INTRODUCTION

SOLIDWORKS® 2018 Reference Guide with video instruction is a comprehensive text written to assist beginner to intermediate users of SOLIDWORKS. SOLIDWORKS is an immense software package, and no one book can cover all topics for all users. The book provides a centralized reference location to address many of the System and Document properties, FeatureManager, PropertyManager, ConfigurationManager, and RenderManager along with 2D and 3D sketch tools, sketch entities, 3D feature tools, Sheet Metal, Motion Study, SOLIDWORKS Simulation, PhotoView 360, Pack and Go, 3D PDFs, Intelligent Modeling techniques, 3D printing and more.

Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS® 2018 software. If you are completely new to SOLIDWORKS, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2 and Lesson 3 in the SOLIDWORKS Tutorials.

If you are familiar with an earlier release of SOLIDWORKS, you still might want to skim Chapter 1 to be acquainted with some of the commands, menus and features that you have not used, or you can simply jump to any section in any chapter.

Each chapter provides detailed PropertyManager information on key topics with individual stand-alone short tutorials to reinforce and demonstrate the functionality and ease of the SOLIDWORKS tool or feature. The book provides access to over 260 models, their solutions and additional support materials. Learn by doing, not just by reading.

Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, design tables and configurations.

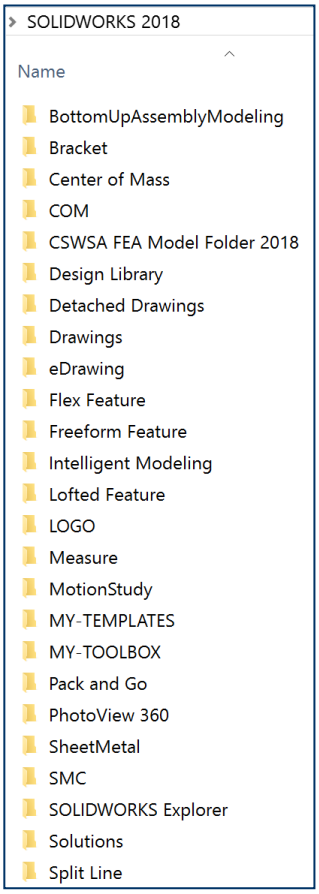
The book is designed to complement the Online Tutorials and Online Help contained in SOLIDWORKS 2018. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs.

Redeem the code on the inside cover of the book. View the provided videos and models to enhance the user experience.

- Start a SOLIDWORKS session.
- Understand the SOLIDWORKS Interface.
- Create 2D Sketches, Sketch Planes and use various Sketch tools.
- Create 3D Features and apply Design Intent.
- Create an Assembly.
- Create fundamental Drawings Part 1 & Part 2.

The author developed the tutorials by combining his own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. These professionals are directly involved with SOLIDWORKS every day. Their responsibilities go far beyond the creation of just a 3D model.

About the Book



You will find a wealth of information in this book. Short stand-alone step-by step tutorials are written for each topic with the new and intermediate user in mind.

The following conventions are used throughout this book:

1. The term document is used to refer to a SOLIDWORKS part, drawing or assembly file.
2. The list of items across the top of the SOLIDWORKS interface is the Main menu. Each item in the Main menu has a pull-down menu. When you need to select a series of commands from these menus, the following format is used: Click **Insert ➢ Reference Geometry ➢ Plane** from the Main menu bar. The Plane PropertyManager is displayed.
3. Screen shots in the book were made using SOLIDWORKS 2018 SP0 running Windows® 10.
4. The book is organized into chapters. You can read any chapter without reading the entire book. Each chapter has short stand-alone step-by-step tutorials to practice and reinforce the subject matter and objectives. Learn by doing, not just by reading.
5. The ANSI overall drafting standard and Third Angle projection is used as the default setting in this text. IPS (inch, pound, second) and MMGS (millimeter, gram, second) unit systems are used.
6. Redeem the code on the inside cover of the book. View the provided videos and models to enhance the user experience. Download and copy the SOLIDWORKS files and folders to your hard drive. The book provides access to over 250 models, their solutions and additional support materials.
7. Compare your results with the tutorial documents in the Summary folders. All models for the stand-alone tutorials are included.

Instructor's information contains over 45 classroom presentations, along with helpful hints, what's new, sample quizzes, avi files of assemblies, projects and all initial and final SOLIDWORKS models.

