SOLIDWORKS® 2018 Quick Start with video instruction introduces the new user to the basics of using SOLIDWORKS 3D CAD software in 5 easy lessons. The book is intended for the student or designer that needs to learn SOLIDWORKS quickly and effectively for senior capstone, machine design, kinematics, dynamics, and other engineering and technology projects that use SOLIDWORKS as a tool.

Engineers in industry who are expected to have SOLIDWORKS skills for their company’s next project. Students who need to learn SOLIDWORKS without taking a formal CAD course.

Desired outcomes and usage competencies are listed for each chapter. The book is divided into three sections with 7 chapters.

Chapter 1 - 5: Explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, configurations, multi-view drawings, and BOMs using basic and advanced features.

Develop a mini Stirling Engine and investigate the proper design intent and constraints. Investigate view orientation based on the Front, Top and Right planes. As you sketch, use tools such as the circle, line, centerline, slot and mirror. Modify sketches and sketch planes. Learn to set document properties, identify sketch states and insert geometric relations and dimensions along with applying proper design intent and technique.
Parts are made up of features. With features you add or remove material. You apply and edit the Extruded Boss/Base, Revolved Boss/Base, Extruded Cut, Circular Pattern and Fillet feature. You explore the Hole Wizard feature with manufacturing parameters and the Mass Properties, Measure, and Appearance tool.

Assemblies are made up of components and sub-assemblies. Incorporate a series of provided parts and your own part to create two assemblies utilizing the Bottom-up approach.

Utilize the following assembly tools: Insert Component, Mate, Hide, Show, Rotate, Move, Modify, Flexible, Ridge, Multiple mate and more. Learn how to add constraints that result in dynamic behavior of the assembly such as linear translation and rotation.

Before you machine or create a rapid prototype of a part or assembly, you should verify the model for clearance, interference, static and dynamic behavior. Verify behavior between the Power Piston, Power Clevis, Connecting Rod and Handle with design modifications using the Stirling Engine Modified assembly.

Apply the Assembly Visualization tool to sort components by mass while creating a motion study (animation) for a formal presentation.

Chapter 6: Review the Certified Associate - Mechanical Design (CSWA) program. Understand the curriculum and categories of the CSWA exam and the required model knowledge needed to successfully take the exam.

Exam categories are: Drafting Competencies, Basic Part Creation and Modification, Intermediate Part Creation and Modification, Advanced Part Creation and Modification, and Assembly Creation and Modification.

Download the CSWA Sample Exam folder. Follow the instructions to login and take a sample exam.
Chapter 7: Comprehend the Differences of Additive vs. Subtractive Manufacturing. Understand 3D printer terminology along with a working knowledge of preparing, saving, and printing a 3D CAD model on a low cost ($500 - $3,000) printer.

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

Redeem the code on the inside cover of the book. Download the SOLIDWORKS 2018 folder and videos to your local hard drive. View the provided videos 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 book is designed to complement the SOLIDWORKS Tutorials contained in SOLIDWORKS 2018.

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, MS 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 Quick Start with video instruction**

- **SOLIDWORKS® 2017 in 5 Hours with video instruction**, 2016, 2015, and 2014


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

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**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 our book. We value your suggestions and comments. Please contact me with any comments, questions or suggestions on this book or any of our other SOLIDWORKS books. David Planchard, D & M Education, LLC, dplanchard@msn.com or planchard@wpi.edu.

**Note to Instructors**

Please contact the publisher [www.SDCpublications.com](http://www.SDCpublications.com) for additional classroom support materials: PowerPoint presentations, Adobe files along with avi files, additional design projects, quizzes with initial and final SOLIDWORKS models and tips that support the usage of this text in a classroom environment.

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References

- SOLIDWORKS Users Guide, SOLIDWORKS Corporation, 2018
- ASME Y14 Engineering Drawing and Related Documentation Practices
- Lockhart & Johnson, *Engineering Design Communications*, Addison Wesley, 1999
- Olivo C., Payne, Olivo, T, *Basic Blueprint Reading and Sketching*, Delmar, 1988

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.
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Chapter 1: Overview of SOLIDWORKS and the User Interface

SOLIDWORKS is a design software application used to create 2D and 3D sketches, 3D parts and assemblies and 2D drawings.

Chapter 1 introduces the user to the SOLIDWORKS User Interface (UI) and CommandManager: Menu bar toolbar, Menu bar menu, Drop-down menus, Context toolbars, Consolidated drop-down toolbars, System feedback icons, Confirmation Corner, Heads-up View toolbar, Document Properties and more.

How do you start a SOLIDWORKS session? How do you open a new or existing part? How do you start a model in SOLIDWORKS? What is design intent?

Chapter 2: 2D Sketching, Features and Parts

Learn about 2D Sketching and 3D features. Create a new part called Wheel with user defined document properties.

Create the Wheel for the Fly Wheel sub-assembly. Utilize the Fly Wheel sub-assembly in the final Stirling Engine assembly.

Apply the following sketch and feature tools: Circle, Line Centerline, Centerpoint Straight Slot, Mirror Entities, Extruded Boss, Extruded Cut, Revolved Boss, Circular Pattern, Hole Wizard and Fillet.

Incorporate design change into a part using proper design intent, along with applying multiple geometric relations: Coincident, Vertical, Horizontal, Tangent and Midpoint and feature and sketch modifications.

Utilize the Material, Mass Properties and Appearance tool on the Wheel.
Chapter 3: Assembly Modeling - Bottom-up method

Learn about the Bottom-up assembly method and create two new assemblies with user defined document properties:

- Fly Wheel.
- Stirling Engine.

Insert the following Standard and Quick mate types: Coincident, Concentric, Distance and Tangent.

