Introducing SolidWorks

Document Number: SWMISENG0902

Contents

Chapter 3 Assemblies

Chapter 4 Drawings

Chapter 5 Engineering Tasks

Introduction

The SolidWorks Software

The SolidWorks® software is a mechanical design automation application that takes advantage of the Microsoft® Windows® graphical user interface. This software makes it possible for designers to quickly sketch out ideas, experiment with features and dimensions, and produce models and detailed drawings.

Introducing SolidWorks discusses some basic concepts and terminology used throughout the SolidWorks application. It familiarizes you with the commonly used functions of SolidWorks.

This chapter discusses the following topics:

- Intended audience
- System requirements
- Book structure
- Conventions used in this book

Intended Audience

The *Introducing SolidWorks* book is for new SolidWorks users but it assumes that you have basic Windows skills.

In this book, you are introduced to concepts and design processes in a high-level approach. You are not given step-by-step procedures on how to create models.

System Requirements

For the most recent information about system requirements, refer to SolidWorks *Read This First*, which is included in the box that contains the SolidWorks software CDs.

Book Structure

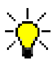
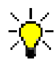
This book is organized to reflect the way that you use the SolidWorks software. It is structured around the basic SolidWorks document types: parts, assemblies, and drawings. For example, you create a part before you create an assembly. Therefore, the Parts chapter precedes the Assemblies chapter.

Throughout the book, a bathroom vanity (including a cabinet, a countertop, a faucet, and pipes) illustrates various tools and functions available to you in the software. The chapters are organized as follows:

Chapter	Title	Topics Discussed
1	Fundamentals	Introduces design concepts, SolidWorks terminology, and an overview of help options
2	Parts	Demonstrates design methods, tools, and features commonly used to make parts
3	Assemblies	Shows how to add parts to an assembly, specify mates, and use in-context design methods
4	Drawings	Discusses drawing sheet formats, views, dimensions, annotations, and bills of materials
5	Engineering Tasks	Examines add-in applications, utilities, and other resources to complete advanced tasks

Conventions Used in this Book

This book uses the following conventions:

Convention	Meaning	Example
Bold	Additional SolidWorks functionality that is not a menu item	**Measure**. Measure the distance between two entities.
Bold Sans Serif	Any SolidWorks tool, menu item, note, help topic, or online tutorial name	Create a sketch with the **Line** tool.
Italic	References to books and other documents, or to emphasize text	Refer to the SolidWorks *Read This First*.
	Reference to online tutorial **NOTE**: You access the *Online Tutorial* from the **Help** menu in the SolidWorks software.	 For a lesson on lofts, see **Loft Features** in the *Online Tutorial*.
	Reference to online help **NOTE**: You access the *SolidWorks Online User's Guide* from the **Help** menu in the SolidWorks software.	 For more information on chamfers, see **Chamfer Feature** in the *SolidWorks Online User's Guide*.
	Tip	 When you create a 3D model, first make the 2D sketch, then create the extruded 3D feature.

1

SolidWorks Fundamentals

SolidWorks Fundamentals introduces you to the following areas:

- **Concepts**. Review the principal concepts found in the SolidWorks software.
- **Terminology**. List the common SolidWorks terms used in the design process.
- **User interface**. Describe the graphical user interface.
- **Design intent**. Examine model design in the context of the SolidWorks software.
- **Design method**. Create a basic 3D model.
- **Model editing**. Review multiple editing options.
- **SolidWorks resources**. Describe the SolidWorks resources.

Concepts

The SolidWorks software enables you to design models quickly and precisely. SolidWorks designs are:

- Defined by 3D design
- Based on components

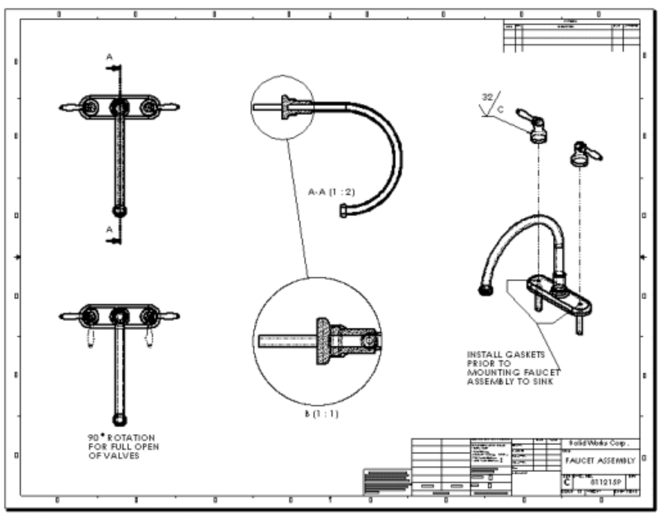
3D Design

SolidWorks uses a 3D design approach. As you design a part, from the initial sketch to the final model, you create a 3D entity. From this 3D entity, you can create 2D drawings, or you can mate different components to create 3D assemblies. You can also create 2D drawings of 3D assemblies.

When designing a part using SolidWorks, you can visualize it in three dimensions, the way the part exists once it is manufactured.

SolidWorks: 3D part

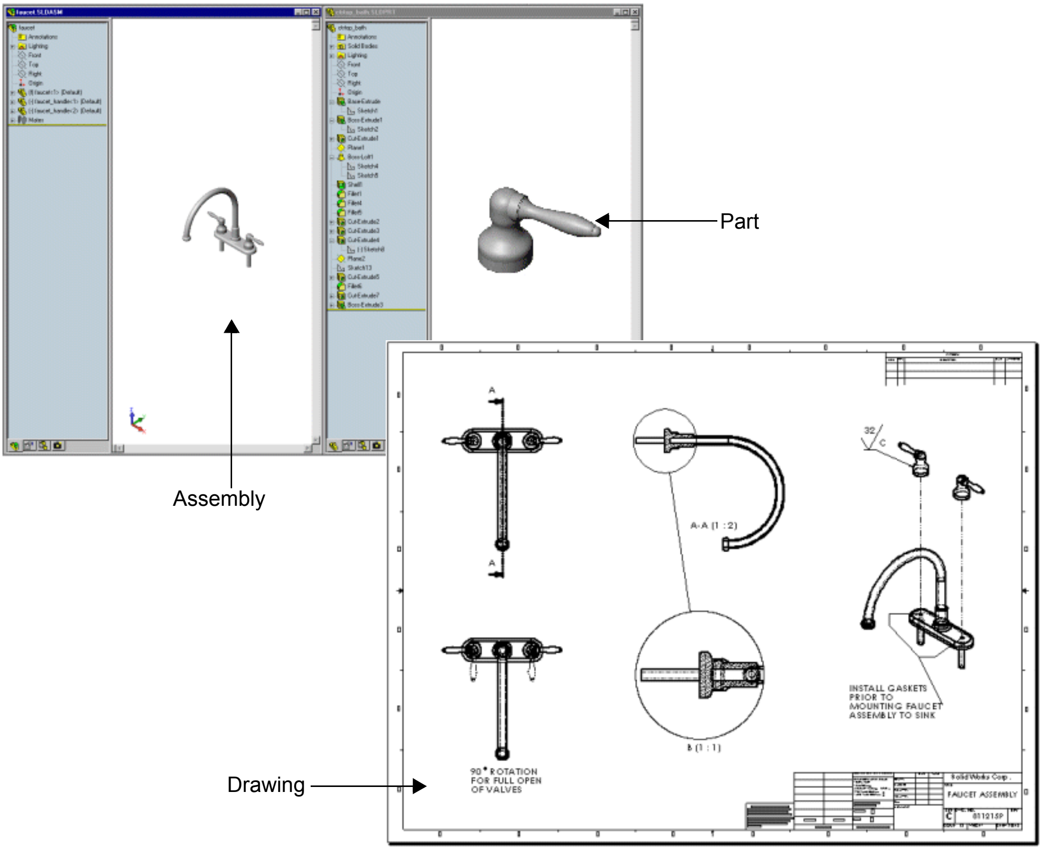
SolidWorks: 3D assembly



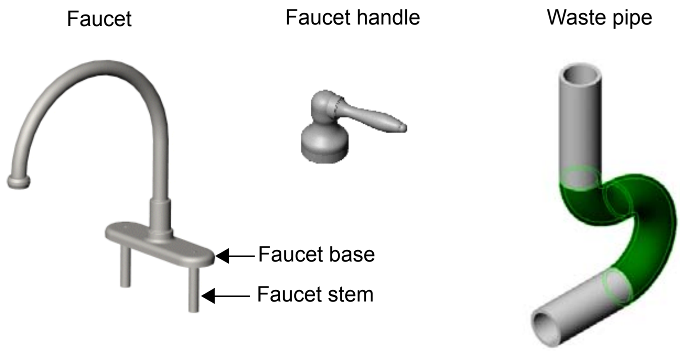
2D drawing generated from 3D model

Component Based

One of the most powerful features in the SolidWorks application is that any change you make to a part is reflected in any associated drawings or assemblies.



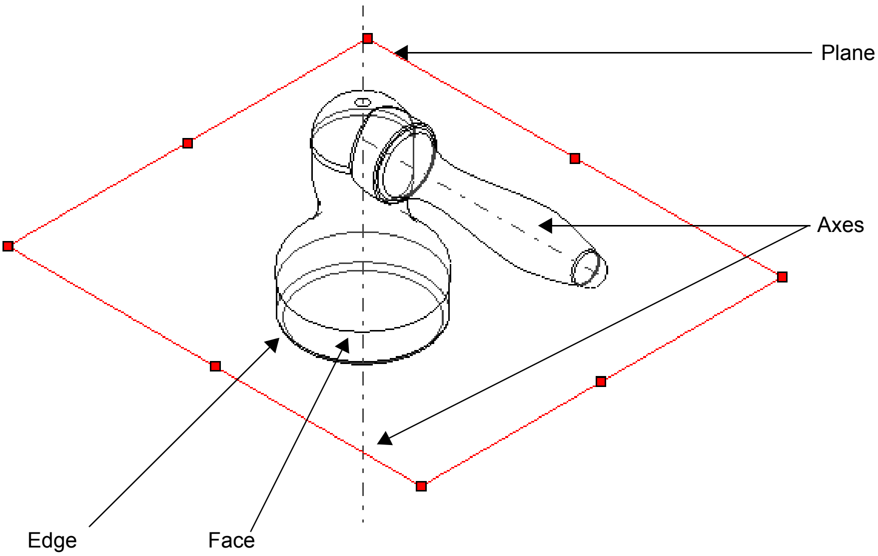
This chapter uses the following terminology:



Terminology

The following terms appear throughout the SolidWorks software and the documentation:

- **Origin**. Appears as two gray arrows and represents the (0,0,0) coordinate of the model. When a sketch is active, a sketch origin appears in red and represents the (0,0,0) coordinate of the sketch. You can add dimensions and relations to a *model* origin, but not to a sketch origin.
- **Plane**. Flat construction geometry. You can use planes for adding a 2D sketch, section view of a model, a neutral plane in a draft feature, and so on.
- **Axis**. Straight line used to create model geometry, features, or patterns. You can create an axis in a number of different ways, including intersecting two planes.
- **Face**. Boundaries that help define the shape of a model or a surface. A face is a selectable area (planar or non-planar) of a model or surface. For example, a rectangular solid has six faces.
- **Edge**. Location where two faces or surfaces meet along a distance. You can select edges for sketching, dimensioning, and many other operations.
- **Vertex**. Point at which two or more lines or edges intersect. You can select vertices for sketching, dimensioning, and many other operations.



User Interface

The SolidWorks application includes a variety of user interface tools and capabilities to help you create and edit models efficiently. These tools and capabilities include the following:

- Windows functions
- SolidWorks document windows
- Function selection and feedback

Windows Functions

The SolidWorks application includes familiar Windows functions, such as dragging and resizing windows, and so on. Many of the same icons, such as print, open and save, cut and paste, and so on, are also part of the SolidWorks application.

Some common Windows-related functions include:

- **Open a document**. Drag a part from Windows Explorer into a blank SolidWorks document to open a part.
- **Open and save to a web folder**. Open or save files to a web folder. A web folder is a SolidWorks tool that allows multiple users to share and work on a SolidWorks part, assembly, or drawing document, as well as other file formats, across the Internet.
- **Create a drawing**. Drag a part into a blank drawing document to create one or more drawing views of the part. A view is an orientation, such as front, top, isometric, and so on.
- **Create an assembly**. Drag components into a blank assembly document to mate the various components and create an assembly. An assembly is a collection of related parts saved in one SolidWorks document.
- **Use keyboard shortcuts**. Use keyboard shortcuts for all menu items. For example: **Ctrl+O** to open a file, **Ctrl+S** to save a file, and **Ctrl+Z** to undo a task.

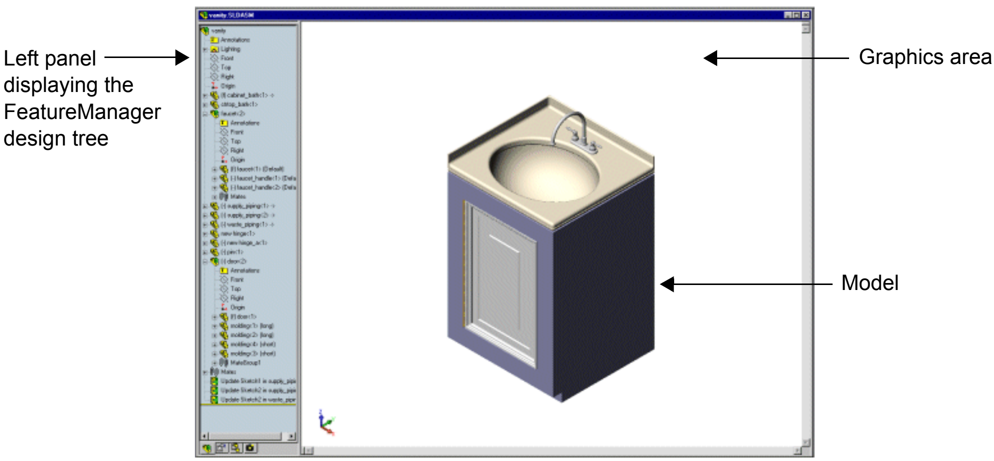
SolidWorks Document Windows

SolidWorks document windows have two panels. The left panel contains the following:

- **FeatureManager® design tree**. Lists the structure of the part, assembly, or drawing. When you select an element from the FeatureManager design tree, you can edit the underlying sketch, edit the feature, suppress and unsuppress the feature or component, and so on.
- **PropertyManager**. Displays relevant information along with user-interface capabilities for many functions such as sketches, fillet features, assembly mates, and so on.
- **ConfigurationManager**. Helps create, select, and view multiple configurations of parts and assemblies in a document.
- **Customized third-party add-in panels**. Includes information for add-ins.

You can split the left panel to display more than one tab at a time. For example, you can display the FeatureManager design tree on the top portion, and the PropertyManager tab for a feature you want to implement, on the bottom portion.

The right panel is the graphics area, where you create and manipulate the part, assembly, or drawing.



Function Selection and Feedback

The SolidWorks application allows you to perform tasks in different ways. It also provides feedback as you perform a task such as sketching an entity or applying a feature. Feedback includes pointers, inference lines, previews, and so on.

Menus

You can access all SolidWorks commands using menus. SolidWorks menus use Windows conventions, including sub-menus, check marks to indicate that an item is active, and so on. You can also use context-sensitive shortcut menus.

Toolbars

You can access SolidWorks functions using toolbars. Toolbars are organized by function; for example, the Sketch or the Assembly toolbar. Each toolbar comprises individual icons that represent specific tools, such as **Rotate View**, **Circular Pattern**, **Circle**, and so on.

You can display or hide toolbars, dock them around the four borders of the SolidWorks window, or float them in the graphics area. The SolidWorks software remembers the state of the toolbars from session to session. You can also add or delete tools to customize the toolbars.

Mouse Buttons

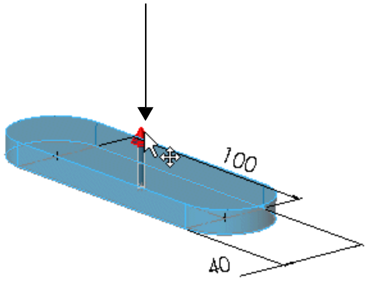
Mouse buttons operate in the following ways:

- **Left**. Selects menu items, entities in the graphics area, and objects in the FeatureManager design tree.
- **Right**. Displays the context-sensitive shortcut menus.
- **Middle**. Rotates, pans, and zooms a part or an assembly, and pans in a drawing.

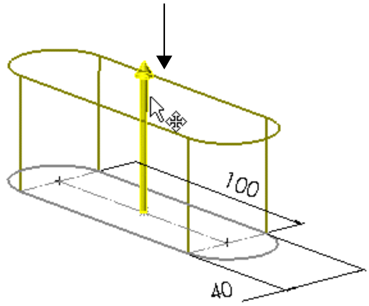
Handles

You can use the PropertyManager to set the values such as the depth of an extrude. You can also use graphic handles that allow you to drag and set certain parameters dynamically without leaving the graphics area.

Grab handle to drag

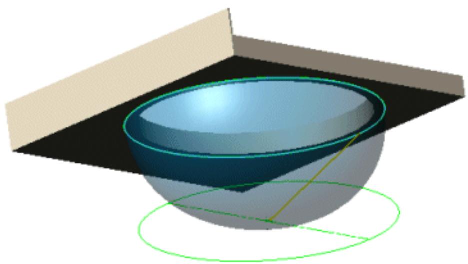


Drag and view the preview



Previews

With most features, the graphics area displays a preview of the feature you want to create. Previews are displayed with features such as base or boss extrudes, cut extrudes, sweeps, lofts, patterns, surfaces, and so on.

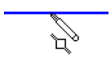


Loft thin preview

Pointer Feedback

As you create a sketch, the pointer changes dynamically to provide data about the type of sketch entity, and the position of the pointer relative to other sketch entities. For example:

 Pointer indicates rectangular sketch.

 Pointer indicates the midpoint of a sketch line or an edge.

Selection Filters

Selection filters help you select a particular type of entity, thereby excluding selection of any other entity type in the graphics area. For example, if you need to select an edge in a complex part or assembly, select **Filter Edges** to exclude any other entity.

Filters are not restricted to entities such as faces, surfaces, or axes. You can also use the selection filter to select specific drawing annotations, such as notes and balloons, weld symbols, geometric tolerances, and so on.

Additionally, you can select multiple entities using selection filters. For example, to apply a fillet, a feature that rounds off edges, you can select a loop composed of multiple adjacent edges.

For more information on using filters, see **Selection Filters** in the *SolidWorks Online User's Guide*.

Select Other

Use the **Select other** function to scroll through the multiple edges or faces in a part, including edges and faces that are hidden.

Design Intent

Design intent determines how you want your model to react as a result of any changes you need to make to the model. Design intent is primarily about planning. Deciding how to create the model determines how changes affect the model. The closer your design implementation is to your design intent, the greater the integrity of the model.

Various factors contribute to the design process, including:

- **Current needs**. Understand the purpose of the model to design it efficiently.
- **Future considerations**. Anticipate potential requirements to minimize redesign efforts when changing the model.

