Introduction

Engineering Design with SOLIDWORKS® 2018 and video instruction is written to assist students, designers, engineers and professionals. The book provides a solid foundation in SOLIDWORKS by utilizing projects with step-by-step instructions for the beginner to intermediate SOLIDWORKS user featuring machined, plastic and sheet metal components.

Desired outcomes and usage competencies are listed for each project. The book is divided into five sections with 11 projects.

Project 1 - Project 6: 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. Additional techniques include the edit and reuse of features, parts, and assemblies through symmetry, patterns, configurations, SOLIDWORKS 3D ContentCentral and the SOLIDWORKS Toolbox.

Project 7: Understand Top-Down assembly modeling and Sheet Metal parts. Develop components In-Context with InPlace Mates, along with the ability to import parts using the Top-Down assembly method. Convert a solid part into a Sheet Metal part and insert and apply various Sheet Metal features.

Project 8 - Project 9: Recognize SOLIDWORKS Simulation and Intelligent Modeling techniques. Understand a general overview of SOLIDWORKS Simulation and the type of questions that are on the SOLIDWORKS Simulation Associate - Finite Element Analysis (CSWSA-FEA) exam. Apply design intent and intelligent modeling techniques in a sketch, feature, part, plane, assembly and drawing.

Project 10: 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.
**Project 11:** 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.

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 ENGDESIGN-W-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, 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 dplanchard@msn.com or planchard@wpi.edu.

**Note to Instructors**

Please contact the publisher [www.schroff.com](http://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**

- 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.
# Table of Contents

## Introduction
- About the Author
- Acknowledgements
- Contact the Author
- Note to Instructors
- Trademarks, Disclaimer and Copyrighted Material
- References
- Table of Contents

## Project 1 - Overview of SOLIDWORKS and the User Interface
- Project Objective
- What is SOLIDWORKS?
- Basic concepts in SOLIDWORKS
- Start a SOLIDWORKS Session
  - *Tutorial: Start a SOLIDWORKS Session*
    - Welcome dialog box
    - Home Tab
    - Recent Tab
    - Learn Tab
    - Alerts Tab
- SOLIDWORKS User Interface (UI) and CommandManager
  - Menu Bar toolbar
  - Menu Bar menu
  - Drop-down menu
  - Create a New Part Document
    - Novice Mode
    - Advanced Mode
  - Graphic Window (Default)
    - View Default Sketch Planes
  - Open a Part
    - Part FeatureManager
    - FeatureManager Rollback Bar
  - Heads-up View toolbar
    - Dynamic Annotation Views
    - Zoom to Fit
    - Zoom to Area
    - Window-Select
    - Rotate
    - Front View
    - Right View
    - Top View
    - Trimetric view
  - SOLIDWORKS Help
  - SOLIDWORKS Tutorials
SOLIDWORKS Icon Style | 1-17  
Additional User Interface Tools | 1-17  
Right-click | 1-18  
Consolidated toolbar | 1-18  
System feedback icons | 1-18  
Confirmation Corner | 1-19  
Heads-up View toolbar | 1-19  
CommandManager (Default Part tab) | 1-22  
CommandManager (Default Drawing tab) | 1-23  
CommandManager (Default Assembly tab) | 1-24  
CommandManager (Float/Fit) | 1-25  
Selection Enhancements | 1-25  
FeatureManager Design Tree | 1-26  
FeatureManager design tree tab | 1-26  
PropertyManager tab | 1-26  
Configuration Manager tab | 1-26  
DimXpertManager tab | 1-26  
DisplayManager tab | 1-26  
Fly-out FeatureManager | 1-28  
Task Pane | 1-29  
SOLIDWORKS Resources | 1-29  
Design Library | 1-30  
File Explorer | 1-30  
Search | 1-31  
View Palette | 1-31  
Appearances, Scenes and Decals | 1-32  
Custom Properties | 1-32  
SOLIDWORKS Forum | 1-32  
User Interface for Scaling High Resolution Screens | 1-32  
Motion Study tab | 1-33  
3D Views tab | 1-34  
Dynamic Reference Visualization | 1-34  
Mouse Movements | 1-35  
Single-Click | 1-35  
Double-Click | 1-35  
Right-Click | 1-35  
Scroll Wheel | 1-35  
Summary | 1-36

