Creo Practise In

SOLIDWORKS Exercises - Learn by Practicing (3rd Edition) book is designed to help engineers and designers interested in learning SOLIDWORKS by practicing 100 real-world mechanical models. This book does not simply provide step-by-step instructions to design 3D models, instead it is a practice book that challenges users to first analyze the drawings and then create the models using the powerful toolset of SOLIDWORKS. This approach helps users to enhance their design skills and take it to the next level. You can also access the video instruction for creating each exercise of the book. This book is written with a wide range of SOLIDWORKS users in mind, varying from beginners to advanced users. In addition to SOLIDWORKS, each exercise of this book can also be designed on any other CAD software such as CATIA, Creo Parametric, NX, Autodesk Inventor, and Solid Edge. NOTE: The exercises/models available for download are created in SOLIDWORKS 2021 and cannot be opened in the lower version of SOLIDWORKS.

The Creo Parametric 7.0: Surface Design learning guide focuses on the creation of complex geometry that cannot be created easily using solid features. It provides users with a basic understanding of surface modeling styles and extensive practices to practice the new functionality used to create complex geometry. Topics Covered Surface Basics Reference Geometry Splines and Conics Creating Simple Surfaces Surface Operations Creating

Surfaces from Boundaries Analysis Tools Advanced Surfaces (Curvature Continuous Surfaces, Blend Tangent to Surfaces, Ribbon Surface) Advanced Swept Surfaces Offset Surfaces Introduction to Data Exchange (Import Data Doctor) Prerequisites Access to the Creo Parametric 7.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition. Completion of Creo Parametric 7.0: Introduction to Solid Modeling and Creo Parametric 7.0: Advanced Part Design or equivalent Creo Parametric experience is recommended.
Note: To complete this course, "Creo Parametric 2.0: Introduction to Solid Modeling - Part 2" is required. Learn the process of designing models with Creo Parametric 2.0 from 2D sketching, through to solid part modeling, assembly creation, and drawing production. Gain an understanding of the design philosophy of Creo Parametric 2.0 through this extensive hands-on course with numerous practice exercises. It is expected that all new users of Creo Parametric 2.0 will require this course. Topics include: Creo Parametric fundamentals and interface Principles behind design intent Manipulating a model Creo Parametric file management Part creation and modification Sketching and creating geometry Sketcher mode functionality (sketching and dimensioning) Datum features Duplication techniques (patterns, mirroring) Creating relations to capture design intent Creo Parametric customization Design documentation and detailing Feature management

Sweeps and blends Assembly creation and manipulation Parent/Child relationships in Creo Parametric models Model Analysis Feature failure resolution Effective modeling techniques Prerequisites: Experience in mechanical design and drawing production is recommended.

The Creo Parametric 6.0: Introduction for Non-Designers learning guide provides reviewers or downstream users of Creo Parametric data with the knowledge to investigate, manipulate, and annotate existing models. It is targeted at students who require less training about geometry creation techniques. The student learns to open models for the purpose of providing feedback, verification, image capture, and taking data into specialized modules. This learning guide provides a good introduction to Creo Parametric for users who are evaluating the software or need a high-level understanding of software's capabilities. This guide was developed against Creo Parametric 6.0 Build 6.0.4.0. Topics Covered Creo Parametric interface Obtaining model information Display control Creating datum features View manager 3D annotations Creating planar and offset cross-sections View creating and detailing File management Prerequisites Access to the Creo Parametric 6.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition.

Note: This learning guide is the first of a two-part series, with each guide sold separately. The Creo Parametric

6.0: Introduction for Experienced 3D CAD Users learning guide is intended to provide accelerated introductory training in Creo Parametric 6.0 software. This learning guide is designed for users that have 3D modeling design experience with other 3D CAD software packages (e.g., CATIA(TM), Inventor(TM), NX(TM), SolidWorks(R), etc.). By leveraging the experience users gain in working with other 3D modeling software packages, this hands-on, practice-intensive guide is developed so that users who are new to Creo Parametric can benefit from a shorter, introductory-level, learning guide. You are taught how to find and use the modeling tools associated with familiar modeling strategies that are used in other 3D CAD software. You will acquire the knowledge necessary to complete the process of creating models from conceptual sketching, through to solid modeling, assembly design, and drawing production. Topics Covered Creo Parametric fundamentals and interface Manipulating a model Creo Parametric file management Part creation and modification Sketching and creating geometry Sketcher mode functionality (sketching and dimensioning) Datum features Duplication techniques (patterns, mirroring) Creating relations to capture design intent Creo Parametric customization Design documentation and detailing Feature management Sweeps and blends Assembly creation and manipulation Parent/child relationships in Creo Parametric models Model analysis Feature failure resolution Prerequisites Access to the Creo Parametric 6.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included

