CHAPTER

1

Introducing SolidWorks

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In SolidWorks, you build 3D parts from a series of simple 2D sketches and features such as extrude, revolve, fillets, cuts, and holes, among others. You can put the parts together into assemblies. You can then create 2D drawings from the 3D parts and assemblies.

This chapter will familiarize you with some of the tools available to make the transition to SolidWorks and with some of the basic facts and concepts that you need to know to get the most out of the software.

VIDEO

You can view videos for each chapter on the Wiley website, in addition to downloading sample files. View the Chapter 1 Introduction video to get started.

Installing SolidWorks for the First Time

Some of you will have SolidWorks installed for you by people in your company or by SolidWorks reseller experts, and some of you will do the installation on your own. Regardless, it is best to make sure that your hardware and software are compatible with the SolidWorks system requirements, available on the SolidWorks website at www.solidworks.com/sw/support/SystemRequirements.html.

SolidWorks installs natively on both 32- and 64-bit operating systems. It is supported only for Windows Vista and Windows 7. SolidWorks 2013 does not install on Windows XP. The 64-bit operating system is recommended. In all cases, the professional-level OS is recommended. Although it is possible to install and run SolidWorks under Parallels and Boot Camp on Apple hardware, that configuration is not supported or tested by SolidWorks Corporation or its resellers.

You can find video card requirements at the above link for system requirements. The main concern with a video card for SolidWorks is that it must be compatible with OpenGL, and be SolidWorks certified. Hardware changes too rapidly for me to give specific recommendations here, but generally nVidia brand boards in the Quadro line are acceptable, as are AMD/ATI brand boards in the FirePro line. You should expect to pay $100 to $500 for a serviceable low- to mid-range video card. Cards that are marketed as game cards, such as the Radeon or GeForce, have known limitations and do not work well with SolidWorks.

In addition to installing the correct hardware, you also must have a compatible driver version installed. Again, refer to the SolidWorks system requirements website.

TIP

Whether you are installing SolidWorks or someone is installing it for you, you should consider purchasing a copy of the *SolidWorks Administration Bible* (Wiley, 2010), which contains all the information you need on installing, configuring, and troubleshooting SolidWorks.

Alternatively, you might not want to get that involved at first. You can install the software with all defaults just to get started, using it as a practice installation, especially if you intend to learn as much as you can about it, and then come back and do a more thorough job of implementing it later. If this is the case, just put the DVD in the drive (or use the automatic download installers) and accept all the defaults.

You should count on the installation requiring about 6GB of space on your hard drive, depending on the options you select to install. The locations of files on your computer will vary by SolidWorks version, by your operating system, and by your own installation choices, but the bulk of the files are placed into two separate folders: the Program Files folder (`C:\Program Files\SolidWorks Corp\SolidWorks`) and the Toolbox Data folder (`C:\SolidWorks Data`).

When installing any software, be sure to exit out of all other software first, turn off antivirus software, make sure you have enough hard drive space and otherwise meet the system requirements outlined on the SolidWorks System Requirements web page, and reboot when the installation is complete.

You also can install without a DVD, using downloaded data instead. To do this, you need a subscription account for the Customer Portal area of the SolidWorks website at www.solidworks.com/sw/support/Subscription%20Services.html.

Trial installations, network licenses, student versions, administrative images, and other special cases may require that you contact your reseller for technical support.

Starting SolidWorks for the First Time

SolidWorks has many tools for beginning users that are available when the software is installed. A default installation presents you with several options when the software is started the first time. This section includes a description of these options and how you can most benefit from them. If you plan to go to formal SolidWorks reseller-based training classes, be sure to go through some of the tutorials mentioned in this section first, so you are prepared to ask educated questions and have a leg up on the rest of the class. You will get more out of the training with the instructor if you have seen the material once before.

NEW FEATURE

While SolidWorks opens, it displays a splash screen, which features a Cancel button. SolidWorks 2013 enables you to cancel out of the software opening before it actually opens. SolidWorks 2013 also starts some components when you start your computer to make startup of the software faster. If you prefer to disable these components from loading when you start your computer, go to the Windows Start menu ➪ All Programs ➪ Startup, right-click SolidWorks Fast Start, and select Delete.

Examining the SolidWorks license agreement

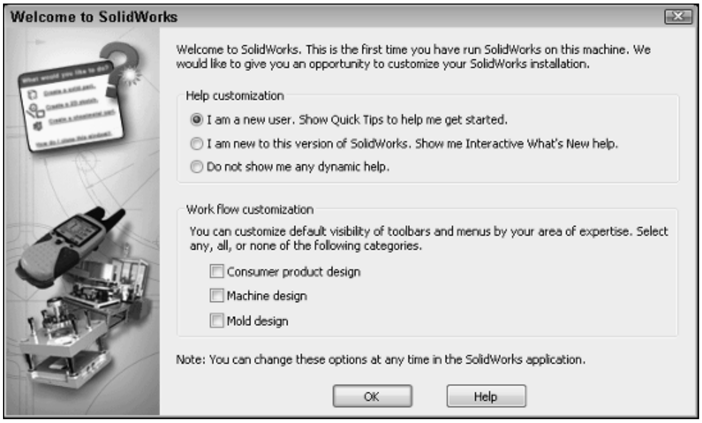
Becoming familiar with what the license agreement says can be useful, but the agreement does not have any bearing on learning how to use the software other than the fact that it allows for a Home Use License. Many users find this part of the license agreement helpful. The primary user of the license at work is also allowed to use the license at home or on a portable computer. This is often a good option for learning techniques, doing additional practice, or completing the design of that deck or soapbox derby car. If your employer uses floating licenses or you are outside of North America, the rules are somewhat different. Contact your reseller for details.