The design process usually involves the following steps:

- Identify needs
- Conceptualize model based on identified needs
- Develop model based on the concepts
- Analyze model development results
- Prototype the model
- Construct the model
- Edit the model if needed

Design Method

After you identify needs and isolate the appropriate concepts, you can develop the model using the following steps:

- **Sketches**. Create the sketches, and decide how to dimension, where to apply relations, and so on.
- **Features**. Select the appropriate features, determine the best features to apply, decide in what order to apply those features, and so on.
- **Assemblies**. If the model is an assembly, select what components to mate, what types of mates to apply, and so on.

A model always includes one or more sketches, and one or more features. Not all models, however, include assemblies.

Sketches

Creating a model begins with a sketch. From the sketch, you can create features. You can combine one or more features to make a part. Then, you can then combine and mate the appropriate parts to create an assembly. From the parts or assemblies, you can then create drawings.

A sketch is a 2D profile or cross section. To create a 2D sketch, you use a plane or a planar face. In addition to 2D sketches, you can also create 3D sketches that include a Z axis, as well as the X and Y axes.

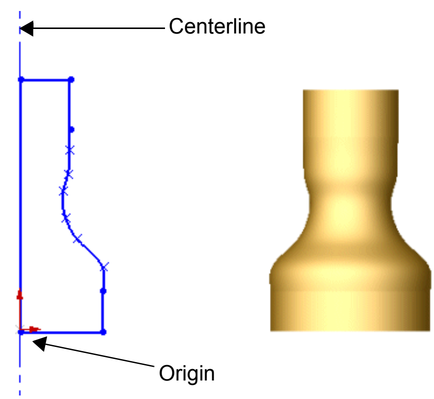
There are various ways of creating a sketch. All sketches include the following elements:

- Origin
- Planes
- Dimensions
- Relations

Origin

In many instances, you start the sketch at the origin. This provides an anchor for the sketch. In other situations, you can use the origin differently.

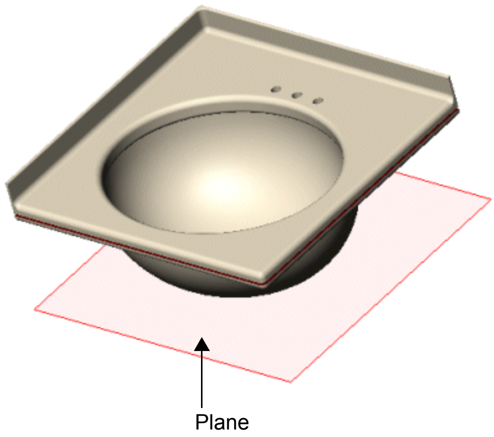
The sketch on the right also includes a centerline. The centerline is sketched through the origin, and is used to create the revolve.



Although a centerline is not always needed in a sketch, a centerline helps to establish symmetry. You can also use a centerline to apply a mirror relation, and to establish equal and symmetrical relations between sketch entities.

Planes

You can create planes in part or assembly documents. You can sketch on planes with sketch tools such as the **Line** or **Rectangle** tool, create a section view of a model, and so on. On some models, the plane you select on which to sketch affects only the way you want the model to appear using a standard isometric view (3D). It does not affect the design intent. With other models, selecting the correct initial plane on which to sketch can help you create a more efficient model.



The **Front** plane is the default plane for the first sketch in a new part. The two other standard planes use top and right orientations. You can also add and position planes as needed.

For more information on planes, see **Creating Planes** in the *SolidWorks Online User's Guide*.

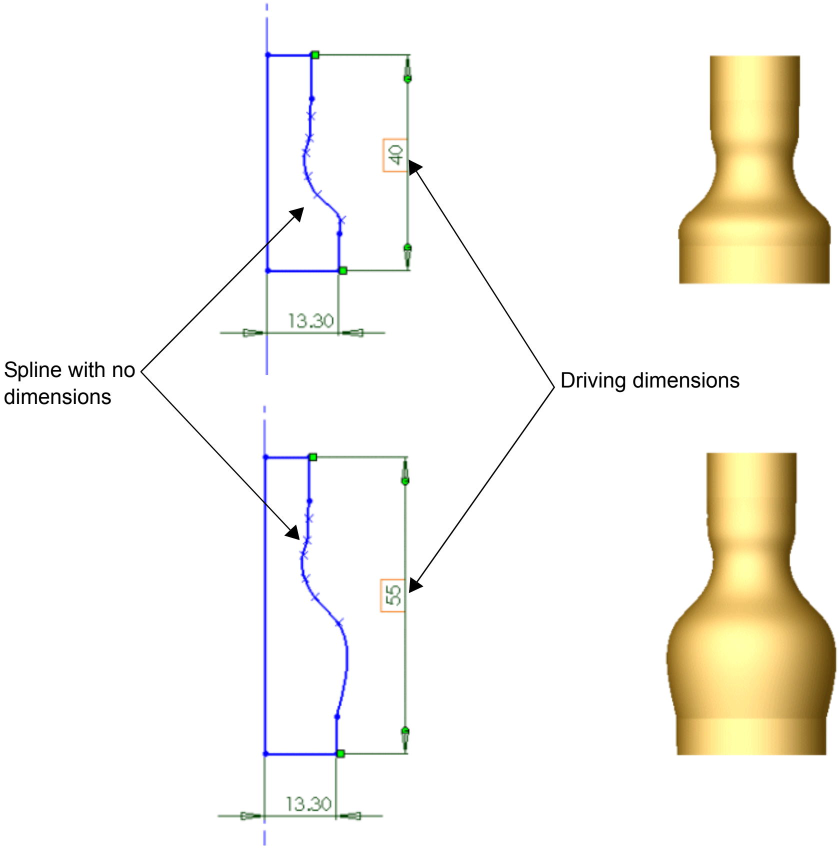
Dimensions

You can specify dimensions and geometric relations between entities. Dimensions define length, radius, and so on. When you change dimensions, the size and shape of the part changes. Depending on how you dimension the part, you can preserve the design intent. See **Design Intent** on page 1-10.

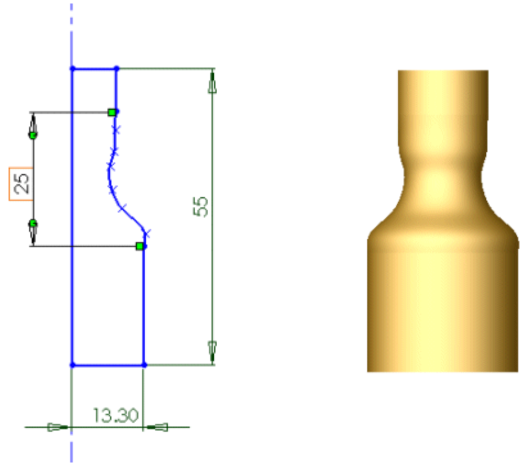
One way to retain design intent is to keep one dimension constant while you change other dimensions. In this context, there are driving dimensions, and driven dimensions (see the next section).

Driving Dimensions

You create driving dimensions with the **Dimension** tool. Driving dimensions change the size of the model when you change their value. For example, in the faucet handle, you can change the height of the faucet handle from 40mm to 55mm, but the radius of 13.30mm at the bottom remains constant. Note how the shape of the revolved part changes because the spline is *not* dimensioned.



To maintain a uniform shape generated by the spline, you need to dimension the spline. You can still change the height of the faucet handle base.

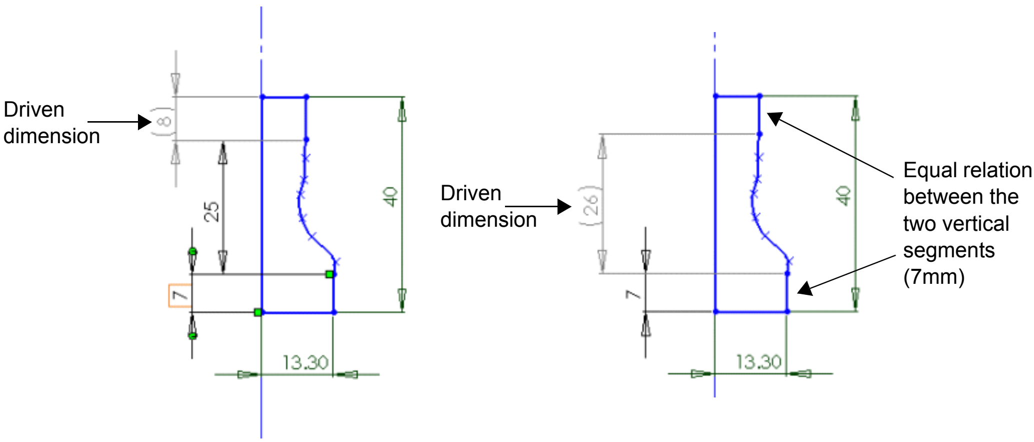


Driven Dimensions

Some dimensions associated with the model are driven. Driven dimensions are created by the SolidWorks software, and are used for information only. You can delete driven dimensions, but you cannot modify them. Driven dimensions change when you modify driving dimensions.

In the faucet handle, if you dimension the total height (40mm), the vertical section below the spline (7mm), and the spline segment (25mm), the vertical segment above the spline (8mm) is a driven dimension.

Deciding which dimensions are the driving dimensions and which are the driven dimensions affects your design. For example, if you dimension the total height (40mm), and create an *equal* relation (see next section) between the top and bottom vertical segments (7mm), the spline becomes a driven dimension (26mm).



Sketch Definitions

Sketches can be fully defined, under defined, or over defined. In fully defined sketches, all the lines and the curves in the sketch, and their positions, are described by dimensions, or relations (see the following section), or both. It is not necessary to fully define sketches before you use them to create features. However, you should fully define sketches to complete a part.

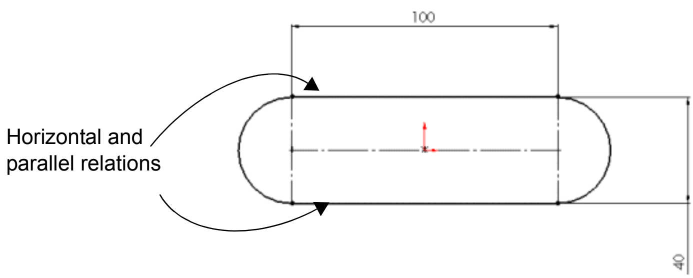
By displaying the entities of the sketch that are under defined, you can determine what you need to add (dimension or relations), to fully define the sketch. You can use the color cues to determine if a sketch is under defined. In addition to color cues, entities in under defined sketches are not fixed within the sketch, so you can drag them.

Over defined sketches include redundant dimensions or relations. You can delete over defined dimensions or relations, but you cannot edit them.

Relations

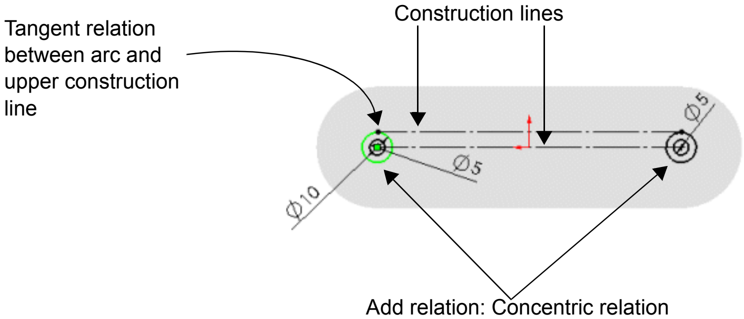
Relations establish geometric relations (equal, tangent, and so on) between sketch entities. For example, you can establish equality between the two horizontal 100mm entities below. You could dimension each horizontal entity individually, but by establishing an equal relation between the two horizontal entities, you need to update only one dimension if the length changes. Relations are saved with the sketch. You can apply relations in the following ways:

- **Inference**. Some relations are created by inference. For example, as you sketch the two horizontal entities to create the base extrude for the faucet base, horizontal and parallel relations are created by inference.



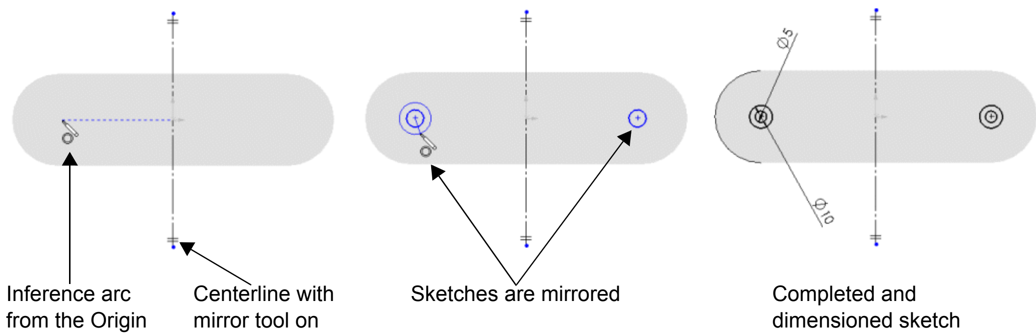
- **Add relations**. You can also use the **Add Relations** tool where necessary. For example, to create the faucet stems, you sketch a pair of arcs for each stem.

To position the stems, You add a tangent relation between the outer arcs and the top construction line horizontal (displayed as a broken line). For each stem, you also add a concentric relation between the inner and outer arcs.



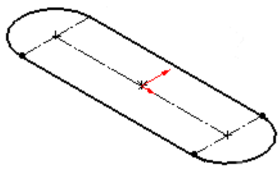
Sketch Complexity

A simple sketch is easy to create and update, and it rebuilds quicker. One way to simplify sketching is to apply relations as you sketch (see the preceding section). You can also take advantage of repetition and symmetry. For example, the faucet stems on the faucet base include repeated sketch entities. Repetition allows you to mirror the left and the right sides by using a Centerline, which creates some automatic relations and dimensions. You can inference the position of the arcs from the Origin. You can dimension and add a concentric relation between one of the inner arcs and the outer arc of the base once.

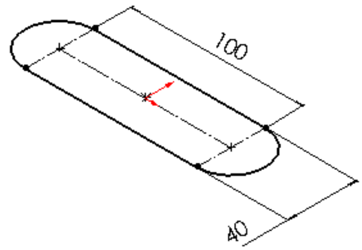


Features

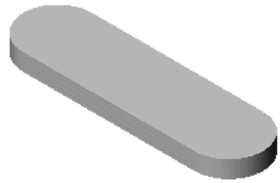
Once you complete the sketch, you can create a 3D model using features such as an extrude (the base of the faucet) or a revolve (the faucet handle).



Create the sketch

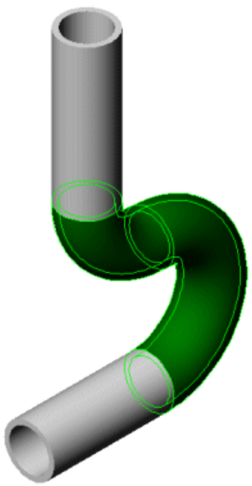


Dimension the sketch

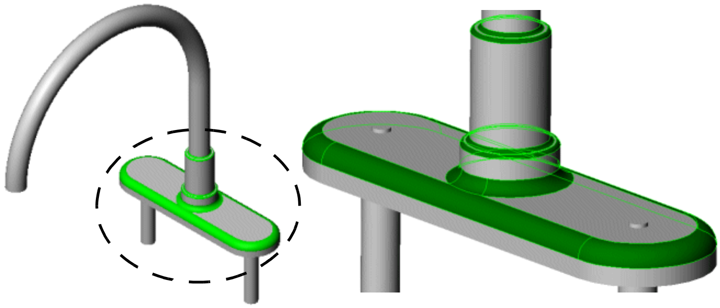


Extrude the sketch 10mm

Some sketch-based features are shapes (bosses, cuts, holes, and so on). Other sketch-based features such as lofts and sweeps, use a profile along a path. Another type of feature is the applied feature such as fillets, chamfers, or shells. All parts include sketch-based features, and most parts also include applied features.



Sketch-based features:
Base sweep for waste pipe



Applied feature: Fillets

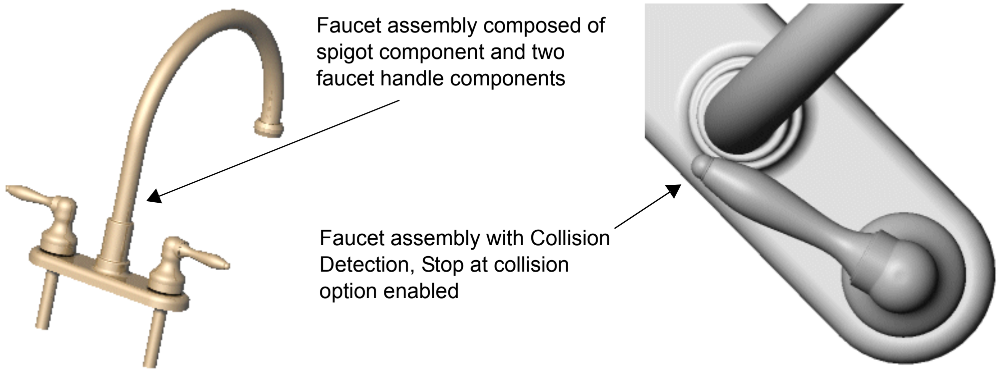
Several factors influence how you apply features. These factors include selecting between different features such as a sweep or a loft to achieve the same results, applying features to a model in a specific order, and so on. For more on features, see Chapter 2, “Parts.”

Assemblies

You can create multiple parts that fit together to create assemblies.

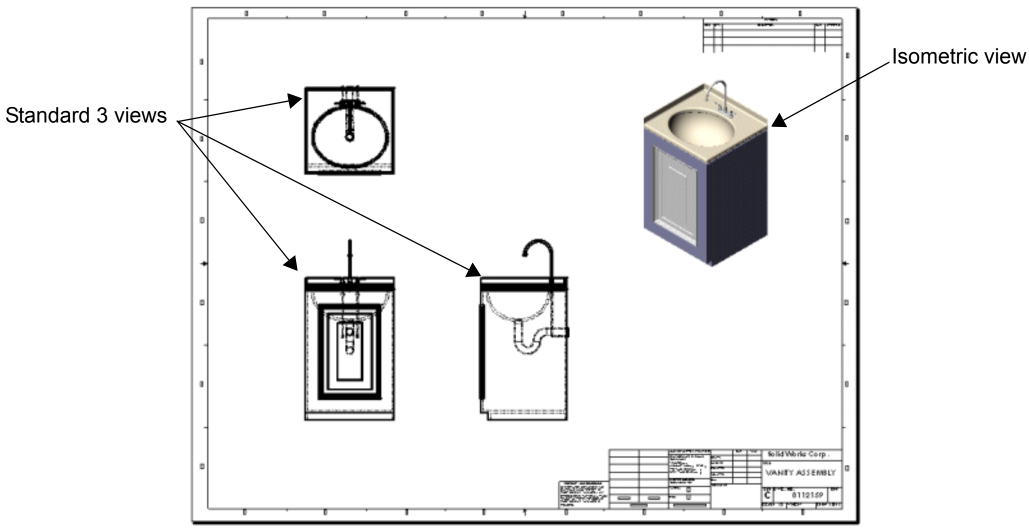
You integrate the parts in an assembly using **Mates**, such as **Coincident** or **Colinear**. With tools such as **Move Component** or **Rotate Component**, you can see how the parts in an assembly function in a 3D context.

