INTRODUCTION

The **Official Certified SolidWorks Associate** (**CSWA**) **Examination Guide** is written to assist the SolidWorks user to pass the CSWA exam.

DS SolidWorks Corp. offers various stages of certification. Each stage represents increasing

levels of expertise in 3D CAD design as it applies to engineering: *Certified SolidWorks Associate CSWA, Certified SolidWorks Professional CSWP and Certified SolidWorks Expert CSWE* along with specialty fields in Simulation, Sheet Metal, and Surfacing.

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 on-line proctored 180 minute exam (minimum of 165 out of 240 points). The new CSWA exam consists of fourteen questions in five categories. Passing this exam provides students the chance to prove their knowledge and expertise and to be part of a worldwide industry certification standard.

Copy the corresponding CSWA Model Folder from the CD in the book that matches your release of SolidWorks to your hard drive. Work directly from your hard drive on the tutorials in this book. SolidWorks Model files for 2009, 2010, and 2011 are provided.

Goals

The primary goal is not only to help you pass the CSWA exam, but also to ensure that you understand and comprehend the concepts and implementation details of the CSWA process.

The second goal is to provide the most comprehensive coverage of CSWA exam related topics available, without too much coverage of topics not on the exam.

The third and ultimate goal is to get you from where you are today to the point that you can confidently pass the CSWA exam.







CSWA Audience

The intended audience for this book and the CSWA exam is anyone with a minimum of 6 - 9 months of SolidWorks experience and basic knowledge of engineering fundamentals and practices. SolidWorks recommends that you review their SolidWorks Tutorials on Parts, Assemblies, Drawings as a prerequisite and have at least 45 hours of classroom time learning SolidWorks or using SolidWorks with basic engineering design principles and practices.

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, Society of Automotive Engineering (SAE) and the American Society of Engineering Education (ASEE). David holds a BSME, MSM with the following Professional Certifications: CCAI, CCNA, CCNP, CSWA, and CSWP. David is a SolidWorks Solution Partner and an Adjunct Faculty member at Worcester Polytechnic Institute in the Mechanical Engineering department.

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David and Marie Planchard are co-authors of the following books:

- A Commands Guide for SolidWorks[®] 2012, 2011, 2010, 2009 and 2008
- Assembly Modeling with SolidWorks[®] 2012, 2010, 2008, 2006, 2005-2004, 2003 and 2001Plus
- Drawing and Detailing with SolidWorks[®] 2012, 2010, 2010, 2009, 2008, 2007, 2006, 2005, 2004, 2003, 2002 and 2001/2001Plus
- Engineering Design with SolidWorks[®] with Multimedia CD 2012, 2011, 2010, 2009, 2008, 2007, 2006, 2005, 2004, 2003, 2001Plus, 2001 and 1999
- Engineering Graphics with SolidWorks[®] with Multimedia CD 2012, 2011, 2010
- SolidWorks[®] The Basics with Multimedia CD 2009, 2008, 2007, 2006, 2005, 2004 and 2003

- SolidWorks[®] Tutorial with Multimedia CD 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 Multimedia DVD, 2011, 2010
- 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 M. Sullivan and Professor Robert L. 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 text.

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 and Professor Dennis Hance of Wright State 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 3rd 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.

Note to Instructors

Please contact the publisher **www.schroff.com** for additional classroom support materials (PowerPoint 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.

Trademarks, Disclaimer and Copyrighted Material

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The Y14 ASME Engineering Drawing and Related Documentation Publications utilized in this text are as follows: ASME Y14.1 1995, ASME Y14.2M-1992 (R1998), ASME Y14.3M-1994 (R1999), ASME Y14.41-2003, ASME Y14.5-1982, ASME Y14.5M-1994, and ASME B4.2. 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.

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During the initial SolidWorks installation, you are requested to select either the ISO or ANSI drafting standard. ISO is typically a European drafting standard and uses First Angle Projection. The book is written using the ANSI (US) overall drafting standard and Third Angle Projection for drawings.



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What is SolidWorks?

SolidWorks is a design automation software package used to produce parts, assemblies and drawings. SolidWorks is a Windows native 3D solid modeling CAD program. SolidWorks provides easy to use, highest quality design software for engineers and designers who create 3D models and 2D drawings ranging from individual parts to assemblies with thousands of parts.

The SolidWorks Corporation, headquartered in Concord, Massachusetts, USA develops and markets innovative design solutions for the Microsoft Windows platform. Additional information on SolidWorks and its family of products can be obtained at their URL, www.SolidWorks.com.

In SolidWorks, you create 3D parts, 3D assemblies and 2D drawings. The part, assembly and drawing documents are related.



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Features are the building blocks of parts. Use features to create parts, such as: Extruded Boss/Base and Extruded Cut. Extruded features begin with a 2D sketch created on a Sketch plane.

The 2D sketch is a profile or cross section. Sketch tools such as: lines, arcs and circles are used to create the 2D sketch. Sketch the general shape of the profile. Add Geometric relationships and dimensions to control the exact size of the geometry.

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

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

Apply Geometric relationships: Vertical, Horizontal, Parallel, etc. to maintain Design intent.

Create a hole that penetrates through a part. SolidWorks maintains relationships through the change.

The step-by-step approach used in this text allows you to create parts, assemblies and drawings by doing, not just by reading.

The book provides the knowledge to modify all parts and components in a document. Change is an integral part of design.









Design Intent

What is design intent? All designs are created for a purpose. Design intent is the intellectual arrangements of features and dimensions of a design. Design intent governs the relationship between sketches in a feature, features in a part and parts in an assembly.