Viewing the Welcome to SolidWorks screen

The Welcome to SolidWorks screen, shown in Figure 1.1, is the next thing to greet you. This helps you establish what type of tools you would like to see in the interface and gives you some help options. You may not get the chance to see this dialog box if someone else, such as an IT person, has installed and done an initial test on your software for you.

FIGURE 1.1

The Welcome to SolidWorks screen



If you make a choice that you would like to change later, the options presented in this dialog box also are available elsewhere. Although you will not see this dialog box again, the interface is highly customizable and options exist for most things you might want to change. Chapter 2 covers interface customization in more detail.

Using Quick Tips

Quick Tips become available only when a document window is active. The Quick Tips setting enables balloons with tips to help you get started with several tasks.

You can turn Quick Tips on or off by clicking the small square on the status bar near the lower-right corner, as shown in Figure 1.2. You can turn the status bar on or off in the View menu; however, the status bar serves many useful purposes for all users, so I recommend you leave it active. You also can turn Quick Tips off in the Help menu by deselecting Quick Tips. The on/off setting is document-type sensitive, so if you deselected Quick Tips off in part mode, you need to do it again for assemblies and drawings as well. Quick Tips are a great way to get going or to get a little refresher if it has been a while since you last saw the software.

Using Interactive What's New

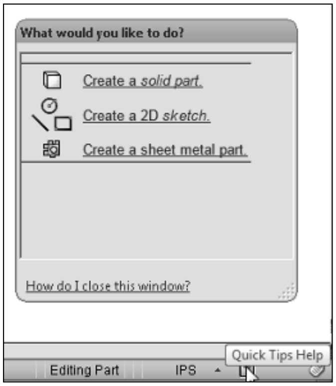
Interactive What's New help is another option that you can change later from the Help menu (Help ➪ What's New ➪ Interactive). This adds a question mark with an asterisk symbol next to menu items that are new and have special Help file entries.

Customizing Workflow

The Workflow Customization options in the Welcome to SolidWorks dialog box activate certain toolbars in the interface by default. It does not customize the toolbars at all. All these options can be changed later in a more complete interface customization.

FIGURE 1.2

Turning Quick Tips on or off



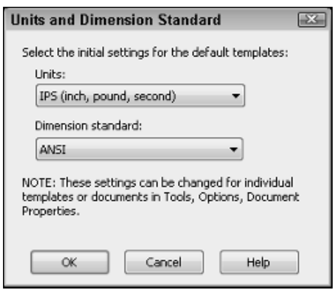
Creating a new document

To start a new SolidWorks document, click the New icon in the title bar of the SolidWorks application. With standard functions such as creating a new document, SolidWorks works just like a Microsoft Office application, and the icons even look the same.

The first time you create a document, SolidWorks prompts you to select units for your default templates, as shown in Figure 1.3. This is an important step, although you can make changes later if needed. SolidWorks stores most of the document-specific settings in document templates for each type of document—parts, assemblies, and drawings. More information on part and assembly templates can be found later in this chapter.

FIGURE 1.3

The default template units selection



One of the most common questions new users ask is how they can change the default so new documents come up with a certain type of units every time. Units in new documents are set within the templates as a part of the drafting standards. To create a part with inch units, use a template with inch units. You can have as many templates as you want, with a different template for each type of units you might use.

ISO (International Organization for Standardization) and ANSI (American National Standards Institute) drafting standards use different methods of displaying dimensions, tolerances, drawing views, symbols, and projecting views, and these standards are controlled by templates. ISO is typically a European standard and uses first angle projection, while ANSI is an American standard and uses third angle projection. The standard used throughout this book is ANSI, except where noted.

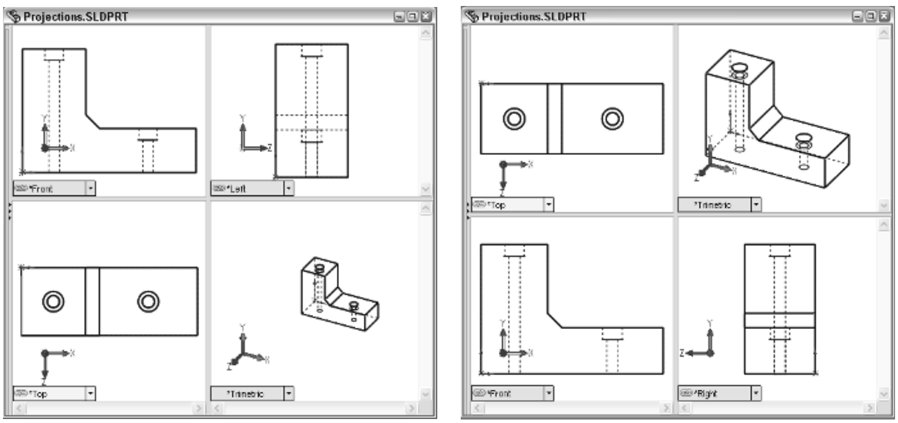
The difference between third and first angle projections can cause parts to be manufactured incorrectly. Figure 1.4 demonstrates the difference between the two projection types.

Third angle projections are created as if the observer moves to the named side of the part, and the named view is placed in the named orientation with respect to the Front view. So the Right view is created by looking at the part from the right side, and then placing the Right view to the right of the Front view.

First angle projections are the same as third angle except that the Right view is placed on the left of the Front view; the Top view is placed on the bottom. The effect is that if you are reading a first angle drawing but expecting a third angle drawing, the views appear to be incorrect.

FIGURE 1.4

Differences between first (left) and third (right) angle projections



First angle projection   Third angle projection

For more information on first and third angle projections, refer to this Wikipedia article: http://en.wikipedia.org/wiki/Multiview_orthographic_projection.

Be sure to get the option correct. If someone else, such as a computer specialist who is not familiar with mechanical drafting standards, initially sets up SolidWorks on your computer, verify that the default templates use the correct standards, units, and projection method.