with this guide are compatible with the commercial version of the software, but not the student edition. Experience in mechanical design and drawing production using 3D CAD software. This content was developed using Creo Parametric 6.0 Build 6.0.4.0.

The Creo Parametric 8.0 Black Book is the 6th edition of our series on Creo Parametric. With lots of additions and thorough review, we present a book to help professionals as well as learners in creating some of the most complex solid models. The book follows a step by step methodology. In this book, we have tried to give real-world examples with real challenges in designing. We have tried to reduce the gap between university use of Creo Parametric and industrial use of Creo Parametric. In this edition of book, we have included new enhancements of Creo Parametric 8.0 interface. We have included an introductory chapter on Live Simulation in this edition. The book covers almost all the information required by a learner to master Creo Parametric. The book starts with sketching and ends at advanced topics like Sheetmetal, Surface Design, 3D Printing, MBD, Sheet metal NC manufacturing, and Live Simulation. Some of the salient features of this book are: In-Depth explanation of concepts Every new topic of this book starts with the explanation of the basic concepts. In this way, the user becomes capable of relating the things with real world. Topics Covered Every chapter starts with a list of topics being covered in that chapter. In this way, the users can easy find the topic of their interest easily. Instruction through illustration The instructions to perform any action are provided by maximum number of

illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about 1500 small and large illustrations that make the learning process effective. Tutorial point of view At the end of concept's explanation, the tutorial make the understanding of users firm and long lasting. Almost each chapter of the book has tutorials that are real world projects. Most of the tools in this book are discussed in the form of tutorials. Project Projects and exercises are provided to students for practicing. For Faculty If you are a faculty member, then you can ask for video tutorials on any of the topic, exercise, tutorial, or concept.
Designing with Creo Parametric 3.0 provides the high school student, college student, or practicing engineer with a basic introduction to engineering design while learning the 3D modeling Computer-Aided Design software called Creo Parametric from PTC. The topics are presented in tutorial format with exercises at the end of each chapter to reinforce the concepts covered. It is richly illustrated with computer screen shots throughout. Above all, this text is designed to help the reader expand their creative talents and communicate their ideas through the graphics language. Because it is easier to learn new information if you have a reason for learning it, this textbook discusses design intent while you are learning Creo Parametric. At the same time, it shows how knowledge covered in basic engineering courses such as statics, dynamics, strength of materials, and design of mechanical components can be applied to design. You do not need an engineering degree nor be working toward a degree in engineering to use this

textbook. Although FEA (Finite Element Analysis) is used in this textbook, its theory is not covered. The first two chapters of this book describe the design process. The meat of this text, learning the basic Creo Parametric software, is found in Chapters 3 through 6. Chapters 7, 8, and 12 deal with dimensioning and tolerancing an engineering part. Chapters 9 and 10 deal with assemblies and assembly drawings. Chapter 11 deals with family tables used when similar parts are to be designed or used. Chapter 13 is an introduction to Creo Simulate and FEA.

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world. Topics Covered: Every chapter starts with a list of topics being covered in that chapter. In this way, the user can easy find the topic of his/her interest easily. Instruction through illustration: The instructions to perform any action are provided by maximum number of illustrations so that the user can perform the actions discussed in the book easily and effectively. There are about 900 illustrations that make the learning process effective. Tutorial point of view: At the end of concept's explanation, the tutorial make the understanding of users firm and long lasting. Almost each chapter of the book has tutorials that are real world projects. Project: Free projects and exercises are provided to students for practicing. For Faculty: If you are a faculty member, then you can ask for video tutorials on any of the topic, exercise, tutorial, or concept. Table of Contents: Starting with Creo Parametric Sketching Advanced Sketching & Practicals 3D Modeling Basics 3D Modeling Practical & Practice 3D Modeling Advanced 3D Modeling Advanced Practical and Practice Assembly and Practical Sheetmetal Drawing Surface Design Buy the book from https://www.createspace.com/5141870 and apply my author code: EUAFU3ZM for 50% discount."

