

Chapter 1

Getting to Know (And Love) SolidWorks

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 2006
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SolidWorks is a tool that helps design engineers harness their imagination 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 was not yet widespread. The company's original mission back in 1995 — and a goal that 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 smooth. I also give you the lowdown on the newest features in SolidWorks 2006.

Exploring the Ins and Outs of 3D

As a design engineer, you need to be critical as to how you go about your craft and how you can do it better. As tools and technology continue to improve, you too need to evolve. And that means staying abreast of the latest design tool innovations.

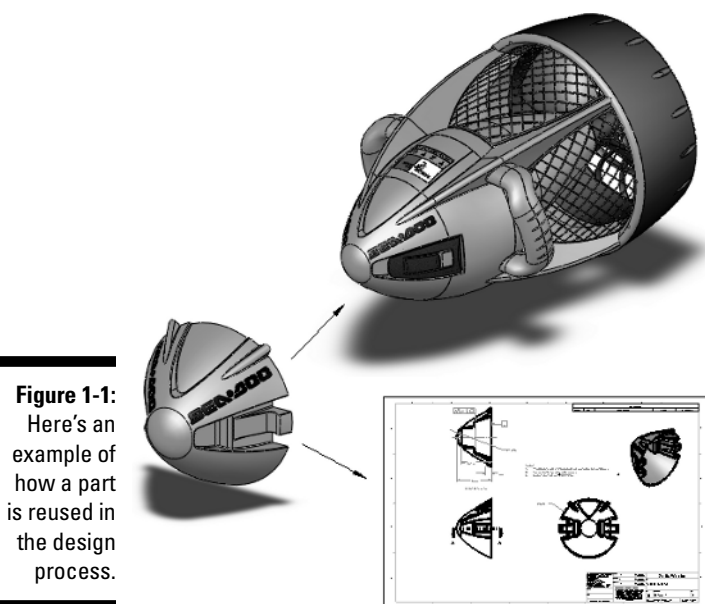
Just as I wouldn't pound nails with a shovel, I also believe in using CAD tools for the purpose for which they were designed. Throughout this book, I emphasize how to use SolidWorks more effectively. It may be that designing in SolidWorks is different from how you've designed in the past. But keep an open mind to how you can sharpen your skills with 3D tools.

One of the biggest advantages 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. It also allows you to reuse design information downstream, for example, when it comes time to manufacture the design.

The SolidWorks advantage

When I reflect back on my early days as a designer, I realize that my greatest satisfaction 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, this goal isn't easily attainable.

SolidWorks lets you design 3D parts (see Figure 1-1) and assemblies and use them to generate manufacturing drawings. The drawings are based on the geometry defined in the original part or assembly.



Embracing the virtual prototype

The virtual prototype is a concept that I refer to throughout this book. In the old days (I'm talking 20 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, it was back to the drawing board. It took a lot of time building all those prototypes.

Nowadays, you do all that testing and simulating on a *virtual prototype*. 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, and so on). 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 many, many, many design ideas quickly on your computer. And though the physical prototype has not vanished, you at least don't need so many.

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 design, 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 radius any corner because I had my trusty ellipse template. In other words, I would fudge the corner 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 a 3D part, place that part in an assembly the same way as if they were assembling physical parts, and then create a 2D drawing directly from that information.

Setting Up the System

SolidWorks comes to you on three CDs. When you are ready to install, put the first CD in your CD-ROM drive and follow the instructions. During installation or when you first run SolidWorks, the software asks for a registration code. This is different than the serial number, which is listed on the box. If a registration code does not automatically appear when you register the software, you can get one through your SolidWorks reseller (often called a Value Added Reseller, or VAR) or by visiting the registration page of the SolidWorks support site at www.solidworks.com/pages/services/Registration.html.