**Project 2 - Fundamentals of Part Modeling**  
2-1  
Project Objective | 2-3  
Project Situation | 2-4  
Project Overview | 2-6  
File Management | 2-7  
Start a SOLIDWORKS Session | 2-8  
System Options | 2-8  
Part Document Template and Document Properties | 2-10  
PLATE Part Overview | 2-13  
PLATE Part-New SOLIDWORKS Document | 2-15  
Base Feature | 2-16  
Machined Part | 2-17
Reference Planes and Orthographic Projection 2-18
PLATE Part-Extruded Boss/Base Feature 2-22
PLATE Part-Modify Dimensions and Rename 2-31
Display Modes, View Modes, View tools and Appearances 2-33
PLATE Part-Extruded Cut Feature 2-35
PLATE Part-Fillet Feature 2-41
PLATE Part-Hole Wizard Feature 2-43
ROD Part Overview 2-46
ROD Part-Extruded Boss/Base Feature 2-48
ROD Part-Hole Wizard Feature 2-50
ROD Part-Chamfer Feature 2-51
ROD Part-Extruded Cut Feature & Convert Entities Sketch Tool 2-52
ROD Part-View Orientation, Named Views & Viewport option 2-57
ROD Part-Copy/Paste Function 2-58
ROD Part-Design Changes with Rollback Bar 2-59
ROD Part-Recover from Rebuild Errors 2-61
ROD Part-Edit Part Appearance 2-65
GUIDE Part Overview 2-67
GUIDE Part-Extruded Boss/Base Feature and Dynamic Mirror 2-69
GUIDE Part-Extruded Cut Slot Profile 2-72
GUIDE Part-Mirror Feature 2-76
GUIDE Part-Holes 2-77
GUIDE Part-Linear Pattern Feature 2-80
GUIDE Part-Materials Editor and Mass Properties 2-82
Manufacturing Considerations 2-84
Sketch Entities and Sketch Tools 2-87
Project Summary 2-88
Questions/Exercises 2-90

**Project 3 - Fundamentals of Assembly Modeling** 3-1
Project Objective 3-3
Project Situation 3-4
Project Overview 3-5
Assembly Modeling Approach 3-5
Linear Motion and Rotational Motion 3-6
GUIDE-ROD assembly 3-7
GUIDE-ROD assembly - Insert Components 3-11
FeatureManager Syntax 3-13
Mate Types 3-16
  - Standard Mates 3-16
  - Advanced Mates 3-17
  - Mechanical Mates 3-18
Quick Mate 3-18
GUIDE-ROD Assembly - Mate the ROD Component 3-20
GUIDE-ROD Assembly - Mate the PLATE Component 3-22
GUIDE-ROD Assembly - Mate Errors 3-27
Collision Detection 3-29
Modify Component Dimension 3-30
SOLIDWORKS Design Library 3-31
GUIDE-ROD Assembly - Insert Mates for Flange bolts 3-34
Socket Head Cap Screw Part 3-38
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>SmartMates</td>
<td>3-44</td>
</tr>
<tr>
<td>Coincident/Concentric SmartMate</td>
<td>3-45</td>
</tr>
<tr>
<td>Tolerance and Fit</td>
<td>3-47</td>
</tr>
<tr>
<td>Exploded View</td>
<td>3-51</td>
</tr>
<tr>
<td>Section View</td>
<td>3-51</td>
</tr>
<tr>
<td>Analyze an Interference Problem</td>
<td>3-56</td>
</tr>
<tr>
<td>Save As Copy Option</td>
<td>3-58</td>
</tr>
<tr>
<td>Save as</td>
<td>3-59</td>
</tr>
<tr>
<td>Save as copy and continue</td>
<td>3-59</td>
</tr>
<tr>
<td>Save as copy and open</td>
<td>3-59</td>
</tr>
<tr>
<td>GUIDE-ROD Assembly-Pattern Driven Component Pattern</td>
<td>3-62</td>
</tr>
<tr>
<td>Linear Component Pattern Feature</td>
<td>3-64</td>
</tr>
<tr>
<td>Folders and Suppressed Components</td>
<td>3-66</td>
</tr>
<tr>
<td>Make-Buy Decision-3D ContentCentral</td>
<td>3-67</td>
</tr>
<tr>
<td>CUSTOMER Assembly</td>
<td>3-69</td>
</tr>
<tr>
<td>Copy the CUSTOMER Assembly - Apply Pack and Go</td>
<td>3-75</td>
</tr>
<tr>
<td>Point at the Center of Mass</td>
<td>3-77</td>
</tr>
<tr>
<td>Project Summary</td>
<td>3-79</td>
</tr>
<tr>
<td>Questions/Exercises</td>
<td>3-80</td>
</tr>
</tbody>
</table>

