

## Chapter 1

# Getting to Know (And Love) SolidWorks

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### *In This Chapter*

- ▶ Becoming familiar with SolidWorks
  - ▶ Discovering the advantages of the virtual prototype
  - ▶ Figuring out where to start with SolidWorks
  - ▶ Getting acquainted with new features in SolidWorks 2008
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**S**olidWorks is a tool that helps design engineers harness their imaginations and add creativity to their designs. The true mark of a good tool is when it becomes part of your process without getting in your way. When you design, you need to do just that — design!

When SolidWorks was created, the power of 3D wasn't yet widespread. The company's original mission back in 1995 — and a goal it still pursues today — is to bring the power of 3D to every engineer's desktop. Two early quotes of SolidWorks founders that still hold true today are

- ✓ “No matter how easy it is to use, it is never easy enough.”
- ✓ “No matter how fast we make it, it is never fast enough.”

In this chapter, I introduce you to SolidWorks, the wonderful world of 3D, and the virtual prototype. You discover the basic system requirements for SolidWorks and tips to keep the program running smoothly. I also give you the lowdown on the newest features in SolidWorks 2008.

## *Exploring the SolidWorks Advantage*

As a design engineer, you need to be critical about how you work with your craft and to understand how you can do it better. As tools and technology continue to improve, you also need to evolve. That means staying abreast of

the latest design tool innovations. In this section, you find out how to take advantage of the benefits that 3D and SolidWorks offer.

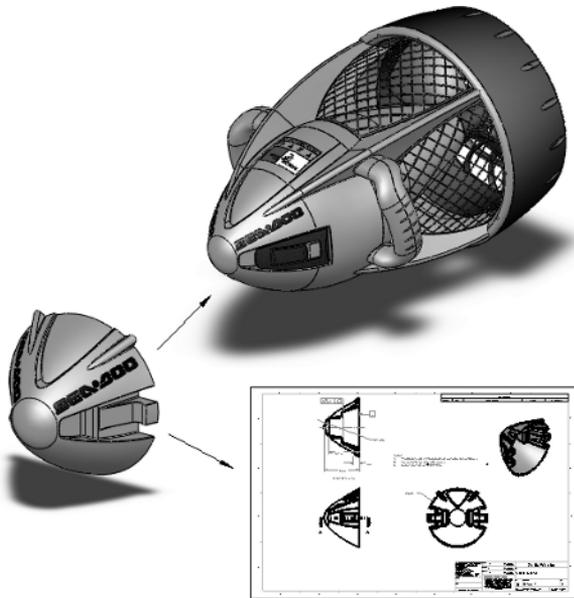
## *Improving the way you work*

Designing in SolidWorks may be different from how you designed in the past. My greatest satisfaction in my early days as a designer came from creating a complex assembly on my computer and then watching the darn thing actually come together on the shop floor just the way I designed it.

Without the ability to create 3D solid models and assemblies, however, your goal isn't easily attainable. The following workflow example shows how modeling in SolidWorks enables you to achieve better results:

### 1. Design 3D parts (such as the one shown in Figure 1-1) and assemblies.

One big advantage to working in 3D is the ability to capture design intent early in the design process. Good designs are built on solid foundations. SolidWorks 3D modeling gives you a better understanding of your design, long before you create the first part.



**Figure 1-1:**  
An example  
of how a  
part is  
reused in  
the design  
process.



*Design intent* is an intelligent arrangement of part features and dimensions, or in the case of assemblies, the location of parts and the interaction between them. Starting your designs by building in good design intent makes reacting to future changes or additions easier.

2. **Test your design as a virtual prototype, using advanced features in SolidWorks to test different ideas more quickly and easily than you can with drawings or traditional prototypes.**

A virtual prototype is such an important idea in using SolidWorks that I discuss it greater detail in the next section.

3. **Generate 2D manufacturing drawings, based on the geometry defined in the original part or assembly.**

Refer to Figure 1-1 for an example. You can find out more about generating drawings in Chapter 7.

