

PREFACE

PURPOSE

My decision to write a Pro/ENGINEER textbook was based on the lack of a comprehensive textbook on this popular computer-aided design package. I focused on several objectives and ideas when I started to develop this project:

1. To write a textbook for an introductory course in engineering graphics.
2. To meet the needs of institutions teaching a course on parametric design and constraint-based modeling.
3. To create a book that would serve as a self-paced, independent study guide for the learning of Pro/ENGINEER for those who do not have the opportunity to take a formal course.
4. To incorporate a tutorial approach to the learning of Pro/ENGINEER in conjunction with detailed reference material.
5. To include topics that make the text a suitable supplement for an upper division course in mechanical design.

APPROACHES TO USING THE TEXTBOOK

This textbook is designed to serve as a tutorial, reference, and lecture guide. Chapters start by covering selected topics in moderate detail. Following the reference portion of each chapter are one or more tutorials covering the chapter's objectives and topics. At the end of each chapter are practice problems used to reinforce concepts covered in the chapter and previously in the book.

I had several ideas in mind when developing this approach to the book:

1. Since Pro/ENGINEER is a menu-intensive, computer-aided design application, the most practical pedagogical method to cover Pro/ENGINEER's capabilities (that would be the most beneficial both to students and instructors) would be to approach this book as a tutorial.
2. The book provides detailed reference material. A typical approach to teaching Pro/ENGINEER would be to provide a tutorial exercise followed by a nontutorial practice or practical problem. Usually students can complete the tutorial, but they may run into problems on the practice exercise. One of the problems that Pro/ENGINEER students have is digging back through the tutorial to find the steps for performing specific modeling tasks. The reference portion of each chapter in this text provides step-by-step guides for performing specific Pro/ENGINEER modeling tasks outside of a tutorial environment.
3. A supplemental CD with existing Pro/ENGINEER part and configuration files is available to enhance reference material. Each chapter contains reference guides for performing specific Pro/ENGINEER tasks. When appropriate in the reference guide, Pro/ENGINEER part files have been provided. This serves two purposes. First, the provided model provides a good starting point for instructors lecturing on specific topics of Pro/ENGINEER. Second, a student of Pro/ENGINEER can use a reference guide and part file to practice specific tasks.
4. The book is flexible in the order that topics are covered. Chapters 1 and 2 are primarily reference material. The first in-depth modeling tutorial starts with Chapter 3 (sketching fundamentals), and the first three-dimensional modeling begins in Chapter 4 (extruding features). Using this textbook in a course, an instructor may decide to start with any of Chapters 1, 2, 3, or 4. Additionally, many of the chapters after Chapter 4 can be reordered to meet the needs of an individual instructor.

STUDENTS OF PRO/ENGINEER

One of the objectives of this book is to serve as a stand-alone text for independent learners of Pro/ENGINEER. This book is approached as a tutorial to help meet this objective. Since Pro/ENGINEER is menu intensive, tutorials in this book use numbered steps to guide the selection of menu options. The following is an example of a tutorial step:

STEP 6: **Place Dimensions according to Design Intent.**

Use the Dimension icon to match the dimensioning scheme shown in Figure 4–24. Placement of dimensions on a part should match design intent. With Intent Manager activated (Sketch >> Intent Manager), dimensions and constraints are provided automatically that fully define the section. Pro/ENGINEER does not know what dimensioning scheme will match design intent, though. Due to this, it is usually necessary to change some dimension placements.

MODELING POINT If possible, a good rule of thumb to follow is to avoid modifying the section's dimension values until your dimension placement scheme matches design intent.

The primary menu selection is shown in bold. In this example, you are instructed to use the dimension option (portrayed by the Dimension icon) to create dimensions that match the part's design intent. Following the specific menu selection, when appropriate, is the rationale for the menu selection. In addition, Modeling Points are used throughout the book to highlight specific modeling strategies.