**Project 4 - Fundamentals of Drawing**

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Project Objective</td>
<td>4-3</td>
</tr>
<tr>
<td>Project Situation</td>
<td>4-4</td>
</tr>
<tr>
<td>Project Overview</td>
<td>4-4</td>
</tr>
<tr>
<td>Drawing Template and Sheet Format</td>
<td>4-5</td>
</tr>
<tr>
<td>Sheet Format and Title Block</td>
<td>4-12</td>
</tr>
<tr>
<td>Company Logo</td>
<td>4-17</td>
</tr>
<tr>
<td>Save Sheet Format and Save As Drawing Template</td>
<td>4-18</td>
</tr>
<tr>
<td>GUIDE Part-Modify</td>
<td>4-22</td>
</tr>
<tr>
<td>GUIDE Part Drawing</td>
<td>4-23</td>
</tr>
<tr>
<td>Move Views and Properties of the Sheet</td>
<td>4-26</td>
</tr>
<tr>
<td>Auxiliary View, Section View and Detail View</td>
<td>4-29</td>
</tr>
<tr>
<td>Auxiliary View</td>
<td>4-30</td>
</tr>
<tr>
<td>Section View</td>
<td>4-31</td>
</tr>
<tr>
<td>Detail View</td>
<td>4-32</td>
</tr>
<tr>
<td>Partial Auxiliary Drawing View - Crop Drawing View</td>
<td>4-33</td>
</tr>
<tr>
<td>Display Modes and Performance</td>
<td>4-35</td>
</tr>
<tr>
<td>Detail Drawing</td>
<td>4-37</td>
</tr>
<tr>
<td>Move Dimensions in the Same View</td>
<td>4-40</td>
</tr>
<tr>
<td>Move Dimensions to a Different View</td>
<td>4-44</td>
</tr>
<tr>
<td>Dimension Holes and the Hole Callout</td>
<td>4-45</td>
</tr>
<tr>
<td>Center Marks and Centerlines</td>
<td>4-48</td>
</tr>
<tr>
<td>Modify the Dimension Scheme</td>
<td>4-50</td>
</tr>
<tr>
<td>GUIDE Part-Insert an Additional Feature</td>
<td>4-54</td>
</tr>
<tr>
<td>General Notes and Parametric Notes</td>
<td>4-56</td>
</tr>
<tr>
<td>Revision Table</td>
<td>4-59</td>
</tr>
<tr>
<td>Part Number and Document Properties</td>
<td>4-61</td>
</tr>
<tr>
<td>Exploded View</td>
<td>4-67</td>
</tr>
<tr>
<td>Balloons</td>
<td>4-69</td>
</tr>
<tr>
<td>Bill of Materials</td>
<td>4-71</td>
</tr>
<tr>
<td>Insert a Center of Mass Point into a drawing</td>
<td>4-76</td>
</tr>
<tr>
<td>Project Summary</td>
<td>4-78</td>
</tr>
<tr>
<td>-----------------</td>
<td>------</td>
</tr>
<tr>
<td>Questions/Exercises</td>
<td>4-78</td>
</tr>
</tbody>
</table>

