INTRODUCTION

Assembly Modeling with SolidWorks is written to assist the beginning SolidWorks user with a few months of design experience to the *intermediate* SolidWorks user who desires to enhance their skill sets in assembly modeling. The book provides a solid foundation in assembly modeling using competency-based projects. In step-by-step instructions, the book provides examples to:

- Start a SolidWorks 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, CommandManager, and more*.
- Set System Options and Document Properties as they applied to a part and assembly template. Create new SolidWorks folder locations: *Document Templates, Reference Documents, and Design Library*.
- Download components from 3D ContentCentral and rename and save components using SolidWorks Explorer.
- Apply the Bottom-up assembly approach with two levels of configurations using the Configure Component tool, the Configure Dimension tool, Design Tables, and the Add Configuration tool.
- Create new parts based on component features utilizing the Bottom-up assembly approach.
- Apply Standard Mates, SmartMates, and the Design Library Toolbox.
- Apply the Top-down assembly approach with two levels of configurations with In-Context components.
- Understand the following: *Out-of-Context components, External References, InPlace Mates, redefining and replacing components and motion studies.*
- Apply the Derived Feature Component Pattern tool, Linear Component Pattern tool, and the Mirror Component tool along with the Explode Line Sketch tool.
- Create a multi sheet, multi view assembly drawing. Knowledge of Custom Properties in a part/assembly and linked notes, with the ability to incorporate configurations of an Exploded view, Bill of Materials, Revision tables, and more.
- Address the Layout-based assembly approach and Global Variables and Equations to control relationships.

Each chapter begins with the desired outcomes and usage competencies. Explore assembly modeling techniques through a series of design situations, industry scenarios, projects and objectives.

Chapter 9 provides a bonus section on the *Certified SolidWorks Associate CSWA program* with sample exam questions and initial and final SolidWorks models. Passing the CSWA exam proves to employers that you have the necessary fundamental engineering graphics and SolidWorks competencies.

The book compliments and enhances the **SolidWorks Tutorials**. Although over 150 SolidWorks tools and commands are utilized in **Assembly Modeling with SolidWorks 2012,** the book is not a reference guide.

The book is a self-paced tutorial in a realistic design setting. Complex models expose you to large assembly modeling techniques. You focus on the design process while learning the commands relative to assemblies.

To obtain the most from this text, you should be familiar with the SolidWorks User Interface or other parametric modeling software application. Your skill sets should include the ability to create simple parts, assemblies, and drawings and manipulate documents through the Windows operating system.

The authors developed the industry scenarios by combining their own industry experience with the knowledge of engineers, department managers, vendors and manufacturers. These professionals are directly involved with SolidWorks everyday. They create assemblies with thousands of components and drawings with hundreds of sheets. Their responsibilities go far beyond the creation of just a 3D model.

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 for SolidWorks® 2012**, 2011, 2010, 2009 and 2008
- **A Commands Guide Reference Tutorial for SolidWorks® 2007**
- **Assembly Modeling with SolidWorks® 2012**, 2010, 2009, 2008, 2006, 2005-2004, 2003 and 2001Plus
- **Drawing and Detailing with SolidWorks® 2012**, 2010, 2009, 2008, 2007, 2006, 2005, 2004, 2003, 2002 and 2001/2001Plus
- **Engineering Design with SolidWorks® with Video Instruction 2012**, 2011, 2010, 2009, 2008, 2007, 2006, 2005, 2004, 2003, 2001Plus, 2001 and 1999
- **Engineering Graphics with SolidWorks with Video Instruction 2012**, 2011, and 2010
- **SolidWorks® The Basics with Multimedia CD 2009**, 2008, 2007, 2006, 2005, 2004 and 2003
- **SolidWorks® Tutorial with Video Instruction, 2012**, 2011, 2010, 2009, 2008, 2007, 2006, 2005, 2004, 2003 and 2001/2001Plus
- **The Fundamentals of SolidWorks®: Featuring the VEXplorer robot**, **2008** and 2007
- **Official Certified SolidWorks® Associate Examination Guide, Version 3; 2011, 2010, 2009,** Version 2; 2010, 2009, 2008, Version 1; 2007
- **Official Certified SolidWorks® Professional (CSWP) Certification Guide with Video Instruction DVD, 2011, 2010**
- **Applications in Sheet Metal Using Pro/SHEETMETAL & Pro/ENGINEER**

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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.

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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 7th edition of the 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 contact us with any comments, questions, or suggestions on this book or any of our other SolidWorks books. David Planchard, D & M Education, LLC, dplanchard@msn.com or visit our website at **www.dmeducation.net**.

Note to Instructors

Please contact the publisher: http://www.schroff.com for additional materials that will support the usage of this text in your classroom.

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- Simpson Strong Tie Product Manual, Simpson Strong Tie, CA, 2009.