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 is too slow, you can add a second one later. But 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:** SolidWorks lists minimum requirements, but I tell folks that 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 e-mail, word processing, a Web browser, and so on. 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 (Ctrl+Alt+Delete) and click the Performance tab. Check the available memory under Physical Memory. If available memory is not greater than 0, you need more RAM.



The amount of RAM you have is important. If Windows runs out of physical memory, bad things happen. The system becomes sluggish and less stable. RAM is cheap, so make sure that you have plenty.

- ✓ **Graphics card:** Make sure that you have a certified graphics card and driver version. The SolidWorks support Web site lists certified graphics cards/driver combinations. For a listing of currently supported graphics cards/drivers, visit the hardware page of the SolidWorks support site at www.solidworks.com/pages/services/VideoCardTesting.html.
- ✓ **Hard drive:** Big and fast is where it's at. These days, folks who use SolidWorks typically have 60 to 80GB hard drives.

Keeping Your Computer Happy

A commonly overlooked means of making sure that your computer stays happy and healthy is regular system maintenance. The two most important items on this list are making sure you have sufficient disk space and routine disk defragmentation.



You should also check the backup settings in SolidWorks (Tools⇨Options⇨Backups). Check the number of backup copies per document and the location of the backups. If you set the number of backup copies, keep this number low (1 or 2) because you will be creating copies of every document opened. It's best to store backups on a different computer or at a different site. Remember to check your backup directory regularly to make sure that you have enough disk space. If you run low on space, it's time to clean up your hard drive or buy a bigger one.

During your normal work on a computer, the disks become fragmented, which means that Windows can't store all of a file in one 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 (Start⇨All Programs⇨Accessories⇨System Tools). I run a complete scan on a weekly basis. It does make a difference.



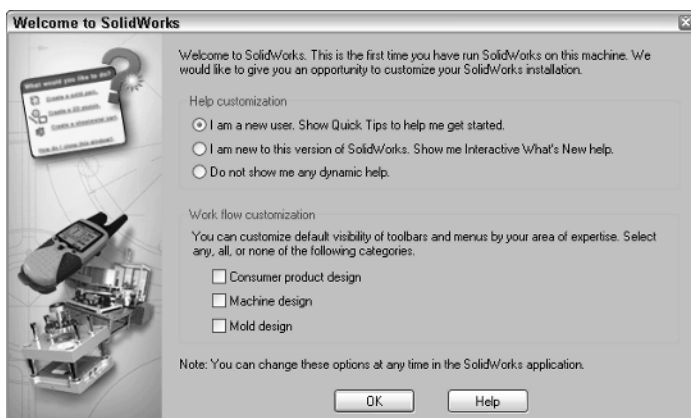
For more information on many of these administrative tasks, check out the *SolidWorks Express* archive. *SolidWorks Express* is a bimonthly electronic newsletter for the SolidWorks community. To view technical tips in the archive, visit www.solidworks.com/swexpress.

Things to Know Before You Start Designing

Before I start talking about SolidWorks, I want to explain a few basic items.

The first time you run SolidWorks, the Welcome to SolidWorks dialog box appears (see Figure 1-2) to ask you how you want to configure the Help and Workflow customization. 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 Help Customization. If you're a new user, you can activate Quick Tips. *Quick Tips* are a set of pop-up messages that appear while you create SolidWorks documents. They 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 Quick Tips on in each type of document (drawing, part, or assembly) by choosing Help⇨Quick Tips.

The second section is Work Flow Customization. If you make a selection here, SolidWorks displays toolbars and menus that relate to the type of work you do. The choices are Consumer Product Design, Machine Design, and 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 (see Figure 1-4). The *CommandManager* is a context-sensitive toolbar that dynamically updates based on the toolbar you want to access.

Figure 1-4:
The
Command-
Manager
automat-
ically
updates
to display
the toolbar
you need.
Here, the
Surface
toolbar
appears.





