SOLIDWORKS® 2018 Tutorial with video instruction is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text provides a step-by-step project based learning approach featuring machined components with additional semester design projects.

Desired outcomes and usage competencies are listed for each chapter. The book is divided into four sections with 11 Chapters.

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

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, apply proper design intent, design tables, and configurations.

**Chapter 6:** Create the final ROBOT Assembly. The physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components for the final ROBOT assembly are provided.
Chapter 7 - Chapter 10: Prepare for the Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles.

Each chapter addresses one of the five categories in the CSWA exam: Drafting Competencies, Basic Part Creation and Modification, Intermediate Part Creation and Modification, Advanced Part Creation and Modification, Assembly Creation and Modification.

Chapter 11: 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.

Redeem your code on the inside cover of the book. Download the models and videos to your hard drive. View the provided videos to enhance the user experience: Start a SOLIDWORKS session, Understand the SOLIDWORKS User Interface, Create 2D Sketches, Utilize various Sketch tools, Create 3D Features, Apply Design Intent, Create an Assembly and Create fundamental Drawings Part 1 & Part 2.

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 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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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 dplanchar@msn.com or planchar@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.

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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

- Walker, James, Machining Fundamentals, Goodheart Wilcox, 1999.
- SMC Corporation of America, Product Manuals, Indiana, USA, 2012.

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 your code on the inside cover of the book. All templates, logos and model documents along with additional support materials for this book are available. View the video instruction files for additional information.
Additional semester design projects are included in the exercise section of Chapter 6. Copy the components from the Chapter 6 Homework folder. 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.
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- Move views and Properties of the Sheet
- FLATBAR Drawing-Position views
- Detail Drawing
- FLATBAR Drawing-Dimensions and Annotations
- FLATBAR Drawing-Part Number and Document Properties
- FLATBAR Drawing-Linked Note

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**LINKAGE Assembly Drawing-Exploded view**

**LINKAGE Assembly Drawing-Animation**

**LINKAGE Assembly Drawing-Bill of Materials**

**LINKAGE Assembly Drawing-Automatic Balloons**

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**LINKAGE Assembly Drawing-Sheet2 Section view**

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- Build an Assembly from a Detailed Dimensioned Illustration
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## Chapter 11 - Additive Manufacturing - 3D Printing

## Chapter Objective

- Additive vs. Subtractive Manufacturing
- Cartesian Printer vs. Delta Printer
- Create an STL file in SOLIDWORKS
- Print Directly from SOLIDWORKS
- Print Material
  - ABS - Storage
  - ABS - Part Accuracy
  - PLA - Storage
  - PLA - Part Accuracy
  - Nylon - Storage
  - Nylon - Part Accuracy
- Build Plate
  - Non-Heated
  - Heated
  - Clean
  - Level
  - Temperature
- Filament
- Prepare the Model
  - Example 1: Part Orientation
  - Example 2: Part Orientation
- 3D Terminology
  - Stereolithography (SL or SLA)
  - Fused Filament Fabrication (FFF)
  - Fused Deposition Fabrication (FDM)
  - Digital Light Process (DLP)
  - Raft, Skirt, Brim
  - Support, Touching Build Plate
  - Slicer Engine
  - G-code
  - Infill
  - Infill Pattern/Shape
  - Shells/Parameters
  - Lay Height
  - Influence of Percent Infill
  - Remove the Model from the Build Plate
  - Know the Printer’s Limitation
  - Tolerance for Interlocking Parts

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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.

Redeem your code on the inside cover of the book. View the provided videos to enhance the user experience: Start a SOLIDWORKS session, Understand the SOLIDWORKS User Interface, Create 2D Sketches, Sketch Planes and utilize various Sketch tools, Create 3D Features and apply Design Intent, Create an Assembly and Create fundamental Drawings Part 1 & Part 2.
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 Simulations, 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.

This 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 arrangement 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 16.5mm ≤ 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 16.5mm ≤ 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 separately 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 reuse 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.

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

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.
Overview of Chapters

Chapter 1: Overview of SOLIDWORKS and the SOLIDWORKS 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 2018 Welcome dialog box, User Interface (UI) and the 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.

Start a new SOLIDWORKS Session. Create a new part. Open an existing part and view the created features and sketches using the Rollback bar. Design the part using proper design intent.

Chapter 2: Parts and Assembly Creation.

Create three parts: AXLE, SHAFT-COLLAR and FLATBAR

Understand and apply the following Sketch tools: Circle, Smart dimension and Centerpoint Straight Slot.

Insert geometric relations and dimensions: Equal, Vertical, Horizontal, Parallel, Perpendicular, Coincident and MidPoint.

