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Mold Design Using Solidworks Manual

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? SOLIDWORKS TUTORIAL || mold tools || Simple mold design using cavity feature.

SOLIDWORKS Tutorial \"Mold Design\" (10/10)

Solidworks Mold tools tutorial | Introduction of Mold tools in Solidworks

MOLD | How to Create CORE-CAVITY || SOLIDWORKS TUTORIALS SOLIDWORKS Mold Design Webinar Archives: Designing with SOLIDWORKS Mold Tools ? **SOLIDWORKS TUTORIAL || Mold tools || Design a mold for soap case. How to Create a Mold in SolidWorks 2010 2 Plate Injection Mold Designing in Solidworks (Mould Design Tutorials in Solidworks)** ? ~~SOLIDWORKS TUTORIAL || mold tools || Draft analysis for mold design.~~ Mold Tools in Solidworks Tutorial |

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Solidworks | Mold Tools | core cavity

Mold Design and Moldmaking - Actuators (excerpt) ~~SolidWorks~~ :

~~Freeform~~ Mold tools || Design a Mold Product in Solidworks

Solidworks Tutorials #34 How to Make a Pillow Design in

Solidworks BY SOLIDWORKS_EASY_DESIGN. SDNR Valve

Design | Complete Assembly | Part 1 - Body | Solidworks Tutorials

SolidWorks Mold Tools Tutorial #319 : plastic mobile cover mold

(mold tools course) Solidworks tutorial die design

Simple Solidworks Mold Making Tutorial

Solidworks Sheet Metal tutorial forming tool SolidWorks Surfaces

Tutorial Bottle Assembly ~~Solidworks tutorial | Mold Design in~~

~~Solidworks | Cavity and Core in Solidworks SOLIDWORKS Mold~~

~~Design | Solidworks Mold Tutorial~~ Solidworks Tutorial # Mold

Design ~~SOLIDWORKS Simulation - Mold Design in~~

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~~SOLIDWORKS 2015 Introduction to SolidWorks Mold Design
Solidworks Plastics Simulation Tutorial 1: Injection Molding Flow,
Sink Marks, Shrinkage, Weld Lines Solidworks tutorial | Mold
Design in Solidworks | Cavity and Core in Solidworks~~

Lunch \u0026 Learn - SOLIDWORKS Mold Tools Mold Design
Using Solidworks Manual

Mold Design Using SOLIDWORKS . Length: 2 day. To locate an Authorized Training Center near you, click [here](#). Prerequisites: Advanced Part Modeling. Description: Mold Design Using SOLIDWORKS teaches you several manual mold creation techniques and how to use the Mold Tools in SOLIDWORKS mechanical design automation software.

Mold Design Using SOLIDWORKS | Training Courses |

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SOLIDWORKS

Mold Design Using SOLIDWORKS Length: 2 days Prerequisite: Advanced Part Modeling Description: Mold Design Using SOLIDWORKS teaches you several manual mold creation techniques and how to use the Mold Tools in SOLIDWORKS mechanical design automation software. Topics covered in this course are: Introduction About This Course Using this Book

Mold Design Using SOLIDWORKS - CADimensions, Inc.
Mold Design You create a mold using a sequence of integrated tools that control the mold creation process. You can use these mold tools to analyze and correct deficiencies with either SOLIDWORKS or imported models of parts to be molded. Mold tools span from initial analysis to creating the tooling split.

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2020 SOLIDWORKS Help - Mold Design
SOLIDWORKS Mold Design Using SOLIDWORKS Dassault
Systèmes SolidWorks Corporation 175 Wyman Street Waltham,
MA 02451 U.S.A.

Mold Design Using SolidWorks - Hawk Ridge Systems
SOLIDWORKS Mold Design Using SOLIDWORKS Dassault
Systèmes SolidWorks Corporation 175 Wyman Street Waltham,
MA 02451 U.S.A.

Mold Design Using SolidWorks
Mold Design Using SOLIDWORKS teaches you several manual
mold creation techniques and how to use the Mold Tools in

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SOLIDWORKS mechanical design automation software. The main topics covered include:

Duration - Pentagon Solutions

BestMoldComponents.com are Solidworks 3D models of HASCO mold components. HASCO is one of many injection mold base suppliers. If you are ready for mold design using SolidWorks, just follow the instructions below. 01. Download. The HASCO mold base layout 1 consists of 22 files. 3 SolidWorks assemblies and 19 SolidWorks part files.