To ensure that the assembly functions correctly, you can use assembly tools such as **Collision Detection** or **Interference Detection**.



Drawings

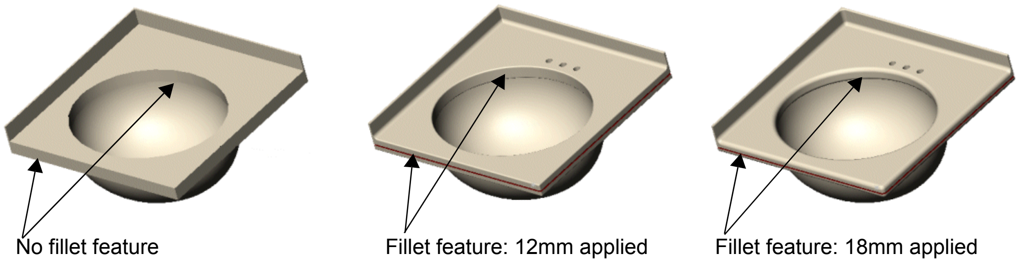
You create drawings from part or assembly models. Drawings are available in multiple views.Views include a set of standard 3 views, isometric view (3D), and so on. You can import the dimensions from the model document, add annotations such as datum target symbols, and so on.



Model Editing

Use the SolidWorks FeatureManager design tree and the PropertyManager, to edit sketches, drawings, parts, or assemblies. Editing capabilities include the following:

- **Edit sketch**. You can select a sketch in the FeatureManager design tree, then edit the sketch.You can edit the sketch entities, change the dimensions, view or delete existing relations, add new relations between the sketch entities, change the size of dimension displays, and so on.
- **Edit feature**. Once you create a feature, you can change most of the values associated with that feature. To edit a feature, use **Edit Definition** to display the appropriate PropertyManager. For example, if you apply a **Constant radius** fillet to an edge, you display the Fillet PropertyManager where you can change the radius.



- **Hide and show**. With certain geometry, for example multiple surface bodies in a single model, you can hide or show one or more surface bodies. You can also hide and show sketches in all documents, and views, lines, and components in drawings.
- **Suppress and unsuppress**. You can select any feature from the FeatureManager design tree, and suppress the feature to view the model without that feature. You can then unsuppress the feature to display the model in its original state. You can suppress and unsuppress components in assemblies as well (see **Assembly Design Methods** on page 3-2).
- **Rollback**. When you want to display a model with multiple features, you can roll the FeatureManager design tree back to a prior state. This displays all features in the model up to the rollback state, until you reverse the FeatureManager design tree back to its original state.

SolidWorks Resources

In addition to this book, you have access to many other resources to help you design models. These include:

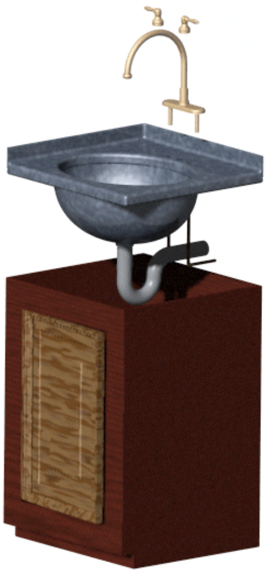
- **Online Tutorial**. The *Online Tutorial* includes practical, step-by-step examples for you to build. There are over twenty different models. The exercises offer a range of subject matter and difficulty. You can access the *Online Tutorial* through the SolidWorks **Help** menu.
- **Design Portfolio**. The *Design Portfolio* showcases various models from a conceptual viewpoint. It analyzes several models, and shows examples of why the model builder followed a certain design method.
- **Online help**. The *SolidWorks Online User's Guide* addresses all functionality in the SolidWorks software. It includes context-sensitive topics that you access from the PropertyManager, as well as additional topics, graphic examples, and animations. You can use search, the Table of Contents, or the Index. There is also a Glossary for SolidWorks terminology.
- **Additional online help**. All of the add-ins include online help. There is also online help for the Application Program Interface (API) and Help for AutoCAD Users.
- **Additional resources**. Other resources are available through the SolidWorks web site. They include design samples, frequently asked questions, a list of related books, and so on.

2

Parts

Parts are the building blocks of every SolidWorks model. Each assembly and drawing you create is made from parts. In this chapter, you learn about the following:

- **Overview.** Learn about some of the common tools used to make parts in SolidWorks.
- **Countertop.** Explore the extrude, sweep, shell, and fillet tools.
- **Faucet.** See additional uses of the sweep and fillet tools.
- **Faucet handle.** Use more detailed sketches and the revolve tool.
- **Cabinet door.** Create beveled edges with the chamfer tool.
- **Moldings.** Design your model with mirroring and configurations.
- **Hinge.** Make a sheet metal part with the base-flange, tab, linear pattern, and hem tools.



Overview

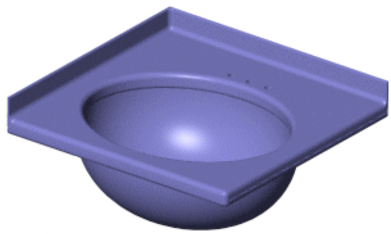
In this chapter, you learn about some of the common tools used to make parts in the SolidWorks software. Many parts in this chapter use the same tools (such as an extrude), so these tools are discussed in detail only the first time they appear.

The beginning of each section shows the design approach used for each part. This is a high-level overview of the tools that create the part. The overview gives you an outline of the features, so you can skim those that you already understand.

> The cabinet, waste pipe, and supply pipes used in the vanity are not discussed in this chapter because they repeat the tools already presented. You will see these parts in later chapters.

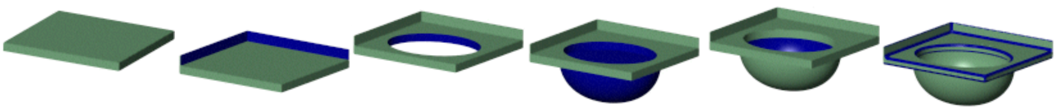
Countertop

The countertop is a single part that includes a sink and counter. First you create the counter, then you make the sink. This countertop uses several tools that are commonly used in the SolidWorks software, including extrusions, a sweep, a shell, and fillets.



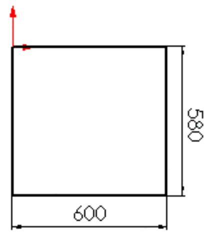
Design Approach

Extrude → **Extrude** → **Cut-Extrude** → **Sweep** → **Shell** → **Fillet**

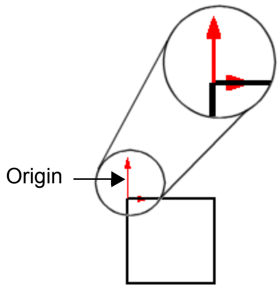


Create the Base Feature with an Extrude

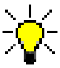
Before you create an extrude feature, you need to make a sketch. This rectangular sketch is dimensioned at 600mm x 580mm.



The sketch begins at the Origin. The Origin is a helpful reference point for sketches. It is the (0,0) coordinate of a 2D sketch. If you begin a sketch at the origin, the sketch position is set. When you add dimensions and relations to the sketch, it becomes fully defined.



After you sketch the rectangle, use the **Extrude** tool to create a 3D base feature. The sketch is extruded normal to the sketch plane. This model is displayed in an isometric view so it is easy to see the model structure.



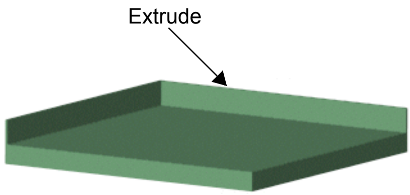
When you design a 3D model, first make the 2D sketch, then create the 3D feature.

Add an Extrude to the Base

The second extrude adds material to a part by building upon the base. In this example, you extrude the countertop edges.

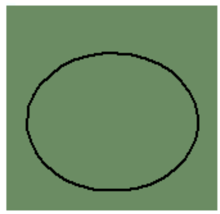
First, you create the sketch for the extrude. This is an L-shaped sketch made with the **Line** tool.

Next, you use the **Extrude** tool to create the countertop edges.

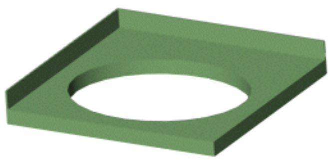


Remove Material with the Cut-Extrude

The **Cut-Extrude** tool is similar to an extrude feature, except that it removes material from the model instead of adding material. First you create a 2D sketch, then you make the cut-extrude. In this example, you use the **Ellipse** tool to make an oblong sketch.



When the cut-extrude is complete, the countertop has an opening for the sink.

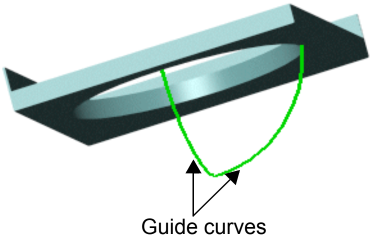


For a lesson that includes extrude features, see **Welcome to Lesson 1 of the Running Start** in the *Online Tutorial*.

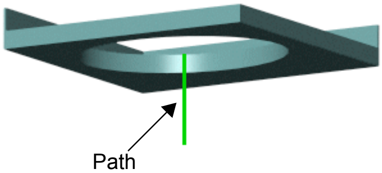
Use a Sweep to Make a Solid

After you make the cut-extrude feature, you create the sink with the **Sweep** tool. A sweep creates a feature by moving a profile along a path.

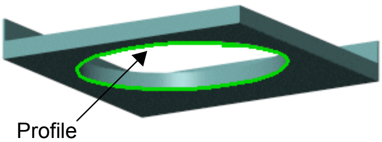
First, sketch two guide curves with the **3 Point Arc** tool. These 2D guide curves control the intermediate profiles as the sketch is swept along the path.



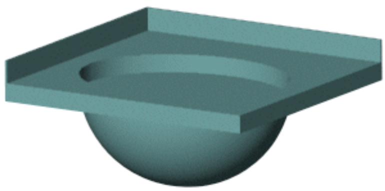
Second, sketch the path that you want the profile to follow. The sink is below the countertop, so you sketch a downward vertical line starting from the center of the ellipse.



Next, create the profile. You sketch the sink profile with the **Ellipse** tool.



Finally, use the **Sweep** tool to run the profile along the path, bounded by the guide curves.



For a lesson on sweeps, see **Revolve and Sweep Features** in the *Online Tutorial*.

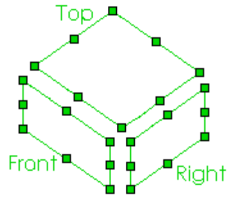
Use a Loft to Make a Solid - Alternate Design Approach

There is no right or wrong way to design a part, assembly, or drawing. However, some design methods are more efficient than others.

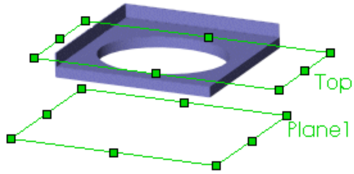
As an alternative to a sweep, you could use the **Loft** tool to create the sink. A loft is a solid feature that connects two or more sketch profiles. In this example, the loft creates the sink by connecting an elliptical sketch and a sketched point.

When you create a loft, the sketch profiles must be on different planes. The 2D sketches must reside on a plane or a planar face.

By default, each SolidWorks part has a front, top, and right plane. You can sketch on any of the planes.

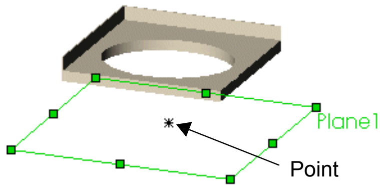


In this example, create a new plane, **Plane1**, by offsetting it from the **Top** plane. **Plane1** is parallel to the **Top** plane.

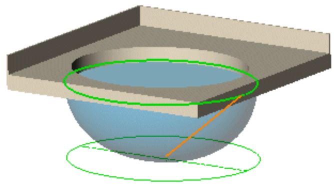


First, sketch an ellipse on the bottom of the countertop with the **Convert Entities** sketch tool. This tool creates a sketch by projecting the cut-extrude sketch onto the **Front** plane.

Next, use the **Point** tool to sketch a point on **Plane1**.



Now that you have two sketch profiles, use the **Loft** tool to connect them. The SolidWorks software uses shaded previews to illustrate what the model will look like before you accept the feature.

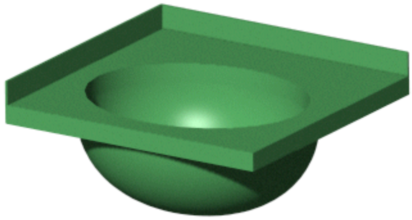


For a lesson on lofts, see **Loft Features** in the *Online Tutorial*.

It is better to use a sweep instead of a loft because a sweep rebuilds faster, which improves system performance. Sweeps are also easier to create when you want a consistent cross-sectional shape in the model.

Shell the Part

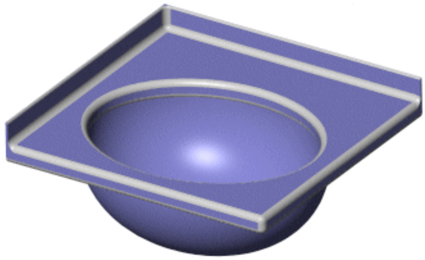
Because the loft creates a solid feature, you need to cut out material to make the sink. The **Shell** tool hollows out the sink and removes the top face. When you shell a part in SolidWorks, any selected faces are removed, and thin faces remain on the rest of the part.



For a lesson that includes shells, see **Welcome to Lesson 1 of the Running Start** in the *Online Tutorial*.

Round Sharp Edges with Fillets

To complete the countertop, you add fillet features to your model to round off sharp edges. A fillet is an internal rounding of an edge on a surface or solid. When you create a fillet, you set the radius to determine the smoothness of the edges.



It is best to save cosmetic fillets for the last step, after all of the geometry is in place. Models rebuild faster when fillets are made at the end of the design process.

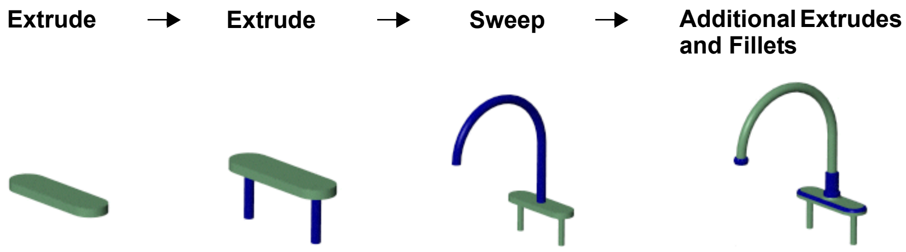
Fillets are applied features, not sketch features. This means that fillets do not require you to create a sketch. Instead, you select the edges of an existing feature to fillet, set the fillet radius, and create the fillet. As you increase the radius, the edges or faces become more round.

For a lesson on fillets, see **Fillet Features** in the *Online Tutorial*.

Faucet

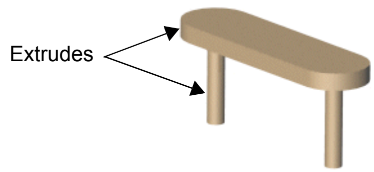
Most parts have extrude and fillet features. The faucet also uses these tools, in addition to a sweep. In the following example, a sweep creates the faucet spigot.

Design Approach



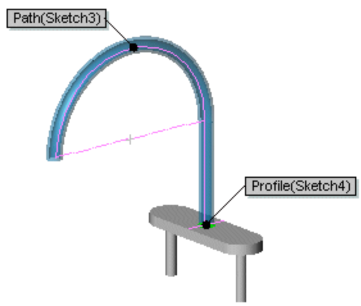
Create the Sweep

After you create the faucet base with two extrudes, the model appears as shown.



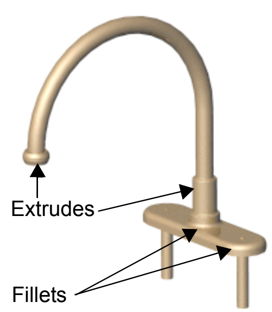
Use the **Sweep** tool to make the spigot. In this example, the profile is a circular sketch, and the path is a sketched arc that is tangent to a vertical line. There is no need to use guide curves, because the circular profile remains the same shape and diameter for the entire sweep.

> When you sketch the profile and path, it is important that the starting point of the path lie on the same plane as the profile.



Sweep preview

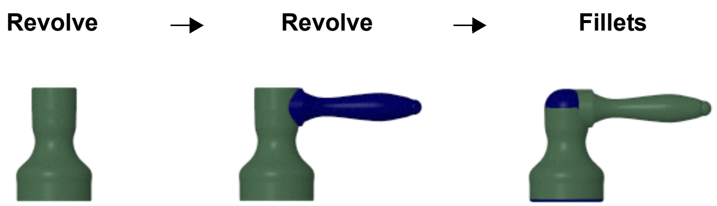
After you create some additional extrudes and fillets as shown, the faucet is complete.



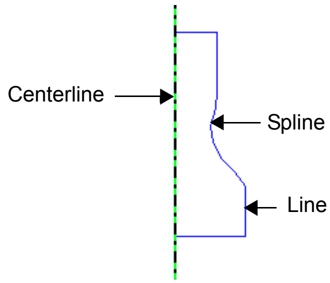
Faucet Handle

The faucet handle is built with two revolve features. The model uses a simple design approach, although the revolves require detailed sketches. The **Revolved Boss/Base** tool revolves a sketch profile around a centerline at a specified angle. In the following examples, the revolve angles are set to 360°.

Design Approach



Revolve the Sketch



A revolve creates the base of the handle, and completes the first feature in the faucet handle.

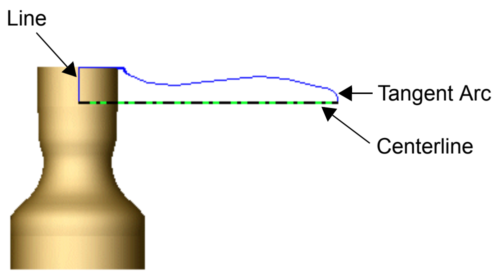
First, you create a sketch with the **Line** and **Spline** tools. In the same sketch, you make an axis of revolution with the **Centerline** tool. A centerline creates an axis that is construction geometry; it is not built into the feature.

You then use the **Revolved Boss/Base** tool to rotate the sketch and create a solid feature.

Create the Second Revolve

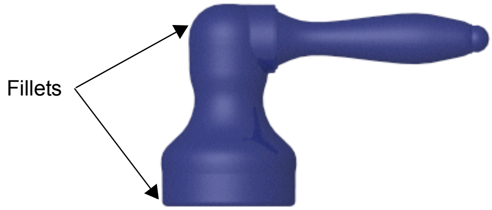
You create a second revolve feature to add material to the faucet handle.