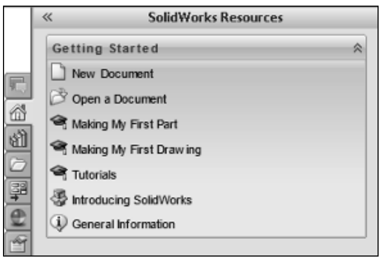
Another setting affecting projections that you will want to check can be accessed by choosing Tools ➪ Options ➪ Display/Selection ➪ Projection type for four-view viewport. This does not follow the drafting standard selected for the default templates or the country in which the software is installed.

Exploring SolidWorks documentation

Several types of documentation are available to SolidWorks users. A great place to start is the SolidWorks Resources tab in the Task Pane (on the right side of the screen). This is the tab with the Home icon. The Getting Started option in the SolidWorks Resources tab is shown in Figure 1.5.

FIGURE 1.5

The Getting Started panel on the SolidWorks Resources tab of the Task Manager



SolidWorks has two separate Help files. One is stored on the web (and is unavailable during beta testing, when this book is being written), and the other is installed on your computer. The reason for this, according to SolidWorks, is that it is easier to update the files on the web. The option to select which you want to use is at Help ➪ Use SolidWorks Web Help.

Accessing tutorials

You can access several tutorials by selecting the SolidWorks Tutorials option from the Help menu or from the SolidWorks Resources tab in the Task Manager. There, you will find a list of tutorials on subjects from sheet metal to macros in parts, assemblies, and drawings. These tutorials are certainly worth your time and will build your skills and knowledge of basic functionality. This *SolidWorks Bible* distinguishes itself from the manufacturer's tutorials by going into far more detail and depth about each function, adding information such as best practices, performance considerations, and cautionary data, acting as a thorough desk reference. The purpose of this book is not to duplicate all the resources for beginners, but to take the information into far more depth and detail and answer the "why" questions instead of just the "how" questions.

Keeping up with what's new

With every release, SolidWorks publishes a What's New document to help you keep up to speed with the changes. This is typically a PDF file with accompanying example files. If you have missed a version or two, reading through the What's New files can help get you back on track. (You can find every What's New document on Ricky Jordan's blog at http://rickyjordan.com.)

Moving from 2D to 3D

The Help menu contains a selection called Moving from 2D to 3D. It is intended to help transitioning users acclimate to their new surroundings. Terminology is a big part of the switch and figures prominently in the Moving from 2D to 3D help file. Likely, the most helpful sections in Moving from 2D to 3D are "Approach to Modeling" and "Imported AutoCAD Data."

Checking out the Tip of the Day

The SolidWorks Tip of the Day, when activated, is displayed at the bottom of the SolidWorks Resources tab in the Task Pane. Cycling through a few of the tips or using them to quiz coworkers can be a useful skills-building exercise.

Identifying SolidWorks Documents

SolidWorks has three main data type files: parts, assemblies, and drawings; however, there are additional supporting types that you may want to be familiar with if you are concerned with customization and creating implementation standards. Table 1.1 outlines the document types.

TABLE 1.1 Document Types

Design Documents	Description
.sldasm	SolidWorks assembly file type
.slddrw	SolidWorks drawing file type
.sldprt	SolidWorks part file type

Templates and Formats	Description
.asmdot	Assembly template
.asmprp	Assembly custom properties tab template
.drwdot	Drawing template
.drwprp	Drawing custom properties tab template
journal.doc	Design journal template
.prtdot	Part template
.prtprp	Part custom properties tab template
.sldbombt	BOM template (table-based)
.sldtbt	General table template
.slddrt	Drawing sheet format
.sldholtbt	Hole table template
.sldrevtbt	Revision table template
.sldwldtbt	Weldment cutlist template
.xls	BOM template (Excel-based)
Library Files	**Description**
.sldblk	Blocks
.sldlfp	Library part file
Styles	**Description**
.sldgtolfvt	Geometric tolerance style
.sldsffvt	Surface finish style
.sldweldfvt	Weld style
Symbol Files	**Description**
gtol.sym	A symbol file that enables you to create custom symbols
swlines.lin	A line style definition file that enables you to create new line styles
Others	**Description**
.btl	Sheet metal bend table
calloutformat.txt	Hole callout format file
.sldclr	Color palette file
.sldreg	SolidWorks settings file
.sldmat	Material database
.sldstd	Drafting standard
.swb, swp	Macros, macro features
.txt	Custom property file, sheet metal bend line note file
.xls	Sheet metal gauge table

Saving your setup

If you need to reinstall SolidWorks, move to another computer, or duplicate the setup for another user, you need to copy out the files you have used or customized. By default, all these files are located in different folders within the SolidWorks installation directory. Chapter 2 deals with interface settings and creating a registry settings file to copy to other computers or use as a backup.

Best Practice

It is especially important to have copies of any customized template or library files in a location other than the default installation folder when you are doing complex implementations that include templates of various types of tables or customized symbol files. Uninstalling SolidWorks or installing a new version will wipe out all your hard work. Choose Tools ➪ Options ➪ File Location to establish separate library folders on the local hard drive or on a network location.

Using templates

I have included some of my part and assembly templates with the download materials for this book. After you have downloaded the Zip files for each chapter, extract and copy them to the folder specified at Tools ➪ Options ➪ File Locations ➪ Document Templates.

On the Website

Wiley has established a website for the material in this book. You can download files for each chapter from www.wiley.com/go/solidworks2013.

When you begin to create a new document, and the New SolidWorks Document dialog box gives you the option to select one of several files to start from, those files are templates. Think of templates as "start parts" that contain all the document-specific settings for a part (Tools ➪ Options ➪ Document Properties). The same concept applies to assemblies and drawings. Templates generally do not have any geometry in them (although it is possible).