 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

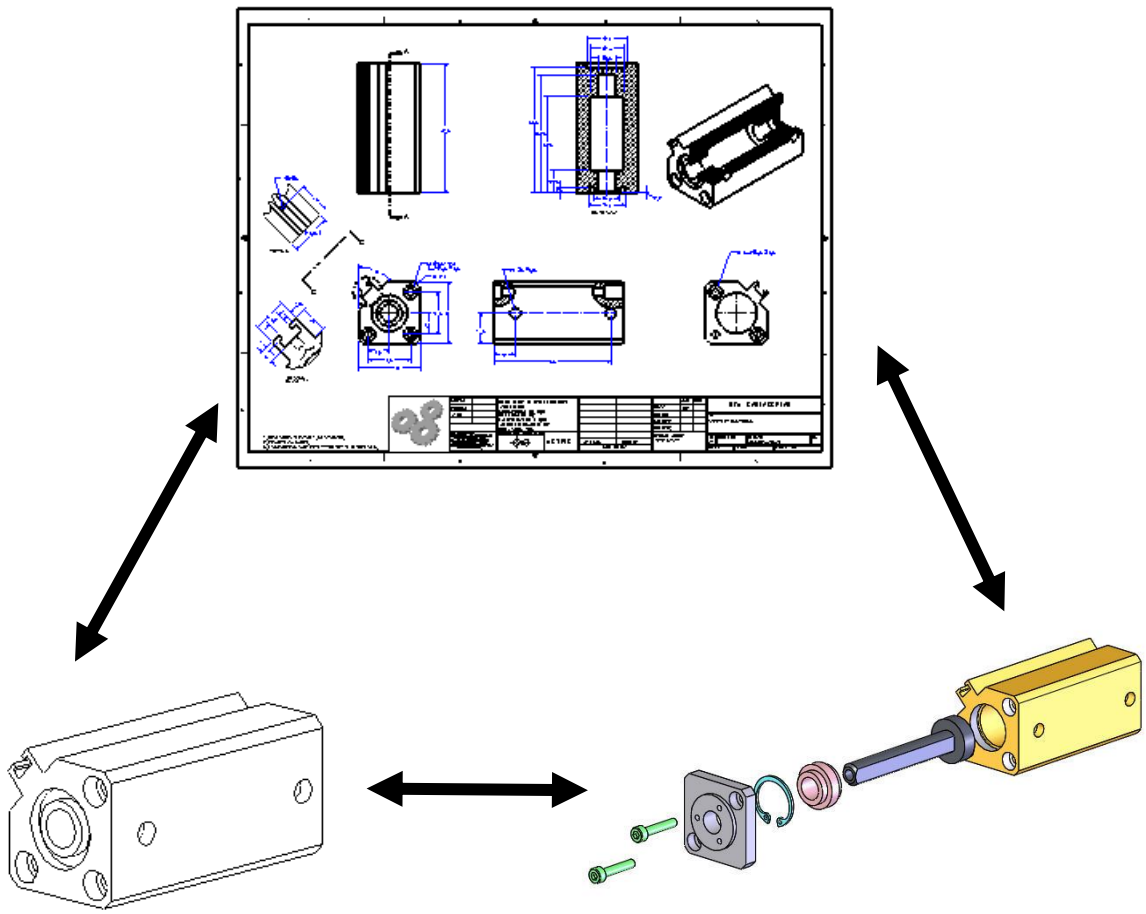
What is SolidWorks?

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.

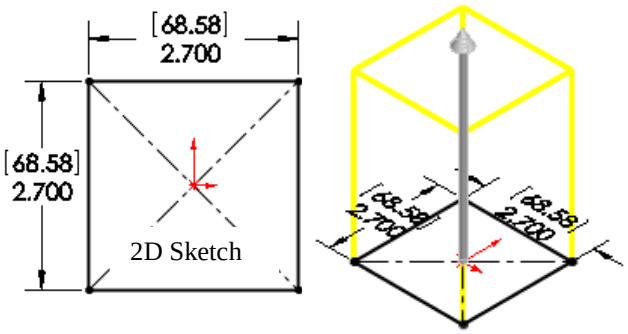
SolidWorks is an easy to learn design and analysis tool, (SolidWorks SimulationXpress, SolidWorks Motion, SolidWorks Flow Simulation, etc.) which makes it possible for designers to quickly sketch 2D and 3D concepts, create 3D parts and assemblies and detail 2D drawings.

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.

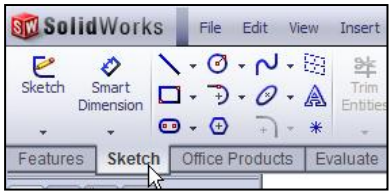


Features are the building blocks of parts. Use feature tools such as: Extruded Boss/Base, Extruded Cut, Fillet, etc. from the Features tab in the CommandManager to create 3D parts.

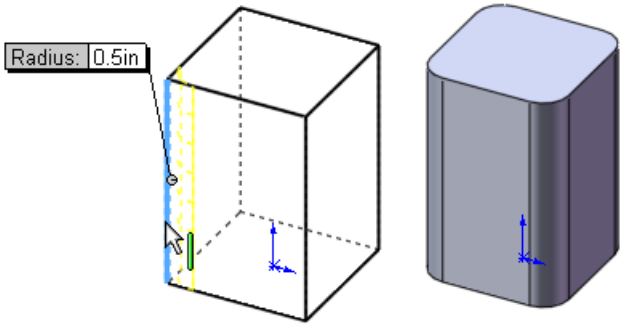


Extruded features begin with a 2D sketch created on a Sketch plane.