Tutorial mold design using SolidWorks mold base assembly
3DQuickMold is a professional plastic mold design software seamlessly integrated with Solidworks. This manual is based on the

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Solidworks 2011. Please do the following after the installation: 1. Click Tools -> Options->Add-Ins, make sure 3DQuickMold2011 is checked 2. 3DQuickMold advises the user to start project with new SolidWorks part. To do this, you

3DQuickMold Training Manual - 3D Tool Design Solutions
IMOLD for SOLIDWORKS is a powerful, completely SOLIDWORKS integrated CAMD tool for streamlining and automating the mold design process. Leveraging SOLIDWORKS, it captures the process specific knowledge of mold engineering to provide designers with expert, highly efficient design tools and step by step guide for the complete mold design process.

IMOLD for SOLIDWORKS - Manusoft

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SOLIDWORKS® product development solutions provide a full range of integrated modeling, simulation, and communication tools that mold, tool, and die designers and manufacturers need to create molds faster, at lower cost, and that are optimized for productivity and longevity. Handle Complex Mold Designs with Ease

Mold, Tool & Die Designers | SOLIDWORKS

DESCRIPTION: Mold Design Using SOLIDWORKS teaches you several manual mold creation techniques and how to use the Mold Tools in SOLIDWORKS mechanical design automation software. All the ways to take SOLIDWORKS mold design

SOLIDWORKS Mold Design Training Course | GoEngineer

Mold Design Using SOLIDWORKS teaches you several manual

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mold creation techniques and how to use the Mold Tools in SOLIDWORKS mechanical design automation software.

Prerequisites: Advanced Part Modeling. Length: 2 days

Mold Design Using SOLIDWORKS Course – SOLIDWORKS Malaysia ...

her we will learn about Cavity & core design in Solidworks with the help of Solidworks Mold & assembly tool, & we will design a simple Mold in Solidworks, Pl...

Solidworks tutorial | Mold Design in Solidworks | Cavity ...

Mold Design Using SOLIDWORKS teaches you several manual mold creation techniques and how to use the Mold Tools in SOLIDWORKS mechanical design automation software.

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Prerequisites: Advanced Part Modeling. Length: 2 days

Mold Design Using SOLIDWORKS Course - SOLIDWORKS Singapore ...

Mold Design Using SOLIDWORKS teaches you several manual creation techniques and how to use the Mold Tools in SOLIDWORKS mechanical design automation software.

Mold Design Using SOLIDWORKS - Online

Mold Design Using SOLIDWORKS Length 2 Days Cost \$800.00

Recommended Experience Level Advanced Part Modeling

Description When working with casting or injection molds, it's important to be able to utilize all of the tools available to engineers in SOLIDWORKS. With proper training, mold makers are able to

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learn best practices from the start.

Mold Design Using SOLIDWORKS | CADDimensions

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Use our HASCO standard mold components together with the features of SolidWorks mold tools and your mold design will be fast. Open the SolidWorks mold assembly Layout 1 and add your

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(part) cavity in SolidWorks and you are up and running. These mold components can be found here at BestMoldComponents.com and they are free to download.

The Complete Guide to Mold Making with SOLIDWORKS 2020 is a quick paced book written to provide experienced SOLIDWORKS users with in-depth knowledge of the mold tools provided by SOLIDWORKS. Throughout this book you will learn the procedures necessary for using these tools to create and analyze effective mold designs. Utilizing step-by-step instructions, each chapter of this book will guide you through different tasks, from designing or repairing a mold, to developing complex parting lines;

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from making a core in the part mode to advancing through more complex tasks in the assembly mode. Throughout this book you will be introduced to using surfacing tools to repair models and prepare them for the mold making process. Towards the end of this book, you will learn how to work with SOLIDWORKS Plastics and Flow Simulation to simulate the way melted plastics flow during the injection molding process. You will also learn to analyze the thick-thin wall regions to predict defects on plastic parts and molds. Learning how to analyze plastic parts for errors and correct them early in the design stage is a valuable skill, which can save a significant amount of time throughout the span of the entire design process. Every project in this book is based on real world products. Each of these projects have been broken down and developed into simple, comprehensible steps. Furthermore, every mold design is

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explained very clearly in short chapters, ranging from 15 to 25 pages. Each step comes with the exact screen shot to help you understand the main concept of the design. Learn the mold designs at your own pace, as you progress from simple core and cavity creation to more complex mold design challenges. This book will also teach you to use various surfacing tools such as: Ruled Surface Planar Surface Knit Surface Filled Surface Extend Surface Trim Surface Lofted Surface Who This Book Is For This book is for users already familiar with SOLIDWORKS who want to expand their knowledge of mold design. To get the most out of this mold design book, it is strongly recommended that you have completed all the lessons in the SOLIDWORKS Advanced Techniques book or have comparable knowledge. More CAD literate individuals, who want to expand their knowledge of the different features that

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SOLIDWORKS 2020 has to offer, will also find this book to be a great resource.

The Complete Guide to Mold Making with SOLIDWORKS 2021 is a quick paced book written to provide experienced SOLIDWORKS users with in-depth knowledge of the mold tools provided by SOLIDWORKS. Throughout this book you will learn the procedures necessary for using these tools to create and analyze effective mold designs. Utilizing step-by-step instructions, each chapter of this book will guide you through different tasks, from designing or repairing a mold, to developing complex parting lines; from making a core in the part mode to advancing through more complex tasks in the assembly mode. Throughout this book you will be introduced to using surfacing tools to repair models and prepare

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them for the mold making process. Towards the end of this book, you will learn how to work with SOLIDWORKS Plastics and Flow Simulation to simulate the way melted plastics flow during the injection molding process. You will also learn to analyze the thick-thin wall regions to predict defects on plastic parts and molds. Learning how to analyze plastic parts for errors and correct them early in the design stage is a valuable skill, which can save a significant amount of time throughout the span of the entire design process. Every project in this book is based on real world products. Each of these projects have been broken down and developed into simple, comprehensible steps. Furthermore, every mold design is explained very clearly in short chapters, ranging from 15 to 25 pages. Each step comes with the exact screen shot to help you understand the main concept of the design. Learn the mold designs

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at your own pace, as you progress from simple core and cavity creation to more complex mold design challenges. This book will also teach you to use various surfacing tools such as: • Ruled Surface • Planar Surface • Knit Surface • Filled Surface • Extend Surface • Trim Surface • Lofted Surface

Who This Book Is For

This book is for users already familiar with SOLIDWORKS who want to expand their knowledge of mold design. To get the most out of this mold design book, it is strongly recommended that you have completed all the lessons in the SOLIDWORKS Advanced Techniques book or have comparable knowledge. More CAD literate individuals, who want to expand their knowledge of the different features that SOLIDWORKS 2021 has to offer, will also find this book to be a great resource.

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The only continuous, step-by-step tutorial for SolidWorks
SolidWorks is a 3D CAD manufacturing software package that has been used to design everything from aerospace robotics to bicycles. This book teaches beginners to use SolidWorks through a step-by-step tutorial, letting you build, document, and present a project while you learn. Tools and functionality are explained in the context of professional, real-world tasks and workflows. You will learn the essential functions and gain the skills to use the software at once. SolidWorks is a popular design software for manufacturing, and this book introduces it in the context of actually creating an object
Begins with an overview of SolidWorks conventions and the interface Explains how to create models and drawings, create a revolved part and subassembly, and model parts within a subassembly Explores modification capabilities and drawing and

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Bill of Materials templates Moves on to top-level assembly models and drawings, Toolbox components and the Design Library, mates, export and printing capabilities, and creating renderings Includes a glossary, a foreword from the SolidWorks product manager, and downloadable tutorial files SolidWorks 2010: No Experience Required quickly turns beginners into confident users of SolidWorks.

This senior undergraduate level textbook is written for Advanced Manufacturing, Additive Manufacturing, as well as CAD/CAM courses. Its goal is to assist students in colleges and universities, designers, engineers, and professionals interested in using SolidWorks as the design and 3D printing tool for emerging manufacturing technology for practical applications. This textbook

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will bring a new dimension to SolidWorks by introducing readers to the role of SolidWorks in the relatively new manufacturing paradigm shift, known as 3D-Printing which is based on Additive Manufacturing (AM) technology. This new textbook: Features modeling of complex parts and surfaces Provides a step-by-step tutorial type approach with pictures showing how to model using SolidWorks Offers a user-Friendly approach for the design of parts, assemblies, and drawings, motion-analysis, and FEA topics Includes clarification of connections between SolidWorks and 3D-Printing based on Additive Manufacturing Discusses a clear presentation of Additive Manufacturing for Designers using SolidWorks CAD software "Introduction to SolidWorks: A Comprehensive Guide with Applications in 3D Printing" is written using a hands-on approach which includes a significant number of

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pictorial descriptions of the steps that a student should follow to model parts, assemble parts, and produce drawings.

Mold Design Using NX 11.0: A Tutorial Approach book is written with the intention of helping the readers effectively design molds and its parts such as gate, runner, and various other standard parts using Mold Wizard of NX. After going through this book, the users will be able to design molds easily and effectively through processes such as analysis and documentation which have been dealt in detail. Also, the chapters in this book are arranged in a pedagogical sequence that makes this book very effective in learning the features and capabilities of the software. Keeping in mind the requirements of the users, the book at first introduces basic terms and analyses and gradually progresses to cover sequential

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method to create mold and documentation. Written with the tutorial point of view and the learn by doing a theme, the book caters to the needs of both novice and advanced users and is ideally suited for learning at your convenience and pace. Salient Features Consists of 10 chapters that are organized in a pedagogical sequence. Cover mold design concepts using NX 11.0. Tutorial approach to explain the concepts of Mold Design using NX 11.0. Summarized content on the first page of the topics that are covered in the chapter.

Hundreds of illustrations for easy understanding of concepts. Step-by-step instructions to guide the users through the learning process. Additional information throughout the book in the form of notes and tips. Self-Evaluation Tests and Review Questions at the end of each chapter to help the users assess their knowledge. Technical support by contacting 'techsupport@cadcim.com' Additional

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learning resources at 'allaboutcadcam.blogspot.com' Table of Contents Chapter 1: Introduction to Mold Design and NX Mold Wizard Chapter 2: Part Analysis Chapter 3: Creating Parting Surface Chapter 4: Creating Core and Cavity Chapter 5: Adding Mold Base and Standard Parts Chapter 6: Creating Gate, Runner, and Layout Chapter 7: Creating Sliders and Lifters Chapter 8: Creating Ejection and Cooling Systems Chapter 9: Creating Electrodes Chapter 10: Documentation Index

Computer-Aided Engineering Design with SolidWorks is designed for students taking SolidWorks courses at college and university, and also for engineering designers involved or interested in using SolidWorks for real-life applications in manufacturing processes, mechanical systems, and engineering analysis. The course material

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is divided into two parts. Part I covers the principles of SolidWorks, simple and advanced part modeling approaches, assembly modeling, drawing, configurations/design tables, and surface modeling. Part II covers the applications of SolidWorks in manufacturing processes, mechanical systems, and engineering analysis. The manufacturing processes applications include mold design, sheet metal parts design, die design, and weldments. The mechanical systems applications include: routing, piping and tubing, gears, pulleys and chains, cams and springs, mechanism design and analysis, threads and fasteners, hinges, and universal joints. The sections on engineering analysis also include finite element analysis. This textbook is unique because it is one of the very few to thoroughly cover the applications of SolidWorks in manufacturing processes, mechanical systems, and engineering

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analysis, as presented in Part II. It is written using a hands-on approach in which students can follow the steps described in each chapter to: model and assemble parts, produce drawings, and create applications on their own with little assistance from their instructors during each teaching session or in the computer laboratory. There are pictorial descriptions of the steps involved in every stage of part modeling, assembly modeling, drawing details, and applications presented in this textbook. Supplementary Material(s) For Users (2 MB)

This senior undergraduate level textbook is written for Advanced Manufacturing, Additive Manufacturing, as well as CAD/CAM courses. Its goal is to assist students in colleges and universities, designers, engineers, and professionals interested in using

