Alex R. Ruiz
with Gabi Jack
Foreword by Josh Mings, SolidSmack.com

SolidWorks® 2010

NO EXPERIENCE REQUIRED

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Alex Ruiz
with Gabi Jack

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Best regards,

Neil Edde
Vice President and Publisher
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To my wife Griselda and my children,Orion, Ian, Venus, and Maya.
—Alex R. Ruiz

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Finally, I want to thank you, the reader. I hope you find this book not only educational but also enjoyable. I hope you have as much fun using this book as I had writing it.

—Alex R. Ruiz

About the Authors

Alex R. Ruiz is a Certified SolidWorks Professional (CSWP) and engineering manager for a leading medical device manufacturer. He designs new products and trains and supports more than 100 SolidWorks users. Alex has close ties to the development team and is very well known in the SolidWorks community as the SolidWorks Geek, which is the name of his blog (www.TheSWGeek.com).

Gabi Jack is a Certified SolidWorks Professional and mechanical engineer who maintains a popular blog about solid modeling, design, and engineering (www.GabiJack.com).

Foreword

Do you really want to read a Foreword? Really? If I were you, I'd be skipping to the chapter on creating impressive looking surfaces, but since you're here, I've got something really important to say to you. You are living in an amazing time. Way more amazing than the last "amazing time" someone told you about. This is the year you are learning SolidWorks. There's a lot to be said about taking that on, but I'll whittle it down to this: You're in for a ride.

When people go about learning a 3D modeling application like SolidWorks there's a hesitation that simmers in the idea that there's just way too much to learn. It's true. There is too much to learn—in one day. However, just as languages have a set of commonly used words, so too does SolidWorks have a set of commonly used features. Features which, after a week of practicing using this book, will give you all the abilities needed to introduce SolidWorks to your team, verify a design, get a better job, or create a new idea. I imagine one of those abilities is going to be very important for you over the coming years. What's even more important is how you start gaining those abilities.

You could have started modeling 3D geometry 15 years ago when SolidWorks first came on the scene. You actually may have, but don't kick yourself if you didn't. Over those past 15 years, developing products in 3D has changed immensely. SolidWorks has been a big part of making that happen. I've seen the program gain an enormous feature set that has expanded from basic modeling features to complex assembly and surface design with added functionality for rich simulation and motion analysis. It is a program that spans an increasing number of industries and disciplines. With all of that it truly becomes a program which, in the right hands, would turn a capable person like

yourself into a modeling genius. That’s the potential you have starting right now. Plus, you’re not alone. I’ve seen the user base grow to become one of the largest in the mechanical design communities in the world. People from around the world interact daily within online resources spanning video tutorials and model data across a growing number of web sites, blogs, and forums. If you want to model it in SolidWorks, there’s likely someone out there doing it.

So, you probably get the idea that there’s a lot of information out there about SolidWorks. It can get distracting and overwhelming. So, I want to give you a challenge. Focus intently on this book for a week with all the passion you can muster. Write in the margins, highlight the text, and apply each topic to what you design. This is it where it gets real. This is where you start to not only learn more about how to master SolidWorks, but also how to apply what you can do with SolidWorks daily to expanding your career. Many times the only thing missing is the right book. I’m convinced this book, is that book. So now, you have a great program, a great book, and a challenge to complete. You won’t be disappointed.

—Josh Mings

SolidSmack.com

Introduction

Whether you are a new user of SolidWorks or a professional who wants to improve your skills, this book was written for you. Learning any software can be difficult at times. You launch the software for the first time, and you feel overwhelmed, not knowing how to even start a new document. In 3D CAD programs, it can be especially difficult. Many times a whole new vocabulary and a whole new creative environment are introduced.

In this book, you will learn how to use the software—it covers everything from what you see when you open SolidWorks for the first time to 3D solid modeling and to how to create high-resolution renderings of the desk-top lamp that you will create by following the examples. With plain-English step-by-step tutorials, you will create 3D parts, assemblies, and drawings. Not only will you learn how to create models and drawings, but you will be introduced to some of the reasons why certain techniques are used and how to put them to use in your daily job.