The Creo Parametric 7.0: Design Documentation and Detailing learning guide is designed for all draftspersons that document designs using Creo Parametric. It focuses on how to use Creo

Parametric to communicate design information from your part and assembly models. This content was developed using Creo Parametric 7.0, Build 7.0.2.0. Topics Covered View creation View manipulation Detailing a drawing Drawing notes Tolerances Assembly drawings Drawing tables 2D sketching Symbols Prerequisites Access to the Creo Parametric 7.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition. Completing Creo Parametric 7.0: Introduction to Solid Modeling or equivalent Creo Parametric 7.0 experience. PTC CREO ASSEMBLY DRAWINGS This book has been designed for self-paced learning by doing assembly practice exercises. This book doesn't provide you with a step by step tutorial. This book is intended to provide cad assembly practice exercises. What's included in the PTC CREO ASSEMBLY DRAWINGS book? Whether you are a beginner, intermediate, or an expert, these CAD Assembly exercises will challenge you. The book has various cad assembly exercises. Each exercise contains images of the final Assembly design and exact measurements needed to create the design. Each exercise can be designed on any CAD software which you desire. It can be done with AutoCAD, SolidWorks, CATIA, DraftSight, Fusion 360, Solid

Edge, NX and other feature-based CAD modeling software. It is intended to provide Drafters, Designers and Engineers with enough CAD Assembly exercises for practice on any cad program. It includes almost all types of exercises that are necessary to provide, clear, concise and systematic information required on industrial machine part drawings. Third Angle Projection is intentionally used to familiarize Drafters, Designers and Engineers in Third Angle Projection to meet the expectation of worldwide Engineering drawing print. This book is for Beginner, Intermediate and Advance CAD users. Clear and well drafted drawing help easy understanding of the design. These exercises are from Basics to Advance level. Each exercises can be assigned and designed separately. No Exercise is a prerequisite for another. All dimensions are in mm. Prerequisite To design & develop models, you should have knowledge of PTC CREO program. Student should have knowledge of Orthographic views and projections. Student should have basic knowledge of engineering drawings.

Designing with Creo Parametric 8.0 provides the high school student, college student, or practicing engineer with a basic introduction to engineering design while learning the 3D modeling Computer-Aided Design software called Creo Parametric from PTC. The topics are presented in tutorial format with exercises at the end of each chapter to reinforce the

concepts covered. It is richly illustrated with computer screen shots throughout. Above all, this text is designed to help you expand your creative talents and communicate your ideas through the graphics language. Because it is easier to learn new information if you have a reason for learning it, this textbook discusses design intent while you are learning Creo Parametric. At the same time, it shows how knowledge covered in basic engineering courses such as statics, dynamics, strength of materials, and design of mechanical components can be applied to design. You do not need an engineering degree nor be working toward a degree in engineering to use this textbook. Although FEA (Finite Element Analysis) is used in this textbook, its theory is not covered. The first two chapters of this book describe the design process. The meat of this text, learning the basic Creo Parametric software, is found in Chapters three through six. Chapters seven, eight, and 12 deal with dimensioning and tolerancing an engineering part. Chapters nine and ten deal with assemblies and assembly drawings. Chapter 11 deals with family tables used when similar parts are to be designed or used. Chapter 13 is an introduction to Creo Simulate and FEA. Table of Contents 1. Computer Aided Design 2. Introduction 3. Sketcher 4. Extrusions 5. Revolves 6. Patterns 7. Dimensioning 8. Engineering Drawings 9. Assemblies 10. Assembly Drawings 11. Relations