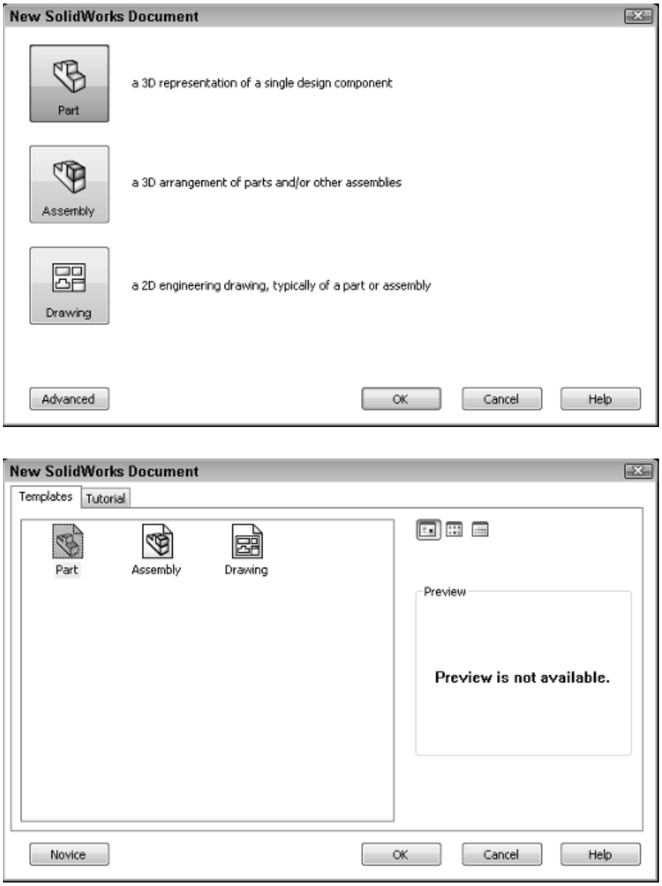
Tip

The Novice interface for the New SolidWorks Document dialog box (File ➪ New SolidWorks Document) enables you to select default templates only. The Advanced interface enables you to select any available template.

As shown in Figure 1.6, several tabs can be displayed on the Advanced interface. Each of these tabs results from creating a folder in the template directory specified in the Options dialog box (Tool ➪ Options). To switch from the Novice interface to the Advanced interface, click the Advanced button. To switch from Advanced to Novice, click the Novice button.

FIGURE 1.6

The Novice and Advanced interfaces for the New SolidWorks Document dialog box



Having multiple document templates available

Having multiple templates available gives you many options when starting a new document. This offers an advantage in many situations, including the following:

- Standardization for a large number of users
- Working in various units
- Preset materials
- Preset custom properties
- Parts with special requirements, such as sheet metal or weldments
- Parts and assemblies with standardized background colors

- Drawings of various sizes with formats (borders) already applied
- Drawings with special notes already on the sheet

Drawing templates and formats are complex enough that I devote a chapter to them. Chapter 24 discusses the differences between drawing templates and formats and how to use them to your advantage. This chapter addresses part and assembly templates.

Depending on your needs, it might be reasonable to have templates for metric parts and assemblies and Imperial parts and assemblies, templates for steel and aluminum, and templates for sheet metal parts and weldments, if you design these types of parts. If your firm has different customers with different requirements, you might consider using separate templates for each customer. Over time, you will discover the types of templates you need, because you will find yourself making the same changes repeatedly.

To create a template, open a document of the appropriate type (part, assembly, or drawing), and make the settings you want the template to have; for example, units are one of the most common reasons to make a separate template, although any Document Property setting is fair game, from the dimensioning standard used to the image quality settings. You can find these settings through the menus at Tools➪Options➪Document Properties.

Some document-specific settings do not appear in the Document Properties dialog box, such as the names of standard planes or the use of axes as reference geometry. Still, these settings are saved with the template. Settings that fall into this category are the View menu entity type visibility option and the Tools➪Sketch Settings menu options.

Custom Properties are another piece of the template puzzle. If you use or plan to use BOMs (Bills of Materials), PDM (Product Data Management), or linked notes on drawings, you need to take advantage of the automation options available with custom properties.

Locating templates

You can set the location of the templates folder at Tools➪Options➪File Locations➪Document Templates. The folder location may be a local folder or a shared network folder. Multiple folders may be specified in the list box, each corresponding to a tab in the New Document's Advanced interface.

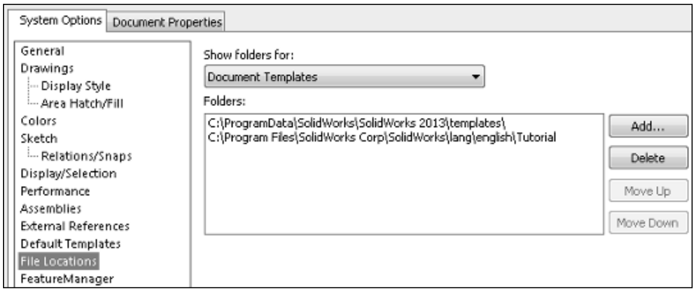
After all the Document Properties, custom properties, and other settings are set to your liking and you are ready to save the file as a template, choose File➪Save As and select Part Templates in Files of Type. SolidWorks prompts you to save the template in the first folder listed in the File Locations list. You can create assembly templates the same way, except that you change the settings for an assembly document.

You also can create additional tabs on the New SolidWorks Document dialog box by making subfolders in the main folder in the File Locations area, as shown in Figure 1.7.

Adding subfolders to either of the locations listed in File Locations results in additional tabs in the New SolidWorks Document dialog box.