Again, you begin with a sketch, as shown, then create a 3D solid with the revolve. This sketch uses the **Line**, **Tangent Arc**, and **Spline** tools.



When you use the **Revolved Boss/Base** tool, the SolidWorks software revolves the sketch to generate a solid.

After you add cosmetic fillets, the faucet handle is complete.



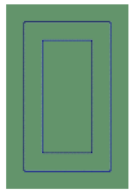
For a lesson on revolves, see **Revolve and Sweep Features** in the *Online Tutorial*.

Cabinet Door

The cabinet door uses an extrude and a cut-extrude to make the exterior detail.

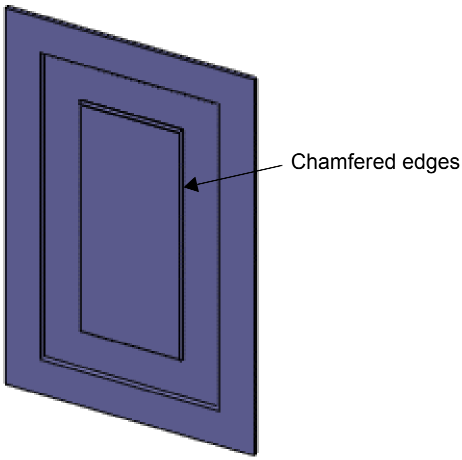
Design Approach

Extrude → **Cut-Extrude** → **Chamfer**



Create Beveled Edges with the Chamfer Tool

The **Chamfer** tool creates beveled faces, as shown. A chamfer, like a fillet, is an applied feature, and does not require you to make a sketch to create the feature. In this example, the face with the extruded cut has chamfered edges.

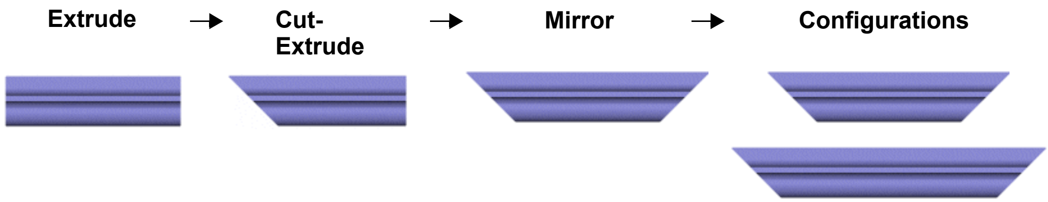


For more information on chamfers, see **Chamfer Feature** in the *SolidWorks Online User's Guide*.

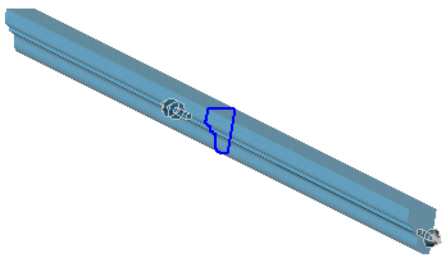
Moldings

The moldings around the edges of the door use an extruded sketch, an extruded cut, and a mirror feature. Only one part file was created, although there are four pieces of molding on the door. With configurations, you can create the different molding lengths within one part.

Design Approach



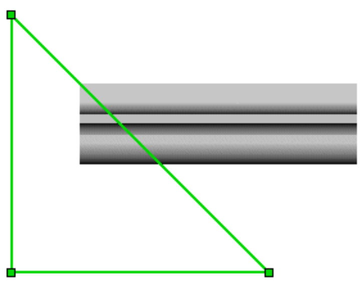
Design a Mid Plane Extrude



The molding sketch uses a mid plane extrusion. This means that instead of extruding the sketch in one direction, you extrude the sketch equally in both directions perpendicular to the sketch plane.

Although you do not have to use a mid plane extrusion, it ensures that you have equal lengths of material on both sides of the sketch.

Sketch a Profile for the Cut-Extrude

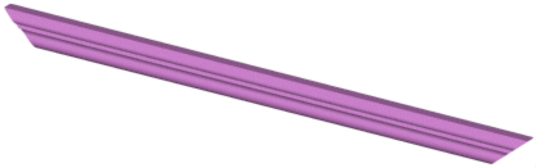


Next, you cut the molding at a 45° angle. The 45° cut ensures that the molding pieces fit together accurately.

When you sketch a profile to cut, it is recommended that you make the sketch larger than the model. This makes a clean cut through the entire molding.

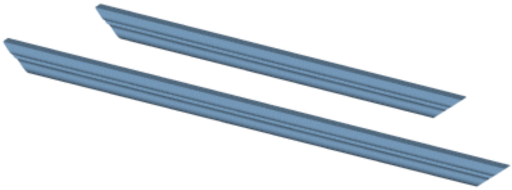
Mirror the Part

Finally, to cut the model at the same angle on the opposite side, use the **Mirror** tool to mirror the original cut about the plane of symmetry.



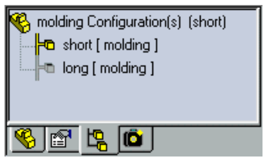
Use Configurations of a Part

Configurations allow you to create multiple variations of a part within a single part file. When you design a part, the SolidWorks software automatically creates the **Default** configuration. In the molding that you created, the default configuration matches the length of the shorter sides of the door. To easily identify the configuration, rename the default configuration to **short**.



In the same document, you create another configuration and name it **long**. In this configuration, increase the length to match the longer sides of the door.

The SolidWorks ConfigurationManager displays the two configurations in the document. When you double-click a configuration name, the graphics area displays that configuration. Later in this book, you insert different configurations of the same part into an assembly.



For a lesson that includes mirroring and configurations, see **Advanced Design Techniques** in the *Online Tutorial*.

Hinge

The hinge is a sheet metal part. By definition, sheet metal parts are constructed of uniform thickness, and have a specified bend radius and bend angle.

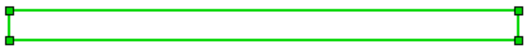
When you design sheet metal in the SolidWorks software, you use a base-flange instead of an extrude to create the base of the part. The base-flange is the first feature in a sheet metal part, and it designates the part as sheet metal. The SolidWorks software has several tools that are specific to sheet metal, including the tab and the hem, which you use in the hinge design.

Design Approach

Base-Flange → **Tab** → **Linear Pattern** → **Hem**

Create Sheet Metal with the Base-Flange

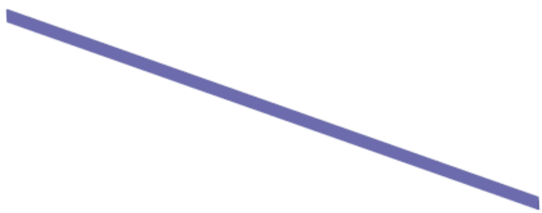
As with other base features, you first create a sketch. In the hinge, you make a sketch with the **Rectangle** tool.



Sketch for base-flange

The base of the hinge is an example where a simple sketch allows for easier creation of the model.

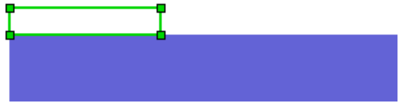
Next, you use the **Base-Flange/Tab** tool to automatically create a sheet metal part.



Make the Tab

The **Tab** tool adds a tab to the sheet metal part. The depth of the tab automatically matches the thickness of the sheet metal part. The direction of the depth automatically coincides with the sheet metal part to prevent a disjoint body.

When you make the sketch for the tab, you sketch on the face where you want the tab to appear. You make this sketch with the **Rectangle** tool on the front face.



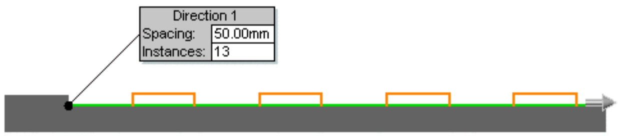
After you complete the sketch, use the **Base-Flange/Tab** tool to add the tab.

For more information on tabs, see **Sheet Metal Tab** in the *SolidWorks Online User's Guide*.

Generate the Linear Pattern

To make tabs that span the length of the hinge, use the **Linear Pattern** tool to copy the original tab for a specified number of times. The linear pattern creates multiple instances of a selected feature along a linear path.

When you make a linear pattern, you specify the number of instances and the distance between each tab. In the hinge, there are 13 tabs separated by 50mm.

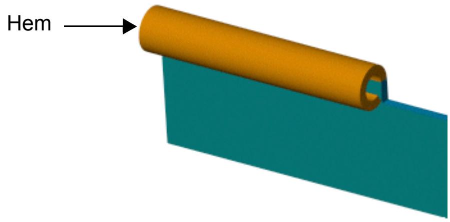


This is the first piece of the hinge. When you create the second piece, you change the location of the tabs so both pieces fit together.

For more information, see **Linear Pattern** in the *SolidWorks Online User's Guide*.

Add the Hem

The **Hem** is a sheet metal-specific tool that uses the same model thickness as the base-flange. In this example, you add a rolled hem to each tab to curl the sheet metal.

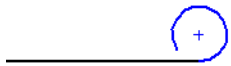


For a lesson on sheet metal, see **Sheet Metal** in the *Online Tutorial*.

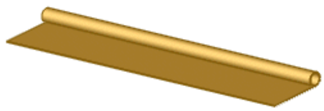
Design Sheet Metal in the Folded State - Alternate Design Approach

You can also create sheet metal parts in the folded state, so that the part is already curled. However, this requires several cuts to make the tabs:

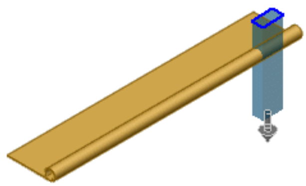
First, create a sketch with the **Line** and **Tangent Arc** tools.



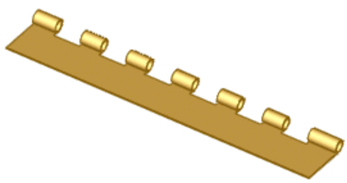
Second, extrude the sketch with the **Base-Flange** tool.



Next, create the first tab with an extruded cut.



Finally, use the **Linear Pattern** tool to create multiple cuts.



Although you can create the hinge in the folded state, the **Hem** tool is the preferred method for curling sheet metal. When you use a hem, it is easier to update its type or radius than it is to edit a sketch such as the one above.

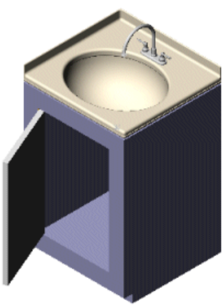
3

Assemblies

In this chapter, you use the vanity cabinet parts described and built in Chapter 2 “Parts” to build sub-assemblies, such as the spigot and the faucet handles. Then you bring the sub-assemblies together to create an assembly - the vanity.

This chapter covers the following topics:

- **Assembly definition.** Describe assembly elements.
- **Assembly design methods**. Define bottom-up and top-down design.
- **Prepare an assembly**. Anchor the first assembly component, and position additional assembly components.
- **Mates**. Complete portions of the full assembly using mates.
- **In-context design**. Reference existing geometry to create additional components.
- **Load an assembly**. Load an existing assembly, and consider performance.
- **Examine the assembly**. Use SolidWorks assembly tools such as exploded view, component visibility, component suppression, and Collision Detection.



Assembly Definition

An assembly is a collection of related parts saved in one SolidWorks document file with an **.sldasm** extension. Assemblies have many characteristics, including the following:

- Contains anywhere from two components to over one thousand components, which can be parts or other assemblies called sub-assemblies
- Displays movement between related parts within their degrees of freedom

The components in an assembly are defined in relation to each other using assembly mates. You mate the assembly components together using various types of mates such as coincident, concentric, and distance mates. For example, the faucet handle components are mated to the faucet base component using concentric and coincident mates. The mated components create the spigot sub-assembly. Later, you include this sub-assembly in the main vanity assembly, mating it to the other components in the vanity assembly.

Assembly Design Methods

You create assemblies using two basic methods: bottom-up design and top-down design. You can also use a combination of the two. With either method, your objective is to mate the components to create the assembly or sub-assembly (see **Mates** on page 3-5).

Bottom-up Design

In bottom-up design, you create parts, insert them into an assembly, and mate them as required by your design. Bottom-up design is the preferred technique when you use previously constructed, off-the-shelf parts.

An advantage of bottom-up design is that because components are designed independently, their relationships and regeneration behavior are simpler than in top-down design. Working bottom-up allows you to focus on the individual parts. It is a good method to use if you do not need to create references that control the size or shape of parts with respect to each other.

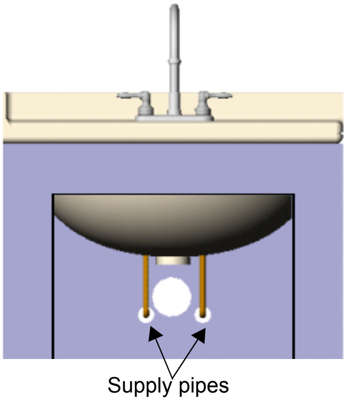
Most of the vanity cabinet uses bottom-up design. You create the components such as the sink and the spigot in their own part windows. Then you open an assembly document, bring the components into the assembly, and add various mates.

Top-down Design

Top-down design is different because you start your work in the assembly. You can use the geometry of one part to help define other parts, or to create machined features that are added only after the parts are assembled. You can start with a layout sketch, define fixed part locations, planes, and so on, then design the parts referencing these definitions.

For example, you can insert a part in an assembly, then build a fixture based on this part. Working top-down, creating the fixture in context, allows you to reference model geometry, so you can control the dimensions of the fixture by creating geometric relations to the original part. That way, if you change a dimension of the part, the fixture is updated automatically.

The vanity cabinet also uses top-down design. You create the two supply pipes within the context of the assembly. Then you reference the size and location of the faucet sub-assembly and the vanity cabinet to define the supply pipes.

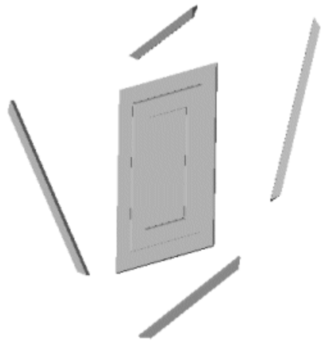


Prepare an Assembly

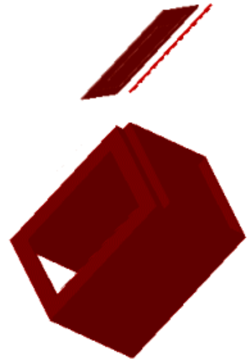
Throughout this chapter, you use the parts for the vanity cabinet created in Chapter 2 "Parts". The vanity includes the following sub-assemblies:

- Faucet and faucet handles
- Door and moldings
- Door sub-assembly, cabinet, and hinge

Spigot and faucet handles



Door and moldings



Door sub-assembly, cabinet, and hinge

For each sub-assembly document, you do the following prior to mating the components:

- Load and anchor the first component to the assembly origin
- Load the additional components
- Move and position the components

Mates

Mates position the components in an assembly precisely with respect to each other. Positioning the components defines how the components move and rotate with respect to each other. Mates create geometric relations, including coincident, perpendicular, tangent, and so on. Each mate is valid for specific combinations of geometry such as cones, cylinders, planes, extrusions, and so on. For example, if you mate a cone to another cone, the valid types of mates you can use include coincident, concentric, and distance (see **Coincident Mate** on page 3-7).

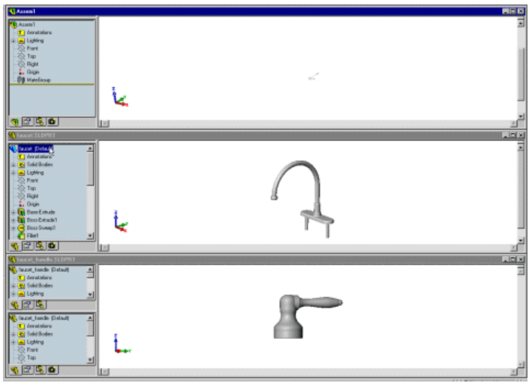
Faucet Sub-Assembly

Depending on the complexity of the assembly (the number of separate components), you can open one or all of your components. In the faucet example, there are only two components (the spigot and the handle), so you can tile the two documents. After you open the components, you need to open a new assembly document into which you bring the components.

New assembly document

Faucet component

Handle component



You can add more than one instance of the same **.sldprt** document to an assembly. You do not have to create a unique **.sldprt** document for each component in the assembly.

You want to place the bottom of the handle component on the flat base of the faucet component, so the handle sits on the faucet. You also need to center the handle components over the faucet stems to position them correctly. To position the components, you apply a coincident mate and a concentric mate.

Load the First Assembly Component

When creating an assembly, start with the component that does not move with respect to the other components. This is the component you anchor or fix to the assembly origin. In the example of the faucet sub-assembly, you want to anchor the spigot component.

Anchoring the first component ensures that the planes in both documents are aligned.

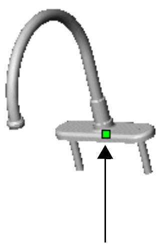
Bring the *first* component into the **.sldasm** document as follows:

- Select the component name in the FeatureManager design tree of the **.sldprt** document, and drag it into the **.sldasm** document.
- Use the pointer feedback to position the first component on the origin of the **.sldasm** document.

As you bring each component into the **.sldasm** document, the component appears in the FeatureManager design tree.

Load the Additional Components

You load the other components of the assembly by selecting the component in the FeatureManager design tree of the **.sldprt** document, and dragging the component into the graphics area of the **.sldasm** document. In the example of the faucet sub-assembly, you drag in two instances of the handle.



Faucet component with origin (assembly origin and component origin)

Faucet component with first handle component

Dragging the second handle component into the assembly

Position the Additional Components

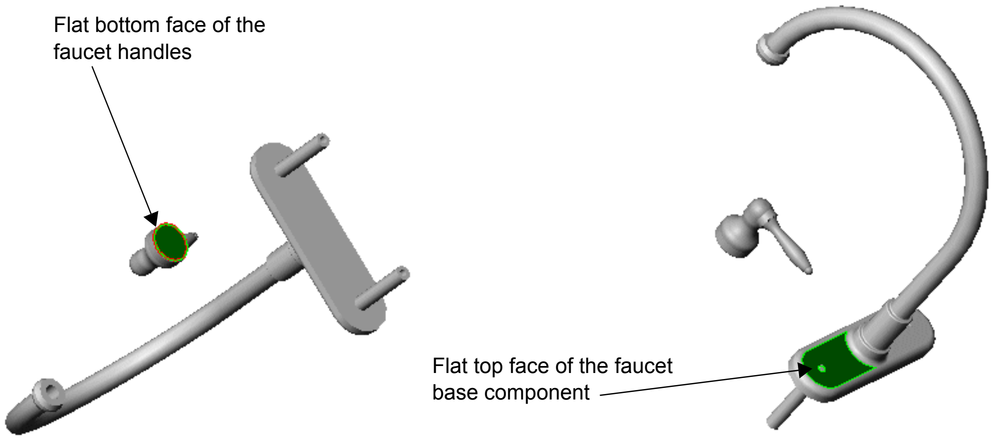
When you bring the additional components into the assembly, you can position them anywhere in the graphics area. Then you can use the **Move Component** tool to drag each component closer to the first, anchored component.

