**INTRODUCTION**

**SolidWorks Tutorial with Video Instruction** is target towards a technical school, two year college, four year university or industry professional that is a beginner or intermediate CAD user. The text provides a student who is looking for a step-by-step project based approach to learning SolidWorks with an enclosed 1.5 hour video instruction DVD, SolidWorks model files, and preparation for the CSWA exam.

The book is divided into two sections. Chapters 1 - 7 explore the SolidWorks User Interface and CommandManager, Document and System properties, simple machine parts, simple and complex assemblies, design tables, configurations, multi-sheet, multi-view drawings, BOMs, Revision tables using basic and advanced features along with Intelligent Modeling Techniques, SustainabilityXpress, SimulationXpress and DFMXpress.

Chapters 8 - 11 prepare you for the new Certified SolidWorks Associate Exam (CSWA). The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles.

Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. 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. Learn by doing, not just by reading!

Desired outcomes and usage competencies are listed for each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SolidWorks in industry.

**LINKAGE Assembly**
*Courtesy of Gears Educational Systems*
About the Cover

Create the final ROBOT assembly illustrated on the cover. The physical components and corresponding Science, Technology, Engineering, and Math (STEM) curriculum are available from Gears Educational Systems www.gearseds.com. All assemblies and components for the final ROBOT assembly are located on the DVD under the Chapter 5 model folder.

About the Authors

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 and degrees. He holds five U.S. patents and one international patent. 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, CCNA, CCNP, CSWA, and CSWP. David is a SolidWorks Solution Partner, an Adjunct Faculty member and the SAE advisor at Worcester Polytechnic Institute in the Mechanical Engineering department.

Marie Planchard is the Director of World Education Markets at DS SolidWorks Corp. Before she joined SolidWorks, Marie spent over 10 years as an engineering professor at Mass Bay College in Wellesley Hills, MA. She has 14 plus years of industry software experience and held a variety of management and engineering positions. Marie holds a BSME, MSME and a Certified SolidWorks Professional (CSWP) Certification. She is an active member of the American Society of Mechanical Engineers (ASME) and the American Society for Engineering Education (ASEE).

David and Marie Planchard are co-authors of the following books:

- **A Commands Guide Reference Tutorial for SolidWorks® 2007**
SolidWorks 2012 Tutorial

Introduction


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

Acknowledgments

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 Sullivan and Robert Norton 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 book.

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Thanks also to Professor Richard L. Roberts of Wentworth Institute of Technology, Professor Dennis Hance of Wright State University, and Professor Jason Durfess of Eastern Washington University who provided insight and invaluable suggestions.

Finally to my wife, who is infinitely patient for her support and encouragement and to our loving daughter Stephanie who supported me during this intense and lengthy project.

Contact the Authors

This is the 10th edition of this book. 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 our book. We value your suggestions and comments. Please visit our website at www.dmeducation.net or contact
us 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

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The physical components and corresponding Science, Technology, Engineering, and Math (STEM) curriculum are available from Gears Educational Systems www.gearseds.com. Additional information (four chapters) on the SolidWorks Certified Associate CSWA exam is provided at the end of the book, to assist the user to take and pass the CSWA exam (a recognized industry CAD standard).

Please contact the publisher www.schroff.com for additional classroom support materials: PowerPoint presentations, Adobe files along with avi files, term projects, quizzes with initial and final SolidWorks models and tips that support the usage of this text in a classroom environment.

Trademarks, Disclaimer and Copyrighted Material

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

References

- ASME Y14 Engineering Drawing and Related Documentation Practices
- Jensen, Cecil, Interpreting Engineering Drawings, Glencoe, 2002
- Planchard & Planchard, Drawing and Detailing with SolidWorks, SDC Pub., Mission, KS 2010
- Walker, James, Machining Fundamentals, Goodheart Wilcox, 1999
- 80/20 Product Manual, 80/20, Inc., Columbia City, IN, 2009
- Simpson Strong Tie Product Manual, Simpson Strong Tie, CA, 2009
- SMC Corporation of America, Product Manuals, Indiana, 2011
- Emhart - A Black and Decker Company, On-line catalog, Hartford, CT, 2009

Every license of SolidWorks contains a copy of SolidWorks SustainabilityXpress. SustainabilityXpress calculates environmental impact on a model in four key areas: Carbon Footprint, Energy Consumption, Air Acidification and Water Eutrophication. Material and Manufacturing process region and Transportation Usage region are used as input variables.