FIGURE 1.7

The Tools➪Options➪File Locations list

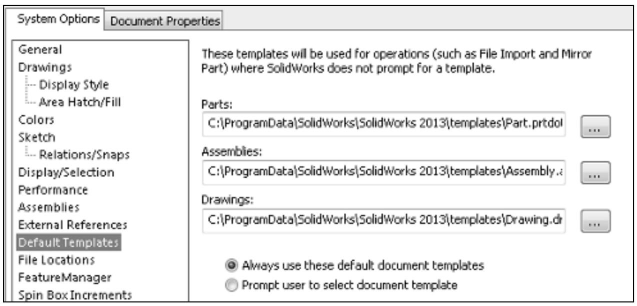


Using default templates

Default templates are established at Tools➪Options➪Default Templates. The default templates must be in one of the paths specified in File Locations. Figure 1.8 shows the Default Templates settings.

FIGURE 1.8

The Tools➪Options➪Default Templates settings



Two Default Template options are available: Always use these default document templates and Prompt user to select document template. The Default Template options apply to situations when a template is required by an automatic feature in the software, such as an imported part or a mirrored part. In this situation, depending on the option selected, the system automatically uses the default template or the user is prompted to select a template.

PERFORMANCE

Allowing the software to apply the default template automatically can have a great impact on speed. This is especially true in the case of imported assemblies, which would require you to select templates manually for each imported part in the assembly if the Prompt user to select document template option is selected.

Sharing templates

If you are administering an installation of a large number of users, or even if just a couple of users are working on similar designs, shared templates are necessary. If every user does what she thinks is best, you may get an adverse combination of conflicting ideas, and the consistency of the company's documentation may suffer. Standardized templates cannot make users model, assemble, and detail in exactly the same way, but they do help users start on the same foot.

To share templates among several users, create a folder for templates on a commonly accessible network location, preferably with read-only access for users and read-write permissions for administrators. Then point each user's File Locations and Default templates to that location. Access problems due to multiple users accessing the same files do not arise in this situation because users copy templates to create new documents and do not use them directly.

Caution

One of the downfalls of this arrangement is that if the network goes down, users no longer have access to their templates. This can be averted by also putting copies of the templates on the local computers; however, this has the tendency to undermine the goal of consistent documentation. Users may tend to use and customize the local templates rather than use the standardized network copies.

CAD administration and organizing any group of people on some level always comes down to training and then trusting employees to do the right thing. There is no way to completely secure any system.

Understanding Feature-Based Modeling

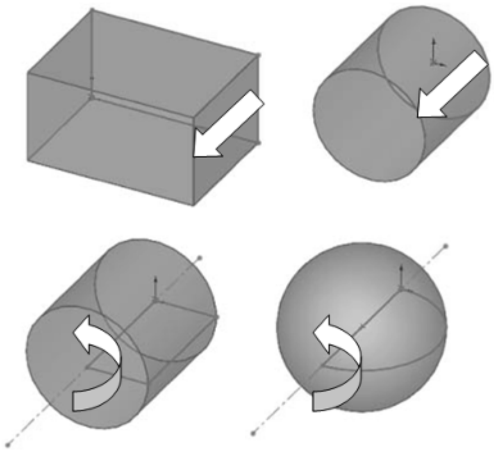
You need to be familiar with some terminology before diving into building models with SolidWorks. Notice that I talk about "modeling" rather than "drawing" or even "design." This is because SolidWorks is virtual prototyping software. Whether you are building an assembly line for automotive parts or designing decorative perfume bottles, SolidWorks can help you visualize your geometrical production data in the most realistic way possible without actually having it in your hand. This is more akin to making a physical model in the shop than drawing on paper.

"Feature-based" modeling means that you build the model by creating 2D sketches and applying processes (features) to create the 3D shape. For example, you can create a simple box by using the Extrude process, and you can create a sphere using the Revolve process. However, you can make a cylinder using either process, by revolving a rectangle or extruding a circle. You start by visualizing the 3D shape, and then you apply a 3D process to a 2D sketch to create that shape. This concept on its own is half of what you need to know to create models with SolidWorks.

Figure 1.9 shows images of simple feature types along with the 2D sketches from which they were created.

FIGURE 1.9

Simple extruded and revolved features



Many different feature types in SolidWorks enable you to create everything from the simplest geometry shown in Figure 1.10 to more complex shapes. In general, when I talk about modeling in this book, I am talking about *solid* modeling, although SolidWorks also has a complete complement of surfacing tools. I discuss the distinction between solid and surface modeling in Chapter 33.

Table 1.2 lists some of the most common features in SolidWorks and classifies them according to whether they always require a sketch, a sketch is optional, or they never require a sketch. As an example of a sketch optional feature, a sweep can use a model edge as a sweep path.

TABLE 1.2 **Feature Types**

Sketch Required	Sketch Optional	No Sketch (Applied Features)
Extrude	Loft	Fillet
Revolve	Sweep	Chamfer
Rib	Dome	Draft
Hole Wizard	Boundary	Shell
Wrap	Deform	Flex

In addition to these features, other types of features create reference geometry, such as curves, planes, axes, and surface features (Chapter 32); specialty features for techniques like sheet metal (Chapter 34); and plastics/mold tools (Chapters 38 and 39).