As with previous releases, SolidWorks 2010 has introduced many new tools and commands to make your daily life easier. You will be introduced to the new tools in parts, assemblies, and drawings, including the new mouse gesture support, the changes to reference planes, the Dimension Palette in drawings, and many more. You will also learn how to create various templates and how to customize your workspace, all meant to increase your productivity. Although many of the more advanced modeling techniques are not covered in this book, any level of user will still be able to find something new about the software.

At the end of it all, you'll use the model of the lamp to create photorealistic renderings using the newly updated PhotoView 360. In 2010, PhotoView 360 was updated with

even more tools to create renderings rivaling that of PhotoWorks. With new ways of controlling scenes, support for background images and custom environments, and new camera effects, you can create images that bring your models to life. As with all the chapters in the book, you will learn how to create your own images with step-by-step tutorials.

What You Will Learn in This Book

Each chapter was written to gradually introduce new tools and concepts as the design progresses. Each subsequent chapter will describe progressively more advanced techniques. Specifically, the book is structured as follows:

Chapter 1 describes the SolidWorks user interface, including the menus and toolbars, the CommandManager, the FeatureManager design tree. You'll also learn about ways to improve productivity with shortcut keys and mouse gestures.

In **Chapter 2**, you will learn the basics of using SolidWorks, including the various document types, how they relate to each other, and how parts, assemblies, and drawings are created.

In **Chapter 3**, you will create your first 3D model of the lamp base. You will learn how to create a fully defined sketch and how to use it to create extrusions that form the model.

In **Chapter 4**, using the model created in Chapter 3, you will create a 2D drawing suitable for manufacturing using a variety of drawing methods, including importing annotations from the model, using the Dimension Palette to tolerance a dimension, and creating various drawing view types.

Chapter 5 will demonstrate how to create the shaft for the lamp base by using a revolved feature.

Chapter 6 will continue to explore the revolve command to create a washer and washer cover for the lamp shade. Then, after creating the two models, you will learn how to create your first assembly, and you will be introduced to assembly mates.

In **Chapter 7**, using the assembly created in Chapter 6, you will create an assembly drawing and learn the basics of how to use a bill of materials (BOM).

In **Chapter 8**, you will learn even more modeling techniques, including how to create a swept feature, add reference planes, and use mirrored features. You will even learn how to create a modeled thread on the part.

Chapter 9 will show you how to create in-context models within a subassembly when you create the lamp's shade. More assembly mates will then be demonstrated when you create multiple configurations to define the shade's positions.

Chapter 10 will demonstrate some of the methods used to modify existing SolidWorks documents using a variety of techniques.

In **Chapters 11** and **12**, you will put everything together to create the top-level assembly of the desk lamp. After creating the assembly, you will learn how to add a BOM to the environment as well as create an exploded view of the top-level assembly to see how it all goes together.

Chapter 13 will show you how to create the final drawing for the desk lamp project and some additional drawing techniques meant to increase productivity.

In **Chapter 14**, you will learn various techniques for sharing your model and drawings with other users, manufactures, vendors, and sales teams.

Chapters 15 and **16** will describe the process for creating your own templates in SolidWorks.

Chapter 17 will introduce you to PhotoView 360 and will show you how to create photorealistic renderings of the desk lamp using new enhancements in the rendering software.

At the end of the book is a glossary of terms that are used in the book and that are related to SolidWorks and mechanical design, followed by an index.

Files on the Website

A few exercises in this book require additional files such as templates, tables, and some models not created in the exercise. The entire project including each part, assembly, and drawing is also available for download. You can download the accompanying files from this book's page on Sybex's website at www.sybex.com/go/solidworks2010ner. Click the Downloads button on that page to access the files. You can also find the same files as well as additional content, forums, and more examples at www.swner.com.