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.
All templates, logos, and needed models for this book are included on the enclosed DVD. Copy the model folders from the DVD to your local hard drive. Work from your local hard drive. View the 1.5 hour Video Instruction DVD for additional help.

Additional projects are included in the exercise section of Chapter 5. Copy the components from the Chapter 5 Homework folder located on the DVD. View all components. Create an ANSI Assembly document.

Insert and create all needed components and mates to assemble the assembly and to simulate proper movement per the provided avi file located in the folder.
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The Instructors DVD contains PowerPoint presentations, Adobe files along with avi files, Term Projects, quizzes with initial and final SolidWorks models.
What is SolidWorks?

SolidWorks® is a mechanical design automation software package used to build parts, assemblies and drawings that takes advantage of the familiar Microsoft® Windows graphical user interface.

SolidWorks is an easy to learn design and analysis tool, (SolidWorks SimulationXpress, SolidWorks Motion, SolidWorks Flow Simulation, etc.) which makes it possible for designers to quickly sketch 2D and 3D concepts, create 3D parts and assemblies and detail 2D drawings.

In SolidWorks, you create 2D and 3D sketches, 3D parts, 3D assemblies and 2D drawings. The part, assembly and drawing documents are related. Additional information on SolidWorks and its family of products can be obtained at their URL, www.SolidWorks.com.
Features are the building blocks of parts. Use features to create parts, such as: Extruded Boss/Base and Extruded Cut. Extruded features begin with a 2D sketch created on a Sketch plane.

The 2D sketch is a profile or cross section. Sketch tools such as: lines, arcs and circles are used to create the 2D sketch. Sketch the general shape of the profile. Add Geometric relationships and dimensions to control the exact size of the geometry.

Create features by selecting edges or faces of existing features, such as a Fillet. The Fillet feature rounds sharp corners.

Dimensions drive features. Change a dimension, and you change the size of the part.

Apply Geometric relationships: Vertical, Horizontal, Parallel, etc. to maintain Design intent.

Create a hole that penetrates through a part. SolidWorks maintains relationships through the change.

The step-by-step approach used in this text allows you to create parts, assemblies and drawings by doing, not just by reading.

The book provides the knowledge to modify all parts and components in a document. Change is an integral part of design.
Design Intent

What is design intent? All designs are created for a purpose. Design intent is the intellectual arrangements of features and dimensions of a design. Design intent governs the relationship between sketches in a feature, features in a part and parts in an assembly.

The SolidWorks definition of design intent is the process in which the model is developed to accept future modifications. Models behave differently when design changes occur.

Design for change! Utilize geometry for symmetry, reuse common features, and reuse common parts. Build change into the following areas that you create:

- Sketch
- Feature
- Part
- Assembly
- Drawing

💡 When editing or repairing geometric relations, it is considered best practice to edit the relation vs. deleting it.

Design Intent in a sketch

Build design intent in a sketch as the profile is created. A profile is determined from the Sketch Entities. Example: Rectangle, Circle, Arc, Point, Slot etc. Apply symmetry into a profile through a sketch centerline, mirror entity and position about the reference planes and Origin. Always know the location of the Origin in the sketch.

Build design intent as you sketch with automatic Geometric relations. Document the decisions made during the up-front design process. This is very valuable when you modify the design later.

A rectangle (Center Rectangle Sketch tool) contains Horizontal, Vertical and Perpendicular automatic Geometric relations.

Apply design intent using added Geometric relations if needed. Example: Horizontal, Vertical, Collinear, Perpendicular, Parallel, Equal etc.
Example A: Apply design intent to create a square profile. Sketch a rectangle. Apply the Center Rectangle Sketch tool. Note: No construction reference centerline or Midpoint relation is required with the Center Rectangle tool. Insert dimensions to fully define the sketch.

Example B: If you have a hole in a part that must always be $16.5\text{mm} \leq$ from an edge, dimension to the edge rather than to another point on the sketch. As the part size is modified, the hole location remains $16.5\text{mm} \leq$ from the edge as illustrated.