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!



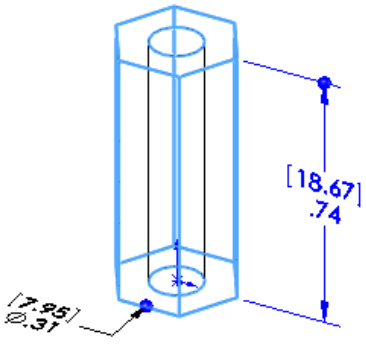
Create features by selecting edges or faces of existing features, such as a Fillet. The Fillet feature rounds sharp corners.



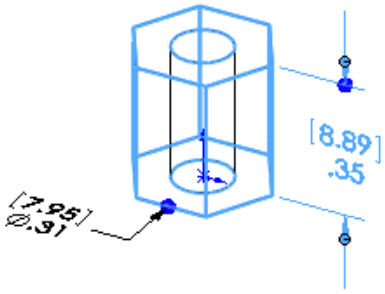
Dimensions drive features. Change a dimension, and you change the size of the part.

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

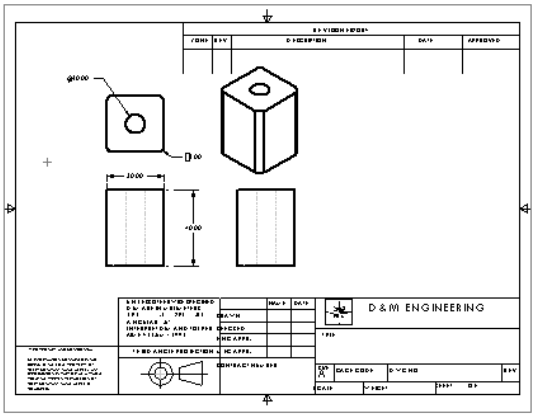
Create a hole that penetrates through a part (Through All). SolidWorks maintains relationships through the change.

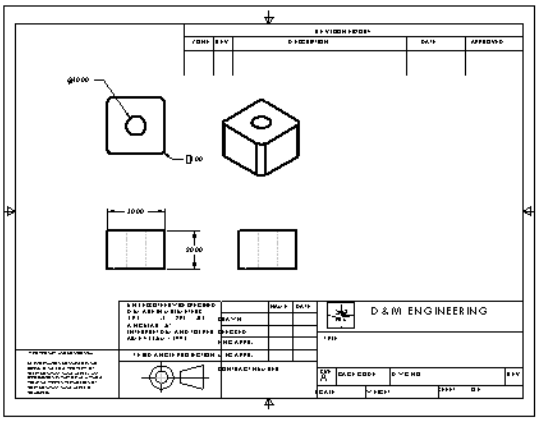


The step-by-step approach used in this text allows you to create, edit and modify parts, assemblies and drawings. Change is an integral part of design!



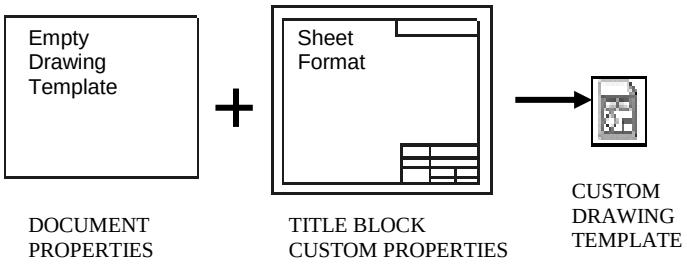
The drawing reflects the changes of the part.





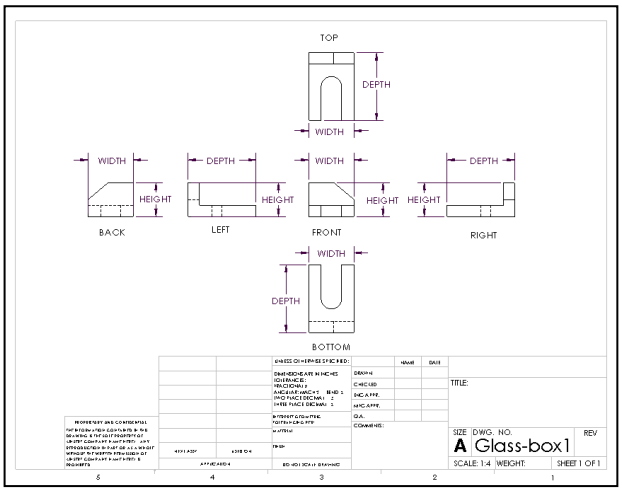
A Drawing template is the foundation for drawing information. Specified drawing standards and size, company information, manufacturing and or assembly requirements and more are included in a drawing template.

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.



