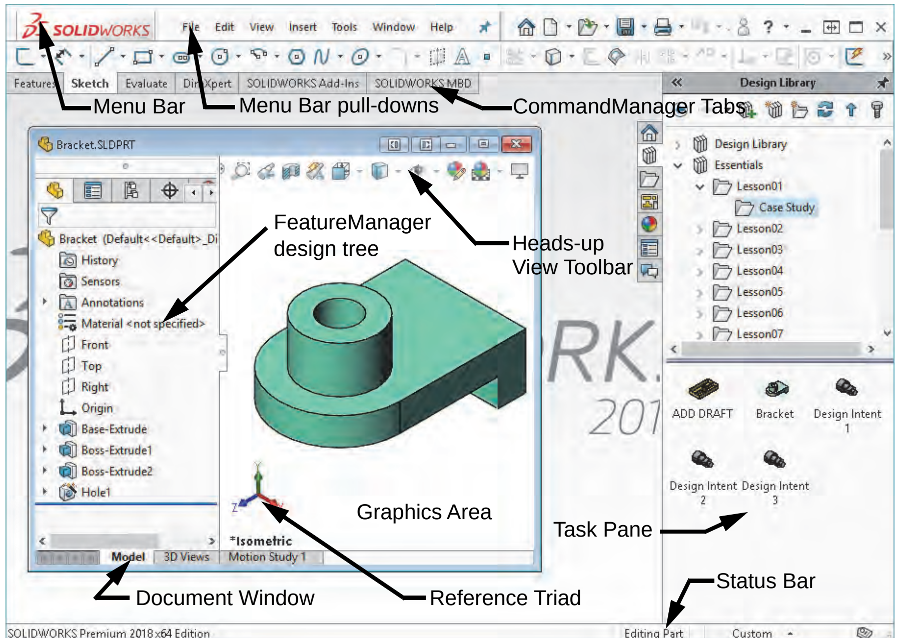
Random Access Memory (RAM) is the volatile memory of the computer. This memory only stores information when the computer is operating. When the computer is turned off, any information in RAM is lost.

Fixed Memory

Fixed memory is all the non-volatile memory. This includes computer hard drives, flash drives and CD/DVD drives. Fixed memory holds its information even when the computer is not running.

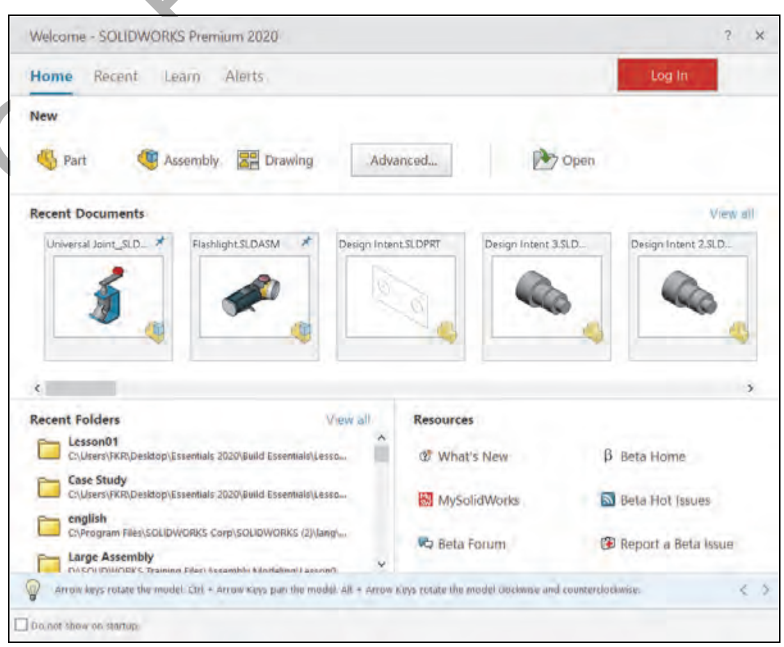
The SOLIDWORKS User Interface

The SOLIDWORKS user interface is a native Windows interface, and as such behaves in the same manner as other Windows applications. Some of the more important aspects of the interface are identified below.



Welcome Dialog Box

The **Welcome** dialog box opens with SOLIDWORKS to provide convenient ways to create new documents, open existing documents, and access SOLIDWORKS resources and news.

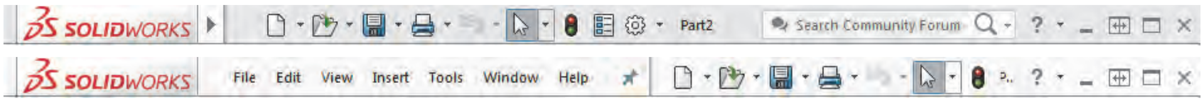


Note

This dialog box can also be set to **Do not show on startup**.

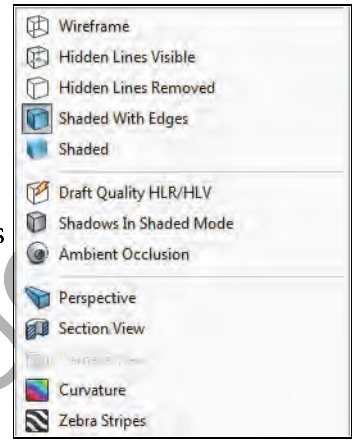
Pull-down Menus

The Pull-down menus provide access to many of the commands that the SOLIDWORKS software offers. Float over the right facing arrow to access the menus. Click the pushpin to keep the menus open.



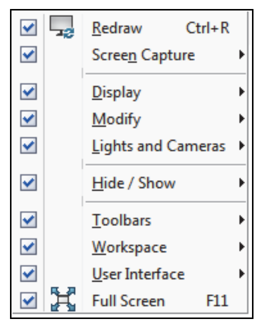
When a menu item has a right-pointing arrow like this: Display ▸, it means that there is a sub-menu associated with that choice.

When a menu item is followed by ellipses like this: Orientation... SpaceBar, it means that the option opens a dialog box with additional choices or information.



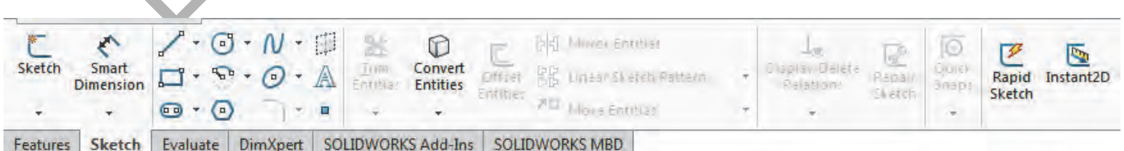
Customizing Pull-down Menus

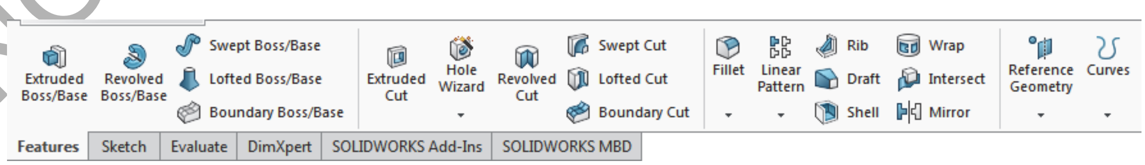
When the **Customize Menu** item is selected, each item appears with a check box. Clearing the check box removes the associated item from the menu.



Using the Command Manager

The **CommandManager** is a set of icons divided into tabs that are geared towards specific tasks. For example, the part version has several tabs to access commands related to features, sketches, and so on.



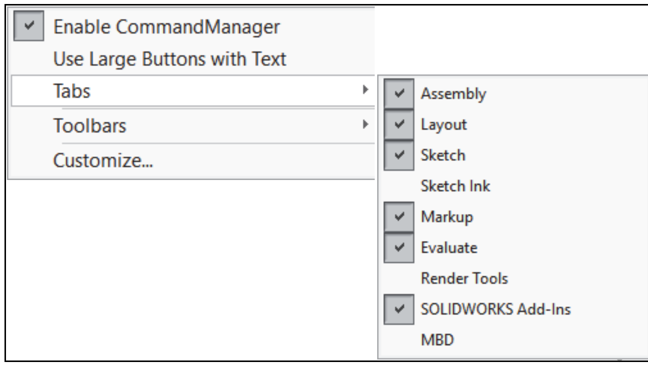


Note

The CommandManager can be displayed with or without text on the buttons. These images show the **Use Large Buttons with Text** option.

Adding and Removing CommandManager Tabs

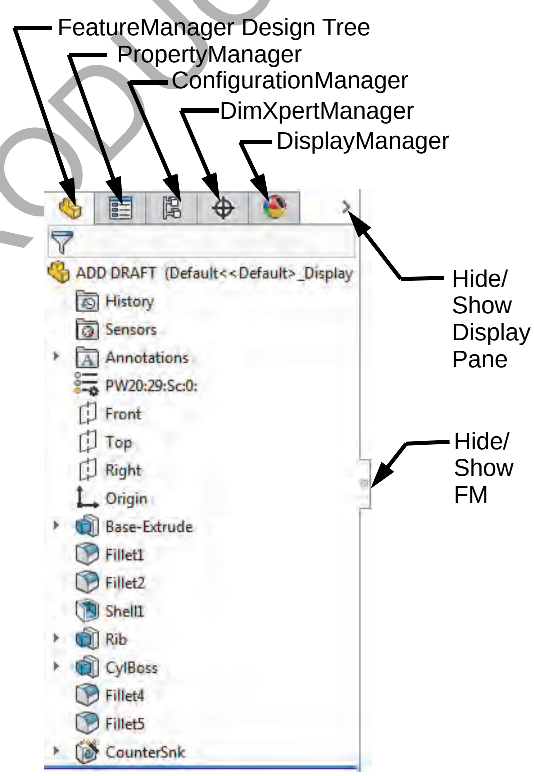
The default settings display multiple CommandManager tabs for a part file. Others can be added or removed by right-clicking on any tab, clicking **Tabs**, and clicking or clearing the tab by name.



There are different sets of tabs for part, assembly and drawing files.

FeatureManager Design Tree

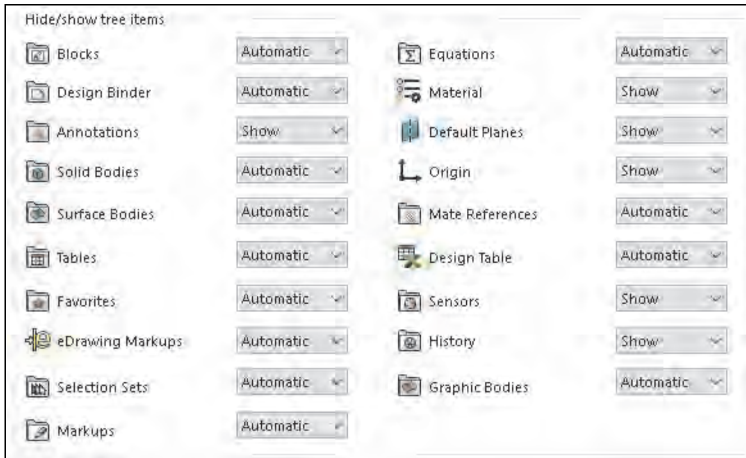
The FeatureManager design tree is a unique part of the SOLIDWORKS software that visually displays all the features in a part or assembly. As features are created they are added to the FeatureManager design tree. As a result, the FeatureManager design tree represents the chronological sequence of modeling operations. The FeatureManager design tree also allows access to the editing of the features (objects) that it contains.



Show and Hide FeatureManager Items

Many FeatureManager items (icons and folders) are hidden by default. In the image above, only the History, Sensors and Annotations folders are shown.

Click **Tools**, **Options**, **System Options**, and **FeatureManager** to control their visibility using one of the three settings explained below.



- **Automatic** - Hide the item when it is empty.
- **Hide** - Hide the item at all times.
- **Show** - Show the item at all times.

Tip

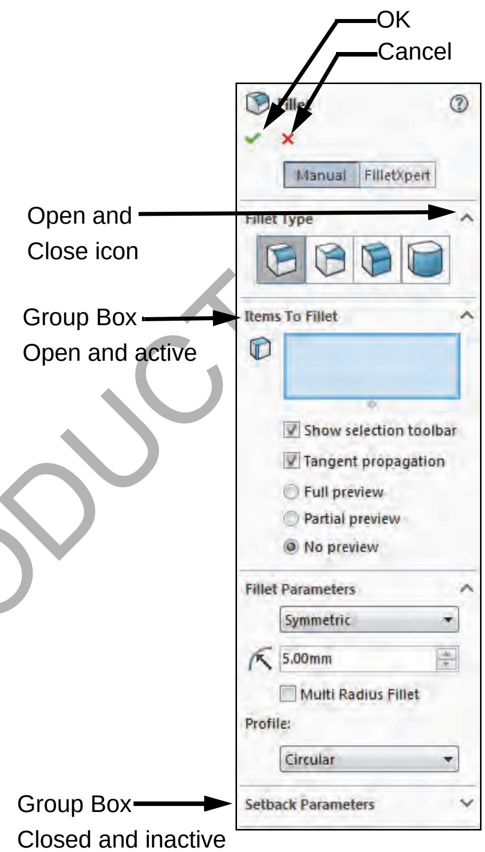
The CommandManager or PropertyManager can be dragged and docked on the top, side or outside of the SOLIDWORKS window or to a different monitor.

PropertyManager

Many SOLIDWORKS commands are executed through the PropertyManager. The PropertyManager occupies the same screen position as the FeatureManager design tree and replaces it when it is in use.

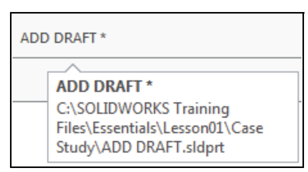
The top row buttons contain the standard **OK** and **Cancel** buttons.

Below the top row of buttons are one or more **Group Boxes** that contain related options. They can be opened (expanded) or closed (collapsed) and in many cases made active or inactive.



Full Path Name

The full path name of the document can be seen as a tool tip when floating the cursor over the file name.



Selection Breadcrumbs

Selection Breadcrumbs show the hierarchy of objects based on a selected piece of geometry. For example, selecting a face can lead to a series of objects including the feature, sold body, component, subassembly, and finally to the top level assembly.

It also leads to the sketch of the feature and the mates attached to that component.

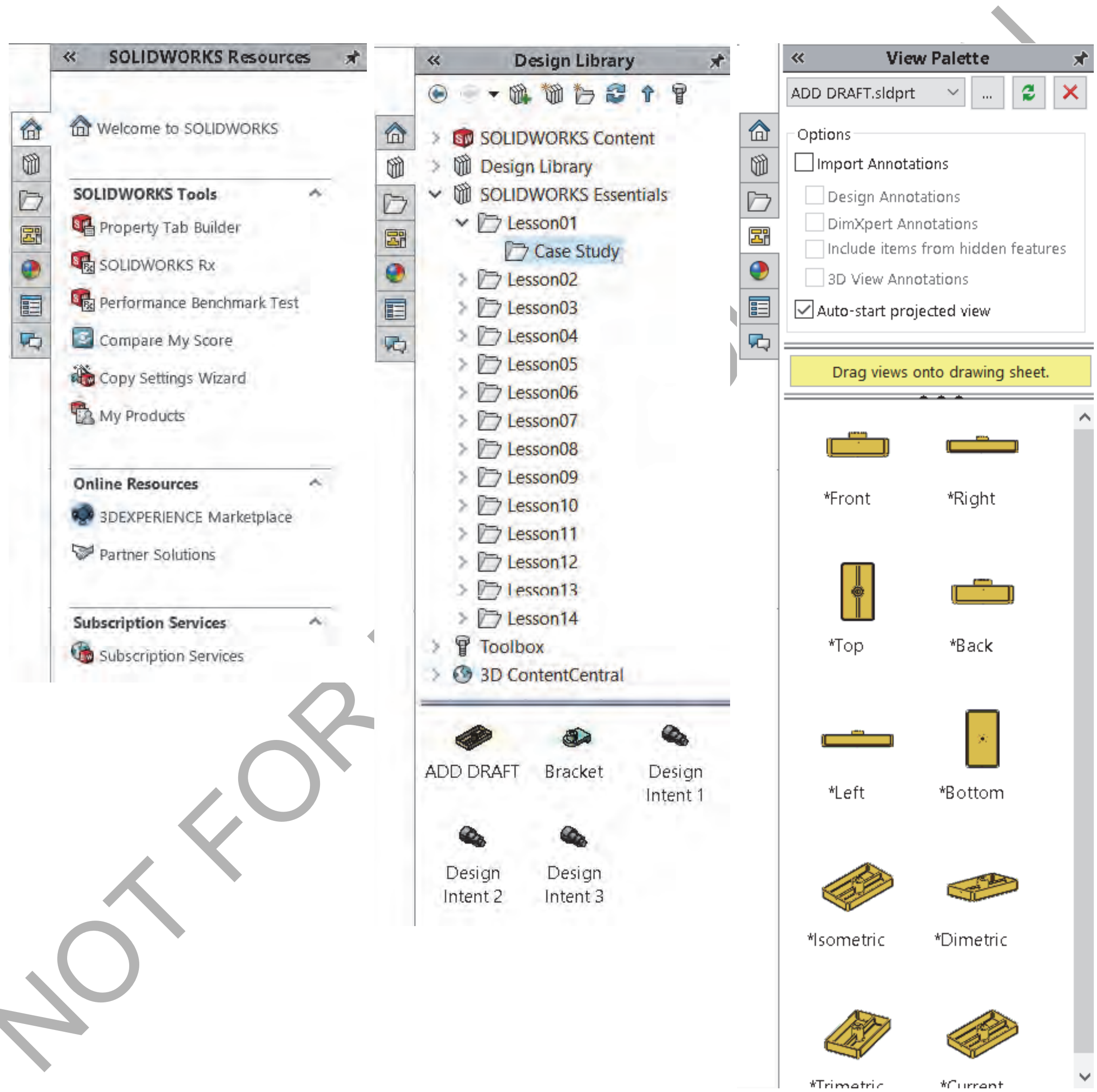
These visual objects can also be used for access. Right-clicking on the boss feature offers several editing tools including **Edit Feature** and **Hide**.

Note

These objects and tools will be discussed in later lessons.

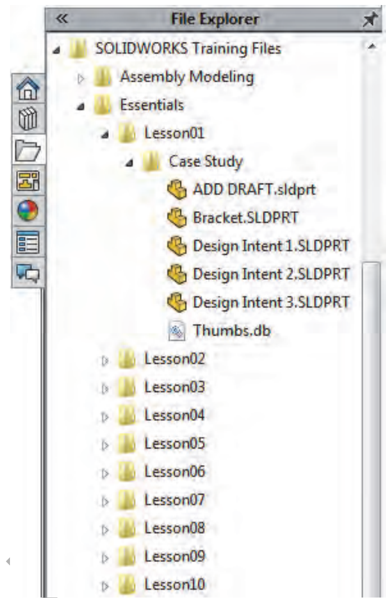
Task Pane

The **Task Pane** window contains **SOLIDWORKS Resources**, **Design Library**, **File Explorer**, **View Palette**, **Appearances, Scenes, and Decals**, **Custom Properties**, and the **SOLIDWORKS Forum** options. The window appears on the right by default but it can be moved and resized. It can be opened/ closed, tacked or moved from its default position on the right side of the interface.



Opening Labs with the File Explorer

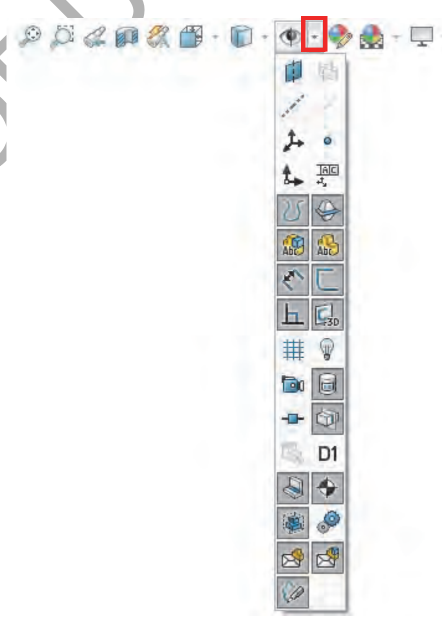
You can open parts and assemblies required for lab exercises using the File Explorer.



- Open the **Task Pane**.
- Click **File Explorer** .
- Expand the `Essentials` folder used for the class files. It should be found under the `SOLIDWORKS Training Files` folder.
- Expand the lesson folder (`Lesson01` for example) followed by either the `Case Study` or `Exercises` folder.
- Double-click a part or assembly file to open it.

Heads-up View Toolbar

The **Heads-up View** toolbar is a transparent toolbar that contains many common view manipulation commands. Many of the icons (such as the **Hide/ Show Items** icon shown) are **Flyout Tool** buttons that contain other options. These flyouts contain a small down arrow to access the other commands.



Unselectable Icons

At times you will notice commands, icons, and menu options that are grayed out and unselectable. This is because you may not be working in the proper environment to access those options. For example, if you are working in a sketch (**Edit Sketch** mode), you have full access to all the sketch tools. However, you cannot select the icons such as fillet or chamfer on the Features tab of the CommandManager. This design helps the inexperienced user by limiting the choices to only those that are appropriate.

To Preselect or Not?

As a rule, the SOLIDWORKS software does not require you to preselect objects before opening a menu or dialog box. For example, if you want to add some fillets to the edges of your model, you have complete freedom – you can select the edges first and then click the **Fillet** tool or you can click the **Fillet** tool and then select the edges. The choice is yours.

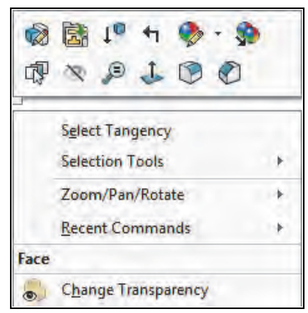
Mouse Buttons

The left, right and middle mouse buttons have distinct meanings in SOLIDWORKS.

- **Left**
  Select objects such as geometry, menus buttons, and objects in the FeatureManager design tree.
- **Right**
  Activates a context sensitive shortcut menu. The contents of the menu differ depending on what object the cursor is over. These menus also represent shortcuts to frequently used commands.

Shortcut Menu

At the top of the **Shortcut Menu** is the **Context Toolbar**. It contains some of the most commonly used commands in icon form.



Below it is the pull-down menu. It contains other commands that are available in the context of the selection, in this example a face.

Note

The Context toolbar will also become available as you make selections with the left mouse button. It provides quick access to common commands.

- **Middle**
  Dynamically rotates, pans or zooms a part or assembly. Pans a drawing.

Keyboard Shortcuts

Some menu items indicate a keyboard shortcut like this:

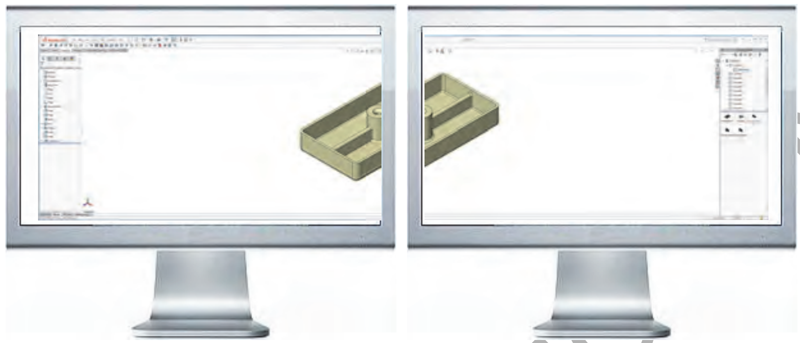
SOLIDWORKS conforms to standard Windows conventions for such shortcuts as **Ctrl+O** for **File, Open**; **Ctrl+S** for **File, Save**; **Ctrl+Z** for **Edit, Undo** and so on. In addition, you can customize SOLIDWORKS by creating your own shortcuts.

Multiple Monitor Displays

SOLIDWORKS can take advantage of multiple monitor displays to span monitors and to move document windows or menus to a different monitor.

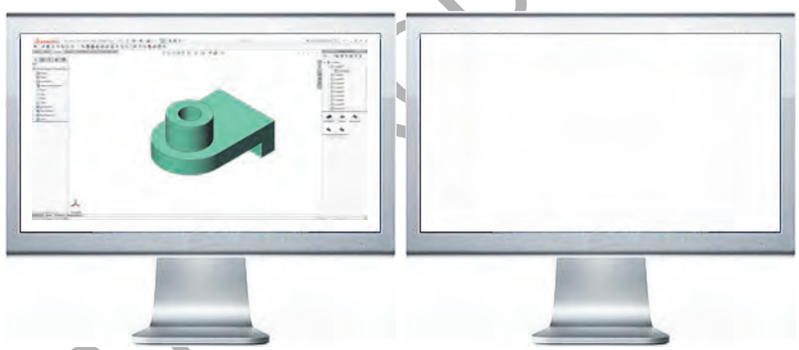
Spanning Monitors

Click **Span Displays** on the top bar of the SOLIDWORKS window to stretch the display across both monitors.



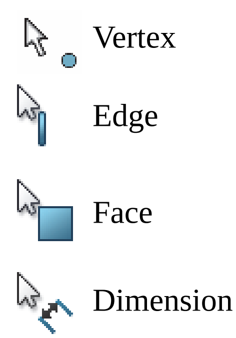
Fitting to a Monitor

Click either **Click to Tile Left** or **Click to Tile Right** on the top bar of a document to fit it to the left or right monitor.



System Feedback

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 will come in the form of symbols, riding next to the cursor. The illustration at the right shows some of the symbols: vertex, edge, face and dimension.

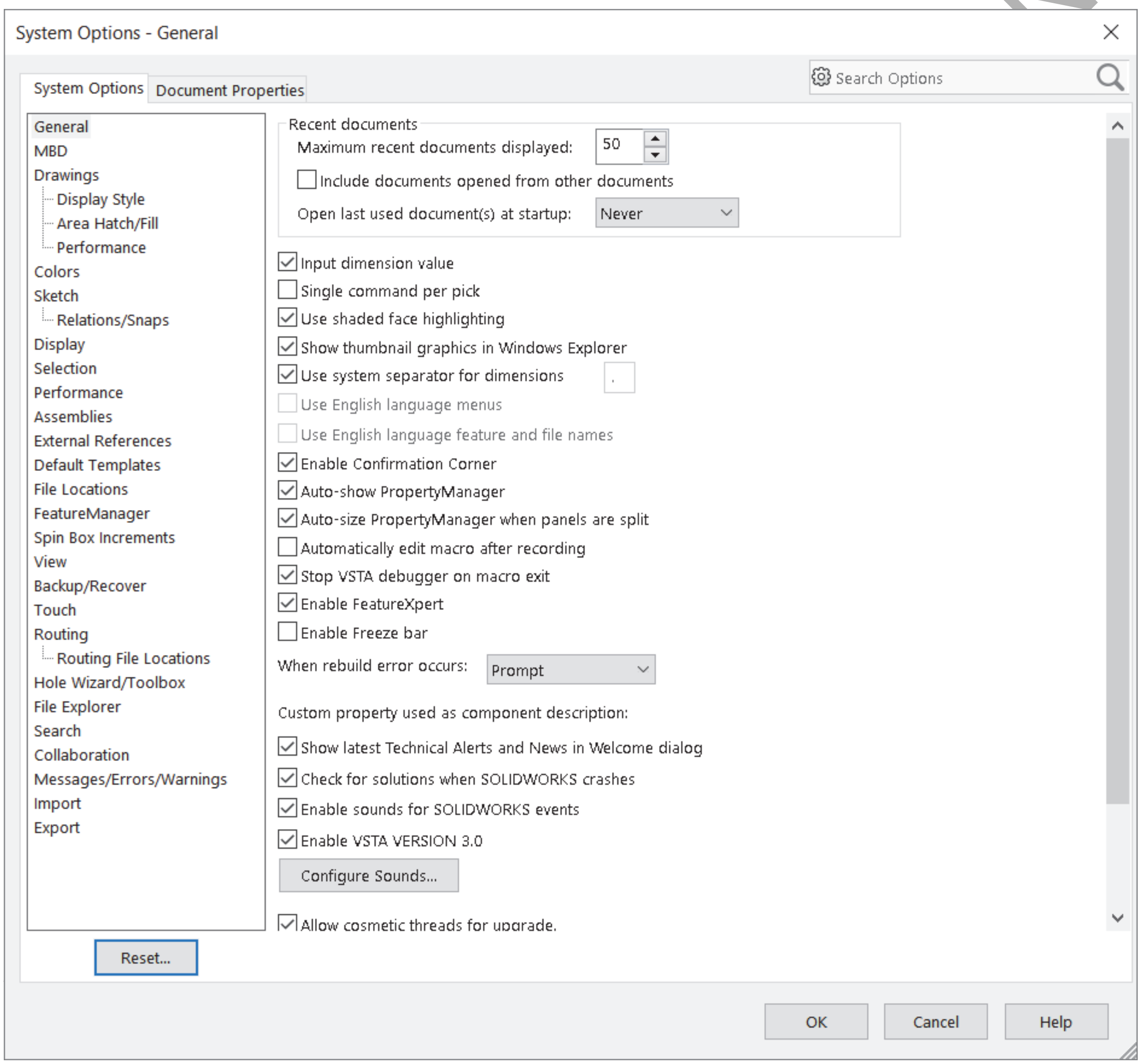


Options

Located on the **Tools** menu, the **Options** dialog box enables you to customize the SOLIDWORKS software to reflect such things as your company's drafting standards as well as your individual preferences and work environment.