**Design intent in a feature**

Build design intent into a feature by addressing End Conditions (Blind, Through All, UpToVertex, etc.) symmetry, feature selection, and the order of feature creation.

Example A: The Extruded Base feature remains symmetric about the Front Plane. Utilize the Mid Plane End Condition option in Direction 1. Modify the depth, and the feature remains symmetric about the Front Plane.

Example B: Create 34 teeth in the model. Do you create each tooth separate using the Extruded Cut feature? No. Create a single tooth and then apply the Circular Pattern feature. Modify the Circular Pattern from 32 to 24 teeth.

**Design intent in a part**

Utilize symmetry, feature order and reusing common features to build design intent into a part. Example A: Feature order. Is the entire part symmetric? Feature order affects the part.

Apply the Shell feature before the Fillet feature and the inside corners remain perpendicular.
**Design intent in an assembly**

Utilizing symmetry, reusing common parts and using the Mate relation between parts builds the design intent into an assembly.

Example A: Reuse geometry in an assembly. The assembly contains a linear pattern of holes. Insert one screw into the first hole. Utilize the Component Pattern feature to copy the machine screw to the other holes.

**Design intent in a drawing**

Utilize dimensions, tolerance and notes in parts and assemblies to build the design intent into a drawing.

Example A: Tolerance and material in the drawing. Insert an outside diameter tolerance +.000/- .002 into the TUBE part. The tolerance propagates to the drawing.

Define the Custom Property Material in the Part. The Material Custom Property propagates to your drawing.

Create a sketch on any of the default planes: Front, Top, Right or a created plane.

Additional information on design process and design intent is available in SolidWorks Help.

The book is design to expose the new user to many tools, techniques and procedures. It may not always use the most direct tool or process.

Every license of SolidWorks contains a copy of SolidWorks SustainabilityXpress. SustainabilityXpress calculates environmental impact on a model in four key areas: *Carbon Footprint, Energy Consumption, Air Acidification and Water Eutrophication*. Material and Manufacturing process region and Transportation Usage region are used as input variables.
Overview of Chapters

Chapter 1: Linkage Assembly

Chapter 1 introduces the basic concepts behind SolidWorks and the SolidWorks 2012 User Interface.

Create a file folder to manage projects. Create three parts: AXLE, SHAFT-COLLAR, and FLATBAR. Utilize the following features: Extruded Boss/Base, Extruded Cut and Linear Pattern.

Create the LINKAGE assembly. The LINKAGE assembly utilizes the SMC AirCylinder component located on the enclosed DVD in the book. Note: Copy all SolidWorks files from the DVD to your hard drive. Work from your hard drive.

Chapter 2: Front Support Assembly

Chapter 2 introduces various Sketch planes to create parts. The Front, Top and Right Planes each contain the Extruded Boss/Base feature for the TRIANGLE, HEX-STANDOFF and ANGLE-13HOLE parts.

Utilize Geometric relationships in your sketch.

Create the SCREW part using the following features: Revolved Base, Extruded Cut, Fillet and Circular Pattern.

Create the FRONT-SUPPORT assembly. Utilize additional parts from the Web or the enclosed DVD to create the RESERVOIR SUPPORT assembly in the Chapter exercises.
Chapter 3: Fundamentals of Drawing

Chapter 3 covers the development of a customized Sheet format and Drawing template.

Review the differences between the Sheet and the Sheet format. Develop a company logo from a bitmap or picture file.

Create a FLATBAR drawing. Insert dimensions created from the part features. Create the LINKAGE assembly drawing with multiple views and assemblies.

Develop and incorporate a Bill of Materials into the drawing Custom Properties in the parts and assemblies. Add information to the Bill of Materials in the drawing. Insert a Design Table to create multiple configurations of parts and assemblies.

Chapter 4: Advanced Features

Chapter 4 focuses on creating six parts for the PNEUMATIC-TEST-MODULE Assembly: WEIGHT, HOOK, WHEEL, HEX-ADAPTER, AXLE-3000 and SHAFTCOLLAR-500.

Apply the following Advanced model features: Plane, Lofted Base, Extruded Cut, Swept Base, Dome, Helix and Spiral, Swept Cut, Extruded Boss/Base, Revolved Cut, Extruded Cut, Circular Pattern, Axis and Hole Wizard.

