

Creo Parametric 20 Advanced Assembly Design And Management

Advanced Assembly Tools in Creo
 Creo Advanced Assembly Tools How to Use Advanced Assembly in PTC
 Creo Advanced Assembly Design with Creo - Introduction and Overview (Part 1 of 4)
 Creo Advanced Assembly - Simplify and Control Complex Designs E5
 CREO Parametric 2.0 Assembly Basics 1
 Complex part design for beginners | Machine part 21 in Creo Parametric
 Nut and Bolt Assembly design with Animation | Part design with custom thread in
 Creo Parametric
 Creo Advanced Assembly Extension (AAX) Techniques How to set up the XP pen quick key remote
 E5
 CREO Parametric 2.0 - Basic Assemblies 1
 Complex part design with Drawing tutorial for beginners | Machine part 23 in
 Creo Parametric
 VLSI vs Software Jobs || Which one is better? Large Assembly Management using
 Advanced Assembly Extension Webinar
 Creo Parametric 11.0 - Full Multibody for Sheet Metal (Part 1)
 PTC
 Creo Advanced Assembly Extension - PTC
 Creo Showcase Advanced Assembly Techniques for a Remote Workforce
 V6 Turbo Engine - Complete CAD Build!
 PTC
 CREO Parametric - Solidworks Compatible - CREO with Chris Large
 Assembly Management and Top-Down Design Webinar
 Creo Design Advanced Highlight Reel
 Creo Complete College Course for Beginners with Training Guide
 How to Run a Top Down Design Process!
 Assembly Modeling with PTC's
 Creo Parametric - PTC
 Creo Parametric - Mechanisms - Schmidt Coupling

Parametric Modeling With Creo Parametric 1.0

Creo Parametric 6.0 Tutorial

Parametric Modeling with Creo Parametric 6.0

Autodesk Inventor Professional 2020 for Designers, 20th Edition

Presenting Creo Parametric 1.0

Advanced SOLIDWORKS 2022 for Designers, 20th Edition

Creo Parametric 7.0 Advanced Tutorial

Parametric Modeling with Creo Parametric 3.0

Autodesk Inventor Exercises

Creo Parametric 9.0 Advanced Tutorial

Creo Parametric 9.0 Tutorial

Computer Aided Virtual Manufacturing Using Creo Parametric

Parametric Modeling with Creo Parametric 10.0

Parametric Modeling with Creo Parametric 9.0

Parametric Modeling with Creo Parametric 4.0

Machine Drawing

Creo Parametric 6.0

e-Design

Advanced Tutorial for Creo Parametric Releases 1.0 & 2.0

Parametric Modeling with Creo Parametric 7.0

Modeling Using Creo Parametric 2.0

Creo Parametric 20 Advanced Assembly Design And Management

OMB No. 8491250625973 edited by

ESMERALDA ARROYO

Parametric Modeling With Creo Parametric 1.0 SDC Publications

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.

Creo Parametric 6.0 Tutorial SDC Publications

Autodesk Inventor Professional 2020 for Designers is a comprehensive book that introduces the users to Autodesk Inventor 2020, a feature-based 3D parametric solid modeling software. All environments of this solid modelling software are covered in this book with a thorough explanation of commands, options, and their applications to create real-world products. The mechanical engineering industry examples that are used as tutorials and the related additional exercises at the end of each chapter help the users to understand the design techniques used in the industry to design a product. Additionally, the author emphasizes on the solid modelling techniques that will improve the productivity and efficiency of the users. After reading this book, the users will be able to create solid parts, sheet metal parts, assemblies, weldments, drawing views with bill of materials, presentation views to animate the assemblies and apply direct modelling techniques to facilitate rapid design prototyping. Also, the users will learn the editing techniques that are essential for making a successful design. Salient Features: Comprehensive book consisting of 19 chapters organized in a pedagogical sequence. Detailed explanation of all concepts, techniques, commands, and tools of Autodesk Inventor Professional 2020. Tutorial approach to explain the concepts. Step-by-step instructions that guide the users through the learning process. More than 54 real-world mechanical engineering designs as tutorials and projects. Self-Evaluation Test, Review Questions, and Exercises are given at the end of the chapters so that the users can assess their knowledge. Technical support by contacting 'techsupport@cadcim.com'. Table of Contents Chapter 1: Introduction Chapter 2: Drawing Sketches for Solid Models Chapter 3: Adding Constraints and Dimensions to Sketches Chapter 4: Editing, Extruding, and Revolving the Sketches Chapter 5: Other Sketching and Modeling Options Chapter 6: Advanced Modeling Tools-I Chapter 7: Editing Features and Adding Automatic Dimensions to Sketches Chapter 8: Advanced Modeling Tools-II Chapter 9: Assembly Modeling-I Chapter 10: Assembly Modeling-II Chapter 11: Working with Drawing Views-I Chapter 12: Working with Drawing Views-II Chapter 13: Presentation Module Chapter 14: Working with Sheet Metal

Components Chapter 15: Introduction to Stress Analysis Chapter 16: Introduction to Weldments (For free download) Chapter 17: Miscellaneous Tools (For free download) Chapter 18: Working with Special Design Tools For free download) Chapter 19: Introduction to Plastic Mold Design (For free download) Index

Parametric Modeling with Creo Parametric 6.0 SDC Publications

The primary goal of Parametric Modeling with Creo Parametric 7.0 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.

Autodesk Inventor Professional 2020 for Designers, 20th Edition SDC Publications

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.

Presenting Creo Parametric 1.0 Springer

The primary goal of Parametric Modeling with Creo Parametric 3.0 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.

[Advanced SOLIDWORKS 2022 for Designers, 20th Edition](#) John Wiley & Sons

The purpose of this book is to introduce the reader to 3D CAD/CAM modelling using Creo™ Parametric (Creo) software. This concise textbook consists of ten lessons covering the basics in Part and Assembly Modelling, Mould Design, NC Simulation, and Engineering Drawings. Each lesson provides essential knowledge and guides the user through the process of performing a practical exercise or task. The modelling philosophy, implementation of corresponding features, and commands behind each exercise are explained and presented in a step-by-step manner. The material is richly illustrated with screenshots and icons from the software interface to facilitate the learning process. Suitable for beginners and intermediate users, CAD/CAM with Creo Parametric enables the reader to make a quick start in learning how to use complex 3D CAD/CAM software such as Creo in engineering design and manufacturing. The aim is to develop an understanding of the main modelling principles and software tools as a basis for independent learning and solving more complex engineering problems.

CREO PARAMETRIC 7.0 ADVANCED TUTORIAL

SDC Publications

The purpose of Creo Parametric 3.0 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, UDF’s, patterns and family tables), layers, Pro/PROGRAM, and advanced drawing and assembly functions. Creo Parametric 3.0 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.

[Parametric Modeling with Creo Parametric 3.0](#) SDC Publications

Tutorial-based introduction to 3D Modeling with Creo Parametric, including images to be scanned and viewed using an AR mobile app Using a tutorial approach, *Creo Parametric Modeling with Augmented Reality* provides an introduction to the modeling techniques and functionality of Creo Parametric, beginning with an overview of parametric design and Creo’s sketching capabilities and 3D tools; proceeding through design methods and skills related to patterns, dimensions, sections, assemblies, and tolerances and GD&T; and concluding with connecting Creo’s capabilities the more specialized skills of Finite Element Analysis, mechanism animation, and sheet metal design. Each chapter includes highly visual, step-by-step examples that readers can follow to develop their modeling skills. The tutorials can be used on their own or in conjunction with an AR mobile app that allows select images to be viewed as 3D images that can be rotated, scaled, and exploded. The text helps readers to visualize, model, and assess model relationships, history, measurements, and mass properties. Written by a highly qualified professional with experience in both academia and industry, *Creo Parametric Modeling with Augmented Reality* includes information on: Parametric design foundational concepts, sketcher and 3D tools, revolved features and sweeps, patterns, and drawings and dimensions Sections, auxiliary, and detail views, assemblies, assembly drawings, tolerances and GD&T, finite element analysis, and mechanism animations How to use Creo software to interpret and communicate with 3D solid models and define their design intent and constraints How to use current computer aided engineering graphics software, recognize and apply standard graphical principles, and utilize CAD software to create models, drawings, and assemblies With no fluff and many highly helpful visual learning aids, *Creo Parametric Modeling with Augmented Reality* is an essential resource for engineering students learning 3D modeling for the first time, as well as for practicing engineers who need to brush up on their Creo Parametric skills.

[Autodesk Inventor Exercises](#) CAD/CIM Technologies

About the Book: Written by three distinguished authors with ample academic and teaching experience, this textbook, meant for diploma and degree students of Mechanical Engineering as well as those preparing for AMIE examination, incorporates the latest st

[Creo Parametric 9.0 Advanced Tutorial](#) SDC Publications

The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 7.0. 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 where 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.

[Creo Parametric 9.0 Tutorial](#) SDC Publications

The primary goal of Parametric Modeling with Creo Parametric 1.0 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.

[Computer Aided Virtual Manufacturing Using Creo Parametric](#) SDC Publications

The purpose of Creo Parametric 4.0 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, UDF’s, patterns and family tables), layers, Pro/PROGRAM, and advanced drawing and assembly functions. Creo Parametric 4.0 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.

[Parametric Modeling with Creo Parametric 10.0](#) SDC Publications

This book constitutes the refereed proceedings of the 13th IFIP WG 5.1 International Conference on Product Lifecycle Management, PLM 2016, held in Columbia, SC, USA, in July 2016. The 57 revised full papers presented were carefully reviewed and selected from 77 submissions. The papers are organized in the following topical sections: knowledge sharing, re-use and preservation; collaborative development architectures; interoperability and systems integration; lean product development and the role of PLM; PLM and innovation; PLM tools; cloud computing and PLM tools; traceability and performance; building information modeling; big data analytics and business intelligence; information lifecycle management; industry 4.0; metrics, standards and regulation; and product, service and systems.

[Parametric Modeling with Creo Parametric 9.0](#) Creo Parametric 7.0 Advanced Tutorial

This practical resource provides a series of Inventor® exercises covering several topics, including: sketches part models assemblies drawing layouts presentations sheet metal design welding for users with some familiarity with Autodesk® Inventor, or other similar feature-based modelling software such as Solid Works®, CATIA®, Pro/ENGINEER and Creo Parametric, and who want to become proficient. Exercises are set out in a structured way and are suitable for releases of Inventor from versions 7 to 13.

PARAMETRIC MODELING WITH CREO PARAMETRIC 4.0

Independently Published

- Uses concise, individual, step-by-step tutorials
- Covers the most important advanced features, commands, and functions of Creo Parametric
- Explains not only how but also why commands are used
- Contains an ongoing project throughout the book

The purpose of *Creo Parametric 9.0 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 9.0 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.

[Machine Drawing](#) CAD/CIM Technologies

The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 6.0. 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 where 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

Creo Parametric 6.0 Lulu.com

The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 9.0. 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 where 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

e-Design SDC Publications

The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 4.0. 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

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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 where 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.

Advanced Tutorial for Creo Parametric Releases 1.0 & 2.0 SDC Publications

The purpose of *Creo Parametric 6.0 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 6.0 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.

Parametric Modeling with Creo Parametric 7.0 SDC Publications

Designing with Creo Parametric 7.0 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.