You should leave some space between the components to view the relevant component areas. In many instances, you use the **Rotate View** and **Pan** tools to change the orientation of the components. This facilitates selecting the edge, face, or other entity needed to apply mates. You can also use the **Zoom to Area** tool.

There are many methods to bring components into an assembly. For example, you can drag a component from Windows Explorer, from a hyperlink in Internet Explorer, from within the assembly, or from a Feature Palette window. For more information, see **Adding Components to an Assembly** in the *SolidWorks Online User's Guide*.

Coincident Mate

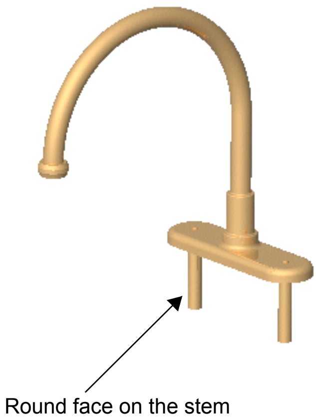
To create a coincident mate between the handle component and the faucet component, attach the flat bottom face of the handles to the flat top face of the spigot.

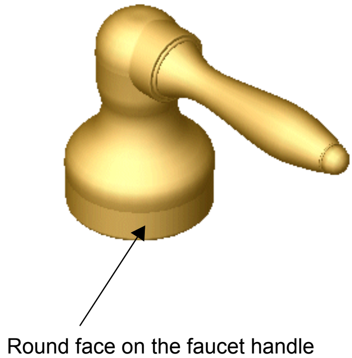


When you apply the coincident mate, it brings the faucet handle component closer to the faucet component. Note that you can still slide the handle anywhere along the top face of the faucet with the **Move Component** tool. This indicates that a second mate is required to further define the position of the two components.

Concentric Mate

Select any round face on the faucet handle. Then select the round face of the faucet stem (the portion of the component that slides into the counter top, and connects to the supply pipe).





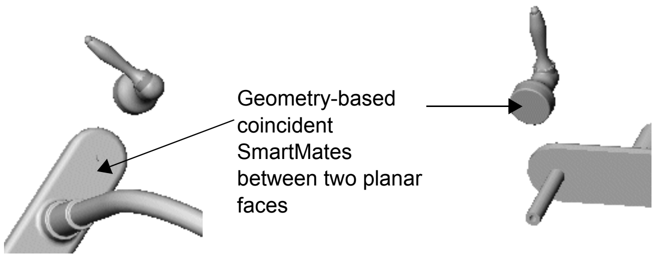
Once you apply the concentric mate between the faucet handle component and the spigot component, you can no longer move the faucet handle along the top face of the spigot to shift its position. You can, however, use the **Move Component** tool to rotate the faucet handle on its axis.

For a lesson on assembly mates, see **Assembly Mates** in the *Online Tutorial*.

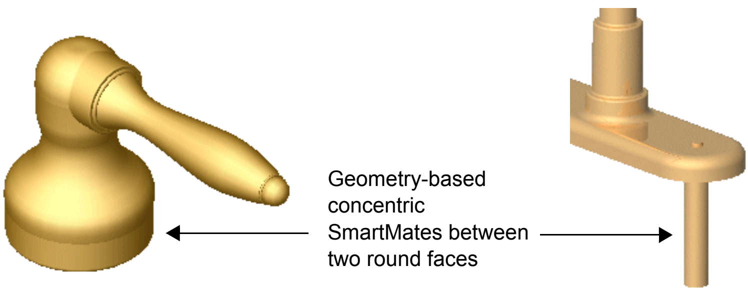
Faucet Sub-Assembly - Alternate Design Approach

Another approach to mating the faucet and handle components is to use SmartMates. With SmartMates, the system automatically creates some mates. SmartMates are based on the entity you use to drag the component.

When you drag components into assemblies, you inference the geometry of existing components to create mates as you drop the new components. There are different types of SmartMates. You can use geometry-based SmartMates to create coincident mates between planar faces. For example, use SmartMates to create a coincident mate between the faucet component and each of the faucet handles in the faucet sub-assembly.



You can use another type of geometry-based SmartMate to create the concentric mate between the two round faces, which are needed to completely define the spigot sub-assembly.



There are other types of SmartMates, including feature-based SmartMates, and pattern-based SmartMates. For more information, see **SmartMates** in the *SolidWorks Online User's Guide*.

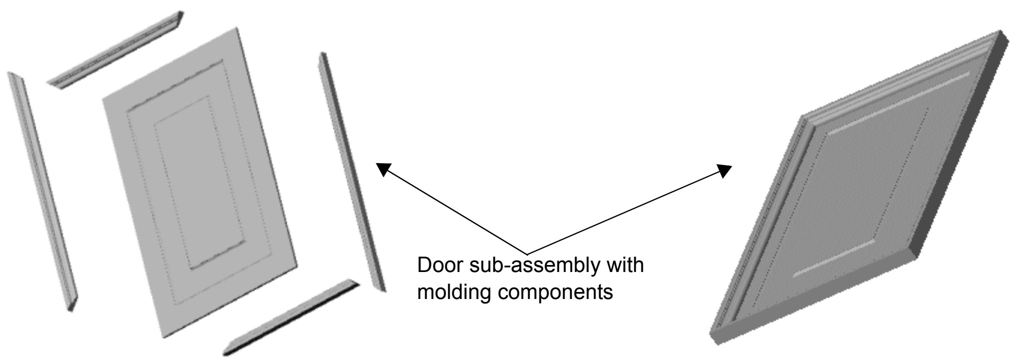
Door Sub-Assembly

The cabinet door uses coincident mates between the door component and the four molding components. It also uses a configuration of the molding as a time-saving design step.

Configurations allow you to create multiple variations of a part or an assembly within a single document. Configurations provide a convenient way to develop and manage families of models with different dimensions, components, or other parameters (see **Use Configurations of a Part** on page 2-13).

As stated earlier, you can use the same **.sldprt** document more than once in an assembly. Each instance of the **.sldprt** document can also use a different configuration.

The door sub-assembly uses configurations. There are four instances of the molding component. Two of the instances use the **short** configuration and fit across the short sides of the door. The other two instances use the **long** configuration.



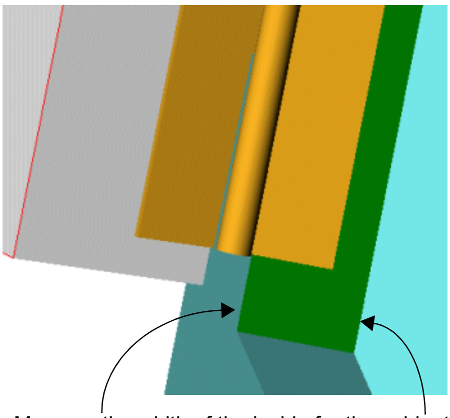
Cabinet Sub-Assembly

The cabinet sub-assembly uses concentric and coincident mates (discussed earlier with the faucet). It also uses a distance mate between the cabinet and one of the hinge components.

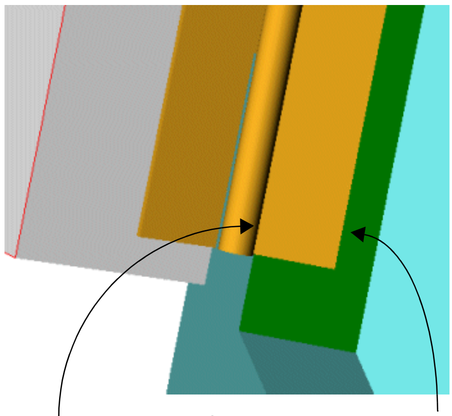
Distance Mate

A distance mate uses a value you assign between two entities. In the vanity cabinet, the distance mate positions the hinge optimally, so that it functions freely. You determine the correct mate distance using the **Measure** tool.

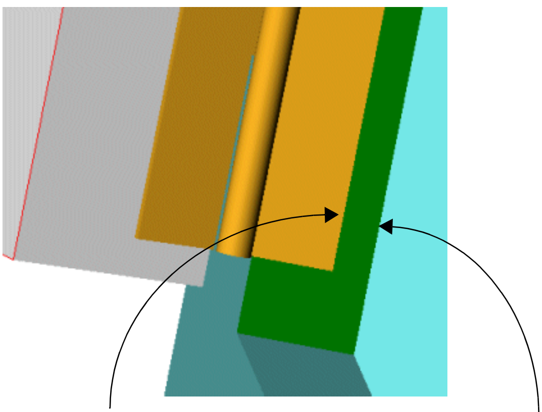
By measuring the entities of different components, you can determine at what position to place the hinge so that it does not bind when you open the cabinet door. Once you know the thickness of the door opening and the width of the hinge, you can position the hinge using a distance mate.



Measure the width of the inside for the cabinet door opening



Measure the width of the hinge that you attach to the inside of the cabinet door opening



Distance mate, based on measurements of the cabinet and of the hinge

In-Context Design

In addition to creating or editing components in their own part windows, the SolidWorks software allows you to create or edit components *in the assembly window*. The advantage is that you can reference the geometry of one component to create or modify another component. By referencing the geometry of another component, you ensure that the components fit together correctly. This method of design is called in-context design because you are working in the context of the assembly.

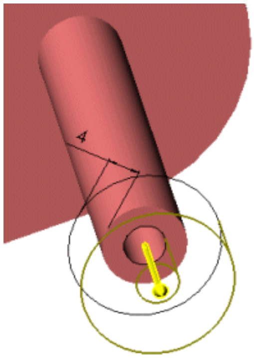
In the vanity assembly, there are two examples of in-context design. One example is the diameter of the supply pipe component and the waste pipe component. The pipe components are both new parts that you create in the context of the assembly. The other example is the cut feature for the holes in the back of the vanity cabinet. The vanity cabinet is an existing part that you edit in the context of the assembly. These examples are discussed in the next two sections.

As you create an in-context part, the software adds the sketches and features to the FeatureManager design tree as if the part were created in its own window. The software also adds an item to the FeatureManager design tree that lists information about the part such as what was used to create it and the referenced assembly component.

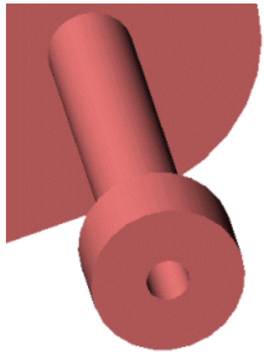
For more information on creating in-context components, see **Creating a Part in an Assembly** in the *SolidWorks Online User's Guide*.

Create an Assembly Component In-Context

The diameter of the supply pipe component depends on the diameter of the faucet stem. It is a good idea to create the supply pipe component in the assembly so you can reference the geometry of the faucet stem. You use the **Convert Entities** and **Offset Entities** sketch tools to reference the geometry of the faucet stem for a sketch in the supply pipe component. This ensures that the size of the supply pipe changes if you change the size of the faucet stem. You can use the same method to create the waste pipe component, which depends on the diameter of the exit stem at the bottom of the basin.



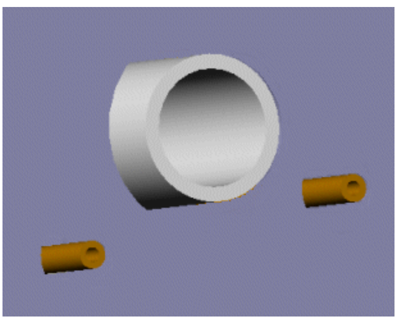
Use convert entities and offset entities to create the sleeve between the faucet stem and the supply pipe



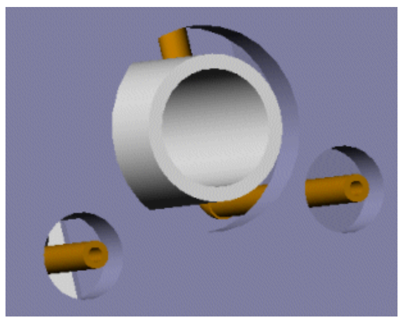
Extrude the sketch to create the sleeve between the faucet stem and the supply pipe

Modify a Part In-Context of an Assembly

The positions of the holes in the back of the vanity cabinet depend on the length of the supply pipe and the waste pipe components. It is a good idea to edit the vanity cabinet component in the assembly so you can reference the geometry of the supply pipes and waste pipe. You use the **Offset Entities** sketch tool to reference the geometry of the pipes for a sketch of the cut in the vanity cabinet component. This ensures that the position and size of the holes changes if you change the position and size of the supply pipes or waste pipe.



Supply and waste piping before in context cut



Supply and waste piping after in context cut

Load an Assembly

After you create an assembly, you can load it with its active components fully resolved or lightweight.

- When a component is fully resolved, all of its model data is loaded in memory.
- When a component is lightweight, only a subset of its model data is loaded in memory. The remaining model data is loaded on an as-needed basis.

You can improve performance of large assemblies significantly by using lightweight components. Loading an assembly with lightweight components is faster than loading the same assembly with fully resolved components.

Lightweight components are efficient because the full model data for the components is loaded only as it is needed.

Assemblies with lightweight components rebuild faster because fewer details are evaluated. However, mates on a lightweight component are solved, and you can edit existing mates.

The vanity cabinet is a relatively simple assembly, so any performance gains using lightweight components will be minimal.

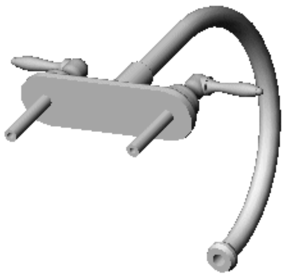
Examine the Assembly

The SolidWorks software includes various assembly tools that can display, test, and measure your assembly components once you apply the mates. Some of the assembly tools include the following:

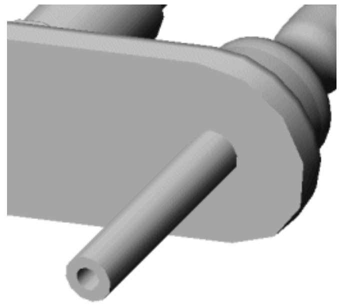
- Show and hide components
- Explode the assembly
- Detect collisions between components

Show and Hide Components

You can hide or show components in the graphics area. Hiding components often facilitates component selection when you add mates or when you create in-context parts. For example, to select the inner and outer diameters of the faucet stems, you can hide all components except the faucet sub-assembly, and then zoom, rotate, or change the view as needed.



Hide all components except the one you need

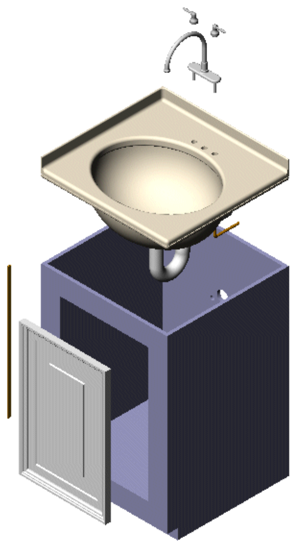


Zoom, rotate, and change the view if necessary to select the feature

Show Components and **Hide Components** does not affect the mates between the components. It affects only the display.

Explode the Assembly

An exploded view separates the components in an assembly to facilitate viewing. Exploded views include many options such as which components to include, what distances to use, and in which direction to display the exploded components. The exploded view is saved with a configuration of the assembly or sub-assembly.



Detect Collisions Between Components

You can detect collisions with other components when you move or rotate a component. The SolidWorks software can detect collisions with the entire assembly or a selected group of components that move as a result of mates.

In the faucet sub-assembly, note how the faucet handles collide with the spigot. You can set the **Stop at Collision** option to determine where the components collide.

Normal position of handles

Collision detection *without* Stop at Collision active. Notice the handle moves inside the spigot.

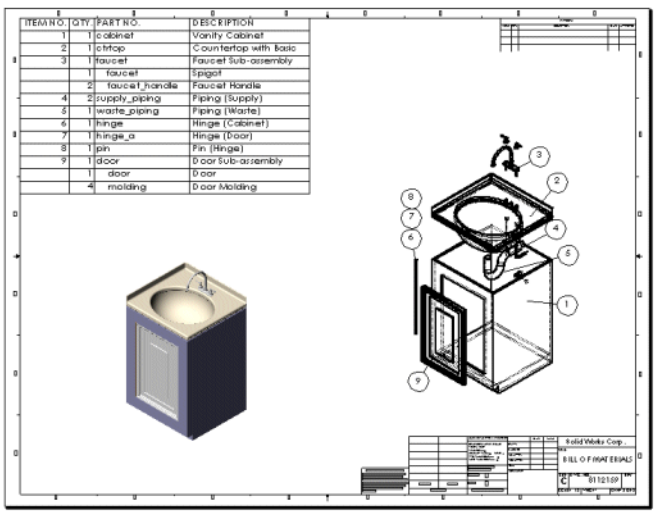
Collision detection *with* Stop at Collision active. Notice the handle cannot move inside the spigot.

4

Drawings

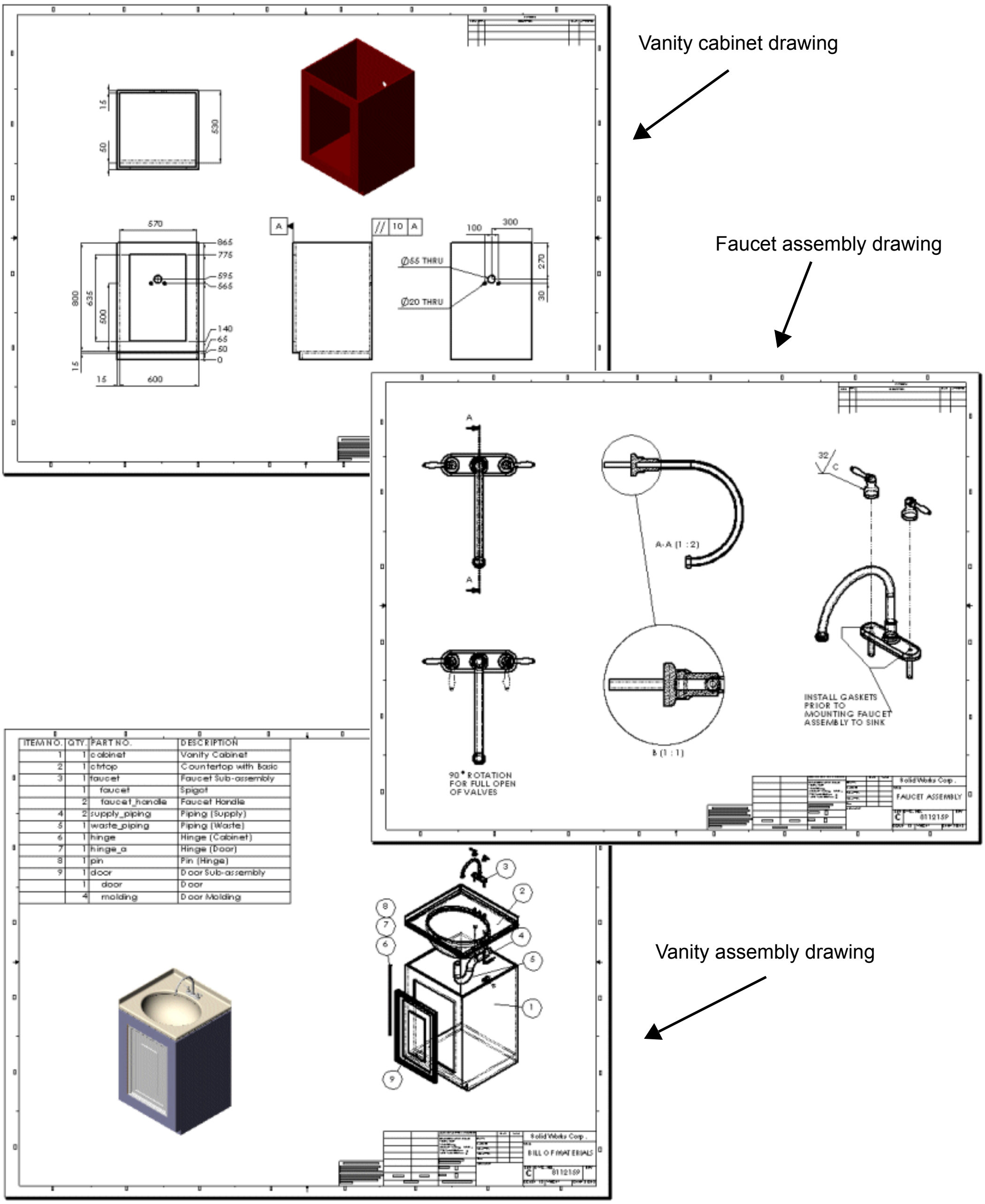
Drawings are 2D documents that convey a design to manufacturing. This chapter covers the following topics while creating three drawing sheets:

- **Drawing documents**. Choose from a set of standard templates, customize sheet formats and templates, and create multiple sheets in a drawing document.
- **Vanity cabinet drawing sheet**. Insert Standard 3 Views and Named Views from a part document. Display views in various modes and align views with each other. Add dimensions and annotations.
- **Faucet assembly drawing sheet**. Create section, detail, and other derived views from standard views. Insert notes and additional annotations.
- **Vanity assembly drawing sheet**. Insert an exploded view, a Bill of Materials, and balloons.