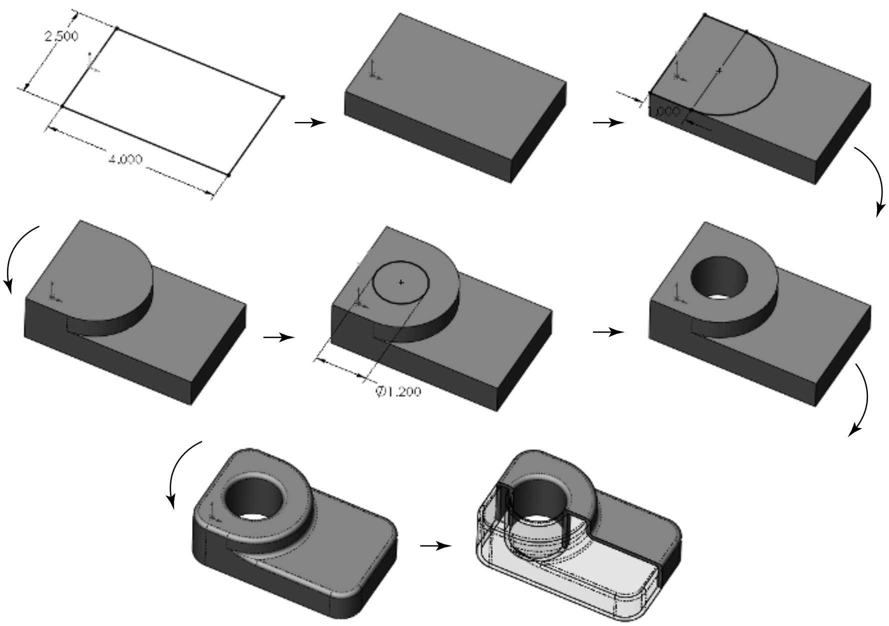
Understanding History-Based Modeling

In addition to being feature-based, SolidWorks is also *history*-based. The process history is shown in a panel to the left side of the SolidWorks window called the *FeatureManager*. The FeatureManager keeps a list of the features in the order in which you have added them. It also enables you to reorder items in the tree (in effect, to change history). Because of this, the order in which you perform operations is important. For example, consider Figure 1.10. This model was created by the following process, left to right starting with the top row:

1. **Create a sketch.**
2. **Extrude the sketch.**
3. **Create a second sketch.**
4. **Extrude the second sketch.**
5. **Create a third sketch.**
6. **Extrude Cut the third sketch.**
7. **Apply fillets.**
8. **Shell the model.**

FIGURE 1.10

Features used to create a simple part



If the order of operations used in the previous part were slightly reordered (by putting the shell and fillet features before Step 6), the resulting part would look slightly different, as shown in Figure 1.11. You can find this part in the download materials for this chapter.

FIGURE 1.11

Using a different order of features for the same part

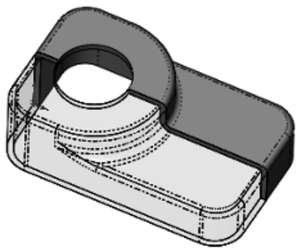
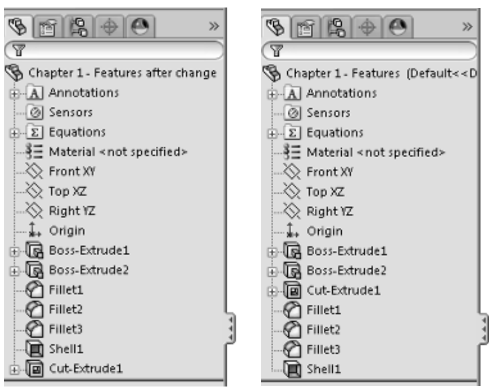


Figure 1.12 shows a comparison of the FeatureManager design trees for the two different feature orders. You can reorder features by dragging them up or down the tree. Relationships between features can prevent reordering; for example, the fillets are dependent on the second extruded feature and cannot be reordered before it. This is referred to as a *Parent/ Child relationship*.

Reordering and Parent/Child relationships are discussed in more detail in Chapter 12.

FIGURE 1.12

Compare the FeatureManager design trees for the parts shown in Figure 1.10 and Figure 1.11.



The order of operations, or *history*, is important to the final state of the part. For example, if you change the order so that the shell comes before the extruded cut, the geometry of the model changes, removing the sleeve inside instead of the hole on top. You can try this for yourself by opening the part indicated previously, dragging the Shell1 feature in the FeatureManager and dropping it just above the Cut-Extrude1 feature.

NOTE
You can drag only one item at a time in the FeatureManager. Therefore, you may drag the shell and then drag each of two fillets, or you could just drag the cut feature down the tree. Alternatively, you can put the shell and fillets in a folder and drag the folder to a new location. Reordering is limited by parent/child relationships between dependent features.

You can read more about reordering folders in Chapter 12.

In some cases, reordering the features in the FeatureManager may result in geometry that might not make any sense; for example, if the fillets are applied after the shell, they might break through to the inside of the part. In these cases, SolidWorks gives an error that helps you to fix the problem.

In 2D CAD programs where you are just drawing lines, the order in which you draw the lines does not matter. This is one of the fundamental differences between history-based modeling and drawing.

Features are really just like steps in building a part; the steps can either add material or remove it. However, when you make a part on a mill or lathe, you are only removing material. Some people choose to model following manufacturing methods, so they start from a piece of stock and apply features that remove material. This approach works best for machining, but doesn't work well for molding, casting, sheet metal, or progressive dies. The FeatureManager is like an instruction sheet to build the part. When you reorder and revise history, you change the order of operations and thus the final result. Some people look at the FeatureManager as a recipe for cooking a dish.

Sketching with Parametrics

Sketching is the foundation that underlies the most common feature types. You will find that sketching in parametric software is vastly different from drawing lines in 2D CAD.

Dictionary.com defines the word *parameter* as "one of a set of measurable factors . . . that define a system and determine its behavior and [that] are varied in an experiment." SolidWorks sketches are parametric. What this means is that you can create sketches that change according to certain rules and maintain relationships through those changes. This is the basis of parametric design. It extends beyond sketching to all the types of geometry that you can create in SolidWorks. Creating sketches and features with intelligence is the basis of the concept of Design Intent, which I cover in more detail later in this chapter.