and Family Tables 12. Tolerancing and GD&T 13. Creo Simulate and FEA Appendix A: Parameters for Drawings Appendix B: Drill and Tap Chart Appendix C: Surface Roughness Chart Appendix D: Clevis Pin Sizes Appendix E: Number and Letter Drill Sizes Appendix F: Square and Flat Key Sizes Appendix G: Screw Sizes Appendix H: Nut Sizes Appendix I: Setscrew Sizes Appendix J: Washer Sizes Appendix K: Retaining Ring Sizes Appendix L: Basic Hole Tolerance Appendix M: Basic Shaft Tolerance Appendix N: Tolerance Zones Appendix O: International Tolerance Grades References Index This book starts with Creo Parametric 3.0 using step-by-step examples. It begins with creating sketches and parts, assembling them, and then creating print ready drawings. This book gives you an idea about how you can design and document various mechanical components, and helps you to learn some advanced tools and techniques. This book also follows some of the best practices in creating parts. In addition to this, there are some additional chapters covering sheet metal and surface design. Each topic in this book has a brief introduction and a step-by-step example. This will help you to learn Creo Parametric 3.0 quickly and easily.* Go through with the User Interface* A step-by-step practice to create sketches and 3D models * Teach you about advance Part Modeling tools * Learn the procedure to create Multiple-body parts* Learn to modify

components at each step* Learn to create assemblies * Learn Top-down assembly design * Learn to create 2D drawings* Learn basic tools available in Sheet Metal and Surface Environment* Create sheet metal drawings* Create complex shapes using surface modeling tools
Understand the full assembly functionality of the Creo Parametric 6.0 software while concentrating on techniques that maximize large assembly management capabilities as well as an introduction to Top Down Design. Creo Parametric 6.0: Advanced Assembly Design and Management is a hands-on learning guide with a substantial amount of time dedicated to practices. Topics Covered Advanced Component Selection and Placement Top Down Design Managing External References Assembly Management Skeleton and Motion Skeleton Models Assembly Duplication Tools Assembly Family Tables Display Styles, Layers and Suppression Restructure Intelligent Fasteners Lite Creating Parts and Features in an Assembly Merge and Cut Out, Intersections Copy Geometry Features Inheritance Features Simplified Representations Interchange Assemblies Prerequisites Access to the Creo Parametric 6.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition. Creo

Parametric 6.0: Introduction to Solid Modeling or equivalent Creo Parametric experience.
Creo Parametric 5.0: Sheet Metal Design enables you to use your introductory modeling skills to create sheet metal models, including walls, bends, notches, and form features. On completion of this course, you will have acquired the skills to confidently manipulate sheet metal geometry, adjust bend developed lengths, and convert solid parts. Creo Parametric 5.0: Sheet Metal Design was developed against Creo Parametric 5.0.3.0. Topics Covered The sheet metal environment Primary and secondary walls Bend relief Corner relief Regular unbends, back bends, and cuts Notches and punches Bend features Unbending complex geometry Sheet metal forms Documenting a sheet metal part Converting solid parts Sheet metal setup Investigating a sheet metal part Prerequisites Access to the Creo Parametric 5.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition. Completion of Creo Parametric: Introduction to Solid Modeling, or similar levels of prior experience using the Creo Parametric software.
In Creo Parametric 5.0: Introduction to Mechanism Design, you will learn how to simulate assembly motion in Creo Parametric using the Mechanism Design extension. You will

also learn to set up your assemblies for motion, and create animations of the assembly using the Design Animation option. This hands-on learning guide contains numerous practices. This content was developed against Creo Parametric 5.0.3.0. Topics Covered MDX interface Basic assembly connections Drag Snapshot configurations Joint axis settings Servo Motors Motion playback Basic Measure analysis Advanced connections Create movies and images Design Animation Key frame sequences Motion envelopes Trace curves Interference checks Prerequisites Access to the Creo Parametric 5.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition. It is highly recommended that you have completed Creo Parametric: Introduction to Solid Modeling or Creo Parametric: Advanced Assembly Design and Management, or have similar levels of prior experience using the Creo Parametric software.

PTC CREO EXERCISESDo you want to learn how to design 2D and 3D models in your favorite Computer Aided Design (CAD) software such as PTC Creo or SolidWorks? Look no further. We have designed 200 CAD exercises that will help you to test your CAD skills.What's included in the PTC CREO EXERCISES book?Whether you are a beginner, intermediate, or an expert, these CAD exercises will challenge you. The book contains 200 3D models and practice drawings or exercises.*Each exercise contains images of the final design and exact measurements needed to create the design.*Each exercise can be designed on any CAD software which you desire. It can be done with AutoCAD, SolidWorks, Inventor, DraftSight, Fusion 360, Solid Edge, Catia, NX and other feature-based CAD modeling software.*It is intended to provide Drafters, Designers and

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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.