## *Embracing the virtual prototype*

In the old days (more than 25 years ago), designers drew their designs on paper. When a designer was done drawing, he built a physical prototype to test his design ideas in the real world. If things didn't work quite right, he went back to the drawing board. Building all those prototypes was time consuming.

Nowadays, you do all that testing and simulating on a *virtual prototype*, which is a concept I refer to throughout this book. When you design a product in SolidWorks, essentially you create a virtual prototype with all the characteristics of the real thing (dimensions, mass properties, screws, and fittings, for example). Your virtual prototype behaves in the computer realm as it would in the real world.



The power of the virtual prototype is that it allows you to test countless design ideas quickly on your computer. And although the physical prototype hasn't vanished, at least you don't need so many of them.

Figure 1-1 shows a virtual prototype. Imagine if you had to draw this baby from scratch. In a drawing, changing the height from 100mm to 120mm requires major effort. In contrast, with a 3D virtual model, you can modify and update key design attributes with ease.

If you think you can work faster in a 2D environment, you're probably right. But that's only because you can't include the detail required to fully manufacture the part. When I made designs on a drawing board with paper and pencil, I could easily add a radius to any corner because I had my trusty ellipse template. In other words, I would "fudge" the corner geometry and let the toolmaker properly fillet the corner. Sometimes this lack of detail is good, and sometimes it's bad.

People use SolidWorks because they can create 3D parts, place them in assemblies in the same way as though they were assembling physical parts, and then create 2D drawings directly from that information.

## Getting Your System Ready for SolidWorks

SolidWorks comes to you on a single DVD. When you're ready to install, put the disc in your DVD drive and follow the instructions provided by the Installation Manager. During installation, you have the opportunity to activate your SolidWorks license automatically over the Internet. (You can also activate your license via e-mail, but that may take a few hours to a few days.) When you first install SolidWorks, you have up to 30 days to activate the license or the software will simply stop working. For more information about activation, or if you experience any difficulties, visit the SolidWorks Customer Portal site at <https://customercenter.solidworks.com>.

The minimum computer requirements boil down to the following:

- ✓ **CPU:** Get the fastest one you can afford. The performance of SolidWorks depends on your CPU speed. If you get a CPU that's too slow, you can add a second one later. Although a second CPU adds some performance increase, it doesn't come close to doubling performance, so it's best to start off big.
- ✓ **RAM:** Although SolidWorks lists the minimum requirements, a better way to gauge how much RAM you need is to open SolidWorks on your computer along with all the other applications you normally have open at one time, such as the ones you use for e-mail, word processing, and Web browsing. Then open a good sampling of SolidWorks documents. (You can find some in the Tutorials folder in the SolidWorks program folder.) Open Windows Task Manager (press Ctrl+Alt+Delete) and click the Performance tab. Check the amount of available memory in the Physical Memory area. If the amount of available memory isn't greater than 0, you need more RAM.  
  
RAM is cheap, so make sure that you have plenty. The amount of RAM you have is important. If Windows runs out of physical memory, bad things happen. Your system becomes sluggish and less stable.
- ✓ **Graphics card:** Make sure that you have a certified graphics card and driver version. The SolidWorks support Web site lists combinations of certified graphics cards and drivers. For a listing of supported graphics cards and drivers, visit the hardware page of the SolidWorks support site at <https://customercenter.solidworks.com>.
- ✓ **Hard drive:** Big and fast is where it's at. These days, folks who use SolidWorks typically have 80 to 120GB hard drives.



## Keeping Your Computer Happy

A commonly overlooked means of making sure that your computer stays happy and healthy is regular system maintenance. Just as craftspeople take good care of their tools, you should treat your computer in much the same way. The two most important tasks are making sure you have sufficient disk space and performing routine disk defragmentation.



You should also check the backup settings in SolidWorks by choosing Tools→Options→System Options→Backup/Recover.

The following settings are particularly important to check:

- ✓ **The number of backup copies per document:** If you set the number of backup copies, keep this number low (one or two) because SolidWorks creates a copy of every document that's opened.
- ✓ **The location of the backups:** You should store backups on a different computer or at a different site. Remember to check your backup drive regularly to make sure that you have enough disk space. If you run low on space, clean up your hard drive or buy a bigger one.