Tip

Use the search bar in the upper right of the **Options** dialog box to find system options and document properties. Type the label of the check box, radio button, or other option to locate the page where the option resides.



Customization

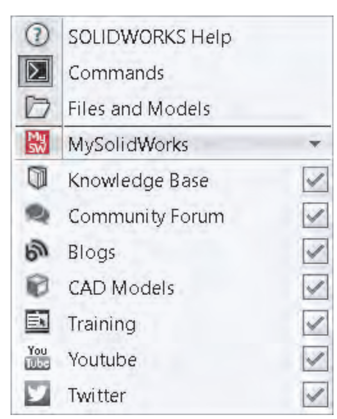
You have several levels of customization. They are:

- **System options**
  The options grouped under the heading **System Options** are saved on your system and affect every document you open in your SOLIDWORKS session. System settings allow you to control and customize your work environment. For example, you might like working with colored viewport background. I don't. Since this is a system setting, parts or assemblies opened on your system would have a colored viewport. The same files opened on my system would not.
- **Document properties**
  These settings are applied to the individual document. For example, units, drafting standards, and material properties (density) are all document settings. They are saved with the document and do not change, regardless of whose system the document is opened on.
- **Document templates**
  Document templates are pre-defined documents that were set up with certain specific settings. For example, you might want two different templates for parts. One with English settings such as ANSI drafting standards and inch units, and one with metric settings such as millimeters units and ISO drafting standards. You can set up as many different document templates as you need. They can be organized into different folders for easy access when opening new documents. You can create document templates for parts, assemblies, and drawings.
- **Object**
  Many times the properties of an individual object can be changed or edited. For example, you can change the default display of a dimension to suppress one or both extension lines, or you can change the color of a feature.

Search

The **Search** option can be used to find information from **SOLIDWORKS Help**, **Commands**, **Files and Models** on your system by searching for any part of the name (requires Windows Desktop Search engine), or **MySolidWorks** information. Search using this procedure:

- Choose which type of search you would like to do.
- Type a name or partial name into the **Search** box and click the search icon.
- For `my.solidworks.com` searches, click **MySolidWorks** and one or more sub options.



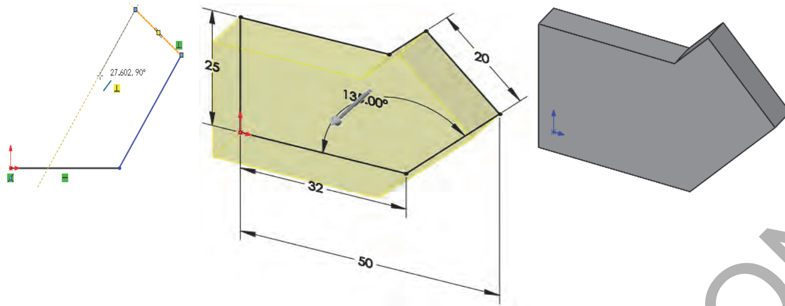
Lesson 2
Introduction to Sketching

Upon successful completion of this lesson, you will be able to:

- Create a new part.
- Insert a new sketch.
- Add sketch geometry.
- Establish sketch relations between pieces of geometry.
- Understand the state of the sketch.
- Extrude the sketch into a solid.

2D Sketching

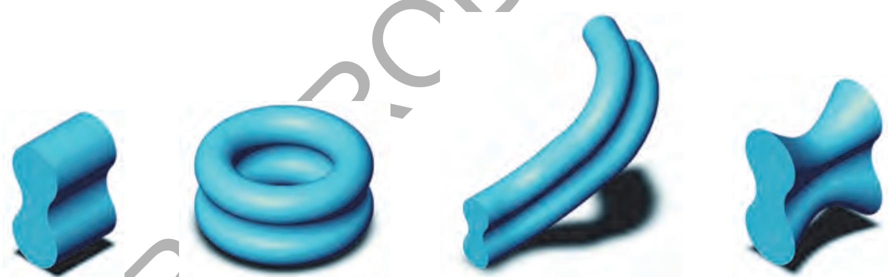
This lesson introduces 2D sketching, the basis of modeling in SOLIDWORKS.



Sketches are used for all sketched features in SOLIDWORKS including:

- Extrusions
- Revolves
- Sweeps
- Lofts

The illustration below shows how a given sketch can form the basis of several different types of features.



Extrude **Revolve** **Sweep** **Loft**

In this lesson, only extruded features will be covered. The others will be covered in detail in later lessons or courses.

Stages in the Process

Every sketch has several characteristics that contribute to its shape, size and orientation.

- **New part**
  New parts can be created in inch, millimeter or other units. Parts are used to create and hold the solid model.
- **Sketches**
  Sketches are collections of 2D geometry that are used to create solid features.
- **Sketch geometry**
  Types of 2D geometry such as lines, circles and rectangles that make up the sketch.
- **Sketch relations**
  Geometric relationships such as horizontal and vertical are applied to the sketch geometry. The relations restrict the movement of the entities.

- **State of the sketch**
  Each sketch has a status that determines whether it is ready to be used or not. The state can be fully-, under- or over defined.
- **Sketch tools**
  Tools can be used to modify the sketch geometry that has been created. This often involves trimming or extending entities.
- **Extruding the sketch**
  Extruding uses the 2D sketch to create a 3D solid feature.

Procedure

The process in this lesson includes sketching and extrusions. To begin with, a new part file is created.

Introducing: New Part

The **New** tool creates a new SOLIDWORKS document from a selection of part, assembly or drawing templates. There are several training templates in addition to the default ones.

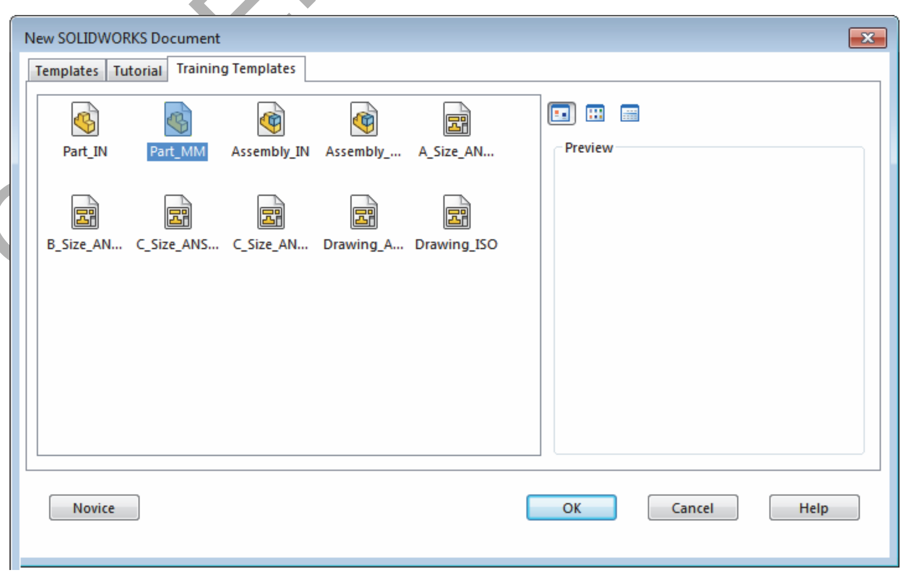
Where to Find It

- Menu Bar: **New**
- Menu: **File, New**
- Keyboard Shortcut: **Ctrl+N**

---

1 New part.

Click **New** and click the Part_MM template from the **Training Templates** tab on the **New SOLIDWORKS Document** dialog box, and click **OK**.



The part is created with the settings of the template including the units. This part template uses millimeters as the units. You can create and save any number of different templates, all with different settings.

Saving Files

Saving files writes the file information in RAM to a location on a fixed disk. SOLIDWORKS provides three options for saving files. Each has a different effect on file references.

Save

Copy the file in RAM to the fixed disk, leaving the copy in RAM open. If this file is being referenced by any open SOLIDWORKS files, there are no changes to the reference.

Where to Find It

- Menu Bar: **Save**
- Menu: **File, Save**
- Keyboard Shortcut: **Ctrl+S**

Save As

Copy the file in RAM to the fixed disk under a new name or file type, replacing the file in RAM with the new file. The old file in RAM is closed *without* saving. If this file is being referenced by any *open* SOLIDWORKS files, you should update the references to this new file.

Save As Copy to Disk

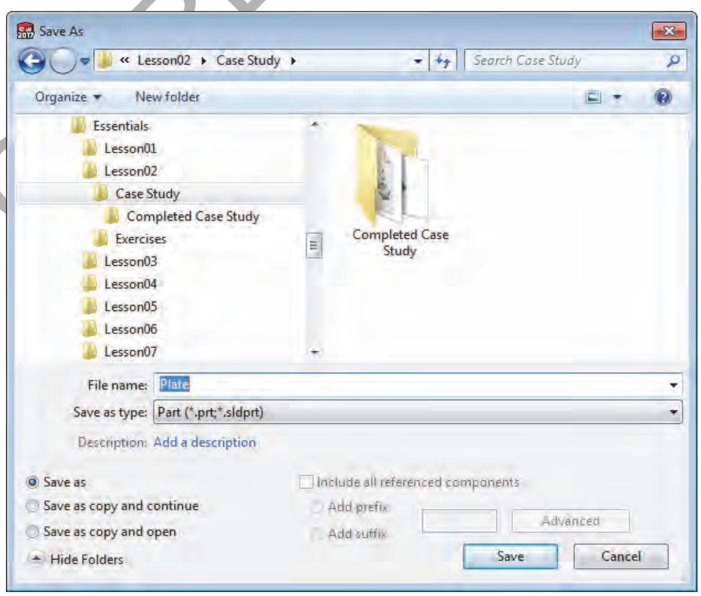
Copy the file in RAM to the fixed disk under a new name or file type, leaving the original in RAM open. If this file is being referenced by any open SOLIDWORKS files, you *should not* update the references to this new file.

Save As Copy and Open

Copy the file in RAM to the fixed disk under a new name or file type, leaving both the copy and the original open.

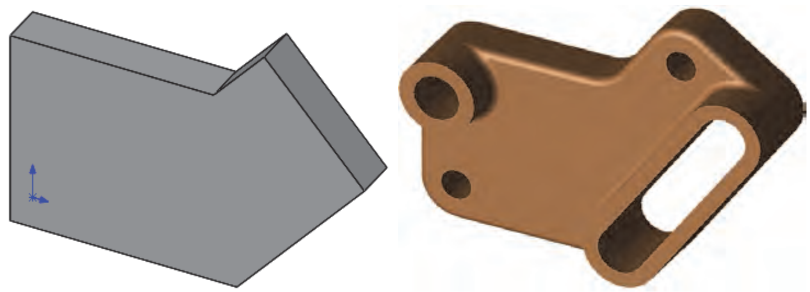
2 Filing a part.

Click **Save** and file the part under the name `Plate`. The extension, `*.sldprt`, is added automatically. Click **Save**.



What are We Going to Sketch?

The first feature of a part will be created in this section. That initial feature is just the first of many features needed to complete the part.



Sketching

Sketching is the act of creating a 2D profile comprised of wireframe geometry. Typical geometry types are lines, arcs, circles and ellipses. Sketching is dynamic, with feedback from the cursor to make it easier.

Default Planes

To create a sketch, you must choose a plane on which to sketch. The system provides three initial planes by default. They are `Front Plane`, `Top Plane`, and `Right Plane`.

Introducing: Sketch

When creating a new sketch, the **Sketch** tool opens the sketcher on the currently selected plane or planar face. You also use the **Sketch** tool to edit an existing sketch.

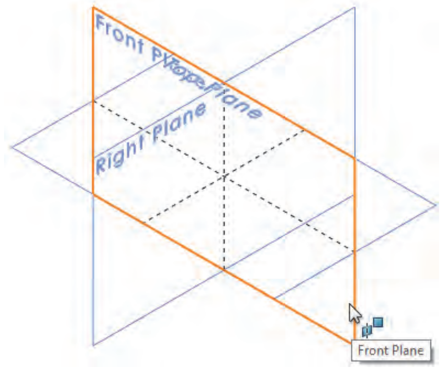
If you have not preselected a face or plane before activating the **Sketch** tool, the cursor appears indicating that you should select a face or plane.

Where to Find It

- CommandManager: **Sketch > Sketch**
- Menu: **Insert, Sketch**
- Shortcut Menu: Right-click a plane or planar face and click **Sketch**

3 Open a new sketch.

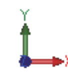
Click . This will show all three default planes for selection in a `Trimetric` orientation. A `Trimetric` orientation is a pictorial view that is oriented so the three mutually perpendicular planes appear unequally foreshortened.



From the screen, choose the `Front Plane`. The plane will highlight and rotate.

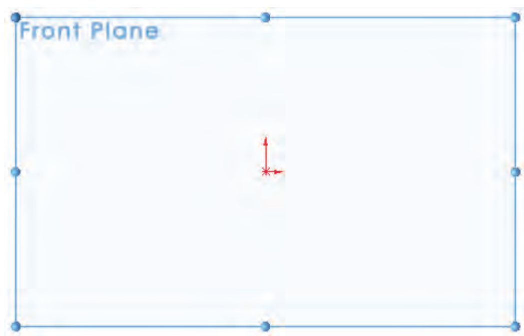
Note

The **Reference Triad** (lower left corner) shows the orientation of the model coordinate axes (red-`X`, green-`Y` and blue-`Z`) at all times. It can help show how the view orientation has been changed relative to the `Front Plane`.



4 Sketch active.

The selected `Front Plane` rotates so it is parallel to the screen.



The symbol represents the sketch origin. It is displayed in the color red, indicating that it is active.

Introducing: Confirmation Corner

When many SOLIDWORKS commands are active, a symbol or a set of symbols appears in the upper right corner of the graphics area. This area is called the **Confirmation Corner**.

Sketch Indicator

When a sketch is active, or open, the **Confirmation Corner** displays two symbols. One looks like a sketch. The other is a red X. These symbols provide a visual reminder that you are active in a sketch. Clicking the sketch symbol exits the sketch and *saves any changes*. Clicking the red X exits the sketch and discards any changes.

When other commands are active, the confirmation corner displays a check mark and an X. The check mark executes the current command. The X cancels the command.

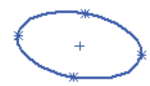
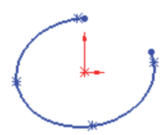
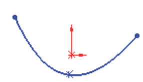
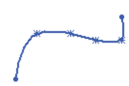
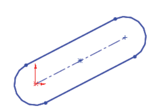
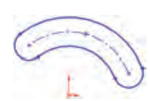
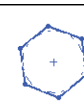
Press the **D** key to move the confirmation corner to the pointer location.

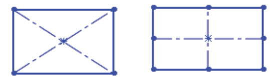
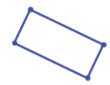
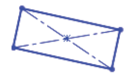
Sketch Entities

SOLIDWORKS offers a rich variety of sketch tools for creating profile geometry. In this lesson, only one of the most basic shapes will be used: **Lines**.

Sketch Geometry

The following chart lists some of the sketch entities that are available:

Sketch Entity	Button	Geometry Example
Line		
Circle		
Perimeter Circle		
Centerpoint Arc		
Tangent Arc		
3 Point Arc		
Ellipse		
Partial Ellipse		
Parabola		
Spline		
Straight Slot		
Centerpoint Straight Slot		
3 Point Arc Slot		
Centerpoint Arc Slot		
Polygon		

Sketch Entity	Button	Geometry Example
Corner Rectangle		
Center Rectangle (Construction geometry can be added to any type)		
3 Point Corner Rectangle		
3 Point Center Rectangle		
Parallelogram		
Point		
Centerline		

Basic Sketching

The best way to begin sketching is by using the most fundamental shape, the **Line**.

The Mechanics of Sketching

To sketch geometry, there are two techniques that can be used:

- **Click-Click**
  Position the cursor where you want the line to start. Click (press and release) the left mouse button. Move the cursor to where you want the line to end. A preview of the sketch entity will follow the cursor like a rubber band. Click the left mouse button a second time. Additional clicks create a series of connected lines.
- **Click-Drag**
  Position the cursor where you want the line to start. Press and hold the left mouse button. Drag the cursor to where you want the sketch entity to end. A preview of the sketch entity will follow the cursor like a rubber band. Release the left mouse button.

Introducing: Insert Line

The **Line** tool creates single line segments in a sketch. Horizontal and vertical lines can be created while sketching by watching for the feedback symbols on the cursor.

Where to Find It

- CommandManager: **Sketch > Line**
- Menu: **Tools, Sketch Entities, Line**
- Shortcut Menu: Right-click in the graphics area and click **Sketch Entities, Line**

Introducing: Sketch Relations

Sketch Relations are used to force a behavior on a sketch element thereby capturing design intent. They will be discussed in detail in *Sketch Relations* on page 41.

---

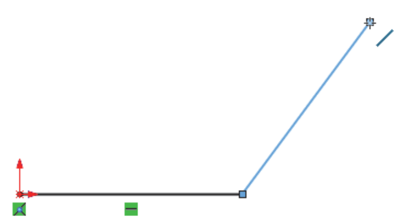
5 Sketch a line.

Click **Line** and sketch a horizontal line from the origin. The symbol appears at the cursor, indicating that a **Horizontal** relation will be automatically added to the line. The number indicates the length of the line. Click again to end the line.

Important!

Do not be too concerned with making the line the exact length. SOLIDWORKS software is dimension driven – the dimensions control the size of the geometry, not the other way around. Make the sketch approximately the right size and shape and then use dimensions to make it exact.

6 Line at angle.
Starting at the end of the first line, sketch a line at an angle.



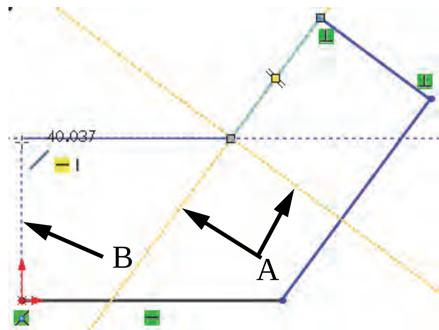
Note

The pencil icon at the cursor will be omitted for clarity.

Inference Lines (Automatic Relations)

In addition to the relation symbols, dashed inference lines will also appear to help you "line up" with existing geometry. These lines include existing line vectors, normals, horizontals, verticals, tangents and centers.

Note that some lines capture actual geometric relations, while others simply act as a guide or reference when sketching. A difference in the color of the inference lines will distinguish them. In the picture at the right, the lines labeled "A" are yellow, and if the sketch line snaps to them, a tangent or perpendicular relationship will be captured.



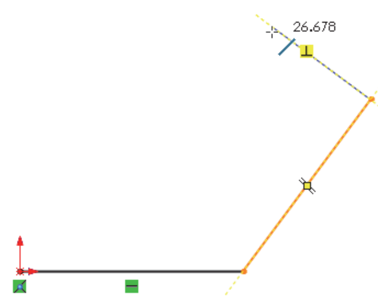
The line labeled "B" is blue. It only provides a reference, in this case vertical, to the other endpoint. If the sketch line is ended at this point, no vertical relation will be captured.

Note

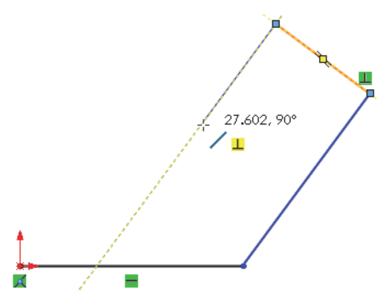
The display of Sketch Relations that appears automatically can be toggled on and off using **View, Hide/Show, Sketch Relations**. It will remain on during the initial phase of sketching.

7 **Inference lines.**
Create a line moving in a direction perpendicular to the previous line. This causes inference lines to be displayed while sketching. A **Perpendicular** relation is created between this line and the last one.

The cursor symbol indicates that you are capturing a perpendicular relation.



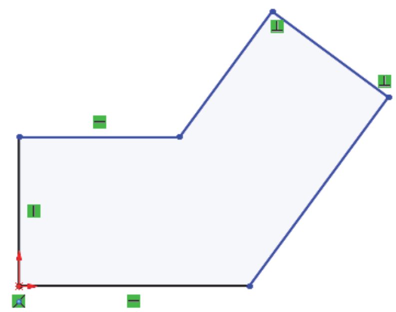
8 **Perpendicular.**
Create another perpendicular line from the last endpoint, again capturing a perpendicular relation.



9 Reference.
Create a horizontal line from the last endpoint. Blue inferences are strictly for reference and do *not* create relations. They are displayed in blue. This reference is used to align the endpoint vertically with the origin.

10 Close.
Close the sketch with a final line connected to the starting point of the first line.

A closed contour is confirmed with shading.

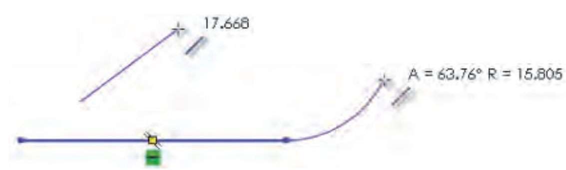


Note

Click **Shaded Sketch Contours** from the **Sketch** CommandManager to toggle the shading on and off.

Sketch Feedback

The sketcher has many feedback features. The cursor will change to show what type of entity is being created. It will also indicate what selections on the existing geometry, such as end, coincident (on) or midpoint, are available using an orange dot when the cursor is on it.



Three of the most common feedback symbols are:

Symbol	Icon	Description
Endpoint		Yellow concentric circles appear at the Endpoint when the cursor is over it.
Midpoint		The Midpoint appears as a yellow square. It changes to orange when the cursor hovers over the line.
Coincident (On Edge)		The quadrant points of the circle appear with a concentric circle over the centerpoint.

Turning Off Tools

Turn off the active tool using *one* of these techniques:

- Menu Bar: **Select**
- CommandManager: Click the active tool to toggle the tool off
- Keyboard Shortcut: **Esc**

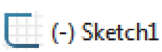
---

11 Turn off the tool.
Press the **Esc** key on the keyboard to turn off the line tool.

Status of a Sketch

Sketches can be in one of five definition states at any time. The status of a sketch depends on geometric relations between geometry and the dimensions that define it. The three most common states are:

Under Defined



The sketch is inadequately defined, but the sketch can still be used to create features. This is good because many times in the early stages of the design process, there isn't sufficient information to fully define the sketch. When more information becomes available, the remaining definition can be added at a later time. Under defined sketch geometry is **blue** (by default).

Fully Defined

The sketch has all the information necessary to fully describe the geometry. Fully defined geometry is black (by default). As a general rule, when a part is released to manufacturing, the sketches within it should be fully defined.

Over Defined

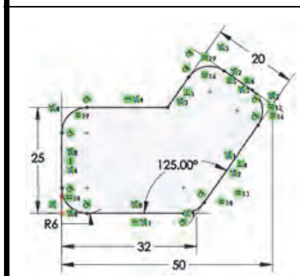
The sketch has duplicate dimensions or conflicting relations and it should not be used until repaired. Extraneous dimensions and relations should be deleted. Over defined geometry is **red** (by default).

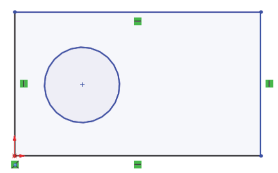
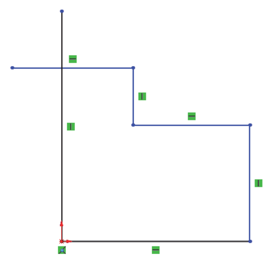
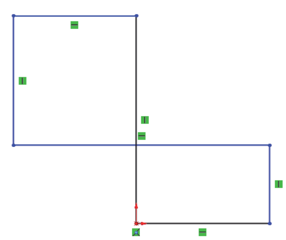
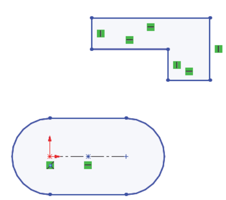
Note

The two other states are **No Solution Found** and **Invalid Solution Found**. They both indicate that there are errors that must be repaired.

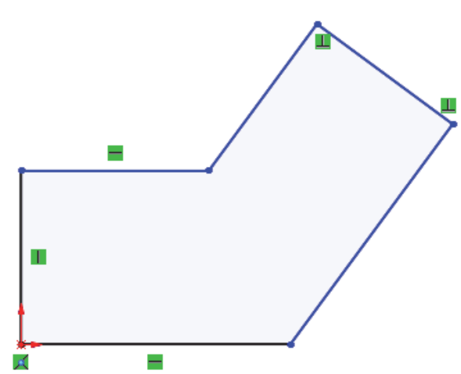
Rules That Govern Sketches

Different types of sketches will yield different results. Several different types are summarized in the table below. It is important to note that some of the techniques shown in the table below are advanced techniques that are covered either later in this course, or in other advanced courses.

Sketch Type	Description	Special Considerations
	A typical "standard" sketch that is a neatly closed contour.	None required.

	Multiple nested contours creates a boss with an internal cut.	None required.
	Open contour creates a thin feature with constant thickness.	None required.
	Corners are not neatly closed. *They should be.*	Use the **Contour Select Tool**. Although this sketch will work, it represents poor technique and sloppy work habits. Do not do it.
	Sketch contains a self-intersecting contour.	Use the **Contour Select Tool**. If both contours are selected, this type of sketch will create a **Multibody Solid**. See *Multibody Solids* in the *Advanced Part Modeling* course. Although this will work, multibodies are an advanced modeling technique that you should not use until you have more experience.
	The sketch contains disjoint contours.	This type of sketch can create a **Multibody Solid**. See *Multibody Solids* in the *Advanced Part Modeling* course. Although this will work, multibodies are an advanced modeling technique that you should not use until you have more experience.

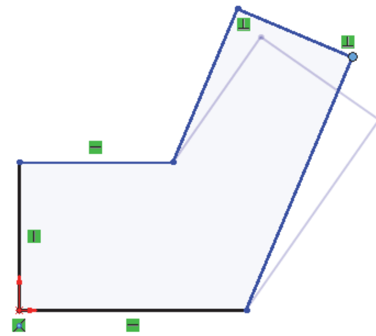
12 Current sketch status.
The sketch is **Under Defined** because some of the geometry is blue. Note that endpoints of a line can be a different color and different state than the line itself. For example, the vertical line at the origin is black because it is (a) vertical, and (b) attached to the origin. However, the uppermost



endpoint is blue because the length of the line is under defined.

13 Dragging.

Under defined geometry (**blue**) can be dragged to new locations. Fully defined geometry cannot. Drag the uppermost endpoint to change the shape of the sketch. The dragged endpoint appears as a blue dot.



14 Undo the change.

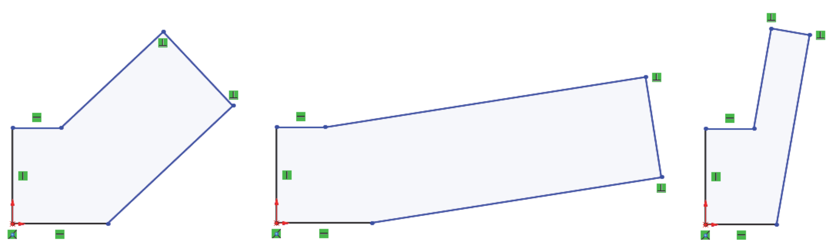
Undo the last command by clicking the **Undo** option. You can see (and select from) a list of the last few commands by clicking the down arrow menu. The keyboard shortcut for **Undo** is **Ctrl+Z**.

Tip

You can also **Redo** a change, which reverts it back to the state prior to undo. The shortcut for redo is **Ctrl+Y**.

Design Intent

The design intent, as discussed earlier, governs how the part is built and how it will change. In this example, the sketch shape must be allowed to change in these ways:



What Controls Design Intent?

Design intent in a sketch is captured and controlled by a combination of two things:

- **Sketch relations**
  Create geometric relationships such as parallel, collinear, perpendicular, or coincident between sketch elements.
- **Dimensions**
  Dimensions are used to define the size and location of the sketch geometry. Linear, radial, diameter and angular dimensions can be added.

To fully define a sketch *and* capture the desired design intent requires understanding and applying a combination of relations and dimensions.