**Project 5 - Extrude and Revolve Features**  
5-1

- **Project Objective**  
5-3
- **Project Overview**  
5-4
- **Design Intent**  
5-6
- **Project Situation**  
5-9
- **Part Template**  
5-11
- **BATTERY Part**  
5-15
- **BATTERY Part - Extruded Boss/Base Feature**  
5-17
- **BATTERY Part - Fillet Feature Edge**  
5-21
- **BATTERY Part - Extruded Cut Feature**  
5-23
- **BATTERY Part - Fillet Feature**  
5-25
- **BATTERY Part - Extruded Boss/Boss Feature**  
5-26
- **Injection Molded Process**  
5-32
- **BATTERYPLATE Part**  
5-33
- **Save As, Delete, Edit Feature and Modify**  
5-34
- **BATTERYPLATE Part - Extruded Boss/Base Feature**  
5-36
- **BATTERYPLATE Part - Fillet Features: Full Round and Multiple Radius Options**  
5-37
- **Multi-body Parts and the Extruded Boss/Base Feature**  
5-40
- **LENS Part**  
5-42
- **LENS Part-Revolved Base Feature**  
5-43
- **LENS Part-Shell Feature**  
5-46
- **Extruded Boss/Base Feature and Convert Entities Sketch tool**  
5-47
- **LENS Part-Hole Wizard**  
5-48
- **LENS Part - Revolved Boss Thin Feature**  
5-50
- **LENS Part - Extruded Boss/Boss Feature and Offset Entities**  
5-52
- **LENS Part - Extruded Boss/Boss Feature and Transparency**  
5-54
- **BULB Part**  
5-56
- **BULB Part - Revolved Base Feature**  
5-57
- **BULB Part - Revolved Boss Feature and Spline Sketch tool**  
5-60
- **BULB Part - Revolved Cut Thin Feature**  
5-61
- **BULB Part - Dome Feature**  
5-63
- **BULB Part - Circular Pattern Feature**  
5-64
- **Customizing Toolbars and Short Cut Keys**  
5-68
- **Design Checklist and Goals before Plastic Manufacturing**  
5-70
- **Mold Base**  
5-72
- **Applying SOLIDWORKS Features for Mold Tooling Design**  
5-72
- **Manufacturing Design Issues**  
5-82
- **Project Summary**  
5-83
- **Questions/Exercises**  
5-84

**Project 6 - Swept, Lofted and Additional Features**  
6-1

- **Project Objective**  
6-3
- **Project Overview**  
6-4
- **Project Situation**  
6-5
- **O-RING Part - Swept Base Feature**  
6-7
- **O-RING Part - Design Table**  
6-8
- **SWITCH Part - Lofted Base Feature**  
6-12
- **SWITCH Part - Dome Feature**  
6-17
## Four Major Categories of Solid Features

- **LENSCAP Part**
  - Extruded Boss/Base, Extruded Cut and Shell Features (6-19)
  - Revolved Thin Cut Feature (6-20)
  - Thread, Swept Feature and Helix/Spiral Curve (6-23)
- **HOUSING Part**
  - Lofted Boss Feature (6-30)
  - Second Extruded Boss/Base Feature (6-37)
  - Shell Feature (6-38)
  - Third Extruded Boss/Base Feature (6-39)
  - Draft Feature (6-40)
  - Thread with Swept Feature (6-42)
  - Handle with Swept Feature (6-47)
  - Extruded Cut Feature with Up To Surface (6-52)
  - First Rib and Linear Pattern Feature (6-54)
  - Second Rib Feature (6-57)
  - Mirror Feature (6-60)
  - Assembly Template (6-64)
- **FLASHLIGHT Assembly**
  - Sub-assembly (6-64)
- **BATTERYANDPLATE Sub-assembly**
- **CAPANLDSENS Sub-assembly**
- **FLASHLIGHT Assembly**
  - Addressing Interference Issues (6-81)
- **Export Files and eDrawings**
- **Project Summary**
- **Questions/Exercises**