Even when you work normally on a computer, the disks become *fragmented*, which means that Windows can't store all of a file in one contiguous spot, so it starts using a number of places on the hard drive to store documents. As you can imagine, fragmentation makes Windows run slower and causes stability issues with the system and the applications running on the computer. To alleviate this problem, use the Windows Disk Defragmenter (choose Start→All Programs→Accessories→System Tools). I run a complete scan weekly. It makes a difference.

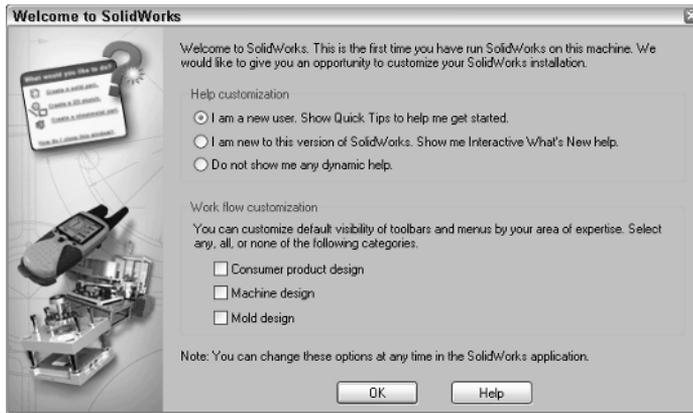


For more information on many of these administrative tasks, check out the *SolidWorks Express* archive. *SolidWorks Express* is a bimonthly electronic newsletter for the SolidWorks community. To view technical tips in the archive, go to <https://www.customercenter.solidworks.com>.

## Starting Up SolidWorks the First Time

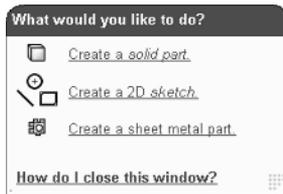
The first time you run SolidWorks, the Welcome to SolidWorks dialog box appears (see Figure 1-2) and asks how you want to configure the help and workflow customizations. You can set up SolidWorks based on your industry and skill level.

**Figure 1-2:**  
Set up  
SolidWorks  
according to  
how you  
work and  
what you  
know and  
don't know.



The first section in the dialog box is labeled Help Customization. If you're a new user, you can activate *Quick Tips*, a set of pop-up messages that appear while you create SolidWorks documents. The messages display hints and options about what to do next. Figure 1-3 shows a Quick Tip that pops up when you create a new part document. The Quick Tip walks you through what is required to perform this task.

**Figure 1-3:**  
Quick Tips  
give you  
an idea  
of where to  
go next.



You can turn on Quick Tips in each type of document (drawing, part, or assembly) by choosing Help → Quick Tips.

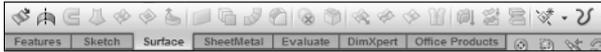
The second section is labeled Work Flow Customization. If you make a selection there, SolidWorks displays toolbars and menus that relate to the type of work you do. The choices are

- ✓ Consumer Product Design
- ✓ Machine Design
- ✓ Mold Design

You can select one, all, or none of these categories.

Based on the type of industry you select, SolidWorks displays a different set of tools in the *CommandManager*, a context-sensitive toolbar that's dynamically updated based on the toolbar you want to access. In Figure 1-4, you see the Surface toolbar.

**Figure 1-4:**  
The Surface  
toolbar  
appears.



To change the icons shown within the CommandManager, right-click the CommandManager and choose *Customize CommandManager*. A menu of available CommandManager icons appears. Select the icons that fit the type of work you most commonly perform.

## Checking Out the Features

SolidWorks 2008 is the 16th version of SolidWorks to be released since the initial release of SolidWorks 95. Version updates, known as *service packs*, come out about every six weeks, whereas major releases occur typically 10 to 12 months apart.

