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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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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

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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

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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

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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

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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

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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

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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:

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Experience in mechanical design and drawing production is recommended.

"Creo Parametric 2.0: Introduction to Solid Modeling - Part 1"

As an experienced user in the basics of Creo Parametric 2.0, this training guide enables you to become more productive by

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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

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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

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the equivalent Creo Parametric
experience: "Creo Parametric 2.0:
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"Creo Parametric 2.0: Introduction to
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Parametric: Core Update, Wildfire 4.0 to
Creo Parametric 2.0"

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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.

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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

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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

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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

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Enhancements Assembly Enhancements
Drawing Enhancements Sheetmetal
Enhancements Prerequisites Creo
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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

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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

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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

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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
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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

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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

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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.

Understand the full assembly functionality

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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

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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,

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Intersections Copy Geometry Features

Inheritance Features Simplified

Representations Assembly Relations

Interchange Assemblies (appendix)

Prerequisites: It is recommended to complete the following, or have the equivalent Creo Parametric experience:

Creo Parametric 2.0: Introduction to Solid

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Parametric

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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

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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

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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

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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,

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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 class covers the fundamentals of Creo

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Simulate: Structural and Thermal Analysis. It provides students with the knowledge to effectively use Creo Simulate for finite element analysis, thereby reducing their design time. Many concepts apply to both Structure and Thermal analysis; but, a half-day is specifically dedicated to Thermal analysis. This is an

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extensive hands-on training guide, in which students have the opportunity to apply their knowledge through real-world scenarios and examples. Topics Covered

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- Basic Modeling and Analysis
- Types of Loads and Constraints
- Idealizations: Shells and

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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 Creo Parametric:
Introduction to Solid Modeling, plus a
minimum of 80 hours of Creo Parametric

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