Design Intent

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

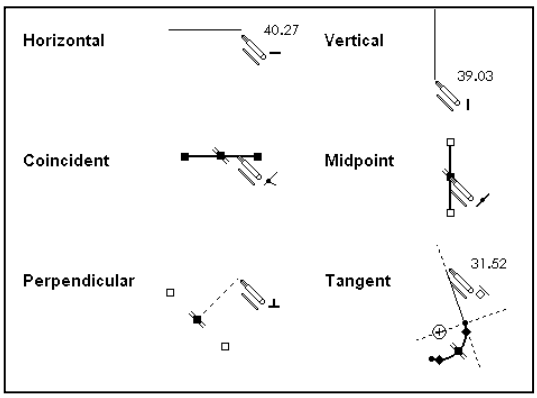
Models behave differently when design changes occur. Design for change. Utilize geometry for symmetry, reuse common features and reuse common parts.

Build change into the following areas:

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

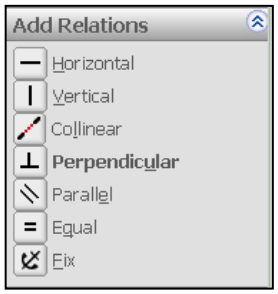
1. Design Intent in the Sketch

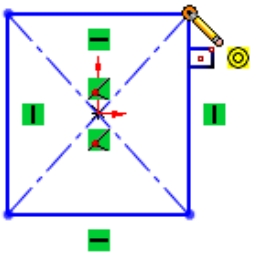
Build design intent in a sketch as the profile is created. A profile is determined from the selected Sketch Entity. Example: Corner Rectangle, Circle, Arc, Point, etc.



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

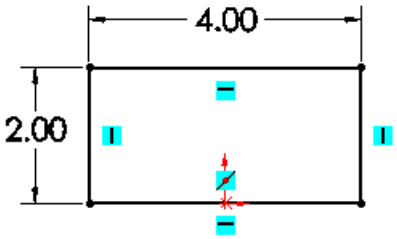
Build design intent as you sketch with automatic Geometric relations. Document the decisions made during the up-front design process. This is very valuable when you modify the design later.





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

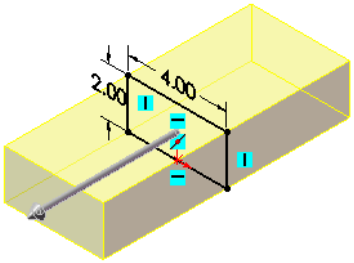
Example A: Apply design intent to create a square profile. Sketch a rectangle. Apply the Center Rectangle tool. Note: No construction reference centerline or Midpoint relation is required with the Center Rectangle tool. Insert dimensions to define the square.



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.

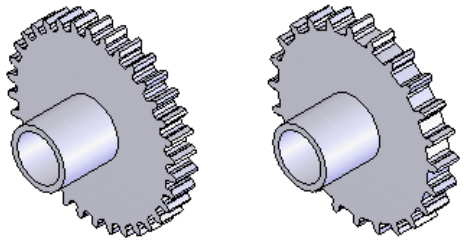
2. Design Intent in the Feature

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



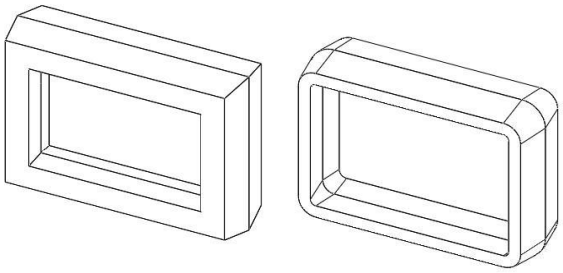
Example A: The Boss-Extrude1 feature (Base feature) remains symmetric about the Front Plane. Utilize the Mid Plane End Condition option in Direction 1. Modify the depth, and the feature remains symmetric about the Front Plane.

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

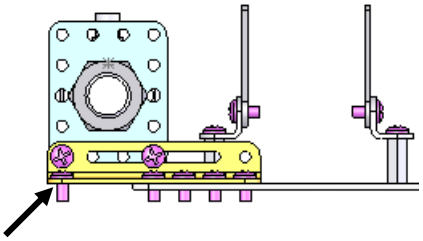
Utilize symmetry, feature order and reusing common features to build design intent into the part.



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

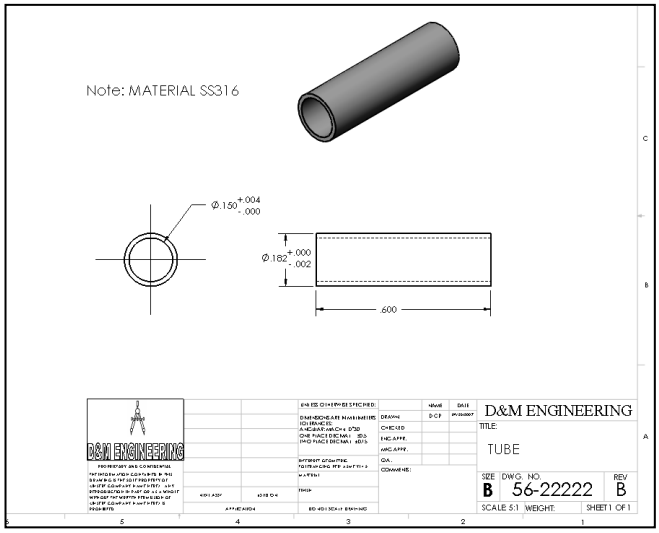
Utilizing symmetry, reusing common parts and using the Mate relation between parts builds the design intent into an assembly.



