INTRODUCTION

Drawing and Detailing with SolidWorks is written to educate and assist students, designers, engineers, and professionals in the drawing and detailing tools of SolidWorks. Explore the learning process through a series of design situations, industry scenarios, projects, and objectives target towards the beginning to intermediate SolidWorks user.

Work through numerous activities to create multiple-view, multiple-sheet, detailed drawings, and assembly drawings. Develop Drawing templates, Sheet formats, and Custom Properties. Construct drawings that incorporate part configurations, assembly configurations, and design tables with equations. Manipulate annotations in parts, drawings, assemblies, Revision tables, Bills of Materials and more.

Apply your drawing and detailing knowledge to over thirty exercises. The exercises test your usage competency as well as explore additional topics with industry examples. Advanced exercises require the ability to create parts and assemblies.

Drawing and Detailing with SolidWorks is not a reference book for all drafting and drawing techniques and tools. The book provides information and examples in the following areas:

- History of engineering graphics, manual sketching techniques, orthographic projection, isometric projection, multi-view drawings, dimensioning practices, fasteners in general, tolerance and fit and the history of CAD leading to the development of SolidWorks.

- Start a SolidWorks 2012 session and to understand the following interfaces: 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.


- Import an AutoCAD file as a Sheet format. Insert SolidWorks System Properties and Custom Properties.

- Create new SolidWorks Document tabs.

- Create multi-sheet drawings from various part configurations and develop the following drawing views: Standard, Isometric, Auxiliary, Section, Broken Section, Detail, Half Section (Cut-away), Crop, Projected Back, with a Bill of Materials and a Revision Table and Revisions.

- Insert and edit: Dimensions, Feature Control Frames, Datums, Geometric Tolerancing, Surface Finishes, and Weld Symbols using DimXpert and manual techniques.
• Create, apply, and save Blocks and Parametric Notes in a drawing.

Chapter 10 provides a bonus section on the *Certified SolidWorks Associate CSWA program* with sample exam questions and initial and final SolidWorks models.


The authors recognize that companies utilize additional drawing standards. The authors developed the industry scenarios by combining industry experience with their knowledge of engineers, sales, vendors and manufacturers. These professionals are directly involved with SolidWorks everyday. Their work goes far beyond a simple drawing with a few dimensions. They create detailed drawings, assembly drawings, marketing drawings and customer drawings. SolidWorks users work between drawings, parts, assemblies and many other documents to complete a project on time.

**Note to Instructors**

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

SolidWorks® Corp. is a Dassault Systèmes S.A. (Nasdaq: DASTY) company that develops and markets software for design, analysis, and product data management applications Microsoft Windows®, Microsoft Office® and its family of products are registered trademarks of the Microsoft Corporation. Other software applications and parts described in this book are trademarks or registered trademarks of their respective owners.

Dimensions of parts and model views are modified for illustration purposes. Every effort is made to provide an accurate text. The authors and the manufacturers shall not be held liable for any parts or drawings developed or designed with this book or any responsibility for inaccuracies that appear in the book. Web and company information was valid at the time of this printing.


Note: By permission of The American Society of Mechanical Engineers, Codes and Standards, New York, NY, USA. All rights reserved.
Additional information references the American Welding Society, AWS 2.4:1997 Standard Symbols for Welding, Braising and Non-Destructive Examinations, Miami, Florida, USA.

**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, CSWP, and CSDA. 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**
• The Fundamentals of SolidWorks®: Featuring the VEXplorer robot, 2008 and 2007


• 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.

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

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 sixth 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.
References

- ASME B4.2 Dimensions Preferred Metric Limits and Fits, ASME, NY.
- Walker, James, Machining Fundamentals, Goodheart Wilcox, 1999.

1 An on-line catalog of ASME Codes and Standards is available on their web site www.asme.org.

2 An on-line catalog of SMC parts and documents is available on their web site www.smcusa.com. Instructions to download additional SMC components are available in the Appendix.

3 An on-line catalog of Emerson-EPT parts and documents is available on their web site www.emerson-ept.com.

4 An on-line catalog of AWS Standards is available on their web site www.aws.org.
Every license of SolidWorks 2012 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.