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Chapter 1: Becoming Familiar with SolidWorks

- Start SolidWorks
- Navigate the SolidWorks Interface
- Use the CommandManager
- Use and Customize the Menus
- Use Toolbars
- Use the Keyboard
- Use the Mouse

SolidWorks 2010 is one of the most popular 3D mechanical computer-aided design (CAD) packages on the market today. Since its introduction in 1995, SolidWorks has become a favorite design tool for many of today's engineers, mechanical designers, and industrial designers. In part because of its easy-to-learn graphical user interface and powerful set of tools, SolidWorks is used by many top companies worldwide to design, engineer, and document their products in a variety of fields.

At the core of SolidWorks is the ability to create parametric 3D solid geometry that is then used to create drawings, manufacturing instructions, instruction manuals, animations, full-color renderings, and other types of documentation. Regardless of the complexity of the item being created, the creation process is easy and follows the same basic steps. First a *sketch* is created that is turned into a *base feature*. The base feature is then further refined by adding features that add or remove material from the base feature. Individual *part models* can then be used to build *assemblies* that represent the final design. After creating the 3D part or assembly models, *drawings* are made to document the design and manufacturing process.

Learning a new CAD package can be a daunting task. In addition to the new terminology, first-time users may feel a bit overwhelmed with a new user interface, toolbars, and commands. In this chapter, you will spend some time launching SolidWorks for the first time, becoming familiar with the SolidWorks interface, and working with the CommandManager.

Start SolidWorks

Before installing and running SolidWorks for the first time, ensure that you meet the recommended minimum system requirements. SolidWorks currently supports the following operating systems:

- Windows 7 (32-bit) Professional, Ultimate or Enterprise Edition.
- Windows 7 (64-bit) Professional, Ultimate or Enterprise Edition.
- Windows Vista (64-bit) Ultimate, Business, or Enterprise edition, SP0 or newer
- Windows Vista (32-bit) Ultimate, Business, or Enterprise edition, SP0 or newer
- Windows XP Professional (32-bit), SP2 or newer
- Windows XP Professional (64-bit)

And here are the random-access memory (RAM) requirements:

Minimum 1GB RAM when parts contain fewer than 200 features and assemblies contain fewer than 1,000 components

Recommended 2GB RAM or more when parts contain more than 200 features and assemblies contain more than 1,000 components

Once you have verified that your computer is able to support SolidWorks and it is installed onto your system, you

can launch it by selecting Start ⇒ Programs ⇒ SolidWorks 2010 ⇒ SolidWorks 2010 SPX.X ⇒ SolidWorks 2010.

NOTE All images in this book are from SolidWorks running on Windows 7. You might notice a slight difference if you are using another version of windows such as Windows XP.

SolidWorks License Agreement

The first time you launch SolidWorks, you will be presented with the SolidWorks License Agreement. You must accept the license agreement in order to use SolidWorks. After reading the license, click Accept to continue.If for some reason you do not accept the terms of the license agreement, clicking Do Not Accept will exit SolidWorks.

Help and Workflow Customization

After accepting the SolidWorks License Agreement, you will then be presented with the Welcome To SolidWorks window. This screen allows you to customize the appearance of dynamic help as well as the workflow. You will see this only the first time you launch SolidWorks on your computer, but you can make changes to the options anytime you want in the SolidWorks Options window.

Three options are available in the Help Customization section of the screen. Each option will provide the user with a different level of dynamic help, so consider your needs when making your selection.

I Am A New User. Show Quick Tips To Help Me Get Started. This option will provide you with pop-up messages that appear while working in different modes of SolidWorks.

I Am New To This Version Of SolidWorks. Show Me Interactive What's New Help. Experienced SolidWorks users will find this option helpful when they are working in a new version of SolidWorks. When this option is selected, a question-mark icon will be displayed on new menu items and new and changed PropertyManagers and will link to the corresponding sections of the What's New manual. The topics in the What's New manual will then provide more information about the new or updated functionality since the previous release.

Do Not Show Me Any Dynamic Help. For more experienced users, this option will not provide you with any pop-ups or links to the What's New manual while working in SolidWorks.