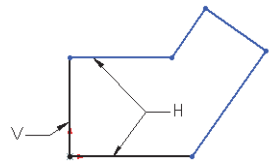
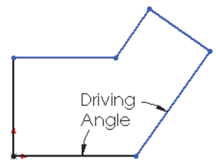
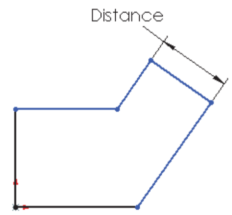
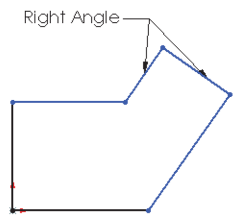
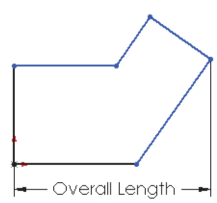
Tip

The relations are visible because **View, Hide/Show, Sketch Relations** is toggled on. If it is turned off, clicking the geometry will show the relations and open the PropertyManager.

The relations will be toggled *off* at this point, but they will still appear on selected geometry.

Desired Design Intent

In order for the sketch to change properly, the correct relations and dimensions are required. The required design intent is listed below:

Horizontal and vertical lines	
Angle value	
Parallel Distance value	
Right-angle corners, or perpendicular lines	
Overall length value	

Note

The shading has been removed from table images for clarity.

Sketch Relations

Sketch Relations are used to force a behavior on a sketch element thereby capturing design intent. Some are automatic, others can be added as needed. In this example, we will look at the relations on one of the lines and examine how they affect the design intent of the sketch.

Automatic Sketch Relations

Automatic relations are added as geometry is sketched. We saw this as we sketched the outline in the previous steps. Sketch feedback tells you when automatic relations are being created.

Added Sketch Relations

For those relations that cannot be added automatically, tools exist to create relations based on selected geometry.

Introducing: Display/Delete Relations

Display/Delete Relations shows the relations in a sketch. It also enables you to remove relations.

Where to Find It

- CommandManager: **Sketch > Display/Delete Relations**
- Menu: **Tools, Relations, Display/Delete**
- **Properties** PropertyManager: **Existing Relations**

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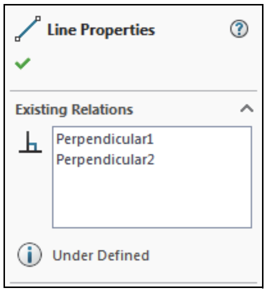
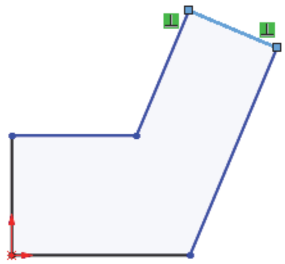
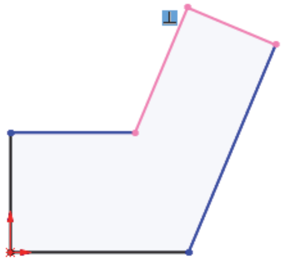
15 Display the relations associated with a line.
Click the uppermost angled line and the PropertyManager opens. The **Existing Relations** box in the PropertyManager lists the geometric relations that are associated with the selected line.

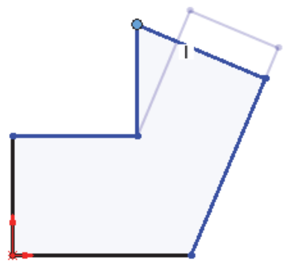




16 Remove the relation.
Remove the uppermost relation by clicking the relation, either the symbol or in the PropertyManager, and pressing the **Delete** key. If the symbol is selected, it changes color and displays the entities it controls.

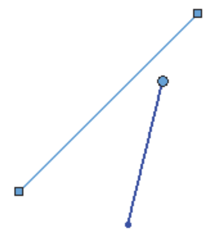
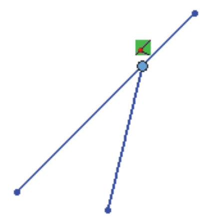
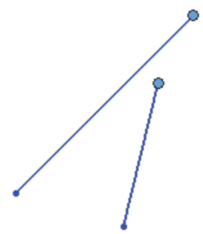
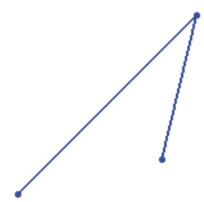
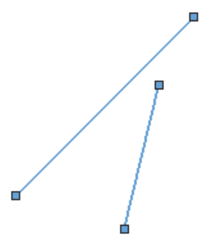
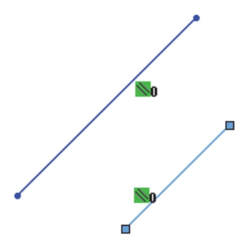
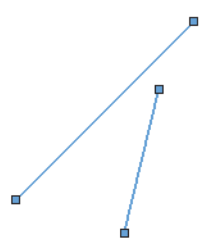
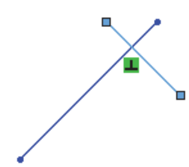
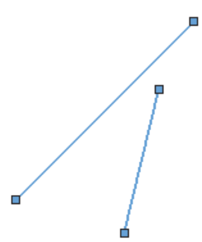
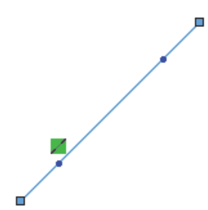


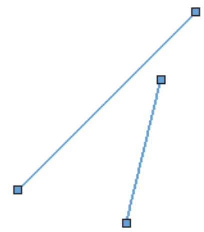
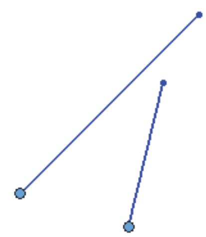
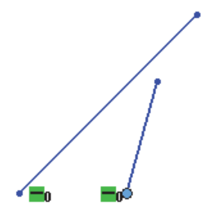
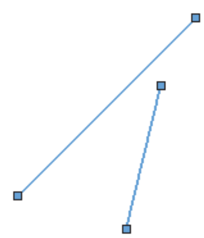
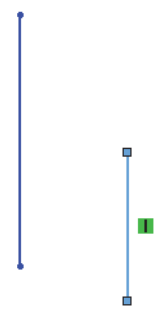
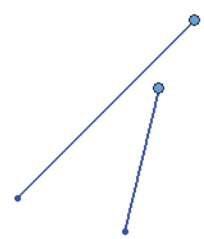
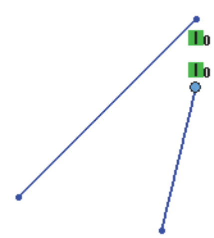
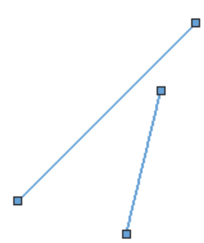
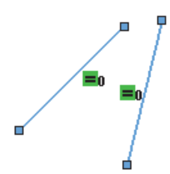
17 Drag the endpoint.
Because the line is no longer constrained to be perpendicular, the sketch will behave differently when you drag it. Compare this to how the sketch behaved when you dragged it in step **13**.

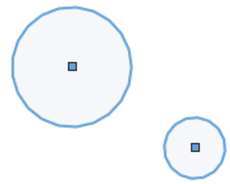
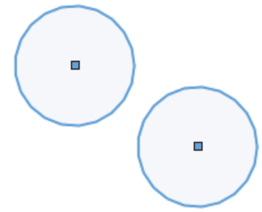
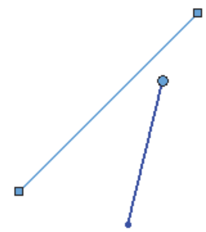
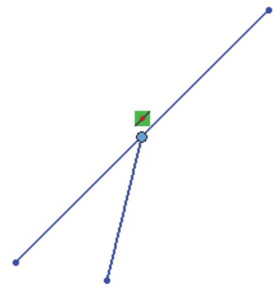
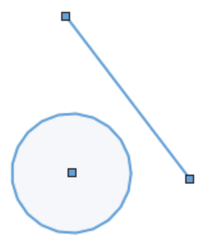
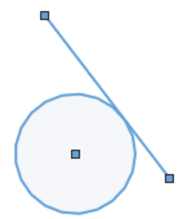
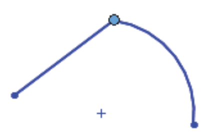
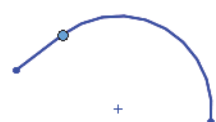


Examples of Sketch Relations

There are many types of **Sketch Relations**. Which ones are valid depends on the combination of geometry that you select. Selections can be the entity itself, endpoints or a combination. Depending on the selection, a limited set of options is made available. The following chart shows some examples of sketch relations. This is not a complete list of all geometric relations. Additional examples will be introduced throughout this course.

Relation	Before	After
Coincident between a line and an endpoint.		
Merge between two endpoints.		
Parallel between two or more lines.		
Perpendicular between two lines.		
Collinear between two or more lines.		

Relation	Before	After
Horizontal applied to one or more lines.		
Horizontal between two or more endpoints.		
Vertical applied to one or more lines.		
Vertical between two or more endpoints.		
Equal between two or more lines.		

Relation	Before	After
Equal between two or more arcs or circles.		
Midpoint between a line and an endpoint.		
Tangent between a line and an arc/circle or two arc/circles.		
Tangent between a line and an arc using the common endpoint.		

Introducing: Add Relations

Add Relations is used to create a geometric relationship such as parallel or collinear between sketch elements.

Where to Find It

- CommandManager: **Sketch > Display/Delete Relations > Add Relation**
- Menu: **Tools, Relations, Add**
- Shortcut Menu: Select one or more sketch objects and click a relation

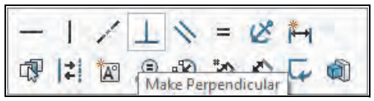
Selecting Multiple Objects

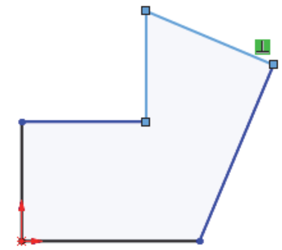
As you learned in a previous lesson, you select objects with the left mouse button. What about when you need to select more than one object at a time? When selecting multiple objects, SOLIDWORKS follows standard Microsoft® Windows conventions: hold down the **Ctrl** key while selecting the objects.

18 Add a relation.

Hold down **Ctrl** and click the two lines. The context menu shows only those relations that are valid for the geometry selected.

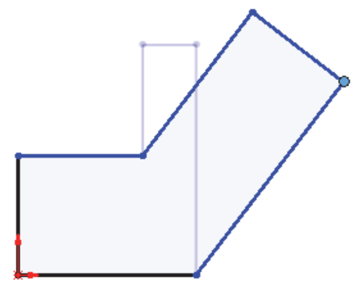
Click **Make Perpendicular**.





19 Drag the sketch.

Drag the sketch back into approximately its original shape.



Dimensions

Dimensions are another way to define geometry and capture design intent in the SOLIDWORKS system. The advantage of using a dimension is that it is used to both display the current value and change it.

Introducing: Smart Dimensions

The **Smart Dimension** tool determines the proper type of dimension based on the geometry chosen, *previewing* the dimension before creating it. For example, if you pick an arc the system will create a radial dimension. If you pick a circle, you will get a diameter dimension, while selecting two parallel lines will create a linear dimension between them. In cases where the **Smart Dimension** tool isn't quite smart enough, you have the option of selecting endpoints and moving the dimension to different measurement positions.

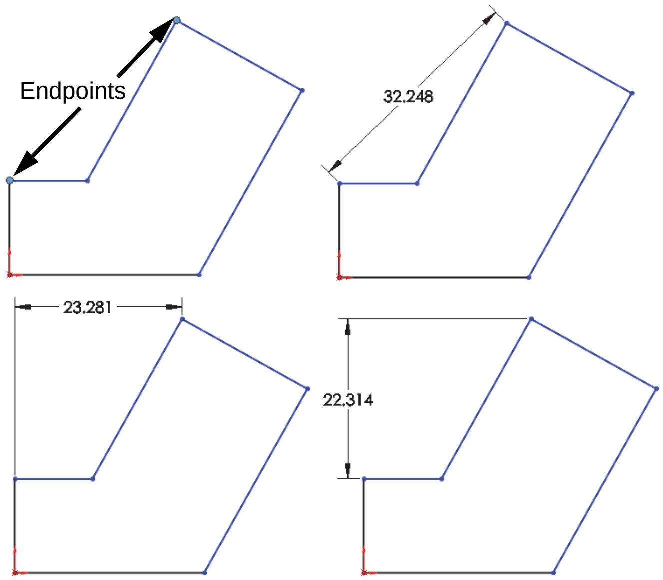
Where to Find It

- CommandManager: **Sketch > Smart Dimension**
- Menu: **Tools, Dimensions, Smart**
- Shortcut Menu: Right-click in the graphics area and click **Smart Dimension**

Dimensioning: Selection and Preview

As you select sketch geometry with the dimension tool, the system creates a preview of the dimension. The preview enables you to see all the possible options by simply moving the mouse after making the selections. Clicking the left mouse button places the dimension in its current position and orientation. Clicking the right mouse button locks only the orientation, allowing you to move the text before final placement by clicking the left mouse button.

With the dimension tool and two endpoints selected, below are three possible orientations for a linear dimension. The value is derived from the initial point to point distance and may change based on the orientation selected.

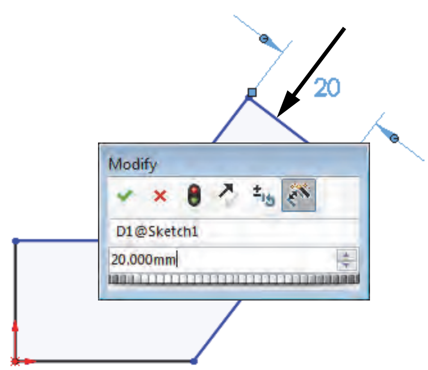


Note Another option is to select the geometry that is to be dimensioned and click **Auto Insert Dimension**.

---

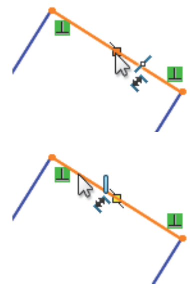
20 Adding a linear dimension.

Click **Smart Dimension** and click the line shown and right-click to lock in the orientation. Click again to place the text as shown. The dimension appears with a **Modify** tool displaying the current length of the line. The thumbwheel is used to incrementally increase/decrease the value using the middle mouse button. Or with the text highlighted, you can type a new value to change it directly.



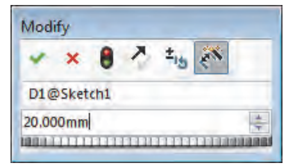
Note

A midpoint location can be inadvertently selected instead of the geometry itself. To avoid this, select the geometry slightly off center.



The Modify Tool

The modify tool that appears when you create or edit a dimension (parameter) has several options. The options available to you are:



Dial the value up or down.

Save the current value and exit the dialog box.

Restore the original value and exit the dialog box.

Rebuild the model with the current value.

Reverse the sense of the dimension.

Change the thumbwheel increment value.

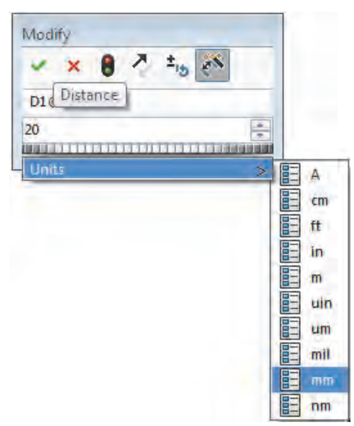
Mark the dimension for drawing import.

Note

The dimension name can be changed in the upper section of the dialog box.

Units in the Modify Tool

Units different from the part units can be selected for the input. When typing the value, select the **Units >** menu and select input units.

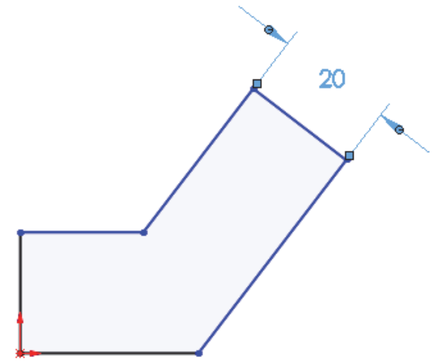


Note

Unit abbreviations and fractions can also be typed into the value field after the numeric value (for instance **0.375in** or **3/8”**).

21 Set the value.

Change the value to **20** and click the **Save** option. The dimension forces the length of the line to be 20mm.

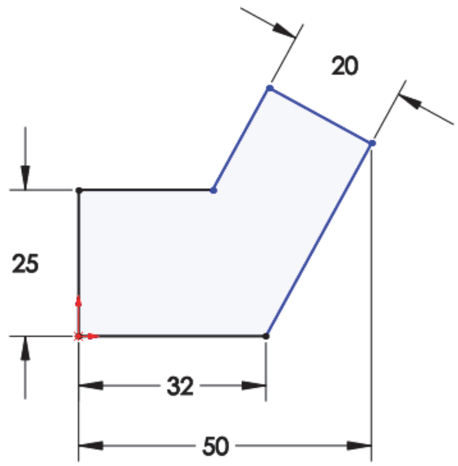


Tip Pressing **Enter** has the same effect as clicking the **Save** button.

22 Linear dimensions.

Add additional linear dimensions to the sketch as shown.

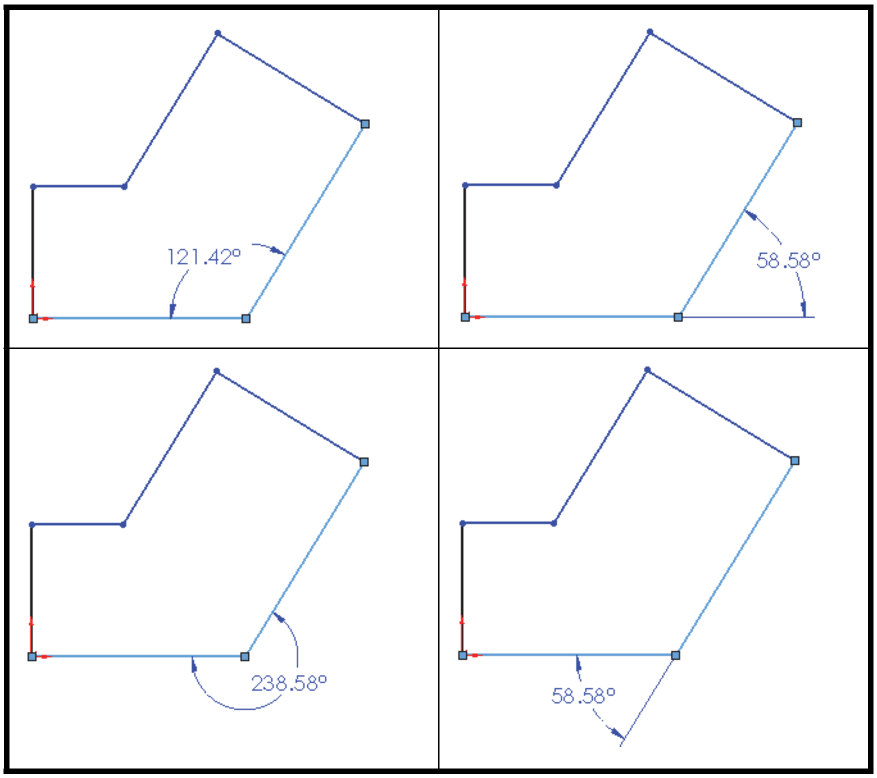
Dimensioning Tip When you dimension a sketch, start with the smallest dimension first, and work your way to the largest.



Angular Dimensions

Angular dimensions can be created using the same dimension tool used to create linear, diameter and radial dimensions. Select either two lines that are both non-collinear and non-parallel, or select three non-collinear endpoints.

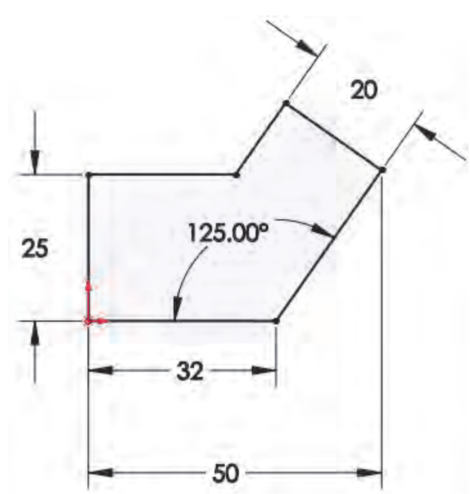
Depending on where you place the angular dimension, you can get the interior or exterior angle, the acute angle, or the oblique angle. Possible placement options:



23 Angular dimension.

Using the dimension tool, create the angular dimension shown and set the value to **125°**.

The sketch is fully defined. See *Fully Defined* on page 38.



Instant 2D

Instant 2D can be used to manipulate sketch dimensions, dynamically changing the values using a graphic **Ruler**.

Note

The ruler is displayed to guide the drag. Moving closer to the ruler gradients allows you to snap to them.

Where to Find It

- CommandManager: **Sketch > Instant 2D**

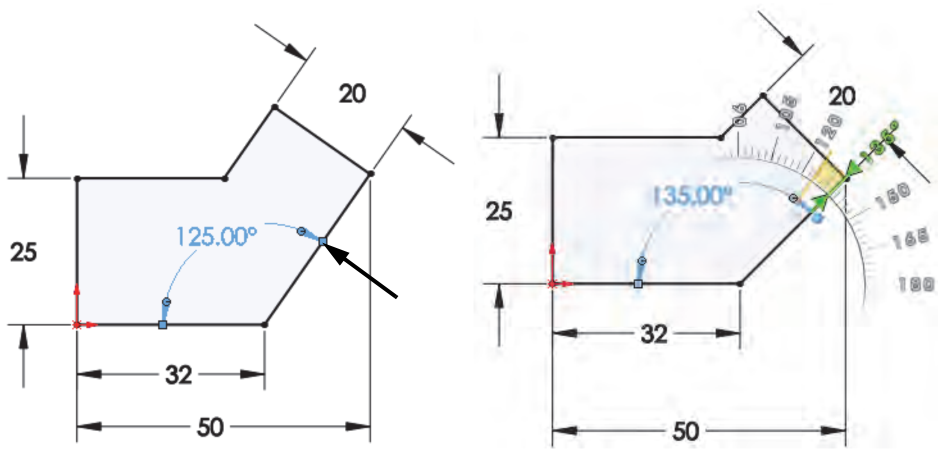
24 Select dimension.

The **Instant 2D** tool is on by default. Select the 125° dimension.

Click and hold the round ball handle at the tip of the arrow.

The value of the dimension, and the geometry, changes dynamically as the handle is dragged.

Drag the value to **135°** using the ruler.



Extrude

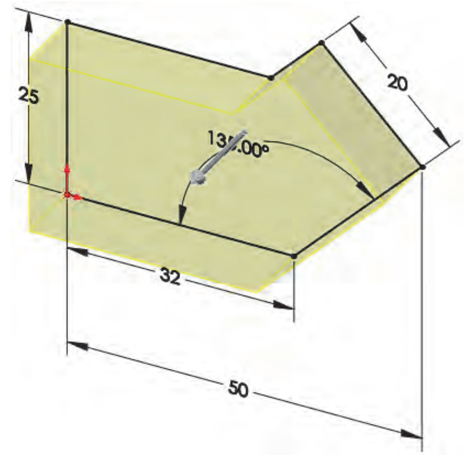
Once the sketch is completed, it can be extruded to create the first feature. There are many options for extruding a sketch including the start and end conditions, draft and depth of extrusion, which will be discussed in more detail in later lessons. Typically, extrusions take place in a direction normal to the sketch plane, in this case the `Front` plane.

Where to Find It

- CommandManager: **Features > Extruded Boss/Base**
- Menu: **Insert, Boss/Base, Extrude**

25 Extrude.

Click **Extruded Boss/Base**.

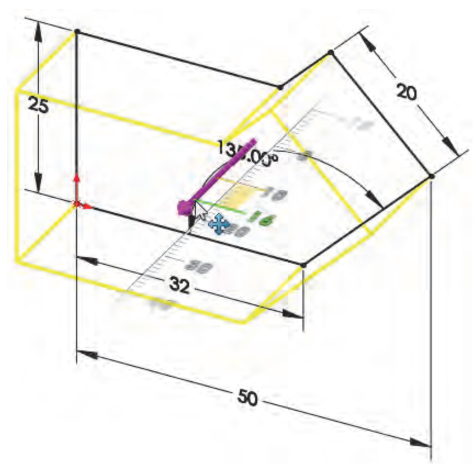


On the **Features** CommandManager tab, the options for other methods of creating features are listed along with **Extrude** and **Revolve**. They are unavailable because this sketch does not meet the conditions necessary for creating these types of features. For example, a **Sweep** feature requires both profile and path sketches. Since there is only one sketch at this time, the **Sweep** option is unavailable.

The view automatically changes to `Trimetric` and a preview of the feature is shown at the default depth.

Drag Handles and Rulers

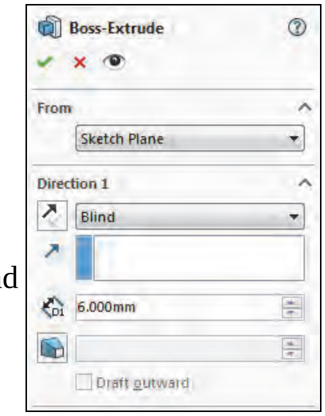
Handles appear that can be used to drag the preview to the desired depth. The handle is colored while dragging in the active direction. A callout shows the current depth value and a ruler.



26 Extrude Feature settings.
Change the settings as shown.

- **End Condition** = **Blind**
- (**Depth**) = **6mm**

Click **OK** to create the feature.

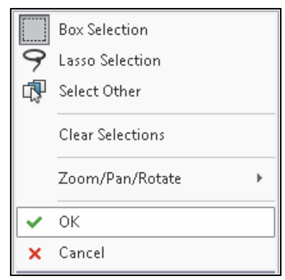


Tip

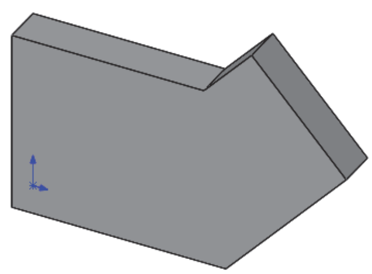
The **OK** button is just one way to accept and complete the process. A second is to press the **Enter** key.

A third method is the set of **OK/Cancel** buttons in the **Confirmation Corner** of the graphics area, or press the **D** key to bring it to the cursor.

A fourth method is to right-click and click **OK** from the shortcut menu.

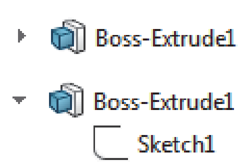


27 Completed feature.
The completed feature is the first solid, or feature of the part. The sketch is absorbed into the `Extrude1` feature.



Note

Click the preceding the feature name to expand the feature and show the sketch.



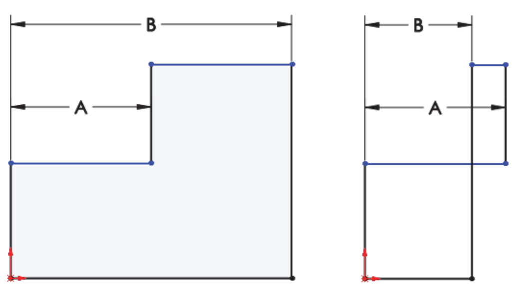
28 Save and close.

Click **Save** then click **File, Close** to close the part.

Sketching Guidelines†

Following is a collection of "rules of thumb" or best practices for sketching of which all SOLIDWORKS users should be aware. Some of these tips are covered in substantial detail in subsequent lessons within this book.

- Keep your sketches simple. Simple sketches are easier to edit, less likely to develop errors, and help with downstream features such as configurations.
- Make use of the origin in your first sketch.
- The first sketch of a new part should represent the main profile of the part.
- Create sketch geometry first, add geometric relationships second, and then add your dimensions last. Dimensions can sometimes interfere with the addition of required relations.
- Use geometric relations wherever possible to maintain design intent.
- Draw the sketch to approximately the right scale to prevent errors or geometry overlap when you start adding dimensions.
- Add or edit dimensions on the closest / smallest geometry first, then work your way to the outer / larger geometry to prevent geometry overlap.

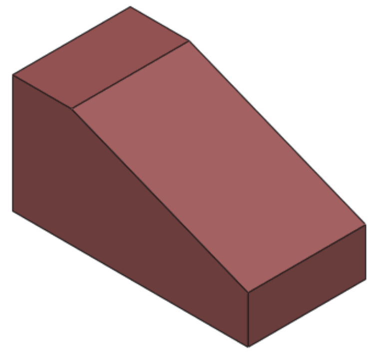


- Use relations, equations, and global variables to reduce the number of independent dimensions needed.
- Take advantage of symmetry. Use the **Mirror** or **Dynamic Mirror** sketch tool to mirror sketch elements and add symmetrical relations.
- Be flexible. It may be necessary to change the order in which you're adding dimensions or relations. Drag the sketch geometry closer to the required location before adding dimensions.
- Fix errors as they occur. Use **SketchXpert** and **Check Sketch for Feature** which can quickly help you identify problems and correct them.