New in SolidWorks 2012 is the What’s New Examples section.

All templates, logos and needed models for this book are included on the enclosed DVD. Copy the information from the DVD to your local hard drive. Work from your local hard drive.
TABLE OF CONTENTS

Introduction I-1
   About the Cover I-2
   About the Authors I-2
   Dedication I-3
   Contact the Authors I-3
   Note to Instructors I-3
   Trademarks, Disclaimers, and Copyrighted Material I-4
   References I-4
   Table of Contents I-7
   What is SolidWorks I-16
   Design Intent I-21
   Overview of Chapters I-24
   Chapter 1: History of Engineering Graphics I-24
   Chapter 2: Isometric Projection and Multi View Drawings I-24
   Chapter 3: Dimensioning Practices, Tolerancing and Fasteners I-24
   Chapter 4: SolidWorks 2012 User Interface I-25
   Chapter 5: Introduction to SolidWorks Part Modeling I-25
   Chapter 6: Drawing Templates and Sheet Formats I-26
   Chapter 7: Drawing and Various Drawing Views I-26
   Chapter 8: Assembly Drawings I-27
   Chapter 9: Datums, Feature Control Frames, Geometric Tolerancing and Symbols I-27
   Chapter 10: Introduction to the CSWA exam I-28
   About the Book I-29
   Windows Terminology in SolidWorks I-30

Chapter 1 - History of Engineering Graphics I-1
   Chapter Overview I-3
   History of Engineering Graphics I-3
   2D Cartesian Coordinate System I-6
   3D Cartesian Coordinate System I-7
   Absolute Coordinates I-9
   Relative Coordinates I-9
   Polar Coordinates I-10
   Cylindrical and Spherical Coordinates I-10
   Free Hand Sketching I-11
   General Sketching Techniques I-12
   Geometric Entities I-13
      Points I-13
      Lines I-13
      Planes I-14
      Circles I-14
      Arcs I-15
   Solid Primitives I-15
   Alphabet of Lines I-16
   Precedence of Line Types I-22
<table>
<thead>
<tr>
<th>Introduction</th>
<th>Drawing and Detailing with SolidWorks 2012</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alphabet of Lines - Exercises</td>
<td>1-24</td>
</tr>
<tr>
<td>Orthographic Projection</td>
<td>1-26</td>
</tr>
<tr>
<td>Glass Box Method</td>
<td>1-27</td>
</tr>
<tr>
<td>Six Principal Orthographic Views</td>
<td>1-27</td>
</tr>
<tr>
<td>Height, Width, and Depth Dimensions</td>
<td>1-30</td>
</tr>
<tr>
<td>Transferring Dimensions</td>
<td>1-30</td>
</tr>
<tr>
<td>Orthographic Projection - Exercises</td>
<td>1-32</td>
</tr>
<tr>
<td>Planes (Normal, Inclined and Oblique)</td>
<td>1-37</td>
</tr>
<tr>
<td>Plane - Exercises</td>
<td>1-38</td>
</tr>
<tr>
<td>Chapter Summary</td>
<td>1-44</td>
</tr>
<tr>
<td>Chapter Terminology</td>
<td>1-44</td>
</tr>
<tr>
<td>Questions / Exercises</td>
<td>1-46</td>
</tr>
</tbody>
</table>

**Chapter 2 - Isometric Projection and Multi View Drawings**  
2-1  
| Chapter Overview | 2-3 |
| Isometric Projections | 2-3 |
| Isometric Sketching | 2-5 |
| Circles drawn in Axonometric Views | 2-7 |
| Additional Projections | 2-9 |
| Arrangement of Views | 2-13 |
| Two View drawing | 2-14 |
| One View drawing | 2-16 |
| Drawing - Exercises | 2-19 |
| Drawing views - Advanced | 2-21 |
| Section View | 2-21 |
| Detail View | 2-23 |
| Broken out View | 2-24 |
| Break or Broken View | 2-25 |
| Crop View | 2-26 |
| Auxiliary View | 2-27 |
| Exercises | 2-27 |
| History of Computer Aided Design (CAD) | 2-28 |
| Boolean operation | 2-29 |
| Chapter Summary | 2-31 |
| Chapter Terminology | 2-31 |
| Questions / Exercises | 2-34 |