Utilize the following features: Extruded Boss/Base, Extruded Cut and Linear Pattern.

Create the LINKAGE assembly. The LINKAGE assembly utilizes the provided AirCylinder assembly. Insert the following Standard mates: Coincident, Concentric and Parallel.

Utilize the Standard and Quick mate procedure.
Chapter 3: Front Support Assembly.

Chapter 3 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 provided components to create the RESERVOIR SUPPORT assembly in the chapter exercises.

Apply the following mate types: Concentric, Coincident, Parallel, Distance, Gear and Cam.

Chapter 4: Fundamentals of Drawing.

Chapter 4 covers the development of a customized Sheet format and Drawing templates.

Review the differences between the Edit Sheet and the Edit Sheet Format modes.

Develop a company logo from a bitmap or picture file.

Create a FLATBAR drawing.

Insert dimensions created from the part features.

Create a LINKAGE assembly drawing with multiple views.

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 assembly drawing.

Insert a Design Table to create multiple configurations of parts and assemblies.
Chapter 5: Advanced Features.

Chapter 5 focuses on creating six parts for the PNEUMATIC-TEST-MODULE Assembly: WHEEL, HOOK, WHEEL, HEX-ADAPTER, AXLE-3000 and SHAFTCOLLAR-500.

Obtain the ability to reuse geometry by modifying existing parts and to create new parts.


Chapter 6: PNEUMATIC-TEST-MODULE Assembly and Final ROBOT Assembly.

Chapter 6 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 created in the Chapter 3 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 Pattern 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, Pattern Driven Component Pattern, Circular Component Pattern, Mirror Components and Replace Components.

Redeem your code on the inside cover of the book. All assemblies and components for the final ROBOT assembly are provided.
Additional projects are included in the exercise section of the chapter. Copy the components from the Chapter 6 Homework folders. 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.
Chapter 7 - 10: Introduction to the Certified Associate - Mechanical Design (CSWA) 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 Drawing, Simulation, Sheet Metal, Surfacing and more.

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.

The CSWA exam consists of 14 questions in the following five 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).

💡 Copy and open the CSWA Sample Exam folder from the SOLIDWORKS CSWA MODEL folder. Explore the provided information. Follow the instructions to login and download a sample exam.
Chapter 11: 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 - $3,000) 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.
  - Address model setup, print orientation, extruder temperature, and bed temperature.
- Comprehend the following 3D printer terminology:
  - (STereoLithography) file - STL.
  - Fused Filament Fabrication - FFF.
  - Fused Deposition Model - FDM.
  - Digital Light Process - DLP.
  - Dissolvable Support System - DDS.
  - Fast Layer Deposition - FLD.
  - Raft, Skirt and Brim.
  - Support and Touching Buildplate.
  - Slicer, G-code.
- Address fit tolerance for interlocking parts.
- Define general 3D Printing tips.
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.
- 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.
- Redeem your code on the inside cover of the book. Download all folders and files to your hard drive. Work from your hard drive. All assemblies and components for the final ROBOT assembly are located in the Chapter 6 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>Bold</td>
<td>• All commands actions.</td>
<td>• Click Options from the Menu bar toolbar.</td>
</tr>
<tr>
<td></td>
<td>• Selected icon button.</td>
<td>• Click Corner Rectangle from the Sketch toolbar.</td>
</tr>
<tr>
<td></td>
<td>• Selected icon button.</td>
<td>• Click Sketch from the Context toolbar.</td>
</tr>
<tr>
<td></td>
<td>• Selected geometry: line, circle.</td>
<td>• Select the centerpoint.</td>
</tr>
<tr>
<td></td>
<td>• Value entries.</td>
<td>• Enter 3.0 for Radius.</td>
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<tr>
<td></td>
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</tr>
<tr>
<td>Capitalized</td>
<td>• Filenames.</td>
<td>• Save the FLATBAR assembly.</td>
</tr>
<tr>
<td></td>
<td>• First letter in a feature name.</td>
<td>• Click the Fillet 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>
</tr>
<tr>
<td>Click inside</td>
<td>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.</td>
</tr>
<tr>
<td>Drag</td>
<td>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.</td>
</tr>
<tr>
<td>Right-click</td>
<td>Press and release the right mouse button. A Shortcut menu is displayed. Use the left mouse button to select a menu command.</td>
</tr>
<tr>
<td>Tool Tip</td>
<td>Position the mouse pointer over an Icon (button). The tool name is displayed below the mouse pointer.</td>
</tr>
<tr>
<td>Large Tool Tip</td>
<td>Position the mouse pointer over an Icon (button). The tool name and a 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, 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 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

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