† Thanks to Joe Medeiros, Javelin Technologies.

Exercise 1: Sketch and Extrude 1

Create this part using the information and dimensions provided. Sketch and extrude profiles to create this part.



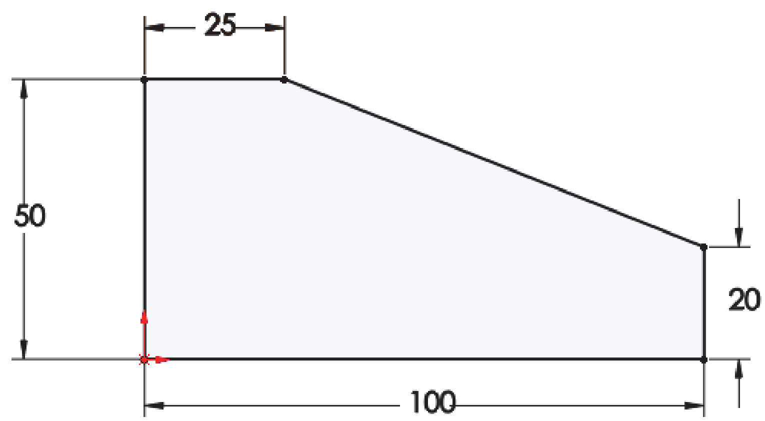
This lab reinforces the following skills:

- *Introducing: New Part* on page 29
- *Sketching* on page 31
- *Inference Lines (Automatic Relations)* on page 36
- *Dimensions* on page 46
- *Extrude* on page 51

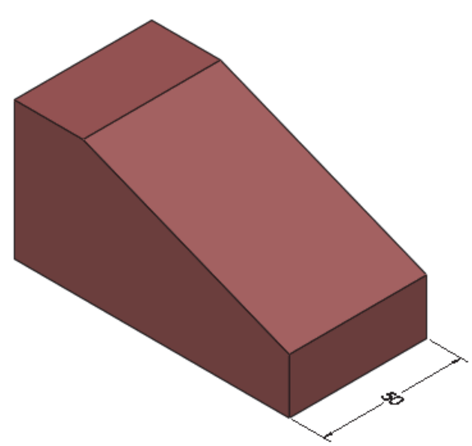
Units: **millimeters**

1 **New part.**
Create a new part using the `Part_MM` template.

2 **Sketch.**
Create this sketch on the `Front Plane` using lines, automatic relations and dimensions.



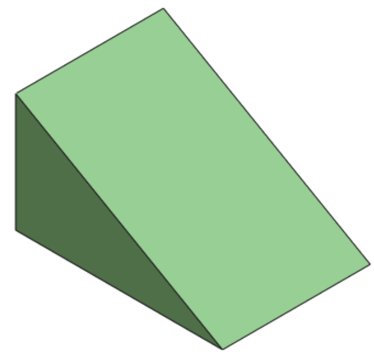
3 **Extrude.**
Extrude the sketch **50mm** in depth.



4 **Save and close the part.**

Exercise 2: Sketch and Extrude 2

Create this part using the information and dimensions provided. Sketch and extrude profiles to create this part.



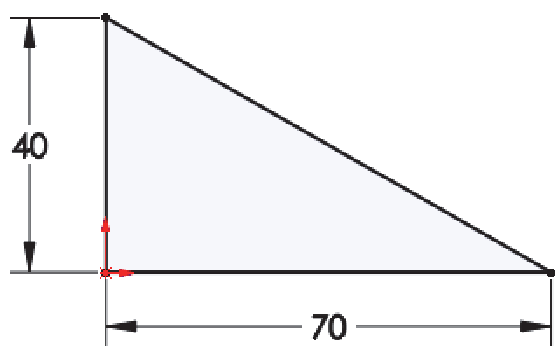
This lab reinforces the following skills:

- *Introducing: New Part* on page 29
- *Sketching* on page 31
- *Inference Lines (Automatic Relations)* on page 36
- *Dimensions* on page 46
- *Extrude* on page 51

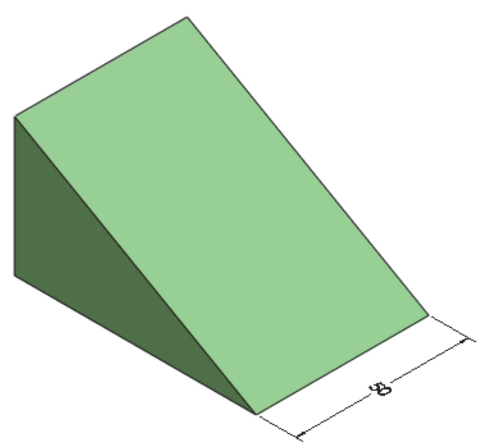
Units: **millimeters**

1 New part.
Create a new part using the `Part_MM` template.

2 Sketch.
Create this sketch on the `Front Plane` using lines, automatic relations and dimensions.



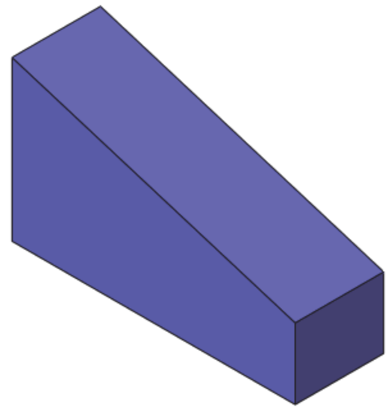
3 Extrude.
Extrude the sketch **50mm** in depth.



4 Save and close the part.

Exercise 3: Sketch and Extrude 3

Create this part using the information and dimensions provided. Sketch and extrude profiles to create this part.



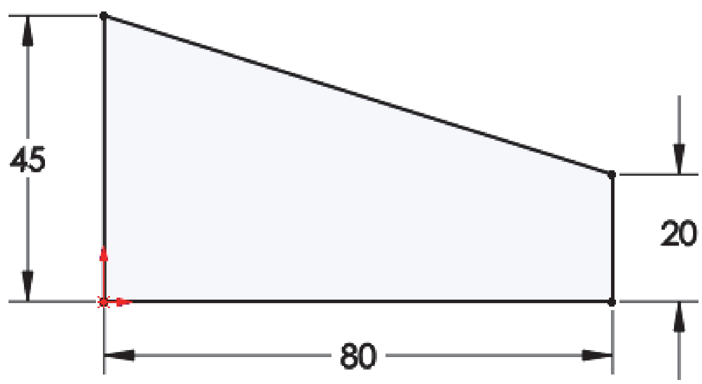
This lab reinforces the following skills:

- *Introducing: New Part* on page 29
- *Sketching* on page 31
- *Inference Lines (Automatic Relations)* on page 36
- *Dimensions* on page 46
- *Extrude* on page 51

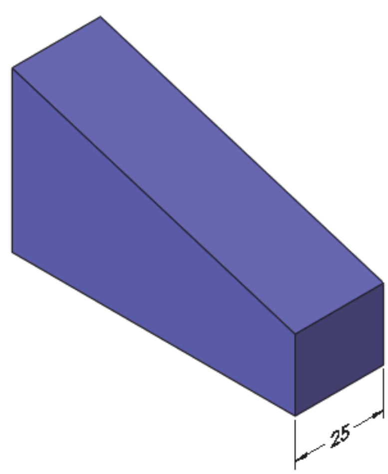
Units: **millimeters**

1 New part.
Create a new part using the `Part_MM` template.

2 Sketch.
Create this sketch on the `Front Plane` using lines, automatic relations and dimensions.



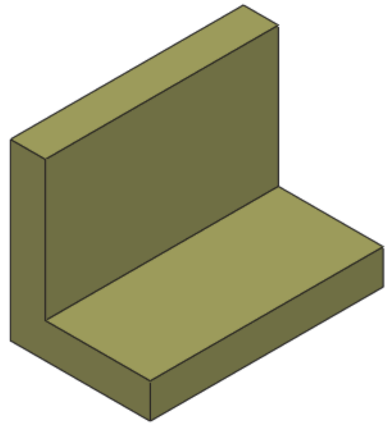
3 Extrude.
Extrude the sketch **25mm** in depth.



4 Save and close the part.

Exercise 4: Sketch and Extrude 4

Create this part using the information and dimensions provided. Sketch and extrude profiles to create this part.



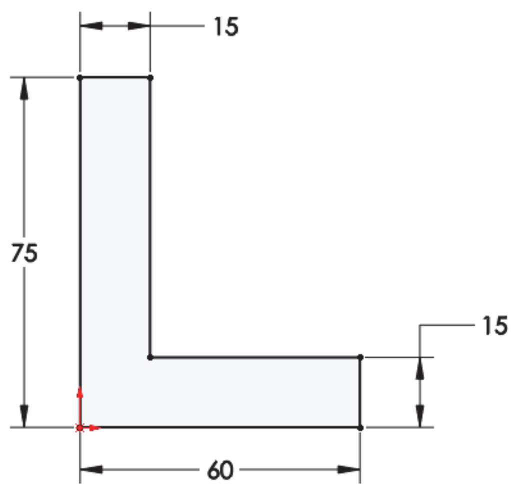
This lab reinforces the following skills:

- *Introducing: New Part* on page 29
- *Sketching* on page 31
- *Inference Lines (Automatic Relations)* on page 36
- *Dimensions* on page 46
- *Extrude* on page 51

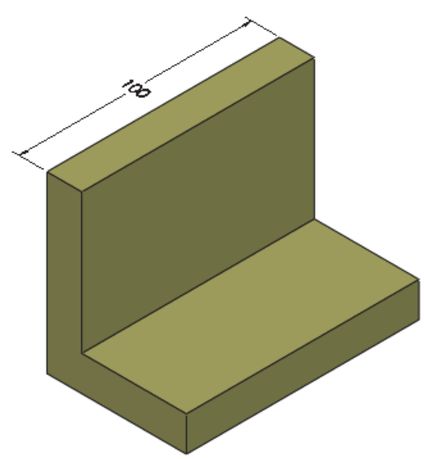
Units: **millimeters**

1 **New part.**
Create a new part using the `Part_MM` template.

2 **Sketch.**
Create this sketch on the `Front Plane` using lines, automatic relations and dimensions.

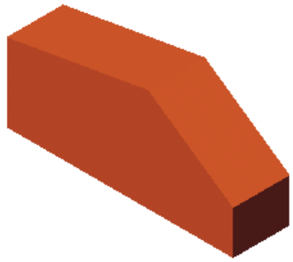


3 **Extrude.**
Extrude the sketch **100mm** in depth.



4 **Save and close the part.**

Exercise 5: Sketch and Extrude 5



Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.

This lab reinforces the following skills:

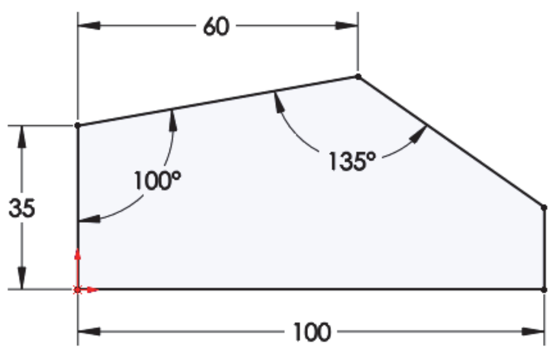
- *Introducing: New Part* on page 29
- *Sketching* on page 31
- *Inference Lines (Automatic Relations)* on page 36
- *Dimensions* on page 46
- *Extrude* on page 51

Units: **millimeters**

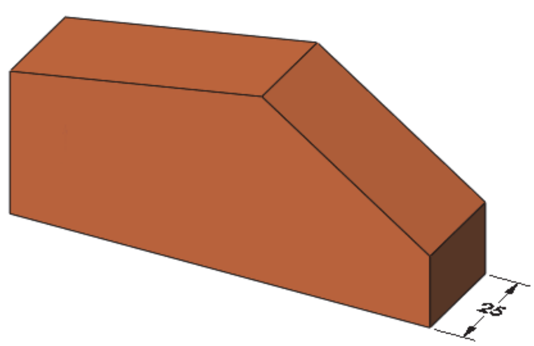
1 New part.
Create a new part using the `Part_MM` template.

2 Sketch.
Create this sketch on the `Front Plane` using lines, automatic relations and dimensions.

Fully define the sketch.

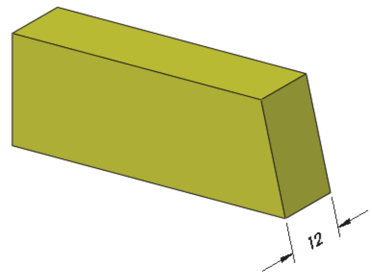


3 Extrude.
Extrude the sketch **25mm** in depth.



4 Save and close the part.

Exercise 6: Sketch and Extrude 6



Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part.

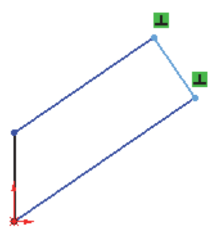
This lab reinforces the following skills:

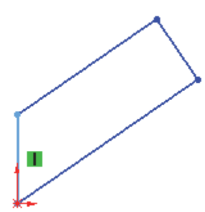
- *Introducing: New Part* on page 29
- *Sketching* on page 31
- *Inference Lines (Automatic Relations)* on page 36
- *Dimensions* on page 46
- *Extrude* on page 51

Units: **millimeters**

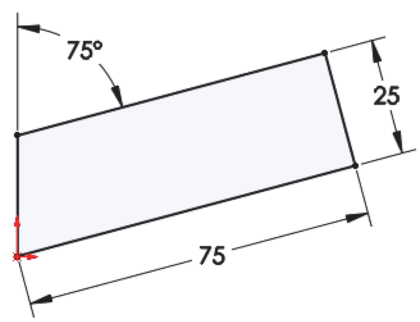
1 New part.
Create a new part using the `Part_MM` template.

2 Automatic relations.
Create this sketch on the `Front Plane` using lines and automatic relations. Show the **Perpendicular** and **Vertical** relations.

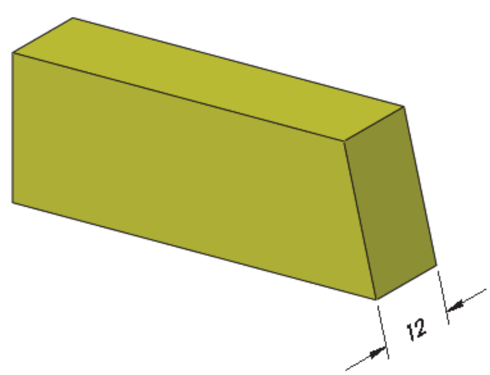




3 Dimensions.
Add dimensions to fully define the sketch.



4 Extrude.
Extrude the sketch **12mm**.



5 Save and close the part.

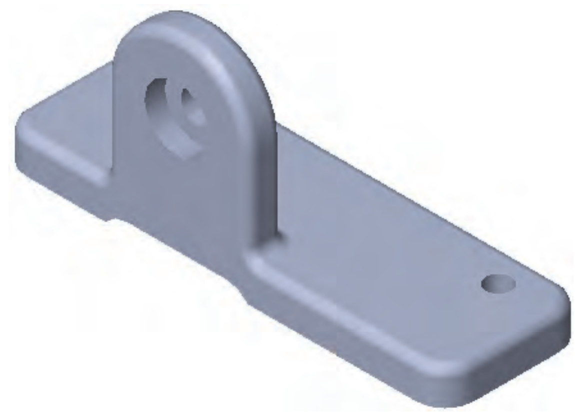
Lesson 3
Basic Part Modeling

Upon successful completion of this lesson, you will be able to:

- Choose the best profile for sketching.
- Choose the proper sketch plane.
- Extrude a sketch as a cut.
- Create Hole Wizard holes.
- Insert fillets on a solid.
- Use the editing tools Edit Sketch, Edit Feature and Rollback.
- Make a basic drawing of a part.
- Make a change to a dimension.
- Demonstrate the associativity between the model and its drawings.

Basic Modeling

This lesson discusses the considerations that you make before creating a part, and shows the process of creating a simple one.



Stages in the Process

The steps in planning and executing the creation of this part are listed below.

- **Terminology**
  What are the terms commonly used when talking about modeling and using the SOLIDWORKS software?
- **Profile choice**
  Which profile is the best one to choose when starting the modeling process?
- **Sketch plane choice**
  Once you've chosen the best profile, how does this affect your choice of sketch plane?
- **Design intent**
  What is design intent and how does it affect the modeling process?
- **New part**
  Opening the new part is the first step.
- **First feature**
  What is the first feature?
- **Bosses, cuts and hole features**
  How do you modify the first feature by adding bosses, cuts and holes?
- **Fillets**
  Rounding off the sharp corners – filleting.
- **Editing tools**
  Use three of the most common editing tools.
- **Drawings**
  Creating a drawing sheet and drawing views of the model.
- **Dimension changes**
  Making a change to a dimension changes the model's geometry. How does this happen?

Terminology

Moving to 3D requires some new terminology. The SOLIDWORKS software employs many terms that you will become familiar with through using the product. Many are terms that you will recognize from design and manufacturing such as cuts and bosses.

Feature

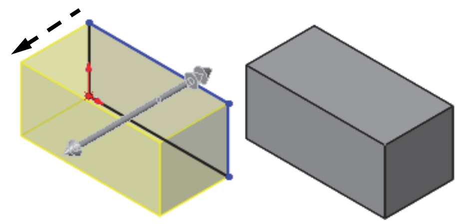
All cuts, bosses, planes and sketches that you create are considered Features. Sketched features are those based on sketches (boss and cut), and applied features are applied directly to existing geometry (fillet).

Plane

Planes are flat and infinite. They are represented on the screen with visible edges. They are used as the primary sketch surface for creating boss and cut features.

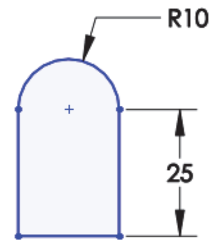
Extrusion

Although there are many ways to create features and shape the solid, for this lesson, only *extrusions* will be discussed. An extrusion will extend a profile along a path typically normal to the profile plane for some distance. The movement along that path becomes the solid model.



Sketch

In the SOLIDWORKS system, the name used to describe a 2D profile is *sketch*. Sketches are created on flat faces and planes within the model. They are generally used as the basis for bosses and cuts, although they can exist independently.



Boss

Bosses are used to *add* material to the model. The critical initial feature is always a boss. After the first feature, you may add as many bosses as needed to complete the design. As with the base, all bosses begin with a sketch.

Cut

A *Cut* is used to *remove* material from the model. This is the opposite of the boss. Like the boss, cuts begin as 2D sketches and remove material by extrusion, revolution, or other methods you will learn about.

Fillets and Rounds

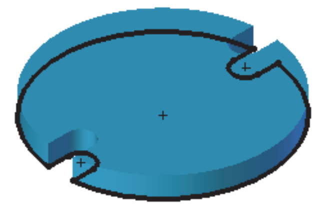
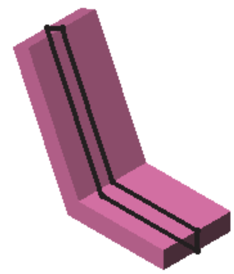
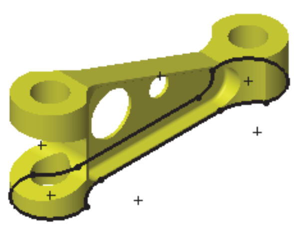
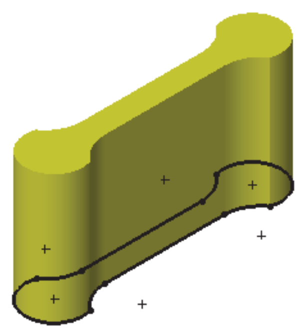
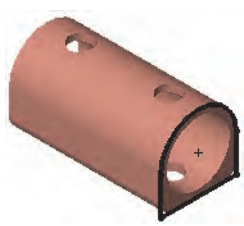
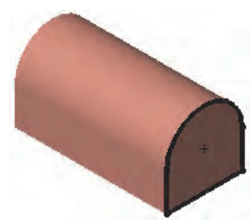
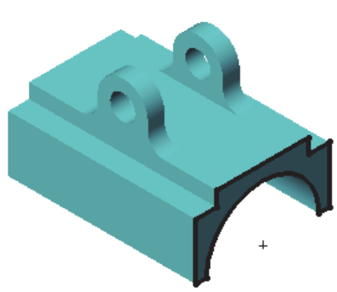
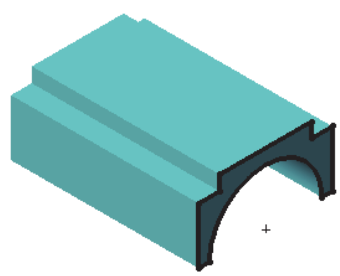
Fillets and *rounds* are generally added to the solid, not the sketch. By nature of the faces adjacent to the selected edge, the system knows whether to create a round (removing material) or a fillet (adding material).

Design Intent

How the model should be created and changed, is considered the design intent. Relationships between features and the sequence of their creation all contribute to design intent.

Choosing the Best Profile

Choose the *"best"* profile for the model's base feature. This profile, when extruded, will generate more of the model than any other. Look at these models as examples.

Part	Best Profile Extruded
	
	
	
	
	

Choosing the Sketch Plane

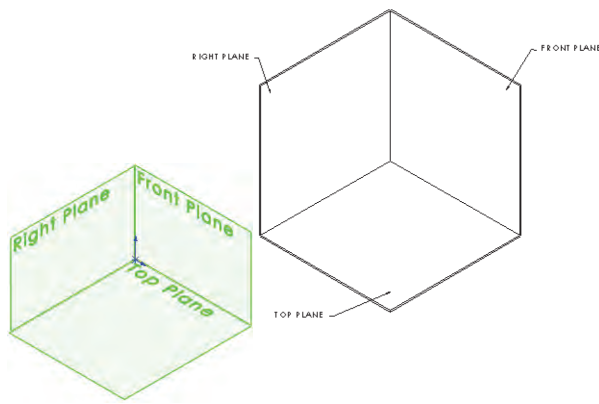
Once the best profile is determined, the next step is to decide which view to use and select the plane with the same name for sketching it. The SOLIDWORKS software provides three planes; they are described below.

Planes

There are three default planes, labeled Front Plane, Top Plane and Right Plane. Each plane is infinite, but has screen borders for viewing and selection. Also, each plane passes through the origin and is mutually perpendicular to the others.

The planes can be renamed. In this course the names Front Plane, Top Plane and Right Plane are used. This naming convention is used in other CAD systems and is comfortable to many users.

Although the planes are infinite, it may be easier to think of them as forming an open box, connecting at the origin. Using this analogy, the inner faces of the box are the potential sketch planes.



Placement of the Model

The part will be placed into the box three times. Each time the best profile will contact or be parallel to one of the three planes. Although there are many combinations, the choices are limited to three for this exercise.

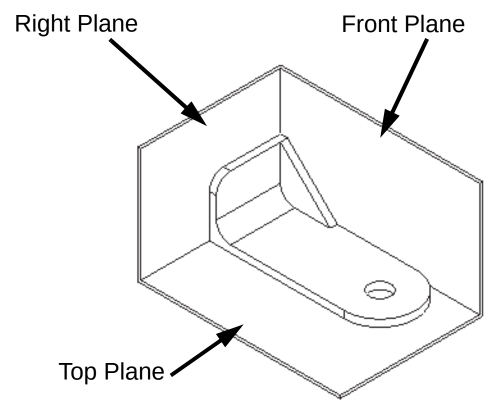
When choosing the sketch plane, consider the part's appearance and its orientation in an assembly. The appearance dictates how the part will be oriented in standard views such as the Isometric. It also determines how you will spend most of your time looking at the model as you create it.

The part's orientation in an assembly dictates how it is to be positioned with respect to other, mating parts.

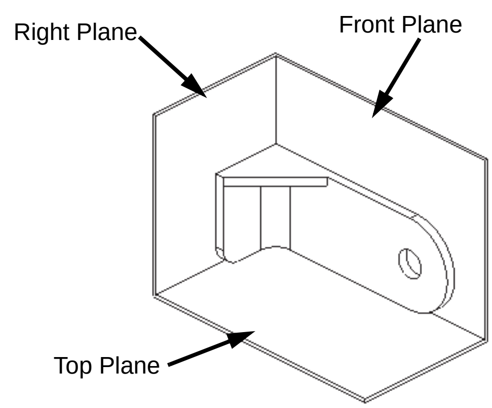
Orient the Model for the Drawing

Another consideration when deciding which sketch plane to use is how you want the model to appear on the drawing when you detail it. You should build the model so that the front of the model is the same as the Front view in the drawing. This saves time during the detailing process because you can use predefined views.

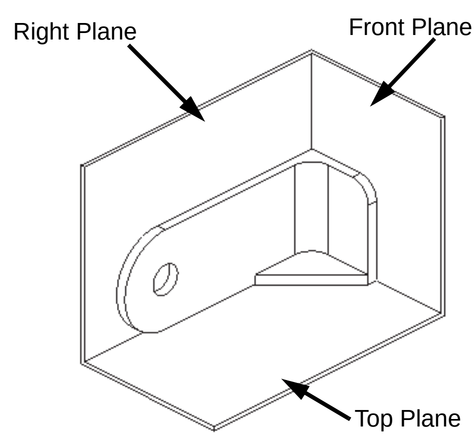
In the first example, the best profile is in contact with the Top plane.



In the second example, it is contacting the Front plane.



The last example shows the best profile in contact with the Right plane.

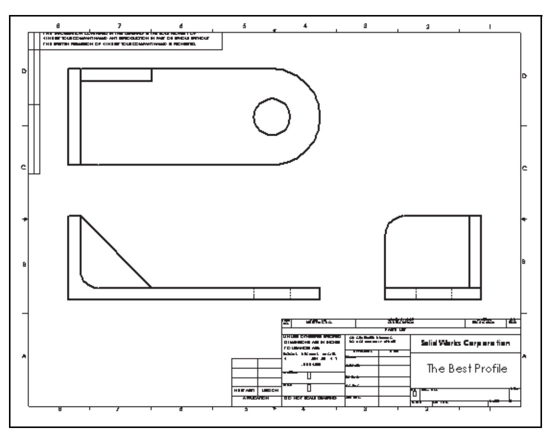


Chosen Plane

The Top plane orientation seems to be the best. This indicates that the best profile should be sketched on the Top plane of the model.

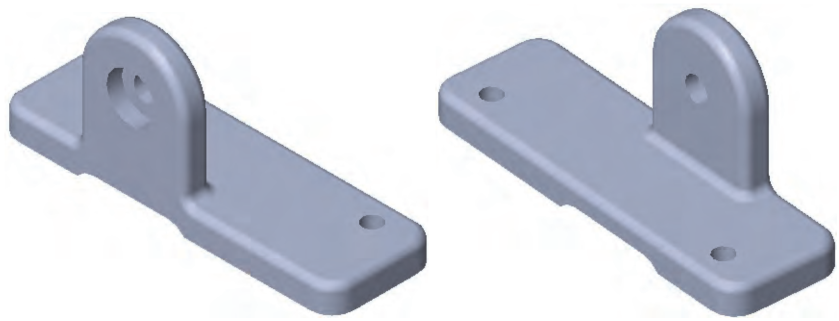
How it Looks on the Drawing

By giving careful thought to which plane is used to sketch the profile, the proper views are easily generated on the detail drawing.



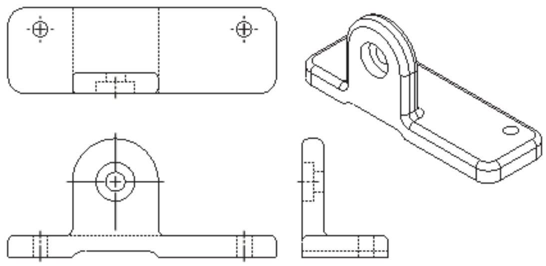
Details of the Part

The part we will be creating is shown below. There are two main boss features, some cuts, and fillets.



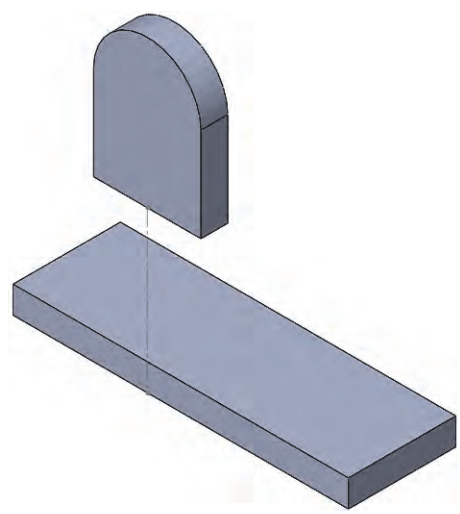
Standard Views

The part is shown here in four standard views.