**Chapter 3 - Dimensioning Practices, Tolerancing and Fasteners**  
3-1  
<p>| Chapter Overview | 3-3 |
| Size and Location Dimensions | 3-3 |
| Dimensioning Systems | 3-4 |
| Standards for Dimensioning | 3-5 |
| Part / Construction Dimensions | 3-5 |
| Two Place Decimal Dimensions | 3-6 |
| Size Dimensions | 3-6 |
| Continuous Dimensions | 3-7 |
| Other Dimension Placements | 3-8 |
| Dimension - Exercises | 3-10 |
| Dimensioning Cylinders | 3-12 |
| Dimensioning a Simple Hole | 3-14 |
| Dimensioning Angles | 3-15 |</p>
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimensioning a Point or a Center</td>
<td>3-16</td>
</tr>
<tr>
<td>Dimensioning equally spaced holes on a Circle</td>
<td>3-16</td>
</tr>
<tr>
<td>Dimensioning Holes not on a circle</td>
<td>3-17</td>
</tr>
<tr>
<td>Dimensioning Arcs</td>
<td>3-17</td>
</tr>
<tr>
<td>Dimensioning Chamfers</td>
<td>3-18</td>
</tr>
<tr>
<td>Dual Dimensioning</td>
<td>3-18</td>
</tr>
<tr>
<td>Dimension - Exercises</td>
<td>3-19</td>
</tr>
<tr>
<td>Precision and Tolerance</td>
<td>3-22</td>
</tr>
<tr>
<td>Tolerance for a drawing</td>
<td>3-22</td>
</tr>
<tr>
<td>General Tolerance - Title Block</td>
<td>3-22</td>
</tr>
<tr>
<td>Local Tolerance - Dimension</td>
<td>3-23</td>
</tr>
<tr>
<td>Limit Tolerance</td>
<td>3-23</td>
</tr>
<tr>
<td>Unilateral Tolerance</td>
<td>3-24</td>
</tr>
<tr>
<td>Bilateral Tolerance</td>
<td>3-24</td>
</tr>
<tr>
<td>Formatting inch Tolerance</td>
<td>3-24</td>
</tr>
<tr>
<td>Metric Dimension Specifications</td>
<td>3-25</td>
</tr>
<tr>
<td>General Nomenclature</td>
<td>3-25</td>
</tr>
<tr>
<td>Fit - Hole Tolerance</td>
<td>3-26</td>
</tr>
<tr>
<td>Fit between Mating Parts</td>
<td>3-26</td>
</tr>
<tr>
<td>Clearance Fit</td>
<td>3-27</td>
</tr>
<tr>
<td>Interference Fit</td>
<td>3-27</td>
</tr>
<tr>
<td>Transition Fit</td>
<td>3-27</td>
</tr>
<tr>
<td>Line Fit</td>
<td>3-27</td>
</tr>
<tr>
<td>Fasteners in General</td>
<td>3-28</td>
</tr>
<tr>
<td>Representing External (Male) Threads</td>
<td>3-28</td>
</tr>
<tr>
<td>Cutting External (Male) Threads</td>
<td>3-29</td>
</tr>
<tr>
<td>Die</td>
<td>3-29</td>
</tr>
<tr>
<td>Lathe</td>
<td>3-29</td>
</tr>
<tr>
<td>Representing Internal (Female) Threads</td>
<td>3-30</td>
</tr>
<tr>
<td>Cutting Internal (Female) Threads</td>
<td>3-30</td>
</tr>
<tr>
<td>Taper</td>
<td>3-31</td>
</tr>
<tr>
<td>Plug</td>
<td>3-31</td>
</tr>
<tr>
<td>Bottoming</td>
<td>3-31</td>
</tr>
<tr>
<td>American National Standard and Unified Screw threads</td>
<td>3-32</td>
</tr>
<tr>
<td>Single vs. Double or Triple Threads</td>
<td>3-32</td>
</tr>
<tr>
<td>Pitch and Major Diameter</td>
<td>3-33</td>
</tr>
<tr>
<td>Thread Class of Fit</td>
<td>3-33</td>
</tr>
<tr>
<td>Class 1</td>
<td>3-33</td>
</tr>
<tr>
<td>Class 2</td>
<td>3-33</td>
</tr>
<tr>
<td>Class 3</td>
<td>3-33</td>
</tr>
<tr>
<td>General Thread Notes</td>
<td>3-34</td>
</tr>
<tr>
<td>Dimensioning a CounterBore Hole</td>
<td>3-35</td>
</tr>
<tr>
<td>Dimensioning a CounterSink Hole</td>
<td>3-35</td>
</tr>
<tr>
<td>Chapter Summary</td>
<td>3-36</td>
</tr>
</tbody>
</table>
Chapter Terminology
Questions / Exercises