Drawing Documents

Drawings of the vanity cabinet, faucet assembly, and vanity assembly introduce drawing and detailing concepts in this chapter.



Drawings open from document templates. Within a drawing document are drawing sheets that contain drawing views. The drawing sheets have underlying formats.

For a lesson on drawing documents, inserting standard views, and adding dimensions to drawings, see **Lesson 3 - Drawing Basics** in the *Online Tutorial*.

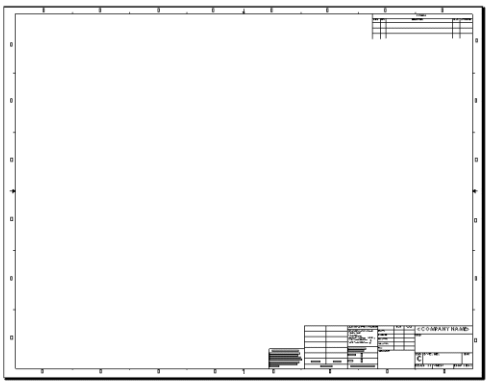
Document Templates

To start the drawing document, you open a document template. Document templates contain basic document information. You choose from document templates supplied with the SolidWorks software (which contain default drawing sheets) or document templates you customize. You can create custom document templates, for example, with any of the following characteristics:

- Drawing sheet size (A, B, C, and so on)
- Drawing standard (ISO, ANSI, and so on)
- Units (millimeters, inches, and so on)
- Company name and logo, author’s name, and other information

Drawing Sheets

For the vanity drawings, a document template with a C size drawing sheet in landscape orientation is appropriate. The standard drawing sheet formats contain borders and title blocks such as shown here for the C-Landscape format.



Drawing sheets have a specified size, orientation, and format. When you open the standard drawing template, or when you add a sheet to a drawing document, you choose among the following types of sheet formats:

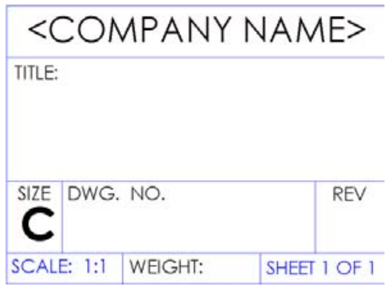
- **Standard**. Standard sizes and orientations.
- **Custom**. A sheet format that you have edited and saved.
- **None**. Only the size of the sheet specified.

The drawing document for the vanity contains three sheets. You can have any number of drawing sheets in a drawing document, like a set of drawings. You can add sheets at any time, using any format regardless of the format for other sheets in the document. Tabs with sheet names appear at the bottom of the graphics area.

The properties that you can edit include the name of the sheet, paper size, sheet format, scale, type of projection, view labels, and the source of custom property values.

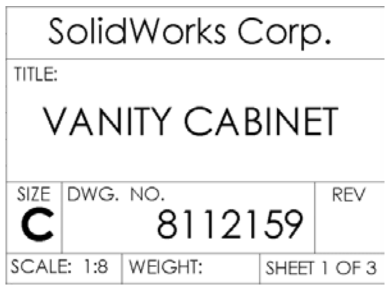
Sheet Formats

The bottom right corner of the default sheet format contains a title block, which is shown at the right.



After changing the sheet scale, adding two sheets, and editing and adding notes, the title block appears as shown below. The scale and page numbers are linked to system variables and updated automatically.

You can also add, delete, and change the weight of lines in the sheet format.



The sheet format underlies the drawing sheet and is separate from the drawing sheet. You edit the sheet format separately from the drawing sheet. Drawing sheet formats can contain lines, note text, bitmaps, Bill of Materials anchor point, and so on. You can link the notes to system properties and custom properties.

You enter custom properties in the summary information dialog box in part documents. The properties are then available in a listing in the dialog box for linking notes to properties.

Drawing Views

Drawing views are placed on drawing sheets and contain the images of the models, plus dimensions and annotations. Drawings begin with standard views. From those views you can derive other types of views, such as projected, section, detail, and so on.

The various types of views are described in the next sections.

? For more information on document templates, drawing sheets, and drawing views, see the *SolidWorks Online User's Guide*.

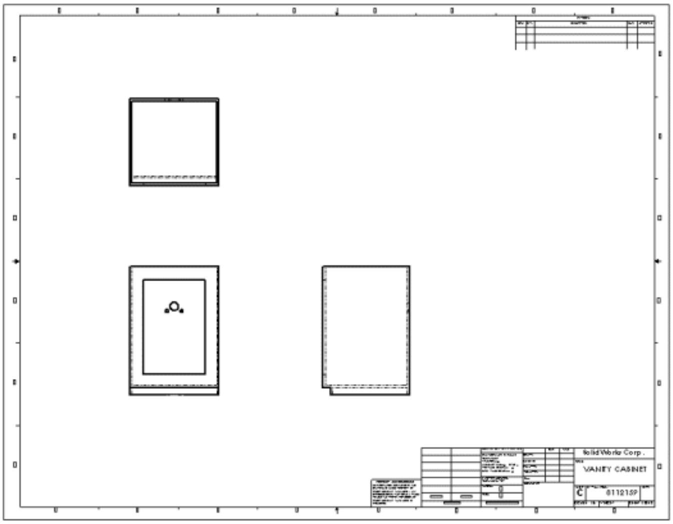
Vanity Cabinet Drawing Sheet

Standard Views

Drawings generally start with a Standard 3 View or some type of Named View (isometric, or exploded, for example). You can insert these views from other views in the same drawing document, from an open part or assembly document, or from a file.

Standard 3 Views

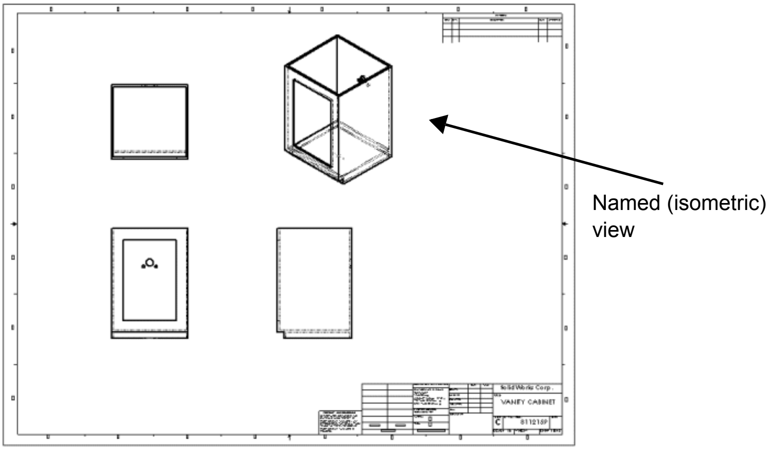
Insert the standard three views of the vanity cabinet onto the Vanity Cabinet drawing sheet.



Standard 3 Views, as the name implies, comprise three views: front, top, and right (third angle projection) or front, top, and left (first angle projection). In third angle projection, the default front view is displayed at the lower left. In first angle projection, the front view is displayed at the upper left. The example in this chapter uses third angle projection.

Named Views

Add an isometric view of the cabinet (a Named View) to the drawing sheet.



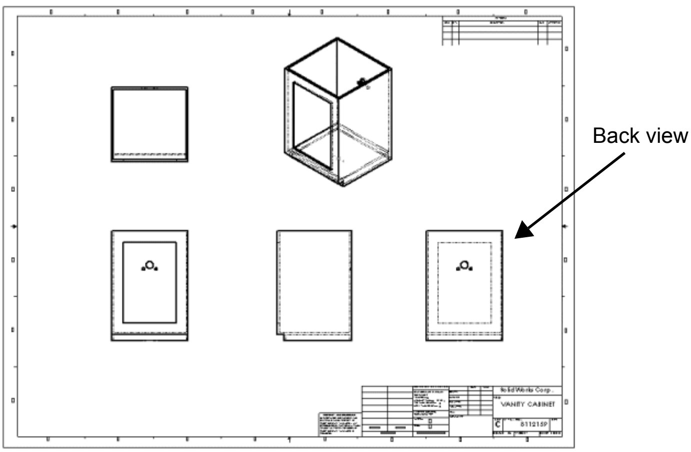
Views are named in the model documents. Named views include:

- Standard orientations (Front, Top, Isometric, and so on)
- Current model view
- Custom named views

You select the view orientation as you bring the view into the drawing.

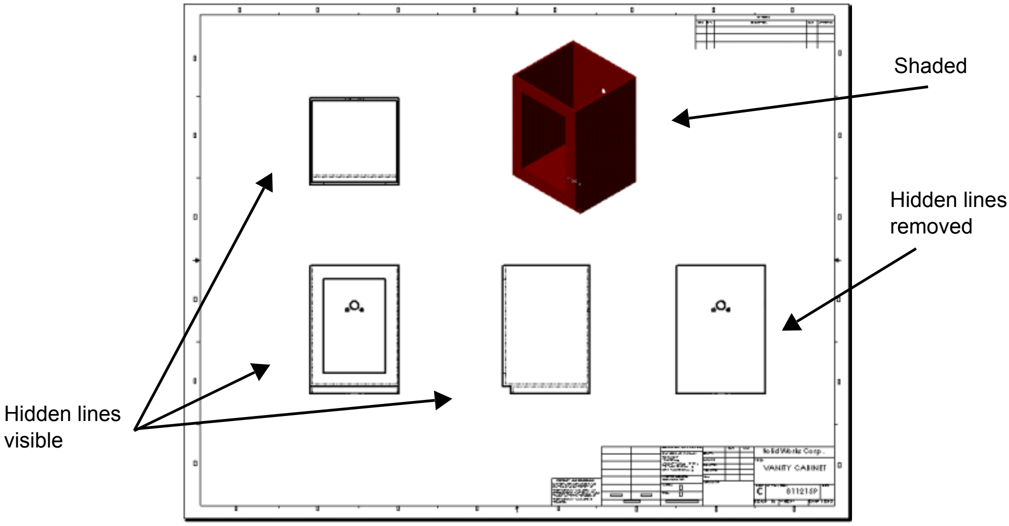
Projected Views

The vanity has details in the back that are important to show. To create a Back view, project the Right view and place it to the right. Projected Views are orthographic projections of existing views.



View Display and Alignment

You can choose various display modes for drawing views. On the Vanity Cabinet sheet, the back view is in Hidden Lines Removed. The Standard 3 Views show Hidden Lines Visible. (Hidden lines appear gray on screen, but as dashed lines when printed.) The isometric view is shown in Shaded mode.



Some views are aligned automatically; however, you can break the alignments. Standard 3 Views are aligned so that if you drag the front view, both the top and right view move with it. The right view moves independently in the horizontal direction, but not vertically. The top view moves independently in the vertical direction, but not horizontally.

Section Views, Projected Views, and Auxiliary Views automatically align in the direction of the view arrows. Detail Views are not aligned by default.

You can align views that are not automatically aligned. For example, the back view of the cabinet is aligned horizontally with the right view, which is aligned with the front view by default.

For more information on displaying, hiding, and aligning views, see **Drawing View Alignment and Display** in the *SolidWorks Online User's Guide*.

Dimensions

Dimensions in a SolidWorks drawing are associated with the model. Changes in the model are reflected in the drawing, and vice versa.

Typically, you create dimensions as you create each feature in a part, then insert those dimensions into the drawing views. Changing a dimension in the model updates the drawing, and changing a model dimension in a drawing changes the model.

You can also add dimensions in the drawing document, but these are *reference* dimensions, and are driven; you cannot edit the value of reference dimensions to change the model. However, the values of reference dimensions change when the model dimensions change.

You can set the units (millimeters, inches, and so on) and drawing standard (ISO, ANSI, and so on) in the options for detailing. The vanity is in millimeters in the ISO standard.

For more information about dimensions in drawings, see **Dimensions Overview** in the *SolidWorks Online User's Guide*.

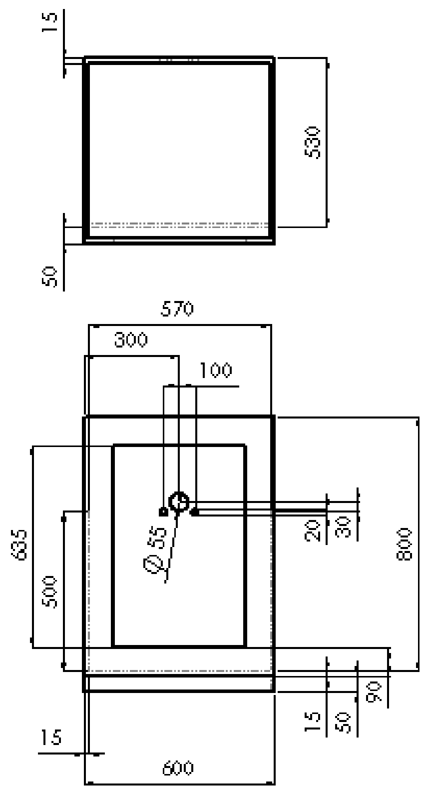
Insert Model Items

Insert the model dimensions into the cabinet drawing in one operation with the **Insert Model Items** tool. You can insert items for a selected feature, an assembly component, a drawing view, or all views.

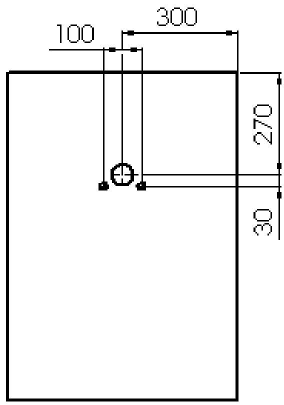
When inserted into all views (as in the example), dimensions and annotations appear in the most appropriate view. Features that appear in partial views, such as Detail or Section views, are dimensioned in those views first.

Once you insert the dimensions, you can drag them into position, drag them to other views, hide them, edit properties, and so on.

If the model contains annotations, you can also insert the annotations into drawings by the same procedure.

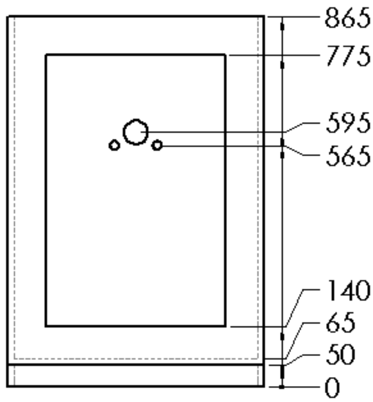


Reference Dimensions



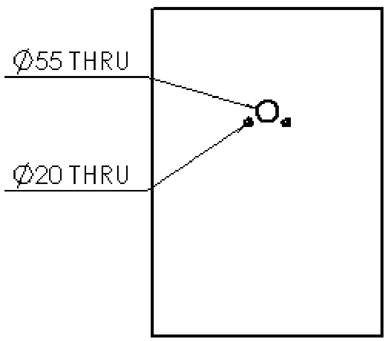
The back view on the Vanity Cabinet sheet is included to show the dimensions of the holes in the cabinet for the supply and waste pipes.

Add reference dimensions to locate the holes. You can choose in the options for dimensions whether to enclose reference dimensions in parentheses automatically.



Other types of reference dimensions that might be added to a drawing view are Baseline Dimensions and Ordinate Dimensions. For example, you might add Ordinate Dimensions to the front view of the cabinet as shown. You can dimension to edges, vertices, and arcs. The dimensions jog automatically to avoid overlapping. You can display ordinate dimensions without the chain (the arrows between the dimension extension lines).

Hole Callouts



To specify the size and depth of the holes in the cabinet, add Hole Callouts (annotations that are also dimensions) to the back view.

You can specify Hole Callouts when creating holes in models with the Hole Wizard. (The Hole Wizard creates and positions user-defined holes for fasteners such as counterbore and countersunk screws, tap holes, and so on). The information on diameter, depth, counterbore, and so on that is part of the Hole Wizard design data becomes part of the Hole Callout automatically.

Annotations

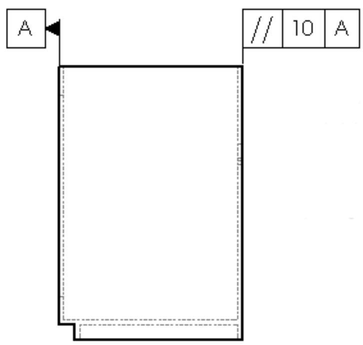
Many annotations are available from tools and menus, including the following:

- Note
- Geometric Tolerance Symbol
- Datum Feature Symbol
- Center Mark
- Surface Finish Symbol
- Datum Target Symbol
- Weld Symbol
- Balloon and Stacked Balloon
- Block
- Multi-jog Leader
- Area Hatch
- Dowel Pin Symbol

Most annotations can be added in part and assembly documents and inserted automatically into drawings in the same way that dimensions are inserted into drawings. Some annotations (balloons, center marks, blocks, multi-jog leaders, hole callouts, area hatch, and dowel pin symbols) are available in drawings only.

For more information on annotations, see **Annotations Overview** in the *SolidWorks Online User's Guide*.

Geometric Tolerance and Datum Feature Symbols



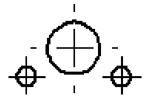
In the right view of the cabinet, the back edge of the cabinet is specified with a Geometric Tolerance Symbol to be parallel to the front edge within 10mm.