NOTE As you become more familiar with working in SolidWorks, you may decide to disable the Quick Tips. You can disable them by selecting Help ⇒ Quick Tips or by clicking the question-mark icon in the status bar. After becoming familiar with the updates made to the new release of SolidWorks, you can disable the display of the link by selecting Help ⇒ Interactive What's New.

The *Workflow Customization* section of the Welcome To SolidWorks window allows you to hide and display tools, links, and menus items based on your usage of SolidWorks. You can select one, two, all, or none of the following categories:

- Machine Design
- Mold Design
- Consumer Product Design

When you select an option in the Workflow Customization section of the window, the following changes will occur in your part document environment:

Machine Design The Machine Design Overview, Machine Design Tutorials, and SolidWorks SimulationXpress links

will be displayed on the SolidWorks Resources tab of the task pane. Sheet Metal and Weldments tabs will be added to the CommandManager. The Molds menu item will be hidden in the Insert menu. Draft Analysis, Undercut Detection, and Deviation Analysis will also be hidden in the Tools menu.

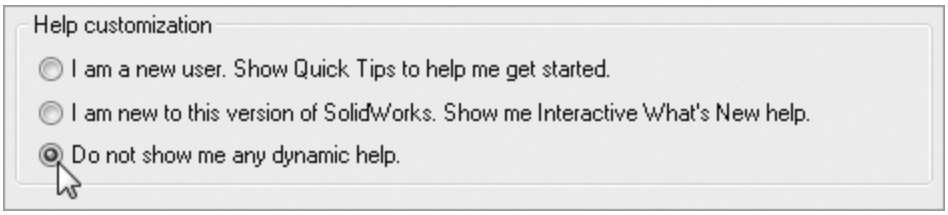
Mold Design The Mold Design Overview, Mold Design Tutorials, and Import File links will be displayed on the SolidWorks Resource tab of the task pane. Surfaces and Molds tabs will be added to the CommandManager. The Weldments menu item will be hidden in the Insert menu.

Consumer Product Design A Consumer Product Tutorials link will be displayed on the SolidWorks Resources tab of the task pane. The Surfaces tab will be added to the CommandManager. The Weldments menu item will be hidden in the Insert menu. The Undercut Detection menu item will be hidden in the Tools menu.

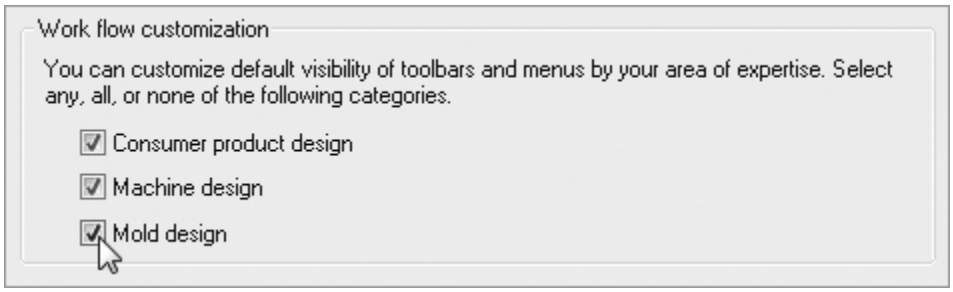
NOTE You can adjust your workflow customization at any time while in a part file by selecting Tools ⇒ Customize and select the Options tab. In the Work flow Customization section, select or deselect the appropriate options.