Chapter 4 - SolidWorks 2012 User Interface
Chapter Overview
Start a SolidWorks 2012 Session
SolidWorks UI and CommandManager
  Menu bar toolbar
  Menu bar menu
  Drop-down menu
  Right-click Context toolbar
  Fly-out tool buttons / Consolidated menu
  System feedback icons
  Confirmation Corner
  Heads-up View toolbar
  CommandManager
  FeatureManager Design Tree
  Fly-out FeatureManager
  Task Pane
    SolidWorks Resources
    Design Library
    File Explorer
    View Palette
    Appearances, Scenes, and Decals
    Custom Properties
    Document Recover
    Motion Study tab
  New Part
  SolidWorks Default Graphics area
  SolidWorks Help
  SolidWorks Tutorials
  Chapter Summary
  Chapter Terminology

Chapter 5 - Drawing Templates and Sheet Formats
Chapter Objective
Chapter Overview
Engineering Drawing and Related Documentation Practices
File Management
Default Drawing Templates, Sheet Format and Sheet Size
  New SolidWorks Document
  Sheet Format/Size
    ASME Y14.1 Drawing Sheet Size and Format
Line Format Toolbar
Cursor Feedback
Sheet Properties
Display Styles / Modes
System Options
Document Properties
  Font
  Arrowheads
Line Widths 5-22
Line Font 5-23
Document Properties-Dimensions 5-24
Drafting (Dimensioning) Standard 5-25
Dual Dimensions Display Option 5-25
Fixed Size Weld Symbols Option 5-25
Display Datums per 1982 Option 5-26
Leading Zeroes and Trailing Zeroes Option 5-26
Alternative Section Display Option 5-26
Centerline Extension and Center Marks Option 5-26
Auto Insert on Vies Creation Option 5-27
Extension Lines Option 5-27
Datum Feature Option 5-28
Surface Finish Symbols 5-28
Break Line Option 5-28
Automatic Update on BOM Option 5-28
Cosmetic Thread Display Option 5-28
Document Properties, Annotations Font 5-31
Notes Font 5-31
Dimension Font 5-31
Auxiliary View / Auxiliary View Label Font 5-32
Detail View / Detail View Label Font 5-32
Section View / Section View Label Font 5-33
Annotations Arrow Font 5-33
Tables Font 5-34
Document Properties, Dimension Option 5-35
Offset Distances Option 5-35
Arrows Option 5-36
Break Dimension / Extension Option 5-36
Bent Leader Length Option 5-36
Document Properties-Notes and Balloons Option 5-38
Document Properties - Arrows 5-39
Document Properties - Line Font 5-40
Predefined and Projected Views 5-41
Save As 5-43
Sheet Format 5-46
Title Block Notes and Properties 5-53
System Properties 5-53
User Defined Properties 5-55
Linked Notes 5-55
Size, Sheet and Scale Properties 5-58
Custom Property and Logo Picture 5-60
User Defined Custom Properties 5-62
Copy / Paste Custom Properties 5-62
Custom Properties in a Part or Assembly 5-64
General Notes 5-66
Tables 5-67
Save Sheet Format and Drawing Template 5-71
A (ANSI) Size Drawing Template 5-73
Chapter Summary 5-74
Primary Value 7-63
Text Dimension 7-63
Dual Dimension 7-64
Dimension Leaders PropertyManager 7-65
Witness / Leader Display 7-65
Leader Style 7-65
Break Line 7-66
Custom Text Position 7-66
Arc Condition 7-66
Dimension Other PropertyManager 7-66
Override Units 7-67
Text Fonts 7-67
Options 7-67
Layer 7-68
Dimension Properties Dialog Box 7-68
Hide Dimension Line, Hide Extension Line and Driven 7-69
Dimension Schemes 7-70
Foreshortened Radii 7-70
Partially Rounded Ends, Center/Min/Max Arc Condition 7-71
Display Option, Offset Text 7-72
Slotted Holes 7-72
Grid/Snap 7-74
Location of Features 7-74
Base Line Dimensioning 7-74
Ordinate Dimension 7-75
View Layout Toolbar and Annotation Toolbar 7-76
Chapter Summary 7-77
Chapter Terminology 7-78
Questions / Exercises 7-82