Main Bosses

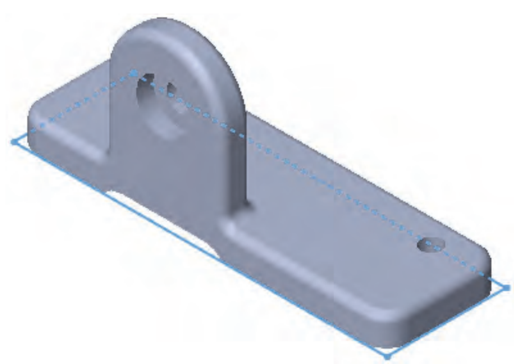
The two main bosses have distinct profiles in different planes. They are connected as shown in the exploded view at right.



Best Profile

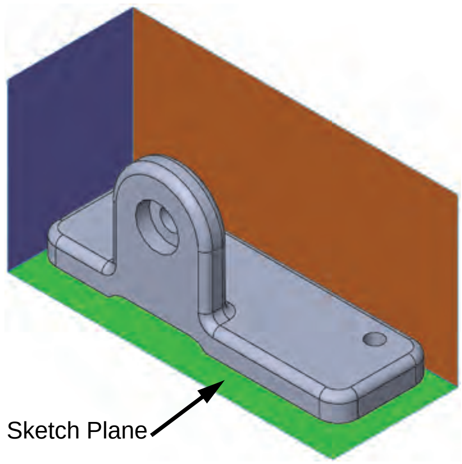
The first feature of the model is created from the rectangular sketch shown overlaid on the model. This is the best profile to begin the model.

The rectangle will then be extruded as a boss to create the solid feature.



Sketch Plane

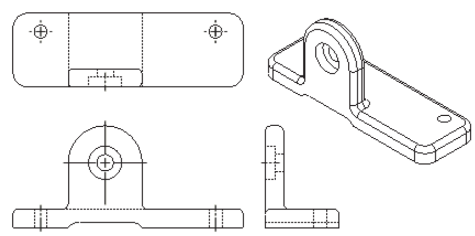
Placing the model "in the box" determines which plane should be used to sketch on. In this case it will be the Top plane.



Design Intent

The design intent of this part describes how the part's relationships should or should not be created. As changes to the model are made, the model will behave as intended.

- All holes are through holes.
- The slot is aligned with the tab.
- The counterbored hole in the front shares the same center point as the rounded face of the tab.



Procedure

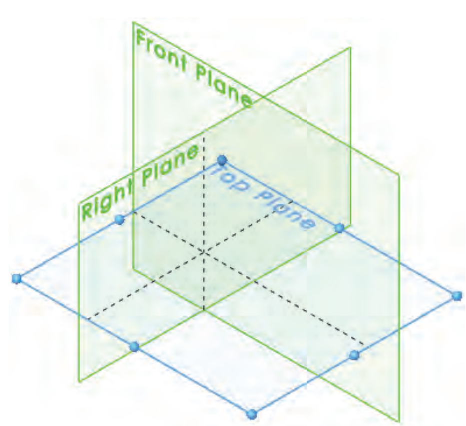
The modeling process includes sketching and creating bosses, cuts and fillets. To begin with, a new part file is created.

1 New part.

Click **New**, or click **File, New**. Create a new part using the Part_MM template and **Save** it as Basic.

2 Select the sketch plane.

Insert a new sketch and choose the Top Plane.



Tip

A plane doesn't have to be shown in order to be used; it can be selected from the FeatureManager design tree.

Sketching the First Feature

Create the first feature by extruding a sketch into a boss. The first feature is always a boss, and it is the first solid feature created in any part. Begin with the sketch geometry, a rectangle.

Introducing: Corner Rectangle

Corner Rectangle is used to create a rectangle in a sketch. The rectangle is comprised of four lines (two horizontal and two vertical) connected at the corners. It is sketched by indicating the locations of two diagonal corners. There are several other rectangle/parallelogram tools available:

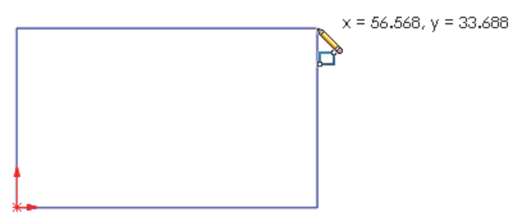
- **Center Rectangle** - Uses a center point and corner to create a rectangle with horizontal and vertical lines.
- **3 Point Center Rectangle** - Creates a rectangle based on a center point, midpoint of edge and corner. Lines are perpendicular at corners.
- **3 Point Corner Rectangle** - Uses three corners to define a rectangle. Lines are perpendicular at corners.
- **Parallelogram** - Uses three corners to define a *parallelogram* (corners are not perpendicular).

Where to Find It

- CommandManager: **Sketch > Corner Rectangle**
- Menu: **Tools, Sketch Entities, Corner Rectangle**
- Shortcut Menu: Right-click in the graphics area and click **Sketch Entities, Corner Rectangle**

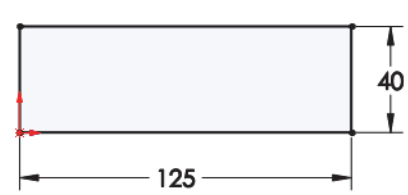
---

3 **Sketch a rectangle.**
Click **Corner Rectangle** and begin the rectangle at the origin.



Make sure the rectangle is locked to the origin by looking for the coincident icon next to the cursor as you begin sketching. Do not worry about the size of the rectangle. Dimensioning it will take care of that in the next step.

4 **Fully defined sketch.**
Add dimensions to the sketch. The sketch is fully defined.



Extrude Options

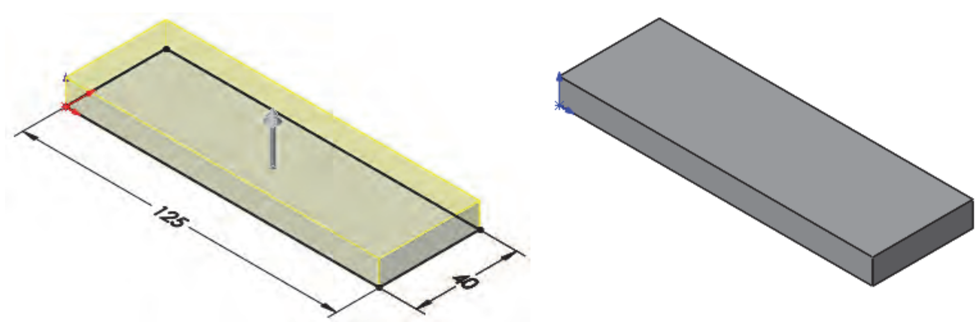
An explanation of some of the more frequently used **Extrude** options is given below (see *Extrude* on page 51). Other options will be discussed in later lessons.

- **End Condition Type**
  A sketch can be extruded in one or two directions. Either or both directions can terminate at some blind depth, up to some geometry in the model, or extend through the whole model.
- **Depth**
  The distance for a blind or mid-plane extrusion. For mid-plane, it refers to the total depth of the extrusion. That would mean that a depth of 50mm for a mid-plane extrusion would result in 25mm on each side of the sketch plane.
- **Draft**
  Applies draft to the extrusion. Draft on the extrusion can be inwards (the profile gets smaller as it extrudes) or outward.

---

5 Extrude.

Click **Extrude** and extrude the rectangle **10mm** upwards. Click **OK**.



Renaming Features

Any feature that appears in the FeatureManager design tree (aside from the part itself) can be renamed using the procedure below. Renaming features is a useful technique for finding and editing features in later stages of the model. Well chosen, logical names help you to organize your work and make it easier when someone else has to edit or modify your model.

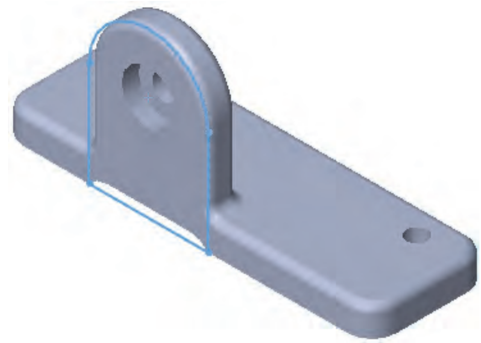
6 Rename the feature.

It is good practice to rename important features that you create with some meaningful name. In the FeatureManager design tree, use a very slow double-click to edit the feature `Boss-Extrude1`. When the name is highlighted and editable, type `BasePlate` as the new feature name. All features in the SOLIDWORKS system can be edited in the same way.

Tip Instead of using a slow double-click to edit the name, you can select the name and press **F2**.

Boss Feature

The next feature will be the boss with a curved top. The sketch plane for this feature will be a planar face of the model instead of an existing plane. The required sketch geometry is shown overlaid on the finished model.

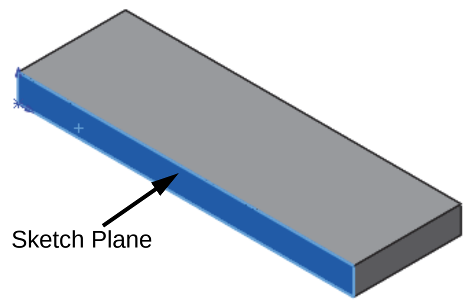


Sketching on a Planar Face

Any planar (flat) face of the model can be used as a sketch plane.

Simply select the face and click **Sketch**. Where faces are difficult to select because they are obscured by other faces, the **Select Other** tool can be used to choose a face without reorienting the view. In this case, the planar face on the front of the `BasePlate` is used.

7 **Insert new sketch.**
Select the indicated face and click **Sketch**.



Note

Make sure that **Features > Instant 3D** is turned off. Leaving it on will cause several handles and axes that we are not currently using to appear on the face.

Sketching

SOLIDWORKS offers a rich variety of sketch tools for creating profile geometry. In this example, **Tangent Arc** is used to create an arc that begins tangent to a selected endpoint on the sketch. Its other endpoint can be placed in space or on another sketch entity.

Introducing: Tangent Arc

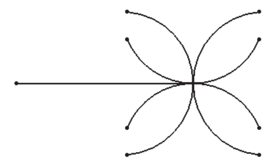
Tangent Arc is used to create tangent arcs in a sketch. The arc must be tangent to some other entity, line or arc, at its start.

Where to Find It

- CommandManager: **Sketch > Arc > Tangent Arc**
- Menu: **Tools, Sketch Entities, Tangent Arc**
- Shortcut Menu: Right-click in the graphics area and click **Sketch Entities, Tangent Arc**

Tangent Arc Intent Zones

When you sketch a tangent arc, the SOLIDWORKS software infers from the motion of the cursor whether you want a tangent or normal arc. There are four intent zones, with eight possible results as shown.



You can start sketching a tangent arc from the end point of any existing sketch entity (line, arc, spline, and so on). Move the cursor away from the end point.

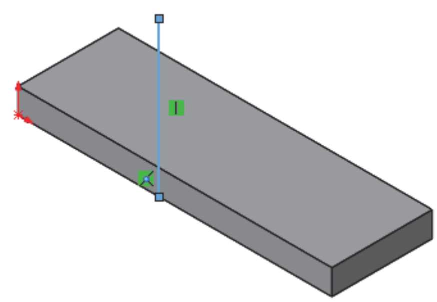
- Moving the cursor in a tangent direction creates one of the four tangent arc possibilities.
- Moving the cursor in a normal direction creates on of the four normal arc possibilities.
- A preview shows what type of arc you are sketching.
- You can change from one type of tangent arc to the other by returning the cursor to the endpoint and moving away in a different direction.

Autotransitioning Between Lines and Arcs

When using **Line**, you can switch from sketching a line to sketching a tangent arc, and back again, without clicking **Tangent Arc**. You can do this by returning the cursor to the endpoint and moving away in a different direction or by pressing the **A** key on the keyboard.

8 Vertical line.

Click **Line** and start the vertical line at the lower edge capturing a **Coincident** relation at the lower edge and **Vertical** relation.



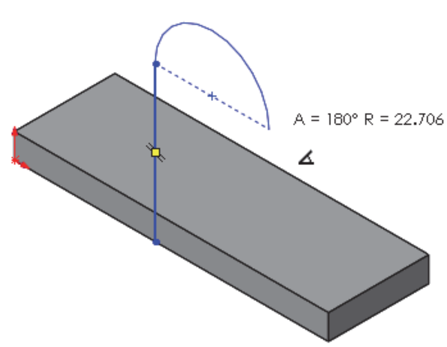
9 Autotransition.

Move the cursor back to the endpoint and move away in a different direction.You are now in tangent arc mode.

10 Tangent arc.

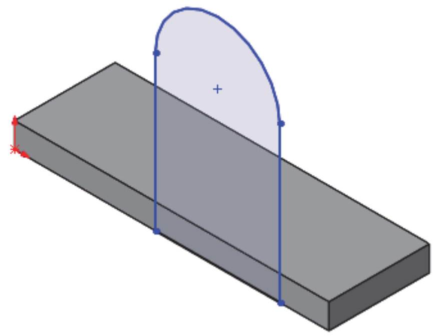
Sketch a 180° arc tangent to the vertical line. Look for the inference line indicating that the end point of the arc is aligned horizontally with the arc's center.

When you finish sketching, the sketch tool automatically switches back to the line tool.



11 Finishing lines.

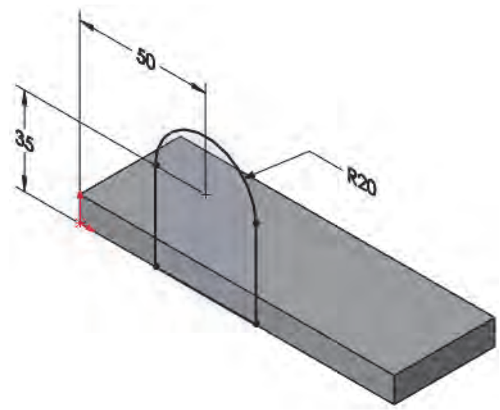
Create a vertical line from the arc end to the base, and one more line connecting the bottom ends of the two vertical lines.



Note that the horizontal line is black, but its endpoints are not.

12 Add dimensions.

Add linear and radial dimensions to the sketch.

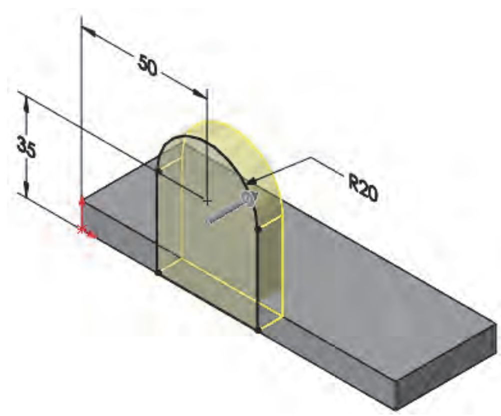


As you add the dimensions, move the cursor around to view different possible orientations.

Always dimension to an arc by selecting on its circumference, rather than center. This makes other dimensioning options (min and max) available.

13 Extrude direction.

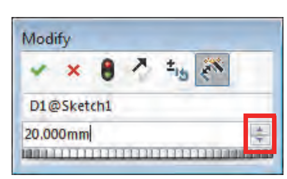
Click **Extrude** and set the **Depth** to **10mm**. Note that the preview shows the extrusion going into the base, in the proper direction.



If the direction of the preview is away from the base, click **Reverse Direction** .

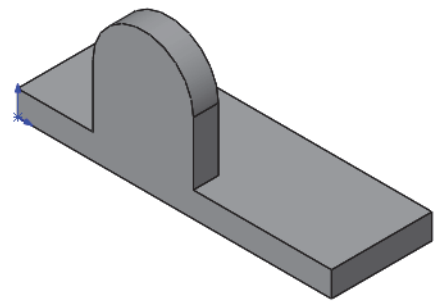
Note

When using the **Spin Increment** arrows, the default up and down increment is 10mm. Pressing the **Alt** key with an arrow drops it to 1/10X, or 1mm. Using it with **Ctrl** key it increases it 10X to 100mm.



14 Completed boss.
The boss merges with the previous base to form a single solid.

Rename the feature `VertBoss`.



Cut Feature

Once the two main boss features are completed, it is time to create a cut to represent the removal of material. Cut features are created in the same way as bosses - in this case with a sketch and extrusion.

Introducing: Cut Extrude

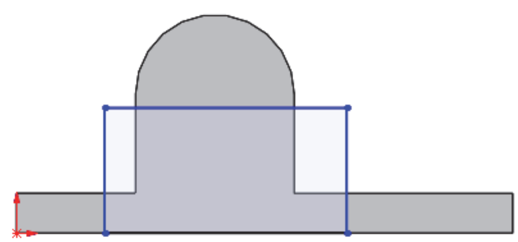
The menu for creating a cut feature by extruding is identical to that of creating a boss. The only difference is that a cut removes material while a boss adds it. Other than that distinction, the commands are the same. This cut represents a slot.

Where to Find It

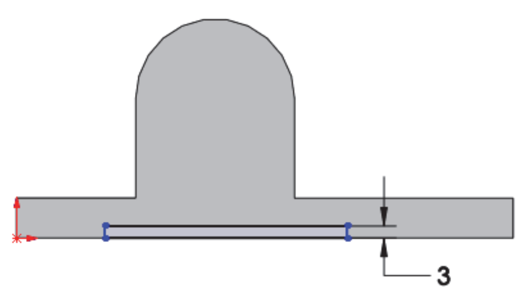
- CommandManager: **Features > Extruded Cut**
- Menu: **Insert, Cut, Extrude**

15 Rectangle.
Press **Space bar** and click `Front`. Start a sketch on this large face and add a rectangle **Coincident** with the bottom model edge.

Turn off the rectangle tool.



16 Dimensions.
Add a dimension as shown.

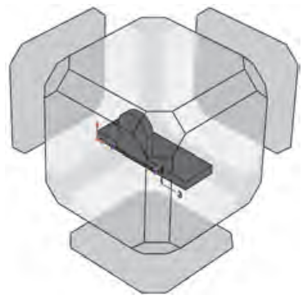


Note

The sketch is under defined, but it will be made fully defined later in this lesson. See *Status of a Sketch* on page 38.

View Selector

The **View Selector** helps to visualize how views of the model will appear by using a transparent cube surrounding the model.



Select a face of the cube to look at the model through the cube, normal to that face or select a view orientation by name.

The cube can also be rotated prior to selecting a face.

Where to Find It

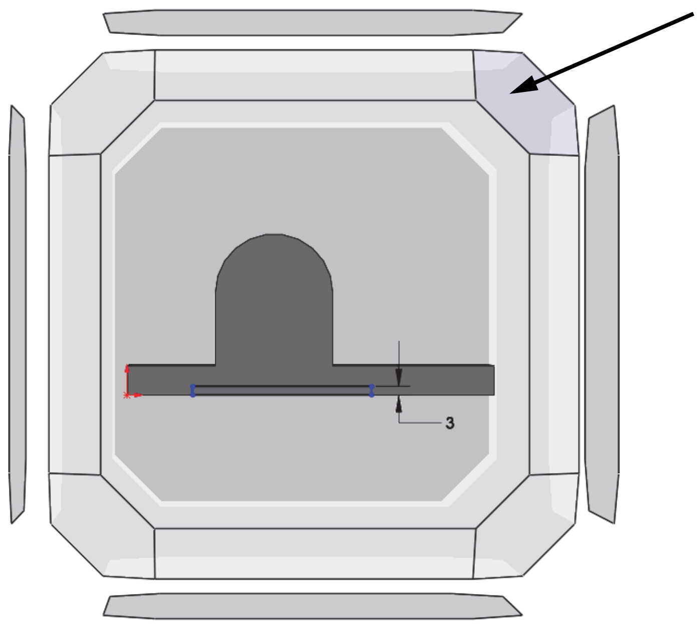
- Heads-up View Toolbar: **View Orientation** and **View Selector**
- Keyboard Shortcut: **Space bar**

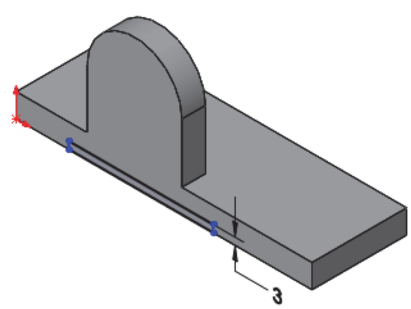
Note

Pressing the **Space Bar** opens the **View Selector** and the **Orientation** dialog box. Pressing **Ctrl+Space bar** opens *only* the **View Selector**.

17 View Selector.

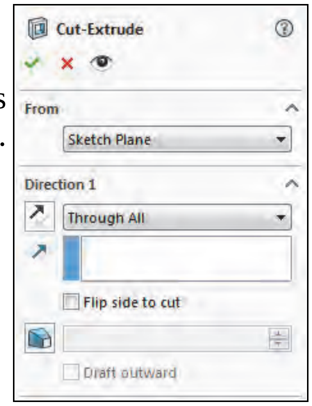
Press **Space Bar** and click the corner of the cube that is labeled Isometric.

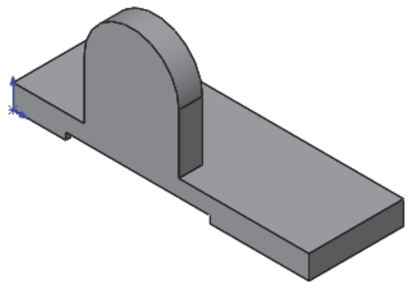




18 Through All Cut.

Click **Extruded Cut** . Choose **Through All** and click **OK**. This type of end condition always cuts through the entire model no matter how far. No depth setting was needed. Rename the feature BottomSlot.





Using the Hole Wizard

The **Hole Wizard** is used to create specialized holes in a solid. It can create simple, tapered, counterbored and countersunk holes using a step by step procedure. I n this example, the **Hole Wizard** will be used to create a standard hole.

Creating a Standard Hole

You can choose the face to insert the hole onto, define the hole's dimensions and locate the hole using the **Hole Wizard**. One of the most intuitive aspects of the **Hole Wizard** is that you specify the size of the hole by the fastener that goes into it.

Tip

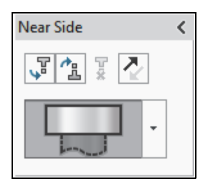
You can also place holes on planes and non-planar faces. For example, you can create a hole on a cylindrical face.

Counterbore Hole

A counterbore hole is required in this model. Using the front face of the model and a relation, the hole can be positioned.

Note

The **Advanced Hole Wizard** (**Insert, Features, Advanced Hole**) is similar to the Hole Wizard, but allows you to design a stack of hole styles including counterbores, countersinks, tapered, tapped, and standard holes.



Introducing: The Hole Wizard

The **Hole Wizard** creates shaped holes, such as countersunk and counterbore types. The process creates two sketches. One defines the shape of the hole. The other, a point, locates the center.

Where to Find It

- CommandManager: **Features > Hole Wizard**
- Menu: **Insert, Features, Hole Wizard**

19 Select Counterbore.
Select the face indicated and click **Hole Wizard**. Set the properties of the hole as follows:

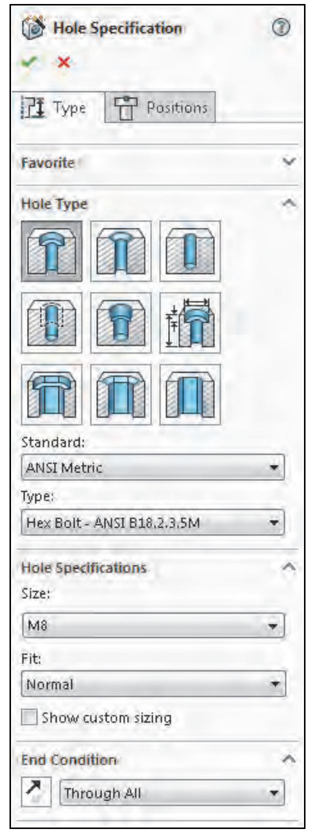
Type: Counterbore

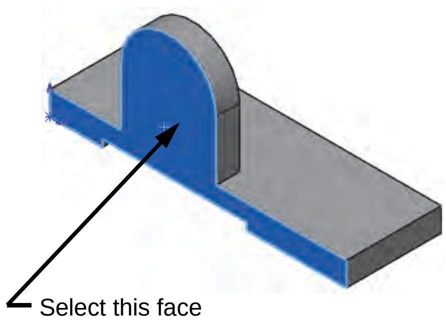
Standard: ANSI Metric

Type: Hex Bolt - ANSI B18.2.3.5M

Size: M8

End Condition: Through All

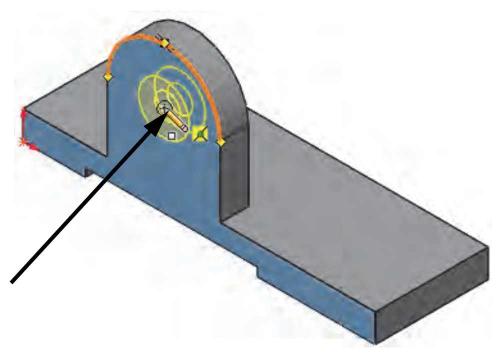




20 Wake up the centerpoint.
Click the **Positions** tab.

Hover the cursor over the circumference of the large arc. *Do not click.*

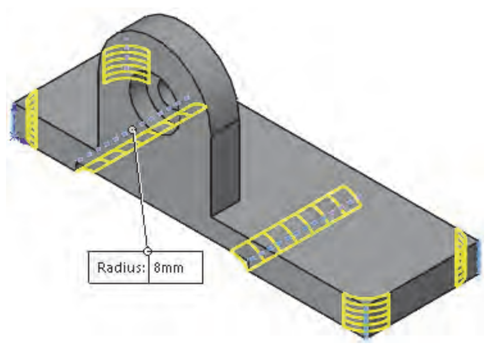
When the **Coincident** symbol appears, the center point of the large arc has been “woken up” and is now a point you can snap to.



Click the point onto the arc’s centerpoint. Look for the feedback that tells you that you are snapping to the arc’s center, a coincident relation. Click **OK** to complete the dialog.

Filleting

Filleting refers to both fillets (adding volume) and rounds (removing volume). The distinction is made by the geometric conditions, not the command itself. Fillets are created on selected edges of the model. Those edges can be selected in several ways, and several options exist for creating different fillet types including constant size, variable size, face and full round fillets. Fillet profile options include circular, conic and curvature continuous.



Note

See the *Advanced Part Modeling* course for more information on fillet types and options.

Filleting Rules

Some general filleting rules are:

1. Leave cosmetic fillets until the end.
2. Create multiple fillets that will have the same radius in the same command.
3. When you need fillets of different radii, generally you should make the larger fillets first.
4. Fillet order is important. Fillets create faces and edges that can be used to generate more fillets.
5. Existing fillets can be converted to chamfers (see *Chamfers* on page 173).

Tip

The *FeatureXpert* can be used to automate the sizing and ordering of fillets.

Selection Toolbar

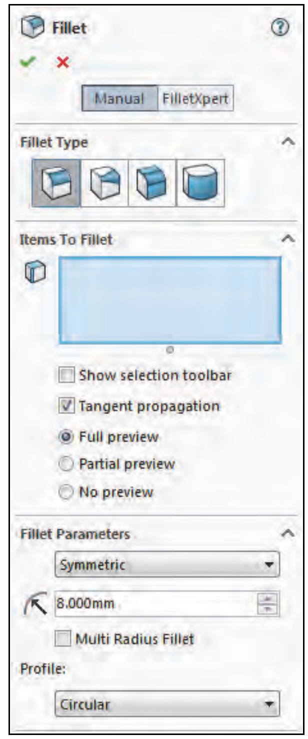
The **Selection Toolbar** can be used to turn a single edge selection into multiple, related, selections. It will not be used in this example, but it will be explained in *Edge Selection* on page 171.

Preview

You have a choice between **Full preview**, **Partial preview** and **No preview** of the fillet. **Full preview**, as shown in the following images, generates a mesh preview on each selected edge. **Partial preview** only generates the preview on the first edge you select. As you gain experience with filleting, you will probably want to use **Partial** or **No preview** because they are faster.

Where to Find It

- CommandManager: **Features > Fillet**
- Menu: **Insert, Features, Fillet/Round**
- Shortcut Menu: Right-click a face or edge and click **Fillet**

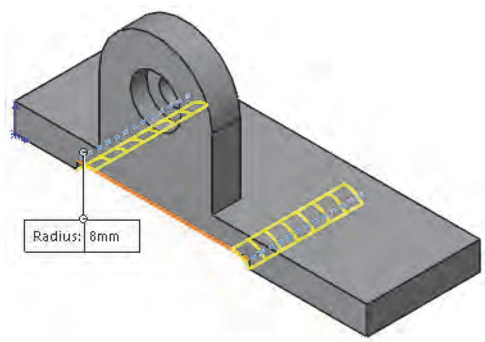


21 Insert Fillet.

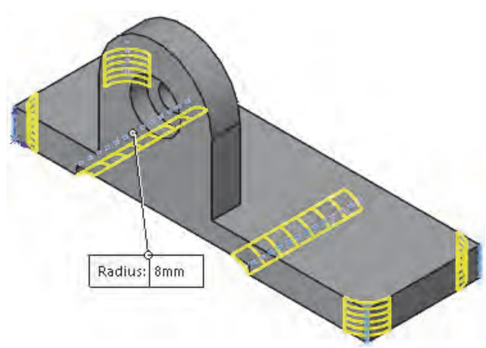
Click **Fillet**. Click **Manual**, click **Constant Size Fillet** and set the radius value to **8mm**.