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.

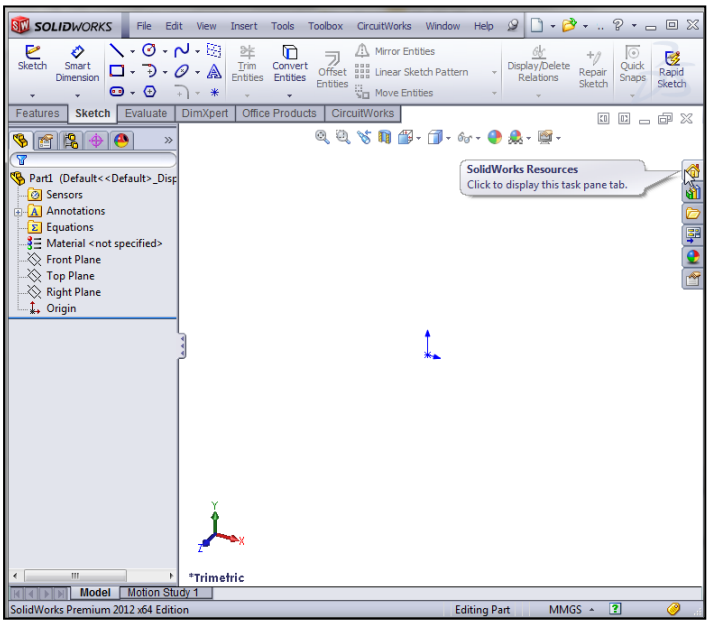


Example A: Tolerance and material in the drawing.

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

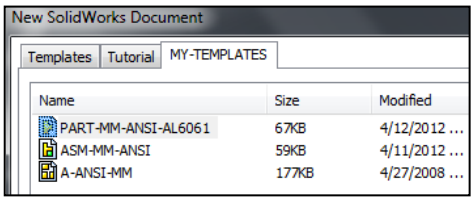


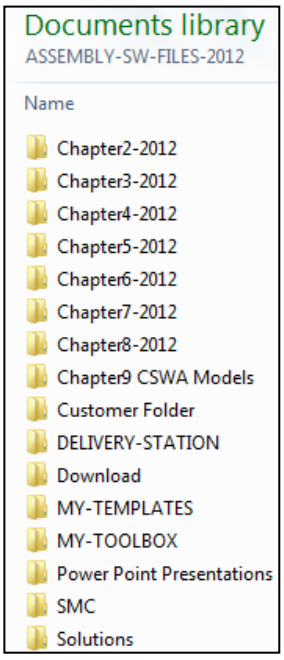
Chapter 1: SolidWorks 2012 User Interface

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

Chapter 1 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 2: File Management, System Options, Templates, SolidWorks Explorer and more





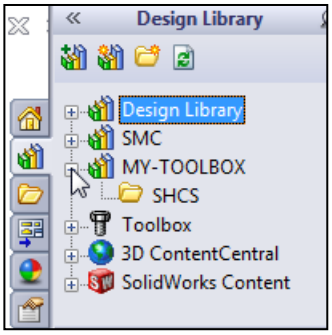
Create two templates: ASM-MM-ANSI Assembly template and the PART-MM-ANSI-AL6061 Part template.

Create new SolidWorks File Locations: Document Templates, Reference Documents, and Design Library folders in the Task Pane.

Use downloaded components from 3ContentCentral on the DVD in the book. Rename and save components using SolidWorks Explorer.

All initial and final models are located on the enclosed DVD. All required 3DContentCentral components are located on the DVD in the SMC folder.

Chapter 3: Assembly Modeling - Bottom - up design approach

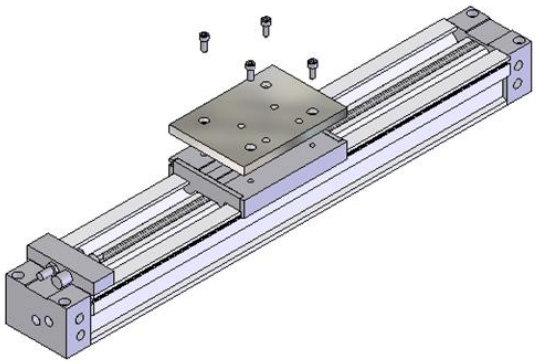


Develop the LINEAR-TRANSFER assembly. The LINEAR-TRANSFER assembly is the first assembly in the 3AXIS-TRANSFER assembly.