## Project 7 - Top-Down Assembly Modeling and Sheet Metal

- **Project Objective**
- **Project Situation**
- **Top-Down Assembly Modeling**
  - BOX Assembly Overview (7-5)
  - InPlace Mates and In-Context features (7-8)
  - Part Template and Assembly Template (7-12)
  - Box Assembly and Layout Sketch (7-13)
  - Global Variables and Equations (7-17)
- **MOTHERBOARD - Insert Component**
- **POWERSUPPLY - Insert Component**
- **Sheet Metal Overview**
- **Bends**
- **Relief**
- **CABINET - Insert Component**
- **CABINET - Rip Feature and Sheet Metal Bends**
- **CABINET - Edge Flange**
- **CABINET - Hole Wizard and Linear Pattern Features**
- **CABINET - Sheetmetal Design Library Feature**
- **CABINET - Louver Forming tool**
- **Manufacturing Considerations**
- **Additional Pattern Options**
- **CABINET - Formed and Flat States**
| CABINET - Sheet Metal Drawing with Configurations | 7-64 |
| PEM Fasteners and IGES Components | 7-70 |
| Pattern Driven Component Pattern | 7-74 |
| MOTHERBOARD - Assembly Hole Feature | 7-76 |
| Assembly FeatureManager and External References | 7-77 |
| Replace Components | 7-79 |
| Equations | 7-82 |
| Design Tables | 7-86 |
| BRACKET Part - Sheet Metal Features | 7-89 |
| BRACKET Part - In-Content Features | 7-91 |
| BRACKET Part - Edge, Tab, Break Corner and Miter Flange Features | 7-93 |
| BRACKET Part - Mirror Component | 7-98 |
| MirrorBRACKET Part - Bends, Fold, Unfold and Jog Features | 7-101 |
| Project Summary | 7-106 |
| Questions/Exercises | 7-107 |

### Project 8 - SOLIDWORKS Simulation

| Project Objective | 8-1 |
| Basic FEA Concepts | 8-1 |
| Simulation Advisor | 8-3 |
| SOLIDWORKS Simulation Help & Tutorials | 8-5 |
| Linear Static Analysis | 8-6 |
| Sequence of Calculations in General | 8-10 |
| Stress Calculations in General | 8-10 |
| Overview of Yield or Inflection Point in a Stress-Strain curve | 8-10 |
| Material Properties in General | 8-11 |
| Connections in General | 8-12 |
| Restraint Types | 8-12 |
| Loads and Restraints in General | 8-14 |
| Meshing in General | 8-15 |
| Meshing Types | 8-16 |
| SOLIDWORKS Simulation Meshing Tips | 8-19 |
| Running the Study | 8-21 |
| Displacement Plot - Output of Linear Static Analysis | 8-21 |
| Adaptive Methods for Static Studies | 8-22 |
| Sample Exam Questions | 8-23 |
| FEA Modeling Section | 8-37 |
| Tutorial: FEA Model 8-1 | 8-37 |
| Tutorial: FEA Model 8-2 | 8-41 |
| Tutorial: FEA Model 8-3 | 8-45 |
| Tutorial: FEA Model 8-4 | 8-49 |
| Tutorial: FEA Model 8-5 | 8-52 |
| Definitions | 8-55 |

### Project 9 - Intelligent Modeling Techniques

| Project Objective | 9-1 |
| Design Intent | 9-3 |
| Sketch | 9-4 |
| Fully Defined Sketch tool | 9-5 |
| SketchXpert | 9-8 |
Equations

- Dimension Driven Equations
  - Explicit Equation Driven Curve
  - Parametric Equation Driven Curve

Curves

- Curve Through XYZ Points
- Projected Composite Curves
- Feature - End Conditions
- Along A Vector
- FeatureXpert (Constant Radius)
- Symmetry
- Bodies to mirror