About the Author

David Planchard is the founder of D&M Education LLC. Before starting D&M Education, he spent over 27 years in industry and academia holding various engineering, marketing, and teaching positions. He holds five U.S. patents. He has published and authored numerous papers on Machine Design, Product Design, Mechanics of Materials, and Solid Modeling. He is an active member of the SOLIDWORKS Users Group and the American Society of Engineering Education (ASEE). David holds a BSME, MSM with the following professional certifications: CCAI, CCNP, CSDA, CSWSA-FEA, CSWP, CSWP-DRWT and SOLIDWORKS Accredited Educator. David is a SOLIDWORKS Solution Partner, an Adjunct Faculty member and the SAE advisor at Worcester Polytechnic Institute in the Mechanical Engineering department. In 2012, David's senior Major Qualifying Project team (senior capstone) won first place in the Mechanical Engineering department at WPI. In 2014, 2015 and 2016, David's senior Major Qualifying Project teams won the Provost award in Mechanical Engineering for design excellence.

David Planchard is the author of the following books:

- **SOLIDWORKS® 2018 Reference Guide with video instruction**, 2017, 2016, 2015, 2014, 2013, 2012, 2011, 2010, and 2009
- **Engineering Design with SOLIDWORKS® 2018 with video instruction**, 2017, 2016, 2015, 2014, 2013, 2012, 2011, 2010, 2009, 2008, 2007, 2006, 2005, 2004, and 2003
- **Engineering Graphics with SOLIDWORKS® 2018 with video instruction**, 2017, 2016, 2015, 2014, 2013, 2012, and 2011
- **SOLIDWORKS® 2018 Quick Start with video instruction**
- **SOLIDWORKS® 2017 in 5 Hours with video instruction**, 2016, 2015, and 2014
- **SOLIDWORKS® 2018 Tutorial with video instruction**, 2017, 2016, 2015, 2014, 2013, 2012, 2011, 2010, 2009, 2008, 2007, 2006, 2005, 2004, and 2003
- **Drawing and Detailing with SOLIDWORKS® 2014**, 2012, 2010, 2009, 2008, 2007, 2006, 2005, 2004, 2003, and 2002
- **Official Certified SOLIDWORKS® Professional (CSWP) Certification Guide with video instruction, Version 4: 2015 - 2017**, Version 3: 2012 - 2014, Version 2: 2012 - 2013, Version 1: 2010 - 2010
- **Official Guide to Certified SOLIDWORKS® Associate Exams: CSWA, CSDA, CSWSA-FEA Version 3: 2015 - 2017**, Version 2: 2012 - 2015, Version 1: 2012 - 2013

- **Assembly Modeling with SOLIDWORKS® 2012**, 2010, 2008, 2006, 2005-2004, 2003, and 2001Plus

- **Applications in Sheet Metal Using Pro/SHEETMETAL & Pro/ENGINEER**

Acknowledgements

Writing this book was a substantial effort that would not have been possible without the help and support of my loving family and of my professional colleagues. I would like to thank Professor John M. Sullivan Jr., Professor Jack Hall and the community of scholars at Worcester Polytechnic Institute who have enhanced my life, my knowledge and helped to shape the approach and content to this text.

The author is greatly indebted to my colleagues from Dassault Systèmes SOLIDWORKS Corporation for their help and continuous support: Avelino Rochino and Mike Puckett.

Thanks also to Professor Richard L. Roberts of Wentworth Institute of Technology, Professor Dennis Hance of Wright State University, Professor Jason Durfess of Eastern Washington University and Professor Aaron Schellenberg of Brigham Young University - Idaho who provided vision and invaluable suggestions.

SOLIDWORKS certification has enhanced my skills and knowledge and that of my students. Thank you to Ian Matthew Jutras (CSWE) who is a technical contributor and the creator of the videos and Stephanie Planchard, technical procedure consultant.

Contact the Author

We realize that keeping software application books current is imperative to our customers. We value the hundreds of professors, students, designers, and engineers that have provided us input to enhance the book. Please contact me directly with any comments, questions or suggestions on this book or any of our other SOLIDWORKS books at dplanchard@msn.com or planchard@wpi.edu.

Note to Instructors

Please contact the publisher **www.schroff.com** for classroom support materials (.ppt presentations, labs and more) and the Instructor's Guide with model solutions and tips that support the usage of this text in a classroom environment.