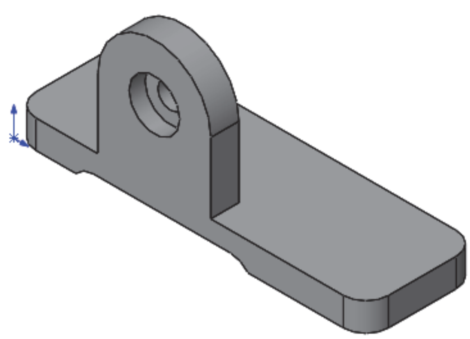
Clear **Show selection toolbar** and click **Full preview**.

22 Select edge.
Select the two hidden edges shown through the model as shown.



23 Additional selections.
Select the additional four corner edges as shown and click **OK**.



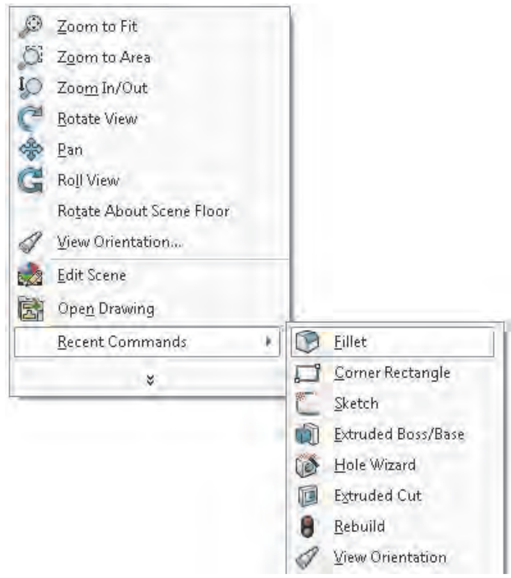


Note

All six fillets are controlled by the same dimension value. The creation of these fillets has generated new edges suitable for the next series of fillets.

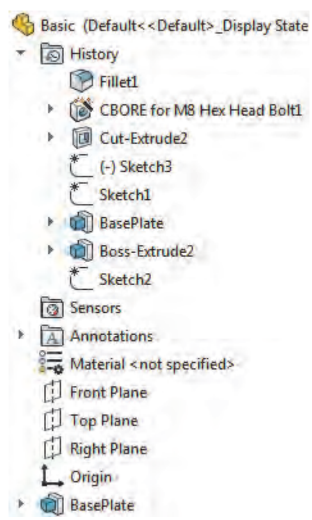
Recent Commands

SOLIDWORKS provides a “just used” buffer that lists the last few commands for easy reuse. The **Enter** key can also be used to re-launch the last used command.



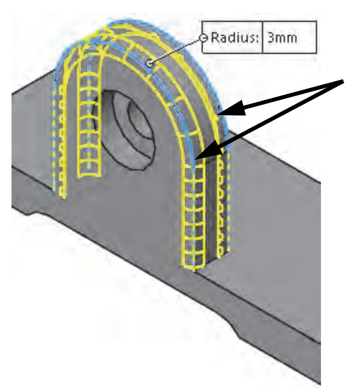
Recent Features

The History folder contains a list of the most recent features that have been created or edited. This is useful for getting access to recent features. See *Editing Tools* on page 81 for more information.



24 Recent Command.
Right-click in the graphics area and click **Recent Commands** and the **Fillet** command from the drop-down list to use it again.

25 Preview and propagate.
Add another fillet, radius **3mm**, using **Full preview**.



Select the edges indicated to see the selected edges and preview.

Click **OK**.

Editing Tools

Three of the most common editing tools are introduced in this lesson: **Edit Sketch**, **Edit Feature** and **Rollback**. They can be used to edit and repair sketches and features as well as specify where, in the FeatureManager design tree, the features are to be created.

Tip

The other editing tools are found later in this lesson: *Editing Features* on page 82 and *Rollback Bar* on page 82.

Editing a Sketch

Once created, sketches can be changed using **Edit Sketch**. This opens the selected sketch so that you can change anything: the dimension values, the dimensions themselves, the geometry or geometric relations.

Introducing: Edit Sketch

Edit Sketch enables you to access a sketch and make changes to any aspect of it. During editing, the model is “rolled back” to its state at the time the sketch was created. The model will be rebuilt when the sketch is exited.

Where to Find It

- Shortcut Menu: Right-click a sketch or feature and click **Edit Sketch**
- Menu: Select a face and click **Edit, Sketch**

---

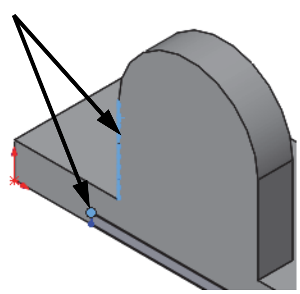
26 Edit the sketch.

Right-click the BottomSlot feature and click **Edit Sketch**. The existing sketch will be opened for editing.

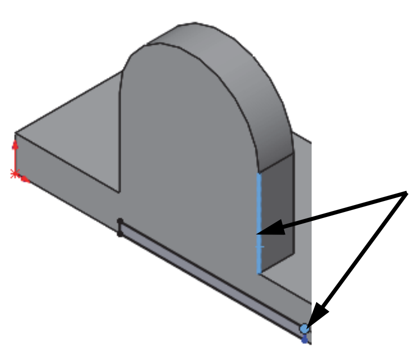
Selecting Multiple Objects

As you learned in *Selecting Multiple Objects* on page 45, when selecting multiple objects, hold down the **Ctrl** key and then select the objects.

27 Relations.
Select the endpoint and edge as shown and add a **Coincident** relation.



28 Repeat.
Repeat the procedure for the endpoint at the other end of the rectangle as shown. The addition of these relations will fully define the sketch.



Note For more information about relations, see *Sketch Relations* on page 41.

29 Exit the sketch.

Click **Exit Sketch** in the upper right (confirmation) corner to exit the sketch and rebuild the part.

Editing Features

The second fillet should also be applied to the top edges of the Base Plate. To do this we will edit the definition of the last fillet feature.

Introducing: Edit Feature

Edit Feature changes how a feature is applied to the model. Each feature has specific information that can be changed or added to, depending on the type of feature it is. As a general rule, the same dialog box used to create a feature is used to edit it.

Fillet Propagation

The **Tangent Propagation** checkbox within the **Fillet** tool allows a fillet feature to flow to tangent edges of the selections made.

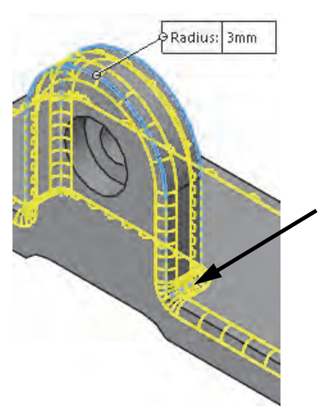
Where to Find It

- Menu: Select a feature and click **Edit, Definition**
- Shortcut Menu: Right-click a feature and click **Edit Feature**

30 Edit the feature.

Right-click the Fillet2 feature and click **Edit Feature**. The existing feature will be opened for editing using the same PropertyManager that was used to create the feature. Make sure that **Tangent Propagation** is clicked.

31 Select additional edge.
Select the additional short edge as shown and the propagation will create the fillets as shown. Click **OK**.



Rollback Bar

The **Rollback Bar** is the blue horizontal bar located at the bottom of the FeatureManager design tree.

The **Rollback Bar** has many uses. It can be used to “walk through” a model showing the steps that were followed to build it or to add features at a specific point in the part’s history. In this example, it will be used to add a hole feature between the existing fillet features.

Using Rollback with Large Parts

The **Rollback Bar** is also useful when editing large parts to limit rebuilding. Roll back to the position just after the feature that you are editing. When the editing is completed, the part is rebuilt only up to the rollback bar. This prevents the entire part from being rebuilt. The part can be saved in a rollback state.

Introducing: The Rollback Bar

You can roll back a part using the **Rollback Bar** in the FeatureManager design tree. The rollback bar is a line which highlights when selected. Drag the bar up or down the FeatureManager design tree to step forward or backward through the regeneration sequence.

Note

To move the rollback bar with the arrow keys, click **Tools, Options, System Options, FeatureManager, Arrow key navigation**. The focus must be set to the rollback bar by clicking on it. If the focus is set to the graphics area, the arrow keys will rotate the model.

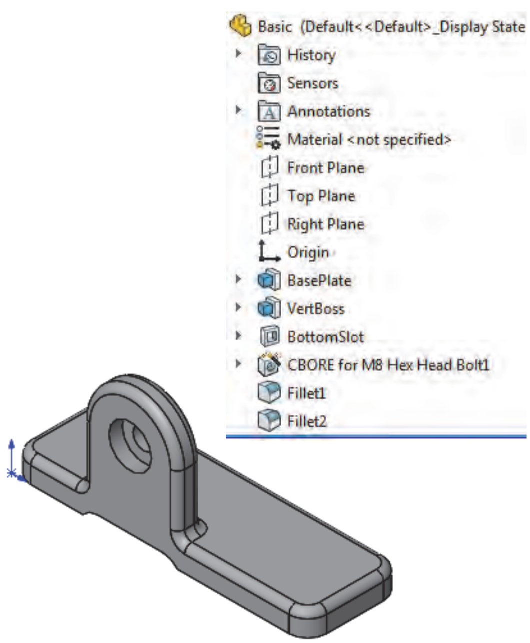
Where to Find It

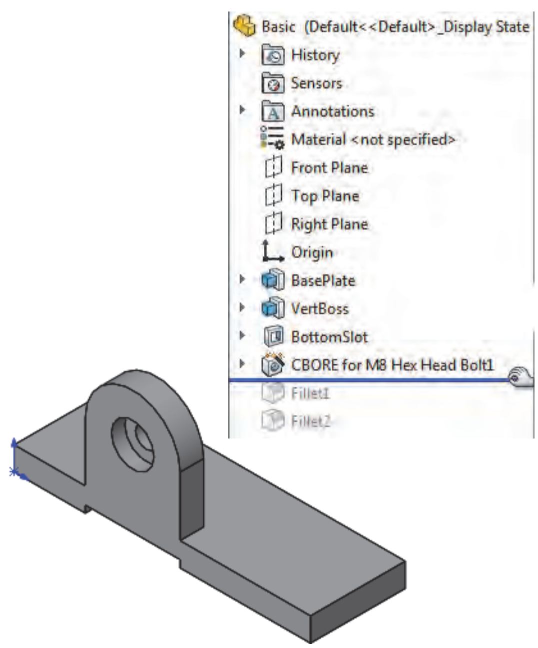
- Shortcut Menu: Right-click a feature and click **Rollback** ↰
- Shortcut Menu: Right-click in the FeatureManager design tree and click **Roll to Previous** or **Roll to End**

---

32 Rollback.

Click on the **Rollback Bar** and drag it upwards. Drop it before the fillet features as shown.



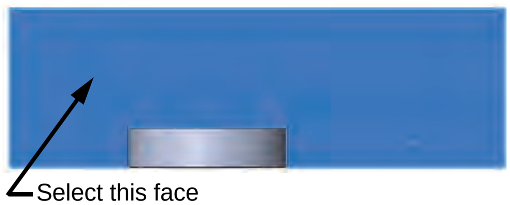


33 Hole Wizard.

Click the **Hole Wizard** and click the **Positions** tab.

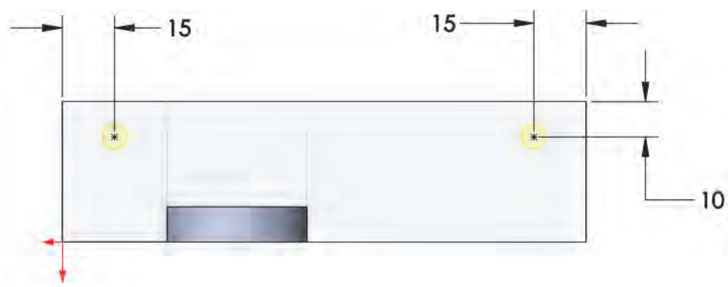
34 Face selection.

Select the face indicated.



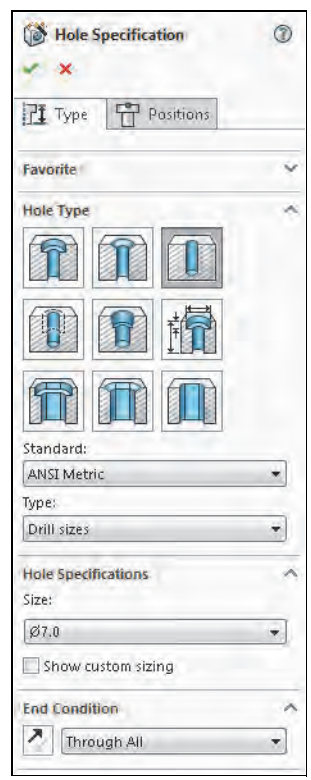
35 Holes.

Add two points and dimension them as shown.



36 Type.

Click the **Type** tab and set the properties of the hole as follows. Click **OK**.



Type: Hole

Standard: Ansi Metric

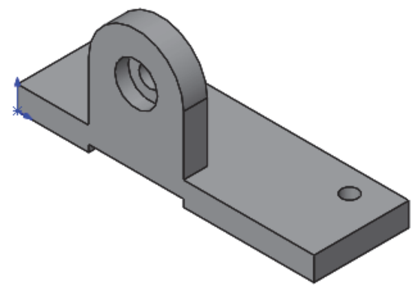
Type: Drill sizes

Size: 7.0

End Condition: Through All

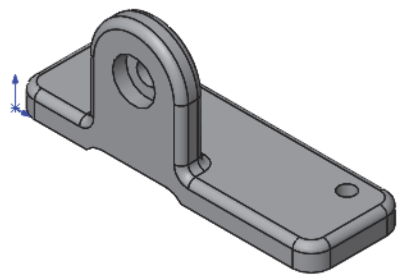
37 Change the view orientation.

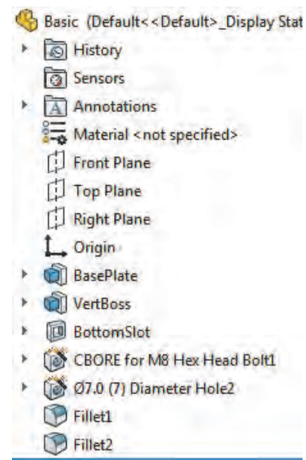
Click **Isometric** to change view orientation.



38 Roll to end.

Right-click on the rollback bar and click **Roll to End**.





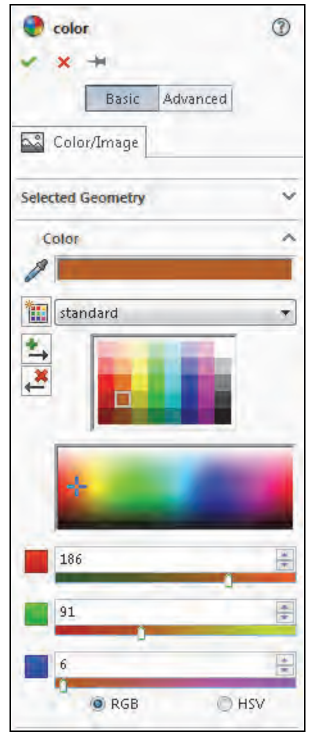
Introducing: Appearances

Use **Appearances** to change the color and optical properties of graphics. Color **Swatches** can also be created for user defined colors.

Where to Find It

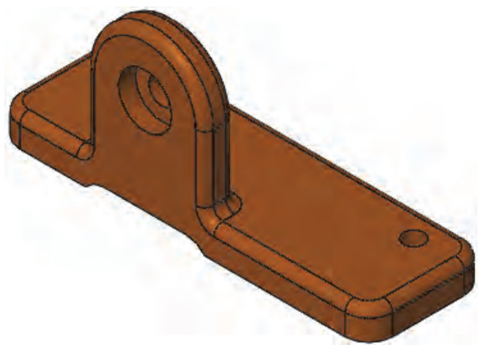
- Shortcut Menu: Right-click a face, feature, body, part, or component, click **Appearances**, and click the item to edit
- Heads-up View Toolbar: **Edit Appearance**

39 Select swatch.

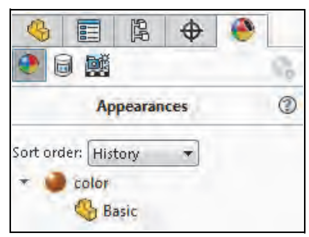


Click **Edit Appearance**. Under the **Color** selection, select the `standard` swatch and one of the colors as shown.

Click **OK**.



40 Display appearances.



Click the **DisplayManager** tab to see the color listed. Click the FeatureManager design tree tab.

Tip

The **DisplayManager** can also be used to view and modify decals, scenes, lights and cameras.

A Note About Color in the User Interface

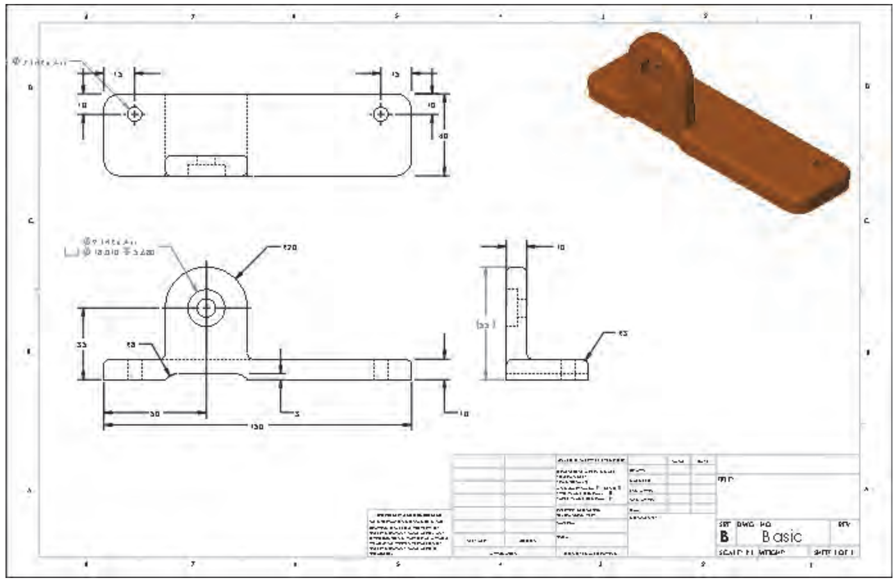
You can customize the colors of the SOLIDWORKS user interface. This is done through **Tools, Options, System Options, Colors**. You can select predefined color schemes, or create your own. In some cases, we have altered colors from their default settings to improve clarity and reproduction quality. As a result, the colors on your system may not match the colors used in this book.

41 Save the results.

Click **Save** to save your work.

Detailing Basics

SOLIDWORKS enables you to easily create drawings from parts or assemblies. These drawings are fully associative with the parts and assemblies they reference. If you change the model, the drawing will update.



Various topics related to making drawings are integrated into several lessons throughout this book. The material presented here is just the beginning. Specifically:

- Creating a new drawing file and sheet.
- Creating drawing views using the View Palette.
- Using dimension assist tools.

A comprehensive treatment of detailing is offered in the course *SOLIDWORKS Drawings*.

Settings Used in the Template

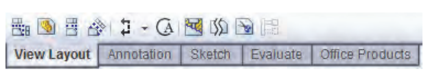
The drawing template used in this section has been designed to include the **Document Properties** shown in the chart below. Settings are accessed through **Tools, Options**. The settings that will be used in this lesson are:

System Options	Document Properties (Set using drawing template)
Drawings, Display Style: • **Display style for new views = Hidden lines removed** • **Tangent Edges = Visible**	Drafting Standard: • **Overall drafting standard = ANSI**
Colors: • **Drawings, Hidden Model Edges = Black**	Dimensions: • **Font = Century Gothic** • **Primary precision = .123** • **Add parentheses by default = Selected**
	Detailing, Auto insert on view creation: • **All options = cleared**
	Units • **Unit system =MMGS**

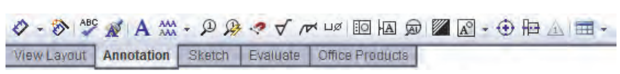
CommandManager Tabs

When working in a drawing document, the CommandManager tabs will update to include toolbars that are specific to the process of detailing and making drawings. They are:

- **View Layout**



- **Annotation**



New Drawing

Drawing files (*.SLDDRW) are SOLIDWORKS files that contain drawing sheets. Each sheet is the equivalent of a single sheet of paper.

Introducing: Make Drawing from Part

Make Drawing from Part takes the current part and steps through the creation of a drawing file, sheet format and initial drawing views using that part.

Where to Find It

- Menu Bar: **New**, **Make Drawing from Part/Assembly**
- Menu: **File, Make Drawing from Part**

1 Create Drawing.

Click **Make Drawing from Part/Assembly** and choose B_Size_ANSI_MM from the **Training Templates** tab.

The sheet format creates a B-size drawing (11” x 17”) arranged with its long edge horizontal. The sheet format includes a border, title block, and other graphics.

Tip

Double-clicking the template will automatically open it, eliminating the need to click **OK**.

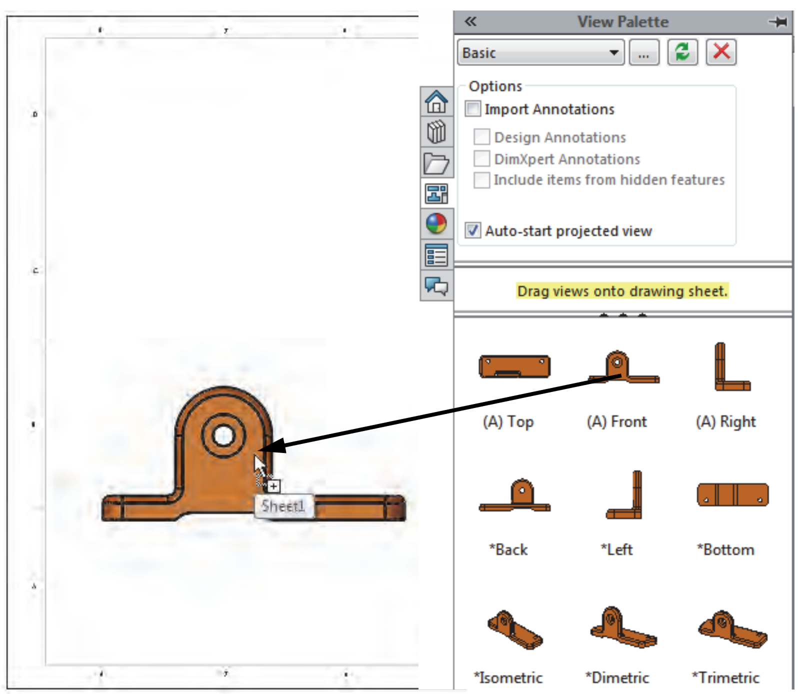
Drawing Views

The initial task of detailing is the creation of views. Using the **Make Drawing from Part/Assembly** tool leads you through the selection of the drawing sheet to the **View Palette**. Previews of the model orientations are shown in the lower pane of the View Palette. Create views on the drawing sheet by using a drag and drop procedure. Additional views can be projected or folded directly from the dropped view.

These options are discussed in detail in the *SOLIDWORKS Drawings* course.

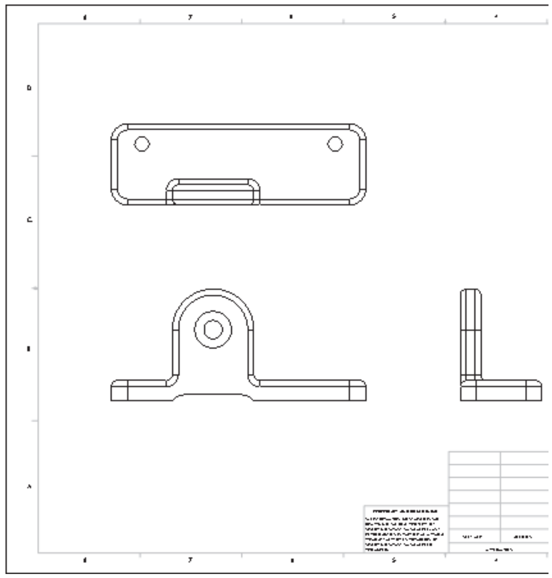
2 View Palette.

Clear **Import Annotations**. Drag the Front view from the **View Palette** and drop it onto the drawing as shown.

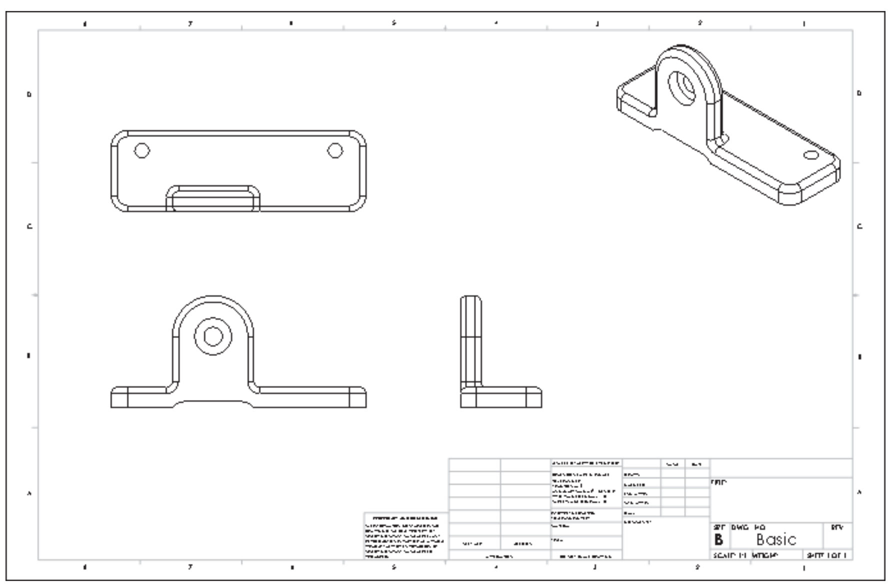


3 Projected views.
Once the first view is placed, **Projected View** become active. Add the Top view by moving the cursor above the view and clicking.

Return the cursor to the Front view and move to the right to create the Right view. Click **OK**.



4 Drawing views.
Add the *Isometric view by dragging and dropping from the palette. Place it in the upper right corner.



Note The part document is still open. You can press **Ctrl+Tab** to switch between the drawing and part document windows.

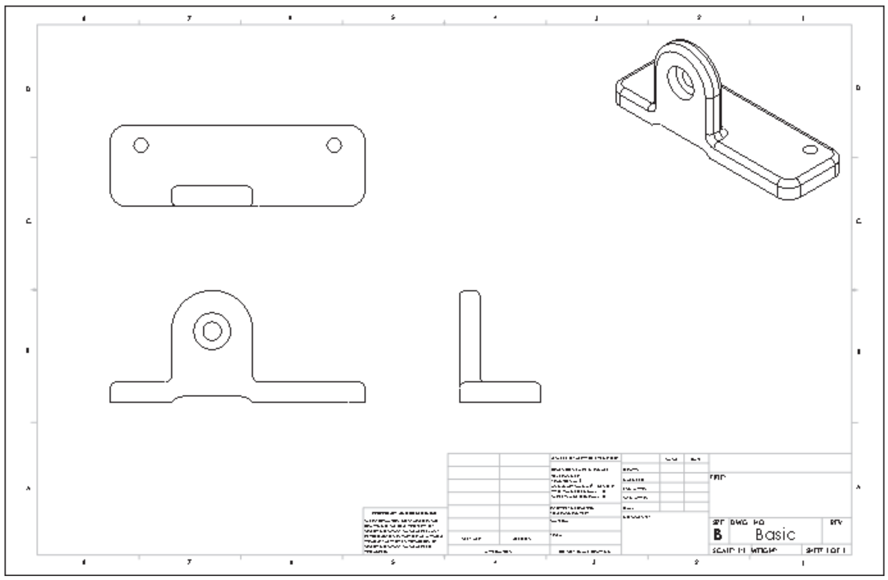
Tangent Edges

Tangent Edges are topological edges of faces that match in tangency. The most commonly seen tangent edges are the edges of fillets. They are often made visible in pictorial views but are removed from orthographic views.

