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UG79F3 - JOVANI HEIDI

SOLIDWORKS 2021 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 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 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.

- Picks up where SOLIDWORKS Basic Tools leaves off
- Uses a step by step tutorial approach with real world projects
- Comprehensive coverage of intermediate SOLIDWORKS tools and techniques
- Expands on Solids, Surfaces, Multibodies, Configurations, Drawings, Sheet Metal and Assemblies
- Features a quick reference guide

This edition includes a new chapter on plastic parts SOLIDWORKS 2020 Intermediate Skills is part of a three part series which builds on the SOLIDWORKS features learned in SOLIDWORKS 2020 Basic Tools. SOLIDWORKS 2020 Intermediate Skills broadens your SOLIDWORKS knowledge base by covering such features as surveys, lofts and boundaries, the use of multibodies, generating engineering drawings and other SOLIDWORKS functions that are critical for the effective use of this powerful software. This book helps prepare you for the advanced features of SOLIDWORKS which are covered in SOLIDWORKS Advanced Techniques. It uses a step by step tutorial approach with real world projects. This book also features a Quick-Reference-Guide to the SOLIDWORKS 2020 commands, icons, and customized hotkeys. Who's this book for? This book is for the mid-level user, who is already familiar with the SOLIDWORKS program. It is also a great resource for the more CAD literate individuals who want to expand their knowledge of the different features that SOLIDWORKS 2020 has to offer.

Engineering Design with SOLIDWORKS 2018 and video instruction is written to assist students, designers, engineers and professionals. The book provides a solid foundation in SOLIDWORKS by utilizing projects with step-by-step instructions for the beginner to intermediate SOLIDWORKS user featuring machined, plastic and sheet metal components. Desired outcomes and usage competencies are listed for each project. The book is divided into five sections with 11 projects. Project 1 - Project 6: Explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables

using basic and advanced features. Additional techniques include the edit and reuse of features, parts, and assemblies through symmetry, patterns, configurations, SOLIDWORKS 3D ContentCentral and the SOLIDWORKS Toolbox. Project 7: Understand Top-Down assembly modeling and Sheet Metal parts. Develop components In-Context with InPlace Mates, along with the ability to import parts using the Top-Down assembly method. Convert a solid part into a Sheet Metal part and insert and apply various Sheet Metal features. Project 8 - Project 9: Recognize SOLIDWORKS Simulation and Intelligent Modeling techniques. Understand a general overview of SOLIDWORKS Simulation and the type of questions that are on the SOLIDWORKS Simulation Associate - Finite Element Analysis (CSWSA-FEA) exam. Apply design intent and intelligent modeling techniques in a sketch, feature, part, plane, assembly and drawing. Project 10: Comprehend the differences between additive and subtractive manufacturing. Understand 3D printer terminology along with a working knowledge of preparing, saving, and printing CAD models on a low cost printer. Project 11: Review the Certified Associate - Mechanical Design (CSWA) program. Understand the curriculum and categories of the CSWA exam and the required model knowledge needed to successfully take the exam. The author developed the industry scenarios by combining his own industry experience with the knowledge of engineers, department managers, vendors and manufacturers. These professionals are directly involved with SOLIDWORKS every day. Their responsibilities go far beyond the creation of just a 3D model.

Most CAD professionals today recognize the need to become certified to prove their skills, prepare for new job searches, and to learn new skills while at their existing job. Specifying a Certified SolidWorks Associate (CSWA) certification on your resume is a great way to increase your chances of landing a new job, getting a promotion, or looking more qualified when representing your company on a consulting job. The primary goal of this book 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. 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.

- 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 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 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 cov-

ers 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 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.

The Commands Guide Tutorial for SolidWorks 2013 is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2013. SolidWorks is an immense software package, and no one book can cover all topics for all users. This book provides a centralized reference location to address many of the tools, features and techniques of SolidWorks 2013. This book covers the following: System and Document properties FeatureManagers PropertyManagers ConfigurationManagers RenderManagers 2D and 3D Sketch tools Sketch entities 3D Feature tools Motion Study Sheet Metal Motion Study Sustainability Sustainability Xpress FlowXpress PhotoView 360 Pack and Go Intelligent Modeling techniques and more. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2013 software. If you are completely new to SolidWorks, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2 and Lesson 3 in the SolidWorks Tutorials. If you are familiar with an earlier release of SolidWorks, you still might want to skim Chapter 1 to become acquainted with some of the commands, menus and features that you have not used; or you can simply jump to any section in any chapter. Each chapter (18 total) provides detailed PropertyManager information on key topics with individual stand alone short tutorials to reinforce and demonstrate the functionality and ease of the SolidWorks tool or feature. All models for the 240 plus tutorials are located on the enclosed book CD with their solution (initial and final). Learn by doing, not just by reading! Formulate the skills to create, modify and edit sketch-

es and solid features. Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, design tables, configurations and more. The book is design to complement the Online Tutorials and Online Help contained in SolidWorks 2013. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The authors developed the tutorials by combining their own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. These professionals are directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of just a 3D model. Analysis of Machine Elements Using SolidWorks Simulation 2014 is written primarily for first-time SolidWorks Simulation 2014 users who wish to understand finite element analysis capabilities applicable to stress analysis of mechanical elements. The focus of examples is on problems commonly found in an introductory, undergraduate, Design of Machine Elements or similarly named courses. In order to be compatible with most machine design textbooks, this text begins with problems that can be solved with a basic understanding of mechanics of materials. Problem types quickly migrate to include states of stress found in more specialized situations common to a design of mechanical elements course. Paralleling this progression of problem types, each chapter introduces new software concepts and capabilities. Many examples are accompanied by problem solutions based on use of classical equations for stress determination. Unlike many step-by-step user guides that only list a succession of steps, which if followed correctly lead to successful solution of a problem, this text attempts to provide insight into why each step is performed. This approach amplifies two fundamental tenets of this text. The first is that a better understanding of course topics related to stress determination is realized when classical methods and finite element solutions are considered together. The second tenet is that finite element solutions should always be verified by checking, whether by classical stress equations or experimentation. Each chapter begins with a list of learning objectives related to specific capabilities of the SolidWorks Simulation program introduced in that chapter. Most software capabilities are repeated in subsequent examples so that users gain familiarity with their purpose and are capable of using them in future problems. All end-of-chapter problems are accompanied by evaluation "check sheets" to facilitate grading assignments.

Explore a practical and example-driven approach to understanding SOLIDWORKS 2020 and achieving CSWA and CSWP certification. Gain comprehensive insights into the core aspects of mechanical part modeling. Get up to speed with generating assembly designs with both standard and advanced mates. Focus on design practices for both 2D as well as 3D modeling and prepare to achieve CWSP and CWSA certification. **Book Description** SOLIDWORKS is the leading choice for 3D engineering and product design applications across industries such as aviation, automobiles, and consumer product design. This book takes a practical approach to getting you up and running with SOLIDWORKS 2020. You'll start with the basics, exploring the software interface and working with drawing files. The book then guides you through topics such as sketching, building complex 3D models, generating dynamic and static assemblies, and generating 2D engineering drawings to equip you for mechanical design projects. You'll also do practical exercises to get hands-on with creating sketches, 3D part models, assemblies, and drawings. To reinforce your understanding of SOLIDWORKS, the book is supplemented by downloadable files that will help you follow up with the concepts and exercises found in the book. By the end of this book, you'll have gained the skills you need to create professional 3D mechanical models using SOLIDWORKS, and you'll be able to prepare effectively for the Certified SOLIDWORKS Associate (CSWA) and Certified SOLIDWORKS Professional (CSWP) exams. What you will learn: Understand the fundamentals of SOLIDWORKS and parametric modeling. Create professional 2D sketches as bases for 3D models using simple and advanced modeling techniques. Use SOLIDWORKS drawing tools to generate standard engineering drawings. Evaluate mass properties and materials for designing parts and assemblies. Understand the objectives and the formats of the CSWA and CSWP exams. Discover expert tips and tricks to generate different part and assembly configurations for your mechanical designs. Who this book is for: This book is for aspiring engineers, designers, drafting technicians, or anyone looking to get started with the latest version of SOLIDWORKS. Anyone interested in becoming a Certified SOLIDWORKS Associate (CSWA) or Certified SOLIDWORKS Professional (CSWP) will also find this book useful.

Drawing and Detailing with SolidWorks 2014 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 targeted towards the beginning to intermediate SolidWorks user. Work through numerous activities to create multiple-view, multi-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. This book is written to assist you with passing the SOLIDWORKS associate level exams. It provides you with detailed information and exercises that will aid you in passing the following exams: Certified SOLIDWORKS Associate (CSWA), Certified SOLIDWORKS Associate Sustainable Design (CSWA-SD), Certified SOLIDWORKS Simulation Associate Finite Element Analysis (CSWSA-FEA) and the Certified SOLIDWORKS Associate Additive Manufacturing (CSWA-AM) exam. There are three goals for this book. The primary goal of this book is not only to help you pass the CSWA, CSWA-SD, CSWSA-FEA and CSWA-AM exams, but also to ensure that you understand and comprehend the concepts and implementation details of the four certification processes. The second goal is to provide the most comprehensive coverage of CSWA, CSWA-SD, CSWSA-FEA and CSWA-AM 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, CSWA-SD, CSWSA-FEA and CSWA-AM exams.

SOLIDWORKS 2019 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 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

responding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and components for the final robot assembly are 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 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.

SOLIDWORKS 2018 Tutorial with video instruction 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, to take and understand the Certified Associate - Mechanical Design (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 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 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.

This book, along with the companion disc that accompanies it, will provide you with a wealth of information about the three segments of the CSWP CORE exam. The intended audience for this book is a person who has passed the CSWA exam and who has eight or more months of SolidWorks training and usage. This guide is not intended to teach you how to use SolidWorks, but is written to provide you with CSWP exam tips, hints and information on sample questions and categories that are aligned with the exam. This guide is written to help you take and pass the CSWP exam. This book comes with a companion disc containing segment videos for you to follow while you use the book. Each segment video provides valuable information, tips and tricks to successfully pass the CSWP CORE exam. SolidWorks model files, in both their initial and final state, are provided on this disc for SolidWorks 2012, 2013 and 2014. The book is organized into three chapters. Each chapter is focused on a segment of the CSWP CORE exam. This is not intended to be a step-by-step book. The Official Guide to Certified SOLIDWORKS Associate Exams: CSWA, CSDA, CSWSA-FEA is written to assist the SOLIDWORKS user to pass the associate level exams. Information is provided to aid a person to pass the Certified SOLIDWORKS Associate (CSWA), Certified SOLIDWORKS Sustainable Design Associate (CSDA) and the Certified SOLIDWORKS Simulation Associate Finite Element Analysis (CSWSA FEA) exam. There are three goals for this book. The primary goal is not only to help you pass the CSWA, CSDA and CSWSA-FEA exams, but also to ensure that you understand and comprehend the concepts and im-

plementation details of the three certification processes. The second goal is to provide the most comprehensive coverage of CSWA, CSDA and CSWSA-FEA 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, CSDA and the CSWSA-FEA exam. The Certified SOLIDWORKS Associate (CSWA) certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. Passing this exam provides students the chance to prove their knowledge and expertise and to be part of a worldwide industry certification standard. The Certified SOLIDWORKS Sustainable Design Associate (CSDA) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of environmental assessment and sustainable design. The Certified SOLIDWORKS Simulation Associate - Finite Element Analysis (CSWSA-FEA) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of stress analysis and the Finite Element Method (FEM).

This book, CSWP exam preparation: All-in-one guidebook with practice exams, is written to assist students in college and universities, engineers, and professionals who are interested in getting a certification of CSWP (Certified SOLIDWORKS Professional). There are so many books available for learning SOLIDWORKS, while there are very few books to help for CSWP exam preparation. To prepare for exams efficiently, future exam takers essentially need exam-oriented preparation books. There are some online/offline training programs. However, those are very expensive to prepare a CSWP exam. This book is divided into four parts. Part I covers introductory principles of CSWP, including fundamental knowledge of the CSWP exam. Part II covers the first segment of the CSWP exam mainly focusing on part modeling, including six practice exam sets. Also, key skills for passing the segment one exam are explained with various hands-on examples. Part III covers the second segment of the CSWP exam focusing on the configuration and design intent. Part IV covers the third segment focusing on the assemblies and mates. Organizing the book in this way is to help the students, engineers, and professionals to prepare the CSWP exam efficiently. This book has a significant number of pictorial descriptions of the steps with screenshots that a student should follow. Since 2018, the format of the CSWP exam has been updated. However, the problem

pattern covered by the exam and the required SOLIDWORKS skill does not change significantly. To help adapt to this new format, two sets of updated tests form are included in each part. The first practice test set for each part is provided with a walk-through instruction that includes a step-by-step problem-solving process, and the rest of the practice test sets are provided with the answers in the end-of-section with video tutorials. Time management is essential in CSWP, so it is recommended that you practice by completing the test within the specified time. It is hoped that students and engineers will find this edition useful for passing the CSWP exam in the first attempt.

Get to grips with leading 3D engineering and product design application to design robust 3D models and achieve CSWA and CSWP certification Key Features Gain comprehensive insights into the core aspects of 3D modeling's mechanical parts Learn how to generate assembly designs with both standard and advanced mates Discover design practices for both 2D as well as 3D modeling and prepare to achieve CSWP and CSWA certification Book Description SOLIDWORKS is the leading choice for 3D engineering and product design applications across industries such as aviation, automobile, and consumer product design. This book helps you to get up and running with SOLIDWORKS and understand each new concept and tool with the help of easy-to-follow exercises. You'll begin with the basics, exploring the software interface and finding out how to work with drawing files. The book then guides you through topics such as sketching, building complex 3D models, generating dynamic and static assemblies, and generating 2D engineering drawings to prepare you to take on any design project. You'll also work with practical exercises to get hands-on experience with creating sketches, 3D part models, assemblies, and drawings. To reinforce your understanding of SOLIDWORKS, the book is supplemented by downloadable files that will help you to understand the concepts and exercises more easily. Finally, you'll also work on projects for 3D modeling objects inspired by everyday life. By the end of this SOLIDWORKS book, you'll have gained the skills you need to create professional 3D mechanical models using SOLIDWORKS and be able to prepare effectively for the Certified SOLIDWORKS Associate (CSWA) and Certified SOLIDWORKS Professional (CSWP) exams. What you will learn Understand the fundamentals of SOLIDWORKS and parametric modeling Create professional 2D sketches as bases for 3D models using simple and advanced modeling techniques Use SOLID-

WORKS drawing tools to generate standard engineering drawings Evaluate mass properties and materials for designing parts and assemblies Join different parts together to form static and dynamic assemblies Discover expert tips and tricks to generate different part and assembly configurations for your mechanical designs Who this book is for This book is for aspiring engineers, designers, makers, draftsmen, and hobbyists looking to get started with SOLIDWORKS and explore the software. Individuals who are interested in becoming Certified SOLIDWORKS Associates (CSWAs) or Certified SOLIDWORKS Professionals (CSWPs) will also find this book useful. No specific background is needed to follow the concepts in the book as it starts from the basics of SOLIDWORKS. However, basic theoretical knowledge of 3D modeling will be helpful to get the most out of this book.

Engineering Design with SolidWorks 2011 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 beginning 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: 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 Multi-media CD. 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 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 everyday. Their responsibilities go far beyond the creation of just a 3D model. The book is designed to complement the SolidWorks Tutorials contained in SolidWorks 2011.

Assembly Modeling with SolidWorks 2012 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 apply 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 be-

gins 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 complements 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.

SOLIDWORKS 2017 Tutorial with video instruction 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, to take and understand the Certified Associate - Mechanical Design (CSWA) exam. The book is divided into three sections. Chapters 1 - 6 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple machine parts, simple and complex assemblies, proper design intent, design tables, configurations, equations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. 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. View Chapter 11 on Additive Manufacturing (3D printing) and its benefits and features. Understand 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 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. Become a Certified SOLIDWORKS Professional (CSWP). Study for CSWP certification by refreshing your SOLIDWORKS skills and practicing sample questions from the 2018 exam.

This book is written to assist you with passing the SOLIDWORKS associate level exams. It provides you with detailed information and exercises that will aid you in passing the following exams: Certified SOLIDWORKS Associate (CSWA), Certified SOLIDWORKS Associate Sustainable Design (CSWA-SD), Certified SOLIDWORKS Associate Simulation (CSWSA-S) and the Certified SOLIDWORKS Associate Additive Manufacturing (CSWA-AM) exam. There are three goals for this book. The primary goal of this book is not only to help you pass the CSWA, CSWA-SD, CSWSA-S and CSWA-AM exams, but also to ensure that you understand and comprehend the concepts and implementation details of the four certification processes. The second goal is to provide the most comprehensive coverage of CSWA, CSWA-SD, CSWSA-S and CSWA-AM 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, CSWA-SD, CSWSA-S and CSWA-AM exams. CSWA Exam The CSWA certification indicates a foundation in and apprentice knowledge of 3D CAD design and engineering practices and principles. The intended audience for this section of the book is anyone trying to take and pass the CSWA exam 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 and 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. CSWA-SD Exam The Certified SOLIDWORKS Associate Sustainable Design (CSWA-SD) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of environmental assessment and sustainable design. This section of the book is intended for anyone interested in Sustainable design as well as life cycle assessment and trying to take and pass the CSWA-SD exam. Although no hands-on usage of SOLIDWORKS is required for the CSWA-SD certification exam, it is a good idea to review the SOLIDWORKS SustainabilityXpress and SOLIDWORKS Sustainability tutorials inside of SOLIDWORKS to better understand the actual workflow. The CSWA-SD is based off the SOLIDWORKS Sustainable Design Guide that incorporates concepts including sustainability, environmental assessment and life cycle impact assessment. CSWSA-S Exam The Certified SOLIDWORKS Associate Simulation (CSWSA-S) certification indicates a foundation in and apprentice knowledge of demonstrating an understanding in the principles of stress analysis and the Finite Element Method (FEM). The CSWSA-S section of the book is for anyone trying to take and pass the CSWSA-S with a minimum of 6 - 9 months of SOLIDWORKS experience and knowledge in the following areas: Engineering Mechanics - Statics, Strength of Materials, Finite Element Method/Finite Element Analysis Theory, Applied concepts in SOLIDWORKS Simulation: namely Static Analysis, Solid, Shell, and Beam elements, Connections and Applying loads and boundary conditions and interpreting results. The purpose of this section in the book is NOT to educate a new or intermediate user on SOLIDWORKS Simulation, but to cover and to inform you on the types of questions, layout and what to expect when taking the CSWSA-S exam. CSWA-AM Exam The Certified SOLIDWORKS Associate Additive Manufacturing (CSWA-AM) certification indicates a foundation in and apprentice knowledge of today's 3D printing technology and market. The intended audience for this section of the book is anyone trying to take and pass the CSWA-AM exam and an interest in Additive Manufacturing. The CSWA-AM exam is meant to be taken after the completion of the 10-part learning path located on MySOLIDWORKS.com. The CSWA-AM exam fundamentally covers two 3D printing technologies: Fused Filament Fabrication (FFF) and STereoLithography (SLA). There are a few questions on Selective Laser Sintering (SLS) technology and available software-based printing aids.

Engineering Graphics with SOLIDWORKS 2018 and Video Instruction is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The book is divided into four sections: Chapters 1 - 3 explore the history of engineering graphics, manual sketching techniques, orthographic projection, Third vs. First angle projection, multi-view drawings, dimensioning practices (ASME Y14.5-2009 standard), line type, fit type, tolerance, fasteners in general, general thread notes and the history of CAD leading to the development of SOLIDWORKS. Chapters 4 - 9 explore the SOLIDWORKS User Interface and CommandManager, Document and System properties, simple machine parts, simple and complex assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. Follow the step-by-step instructions in over 80 activities to develop eight parts, four sub-assemblies, three drawings and six document templates. Chapter 10 provides a section on the Certified Associate - Mechanical Design (CSWA) program with sample exam questions and initial and final SOLIDWORKS models. Chapter 11 helps you understand the differences between additive and subtractive manufacturing. Comprehend 3D printer terminology along with a working knowledge of preparing, saving, and printing a 3D CAD model on a low cost printer. Review individual features, commands, and tools using the video instruction and SOLIDWORKS Help. The chapter exercises analyze and examine usage competencies based on the chapter objectives. The book is designed to complement the SOLIDWORKS Tutorials located in the SOLIDWORKS Help menu. Desired outcomes and usage competencies are listed for each project. Know your objectives up front. Follow the step-by step procedures to achieve your design goals. Work between multiple documents, features, commands, and properties that represent how engineers and designers utilize SOLIDWORKS in industry. 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. SOLIDWORKS 2019 Quick Start introduces the new user to the basics of using SOLIDWORKS 3D CAD software in five easy lessons. This book is intended for the student or designer that needs to learn SOLIDWORKS quickly and effectively for senior capstone, machine design, kinematics, dy-

namics, and other engineering and technology projects that use SOLIDWORKS as a tool. Engineers in industry are expected to have SOLIDWORKS skills for their company's next project. Students need to learn SOLIDWORKS without taking a formal CAD course. Based on years of teaching SOLIDWORKS to engineering students, SOLIDWORKS 2019 Quick Start concentrates on the areas where the new user improves efficiency in the design modeling process. By learning the correct SOLIDWORKS skills and file management techniques, you gain the most knowledge in the shortest period of time. You develop a mini Stirling Engine and investigate the proper design intent and constraints. The mini Stirling Engine is based on the external combustion, closed cycle engine of Scottish inventor Robert Stirling. In addition to 3D modeling, the engine can be used to teach and connect many engineering and physics principles. You begin with an overview of SOLIDWORKS and the User Interface (UI), its menus, toolbars and commands. With a quick pace, you learn the essentials of 2D sketching, part and assembly creation, perform motion study, develop detailed part and assembly drawings and much more.

This book, along with the companion disc that accompanies it, will provide you with a wealth of information about the three segments of the CSWP CORE exam. The intended audience for this book is a person who has passed the CSWA exam and who has eight or more months of SolidWorks training and usage. This guide is not intended to teach you how to use SolidWorks, but is written to provide you with CSWP exam tips, hints and information on sample questions and categories that are aligned with the exam. This guide is written to help you take and pass the CSWP exam. This book comes with a companion DVD containing segment videos for you to follow while you use the book. Each segment video provides valuable information, tips and tricks to successfully pass the CSWP CORE exam. SolidWorks model files, in both their initial and final state, are provided on this DVD for both SolidWorks 2012 and 2013. The book is organized into three chapters. Each chapter is focused on a segment of the CSWP CORE exam. This is not intended to be a step-by-step book.

SolidWorks 2014 Tutorial with video instruction is targeted towards a technical school, two year college, four year university or industry professional that is a beginner or intermediate CAD user. The text provides a student who is looking for a step-by-step project based approach to learning SolidWorks with video instruction, Solid-

Works model files, and preparation for the Certified Associate - Mechanical Design (CSWA) exam. The book is divided into two sections. Chapters 1 - 5 explore the SolidWorks User Interface and CommandManager, Document and System properties, simple machine parts, simple and complex assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, Revision tables using basic and advanced features. Chapters 6 - 9 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. 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 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.

The CSWPA is a set of exams designed to demonstrate your advanced abilities in five distinct areas of SOLIDWORKS. By passing the CSWPA exams you prove to potential employers that you have an advanced skill set within SOLIDWORKS, and you become more desirable in the job market. Certified SOLIDWORKS Professional Advanced Preparation Material is intended for the SOLIDWORKS user who has already passed the CSWP exam, and is ready to advance to the next level. This book covers the five CSWPA examinations: Sheet Metal, Weldments, Surfacing, Mold Tools, and Drawing Tools. The lessons in this book were created based on the actual CSWPA examinations. Each of these projects has been broken down and developed into easy and comprehensible steps for the reader. Every challenge is explained very clearly in short chapters, ranging from 30 to 50 pages. Each step comes with a screen shot to help you understand the main concept of each design more easily. Learn the CSWP Advanced Preparation Materials at your own pace, as you progress from Parts, Assemblies, Drawings and then to more complex design challenges. To get the most out of this CSWPA-Certification Preparation book it is strongly recommend-

ed that you have studied and completed all the lessons in the Basic Tools, Intermediate Skills and Advanced Techniques books. It is also a great resource for the more CAD literate individuals who want to expand their knowledge of the different features that SOLIDWORKS 2019 has to offer. This book is a great resource to prepare for and pass the CSWPA exams which will prove your expertise and further your career. After completing at least four of the five CSWPA exams you will become eligible to try for the highest level SOLIDWORKS certification, the Certified SOLIDWORKS Expert.

Provides an introduction to engineering design using SolidWorks 2010 through step-by-step tutorials that cover such topics as part modeling, assembly modeling, drawing, extrude and revolve features, and top down assembly modeling.

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

SOLIDWORKS 2020 Quick Start introduces new users to the basics of using SOLIDWORKS 3D CAD software in five easy lessons. This book is intended for the student or designer who needs to learn SOLIDWORKS quickly and effectively. This book is perfect for engineers in industry who are expected to have SOLIDWORKS skills for their company's next project or students

who need to learn SOLIDWORKS without taking a comprehensive CAD course. Based on years of teaching SOLIDWORKS to engineering students, SOLIDWORKS 2020 Quick Start concentrates on the areas where new users can improve efficiency in the design modeling process. By learning the correct SOLIDWORKS skills and file management techniques, you gain the most knowledge in the shortest period of time. This book begins with an overview of SOLIDWORKS and the User Interface (UI), its menus, toolbars and commands. With a quick pace, you learn the essentials of 2D sketching, part and assembly creation, perform motion study, develop detailed part and assembly drawings and much more. Throughout this book you develop a mini Stirling Engine and investigate the proper design intent and constraints.

SOLIDWORKS 2017 Advanced Techniques picks up where SOLIDWORKS 2017 Intermediate Skills leaves off. Its aim is to take you from an intermediate user with a basic understanding of SOLIDWORKS and modeling techniques to an advanced user capable of creating complex models and able to use the advanced tools provided by SOLIDWORKS. The text covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds. Every lesson and exercise in this book was created based on real world projects. Each of these projects have been broken down and developed into easy and comprehensible steps. Furthermore, at the end of every chapter there are self test questionnaires to ensure that you have gained sufficient knowledge from each section before moving on to more advanced lessons. This book takes the approach that in order to understand SOLIDWORKS, inside and out, you should create everything from the beginning and take it step by step.

Provides an introduction to SolidWorks 2010 through step-by-step tutorials that cover such topics as linkage assembly, front support assembly, the fundamentals of drawing, and pneumatic test module assembly.

SolidWorks 2015 Part II - Advanced Techniques picks up where SolidWorks 2015 Part I - Basic Tools leaves off. Its aim is to take you from an intermediate user with a basic understanding of SolidWorks and modeling techniques to an advanced user capable of creating complex models and able to use the advanced tools provided by SolidWorks. The text covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds. Every lesson and exercise in this book was created based on real world pro-

jects. Each of these projects have been broken down and developed into easy and comprehensible steps for the reader. Furthermore, at the end of every chapter there are self test questionnaires to ensure that the reader has gained sufficient knowledge from each section before moving on to more advanced lessons. This book takes the approach that in order to understand SolidWorks, inside and out, the reader should create everything from the beginning and take it step by step. Who this book is for This book is for the intermediate user, who has already completed the Part I - Basic Tools book, or someone who is very familiar with the SolidWorks Program and its add ins.

Engineering Design with SolidWorks 2012 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 beginning 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: 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 Projects 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 multimedia 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 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 everyday. Their responsibilities go far beyond the creation of just a 3D model. The book is designed to complement the SolidWorks Tutorials contained in SolidWorks 2012.

- The perfect follow up to SOLIDWORKS Intermediate Skills
- Uses a step by step tutorial approach with real world projects
- Comprehensive coverage of advanced SOLIDWORKS tools and techniques
- Covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds
- Features a quick reference guide and a Certified SOLIDWORKS Professional practice exam

SOLIDWORKS 2020 Advanced Techniques picks up where SOLIDWORKS 2020 Intermediate Skills leaves off. Its aim is to take you from an intermediate user with a basic understanding of SOLIDWORKS and modeling techniques to an advanced user capable of creating complex models and able to use the advanced tools provided by SOLIDWORKS. The text covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds. Every lesson and exercise in this book was created based on real world projects. Each of these projects has been broken down and developed into easy and comprehensible steps. Furthermore, at the end of every chapter there are self test questionnaires to ensure that you have gained sufficient knowledge from each section before moving on to more advanced lessons. This book takes the approach that in order to understand SOLIDWORKS, inside and out, you should create everything from the beginning and take it step by step.

This book is based on the actual CSW-PA-SM exam. It is the most complete and comprehensive book on CSWPA-SM available, and it will thoroughly prepare you to take and pass the CSWPA-SM exam. The CSWPA-SM exam is one of the most challenging of the CSWPA exams and this book will help you prepare for this exam. There are 15 Questions in the exam as is in this book and the exam length is 90 minutes. Each and every step comes with actual exam screen captures and step by step solutions with screen captures from solidworks. Solidworks is one of the most widely used MCAD packages on the planet and it is an excellent example of how certification can distinguish a standout designer or design engineer from the rest of the crowd. The completion of the Certified SOLIDWORKS Professional Advanced Sheet Metal (CSW-PA-SM) exam shows that you have successfully demonstrated your ability to use

SOLIDWORKS Sheet Metal tools and employers can be confident that an individual with this certification understands the set of SOLIDWORKS tools that aid in the design of sheet metal components. A journey of a thousand miles begins with the first step, take the step today by purchasing this eBook or in fact a roadmap to passing your Certified SOLIDWORKS Professional Advanced Sheet Metal (CSWPA-SM) Exam and joining a family of thousands of Certified Solidworks Professionals across the Globe. This book is not only for someone who wants to write and pass the Certified SOLIDWORKS Professional Advanced Sheet Metal (CSWPA-SM) exam but it is also for those who want to learn the best practice in using sheet metal tools in Solidworks as well as those who want to keep their skills honed and or maintain their ingenuity and proficiency with the software. To write the CSWPA-SM exam, one should have passed the CSWA and CSWP exams.

This book will provide you with a wealth of information about the three segments of the CSWP CORE exam. The intended audience for this book is a person who has passed the CSWA exam and who has eight or more months of SOLIDWORKS training

and usage. This guide is not intended to teach you how to use SOLIDWORKS, but is written to provide you with CSWP exam tips, hints and information on sample questions and categories that are aligned with the exam. This guide is written to help you take and pass the CSWP exam. The book is organized into three chapters. Each chapter is focused on a segment of the CSWP CORE exam. This is not intended to be a step-by-step book. Goals of this book The primary goal is not only to help you pass the CSWP CORE exam, but also to ensure that you understand and comprehend the concepts and implementation details of the process. The second goal is to provide the most comprehensive coverage of CSWP CORE 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 all three segments of the CSWP CORE exam. Who this book is for The intended audience for this book and the CSWP exam is a person who has passed the CSWA exam and who has eight or more months of SOLIDWORKS training and usage. However, passing the CSWA exam is not a prerequisite for taking the CSWP exam if you

are a commercial user in industry. For students that take the CSWP exam through their school, you must first pass the CSWA exam.

SOLIDWORKS 2016 Advanced Techniques picks up where SOLIDWORKS 2016 Intermediate Skills leaves off. Its aim is to take you from an intermediate user with a basic understanding of SOLIDWORKS and modeling techniques to an advanced user capable of creating complex models and able to use the advanced tools provided by SOLIDWORKS. The text covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds. Every lesson and exercise in this book was created based on real world projects. Each of these projects have been broken down and developed into easy and comprehensible steps for the reader. Furthermore, at the end of every chapter there are self test questionnaires to ensure that the reader has gained sufficient knowledge from each section before moving on to more advanced lessons. This book takes the approach that in order to understand SOLIDWORKS, inside and out, the reader should create everything from the beginning and take it step by step.