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

Checking Out the Features

SolidWorks 2006 is the 14th version of SolidWorks to come out since the initial release of SolidWorks 95. Version updates, known as *service packs*, come out on about every six weeks, while major releases occur typically 10 to 12 months apart.

You can download service packs from the SolidWorks support Web site at www.solidworks.com/pages/services/downloads.html.

Getting started

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

- ✓ **Online Tutorial:** Online Tutorial offers 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 with SolidWorks and is a good starting point if you haven't ever used the software.
- ✓ **What's New:** A document that describes the new features within the latest version of SolidWorks based on area (features, parts, assemblies, drawings, and so on). You can access the What's New document (PDF) in 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 display. In this tab you find an overview of the industry and industry-specific tutorials.
- ✓ **Tip of the day:** This 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 under the Help menu:

- ✓ **SolidWorks Help Topics:** This takes you directly to the *SolidWorks Online User's Guide*. The guide is organized into chapters with an index you can browse through. A search tool guides you to information on just about any feature in SolidWorks.
- ✓ **Moving from AutoCAD:** An online guide designed to help those moving from AutoCAD to SolidWorks. AutoCAD is a popular 2D design program. Many SolidWorks users come from an AutoCAD background.

Finding out what's new to SolidWorks 2006



I based this book on SolidWorks 2006. Each version of SolidWorks has updates to accommodate new features and requirements. When you save a file in the new 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 in the new version.

Here's a taste of what's new inside SolidWorks 2006:

- ✓ **Spell Checker:** One of my favorites, as I am a typical engineer when it comes to spelling. This feature works similar to Microsoft application spell checkers. It checks a drawing for spelling errors and offers suggestions (see Chapter 7).
- ✓ **Multiple Viewports:** Gives you the ability to see your part or assembly in a two- or four-window configuration. Panning and zooming in one view cause the other views to update in scale and position. If you prefer to gaze over multiple views (isometric, top, front, and right side) at the same time, this feature may appeal to you (see Chapter 2).

If you're used to working with another CAD system where multiple views (isometric, top, front, and right side) appear at the same time, this feature may be for you.
- ✓ **Text-in-a-Box:** Creates a bounding box to control the placement of notes in drawings. This feature makes it easier to place annotations in a drawing because you can define and change the size of the note based on the space available on the drawing (see Chapter 7).
- ✓ **3D Annotations Views (ASME 14.41-2003):** Makes it easier to control and organize the placement of 3D annotations in the model. This feature also automates the placement of 3D annotations in drawings (see Chapter 7).

The dimensions are easier to read, and this standard helps define one of the other concepts that I stress in this book: When creating parts, make your sketches look how they should in the drawing. Then reuse these dimensions to create the drawing.
- ✓ **Display States:** Allow you to save your favorite visual settings (component color, component texture, Hide/Show state, and so on) in a Display States folder for reuse in different assembly components or drawing views (see Chapter 2).
- ✓ **Splines:** Graphical handles on splines (wiggly shaped lines) allow you to better edit and control the shape of the curves (see Chapter 4).
- ✓ **Sketch Blocks:** Similar to Windows block, lets you grab multiple entities in sketch mode and treat them as a single block. You can combine sketch blocks with other entities to create complex layouts. Sketch blocks are fully associative. If you change one instance, all instances update automatically (see Chapter 3).

- ✓ **Fastening Feature:** Adds common plastic part features (mounting bosses, snap hooks, snap grooves, and vents) to plastic parts automatically by selecting the new Fastening Feature command.
- ✓ **Smart Components:** Allow you to relate features and components to a part. For example, when you place a connector in an assembly, you can define smart components that automatically add mounting holes, nuts, and bolts to complete the assembly.
- ✓ **SolidWorks Design Checker:** Checks a drawing for a number of predefined criteria (dimension size, drafting standards, materials, custom properties, and so on) to ensure that the document meets your standards and practices. This option is included in the SolidWorks Office Professional and SolidWorks Office Premium software packages.