In addition to 2D sketching, SolidWorks also makes 3D sketching possible. Of the two methods, 2D sketches are by far more widely used. You create 2D sketches on a selected plane or planar face and then use them to establish shapes for features such as Extrude, Revolve, and others. Relations in 2D sketches often are created between sketch entities and other model edges that may or may not be in the sketch plane. In situations where other entities are not in the sketch plane, the out-of-plane entity is projected into the sketch plane in a direction that is normal to the sketch plane. This does not happen for 3D sketches.

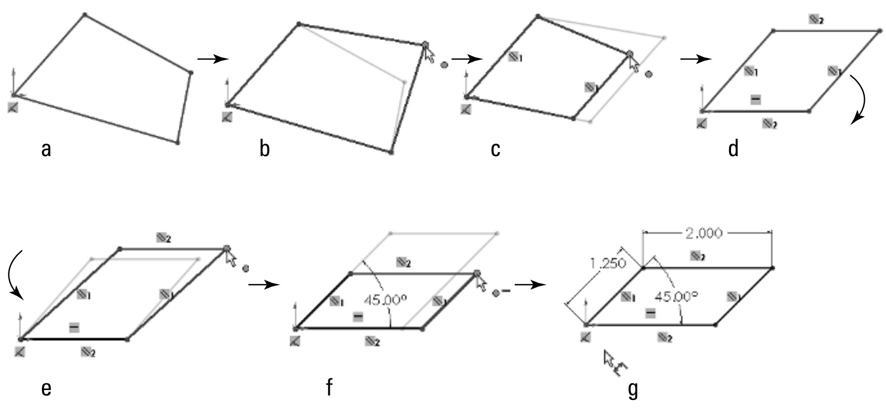
You can use 3D sketches for the Hole Wizard, routing, weldments, and complex shape creation, among other applications.

For more information on 3D sketching, refer to Chapter 6.

For a simple example of working with sketch relations in a 2D sketch, consider the sketch shown in Figure 1.13a. The only relationships among the four lines are that they form a closed loop that is touching end to end and one of the corners is coincident to the part origin. The small square icon near the origin shows the symbol for a *coincident* sketch relation. These sketch relations are persistent through changes and enable you to dynamically move sketch elements with the cursor on the screen. The setting to enable or disable displaying the sketch relation symbols is found at View ➪ Sketch Relations.

FIGURE 1.13

A sketch of four lines



If you drag any of the unconstrained corners (except for the corner that is coincident to the origin), the two neighboring lines follow the dragged endpoint, as shown in Figure 1.13b. Notice the ghosted image left by the original position of the sketch. This is helpful when experimenting with changes to the sketch because you can see both the new and the old states of the sketch. The setting to enable or disable this ghosted position is found at Tools ➪ Options ➪ Sketch ➪ Ghost Image on Drag.

If you add a parallel relation between opposing lines, they now act differently, as shown in Figure 1.13c. You add a parallel relation by selecting the two lines to make parallel and selecting Parallel from the PropertyManager panel. You also can select the Parallel relation from the context bar that pops up in the graphics window when you have both lines selected.

You can read more about the PropertyManager in Chapter 2.

Next, add a second parallel and a horizontal relation, as shown in Figure 1.13d. If you are following along by re-creating the sketch on your computer, you will notice that one line has turned from blue to black.

The line colors represent sketch states. It may be impossible to see this in the black and white printing of this book, but if you are following along on your own computer, you can now see one black line and three blue lines. Sketch states include Underdefined, Overdefined, Fully Defined, Unsolvable, Zero Length, and Dangling, and are described as follows:

- **Blue: Underdefined.** The sketch entity is not completely defined. You can drag a portion of it to change size, position, or orientation.
- **Black: Fully Defined.** The sketch entity is fully defined by a combination of sketch relations and dimensions. A sketch cannot be fully defined without being connected in some way to something external to the sketch, such as the part origin or an edge. (The exception to this rule is the use of the Fix constraint, which, although effective, is not a recommended practice.)
- **Red: Overdefined—Not Solved.** When a sketch entity has two or more relations and one of them cannot be satisfied, the unsatisfied relation will be red. For example, if a line has both Horizontal and Vertical relations, and the line is actually vertical, the Vertical relation will be yellow (because it is conflicting but satisfied), and the Horizontal will be red (because it is conflicting and not satisfied).
- **Yellow: Overdefined—Conflicts.** Solving the sketch relations would result in a zero-length entity; for example, this can occur where an arc is tangent to a line, and the centerpoint of the arc is also coincident to the line.
- **Brown: Dangling.** The relation has lost track of the entity to which it was connected.

Entities with different states can exist within a single sketch. In addition, endpoints of lines can have a different state than the rest of the sketched entity. For example, a line that is sketched horizontally from the origin has a *coincident* at one endpoint to the origin, and the line itself is *horizontal*. As a result, the line and first endpoint are black, but the other endpoint is underdefined because the length of the line is not defined. Sketch states are indicated in the lower-right corner of the graphics window and in the status bar. You can see that dragging one corner allows only the lines to move in certain ways, as shown in Figure 1.13e.

In addition to sketch relations, dimensions applied using the Smart Dimension tool are also part of the parametric scheme. If you apply an angle dimension (by clicking the two angled lines with the Smart Dimension tool) about the origin and try dragging again, as shown in Figure 1.13f, you see that the only aspect that is not locked down is the length of the sides. Notice also that when the angle dimension is added, another line turns black.

Finally, adding length dimensions for the unequal sides completes the definition of the sketch, as shown in Figure 1.13g. At this point, all lines have turned black. This state is called "fully defined." Between the dimensions and sketch relations, there is enough information to re-create this sketch exactly.

Best Practice

A best practice is to fully define all sketches. However, sometimes this is not practical. When you create freeform shapes, generally by using splines, these shapes cannot easily be fully defined, and even if they are fully defined, the extra dimensions are usually meaningless, because it is impractical to dimension splines on manufacturing drawings.