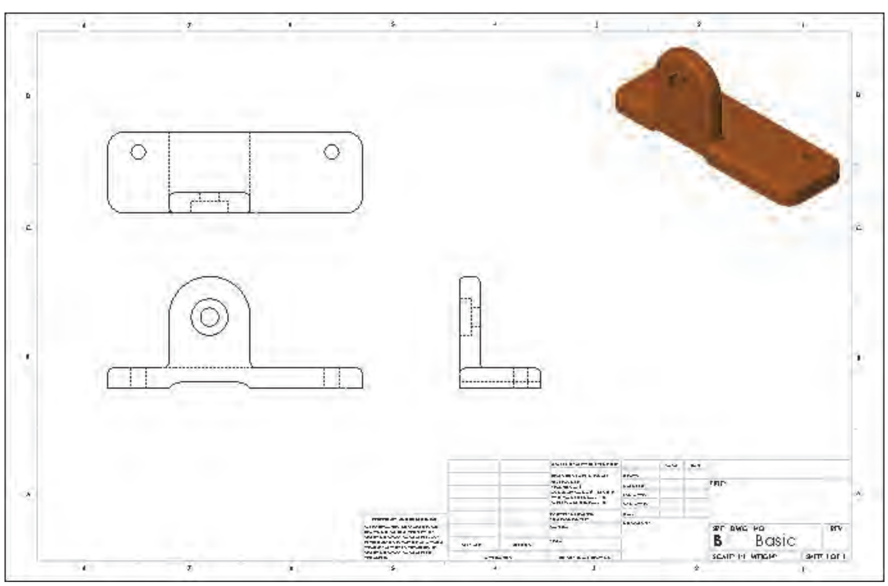
Where to Find It

- Shortcut Menu: Right-click the view and click **Tangent Edge**

5 Remove tangent edges.
Using the **Control** key, select the front, top, and right views. Click **Tangent Edge** and **Tangent Edges Removed**.



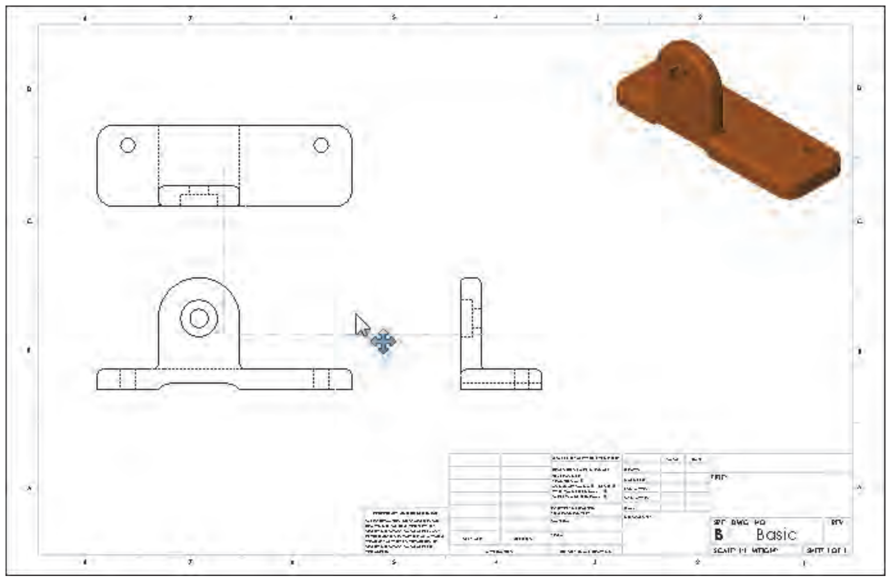
6 Display style.
Click the Isometric view and click **Shaded**. In the other views, click **Hidden Lines Visible**.



Moving Views

Drawing views can be repositioned on the drawing. You place your pointer over the view border, then drag the view. In the standard 3 view arrangement, the `Front` view is the *source* view. This means that moving the front view moves all three views. The `Top` and `Right` views are *aligned* to the `Front`. They can only move along their axis of alignment.

7 Move Aligned Views.
Select the edge and move the `Front` view. It can be moved in any direction and the other views remain aligned.



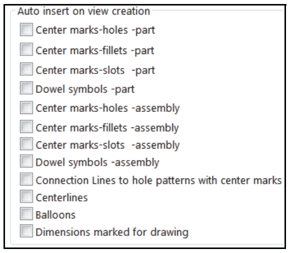
Note

Once the drawing view has been selected, it can be dragged with the mouse or moved with the arrow keys. The distance moved for each press of an arrow key is set under **Tools**, **Options**, **System Options**, **Drawings**, **Keyboard movement increment**. Use **Alt-drag** to select anywhere in the view. Use **Shift-drag** to maintain the spacing between the views while dragging.

Center Marks

Center Marks are attached to circle and arc centers in the drawing view.

Center marks were not inserted into the drawing views automatically. You can turn this option on or off. Set your preference using the **Tools, Options, Document Properties, Detailing** menu.



Where to Find It

- CommandManager: **Annotation > Center Mark**
- Menu: **Insert, Annotations, Center Mark**
- Shortcut Menu: Right-click in the graphics area and click **Annotations, Center Mark**

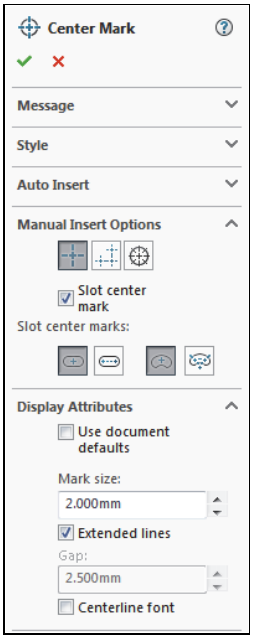
8 Center Mark.

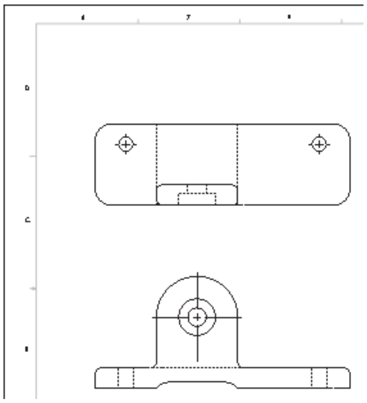
Click **Center Mark** .

Clear **Use document defaults**, check the **Extended lines** option and set the **Mark size** to **2mm** as shown.

Click the large arc in the front view. Continue adding center marks to the two holes in the Top view.

Click **OK**.





Dimensioning

Dimensions can be created in drawing views using several tools. Some dimensions can be related to the dimensions generated in the sketches and features of the model. These are *driving* dimensions. Other dimensions are independent of the sketches and features of the model. These are *driven* dimensions.

Driving Dimensions

Driving dimensions always display the proper values and can be used to change the model. The **Model Items** tool imports the dimensions created in the sketches and features of the model into the drawing.

Driven Dimensions

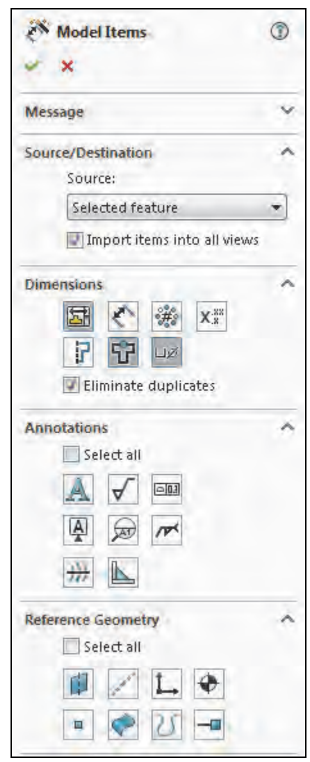
Driven dimensions always display the proper values but cannot be used to change the model. The values of driven dimensions change when the model dimensions change. By default, dimensions of this type appear in a different color and are enclosed in parentheses. Here are two ways to create driven dimensions:

- The **Smart Dimension** tool manually adds dimensions to the model like those in a sketch.
- The **DimXpert** tool adds dimensions working from a datum position.

Introducing: Model Items

The **Model Items** tool assists in adding dimensions to a view or all views using the sketch and feature dimensions of the model.

You can import the dimensions for a selected feature or the entire model. It also has the capability to select and import different types of dimensions as well as many types of **Annotations** and **Reference Geometry** that may exist within the model.



Where to Find It

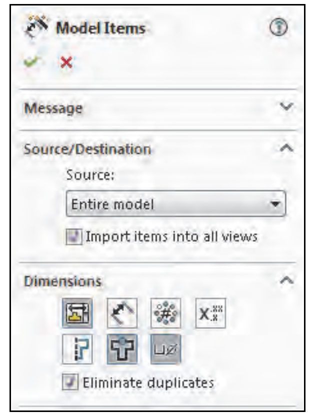
- CommandManager: **Annotation > Model Items**
- Menu: **Insert, Model Items**

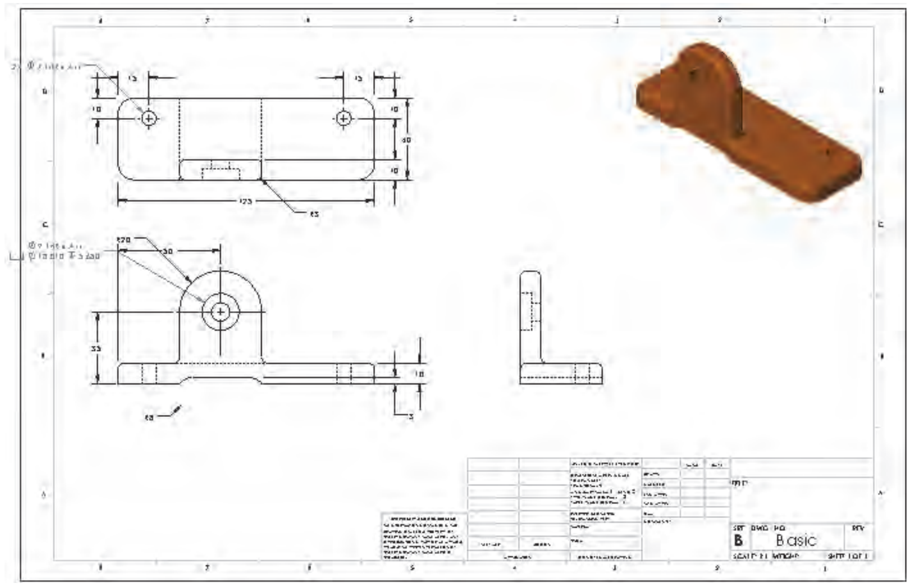
9 Model items.

Click **Model Items** . Click **Entire Model** as the **Source** and **Import items into all views**.

Under **Dimensions**, click **Marked for drawing**, **Hole Wizard Locations, Hole callout** and **Eliminate duplicates**.

Click **OK**.





Note The position of a dimension depends on how the feature was created and where the model dimension was placed. Your results may vary from the image above.

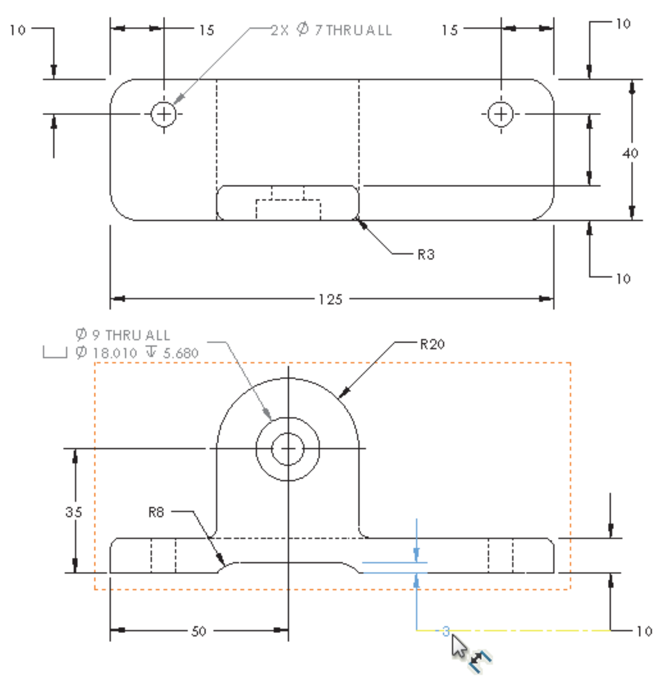
Tip Once the dimensions are inserted, they are associated to that view and will move with it unless you deliberately move them to another view or delete them. For more information, see *Manipulating Dimensions* on page 96.

Manipulating Dimensions

Once dimensions have been added to a view, there are several options as to how they can be manipulated:

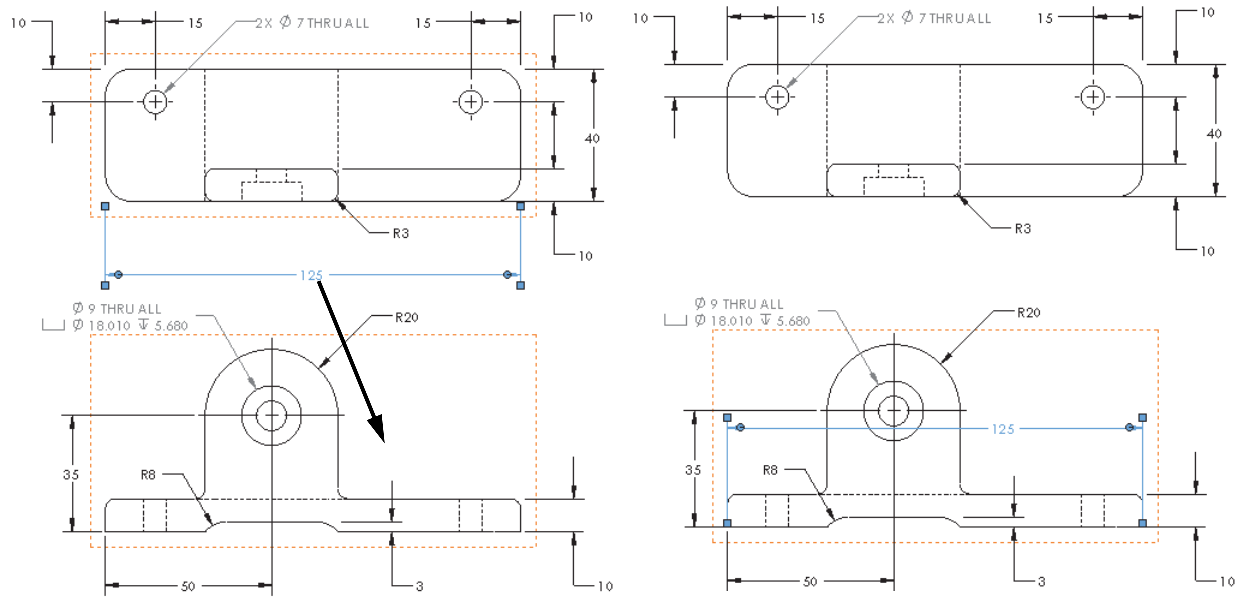
Drag	Drag dimensions by their text to new locations. Use the inference lines to align and position them.
Hide	Right-click the dimension text and click **Hide** from the shortcut menu.
Move to another view	There is generally more than one view where a dimension can be used. To move a dimension, **Shift** + drag the dimension onto another view.
Copy to another view	To copy the dimension, hold down **Ctrl** and drag it into another view and drop it.
Delete	Unwanted dimensions can be deleted from the drawing using the **Delete** key.

10 Drag dimensions.
Drag dimensions within the view to reposition them as shown.

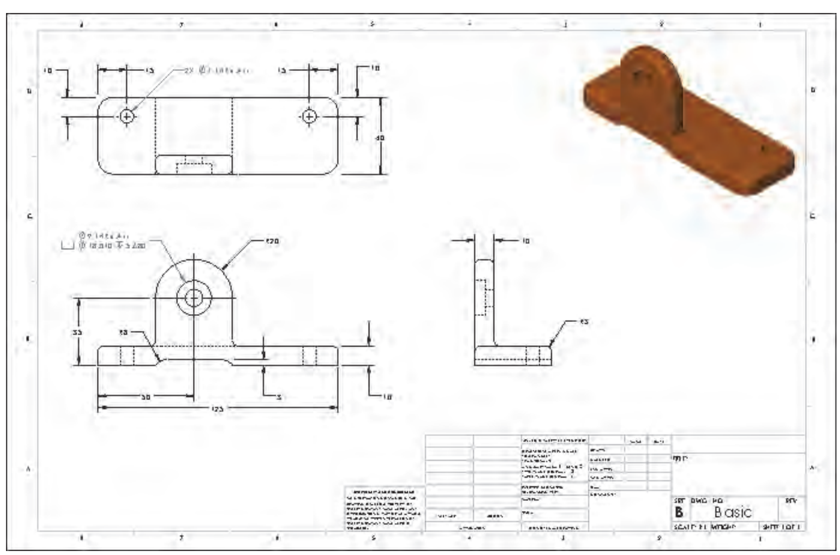


Tip Align dimension text using the yellow guidelines.

11 Move to another view.
Shift + drag the **125mm** dimension to `Drawing View1` and drop it. It will be moved from the original view to the new view.



12 Move remaining dimensions.
Move dimensions to reposition them as shown.



Dimension Palette

The **Dimension Palette** appears near your cursor when you insert a dimension or select one or more dimensions. It can be used to change the dimensions’ properties, formatting, position, and alignment.

Where to Find It

- Select one or more dimensions then click

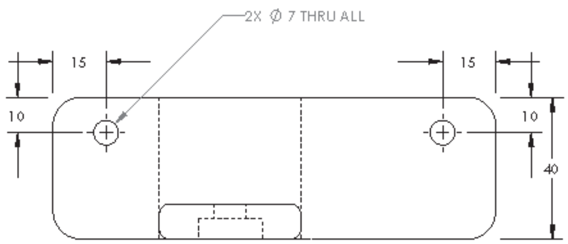
Dimension Assist Tool - Smart Dimensioning

Use the **Smart dimensioning** option of the dimension assist tool to manually add dimensions in the drawing. These dimensions are considered to be *driven* dimensions. See *Driven Dimensions* on page 94.

---

13 Arrange the dimensions.

Select all of the dimensions in the top view and click to open the **Dimension Palette**. Then, click **Auto Arrange Dimensions** to provide better spacing and alignment of the dimensions.

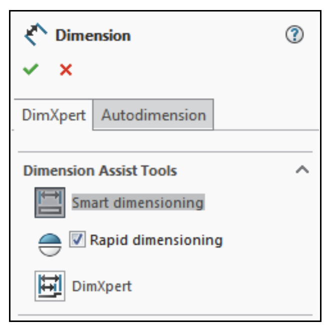


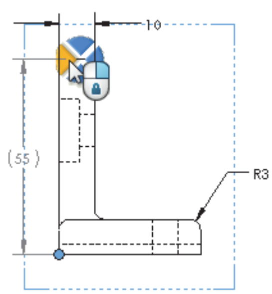
Note Adjustments can be made to dimensions after using arrange.

14 Dimensioning.

Click **Smart Dimension** . Select vertices at the top and bottom and place the dimension to the left of the view.

Click **OK**.

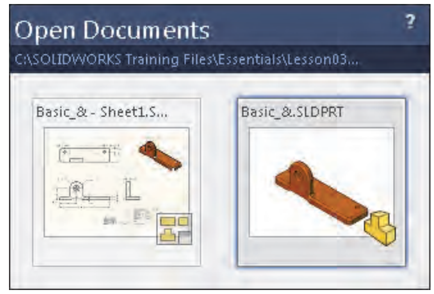




Associativity Between the Model and the Drawing

In the SOLIDWORKS software, everything is associative. If you make a change to an individual part, that change will propagate to any and all drawings and assemblies that reference it.

15 Switch windows.
Press **Ctrl+Tab** and click the part file to switch back to the part document window.



Changing Parameters

SOLIDWORKS makes it very easy to make changes to the dimensions of your part. This ease of editing is one of the principal benefits of parametric modeling. It is also why it is so important to properly capture your design intent. If you don't properly capture the design intent, changes to dimensions may cause quite unexpected results in your part.

Rebuilding the Model

After you make changes to the dimensions, you must rebuild the model to cause those changes to take affect.

Rebuild Symbol

If you make changes to a sketch or part that require the part to be rebuilt, a rebuild symbol is displayed beside the part's name as well as superimposed on the icon of the feature that requires rebuilding BasePlate. Look for the rebuild icon on the Status Bar, also.

The rebuild symbol also is displayed when you are editing a sketch. When you exit the sketch, the part rebuilds automatically.

Introducing: Rebuild

Rebuild regenerates the model with any changes you have made.

Where to Find It

- Menu Bar: **Rebuild**
- Menu: **Edit, Rebuild**
- Keyboard Shortcut: **Ctrl+B**

Tip

The model is also rebuilt when it is saved.

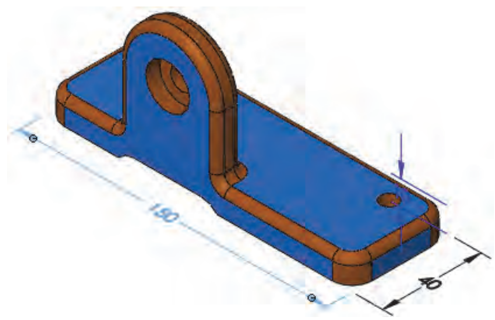
Note

To rebuild *all* features, press **Ctrl+Q**.

16 Double-click on the feature.

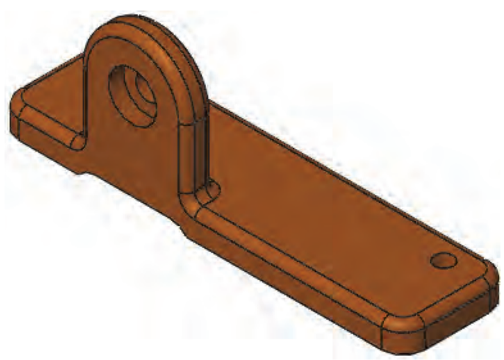
You can double-click on the BasePlate feature either in the FeatureManager design tree or the graphics area. When you do this, the parameters associated with the feature will appear.

Double-click on the **125mm** dimension indicated. The **Modify** dialog box will appear. Enter a new value either by typing it directly or by using the spin box arrows. Enter **150mm** and click **OK**.

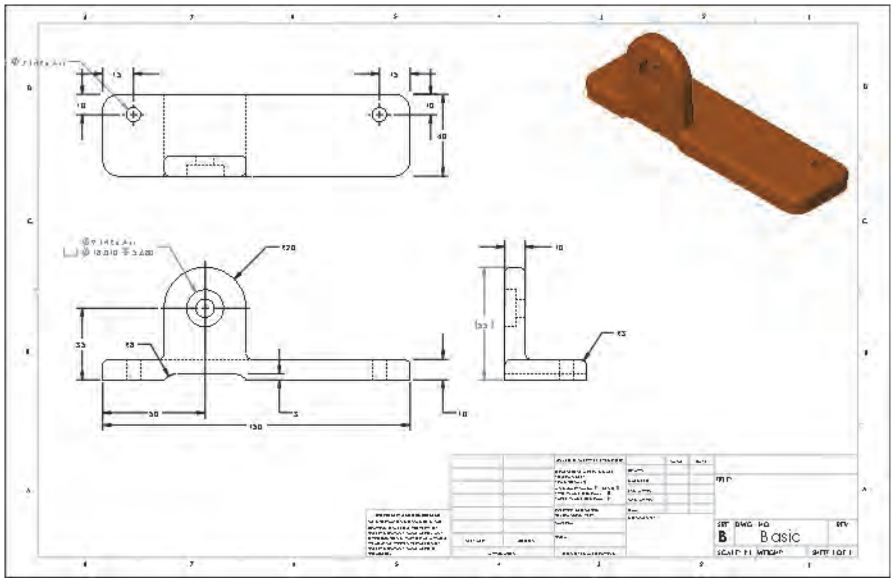


17 Rebuild the part to see the results.

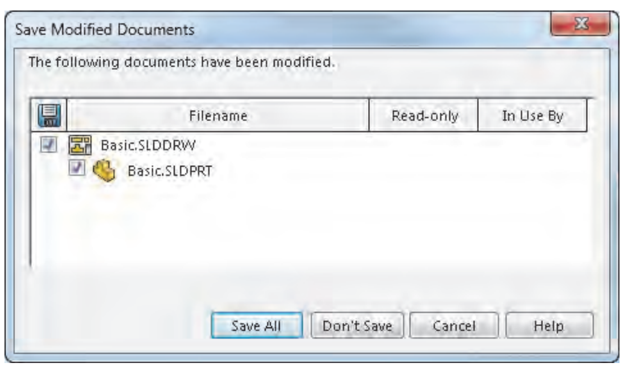
Rebuild the part by clicking **Rebuild** . If you use the one on the **Modify** dialog box, the dialog box will stay open so you can make another change. This makes exploring "what if" scenarios easy.



18 Update the drawing.
Press **Ctrl+Tab** and click the drawing file to switch back to the drawing sheet. The drawing will update automatically to reflect the changes in the model. Dimensions may move during the rebuilding process and require some clean up.



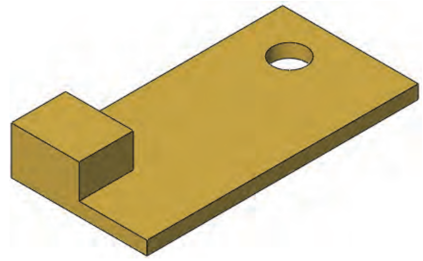
19 Close the drawing.
Click **File, Close** to close the drawing. Click **Save All** to save both the drawing and part files.



20 Confirm.
Click **Yes** to update the drawing views before saving the drawing. Save the drawing file in the same folder as the part.

Exercise 7: Plate

Create this part using the information and dimensions provided. Sketch and extrude profiles to create the part. This lab reinforces the following skills:



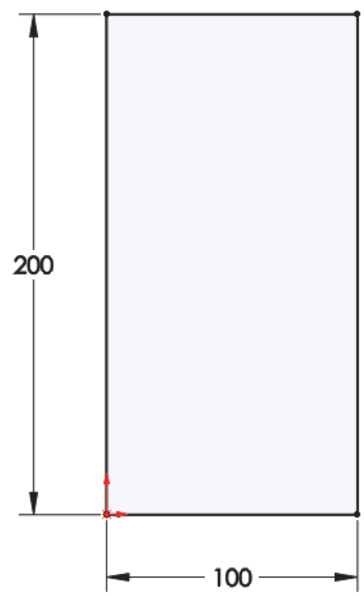
- *Choosing the Best Profile* on page 64
- *Introducing: Corner Rectangle* on page 69
- *Sketching on a Planar Face* on page 71
- *Boss Feature* on page 71
- *Using the Hole Wizard* on page 76

Units: **millimeters**

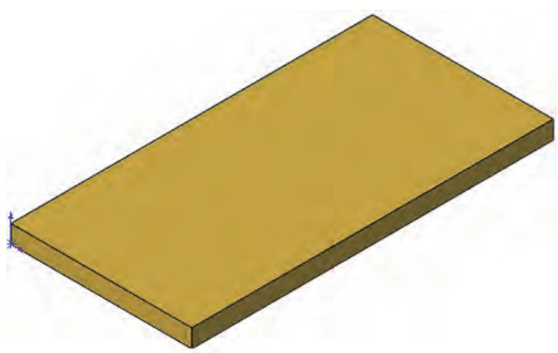
Procedure

Create a new mm part and name it Plate. Create the geometry as shown in the following steps.

1 Sketch base feature.
Create a new sketch on the Top plane. Add the geometry and dimensions as shown.

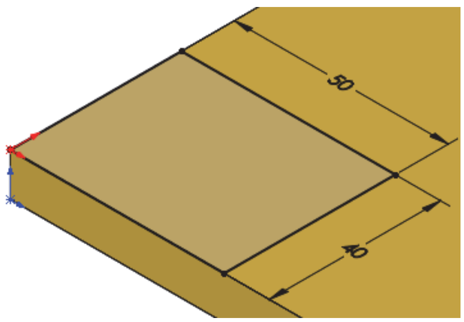


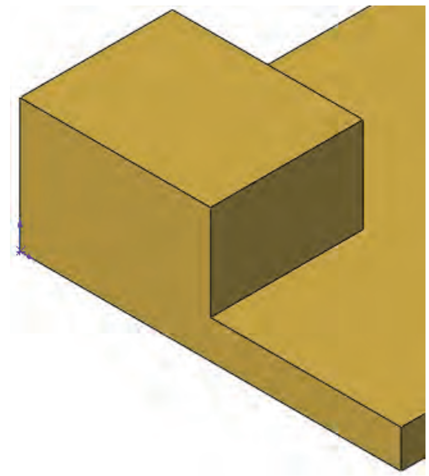
2 Extrude base feature.
Extrude the sketch **10mm** as shown.



3 Boss.
Create a new sketch on the top face of the solid. Add the geometry and dimensions as shown.

Extrude a boss **25mm**.