Planes

- Conic Sections and Planes

Assembly

- Assembly Visualization
- SOLIDWORKS Sustainability - Assembly
- MateXpert

Drawing

- DimXpert

Project Summary

Project 10 - Additive Manufacturing - 3D Printing

- Project 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)
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Raft, Skirt, Brim</td>
<td>10-21</td>
</tr>
<tr>
<td>Support, Touching Build Plate</td>
<td>10-22</td>
</tr>
<tr>
<td>Slicer Engine</td>
<td>10-23</td>
</tr>
<tr>
<td>G-code</td>
<td>10-23</td>
</tr>
<tr>
<td>Infill</td>
<td>10-23</td>
</tr>
<tr>
<td>Infill Pattern/Shape</td>
<td>10-23</td>
</tr>
<tr>
<td>Shells/Parameters</td>
<td>10-23</td>
</tr>
<tr>
<td>Lay Height</td>
<td>10-24</td>
</tr>
<tr>
<td>Influence of Percent Infill</td>
<td>10-24</td>
</tr>
<tr>
<td>Remove the Model from the Build Plate</td>
<td>10-25</td>
</tr>
<tr>
<td>Know the Printer’s Limitation</td>
<td>10-25</td>
</tr>
<tr>
<td>Tolerance for Interlocking Parts</td>
<td>10-25</td>
</tr>
<tr>
<td>General Printing Tips</td>
<td>10-26</td>
</tr>
<tr>
<td>Reduce Infill</td>
<td>10-26</td>
</tr>
<tr>
<td>Control Build Area Temperature</td>
<td>10-26</td>
</tr>
<tr>
<td>Add Pads</td>
<td>10-27</td>
</tr>
<tr>
<td>Unique Shape or a Large Part</td>
<td>10-27</td>
</tr>
<tr>
<td>Safe Zone Rule</td>
<td>10-27</td>
</tr>
<tr>
<td>Wall Thickness</td>
<td>10-27</td>
</tr>
<tr>
<td>Extruder Temperature</td>
<td>10-28</td>
</tr>
<tr>
<td>First Layer Not Sticking</td>
<td>10-28</td>
</tr>
<tr>
<td>Level Build Platform</td>
<td>10-29</td>
</tr>
<tr>
<td>Minimize Internal Support</td>
<td>10-29</td>
</tr>
<tr>
<td>Water-tight Mesh</td>
<td>10-29</td>
</tr>
<tr>
<td>Clearance</td>
<td>10-29</td>
</tr>
<tr>
<td>In General</td>
<td>10-30</td>
</tr>
<tr>
<td>Summary</td>
<td>10-31</td>
</tr>
</tbody>
</table>

**Project 11 - Introduction to the Certified Associated – Mechanical Design Exam**  

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Project Objective</td>
<td>11-3</td>
</tr>
<tr>
<td>Introduction</td>
<td>11-3</td>
</tr>
<tr>
<td>Part 1 of the Exam</td>
<td>11-4</td>
</tr>
<tr>
<td>Basic Part Creation and Modification, Intermediate Part Creation and Modification</td>
<td>11-4</td>
</tr>
<tr>
<td>Assembly Creation and Modification</td>
<td>11-4</td>
</tr>
<tr>
<td>Part 2 of the Exam</td>
<td>11-7</td>
</tr>
<tr>
<td>Introduction and Drafting Competencies</td>
<td>11-7</td>
</tr>
<tr>
<td>Advanced Part Creating and Modification</td>
<td>11-7</td>
</tr>
<tr>
<td>Assembly Creation and Modification</td>
<td>11-9</td>
</tr>
<tr>
<td>Intended Audience</td>
<td>11-10</td>
</tr>
<tr>
<td>During the Exam</td>
<td>11-11</td>
</tr>
<tr>
<td>Drafting Competencies</td>
<td>11-12</td>
</tr>
<tr>
<td>Example 1</td>
<td>11-12</td>
</tr>
<tr>
<td>Example 2</td>
<td>11-12</td>
</tr>
<tr>
<td>Example 3</td>
<td>11-13</td>
</tr>
<tr>
<td>Example 4</td>
<td>11-13</td>
</tr>
<tr>
<td>Example 5</td>
<td>11-13</td>
</tr>
<tr>
<td>Example 6</td>
<td>11-13</td>
</tr>
<tr>
<td>Basic Part Creation and Modification, Intermediate Part Creation and Modification</td>
<td>11-14</td>
</tr>
<tr>
<td>Example 1</td>
<td>11-14</td>
</tr>
<tr>
<td>Example 2</td>
<td>11-15</td>
</tr>
<tr>
<td>Example 3</td>
<td>11-16</td>
</tr>
<tr>
<td>Example 4</td>
<td>11-17</td>
</tr>
</tbody>
</table>
Overview of Projects

Project 1: Overview of SOLIDWORKS and the SOLIDWORKS User Interface

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

- Project 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.