Reuse existing geometry and modify existing parts to create new parts with the Save as copy command.
Chapter 5: PNEUMATIC-TEST-MODULE Assembly and Final ROBOT Assembly

Chapter 5 focuses on the PNEUMATIC-TEST-MODULE Assembly and the final ROBOT Assembly.

Create the WHEEL-AND-AXLE assembly. First, create the 3HOLE-SHAFTCOLLAR assembly and the 5HOLE-SHAFTCOLLAR assembly.

Insert the WHEEL part, AXLE 3000 part, HEX-ADAPTER part and SHAFTCOLLAR-500 part.

Insert the FLAT-PLATE part that was create in the Chapter 2 exercises. Insert the LINKAGE assembly and add components: HEX-STANDOFF, AXLE and SHAFT-COLLAR.

Insert the AIR-RESERVOIR-SUPPORT assembly. Insert the SCREW part. Utilize the Feature Driven Component Pattern tool and the Linear Component Pattern tool.

Insert the FRONT-SUPPORT assembly and apply the Mirror Components tool to complete the Pneumatic Test Module Assembly.

Create the final ROBOT Assembly as illustrated with the Robot-platform sub-assembly, PNEUMATIC-TEST-MODULE sub-assembly, basic_integration sub-assembly and the HEX-ADAPTER component. Add additional components in the Chapter exercises.

Learn the process to work with multiple documents between parts and assemblies and to apply the following Assembly tools: Insert Component, Standard Mates: Concentric, Coincident, and Parallel, Linear Component Pattern, Feature Driven Component Pattern, Circular Component Pattern, Mirror Components and Replace Components.

Note: All assemblies and components for the final ROBOT assembly are located on the DVD under the Chapter 5 models folder.
Additional projects are included in the exercise section of this chapter. Copy the components from the Chapter 5 Homework folder located on the DVD. View all components.

Create an ANSI Assembly document. Insert and create all needed components and mates to assemble the assembly and to simulate proper movement per the avi file located in the folder.
Chapter 6: SolidWorks SimulationXpress, Sustainability, and DFMXpress

Chapter 6 introduces three general SolidWorks analysis tools: SimulationXpress, SustainabilityXpress and DFMXpress.

Execute a SolidWorks SimulationXpress analysis on a part. Determine if the part can support an applied load under a static load condition.

Perform a SustainabilityXpress analysis on a part. View the environmental impact calculated in four key areas: Carbon Footprint, Energy Consumption, Air Acidification and Water Eutrophication.

Material, Manufacturing process region and Usage region are used as input variables.

Compare similar materials and environmental impacts.

Implement DFMXpress on a part. DFMXpress is an analysis tool that validates the manufacturability of SolidWorks parts. Use DFMXpress to identify design areas that may cause problems in fabrication or increase the costs of production.

💡 The book is designed to expose the new user to many tools, techniques and procedures. It may not always use the most direct tool or process.
Chapter 8 - 11: Introduction to the Certified SolidWorks Associate Exam

DS SolidWorks Corp. offers various stages of certification representing increasing levels of expertise in 3D CAD design as it applies to engineering: Certified SolidWorks Associate CSWA, Certified SolidWorks Professional CSWP and Certified SolidWorks Expert CSWE along with specialty fields in Simulation, Sheet Metal, and Surfacing.

The CSWA Certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. The main requirement for obtaining the CSWA certification is to take and pass the on-line proctored 180 minute exam (minimum of 165 out of 240 points). The new CSWA exam consists of fourteen questions in the following categories and subject areas:

- **Drafting Competencies**: (Three questions - multiple choice - 5 points each).
- **Basic Part Creation and Modification**: (Two questions - one multiple choice / one single answer - 15 points each).
- **Intermediate Part Creation and Modification**: (Two questions - one multiple choice / one single answer - 15 points each).
- **Advanced Part Creation and Modification**: (Three questions - one multiple choice / two single answers - 15 points each).
- **Assembly Creation and Modification**: (Two different assemblies - four questions - two multiple choice / two single answers - 30 points each).

Note: Download the needed components during the exam for create the assembly.