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Note: This book is a continuation of BOOK TITLE - Part 1. Both books are required to complete this guide. The Creo Parametric 5.0: Introduction to Solid Modeling learning guide provides you with an understanding of the process of designing models with Creo Parametric 5.0 through a hands-on, practice-intensive curriculum. You will learn the key skills and knowledge required to design models using Creo Parametric 5.0, starting with 2D sketching, through to solid part modeling, assembly creation, and drawing production. This content was developed using Creo Parametric 5.0.3.0. Topics Covered Creo Parametric fundamentals and interface Principles behind design intent Manipulating a model Creo Parametric file management Part creation and modification Sketching and creating geometry Sketcher mode functionality (sketching and dimensioning) Datum features Duplication techniques (patterns, mirroring) Creating relations to capture design intent Creo Parametric customization Design documentation and detailing Feature management Sweeps and blends Assembly creation and manipulation Parent/Child

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Ptc Creo Exercises200 Practice Drawings For CREO and Other Feature-Based Modeling SoftwareIndependently Published

As an experienced user in the basics of Creo Parametric 6.0, this learning guide enables you to create electromechanical cabling systems designed in Creo Parametric using the Piping and Cabling Extension. Utilizing the parametric and associative nature of Creo Parametric, an electromechanical designer can easily create realistic 3D cabling assemblies, wire lists, bill of material tables, and nail-board drawings. The Creo Parametric 6.0: Cable and Harness Design learning guide contains numerous practices to give you practical experience that will improve your job performance. This content was developed using Creo Parametric 6.0, Build 6.0.4.0. Topics Covered Cabling Process Overview CCabling Terminology CEnvironment and Configuration Setup CElectromechanical Model Setup CManual Designation and Parameters CManual Spools CManual Cabling Features CLogical Reference Technique CRouting Methods CModifying Cabling Assemblies CAdditional Routing Features CNetworking CCabling Assembly Deliverables CHARNESS-MFG Prerequisites Access to the Creo Parametric 6.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the learning edition. We recommend that you have completed the Creo Parametric 6.0: Introduction to Solid Modeling learning guide, or have equivalent experience.

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Note: This learning guide is the second of a two-part series, with each guide sold separately. The Creo Parametric 6.0: Introduction for Experienced 3D CAD Users learning guide is intended to provide accelerated introductory training in Creo Parametric 6.0 software. This learning guide is designed for users that have 3D modeling design experience with other 3D CAD software packages (e.g., CATIA(TM), Inventor(TM), NX(TM), SolidWorks(R), etc.). By leveraging the experience users gain in working with other 3D modeling software packages, this hands-on, practice-intensive guide is developed so that users who are new to Creo Parametric can benefit from a shorter, introductory-level, learning guide. You are taught how to find and use the modeling tools associated

with familiar modeling strategies that are used in other 3D CAD software. You will acquire the knowledge necessary to complete the process of creating models from conceptual sketching, through to solid modeling, assembly design, and drawing production. Topics Covered Creo Parametric fundamentals and interface Manipulating a model Creo Parametric file management Part creation and modification Sketching and creating geometry Sketcher mode functionality (sketching and dimensioning) Datum features Duplication techniques (patterns, mirroring) Creating relations to capture design intent Creo Parametric customization Design documentation and detailing Feature management Sweeps and blends Assembly creation and manipulation Parent/child relationships in Creo Parametric models Model analysis Feature failure resolution Prerequisites Access to the Creo Parametric 6.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition. Experience in mechanical design and drawing production using 3D CAD software. This content was developed using Creo Parametric 6.0 Build 6.0.4.0.
Note: To complete this course, "Creo Parametric 3.0: Introduction to Solid Modeling - Part 1" is required. The Creo Parametric 3.0: Introduction to Solid Modeling training guide provides you with an understanding of the process of designing models with Creo Parametric 3.0 through a hands-on, practice-intensive curriculum. You will learn the key skills and knowledge required to design models using Creo Parametric 3.0, starting with 2D sketching, through to solid part modeling, assembly creation, and drawing production. Topics include: Creo Parametric fundamentals and interface Principles behind design intent Manipulating a model Creo Parametric file management Part creation and modification

Sketching and creating geometry Sketcher mode functionality (sketching and dimensioning) Datum features Duplication techniques (patterns, mirroring) Creating relations to capture design intent Creo Parametric customization Design documentation and detailing Feature management Sweeps and blends Assembly creation and manipulation Parent/Child relationships in Creo Parametric models Model Analysis Feature failure resolution Effective modeling techniques Prerequisites: Experience in mechanical design and drawing production is recommended. "Creo Parametric 3.0: Introduction to Solid Modeling - Part 1"

In the Creo Parametric 6.0: Introduction to Mechanism Design learning guide, you will learn how to simulate assembly motion in Creo Parametric using the Mechanism Design extension. You will also learn to set up your assemblies for motion, and create animations of the assembly using the Design Animation option. This hands-on learning guide contains numerous practices. This content was developed against Creo Parametric 6.0.4.0. Topics Covered MDX interface Basic assembly connections Drag Snapshot configurations Joint axis settings Servo Motors Motion playback Basic Measure analysis Advanced connections Create movies and images Design Animation Key frame sequences Motion envelopes Trace curves Interference checks Prerequisites Access to the Creo Parametric 6.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition. It is highly recommended that you have completed the Creo Parametric: Introduction to Solid Modeling or Creo Parametric: Advanced Assembly Design and Management guides, or have similar levels of prior experience using the Creo Parametric software. Understand the full assembly functionality of the Creo

Parametric 5.0 software while concentrating on techniques that maximize large assembly management capabilities as well as an introduction to Top Down Design. Creo Parametric 5.0: Advanced Assembly Design and Management is a hands-on guide with a substantial amount of time dedicated to practices. This guide was developed against Creo Parametric 5.0.3.0. Topics Covered Advanced Component Selection and Placement Top Down Design Managing External References Assembly Management Skeleton and Motion Skeleton Models Assembly Duplication Tools Assembly Family Tables Display Styles, Layers and Suppression Restructure Intelligent Fasteners Lite Creating Parts and Features in an Assembly Merge and Cut Out, Intersections Copy Geometry Features Inheritance Features Simplified Representations Interchange Assemblies Prerequisites Access to the Creo Parametric 5.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition. Creo Parametric 5.0: Introduction to Solid Modeling or equivalent Creo Parametric experience.

Note: To complete this course, "Creo Parametric 3.0: Introduction to Solid Modeling - Part 2" is required. The Creo Parametric 3.0: Introduction to Solid Modeling training guide provides you with an understanding of the process of designing models with Creo Parametric 3.0 through a hands-on, practice-intensive curriculum. You will learn the key skills and knowledge required to design models using Creo Parametric 3.0, starting with 2D sketching, through to solid part modeling, assembly creation, and drawing production. Topics include: Creo Parametric fundamentals and interface Principles behind design intent Manipulating a model Creo Parametric file management Part creation and modification Sketching and creating geometry Sketcher mode functionality

(sketching and dimensioning) Datum features Duplication techniques (patterns, mirroring) Creating relations to capture design intent Creo Parametric customization Design documentation and detailing Feature management Sweeps and blends Assembly creation and manipulation Parent/Child relationships in Creo Parametric models Model Analysis Feature failure resolution Effective modeling techniques Prerequisites: Experience in mechanical design and drawing production is recommended. "Creo Parametric 3.0: Introduction to Solid Modeling - Part 2"

As an experienced user in the basics of Creo Parametric 5.0, Creo Parametric 5:0: Advanced Part Design enables you to become more productive by extending your modeling abilities with advanced functionality and techniques. This extensive hands-on learning guide contains numerous labs and practices to give you practical experience that will improve your job performance. Topics Covered Advanced datum features Advanced bends Sweeps with variable sections and helical sweeps Rotational and swept blends Designing with rounds Advanced round functionality Drafts Basic surface design Part family tables User-defined features (UDFs) Date sharing View Manager Automation (Appendix) Prerequisites Access to the Creo Parametric 5.0 software. The practices and files included with this guide might not be compatible with prior versions. Practice files included with this guide are compatible with the commercial version of the software, but not the student edition. Completing Creo Parametric 5.0: Introduction to Solid Modeling, or the equivalent Creo Parametric experience.