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

SolidWorks® 2010

NO EXPERIENCE REQUIRED

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SOLIDWORKS® 2010

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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**).

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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.