Geometric Tolerance Symbols display various manufacturing specifications, often in conjunction with Datum Feature Symbols as shown in the example. You can insert these symbols in sketches and in part, assembly, and drawing documents.

You can specify the datum letter, a square or round border, and other characteristics of the Datum Feature Symbol.

Geometric Tolerance Symbols can have any number of frames with symbols, numbers, and datum letters. Leaders can be single or multiple and straight, bent, perpendicular, or multi-jog.

Center Marks

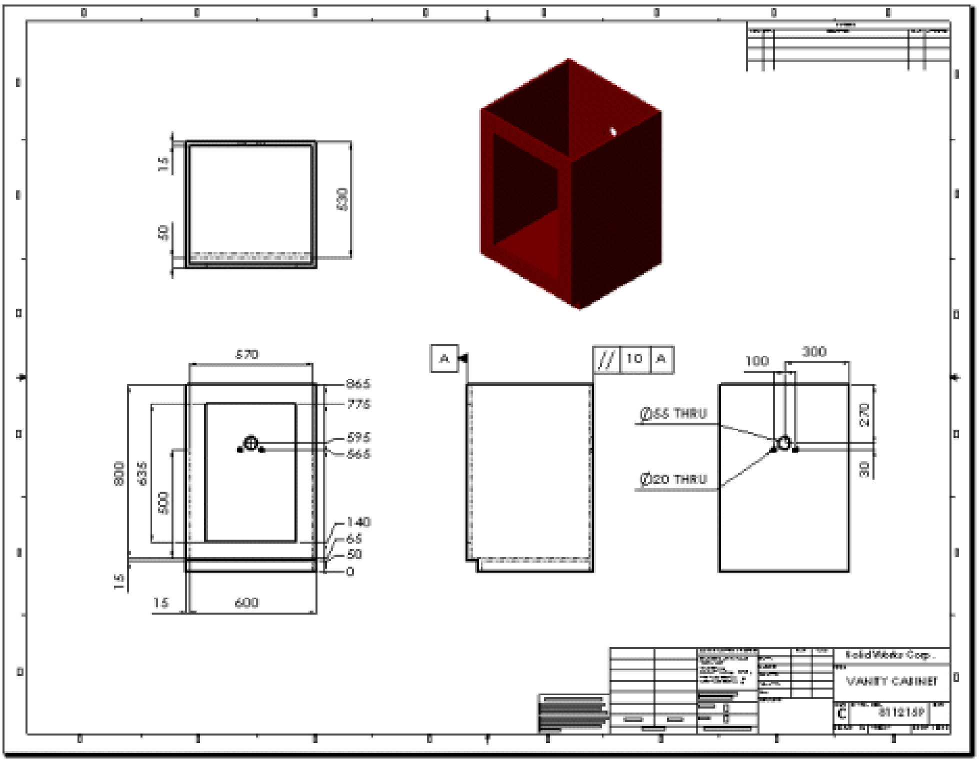


Add Center Marks to the holes in the Back view of the cabinet. You can place center marks on circles or arcs. Center marks can be used as reference points for dimensioning.

You can rotate center marks, specify their size, and choose whether or not to display extended axis lines.

For a lesson on adding derived views, annotations, and exploded views to drawings, see **Advanced Drawings** in the *Online Tutorial*.

Here is the completed Vanity Cabinet drawing sheet.



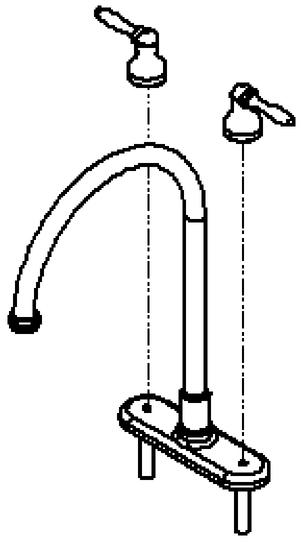
Faucet Assembly Drawing Sheet

The Faucet Assembly drawing sheet displays several derived views and annotations.

Explode Lines

Insert an isometric Named View of the faucet assembly in its exploded configuration, with explode lines, on the right side of the Faucet Assembly drawing sheet. Explode lines show the relationships between assembly components.

You add the explode lines to the assembly document in an Explode Line Sketch. You can also jog the lines as needed. The lines are displayed in phantom line font.

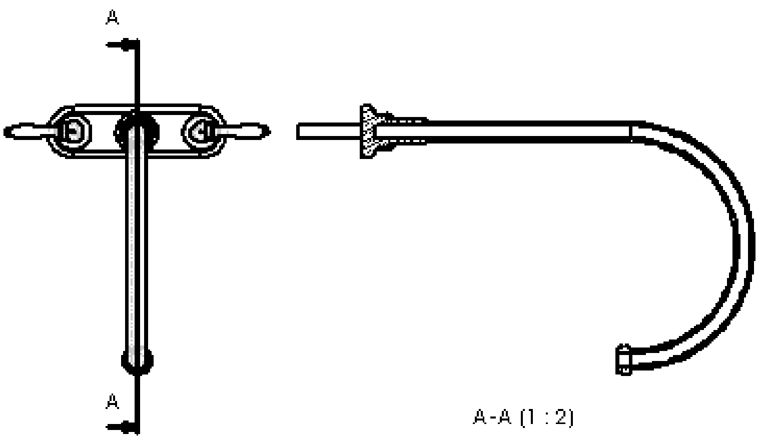


Derived Views

Derived views are created from the standard views. With Standard 3 Views or a Named View in a drawing, you can create other views without going back to the model. Insert a Top view of the faucet assembly at the top left of the drawing sheet.

Section Views

A Section View of the faucet spigot in the Faucet Assembly drawing shows the spigot pipe walls and connections. Insert a Top view of the faucet assembly as the basis for the Section View.



You create a Section View in a drawing by cutting the parent view with a section line. The section view can be a straight cut section or an offset section defined by a stepped section line. The section line can also include concentric arcs.

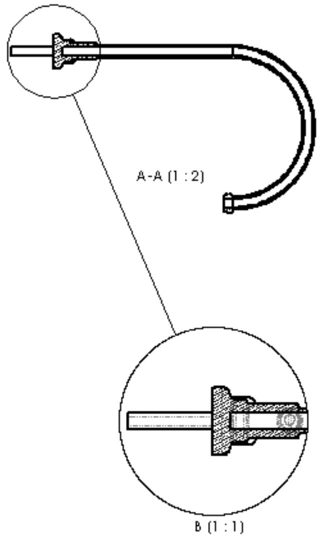
The sectioned components automatically display crosshatching. You can edit the properties of the crosshatching (pattern, scale, and angle).

Detail Views

The connection of the spigot is shown in a Detail View. The parent view is the section view.

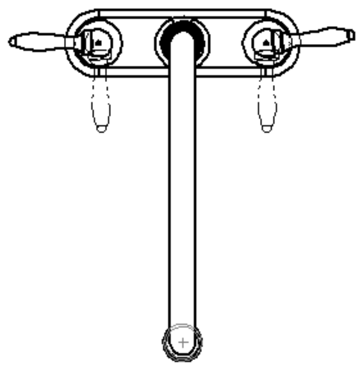
Detail Views show a portion of an orthographic, 3D, or Section View, usually at an enlarged scale.

You can edit the profile style, label, and font. The example shows a connected style. Other styles include a broken circle and the letter with or without a leader. The default follows the detailing standard for the document.



Additional Drawing Views

The faucet handles are shown on the Faucet Assembly sheet in an Alternate Position View to display the range of motion of the handles.



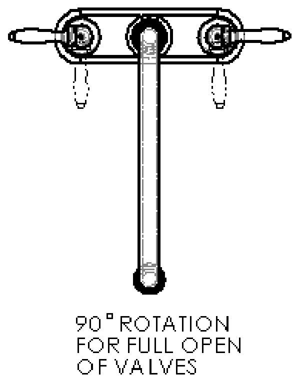
Alternate Position Views display overlays of two or more positions on the same view, often to show range of motion of an assembly component. The overlay views are displayed in the drawing in phantom lines.

Other drawing views available include:

- **Auxiliary View**. A projection normal to a reference edge.
- **Crop View**. Everything outside a sketched profile removed.
- **Broken-out Section**. Material inside a profile removed to expose inner details.
- **Broken View**. Portion of a long part with a uniform cross-section removed.

For more information about drawing views, see **Derived Drawing Views** in the *SolidWorks Online User's Guide*.

Notes and Other Annotations



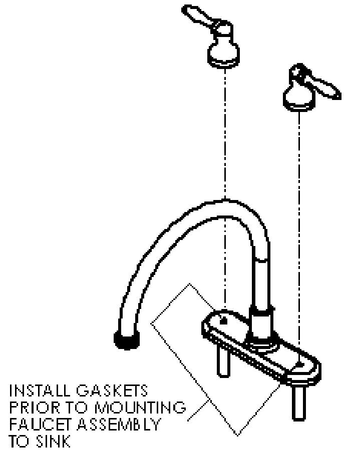
Notes and Multi-jog Leaders

Add a note, inserting the degree symbol, to the right of the Alternate Position View. Add another note, with a multi-jog leader, to the exploded faucet view.

A note can be free floating, as in the first example, or pointing to an item (face, edge, or vertex) in the document, as in the second example.

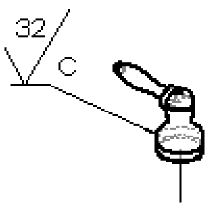
A note can contain text, symbols, a border, parametric text, and hyperlinks. You can justify the text. The leader can be straight, bent, or multi-jog.

You can also link notes to system and custom properties. Linking to properties assures that if the model changes, the drawing updates.



Surface Finish Symbols

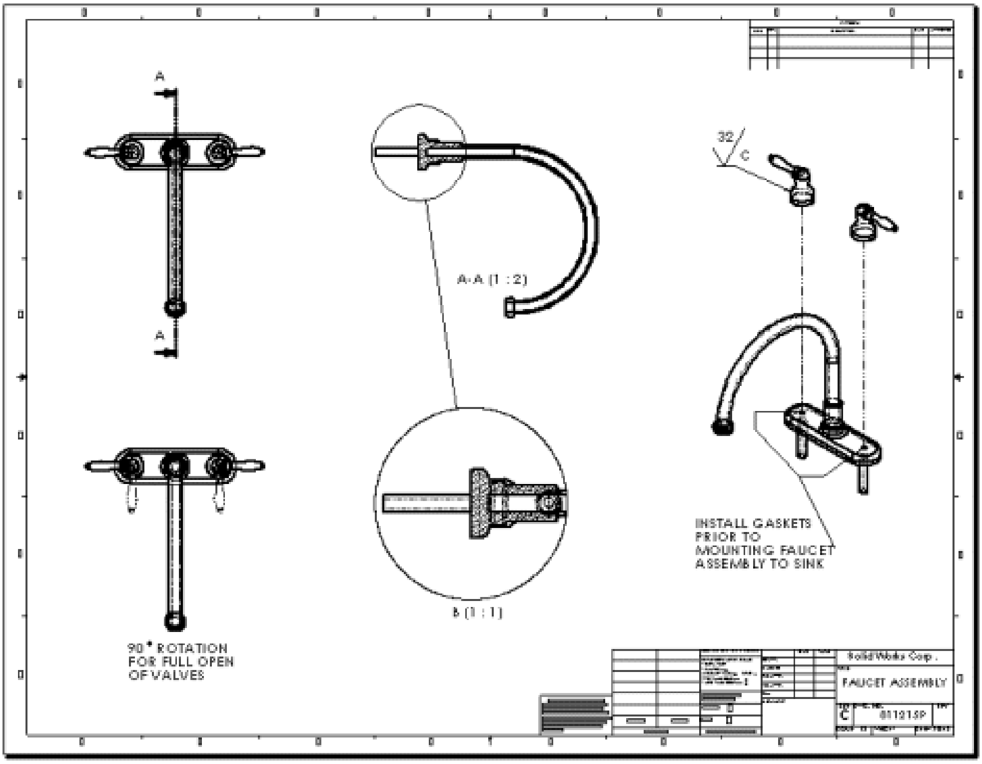
Insert a Surface Finish Symbol to the faucet handle on the exploded view to specify a circular finish and the maximum roughness of the surface.



You can add Surface Finish Symbols to part, assembly, or drawing documents. You can insert multiple symbols and multiple copies of a symbol. Leaders can be straight, bent, or multi-jog.

Some of the characteristics that you can specify for a Surface Finish Symbol include type of symbol, direction of lay, roughness, production method, material removal, and rotation.

Here is the completed Faucet Assembly drawing sheet.



Vanity Assembly Drawing Sheet

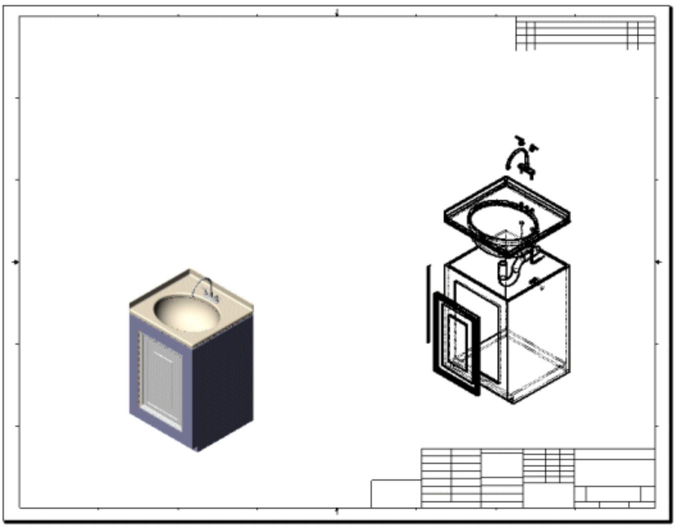
For the Vanity Assembly drawing sheet, insert an exploded view of the vanity assembly, an isometric view, a Bill of Materials table, and balloons.

A Bill of Materials (BOM) is a table listing the components of an assembly along with information needed in the manufacturing process. If the assembly or its components change, the BOM updates to reflect the changes.

Exploded Views

Add an exploded view of the vanity assembly on the right side of the drawing sheet. Exploded Views are versions of Named Views and are defined in configurations in the assembly document.

Also add an isometric Named View of the complete assembly, unexploded, at the lower left.



Bill of Materials

Insert a Bill of Materials into the Vanity Assembly drawing sheet, anchoring it at the upper left corner of the inside border.

ITEM NO.	QTY.	PART NO.	DESCRIPTION
1	1	cabinet	Vanity Cabinet
2	1	ctrtop	Countertop with Basic
3	1	faucet	Faucet Sub-assembly
	1	faucet	Spigot
	2	faucet_handle	Faucet Handle
4	2	supply_piping	Piping (Supply)
5	1	waste_piping	Piping (Waste)
6	1	hinge	Hinge (Cabinet)
7	1	hinge_a	Hinge (Door)
8	1	pin	Pin (Hinge)
9	1	door	Door Sub-assembly
	1	door	Door
	4	molding	Door Molding

When you insert a Bill of Materials, you have a choice of BOM templates with various columns for data such as item number, quantity, part number, description, material, stock size, vendor number, and weight. You can also edit and save a custom BOM template.

The SolidWorks software populates the item number, quantity, and part number columns automatically. The item number is in the sequence in which the model was assembled. In the example, the BOM is edited to add text in the description field.

You set the anchor point for the BOM in the drawing sheet format.

For more information, see **Bill of Materials Overview** in the *SolidWorks Online User's Guide*.

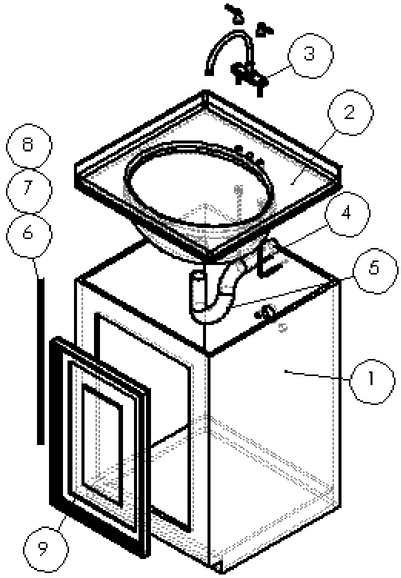
Balloons and Stacked Balloons

Add balloons and stacked balloons to the components of the exploded vanity assembly as shown on the next page. The item numbers appear in the balloons automatically.

Stacked balloons have one leader for a series of balloons. You can stack the balloons vertically or horizontally.

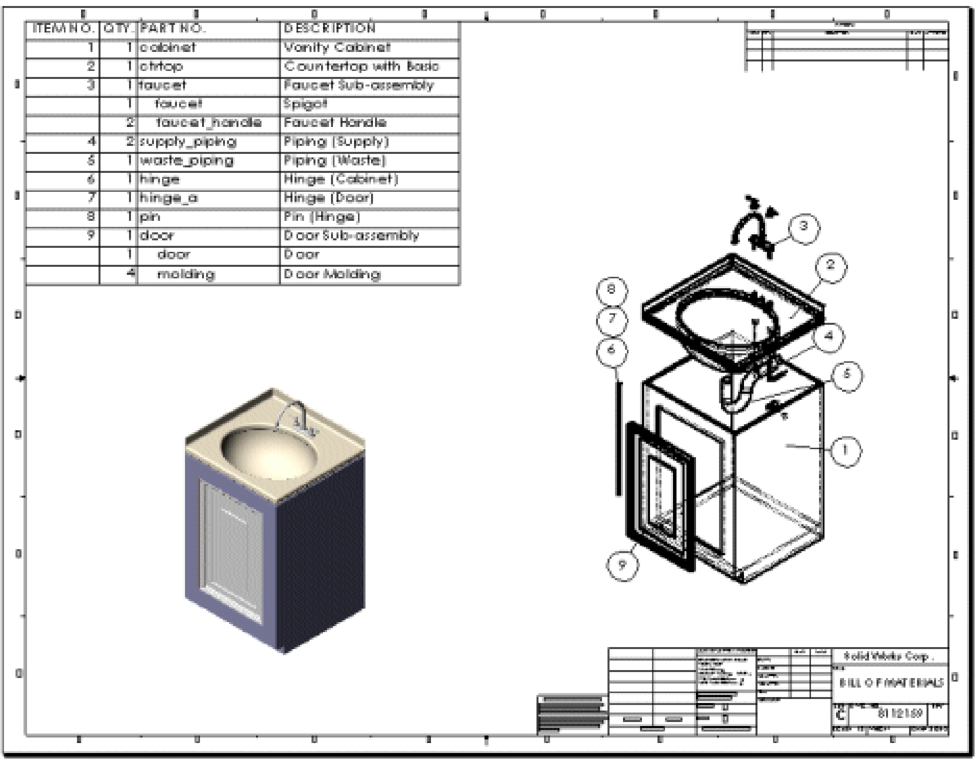
You can insert balloons in assembly and drawing documents. You can set the style, size, and type of information for balloons. In the example, the balloons display the item number corresponding to the BOM in a circle. Some of the balloon shapes available are box, triangle, pentagon, diamond, underline, and split circle.