A total score of 165 out of 240 or better is required to obtain your CSWA Certification.

💡 For detail exam information see the Official Certified SolidWorks Associate Examination Guide book. The primary goal of this book is not only to help you pass the CSWA exam, but also to ensure that you understand and comprehend the concepts and implementation details of the CSWA process.
About the Book

You will find a wealth of information in this book. The book is a project based - step-by-step text written for new and intermediate users. The following conventions are used throughout this book:

- The term document refers to a SolidWorks part, drawing or assembly file.
- The list of items across the top of the SolidWorks interface is the Menu bar menu or the Menu bar toolbar. Each item in the Menu bar 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 Menu bar. The Plane PropertyManager is displayed.
- The book is organized into Chapters. Each Chapter is focused on a specific subject or feature. Use the enclosed DVD to obtain parts and models that are used in this book and to view the features created in each Chapter.
- 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.
- Copy all folders and files from the DVD to your hard drive. Work from the hard drive. All assemblies and components for the final ROBOT assembly are located on the DVD in the Chapter 5 Models folder.

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

<table>
<thead>
<tr>
<th>Format:</th>
<th>Convention:</th>
<th>Example:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Bold</strong></td>
<td>• All commands actions.</td>
<td>• Click <strong>Options</strong> from the Menu bar toolbar.</td>
</tr>
<tr>
<td></td>
<td>• Selected icon button.</td>
<td>• Click <strong>Corner Rectangle</strong> from the Sketch toolbar.</td>
</tr>
<tr>
<td></td>
<td>• Selected icon button.</td>
<td>• Click <strong>Sketch</strong> from the Context toolbar.</td>
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<tr>
<td></td>
<td>• Selected geometry: line, circle.</td>
<td>• Select the <strong>centerpoint</strong>.</td>
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<tr>
<td></td>
<td>• Value entries.</td>
<td>• Enter <strong>3.0</strong> for Radius.</td>
</tr>
<tr>
<td><strong>Capitalized</strong></td>
<td>• Filenames.</td>
<td>• Save the <strong>FLATBAR</strong> assembly.</td>
</tr>
<tr>
<td></td>
<td>• First letter in a feature name.</td>
<td>• Click the <strong>Fillet</strong> feature.</td>
</tr>
</tbody>
</table>
**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:

<table>
<thead>
<tr>
<th>Item:</th>
<th>Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Click</td>
<td>Press and release the left mouse button.</td>
</tr>
<tr>
<td>Double-click</td>
<td>Double press and release the left mouse button.</td>
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<tr>
<td>Click inside</td>
<td>Press the left mouse button. Wait a second, and then press the left mouse</td>
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<tr>
<td></td>
<td>button inside the text box. Use this technique to modify Feature names in</td>
</tr>
<tr>
<td></td>
<td>the FeatureManager design tree.</td>
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<tr>
<td>Drag</td>
<td>Point to an object, press and hold the left mouse button down. Move the</td>
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<tr>
<td></td>
<td>mouse pointer to a new location. Release the left mouse button.</td>
</tr>
<tr>
<td>Right-click</td>
<td>Press and release the right mouse button. A Shortcut menu is displayed. Use</td>
</tr>
<tr>
<td></td>
<td>the left mouse button to select a menu command.</td>
</tr>
<tr>
<td>ToolTip</td>
<td>Position the mouse pointer over an Icon (button). The tool name is displayed</td>
</tr>
<tr>
<td></td>
<td>below the mouse pointer.</td>
</tr>
<tr>
<td>Large ToolTip</td>
<td>Position the mouse pointer over an Icon (button). The tool name and a</td>
</tr>
<tr>
<td></td>
<td>description of its functionality are displayed below the mouse pointer.</td>
</tr>
<tr>
<td>Mouse pointer feedback</td>
<td>Position the mouse pointer over various areas of the sketch, part,</td>
</tr>
<tr>
<td></td>
<td>assembly or drawing. The cursor provides feedback depending on the geometry.</td>
</tr>
</tbody>
</table>

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 Instructors DVD contains PowerPoint presentations, Adobe files along with avi files, Term Projects, quizzes with the initial and final SolidWorks models.

The book is design 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 sites:
http://www.solidworks.com/goedu and