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

- 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.

Project 2: Fundamentals of Part Modeling

Project 2 begins by creating file folders and sub-folders to manage projects. Apply System Options and Document Properties. Develop a Custom Part Template.

- Create three parts: PLATE, ROD, and GUIDE.

- Utilize the following features: Extruded Boss/Base, Instant3D, Extruded Cut, Fillet, Mirror, Chamfer, Hole Wizard and Linear Pattern. Apply materials and appearance.

- Learn the SOLIDWORKS interface, how to select the correct Sketch plane, fully define the sketch, edit sketches and features and copy and paste features.
Project 3: Fundamentals of Assembly Modeling (Bottom up)

Project 3 introduces the fundamentals of Assembly Modeling (Bottom-up) along with creating Standard mates (Coincident, Concentric, Distance, Parallel, Tangent), Mechanical mate (Slot), SmartMates and the Quick mate procedure.

- Create an Assembly template. Review the Assembly FeatureManager syntax.
- Create two assemblies: GUIDE-ROD and CUSTOMER.
- Edit component dimensions and address tolerance and fit.
- Incorporate design changes into an assembly. Obtain additional SOLIDWORKS parts using 3D ContentCentral and the SOLIDWORKS Toolbox.

Project 4: Fundamentals of Drawing

Project 4 covers the development of a customized drawing template.

- Learn the two Sheet Format modes: 1.) Edit Sheet Format and 2.) Edit Sheet
- Develop and insert a Company logo from a bitmap or picture file.
- Create the GUIDE drawing with Custom Properties, various drawing views and a Revision table.
- Create the GUIDE-ROD drawing with Custom Properties, an Exploded Isometric view with a Bill of Materials and balloons.
Project 5: Extrude and Revolve Feature

Project 5 focuses on the customer’s design requirements. Create four key FLASHLIGHT components: BATTERY, BATTERYPLATE, LENS and BULB. Develop an ANSI - IPS Part Template.

- Create the BATTERY and BATTERYPLATE part with the Extruded Boss/Base feature and the Instant3D tool.

- Create the LENS and BULB with the Revolved Boss/Base feature. Utilize the following features: Extruded Boss/Base, Extruded Cut, Revolved Base, Revolved Cut, Dome, Shell, Fillet and Circular Pattern.

- Utilize the Mold tools to create the cavity plate for the BATTERYPLATE.

💡 Tangent edges are displayed for educational purposes in the book.

Project 6: Swept, Lofted and Additional Features

Project 6 develops four additional components to complete the FLASHLIGHT assembly: O-RING, SWITCH, LENSCAP and HOUSING.

- Utilize the following features: Swept Boss/Base, Lofted Boss/Base, Rib, Linear Pattern, Circular Pattern, Draft and Dome.

- Insert the components and sub-assemblies and create the Mates to finish the final FLASHLIGHT assembly.
Project 7: Top-Down assembly and Sheet Metal parts

Project 7 focuses on the Top-Down assembly modeling approach. Develop a Layout Sketch.

- Create components and modify them In-Context of the assembly.
- Create Sheet metal features. Utilize the following features: Rip, Insert Sheet metal Bends, Base Flange, Edge Flange, Miter Flange, Break Corners, Hem and more.
- Utilize the Die Cut Feature and Louver Form tool.
- Add IGES format part files from the Internet.
- Replace fasteners in the assembly and redefine mates.
- Utilize equations, Global Variables and a Design Table to create multiple configurations of the BOX assembly.

💡 The book is designed to expose the new SOLIDWORKS user to many tools, techniques and procedures. It may not always use the most direct tool or process.
Project 8: SOLIDWORKS Simulation

Project 8 provides a general overview of SOLIDWORKS Simulation and the type of questions that are on the SOLIDWORKS Simulation Associate - Finite Element Analysis (CSWSA-FEA) exam. On the completion of this project, you will be able to:

Recognize the power of SOLIDWORKS Simulation.