The type of information in a balloon, in addition to item number, can be quantity or custom. Quantity is often used in the lower portion of a split style balloon. With custom, you can type in any text. Custom balloons are useful, for example, to specify adhesives or liquids.



For a lesson on Bills of Materials and balloons, see **Bill of Materials** in the *Online Tutorial*.

Here is the completed Vanity Assembly drawing sheet.



5

Engineering Tasks

The SolidWorks software contains several tools and functions that help you complete tasks specific to designing models. These tasks include creating variations of parts, importing files from legacy CAD systems to your SolidWorks models, and so on.

This chapter focuses on how the SolidWorks software helps you accomplish various engineering tasks, including:

- **Design tables.** Build configurations with embedded tables.
- **Dimension revisions.** Update referenced documents when a model changes.
- **Import and Export.** Import and export various file formats to and from the SolidWorks application.
- **Reload and Replace.** Refresh shared documents and replace referenced documents.
- **COSMOSXpress.** Simulate design cycles for parts that you create.
- **Application Programming Interface (API).** Customize the SolidWorks software to reduce design time.
- **eDrawings.** Create compact SolidWorks files and allow others to view your models.
- **FeatureWorks.** Recognize features as SolidWorks features from an imported file.
- **PhotoWorks.** Render your models to make photo-realistic images.
- **SolidWorks 3D Instant Website.** Create a web page that includes models from SolidWorks.
- **SolidWorks Animator.** Animate SolidWorks assemblies and generate **.avi** files.
- **SolidWorks Explorer.** Manage and organize your SolidWorks files.
- **SolidWorks Toolbox.** Use a library of standard hardware to insert into assemblies.
- **SolidWorks Utilities.** Examine and edit individual parts, compare the features and geometry among parts, and copy feature parameters from one part to another.

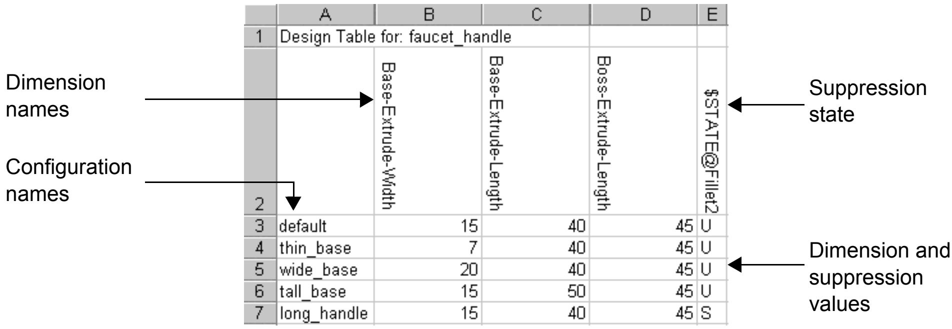
Design Tables

Design tables allow you to build several configurations of a part by applying the values in the table to the dimensions of the part.

In Chapter 2, “Parts,” you used configurations to build two different lengths of the molding in one part file. While configurations help organize the ConfigurationManager, design tables help you organize several configurations in a tabular format.

For example, you may want to create multiple configurations of the faucet handle. After all, not every customer wants the same handle style. In the SolidWorks software, you can create several different handle styles within one part file with a design table.

The following design table shows the parameters used to create variations of the faucet handle:



	A	B	C	D	E
1	Design Table for: faucet_handle				
2		Base-Extrude-Width	Base-Extrude-Length	Boss-Extrude-Length	$STATE@Fillet2
3	default	15	40	45	U
4	thin_base	7	40	45	U
5	wide_base	20	40	45	U
6	tall_base	15	50	45	U
7	long_handle	15	40	45	S

The first column lists the different configuration names. These configuration names describe the type of handle generated from the design table.

It is best to give a meaningful name for each configuration. This saves confusion in complex parts and assemblies, and helps others who use the models.

The next three columns show the dimension names and values. When you change a dimension value in a design table, the configuration updates with the specified value.

The final column shows the suppression state of a fillet feature. In addition to changing dimension values, you can also change the suppression state of a feature in design tables. A feature can be suppressed (**S**) or unsuppressed (**U**).

Each configuration has one value changed from the default configuration. The values and suppression states define each configuration as shown.

For a lesson on design tables, see **Design Table** in the *Online Tutorial*.

Configuration name	Model view
default	
thin_base	
wide_base	
tall_base	
long_handle	

Dimension Revisions

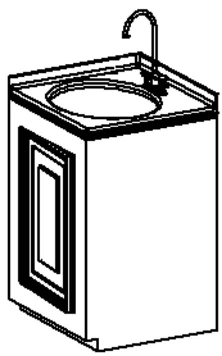
When you change a model dimension, any SolidWorks document that references that model also updates. For example, if you change the length of an extrude in a part, the associated assembly and drawing also change.

More specifically, you designed the faucet spigot to 100mm in length for a vanity countertop. However, your customer needs a longer spigot to accomodate a utility sink. You can modify the dimension of the spigot to make it any length, and the associated assembly and drawing also update.

Original spigot

Revised spigot

Revised assembly



Revised drawing

Import and Export

You can import and export several different file formats to and from the SolidWorks software. SolidWorks allows for open architecture, so you can share files amongst a broad user base.

Consider that your company works with a vendor that uses another CAD system. With the SolidWorks import and export functions, you can share files between companies. This allows you more flexibility in the design process.

For a lesson on importing and exporting files, see **Import/Export** in the *Online Tutorial*.

Reload and Replace

Reload

You can refresh shared documents to load the latest version, including any changes made by one of your colleagues.

Suppose that you are working on a SolidWorks assembly document, and a co-worker just updated one of the assembly components. You can reload the revised component, and the SolidWorks software automatically updates the assembly. This is easier than closing the assembly and re-opening it with the revised part.

Replace

You can replace a referenced document with another document from anywhere on a network.

For example, consider that you are working on the faucet sub-assembly. Meanwhile, another engineer in your group designed a more cost-effective faucet handle. You can globally replace the current handles with the new ones, without having to delete and replace each handle.

Current sub-assembly

New handle

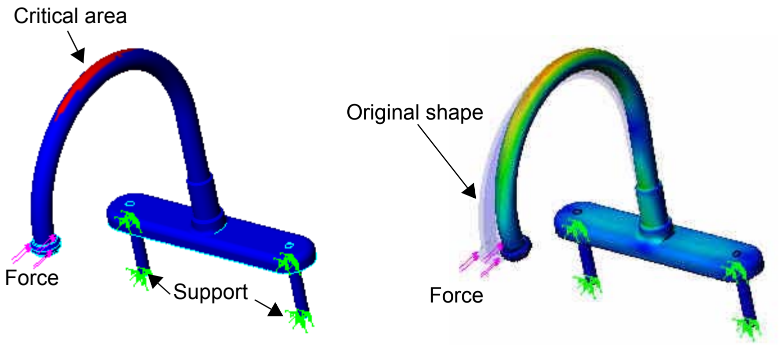
New sub-assembly

When you replace a component, mates used in the original part are applied to the replacement part wherever possible. To ensure that the mates are preserved, rename the corresponding edges and faces on a replacement part to match the edge and face names on the original part.

COSMOSXpress

COSMOSXpress simulates design cycles for parts that you create. With COSMOSXpress, you can reduce expensive and time-consuming prototypes and field tests.

For example, you may want to examine the effects of a force applied to the faucet. COSMOSXpress simulates the design cycle and provides displacement and stress results. It also shows critical areas and safety levels at various regions in the faucet. Based on these results, you can strengthen unsafe regions and remove material from overdesigned areas.



Stress distribution on deformed shape

For a lesson on COSMOSXpress, see **COSMOSXpress** in the *Online Tutorial*.

Application Programming Interface

The SolidWorks Application Programming Interface (API) is an OLE programming interface to the SolidWorks software. The API contains thousands of functions that can be called from Visual Basic, VBA (Excel, Access, and so on), C, C++, or SolidWorks macro files. These functions provide you with direct access to SolidWorks functionality.

With the API, you can customize the SolidWorks application to help reduce design time. You can perform batch operations, automatically populate drawing documents with model views or dimensions, create your own PropertyManagers, and so on.

For example, when you use any software application, you probably set system options to customize your working environment. In SolidWorks, these options may include system colors, default templates, and performance settings. With the API, you can set the system options without setting each one individually. Instead, use the API to automatically set all of your options. This way, you save time by programming the settings only once.

For more information, see the API online help file, or the API Support page on the SolidWorks Web site (http://www.solidworks.com/html/products/api).

eDrawings

eDrawings eliminates the communication barriers that designers and engineers deal with daily. You can create eDrawing files from part, assembly, or drawing documents, then email these eDrawing files to others for instant viewing.

For example, if you work with a client in a remote location, you may need to send a model for their approval. Oftentimes, the file size is too big to send through email. However, if you save your SolidWorks model as an eDrawing file, you can send a much smaller version of the file to your client.

You view eDrawing files with the eDrawings Viewer that you can download from the SolidWorks Web site for free.

eDrawing files have the following features:

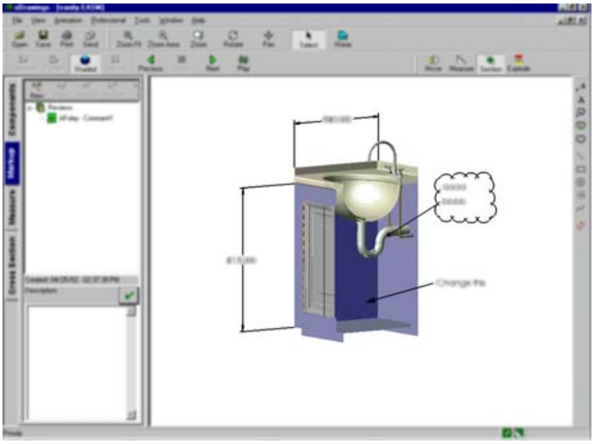
- **Ultra Compact Files**. Send eDrawings through email. Substantially smaller in size than the original files, eDrawings make it practical to send files via email, even over slow connections.
- **Built-in Viewer**. View eDrawings immediately. Anyone with a Windows-based computer can view eDrawings. No additional CAD software is required. You can embed the eDrawings Viewer when you email an eDrawing file.

eDrawing files are also significantly easier to understand than standard 2D drawings. The following features help you overcome common barriers for effective 2D drawing communication:

- **Layouts**. Open individual views in a drawing and arrange them in any way you desire, regardless of how the views were arranged in the original drawing. Layouts enable the eDrawings recipient to print and export any subset of a drawing.
- **Hyperlinking**. Navigate through views automatically, ending searches for views or details. Click the view annotation, and the section view or detail is immediately added to your layout.
- **3D Pointer**. Identify and match geometry in multiple views. The 3D Pointer helps orient you when you check features in multiple views.
- **Animation**. Demonstrate automatically how drawing views relate to each other.

With the optional eDrawings Professional version, you have the following additional capabilities:

- **Cross Sections**. Create cross-section views with a variety of planes to fully examine a model.
- **Markup**. Markup files using clouds, text, or geometric elements. The markup elements are inserted as comments in the file.
- **Measure**. Measure the distance between two entities.
- **Move Components**. Move components in an assembly or drawing file.
- **Configurations**. Save SolidWorks configuration data and see the configurations in the eDrawings Viewer.
- **Exploded Views**. Save SolidWorks exploded view information and see the exploded views in the eDrawings Viewer.



Cross-section view of the vanity with dimensions and comments

For a lesson on the eDrawings software, see **eDrawings** in the *Online Tutorial*.

FeatureWorks

FeatureWorks is an application that recognizes features on an imported solid body in a SolidWorks part document. Recognized features are treated the same as features that you create in the SolidWorks software. You can edit the definition of recognized features to change their parameters. For features that are based on sketches, you can edit the sketches to change the geometry of the features. The FeatureWorks software is intended primarily for machined parts and sheet metal parts.

Suppose you have legacy **.step** files at your company, and you want to use them in the SolidWorks software. You can use the FeatureWorks software to recognize each feature as a SolidWorks feature. This way, you do not have to remodel the same part in the SolidWorks application.

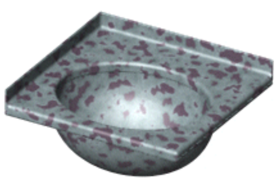
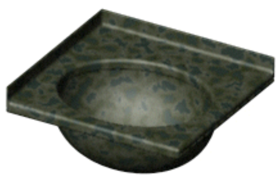
For a lesson on the FeatureWorks software, see **FeatureWorks** in the *Online Tutorial*.

PhotoWorks

PhotoWorks is a rendering application that lets you create photo-realistic images directly from SolidWorks models.

With PhotoWorks, you can specify model surface properties such as color, texture, reflectance, and transparency. PhotoWorks has a library of surface textures (metals, plastics, and so on) and, in addition, you can scan in your own surface bitmaps to use as textures, materials, scenery, and logos.

Perhaps your company catalog shows several marble countertops in different colors. The sinks are the same, except for the marble coloring. In PhotoWorks, you can render the countertops with the different marble colors, then display the images in your catalog.

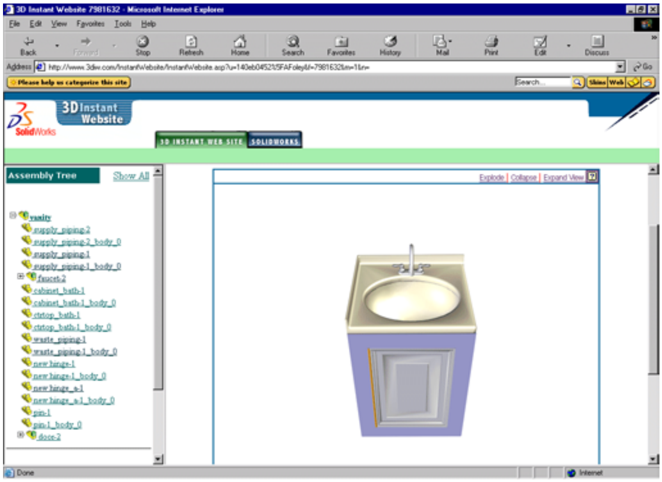



For a lesson on the PhotoWorks rendering software, see **PhotoWorks** in the *Online Tutorial*.

SolidWorks 3D Instant Website

SolidWorks 3D Instant Website allows you to create a web page from the SolidWorks software.

For example, if your company builds vanity assemblies, and your suppliers are in different parts of the country, you can build a web page for the suppliers to see your products. With just one mouse click, you can create a web page on a SolidWorks secured server, or on a local or network drive. This way, several suppliers can view your products, add comments, and so on.



The web page is based on a template and style that you can customize. The default templates that come with SolidWorks 3D Instant Website include:

- Embedded viewers for parts, assemblies, and drawings
- A comment section where multiple reviewers can offer opinions
- Your company's contact information with a link to its home page

SolidWorks Animator

With the SolidWorks Animator application, you can animate and capture SolidWorks assemblies in motion. SolidWorks Animator generates **.avi** files that you can play on any Windows-based computer. In conjunction with the PhotoWorks software, the SolidWorks Animator application can output photo-realistic animations.

Suppose that your company is at a convention with competing companies. To stand out from the competition, you can create **.avi** files that animate your products. This way, your customers can see the vanity door open and close, or see the faucet handles turn on and off. Animation helps your customers visualize models in a real-world situation.

The SolidWorks Animator application allows you to create a fly-around animation, an exploded view animation, or a collapsed view animation. Additionally, you can explicitly create motion paths for various components in your SolidWorks assembly.

For a lesson on SolidWorks Animator, see **SolidWorks Animator** in the *Online Tutorial*.

SolidWorks Explorer

SolidWorks Explorer is a file management tool designed to help you accomplish tasks such as renaming, replacing, and copying SolidWorks documents.

SolidWorks Explorer allows you to:

- View document dependencies for drawings, parts, and assemblies, in a tree structure.
- Copy, rename, or replace referenced documents. You have the option to find and update references to documents.
- View data and previews, or input data, according to the function you have active.

For example, consider that you want to rename the countertop part from **countertop.sldprt** to **countertop_with_sink.sldprt**. If you rename the part in:

- **Windows Explorer**. Any SolidWorks document that references **countertop.sldprt** (such as the vanity assembly) does not recognize that the part name changed. Therefore, the SolidWorks software cannot find the renamed part, and it does not appear in the assembly.
- **SolidWorks Explorer**. The SolidWorks software recognizes that you renamed the part. Any document that references the part updates accordingly with the new name.

SolidWorks Toolbox

SolidWorks Toolbox includes a library of standard parts that are integrated with the SolidWorks software. Select the standard and the type of part you want to insert, then drag the component into your assembly.

For example, when you attach the hinge to the vanity cabinet, or when you fasten the waste pipe to the sink, you can use the standard screws and washers included in SolidWorks Toolbox. This way, you do not have to make additional parts to complete the vanity assembly.

You can customize the SolidWorks Toolbox library of parts to include your company's standards, or to include those parts that you refer to most frequently. You can also make a copy of the SolidWorks Toolbox parts, then edit them as needed.

Solidworks Toolbox supports several international standards including ANSI, BSI, CISC, DIN, ISO, and JIS.

Additionally, SolidWorks Toolbox has several engineering tools:

- **Beam Calculator**. Performs deflection and stress calculations on structural steel cross sections.
- **Bearing Calculator**. Performs bearing calculations to determine capacity ratings and basic life values.
- **Cams**. Creates cams with fully-defined motion paths and follower types. The cam can be either circular or linear with 14 motion types from which to choose. You can also set how the track for the follower is cut, either as a blind cut or a cut through the entire cam.
- **Grooves**. Creates industry standard O-Ring and Retaining ring grooves to your cylindrical model.
- **Structural Steel**. Brings the cross-section sketch of a structural steel beam into a part. The sketch is fully-dimensioned to match industry standard sizes. You can extrude the sketch in the SolidWorks software to create the beam.

For a lesson on SolidWorks Toolbox, see **SolidWorks Toolbox** in the *Online Tutorial*.

SolidWorks Utilities

SolidWorks Utilities is a set of tools that allows you to examine and edit individual parts, and compare the features and solid geometry of pairs of parts.

For example, if you and a co-worker design two similar types of faucet handles, you can use the **Compare Features** utility to compare the parts. This utility identifies the unique features of each part so you can collaborate and decide on the best design methods. Then, you can identify the most efficient designs and combine them in one model.

SolidWorks Utilities includes the following tools:

- **Compare Features**. Compares features of two parts, and finds identical, modified, and unique features.
- **Compare Geometry**. Compares two parts to find their common volumes.
- **Geometry Analysis**. Finds small faces, short edges, sliver faces, and so on.
- **Find/Modify/Suppress**. Finds features of a specific size or any other characteristic you specify, and edits them in a batch mode.
- **Power Select**. Selects entities (edges, loops, faces, or features) in a part that meet the criteria that you define.
- **Feature Paint**. Copies feature parameters (such as depth and size) from one feature to others that you select.

For a lesson on SolidWorks Utilities, see **SolidWorks Utilities** in the *Online Tutorial*.

Index