The SolidWorks definition of design intent is the process in which the model is developed to accept future modifications. Models behave differently when design changes occur.

Design for change! Utilize geometry for symmetry, reuse common features, and reuse common parts. Build change into the following areas that you create:

- Sketch
- Feature
- Part
- Assembly
- Drawing

When editing or repairing geometric relations, it is considered best practice to edit the relation vs. deleting it.

Design Intent in a Sketch

Build design intent in a sketch as the profile is created. A profile is determined from the Sketch Entities. Example: Rectangle, Circle, Arc, Point, Slot etc. Apply symmetry into a profile through a sketch centerline, mirror entity and position about the reference planes and Origin. Always know the location of the Origin in the sketch.

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

A rectangle (Center Rectangle Sketch tool) contains Horizontal, Vertical and Perpendicular automatic Geometric relations.

Apply design intent using added Geometric relations if needed. Example: Horizontal, Vertical, Collinear, Perpendicular, Parallel, Equal etc.





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

Example B: If you have a hole in a part that must always be $16.5 \text{mm} \leq \text{from an edge}$, dimension to the edge rather than to another point on the sketch. As the part size is modified, the hole location remains $16.5 \text{mm} \leq \text{from the}$ edge as illustrated.

Design intent in a Feature

Build design intent into a feature by addressing End Conditions (Blind, Through All, Up To Vertex, etc.) symmetry, feature selection, and the order of feature creation.

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

Example B: Create 34 teeth in the model. Do you create each tooth separate using the Extruded Cut feature? No.

Create a single tooth and then apply the Circular Pattern feature. Modify the Circular Pattern from 32 to 24 teeth.

Design intent in a Part

Utilize symmetry, feature order and reusing common features to build design intent into a part. Example A: Feature order. Is the entire part symmetric? Feature order affects the part.

Apply the Shell feature before the Fillet feature and the inside corners remain perpendicular.





Design intent in an Assembly

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

Example A: Reuse geometry in an assembly. The assembly contains a linear pattern of holes. Insert one screw into the first hole. Utilize the Component Pattern feature to copy the machine screw to the other holes.

Design intent in a Drawing

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

Example A: Tolerance and material in the drawing. Insert an outside diameter tolerance +.000/-.002 into the TUBE part. The tolerance propagates to the drawing.

Define the Custom Property Material in the Part. The Material Custom Property propagates to your drawing.

Create a sketch on any of the default planes: Front, Top, Right or a created plane.

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

The book is design to expose the new user to many tools, techniques and procedures. It may not always use the most direct tool or process.

Every license of SolidWorks 2011 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.







About the Book

You will find a wealth of information in this book. The book is written for new and intermediate users. The following conventions are used throughout this book:

- The term document refers to a SolidWorks part, drawing or • assembly file.
- The list of items across the top of the • SolidWorks interface is the Menu bar menu or the Menu bar toolbar. Each item in the Menu bar has a pull-down menu. When you need to select a series of commands from these menus, the following format is used: Click Insert,





Reference Geometry, **Plane** from the Menu bar. The Plane PropertyManager is displayed.

- The book is organized into Chapters. Each Chapter is focused on a specific subject or ٠ feature. Use the enclosed CD to obtain parts and models that are used in this book and to view the features created in each Chapter.
- Copy the corresponding CSWA Model Folder from the CD in the book that matches • your release of SolidWorks to your hard drive. Work directly from your hard drive on the tutorials in this book. SolidWorks Model files for 2009, 2010, and 2011 are provided.
- 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 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.

| 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 summary of mouse button terminology is displayed below:

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 is planning not to support XP Professional this year and is supporting Windows 7.

| | SolidWorks 2008 | SolidWorks 2009 | SolidWorks 2010 | SolidWorks 2011 |
|---------------------------------------|-----------------|-----------------|-----------------|-----------------|
| Windows 7 ×86 ⁽²⁾⁽¹⁰⁾ | No | No | Yes | Yes |
| Windows 7 x64 ⁽²⁾⁽¹⁰⁾ | No | No | Yes | Yes |
| Vista ×86 ⁽³⁾ | Yes | Yes | Yes | Yes |
| Vista ×64 ⁽³⁾⁽⁴⁾ | Yes | Yes | Yes | Yes |
| XP Professional x86 ⁽⁵⁾⁽¹⁾ | Yes | Yes | Yes | Yes |
| XP Professional x64 ⁽⁴⁾⁽¹⁾ | Yes | Yes | Yes | Yes |

The book does not cover installing SolidWorks 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.co m/sw/docs/EDU_2010_In stallation_Instructions.pdf

Computer and Software Requirements: RAM • Minimum: 1GB RAM • Recommended: 6GB RAM or more on Windows 7 x64 operating system • A certified graphics card and driver (recommended) Video See Graphics Cards Drivers for details CPU Intel® or AMD® processors⁽⁷⁾⁽¹¹⁾ • eDrawings® is supported on Apple Macintosh®-based machines⁽⁶⁾ Other DVD drive • Microsoft Internet Explorer version 6.x or higher • Adobe Acrobat version 7.0.7 or higher Microsoft Office Excel 2002⁽¹²⁾, 2003⁽¹²⁾, 2007, or 2010 Network • SolidWorks is only tested with Microsoft's Windows Networking and Active Directory network environments⁽⁸⁾

Copy the corresponding CSWA Model Folder from the CD in the book that matches your release of SolidWorks to your hard drive. Work directly from your hard drive on the tutorials in this book.



Introduction

Notes: