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Sheetmetal Design using Creo Parametric 2.0 is a comprehensive training course that teaches you how to create sheet metal parts in Creo Parametric. The course builds upon the basic lessons you learned in Introduction to Creo Parametric 2.0 and serves as the second stage of learning. In this course, you will learn how to design sheet metal parts and assemblies, including sheet metal production drawings.

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Note: To complete this course, "Creo Parametric 2.0: Introduction to Solid Modeling - Part 1" 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. "Creo Parametric 2.0: Introduction to Solid Modeling - Part 1"

This training guide enables you to use your introductory modeling skills to create sheet metal models, including wall, bends, notches, and form features. On completion of this course, the student will have acquired the skills to confidently manipulate sheet metal geometry, adjust bend developed lengths, and convert solid parts. Course topics: 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: It is recommended to complete the following courses, or have the equivalent Creo Parametric experience: "Creo Parametric 2.0: Introduction to Solid Modeling - Part 1" "Creo Parametric 2.0: Introduction to Solid Modeling - Part 2" "Creo Parametric: Core Update, Wildfire 4.0 to Creo Parametric 2.0"

As an experienced user in the basics of Creo Parametric 2.0, this training guide enables you to become more productive by extending your modeling abilities with advanced functionality and techniques. This extensive hands-on training guide contains numerous labs and exercises to give you practical experience so that you can improve your job performance. Topics include: Advanced datum features Advanced sweeps Blends and swept blends Designing with rounds Advanced round functionality Drafts Basic surface design Part family tables Advanced feature duplication User-defined features (UDFs) Date sharing Resolving failed features View Manager Automation (appendix) Prerequisites: It is recommended to complete the following courses, or have the

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equivalent Creo Parametric experience: "Creo Parametric 2.0: Introduction to Solid Modeling - Part 1" "Creo Parametric 2.0: Introduction to Solid Modeling - Part 2" "Creo Parametric: Core Update, Wildfire 4.0 to Creo Parametric 2.0"

This class is designed to improve your efficiency with Creo Parametric 2.0 by exploring and practicing the latest functionality in Creo Parametric 2.0. This course focuses on the enhancements within the core modules of Creo Parametric 2.0 and is ideal for those users updating from Wildfire 4.0 to Creo Parametric 2.0. Topics include: General Enhancements Sketcher Enhancements Part Enhancements Advance Part Enhancements Assembly Enhancements Drawing Enhancements Sheet Metal Enhancements Prerequisites: "Pro/ENGINEER: Wildfire Introduction to Solid Modeling I & II" or equivalent Wildfire 4.0 experience.

As an experienced user of Creo Parametric 3.0, the Creo Parametric 4.0: Core Update from Creo Parametric 2.0 learning guide enables you to become familiar with the enhancements that have been made to the core capabilities of Creo Parametric 4.0. This extensive hands-on learning guide contains numerous labs and practices to give you practical experience that will improve your job performance. This guide was developed against build M010 of Creo Parametric 4.0. Topics Covered User Interface Enhancements Part Modeling Enhancements Sketcher Enhancements Assembly Enhancements Drawing Enhancements Sheetmetal Enhancements Prerequisites Creo Parametric 2.0: Introduction to Solid Modeling or equivalent Creo Parametric 2.0 experience. Please note that this learning guide uses commercial practice files which may not be compatible with the Student Edition of Creo Parametric

This student guide covers the fundamentals of Creo Simulate: Structural and Thermal Analysis. It provides you with the knowledge to effectively use Creo Simulate for finite element analysis, thereby reducing design time. Many concepts apply to both Structure and Thermal analysis; a portion of this guide is specifically dedicated to Thermal analysis. This is an extensive hands-on training guide, in which you have the opportunity to apply your knowledge through real-world scenarios and examples. Topics Covered FEA Fundamentals: P-elements and analysis convergence methods Basic Modeling and Analysis Types of Loads and Constraints Idealizations: Shells and Beams Sensitivity and Optimization Studies Assembly Interfaces and Contact Analysis Thermal Analysis Modal Analysis Welds, Springs, and Masses Fasteners and Rigid Links Buckling Analysis Prerequisites Prior to taking this course, we recommend that users complete the Creo Parametric: Introduction to Solid Modeling student guide, plus complete a minimum of 80 hours of Creo Parametric experience.

This learning guide focuses on the creation of complex geometry that cannot easily be created using solid features. It provides students with a basic understanding of surface modeling styles and extensive exercises to practice the new functionality used to create complex geometry. Course topics: Surface Basics Reference Geometry Splines and Conics Creating Simple Surfaces Surface Operations Creating Surfaces from Boundaries Analysis Tools Advanced Surfaces (Curvature Continuous Surfaces, N-Sided) Advanced Swept Surfaces Advanced Surface Options (blend section, blend between surfaces, blend tangent to surfaces) Offset Surfaces Introduction to Data Exchange (Import Data Doctor) Prerequisites: Prerequisites: It is recommended to complete the following, or have the equivalent Creo Parametric experience: Creo Parametric 2.0: Introduction to Solid Modeling - Part 1 Creo Parametric 2.0: Introduction to Solid Modeling - Part 2 Creo Parametric 2.0: Advanced Part Design Creo Parametric: Core Update, Wildfire 4.0 to Creo Parametric 2.0 Please note that this learning guide uses commercial practice files which may not be compatible with the Student Edition of Creo Parametric

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"The Creo Parametric 3.0: Sheet Metal Design" student guide enables you to use your introductory modeling skills to create sheet metal models, including wall, 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. 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 "Creo Parametric: Introduction to Solid Modeling" or equivalent Creo Parametric 3.0 experience.

The primary goal of Parametric Modeling with Creo Parametric 5.0 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 eleven 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.

Understand the full assembly functionality of Creo Parametric 2.0 while concentrating on techniques that maximize large assembly management capabilities as well as an introduction to Top Down Design. This is a hands-on learning guide with a substantial amount of time dedicated to exercises. Topics include: Advanced Component Placement Top Down Design Managing External References Assembly Management Skeleton Models Packaging Assembly Duplication Tools Assembly Family Tables Display Styles Layers and Suppression Restructure Creating Parts and Features in an Assembly Controlling External References Merge and Cut Out, Intersections Copy Geometry Features Inheritance Features Simplified Representations Assembly Relations Interchange Assembles (appendix) Prerequisites: It is recommended to complete the following, or have the equivalent Creo Parametric experience: Creo Parametric 2.0: Introduction to Solid Modeling - Part 1 Creo Parametric 2.0: Introduction to Solid Modeling - Part 2 Creo Parametric: Core Update, Wildfire 4.0 to Creo Parametric 2.0 Please note that this learning guide uses commercial practice files which may not be compatible with the Student Edition of Creo Parametric

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