For the sake of the project being demonstrated in this book, in the Welcome To SolidWorks window select the following:

1. In the Help Customization section, select Do Not Show Me Any Dynamic Help.



2. In the Work flow Customization section, select Consumer Product Design, Machine Design, and Mold Design.



3. Click OK.

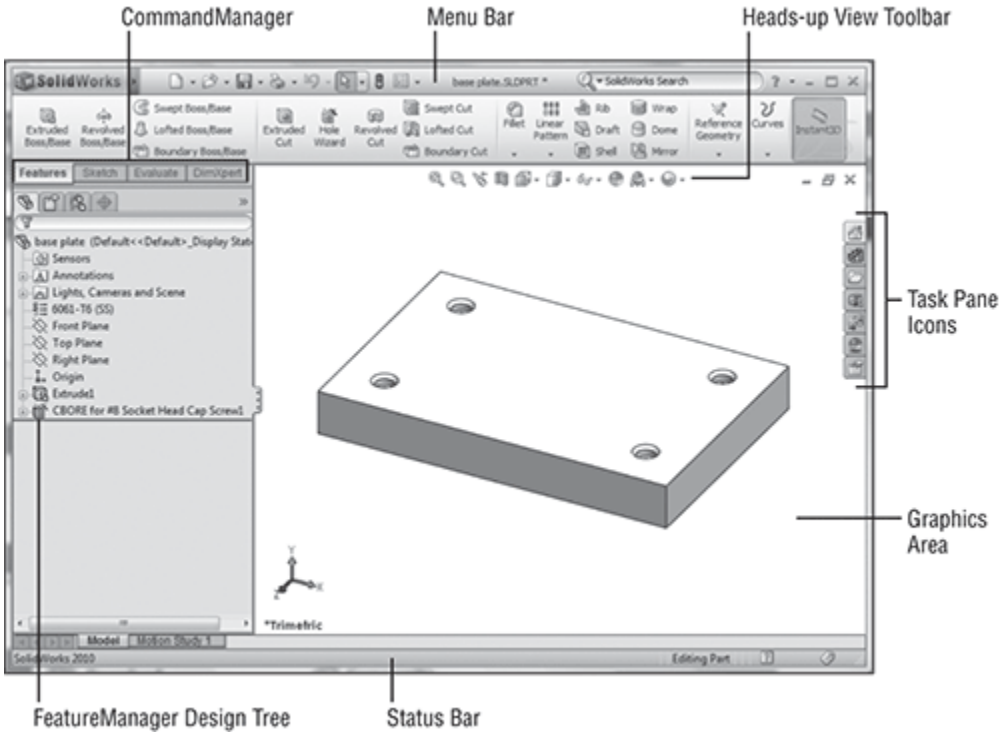
Navigate the SolidWorks Interface

Before using SolidWorks, you should become familiar with the layout of the user interface. Each of the three *document types* in SolidWorks (parts, assemblies, and drawings) has the same basic interface with a few minor differences. To start, we will examine the common elements of the three document types. Figure 1-1 shows the SolidWorks interface when you have a part model open.

Graphics Area

The place where all the action takes place in SolidWorks is the *graphics area*. Here you will be modeling your parts, putting together your assemblies, and creating your drawings. You will be exploring this area in a lot more detail in Chapter 2, “Learning the Basics,” when we cover the three document types in more detail.

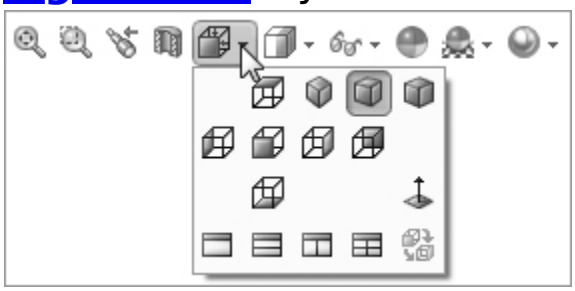
Figure 1-1: SolidWorks 2010 user interface



Heads-up View Toolbar

At the top of the graphics area is the *Heads-up View toolbar*. This transparent toolbar is always available at the top of your graphics area, giving you quick and easy access to the tools necessary to manipulate your views. Icons that display a small downward-pointing arrow provide you with more tools in a flyout, as shown in Figure 1-2.

Figure 1-2: Flyout menu showing additional tools



As you become more comfortable in SolidWorks, you may discover that the tools available on the Heads-up View toolbar may not be what you use most often. The view tools shown by default are not the only tools that are available for