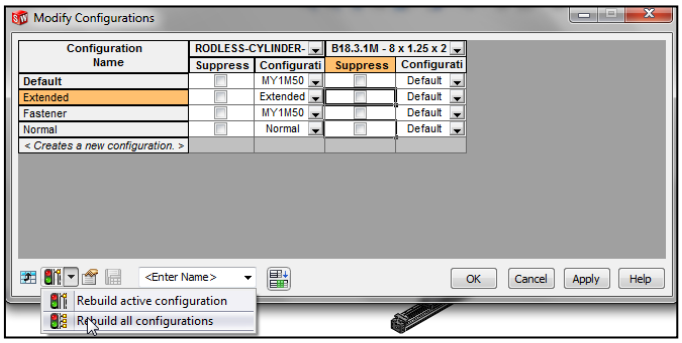
Create the following models: PLATE-A part, and the LINEAR-TRANSFER assembly.

Insert Standard Mates and SmartMates, along with four M8 x 1.25 Socket Head Cap Screws.

Apply the Design Library Toolbox and the Measure tool.



Chapter 4: Bottom-up design assembly approach - Two Levels of Configurations

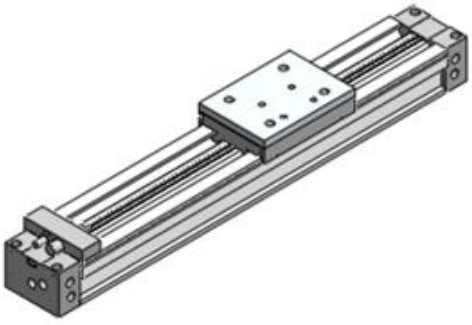


Develop two levels of configurations for the RODLESS-CYLINDER assembly using the Configure component tool to illustrate dynamic motion and physical location.

Create two new RODLESS-CYLINDER configurations: *Normal*, and *Extended.*

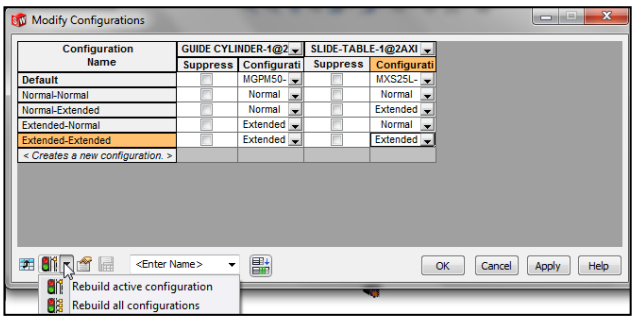
Create three new configurations for the LINEAR-TRANSFER assembly: *Normal, Extended*, and *Fastener*.

Insert a Derived Feature Component Pattern and apply the Collision Detection tool.



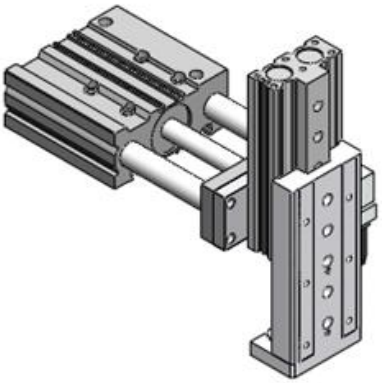
Solutions to each chapter; is provided in the Solutions folder on the DVD in the book.

Chapter 5: Top-down assembly approach - Two Components with Configurations



Create the 2AXIS-TRANSFER assembly. Design the PLATE-B part In-Context of the GUIDE-CYLINDER and SLIDE-TABLE assemblies.

Utilize the Configure Dimension tool to create new configurations for the GUIDE-CYLINDER and SLIDE-TABLE.

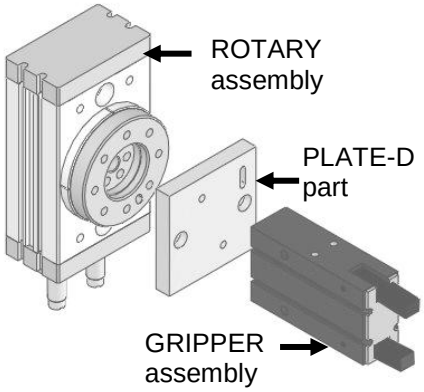


Utilize the Configure Component tool to create configurations for the new 2AXIS-TRANSFER assembly. The 2AXIS-TRANSFER assembly is the second component in the 3AXIS-TRANSFER assembly.

Obtain knowledge of In-Context methods used in Top-down Assembly modeling, Out-of-Context components, External References and InPlace Mates.

Develop two levels of configurations.