Chapter 8 - Assembly Drawings 8-1
Chapter Objective 8-3
Chapter Overview 8-3
CYLINDER Assembly - Exploded View 8-7
CYLINDER Assembly Drawing 8-9
Insert Balloons 8-11
Bill of Materials 8-18
Materials Editor and Mass Properties 8-21
Configuration Properties 8-25
Design Table 8-32
Bill of Materials – Part 2 8-38
Edition Cells 8-38
Edition Columns 8-38
Insert Column 8-40
Edit Header 8-40
Create an Equation 8-43
CYLINDER Assembly - Design Table 8-45
CYLINDER Assembly Drawing - Multiple Configurations 8-51
CYLINDER Assembly Drawing - Revision Table 8-60
CYLINDER Assembly Drawing - Section View, and Broken-out Section View 8-66
Hide Behind Plane 8-67
Introduction

Large Assembly Drawing Performance 8-68
Splitting a Bill of Materials Table 8-68
Dragging a Bill of Materials Table 8-69
Tabulated Bill of Materials Table 8-70
SolidWorks eDrawings 8-70
Export 8-70
Chapter Summary 8-70
Chapter Terminology 8-71
Questions / Exercises 8-75

Chapter 9 - Datums, Feature Control Frames, Geometric Tolerancing and other Drawing symbols 9-1
Chapter Objective 9-3
Chapter Overview 9-4
Drawing Template 9-6
VALVEPLATE1 Part - DimXpert 9-8
Feature Recondition 9-9
Settings 9-9
Reference Features 9-9
Block Tolerance 9-9
General Tolerance 9-9
DimXpert Toolbar 9-16
DimXpert Annotations and Drawings 9-17
DimXper - Part 9-21
VALVEPLATE1 Part - Datums, Feature Control Frames, Geometric Tolerance 9-26
VALVEPLATE1 Part - Surface Finish 9-31
ASME Y14.41 Digital Product Definition Data Practices 9-34
PLATE-TUBE Assembly Drawing and Weld Symbols 9-43
PLATE-CATALOG Drawing, Design Table, an EXCEL Formatting 9-50
Blocks 9-66
Geometric Tolerance Symbols 9-73
Fits - Types 9-73
Chapter Summary 9-74
Chapter Terminology 9-75
Questions / Exercises 9-78

Chapter 10 - Introduction to the Certified SolidWorks Associate Exam 10-1
Chapter Objective 10-3
Introduction 10-3
Intended Audience 10-3
CSWA Exam Content 10-5
About the Exam 10-9
Exam day 10-9
Drafting Competencies 10-14
Basic and Intermediate Part Creation and Modification 10-16
Advanced Part Creation and Modification 10-22
Assembly Creation and Modification 10-28
Appendix
ECO Form A-1
Types of Decimal Dimensions (ASME Y14.5M) A-2
SolidWorks Keyboard Shortcuts A-3
Windows Shortcuts A-3
Helpful On-Line information A-4

Index I-1
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.

Drawing refers to the SolidWorks module used to insert, add, and modify views in an engineering drawing. Detailing refers to the dimensions, notes, symbols, and Bill of Materials used to document the drawing.
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, 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!
The drawing reflects the changes of the part.

A Drawing template is the foundation for drawing information. Specified drawing standards and size, company information, manufacturing and or assembly requirements and more are included in a drawing template.