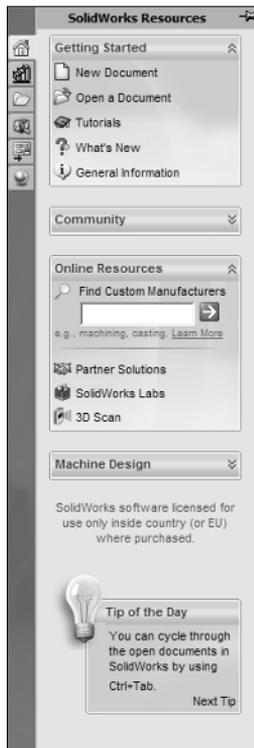
You can download service packs from the SolidWorks Customer Portal Web site at <https://customercenter.solidworks.com>, or by choosing Help → Check for Updates.

## Finding help and tips

A couple of resources can help get you started. The SolidWorks Resources tab (look for a house icon on the far right end of the user interface) in the task pane, as shown in Figure 1-5, displays links to these items:

- ✓ **Tutorials:** Online tutorials offer a group of 30-minute, step-by-step tutorials on a wide variety of topics. The first tutorial, Lesson 1, gives a quick overview of the basic features and functions in SolidWorks and is a good starting point if you haven't ever used the software.
- ✓ **What's New:** This document describes the new features within the latest version of SolidWorks based on area (features, parts, assemblies, and drawings, for example). You can also access the What's New document (a PDF file) from the Help menu.

- ✔ **Machine Design, Mold Design, or Consumer Product Design:** The title and contents of this tab relate to the industry type you chose in Workflow Customization. Figure 1-5 shows resources for machine design. If you chose mold design, other resources are displayed. On this tab, you find an overview of the industry and industry-specific tutorials.
- ✔ **Tip of the Day:** This tip changes each time you open SolidWorks.



**Figure 1-5:** Inside the SolidWorks Resources tab in the task pane, you find a plethora of learning resources.

You can also find the following references on the Help menu:

- ✔ **SolidWorks Help:** This option opens the *SolidWorks Online User's Guide*. The guide is organized into chapters with an index you can browse. A search tool guides you to information on just about any feature in SolidWorks.
- ✔ **Moving from AutoCAD:** This online guide is designed to help you move from AutoCAD, a popular 2D design program, to SolidWorks. Many SolidWorks users come from AutoCAD backgrounds.

## *Finding out what's new in SolidWorks 2008*



We based this book on SolidWorks 2008. Each version of SolidWorks has updates to accommodate new features and requirements. When you save a file in the newest version, keep in mind that older versions can't read these newer files because the new features weren't available in the older version. However, you can open older versions of documents in the new version.

Here's a taste of what's new inside SolidWorks 2008:

- ✔ **Browse Recent Documents:** This new interface feature allows you to select a list of recent documents and see a visual preview in the Recent Document browser. Clicking the preview window opens the document (see Chapter 2).
- ✔ **Context Toolbars:** When you select items in the graphics area or FeatureManager design tree geometry, context toolbars appear and provide access to frequently performed actions for that context — for example, editing the sketch of a selected face (see Chapter 5).
- ✔ **Instant3D:** Drag geometry and dimension manipulators to resize features and make other modifications (see Chapter 9).
- ✔ **AssemblyXpert:** The AssemblyXpert analyzes the performance of assemblies and suggests actions you can take to improve it. This feature is useful when you work with large and complex assemblies (see Chapter 6).
- ✔ **DimXpert:** DimXpert for parts is a set of tools that applies dimensions and tolerances to parts according to the requirements of the ASME Y 14.41-2003 standard. Covering the DimXpert is a bit beyond the scope of this book, but you can find more information in the SolidWorks Help.
- ✔ **TolAnalyst:** The tolerance analysis application, or TolAnalyst, determines the effects that dimensions and tolerances have on parts and assemblies. The TolAnalyst tools let you perform worst-case tolerance stack-up analysis on parts and assemblies (see Chapter 8).
- ✔ **Mates:** Several new mate types have been added, including Screw mates and Universal joint mates (see Chapter 6).
- ✔ **Quick View:** Quick View allows you to open assembly models with only selected parts available for viewing and modifying (see Chapter 6).
- ✔ **Scenes and Appearances:** Adds photorealistic display of your models and environments. Types of scenes that can be displayed include reflective floors, your own photos, and reflections of backgrounds.