Utilize SOLIDWORKS Simulation to:

- Define a Static Analysis Study.
- Apply Material to a part model.
- Work with a Solid and Sheet Metal model.
- Define Solid, Shell and Beam elements.
- Define Standard and Advanced Fixtures and External loads.
- Define Local and Global coordinate systems.
- Understand the axial forces, sheer forces, bending moments and factor of safety.
- Define Connector properties such as Contact Sets, No Penetration and Bonded.
- Set and modify plots to display in the Results folder.
- Work with Multi-body parts as different solid bodies.
- Select different solvers as directed to optimize problems.
- Determine if the result is valid.
- Understand the type of problems and questions that are on the CSWSA-FEA exam.
- Ability to use SOLIDWORKS Simulation Help.

Project 9: Intelligent Modeling Techniques

Project 9 introduces some of the available tools in SOLIDWORKS to perform intelligent modeling.

Intelligent modeling is incorporating design intent into the definition of the sketch, feature, part and assembly or drawing document. Intelligent modeling is most commonly addressed through design intent.

- All needed models for this project are provided.

Screen shots and illustrations in the book display the SOLIDWORKS user default setup.

💡 During the initial SOLIDWORKS installation, you are requested to select either the ISO or ANSI drafting standard. ISO is typically a European overall drafting standard and uses First Angle Projection. The book is written using the ANSI (US) overall drafting standard and Third Angle Projection for drawings.
Project 10: Additive Manufacturing - 3D Printing

Project 10 provides 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 project, 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.
  - Raft, Skirt and Brim.
  - Support and Touching Buildplate.
  - Slicer.
  - G-code.
  - Address fit tolerance for interlocking parts.
  - Define general 3D Printing tips.
Project 11: Introduction to the Certified Associate - Mechanical Design (CSWA) Exam

Project 11 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.

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 Simulation, 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 feature tools such as Extruded Boss/Base, Extruded Cut, Fillet, etc. from the Features tab in the CommandManager to create 3D parts.

Extruded features begin with a 2D sketch created on a Sketch plane.

The 2D sketch is a profile or cross section. Use sketch tools such as Line, Center Rectangle, Slot, Circle Centerline, Mirror, etc. from the Sketch tab in the CommandManager to create a 2D sketch. Sketch the general shape of the profile. Add geometric relationships and dimensions to control the exact size of the geometry and your Design Intent. Design for change.

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.

Use Geometric relationships: Vertical, Horizontal, Parallel, etc. and various End Conditions to maintain the Design Intent.

Create a hole that penetrates through a part (Through All). SOLIDWORKS maintains relationships through the change.

The step-by-step approach used in this text allows you to create, edit and modify parts, assemblies and drawings. Change is an integral part of design.
About the Book

You will find a wealth of information in this book. The following conventions are used throughout the text:

- 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 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 bar. The Plane PropertyManager is displayed.

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

- 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. View the provided videos and models to enhance the user experience. All templates, logos and model documents along with additional support materials are available.

💡 The book is designed to expose the new SOLIDWORKS user to many tools, techniques and procedures. It may not always use the most direct tool or process. Learn by doing, not just by reading.

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 the <strong>Extruded Boss/Base</strong> feature.</td>
</tr>
<tr>
<td></td>
<td>• Selected geometry: line, circle.</td>
<td>• Click <strong>Corner Rectangle</strong> from the Consolidated Sketch toolbar.</td>
</tr>
<tr>
<td></td>
<td>• Value entries.</td>
<td>• Select the <strong>centerpoint</strong>.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Enter <strong>3.0</strong> for Radius.</td>
</tr>
<tr>
<td><strong>Capitalized</strong></td>
<td>• Filenames.</td>
<td>• Save the <strong>FLASHLIGHT</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>
</tr>
<tr>
<td>Click inside</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 the</td>
</tr>
<tr>
<td></td>
<td>FeatureManager design tree.</td>
</tr>
<tr>
<td>Drag</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>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>Tool Tip</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 Tool Tip</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</td>
<td>Position the mouse pointer over various areas of the sketch, part, assembly</td>
</tr>
<tr>
<td>feedback</td>
<td>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. Review various Windows terminology that describe menus, toolbars and commands that constitute the graphical user interface in SOLIDWORKS.
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, avl files of assemblies, projects, and all initial and final SOLIDWORKS model files.