Drawing templates contain Document Properties settings such as millimeter or inch units and ANSI or ISO drawing standards.

Drawing templates also contain information included in the sheet format such as a Title block, company name, company logo, and custom properties.

A drawing is a 2D representation of a 3D part or assembly. SolidWorks utilizes various Orthographic views (Third Angle Projection or First Angle Projection) to display the 3D model on the 2D drawing. Note: All drawings in this book are displayed in Third Angle Projection.
Additional views represent a 3D model or assembly. Insert views from the Drawing tools in SolidWorks such as a Section view, Auxiliary view, or Detail view. Create additional views by combining Drawing tools with different part configurations. The Half Section Isometric view utilizes second configuration that controls the state of an Extruded Cut feature.

Annotations represent a text note or symbol that documents a part, assembly, or drawing.

Insert feature dimensions and annotations from the part or assembly into the drawing. Create additional reference dimensions and annotations in the drawing.

Address extension line gaps, dimension placement and line precedence.
Apply different configurations in a drawing. Assign properties such as material, mass, and cost to individual parts in part and assembly Design Tables. Incorporate multiple properties into the drawing Bill of Materials.

The step-by-step approach used in this text works with multiple parts and assemblies to create and to modify engineering drawings. Understanding design intent assists you in implementing changes.
**Design Intent**

The SolidWorks definition of design intent is the process in which the model is developed to accept future changes.

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:

1. Sketch
2. Feature
3. Part
4. Assembly
5. Drawing

**1. Design Intent in the Sketch**

Build design intent in a sketch as the profile is created. A profile is determined from the selected Sketch Entity. Example: Corner Rectangle, Circle, Arc, Point, etc.

Apply symmetry into a profile through a sketch centerline, mirror entity and position about the reference planes and Origin.

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 contains Horizontal, Vertical, and Perpendicular automatic Geometric relations. Apply design intent using added Geometric relations. Example: Horizontal, Vertical, Collinear, Perpendicular, Parallel etc.

Example A: Apply design intent to create a square profile. Sketch a rectangle. Apply the Center Rectangle tool. Note: No construction reference centerline or Midpoint relation is required with the Center Rectangle tool. Insert dimensions to define the square.

Example B: Develop a rectangular profile. Apply the Corner Rectangle tool. The bottom horizontal midpoint of the rectangular profile is located at the Origin. Add a Midpoint relation between the horizontal edge of the rectangle and the Origin. Insert two dimensions to define the width and height of the rectangle as illustrated.

2. Design Intent in the Feature

Build design intent into a feature by addressing symmetry, feature selection, and the order of feature creation.

Example A: The Boss-Extrude1 feature (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: Do you create each tooth separate using the Extruded Cut feature? No. Create a single tooth and then apply the Circular Pattern feature. Create 34 teeth for a Circular Pattern feature. Modify the number of teeth from 32 to 24.
3. **Design Intent in the Part**

Utilize symmetry, feature order and reusing common features to build design intent into the 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.

4. **Design Intent in the 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.

5. **Design Intent in the Drawing**

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

Chapter 1: History of Engineering Graphics

Chapter 1 provides a broad discussion of the history of Engineering Graphics and the evolution from manual drawing/drafting along with an understanding of general sketching techniques, alphabet of lines, precedence of line types and Orthographic projection.

It also addresses the Glass Box method and the six principle orthographic views along with the difference between First Angle Projection and Third Angle Projection.

Chapter 2: Isometric Projection and Multi View Drawings

Chapter 2 provides a general introduction into Isometric Projection and sketching along with Additional Projections and arrangement of views.

It also covers advanced drawing views and an introduction to the evolution of from manual drafting to early CAD systems and finally to SolidWorks.

Chapter 3: Dimensioning Practices, Tolerancing and Fasteners

Chapter 3 provides a general introduction into dimensioning practices and systems, and the ASME ANSI Y14.5 standards along with fits, fasteners and general tolerancing.
Chapter 4: SolidWorks 2012 User Interface

SolidWorks is a design software application used to model and create 2D and 3D sketches, 3D parts and assemblies, and 2D drawings. Chapter 4 introduces you to the SolidWorks 2012 User Interface 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.