CHAPTERS

The following is a description and rationale for each chapter in the book:

CHAPTER 1 INTRODUCTION TO PARAMETRIC DESIGN This chapter covers the basic principles behind parametric modeling, parametric design, and constraint-based modeling. Discussed is how Pro/ENGINEER can be used to capture design intent, and how it can be an integral component within a concurrent engineering environment.

CHAPTER 2 Pro/ENGINEER's USER INTERFACE This chapter covers basic principles behind Pro/ENGINEER's interface and menu structure. The purpose is to serve as a guide and reference for later modeling activities. A tutorial is provided to reinforce the chapter's objectives.

CHAPTER 3 CREATING A SKETCH Parametric modeling packages such as Pro/ENGINEER rely upon a sketching environment to create most features. This chapter covers the fundamentals behind sketching in Pro/ENGINEER's sketcher mode. Two tutorials are provided along with practice problems.

CHAPTER 4 EXTRUDING, MODIFYING, AND REDEFINING FEATURES Chapter 4 is the first chapter covering Pro/ENGINEER's solid modeling capabilities. Pro/ENGINEER's protrusion and cut commands are introduced, and the extrude option is covered in detail. In addition, modification and datum plane options are introduced. Two tutorials are provided.

CHAPTER 5 FEATURE CONSTRUCTION TOOLS While the protrusion and cut commands are Pro/ENGINEER's basic tools for creating features, this chapter covers additional feature creation tools. Covered in detail are the hole, round, rib, and chamfer commands; creating draft surfaces; shelling a part; cosmetic features; and creating linear patterns. Two tutorials are provided.

CHAPTER 6 REVOLVED FEATURES Many Pro/ENGINEER features are created by revolving around a center axis. Examples include the revolve option found under the protrusion and cut commands, the sketched hole option, the shaft command, the flange command, and the neck command. These options, along with creating rotational patterns and datum axes, are covered in this chapter. Two tutorials are provided.

CHAPTER 7 FEATURE MANIPULATION TOOLS Pro/ENGINEER provides tools for manipulating existing features. Manipulation tools covered include the group option, copying features, user-defined features, creating relations, family tables, and cross sections. In addition, using the model tree to manipulate features is covered.

CHAPTER 8 CREATING A Pro/ENGINEER DRAWING Since Pro/ENGINEER is primarily a modeling and design application, the creation of engineering drawings is considered a downstream task. Despite this, there is a need to cover the capabilities of Pro/ENGINEER's drawing mode. This chapter covers the creation of general and projection views. Other topics covered include sheet formatting, annotating drawings, and creating draft entities. Two tutorials are provided, one of which covers the creation of geometric and dimensional tolerances.

CHAPTER 9 SECTIONS AND ADVANCED DRAWING VIEWS Due to the length and depth of Chapter 8, the creation of section and auxiliary views is covered in a separate chapter.

CHAPTER 10 SWEEP AND BLENDED FEATURES The protrusion and cut commands have options for creating extruded, revolved, swept, and blended features. Extruded and revolved features are covered in previous chapters. This chapter covers the fundamentals behind creating sweeps and blends. Three tutorials are provided.

CHAPTER 11 ADVANCED MODELING TECHNIQUES The protrusion and cut commands have options for creating advanced features. Covered in this chapter are helical sweeps, variable section sweeps, and swept blends. Three tutorials are provided.

CHAPTER 12 ASSEMBLY MODELING This chapter covers the basics of Pro/ENGINEER's assembly mode. Other topics covered include the creation of assembly drawings using report mode, the control of assemblies through layout mode, top-down assembly design, and mechanism design. Three tutorials are provided.

CHAPTER 13 SURFACE MODELING This chapter covers the basics behind creating surface features within Pro/ENGINEER. Two tutorials are provided.

www.mhhe.com Please visit our web page. Ancillary materials are available for reading and download. For instructors, solutions to end-of-chapter Questions and Discussion are available. Also available are solutions to chapter problems and additional problems.

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