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SolidWorks as the design and 3D printing tool for emerging manufacturing technology for practical applications. This textbook will bring a new dimension to SolidWorks by introducing readers to the role of SolidWorks in the relatively new manufacturing paradigm shift, known as 3D-Printing which is based on Additive Manufacturing (AM) technology. This new textbook: Features modeling of complex parts and surfaces Provides a step-by-step tutorial type approach with pictures showing how to model using SolidWorks Offers a user-Friendly approach for the design of parts, assemblies, and drawings, motion-analysis, and FEA topics Includes clarification of connections between SolidWorks and 3D-Printing based on Additive Manufacturing Discusses a clear presentation of Additive Manufacturing for Designers using SolidWorks CAD software "Introduction to SolidWorks: A

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Comprehensive Guide with Applications in 3D Printing" is written using a hands-on approach which includes a significant number of pictorial descriptions of the steps that a student should follow to model parts, assemble parts, and produce drawings.

Engineering Design with SolidWorks 2014 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. Explore the user interface, CommandManager, menus, toolbars and modeling techniques to create parts, assemblies and drawings in an engineering environment. Follow the step-by-step instructions and develop multiple parts and assemblies that combine machined, plastic and

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sheet metal components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, design tables, Bills of Materials, Custom Properties and Configurations. Address various SolidWorks analysis tools: SimulationXpress, Sustainability/SustainabilityXpress and DFMXpress and Intelligent Modeling techniques. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for each project. Know your objective up front. Follow the steps in Project 1 - 8 to achieve the design goals. Work between multiple documents, features, commands and custom properties that represent how engineers and designers utilize SolidWorks in industry. Review individual features, commands and tools with the Video Instruction. The projects contain exercises. The exercises

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analyze and examine usage competencies. Collaborate with leading industry suppliers such as SMC Corporation of America, Boston Gear and 80/20 Inc. Collaborative information translates into numerous formats such as paper drawings, electronic files, rendered images and animations. On-line intelligent catalogs guide designers to the product that meets both their geometric requirements and performance functionality. 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. The book is design to compliment the SolidWorks Tutorials contained in SolidWorks 2014.

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Engineering Design with SolidWorks 2013 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. Explore the user interface, CommandManager, menus, toolbars and modeling techniques to create parts, assemblies and drawings in an engineering environment. Follow the step-by-step instructions and develop multiple parts and assemblies that combine machined, plastic and sheet metal components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, design tables, Bills of Materials, Custom Properties and Configurations. Address various SolidWorks analysis tools:

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SimulationXpress, Sustainability / SustainabilityXpress and DFMXpress and Intelligent Modeling techniques. Learn by doing, not just by reading! Desired outcomes and usage competencies are listed for each project. Know your objective up front. Follow the steps in Project 1 - 8 to achieve the design goals. Work between multiple documents, features, commands and custom properties that represent how engineers and designers utilize SolidWorks in industry. Review individual features, commands and tools with the enclosed Video Instruction DVD. The projects contain exercises. The exercises analyze and examine usage competencies. Collaborate with leading industry suppliers such as SMC Corporation of America, Boston Gear and 80/20 Inc. Collaborative information translates into numerous formats such as paper drawings, electronic files, rendered images and animations. On-line

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intelligent catalogs guide designers to the product that meets both their geometric requirements and performance functionality. 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 every day. Their responsibilities go far beyond the creation of just a 3D model. The book is design to compliment the SolidWorks Tutorials contained in SolidWorks 2013. There are over 2.5 hours of video instructions on the enclosed DVD.

- Uses step-by-step, project based tutorials designed for beginning or intermediate users
- Will prepare you for the Certified SOLIDWORKS Associate Exam
- Includes a chapter introducing

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you to 3D printing SOLIDWORKS 2020 Tutorial is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text provides a step-by-step, project based learning approach. It also contains information and examples on the five categories in the CSWA exam. The book is divided into four sections. Chapters 1 - 5 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter 6 you will create the final robot assembly. The physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components for the final robot assembly are

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provided. Chapters 7 - 10 prepare you for the Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms and technology used in low cost 3D printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, apply proper design intent, design tables and configurations. Learn by doing, not just by reading. Desired outcomes and usage competencies are listed for

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each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SOLIDWORKS in industry.

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