Chapter 6: Part and Assembly Configurations, Custom Properties, Design Tables and References



ROTARY-GRIPPER assembly

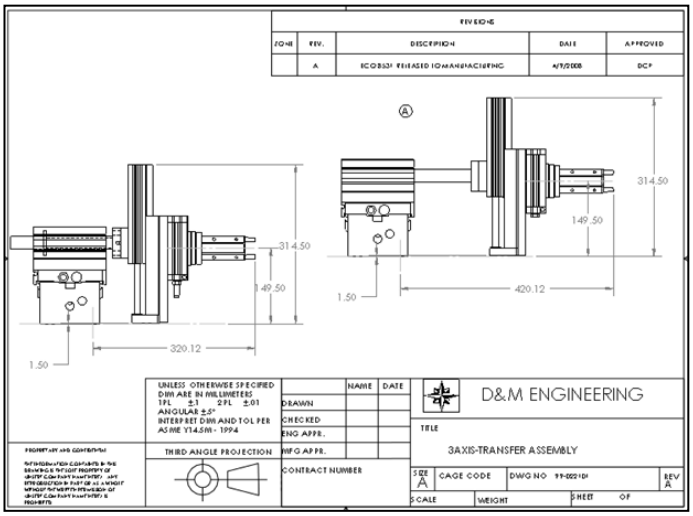
Create the ROTARY-GRIPPER assembly. The ROTARY-GRIPPER assembly is the third component in the 3AXIS-TRANSFER assembly. Create the PLATE-C part. Insert the PLATE-C part into the 2AXIS-TRANSFER assembly with no External References.

Create the PLATE-D part In-Context of the ROTARY and GRIPPER assembly. Delete all InPlace Mates.

Utilize the Add Configuration tool and the Design Table tool to create multi configurations in the ROTARY assembly, ROTARY-GRIPPER assembly, and the PLATE-D part.

Develop Custom Properties for the PLATE-D part and create the 2AXIS-TRANSFER assembly.

Chapter 7: Assembly Drawings with Revision Table and Bill of Materials - Multiple Sheets, Views and Custom Properties



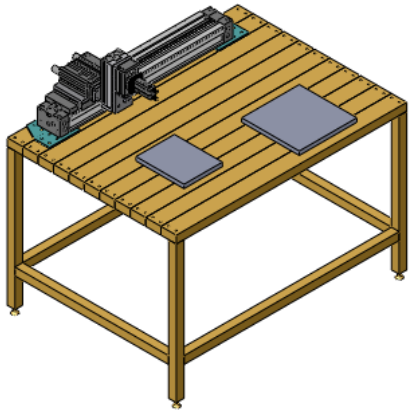
Create the 3AXIS-TRANSFER assembly. Utilize the Configure Component tool to create eight position configurations and a Fastener configuration using the following assemblies: *LINEAR-TRANSFER, 2AXIS-TRANSFER,* and the *ROTARY-GRIPPER.*

Create the 3AXIS-TRANSFER drawing. Insert Fasteners into the 3AXIS-TRANSFER assembly.

Insert Custom Properties to the components in the 3AXIS-TRANSFER assembly. Apply the Replace Component tool to the 3AXIS-TRANSFER assembly.

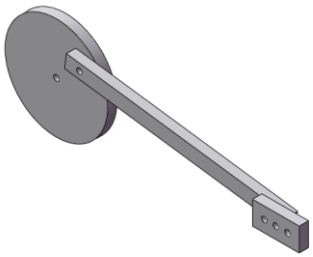
Develop an Exploded Isometric View, Linked Notes, Revision Table, and Bill of Materials in the drawing. Create multiple configurations and multiple sheets.

Chapter 8: Top-down design, Layout Sketches, Blocks, Motion and more



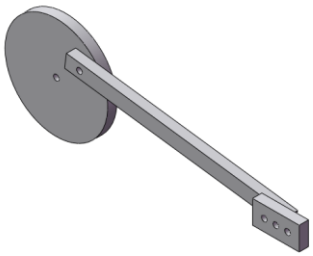
Create the final DELIVERY-STATION assembly utilizing the Top-down assembly approach.

Create the INPUT-BASE-PLATE part and reordered components in the assembly.



Create Global Variables and apply Equations to control relationships along with using the Component Pattern tool, Mirror Components tool, Explode Line Sketch tool, Join feature, and Split feature.

Create two Motion Studies using the Linear motor, Rotary motor, and gravity options.

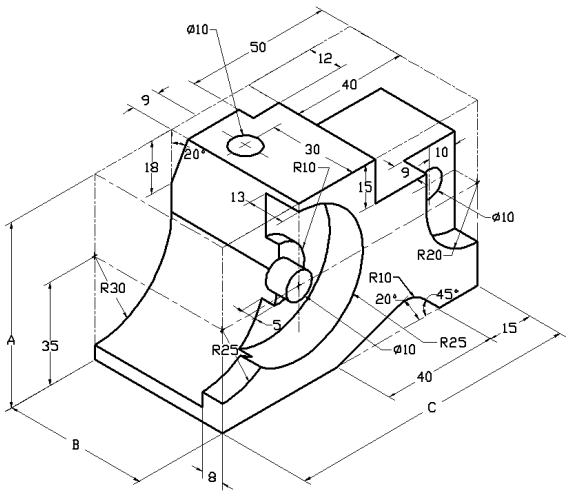


Apply the Layout-based assembly design with blocks to create motion. The AssemblyXpert tool and envelopes are explored.