Utilize the following assembly tools: Insert Component, Suppress, Un-suppress, Mate, Move Component, Rotate Component, Interference Detection, Hide, Show, Flexible, Ridge, and Multiple mate mode.

Create an Exploded View with animation.

Apply the Measure and Mass Properties tool to modify a component in the Stirling Engine assembly.

Chapter 4: Design Modifications

Address clearance, interference, static and dynamic behavior of the Stirling Engine Modified assembly.

Verify the behavior between the following components: Power Piston, Power Clevis, Connecting Rod and Handle in the assembly.

Apply the following assembly tools: Move, Rotate, Collision Detection, Interference Detection, Selected Components, Edit Feature and Center of Mass.

Utilize the Assembly Visualization tool on the Stirling Engine assembly and sort by component mass.

Create a new Coordinate System on the Stirling Engine assembly relative to the default origin.

Run a Motion Study and save the Motion Study AVI file.
Chapter 5: Drawing and Dimensioning Fundamentals

Learn about Drawing and Dimension Fundamentals and create two new drawings with user defined document properties:

- Fly Wheel Assembly.
- Bushing.

Create the Fly Wheel Assembly drawing with an Exploded Isometric view.

Utilize a Bill of Materials, Magnetic lines and Balloons.

Learn about Custom Properties and the Title Block.

Create the Bushing Part drawing utilizing Third Angle Projection with two standard Orthographic views: Front, Top and an Isometric view.

Address imported dimensions from the Model Items tool.

Insert additional dimensions using the Smart Dimension tool along with all needed annotations.

Chapter 6: Introduction to the Certified Associate – Mechanical Design (CSWA) Exam

Chapter 6 provides a basic introduction into the curriculum and categories of the Certified Associate - Mechanical Design (CSWA) exam. Awareness to the exam procedure, process, and required model knowledge needed to take the CSWA exam.
The five exam categories are:

- Drafting Competencies.
- Basic Part Creation and Modification.
- Intermediate Part Creation and Modification.
- Advanced Part Creation and Modification.
- Assembly Creation and Modification.

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 two part on-line proctored exams.

This first exam (Part 1) is 90 minutes, minimum passing score is 80, with 6 questions.

The second exam (Part 2) is 90 minutes, minimum passing score is 80 with 8 questions.

**Chapter 7: Additive Manufacturing - 3D Printing**

Provide a basic understanding between the differences of Additive vs. Subtractive Manufacturing. Comprehend 3D printer terminology along with a working knowledge of preparing, saving, and printing a 3D CAD model on a low cost ($500 - $2,000) printer.

On the completion of this chapter, you will be able to:

- Discuss Additive vs Subtractive Manufacturing.
- Determine the differences between a Cartesian printer and a Delta printer.
- Create a STereoLithography (STL) file in SOLIDWORKS.
- 3D print directly from SOLIDWORKS using an Add-In.
  - Save an STL file to G-code.
- Discuss printer hardware.
- Select the correct filament type:
  - PLA (Polylactic acid), ABS (Acrylonitrile butadiene styrene) or Nylon.
• Prepare the G-code.
  o Address model setup, print orientation, extruder temperature, and bed temperature.

• Comprehend the following 3D printer terminology:
  o (STereoLithography) file - STL.
  o Fused Filament Fabrication - FFF.
  o Fused Deposition Model - FDM.
  o Digital Light Process - DLP.
  o Dissolvable Support System - DDS.
  o Fast Layer Deposition - FLD.
  o Raft, Skirt and Brim.
  o Support and Touching Buildplate.
  o Slicer, G-code.

• Address fit tolerance for interlocking parts.
• Define general 3D Printing tips.

**Book Layout**

The following conventions are used throughout this book:

• The term document is used to refer 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 **View, Hide/Show**, check **Origins** from the Menu bar. The Origins are displayed in the Graphics window.

• The ANSI overall drafting standard and Third Angle projection is used as the default setting in this text. MMGS (millimeter, gram, second) unit system is used.

• The book is organized into various chapters. Each chapter is focused on a specific subject or feature.
• All templates, logos and needed model documents for this book are provided. Redeem your code on the inside cover of the book. View the videos and models to enhance the user experience. Download the SOLIDWORKS 2018 folder to your local hard drive. Work from your local hard drive.

• Additional exercises are provided. Download the SOLIDWORKS 2018 folder to your hard drive. Work from your hard drive.

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

View the provided videos in this book to enhance the user experience.

• Start a SOLIDWORKS session.
• Understand the SOLIDWORKS Interface.
• Create 2D Sketches, Sketch Planes and utilize various Sketch tools.
• Create 3D Features and apply Design Intent.
• Create an Assembly.
• Create fundamental Drawings Part 1 & Part 2.
The following command syntax is used throughout the text. Commands that require you to perform an action are displayed in **Bold** text.

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**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><strong>Click</strong></td>
<td>Press and release the left mouse button.</td>
</tr>
<tr>
<td><strong>Double-click</strong></td>
<td>Double press and release the left mouse button.</td>
</tr>
<tr>
<td><strong>Click inside</strong></td>
<td>Press the left mouse button. Wait a second, and then press the left mouse</td>
</tr>
<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>
</tr>
<tr>
<td><strong>Drag</strong></td>
<td>Point to an object, press and hold the left mouse button down. Move the</td>
</tr>
<tr>
<td></td>
<td>mouse pointer to a new location. Release the left mouse button.</td>
</tr>
<tr>
<td><strong>Right-click</strong></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><strong>Tool Tip</strong></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><strong>Large Tool Tip</strong></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><strong>Mouse pointer feedback</strong></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](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](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.