4 Hole Wizard.
Click **Hole Wizard** and click the face shown.

Click the **Type** tab. Set the properties of the hole as follows:

Type: Hole

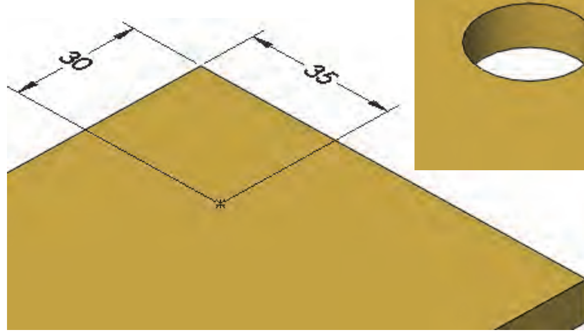
Standard: Ansi Metric

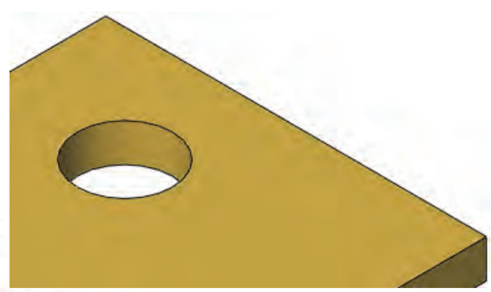
Type: Drill sizes

Size: 25mm

End Condition: Through All

Click the **Positions** tab. Place the points as shown.

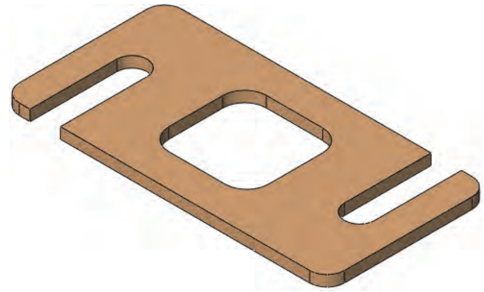




5 Save and close the part.

Exercise 8: Cuts

Use rectangles, tangent arcs and cut features to create the part. This lab reinforces the following skills:



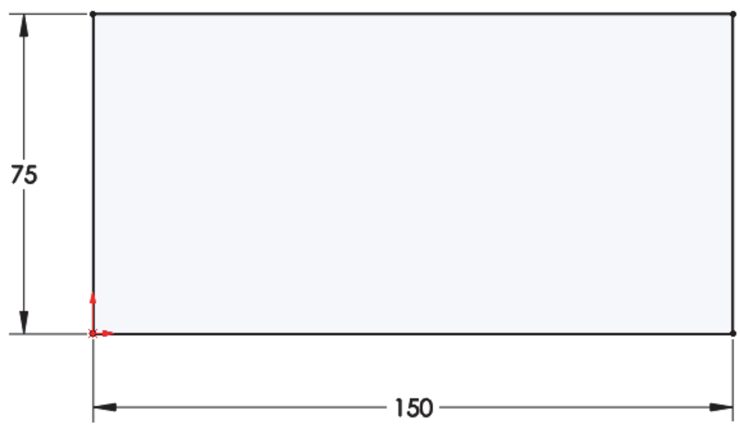
- *Introducing:*
  Corner Rectangle on page 69
- *Tangent Arc Intent Zones* on page 72
- *Cut Feature* on page 74
- *Filleting* on page 78

Units: **millimeters**

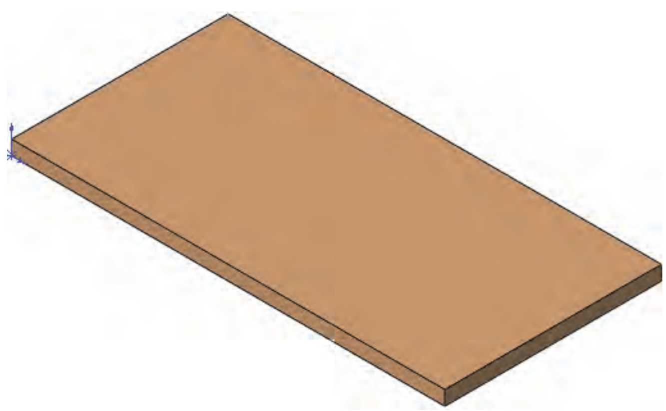
Procedure

Create a new mm part and name it `Cuts`. Create the geometry as shown in the following steps.

1 Sketch base feature.
Create a new sketch on the `Top` plane. Add the geometry and dimensions as shown.

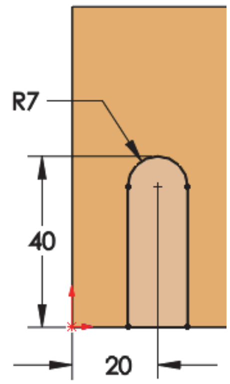


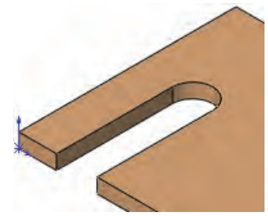
2 Extrude base feature.
Extrude the sketch **5mm** as shown.



3 Cut slot.
Create a new sketch on the top face of the solid. Add the geometry and dimensions as shown.

Extrude a cut using **Through All**.



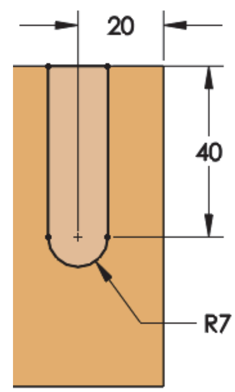


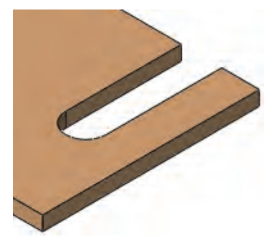
Tip

Remember to create a closed profile by sketching the line across the bottom.

4 Cut another slot.
Create a new sketch using the same face. Add the geometry and dimensions as shown.

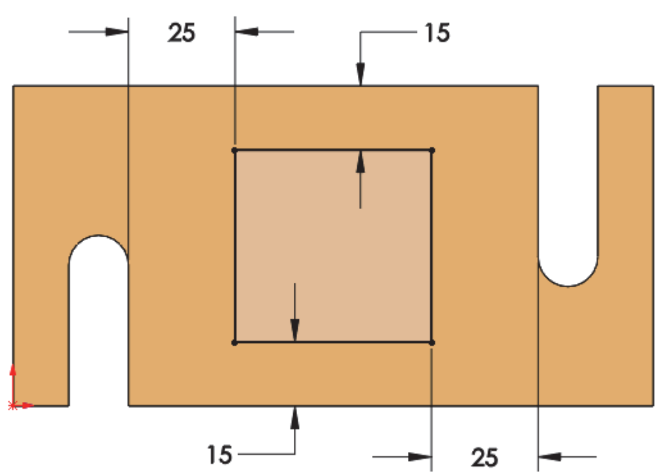
Extrude another cut using **Through All**.





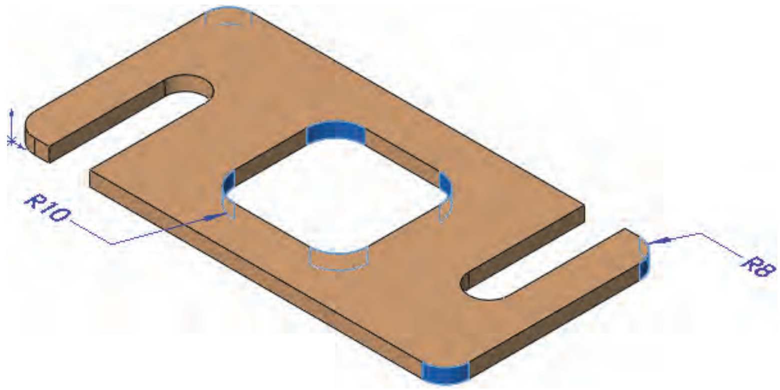
5 Cut rectangle.
Create a new sketch using the same face. Add the geometry and dimensions as shown.

Extrude another cut using **Through All**.



6 Fillets.

Add fillets of **R10mm** and **R8mm** to the edges as shown.



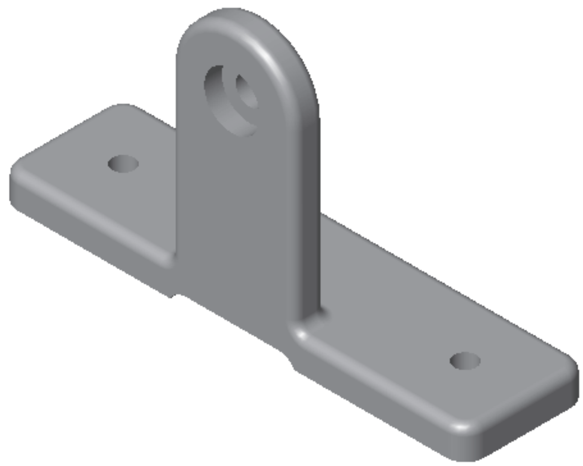
7 Save and close the part.

Exercise 9: Basic-Changes

Make changes to the part created in the previous lesson.

This exercise uses the following skills:

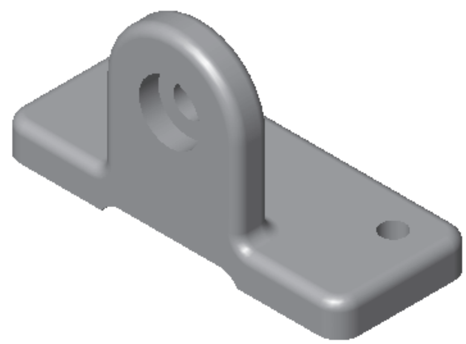
- *Changing Parameters* on page 99
- *Rebuilding the Model* on page 99



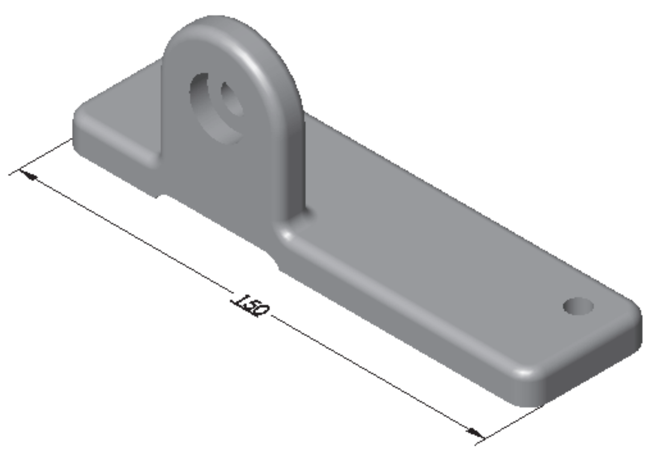
Procedure

Open an existing part and edit it.

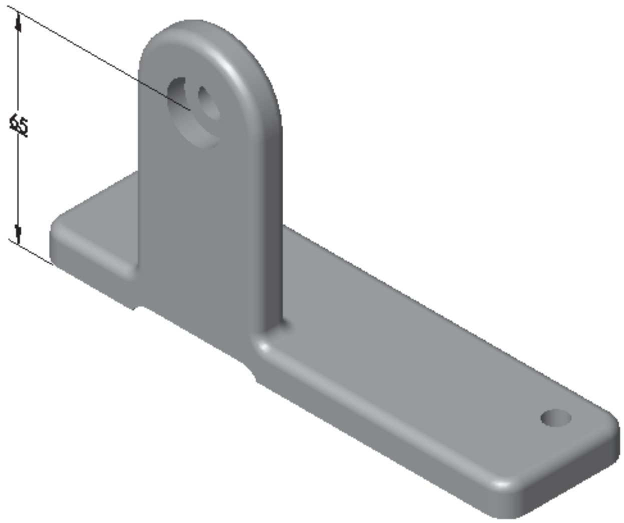
1 Open the part Basic-Changes.
Several changes will be performed on the model to resize it and check the design intent.



2 Overall dimension.
Double-click the first feature (Base Plate) in the FeatureManager design tree or on the screen to access the dimensions. Change the length dimension to **150mm** (shown bold and underlined below) and rebuild the model.



3 Boss.
Double-click the `Vert boss` feature and change the height dimension as shown. Rebuild the part.

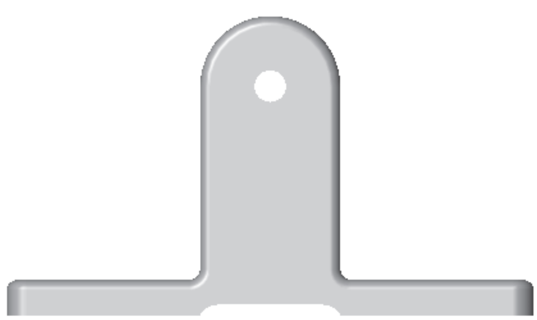


4 Hole locations.
Double-click the `Ø7.0 (7) Diameter Hole1` feature and change the position dimensions to **20mm**. Rebuild the model.

5 Center the `Vert Boss`.
Determine the proper value and change the dimension that centers the `Vert Boss` on the base.

Tip

Optionally, you can delete the dimension and add a relations that centers the `VertBoss` relative to the base.



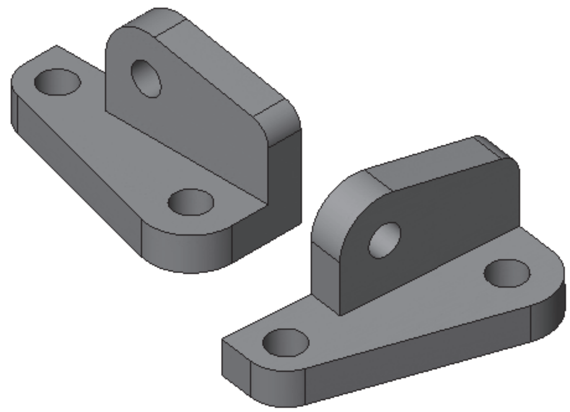
6 Save and close the part.

Exercise 10: Base Bracket

This lab reinforces the following skills:

- *Choosing the Best Profile* on page 64
- *Boss Feature* on page 71
- *Using the Hole Wizard* on page 76
- *Filleting* on page 78

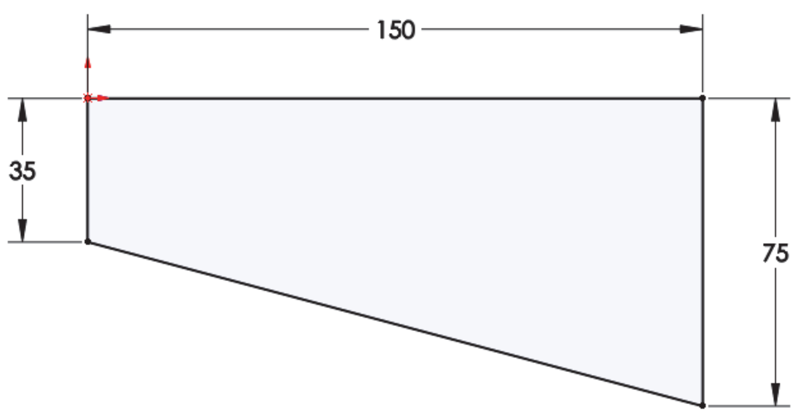
Units: **millimeters**



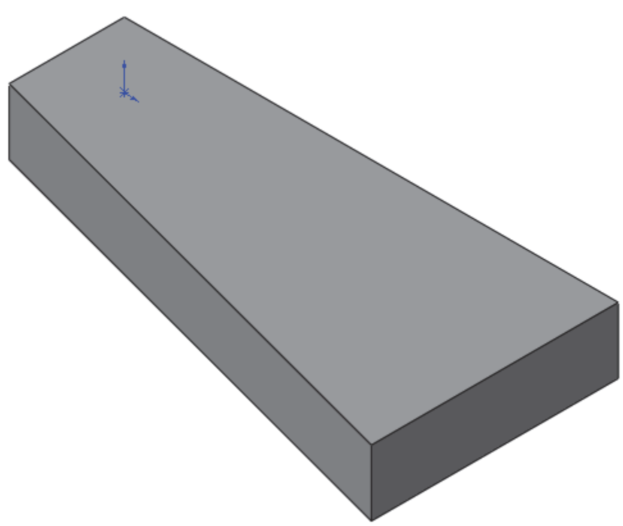
Procedure

Create a new mm part and name it `Base_Bracket`. Create the geometry as shown in the following steps.

1 **Sketch base feature.**
Create a new sketch on the `Top` plane. Add the geometry and dimensions as shown.

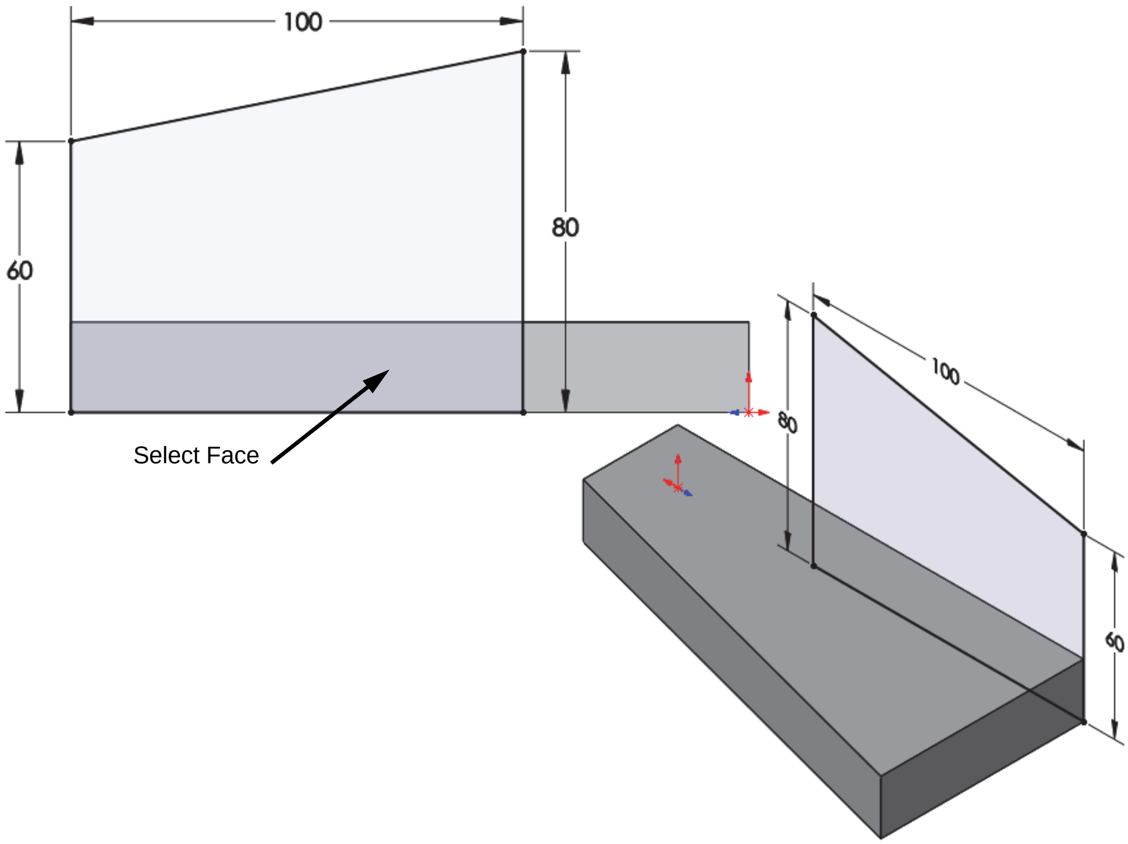


2 **Extrude base feature.**
Extrude the sketch **20mm** to create the base feature as shown.



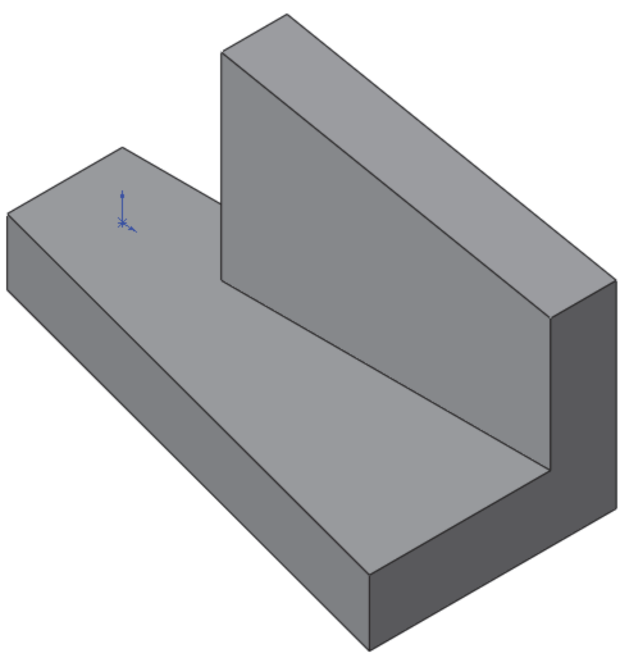
3 Sketch on rear face.

Change to the Back view orientation, select the face indicated and create a new sketch. Add the geometry and dimensions as shown.

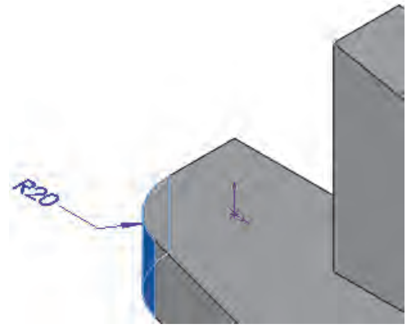
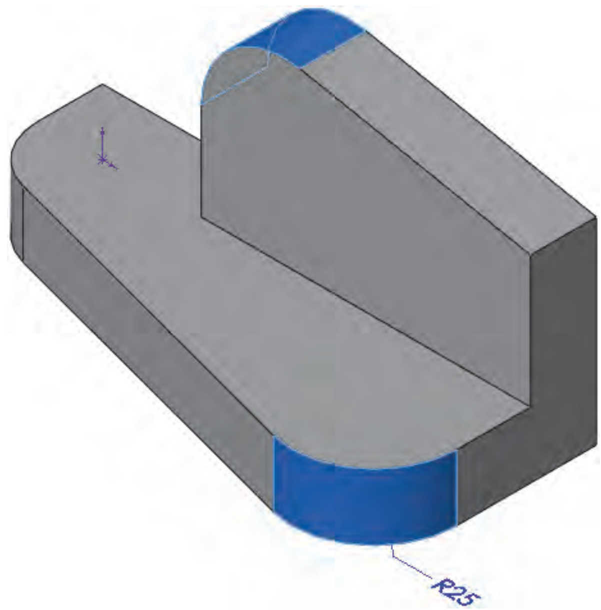
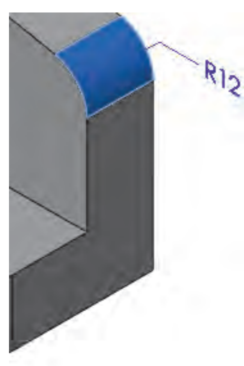


4 Extrude boss.

Extrude to the sketch **20mm** as shown.



5 Fillets.
Add fillets to the edges as shown.

R20mm	
R25mm	
R12mm	

6 Hole Wizard.

Click **Hole Wizard** and click the face shown. Click the **Type** tab and set the properties of the hole as follows:

Type: Hole

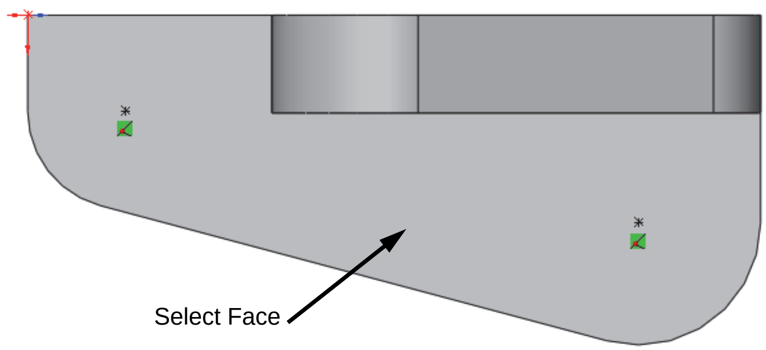
Standard: Ansi Metric

Type: Drill sizes

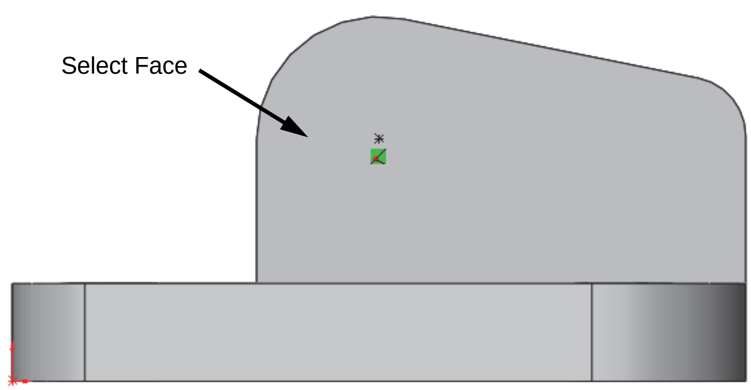
Size: 20mm

End Condition: Through All

Click the **Positions** tab and locate the holes as shown.



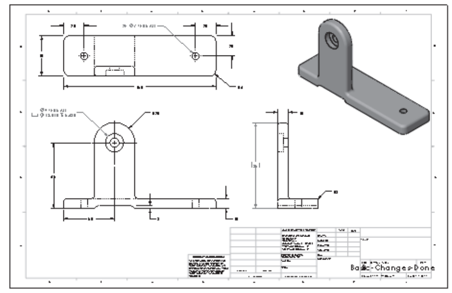
7 Second hole.
Repeat the procedure to create an **18mm** hole on a different face as shown.



8 Save and close the part.

Exercise 11: Part Drawings

Create this part drawing using the information provided.



This lab reinforces the following skills:

- *New Drawing* on page 88
- *Drawing Views* on page 89
- *Center Marks* on page 93
- *Dimensioning* on page 94

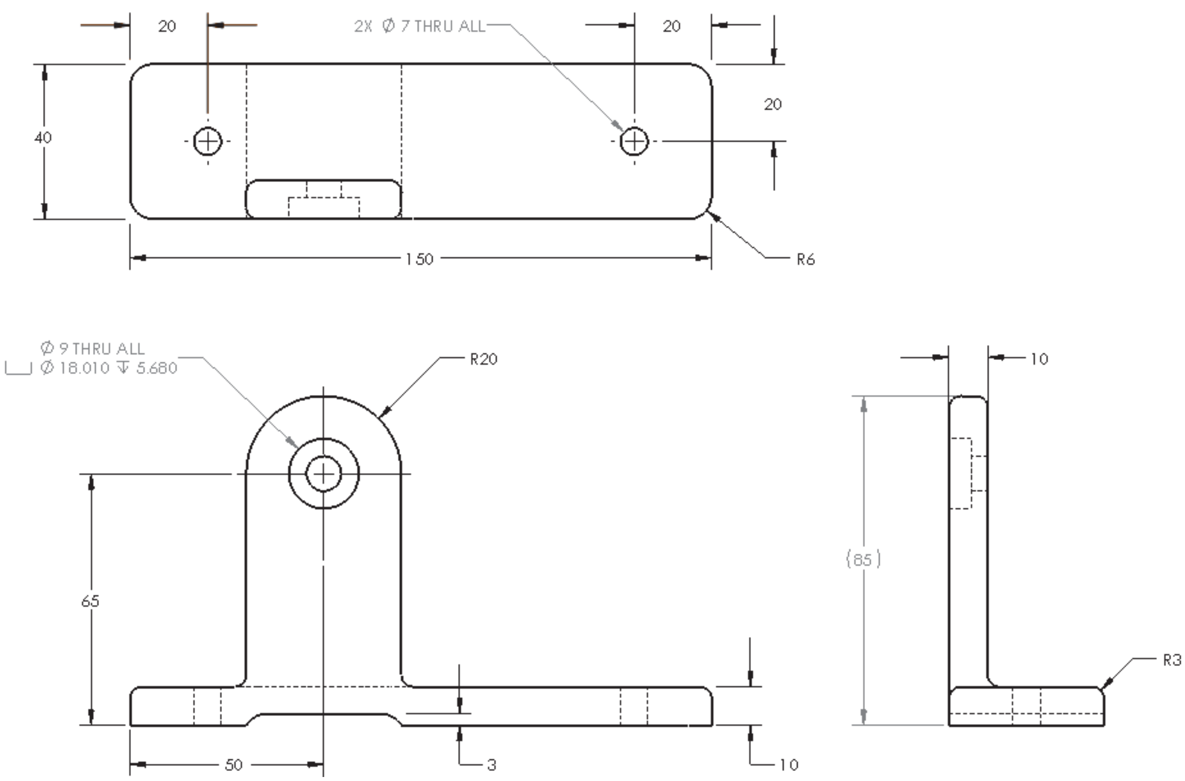
Procedure

Create a new drawing and add the views and dimensions shown in the following steps.

1 **Open part.**
Open the part `Basic-Changes-Done`.

2 **New drawing.**
Use the **Make Drawing from Part** command and the `B_Size_ANSI_MM` template to create the drawing views as shown.

3 **Dimensions.**
Add the annotations and dimensions as shown.



4 **Save and close all files.**