Chapter 5: Drawing Templates and Sheet Formats

Explore the SolidWorks drawing template. Apply Document Properties to reflect the ASME Y14 Engineering Drawing Standards.

Investigate the differences between a Sheet format and a Drawing template. Create two Drawing templates. Create a C-size Drawing template and an A-size Drawing template. Create a C-size Sheet format.

Import an AutoCAD drawing to create a new Sheet format. Apply SolidWorks Properties and Custom Properties in the Sheet format. Combine the Sheet format with an empty drawing template to create a custom Drawing template.
Chapter 6: Drawings and Various Drawing Views

Create three drawings: TUBE, ROD, and COVERPLATE. Insert the following drawing views: Front Top, Right, Isometric, Auxiliary, Detail, Section, Crop, Broken Section, Half Section, Revolved Section, Offset Section, Removed, Projected, Aligned Section, and more.

Insert, modify, suppress, unsuppressed, and delete drawing views and dimensions. Create multi-sheet drawings from various part configurations.

Chapter 7: Fundamentals of Detailing

Insert dimensions and annotations required to detail the TUBE and COVERPLATE drawings.

Insert, add, and modify dimensions for part features. Insert and add notes to the drawing.

Incorporate drawing standards to document specific features.
Chapter 8: Assembly Drawings

Develop the CYLINDER assembly. Combine configurations of the TUBE, ROD and COVERPLATE components.

Obtain an understanding of Custom Properties and SolidWorks Properties.

Combine Properties in a Bill of Materials.

Create a design table in the assembly. Incorporate the Bill of Materials and different configurations into a multi-sheet drawing.

Chapter 9: Datums, Feature Control Frames, Geometric Tolerancing and other Drawing Symbols

Create five drawings: VALVEPLATE1, VALVEPLATE1-GDT, VALVEPLATE1-GDT eDrawing, PLATE-TUBE, PLATE-CATALOG, and modify the ASME14-41 drawing.

Apply DimXpert and the DimXpert Manager. Insert Feature Control Frames, Datum Feature Symbols, Geometric Tolerance, Weld Symbols, Surface Finish Symbols, and more using DimXpert and manual techniques. Format a Design Table in EXCEL.
Chapter 10: Introduction to the Certified SolidWorks Associate Exam

Chapter 10 provides a basic introduction into the curriculum and exam categories for the Certified SolidWorks Associated CSWA Certification program. Review the exam procedure, process and required model knowledge needed to take and pass the exam.

- Review the five exam categories: Drafting Competencies, Basic Part Creation and Modification, Intermediate Part Creation and Modification, Advanced Part Creation and Modification, and Assembly Creation and Modification

All model files for Chapter 10 are located in the Chapter 10 CSWA Models folder on the DVD.

💡 View the Certified SolidWorks Associate CSWA exam pdf file on the enclosed DVD for a sample exam.
**About the Book**

The following conventions are used throughout this book:

- The term document is used to refer 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**, 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. IPS (inch, pound, second) and MMGS (millimeter, gram, second) unit systems are used.

- The book is organized into various chapters. Each chapter is focused on a specific subject or feature. Additional pdf and ppt information and folders/models are provided on the enclosed DVD.

- All templates, logos and needed model documents for this book are included on the enclosed DVD. Copy the information from the DVD to your local hard drive. Work from your local hard drive.

- Screen shots in the book were made using SolidWorks 2012 SP0 running Windows® 7.
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>
</tr>
<tr>
<td></td>
<td>• Selected geometry: line, circle.</td>
<td>• Select the <strong>centerpoint</strong>.</td>
</tr>
<tr>
<td></td>
<td>• Value entries.</td>
<td>• Enter <strong>3.0</strong> for Radius.</td>
</tr>
<tr>
<td>Capitalized</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>
</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>ToolTip</td>
<td>Position the mouse pointer over an Icon (button). The tool name is displayed 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 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 Instructors DVD contains PowerPoint presentations, Adobe files along with avi files, Term projects, quizzes with the initial and final SolidWorks models.

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 sites:
http://www.solidworks.com/goedu and
Notes: