

Introduction To Pro Sheetmetal Sdc Publications

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 • SOLIDWORKS Simulation • PhotoView 360 • Pack and Go • 3D PDFs • 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 sketches 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 designed to compliment 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.

CAD/CAM PARAMETRIC 2.0 designed in direct consultation with PTC to go hand in hand with the latest release of CreoTM Elements/Pro software, formerly known as Pro/ENGINEER. The text acts as a user friendly guide to the program walking the reader through the software and helping them to gain a better understanding of CreoTM Parametric, its assets, and uses. Step by step instructions are provided for utilizing the new capabilities and attributes of the redesigned software. Important Notice: Media content referenced within the product description or the product text may not be available in the ebook version.

Metal protection, including both metal treatments and coating systems, affords mutual protection for both can and contents. This book is the first reference to meld the knowledge of chemical companies and canmaking companies, covering materials and processes used in both protective and decorative aspects of metal packaging. Topics include basic substrates (aluminum and steel), demands of the markets served, basic metal-forming processes, and the specific decorative and protective needs of different packaging types, with emphasis give to the technologies most likely to be used, such as ultraviolet curing. This practical reference gives readers a background and familiarity with terminology and technology and gives insight into why certain technologies are used over others.

A Click-by-click Primer
PTC CreoTM Parametric 3.0
Engineering Design with SolidWorks 2001
AN INTRODUCTION TO PRO/SHEETMETAL
Architectural Modeling and Documentation
Drawing and Detailing with SolidWorks 2007 is written to educate and assist students, designers, engineers and professionals in the following areas: A solid foundation using SolidWorks Drawing Options and SolidWorks Detailing Options. Applying Engineering drawing standards and practices using SolidWorks tools. Building multiple part and assembly configurations that interact with drawings, Bill of Materials and Design Tables. A comprehensive understanding of the differences between Drawing Templates and Sheet Formats. Increase SolidWorks functionality to create view types with various configurations. Combine a series of SolidWorks tools to solve a specific problem using Custom Properties and SolidWorks Properties. The book utilizes a competency-based approach on five projects. Real world parts, projects and tasks are addressed. Commands are presented in a step-by-step progressive approach. The learning process is explored through a series of design situations, industry scenarios, projects and objectives. Table of Contents Introduction 1. Drawing Template and Sheet Format 2. Drawing View 3. Fundamentals of Detailing 4. Assembly Drawing 5. Applied Geometric Tolerancing and Other Symbols Appendix Index

Modeling with Creo Parametric 2.0 synergistically integrates the design process with the specific commands and procedures of Creo Parametric 2.0 through a unique presentation scheme. Users are first provided with the information about the design (part or assembly), and its design intent. Then, they see an overview of steps involved in modeling the part/assembly. This is accompanied by detailed instructions showing goals, steps and commands in a four-column presentation. The consistent approach is supplemented by many illustrations on each page. Each chapter adds new information while reinforcing key concepts.

The SOLIDWORKS 2019 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2019. 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 2019. 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 • SOLIDWORKS Simulation • PhotoView 360 • Pack and Go • 3D PDFs • Intelligent Modeling techniques • 3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2019 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 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. The book provides access to over 260 models, their solutions and additional support materials. Learn by doing, not just by reading. 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, configurations and more. The book is designed to complement the Online Tutorials and Online Help contained in SOLIDWORKS 2019. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The author developed the tutorials by combining his own Parametric Modeling with Creo Parametric 8.0

A Step-by-step Project Based Approach Utilizing 3D Solid Modeling
Drawing and Detailing with SolidWorks 2007
SolidWorks 2003 tutorial
A PROJECT BASED APPROACH RELEASE 20/2000I
The Technology of Metal Protection and Decoration

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 compliment the SolidWorks tutorials contained in SolidWorks 2012.

SOLIDWORKS 2021 Advanced Techniques picks up where SOLIDWORKS 2021 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. Who this book is for This book is for the intermediate to advanced user who has already completed the SOLIDWORKS Basic Tools book and may have also completed the SOLIDWORKS Intermediate Skills book. People who are very familiar with SOLIDWORKS and its add ins will also find this book to be a valuable resource.

This book is intended to familiarize you with the basics of theory and practice in Adams Multibody Dynamics (MBD) modeling. The content has been developed to be beneficial to readers who are students or practicing engineers who are either completely new to MBD modeling or have some experience with MBD modeling. The author's lengthy experience using the Adams software adds a practical and, occasionally, humorous complement to standard documentation and training materials, intended to benefit you while learning Adams. The book features relatively small examples which you can readily build and execute. This book contains an introduction to Adams theory which provides the basics on how Adams models are formulated and then numerically solved. Finally, this book concludes with some success stories taken from industry.

Solidworks 2007 Tutorial with Multimedia CD
Parametric Modeling with Creo Parametric 6.0
Part 2- Lessons 13-22
Engineering Design with SolidWorks 2012

Official Guide to Certified SOLIDWORKS Associate Exams: CSWA, CSWA-SD, CSWSA-FEA, CSWA-AM (2017-2019)
SOLIDWORKS 2021 Advanced Techniques

Provides tutorial style lessons that cover such topics as creating a simple object, modeling utilities, datum planes and sketched tools, patterns and copies, engineering drawings, and assembly operations.

- *Uses step-by-step tutorials designed for novice users • Explains not only how but also why commands are used • Covers part and assembly creation, creating engineering drawings and parametric solid modeling The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 8.0. The tutorial covers the major concepts and frequently used commands required to advance from a novice to an intermediate user level. Major topics include part and assembly creation, and creation of engineering drawings. Also illustrated are the major functions that make Creo Parametric a parametric solid modeler. Although the commands are presented in a click-by-click manner, an effort has been made, in addition to showing/illustrating the command usage, to explain why certain commands are being used and the relation of feature selection and construction to the overall part design philosophy. Simply knowing how commands can be found is only half the battle. As is pointed out numerous times in the text, creating useful and effective models of parts and assemblies requires advance planning and forethought. Moreover, since error recovery is an important skill, considerable time is spent exploring the created models. In fact, some errors are intentionally induced so that users will become comfortable with the “debugging” phase of model creation. At the end of each lesson is a short quiz reviewing the new topics covered in that chapter. Following the quiz are several simple “exercise” parts that can be created using new commands taught in that lesson. In addition to these an ongoing project throughout the book is also included. This project consists of several parts that are introduced with the early lessons and finally assembled at the end. Who this book is for This book has been written specifically with students in mind. Typically, students enter their first CAD course with a broad range of abilities both in spatial visualization and computer skills. The approach taken here is meant to allow accessibility to persons of all levels. These lessons, therefore, were written for new users with no previous experience with CAD, although some familiarity with computers is assumed. The tutorials in this textbook cover the following topics: • Introduction to the program and its operation • The features used in part creation • Modeling utilities • Creating engineering drawings • Creating assemblies and assembly drawings*

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.

SOLIDWORKS 2020 Reference Guide
SolidWorks 2001 Tutorial
A Workbook For SolidWorks 2001/2001Plus

Introduction to Autodesk Land Desktop 2006 and Civil Design 2006
A Competency Project Based Approach Utilizing 3D Solid Modeling
Parametric modeling with pro/ENGINEER

- A comprehensive reference book for SOLIDWORKS 2020 • Contains 260 plus standalone tutorials • Starts with a basic overview of SOLIDWORKS 2020 and its new features • Tutorials are written for each topic with new and intermediate users in mind • Includes access to each tutorial's initial and final state • Contains a chapter introducing you to 3D printing The SOLIDWORKS 2020 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SOLIDWORKS 2020. 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 2020. 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 • SOLIDWORKS Simulation • PhotoView 360 • Pack and Go • 3D PDFs • Intelligent Modeling techniques • 3D printing terminology and more Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SOLIDWORKS 2020 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 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. The book provides access to over 260 models, their solutions and additional support materials. Learn by doing, not just by reading. 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, configurations and more. The book is designed to complement the Online Tutorials and Online Help contained in SOLIDWORKS 2020. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The author developed the tutorials by combining his own industry experience with the knowledge of engineers, department managers, professors, vendors and manufacturers. He is directly involved with SOLIDWORKS every day and his responsibilities go far beyond the creation of just a 3D model.

The primary goal of Parametric Modeling with Creo Parametric 8.0 is to introduce the aspects of Solid Modeling and Parametric Modeling. This text is intended to be used as a training guide for any student or professional wanting to learn to use Creo Parametric. This text covers Creo Parametric and the lessons proceed in a pedagogical fashion to guide you from constructing basic shapes to building intelligent solid models and creating multi-view drawings. This text takes a hands-on, exercise-intensive approach to all the important Parametric Modeling techniques and concepts. This textbook contains a series of 13 tutorial style lessons designed to introduce beginning CAD users to Creo Parametric. The basic premise of this book is that the more designs you create using Creo Parametric, the better you learn the software. With this in mind, each lesson introduces a new set of commands and concepts, building on previous lessons. This book will provide you with a good basis for exploring and growing in the exciting field of Computer Aided Engineering. This book also introduces you to the general principles of 3D printing including a brief history of 3D printing, the types of 3D printing technologies, commonly used filaments, and the basic procedure for printing a 3D model. 3D printing makes it easier than ever for anyone to start turning their designs into physical objects and by the end of this book you will be ready to start printing out your own designs.

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.

Pipe Fitting and Piping Handbook
Referencing the ASME Y14 Engineering Drawing and Related Documentation Practices
Drawing and Detailing with SolidWorks 2003

Pro/MECHANICA Tutorial Structure (release 2000I2 - Integrated Mode)
Official Guide to Certified SOLIDWORKS Associate Exams: CSWA, CSWA-SD, CSWSA-S, CSWA-AM (SOLIDWORKS 2019 - 2021)

The purpose of the Pro/ENGINEER 7.0 Advanced Tutorial is to introduce you to some of the more advanced features, commands, and functions in Creo Parametric. Each lesson concentrates on a few of the major topics and the text attempts to explain the “why’s” of the commands in addition to a concise step-by-step description of new command sequences. This book is suitable for a second course in Creo Parametric and for users who understand the features already covered in Roger Toogood’s Creo Parametric Tutorial. The style and approach of the previous tutorial have been maintained from the previous book and the text picks up right where the last tutorial left off. The material covered in this tutorial represents an overview of what is felt to be the most commonly used and important functions. These include customization of the working environment, advanced feature creation (sweeps, round sets, draft and tweaks, UDFs, patterns and family tables), layers, Pro/PROGRAM, and advanced drawing and assembly functions. Creo Parametric 7.0 Advanced Tutorial consists of eight lessons. A continuing theme throughout the lessons is the creation of parts for a medium-sized modeling project. The project consists of a small three-wheeled utility cart. Project parts are given at the end of each lesson that utilize functions presented earlier in that lesson. Final assembly is performed in the last lesson.

The purpose of Pro Parametric 7.0 Advanced Tutorial is to introduce you to some of the more advanced features, commands, and functions in Creo Parametric. Each lesson concentrates on a few of the major topics and the text attempts to explain the “why’s” of the commands in addition to a concise step-by-step description of new command sequences. This book is suitable for a second course in Creo Parametric and for users who understand the features already covered in Roger Toogood’s Creo Parametric Tutorial. The style and approach of the previous tutorial have been maintained from the previous book and the text picks up right where the last tutorial left off. The material covered in this tutorial represents an overview of what is felt to be the most commonly used and important functions. These include customization of the working environment, advanced feature creation (sweeps, round sets, draft and tweaks, UDFs, patterns and family tables), layers, Pro/PROGRAM, and advanced drawing and assembly functions. Creo Parametric 7.0 Advanced Tutorial consists of eight lessons. A continuing theme throughout Black and White version of Creo Parametric 4.0 (Part 2) (Lessons 13-22) Includes a complete set of Lectures (available on line through YouTube) for Lessons and Projects.

Tutorial and Multimedia CD
Autodesk VIZ 4 for Beginners
Revit 4, 3 Basics

Introduction to Mechanical System Simulation Using Adams
Pro/Engineer Tutorial and MultiMedia CD
Creo Parametric 6.0 Advanced Tutorial

This unique text and video set presents a thorough introduction to Autodesk Inventor for anyone with little or no prior experience with CAD software. It can be used in virtually any setting from four year engineering schools to on-the-job use or self-study. Unlike other books of its kind, it begins at a very basic level and ends at a very advanced level. It’s perfect for anyone interested in learning Autodesk Inventor quickly and effectively using a “learning by doing” approach. Additionally, the extensive videos that are included with this book make it easier than ever to learn Inventor by clearly demonstrating how to use its tools. The philosophy behind this book is that learning computer aided design programs is best accomplished by emphasizing the application of the tools. Students also seem to learn more quickly and retain information and skills better if they are actually creating something with the software program. The driving force behind this book is “learning by doing.” The instructional format of this book centers on making sure that students learn by doing and that students can learn from this book on their own. In fact, this is one thing that differentiates this book from others: the emphasis on being able to use the book for self-study. The presentation of Autodesk Inventor is structured so that no previous knowledge of any CAD program is required. This book uses the philosophy that Inventor is mastered best by concentrating on applying the program to create different types of solid models, starting simply and then using the program to progressively create more complex solid models. The Drawing Activities at the end of each chapter are more complex iterations of the part developed by each chapter’s objectives. Since CAD programs are highly visual, there are graphical illustrations showing how to use the program. This reinforces the “learn by doing” philosophy since a student can see exactly what the program shows, and then step through progressive commands to implement the required operations. Rather than using a verbal description of the command, a screen capture of each command is replicated.

SolidWorks 2007 Tutorial was written to assist students, designers, engineers and professionals. The book provides an introduction to the user interface, menus, toolbars, concepts and modeling techniques of SolidWorks to create parts, assemblies and drawings. Follow the step-by-step instructions and develop multiple assemblies that combine over 80 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, design tables and configurations. Review 2 hours of flash movie files that follow the steps in the book. Desired outcomes and usage competencies are listed for each project. Know your objective up front. Follow the steps in Project 1 through Project 4 to achieve your design goals. Work between multiple documents, features, commands and custom properties that represent how engineers and designers utilize SolidWorks in industry. Table of Contents Introduction 1. Linkage Assembly 2. Front Support Assembly 3. Fundamentals of Drawing 4. Pneumatic Test Module Assembly Appendix Index

The primary goal of Parametric Modeling with Pro/ENGINEER Wildfire 5.0 is to introduce the aspects of solid modeling and parametric modeling. The text is a hands-on, exercise-intensive approach to all the important parametric modeling techniques and concepts. This book contains a series of eleven tutorial style lessons designed to introduce beginning CAD users to the most commonly used features of Pro/ENGINEER. Each lesson introduces a new set of commands and concepts, building on previous lessons. This text guides you from constructing basic shapes to building intelligent solid models and creating multi-view drawings. The basic premise of this book is that the more designs you create, the better you learn the software. This book will establish a good basis for exploring and growing in the exciting field of computer aided engineering. By the end of this book the reader will advance to an intermediate level Pro/ENGINEER user.

Creo Parametric 8.0 Tutorial
SOLIDWORKS 2019 Reference Guide
Canmaking

Creo Parametric 4. 0
Modeling Using Creo Parametric 2.0
A Basic Introduction

Designed in direct consultation with PTC to work hand-in-hand with the latest release of PTC Creo software (formerly known as Pro/ENGINEER), PTC CREOTM PARAMETRIC 3.0 provides step-by-step instructions to help readers understand the uses, assets, attributes, and new capabilities of the redesigned software. This user-friendly guide is the first book on the market on PTC Creo 3.0 and provides all the information, screen shots, and detailed illustrations necessary for effective use of the software as an engineering design tool. The book is enhanced by a free companion website featuring online lessons, online lectures, and a link to the free downloadable PTC Creo Student Edition software. Important Notice: Media content referenced within the product description or the product text may not be available in the ebook version.

The primary goal of Parametric Modeling with Creo Parametric 6.0 is to introduce the aspects of Solid Modeling and Parametric Modeling. This text is intended to be used as a training guide for any student or professional wanting to learn to use Creo Parametric. This text covers Creo Parametric and the lessons proceed in a pedagogical fashion to guide you from constructing basic shapes to building intelligent solid models and creating multi-view drawings. This text takes a hands-on, exercise-intensive approach to all the important Parametric Modeling techniques and concepts. This textbook contains a series of 13 tutorial style lessons designed to introduce beginning CAD users to Creo Parametric. The basic premise of this book is that the more designs you create using Creo Parametric, the better you learn the software. With this in mind, each lesson introduces a new set of commands and concepts, building on previous lessons. This book will provide you with a good basis for exploring and growing in the exciting field of Computer Aided Engineering. This book also introduces you to the general principles of 3D printing including a brief history of 3D printing, the types of 3D printing technologies, commonly used filaments, and the basic procedure for printing a 3D model. 3D printing makes it easier than ever for anyone to start turning their designs into physical objects and by the end of this book you will be ready to start printing out your own designs.

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-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.

CreoTM Parametric 2.0
Official Certified SolidWorks Associate (CSWA) Examination Guide
Parametric Modeling With Pro/Engineer Wildfire 5.0

Pro/ENGINEER Advanced Tutorial
release 2000I sup 2
Pro/ENGINEER Wildfire 5.0
Engineering Design with SolidWorks 2001A Competency Project Based Approach Utilizing 3D Solid ModelingSchroff Development CorporationSolidWorks 2001 TutorialA Basic IntroductionSchroff Development CorporationAN INTRODUCTION TO PRO/SHEETMETAL A PROJECT BASED APPROACH RELEASE 20/2000IParametric Modeling with Pro/Engineer (Release 20/2000I) Parametric Modeling with Pro/Engineer 2.0Crosstong Publishing

A Commands Guide Tutorial for SolidWorks 2007
A Reference Guide
Commands Guide Tutorial for SolidWorks 2013

Autodesk Inventor 2020 A Tutorial Introduction
An Introduction to Creo Parametric 8.0
Drawing and Detailing with SolidWorks