Trademarks, Disclaimer and Copyrighted Material

Additional information references the American Welding Society, AWS 2.4:1997 Standard Symbols for Welding, Braising, and Non-Destructive Examinations, Miami, Florida, USA.

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During the initial SOLIDWORKS installation, you are requested to select either the ISO or ANSI drafting standard. ISO is typically a European drafting standard and uses First Angle Projection. The book is written using the ANSI (US) overall drafting standard and Third Angle Projection for drawings.

Screen shots in the book were made using SOLIDWORKS 2018 SP0 running Windows® 10.

 Redeem the code on the inside cover of the book. View the provided videos and models to enhance the user experience. All templates, logos and model documents along with additional support materials are available.

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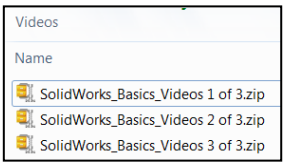
Index **I-1**

Redeem the code on the inside cover of the book. View the provided videos and models to enhance the user experience. The book provides access to over 260 models, their solutions and additional support materials.



Command Syntax

The following command syntax is used throughout the text. Commands that require you to perform an action are displayed in **Bold** text.

Format:	Convention:	Example:
Bold	• All commands actions. • Selected icon button. • Selected geometry: line, circle. • Value entries.	• Click **Options** from the Menu bar toolbar. • Click the **Extruded Boss/Base** feature. • Click **Corner Rectangle** from the Consolidated Sketch toolbar. • Click the **centerpoint**. • Enter **3.0** for Radius.
Capitalized	• Filenames. • First letter in a feature name.	• Save the **Flashlight** assembly. • Click the **Fillet** feature.

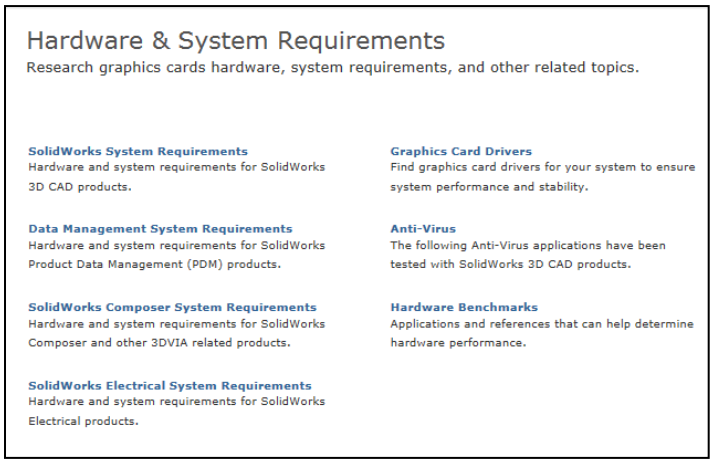
Windows Terminology in SOLIDWORKS

The mouse buttons provide an integral role in executing SOLIDWORKS commands. The mouse buttons execute commands, select geometry, display Shortcut menus and provide information feedback. A summary of mouse button terminology is displayed below:

Item:	Description:
Click	Press and release the left mouse button.
Double-click	Double press and release the left mouse button.
Click inside	Press the left mouse button. Wait a second, and then press the left mouse button inside the text box. Use this technique to modify Feature names in the FeatureManager design tree.
Drag/Drop	Point to an object, press and hold the left mouse button down. Move the mouse pointer to a new location. Release the left mouse button.
Right-click	Press and release the right mouse button. A Shortcut menu is displayed. Use the left mouse button to select a menu command.
Tool Tip	Position the mouse pointer over an Icon (button). The tool name is displayed below the mouse pointer.

A mouse with a center wheel provides additional functionality in SOLIDWORKS. Roll the center wheel downward to enlarge the model in the Graphics window. Hold the center wheel down. Drag the mouse in the Graphics window to rotate the model.

Visit SOLIDWORKS website: http://www.SOLIDWORKS.com/sw/support/hardware.html to view their supported operating systems and hardware requirements.



The book is designed to expose the new user to numerous tools and procedures. It may not always use the simplest and most direct process.

The book does not cover starting a SOLIDWORKS session in detail for the first time. A default SOLIDWORKS installation presents you with several options. For additional information for an Education Edition, visit the following site: http://www.SOLIDWORKS.com/sw/engineering-education-software.htm

The Instructor's information contains over 45 classroom presentations, along with helpful hints, What's new, sample quizzes, avi files of assemblies, projects, and all initial and final SOLIDWORKS model files.