Parametric relations within a sketch control how the sketch reacts to changes from dimensions or relations within the sketch or by some other factor from outside the sketch. Other factors can drive the sketch as well, such as equations, other model geometry that is external to the sketch, and even geometry from another part in an assembly, as you'll see later.

Understanding Design Intent

Design intent is a phrase that you will hear often among SolidWorks users. I like to think of it as "design for change." Design intent means that when you put the parametric sketch relations together with the feature intelligence, you can build models that react to change in predictable ways. This gives you a great deal of control over changes.

An example of design intent could be a statement that describes general aspects that help define the design of a part, such as "This part is symmetrical, with holes that line up with Part A and thick enough to be flush with Part B." From this description, and the surrounding parts, it is possible to re-create the part in such a way that if Part A or Part B changes, the part being described updates to match.

Some types of changes can cause features to fail or sketch relations to conflict. In most situations, SolidWorks has ample tools for troubleshooting and editing that you can use to repair or change the model. In these situations, it is often the design intent itself that is changing.

Best Practice

When editing or repairing relations, a best practice is to edit rather than delete. Deleting often causes additional problems farther down the tree. Many users find it tempting to delete anything that has an error on it.

Editing Design Intent

One of the most prominent aspects of design in general is change. I have often heard it said that you may design something once, but you will change it a dozen times. This concept carries over into solid modeling work. Design intent is sometimes thought of as a static concept that controls changing geometry. However, design intent itself often changes, thus requiring the way in which the model reacts to geometric changes to also change. Fortunately, SolidWorks has many tools to help you deal with changing requirements.

Choosing sketch relations

Seeing the sketch relation symbols is the best tool for visualizing design intent. You can show or hide icons that represent the relations by choosing View ➪ Sketch Relations. When shown, these relations appear as an icon in a small colored box in the graphics area next to the sketch entity. Clicking the icon highlights the sketch elements involved in that relation. Refer to Figure 1.14 for examples of these relations.

TIP

The View Sketch Relations option is an excellent candidate for use with a hotkey, thus enabling you to easily toggle it on and off.

For more information on creating and managing hotkeys, see Chapter 2.

You can use the sketch relation icons on the screen to delete relations by selecting the icon in the graphics area and pressing Delete on the keyboard. You also can use them to quickly determine the status of sketch relations by referring to the colors defined earlier.

Selecting display/delete relations

The Display/Delete Relations tool enables you to list, sort, delete, and repair sketch relations. You can find the Display/Delete Relations tool on the Sketch toolbar. The sketch status colors defined earlier also apply here, with the relations appearing in the appropriate color. (Relations are not shown in blue or black, only the colors that show errors, such as red, yellow, and brown.) This tool also enables you to group relations by several categories:

- All in This Sketch
- Dangling
- Overdefining/Not Solved
- External
- Defined in Context
- Locked

- Broken
- Selected Entities

In the lower Entities panel of the Display/Delete Relations PropertyManager, shown in Figure 1.14, you can replace one entity with another or repair dangling relations.

FIGURE 1.14

The Display/Delete Relations PropertyManager enables you to repair broken relations.

You can read more about repairing dangling entities in Chapter 6.

Using suppressed sketch relations

Suppressing a sketch relation means that the relation is turned off and not used to compute the position of sketch entities. Suppressed relations generally are used in conjunction with configurations.

Configurations are discussed in detail in Chapter 11.

Working with Associativity

In SolidWorks, associativity refers to links between documents, such as a part that has an associative link to a drawing. If the part changes, the drawing updates as well. Bidirectional associativity means that the part can be changed from the part or the drawing document window. One of the implications of this is that you do not edit a SolidWorks drawing by simply moving lines on the drawing; you must change the model, which causes all drawing views of the part or assembly to update correctly.

Other associative links include using *inserted* parts (also called *base* or *derived* parts), where one part is inserted as the first feature in another part. This might be the case when you build a casting. If the part is designed in its "as cast" state, it is then inserted into another part where machining operations are performed by cut features and the part is transformed into its "as machined" state. This technique also is used for plastic parts where a single shape spans multiple plastic pieces. A "master part" is created and split into multiple parts. An example would be a mouse cover and buttons.

One of the most important aspects of associativity is file management. Associated files stay connected by filenames. If a document name is changed and one of the associated files is not updated appropriately, the association between the files can become broken. For this reason, you should use either SolidWorks or SolidWorks Explorer to change names of associated files. Other techniques will work, but you should avoid some techniques.

Best Practice

A poor practice is to change filenames, locations, or the name of a folder in the path of documents that are referenced by other documents with Windows Explorer. Links between parts, assemblies, and drawings can be broken in this way. Using SolidWorks Explorer or a PDM application is the preferred method for changing filenames.

Summary

While product development is about design, it is even more about change. You design something once, but you may modify it endlessly (or it may seem that way sometimes). Similarly, SolidWorks is about design, but it really enables change. Think of SolidWorks as virtual prototyping software that enables you to change your prototype rather than having to make a new one. Virtual prototypes will never completely replace physical models, but they may reduce your dependence on them to some extent.

SolidWorks is also about reusing data. Associativity enables you to model a part once and use it for Finite Element Analysis (FEA), creating 2D drawings, building assemblies, creating photorealistic renderings, and so on. When you make changes to the model, your drawing is automatically updated, and you don't have to reapply FEA materials and conditions or redo the rendering setup. Associativity saves you time by reusing your data. Associativity and change driven by feature-based and history-based modeling can take some getting used to if you have had limited exposure to it, but with some practice it becomes intuitive, and you will see the many benefits for enabling change. Parametric sketching and feature creation help you to maintain Design Intent and to adjust it as necessary.