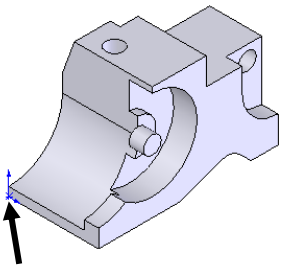
Chapter 9: Introduction to the Certified SolidWorks Associate Exam

Chapter 9 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*



Given:
A = 63, B = 50, C = 100
Material: Copper
Units: MMGS
Density: .0089 g/mm^3
All HOLES THROUGH ALL



Origin

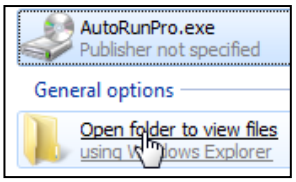
All model files for Chapter 9 are located in the Chapter 9 CSWA Models folder on the DVD.

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

About the Book

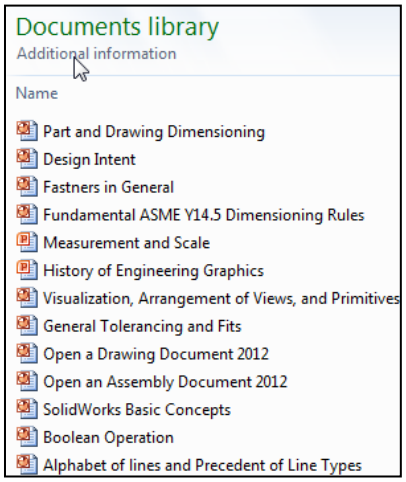
The following conventions are used throughout this book:

- 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 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.

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

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:

Item:	Description:
Click	Press and release the left mouse button.
Double-click	Double press and release the left mouse button.
Click inside	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.
Drag	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.
Right-click	Press and release the right mouse button. A Shortcut menu is displayed. Use the left mouse button to select a menu command.
ToolTip	Position the mouse pointer over an Icon (button). The tool name is displayed below the mouse pointer.
Large ToolTip	Position the mouse pointer over an Icon (button). The tool name and a description of its functionality are displayed below the mouse pointer.
Mouse pointer feedback	Position the mouse pointer over various areas of the sketch, part, assembly or drawing. The cursor provides feedback depending on the geometry.

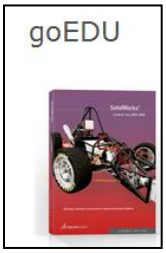
A mouse with a center wheel provides additional functionality in SolidWorks. Roll the center wheel downward to enlarge the model in the Graphics window. Hold the center wheel down. Drag the mouse in the Graphics window to rotate the model.

Visit SolidWorks website: http://www.solidworks.com/sw/support/hardware.html to view their supported operating systems and hardware requirements.

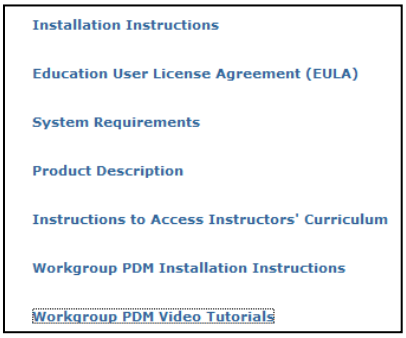
SolidWorks

Operating Systems	SolidWorks 2009	SolidWorks 2010	SolidWorks 2011	SolidWorks 2012	(SolidWorks 2013)
Windows 7	✘	✔*	✔	✔	✔
Windows Vista	✔	✔	✔	✔	✔
Windows XP	✔	✔	✔	✔	✘
Minimum Hardware	Configuring a SolidWorks Workstation				
RAM	2 GB or more				
Disk Space	5 GB or more				
Video Card	Certified cards and drivers				
Processor	Intel or AMD with SSE2 support. 64-bit operating system recommended				
Install Media	DVD Drive or Broadband Internet Connection				

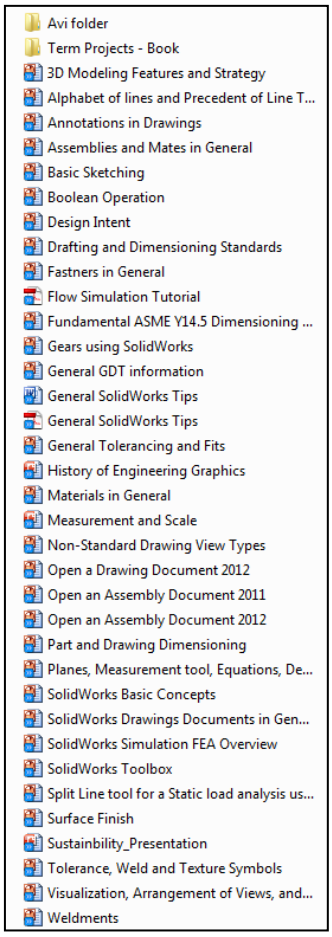
The Instructors DVD contains PowerPoint presentations, Adobe files along with avi files, Term projects, quizzes with the initial and final SolidWorks models.



The book is design to expose the new user to numerous tools and procedures. It may not always use the simplest and most direct process.



The book does not cover starting a SolidWorks session in detail for the first time. A default SolidWorks installation presents you with several options. For additional information for an Education Edition, visit the following sites: http://www.solidworks.com/goedu and http://www.solidworks.com/sw/education/6443_ENU_HTML.htm.



Notes: