SAVE 25%

to Version 15

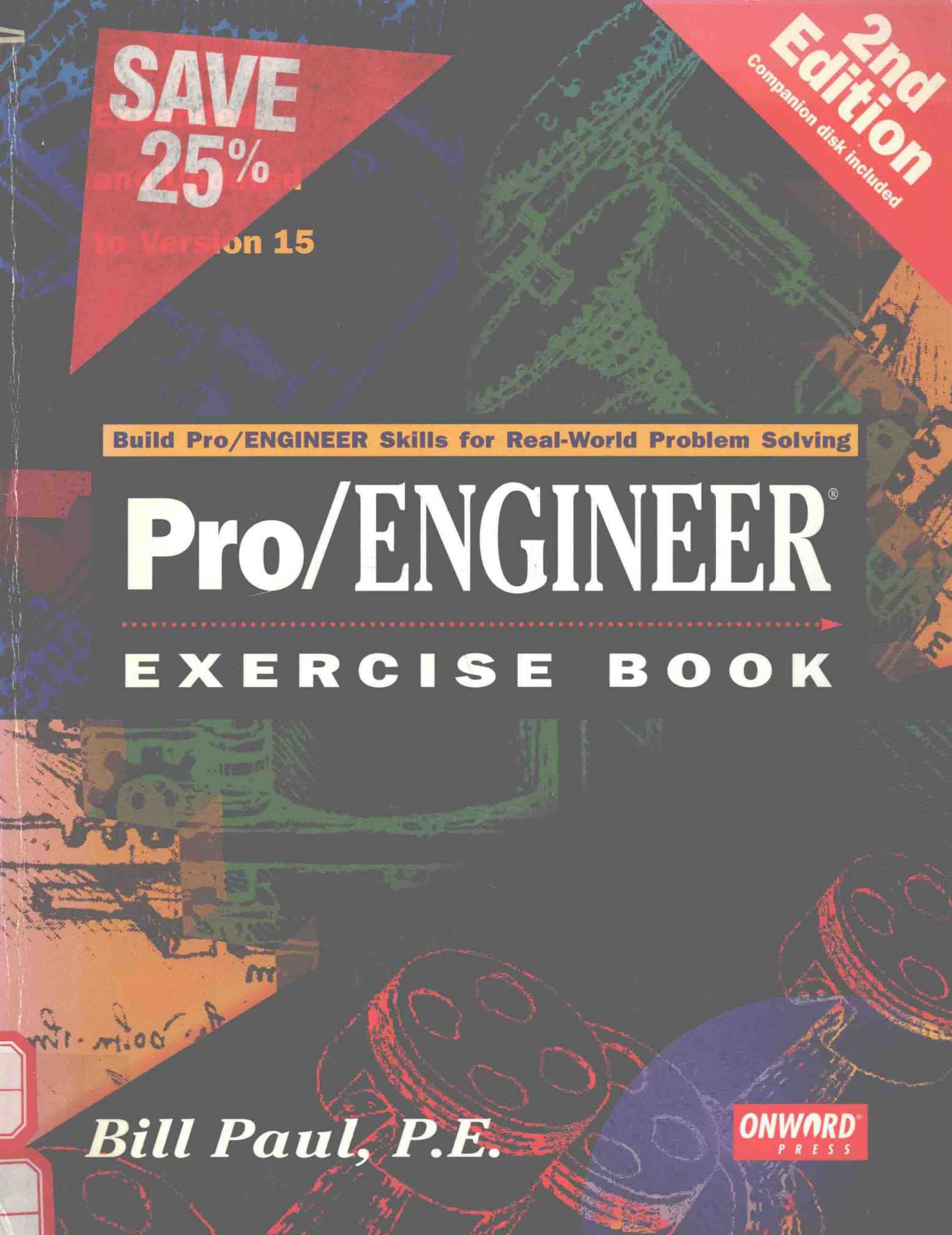
2nd Edition

Companion disk included

Build Pro/ENGINEER Skills for Real-World Problem Solving

Pro/ENGINEER®

EXERCISE BOOK



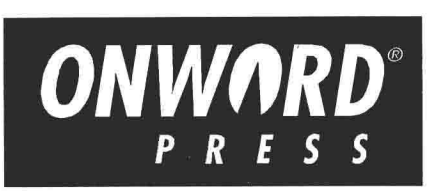
Bill Paul, P.E.

ONWORD PRESS

The Pro/ENGINEER® Exercise Book

Second Edition

Bill Paul, P.E.



The Pro/ENGINEER® Exercise Book

Second Edition

By Bill Paul, P.E.

Published by
OnWord Press
2530 Camino Entrada
Santa Fe, NM 87505-4835 USA

SAN 694-0269
Second Edition, 1995

10 9 8 7 6 5 4 3 2 1

Printed in the United States of America

Cataloging-in-Publication Data

Bill Paul, P.E.
The Pro/ENGINEER Exercise Book
Second Edition

Includes index.

1. Pro/ENGINEER (computer software) 2. Mechanical computer-aided design I. Title

ISBN 1-56690-083-2

OnWord Press

OnWord Press is dedicated to the fine art of professional documentation. In addition to those listed below, other members who contributed to the production and distribution of this book include Joe Adams, James Bridge, Roxsan Meyer, Lisa Meyrick, Jean Nichols, and Ezekiel Olguin.

Dan Raker, President
Gary Lange, Vice President
David Talbott, Acquisitions Editor
Margaret Burns, Project Editor
Carol Leyba, Production Manager
Janet Leigh Dick, Marketing Director
Lynne Egensteiner, Cover Designer

Conventions Used in this Book

The following conventions were used throughout this book:

Menu names appear in all capital letters, e.g., MODIFY SCHEME.

Commands and submenu selections appear in italic type, e.g., *Reroute* command.

Companion Disk Installation

The bonus companion disk is attached to the inside back cover of this book. The disk contains the exercises for you to use as you go through the workbook. The disk is a high-density, 1.44Mb DOS format disk. There are two files on the disk: *proe_x.exe* and *proe_x.z*. The first file contains self-extracting DOS files. The second file contains a compressed UNIX file, *proe_x.tar*.

Installation on a Windows NT Workstation

To install the disk,

1. Insert the disk into your floppy drive.
2. Using File Manager, create a directory on your hard drive.
3. Copy *proe_x.exe* into the new directory.
4. Using File Manager, execute the file.

Installation on a UNIX Workstation that Can Read a DOS Disk

1. Insert the disk into your floppy drive.
2. Mount the floppy drive if it is not already mounted.
3. Create a directory in your home directory.
4. Using a "dos2unix" transfer method, copy the *proe_x.z* to the new directory.
5. Using the uncompress command, uncompress this file to a tar file. (Note: The extension ".tar" does not appear. It was deleted so this file could be put on the DOS disk.)
6. Using the tar command, extract the workbook files.

Installation on a UNIX Workstation that Cannot Read a DOS Disk

If you have a workstation that cannot read DOS disks, but is connected to a network with a PC, you can do the following.

1. Make a temporary directory on the PC.
2. Insert the disk into your floppy drive on the PC.
3. Copy the *proe_x.exe* file into the temporary directory on the PC.
4. Execute the file, creating the workbook files.
5. Delete the *proe_x.exe* file from the temporary directory.
6. Transfer the files to the UNIX workstation via a TCP/IP connection using the bin option.
7. Delete all the files from the temporary directory on the PC.

Consult your manual or system administrator if you are unsure of how to perform any of these operations.

Trademarks

Warning and Disclaimer

About the Author

Bill Paul, P.E., is Manager of MCAD/MCAE training for the Texas Instruments Learning Institute in Dallas. He administers, develops, and teaches Pro/ENGINEER, Pro/PDM, MECHANICA software and other related courses. He has a B.S. and M.S. from Mississippi State University. Bill has over 30 years' experience in mechanical design, analysis, training, and MCAD. He is Chair of the Pro/USER Modeling Technical Committee and a member of the editorial board of *Pro/E: The Magazine*. Bill can be reached by email: wrpa@msg.ti.com or by telephone: (214) 917-1916.

Acknowledgments

We are grateful for the assistance of many who have helped in the preparation of this book. The HRD staff at the Texas Instruments Learning Institute was instrumental in the development of the material on which this book is based. In particular, our thanks go to Dwight Saffel of Saffel Consulting for his invaluable help in preparation of this work. Dwight can be reached at DSafCon@aol.com. We also wish to thank David Bigelow and Peter Barbee for their technical reviews of the manuscript.

Pro/ENGINEER Exercise Book

Table of Contents

4. Lower manufacturing costs
5. Lower testing costs
6. Reduced service costs
7. Enhanced competitiveness
8. Improved profit margins

There really is no choice; we need to improve in these areas or we will lose out to the competitors who choose to improve their design process.

The concept of concurrent engineering is not new. A look at the history of mechanical concurrent engineering shows the evolution of the process. In the 1960s, design was a manual process. There was informal consultation between the designer, drafters, and manufacturers. At the time it was not known as *concurrent engineering*, although the results were similar.

During the 1980s, wireframe CAD was implemented. This created a semiautomated drafting-based design environment. Instead of concurrent design, a formal review procedure was installed between designer and drafter. We are currently in this phase of design automation. This process has created the use of multiple databases in design, analysis, and manufacturing (Figure 1-1). The use of multiple databases has created many problems during the life of the product. The problems can include the following:

1. The drawing is the master design, resulting in 50% to 90% of all significant drawings having missing or incorrect dimensions.
2. The multidatabase product definition environment is the basis for reduced product quality, resulting in a high number of engineering changes.
3. Concurrent engineering is an after-the-fact design review process, resulting in four or more drawing iterations per part.
4. Analysis and manufacturing automation is neutralized by the large amount of data preparation required.

Mechanical Design Process Automation Objectives

To improve these statistics, a mechanical design process automation objective can be established. The objective is to create an engineering-based design environment, using a single product definition database for the design, drafting, analysis, and manufacturing phases. This allows for concurrent data flow, along with the following improvements:

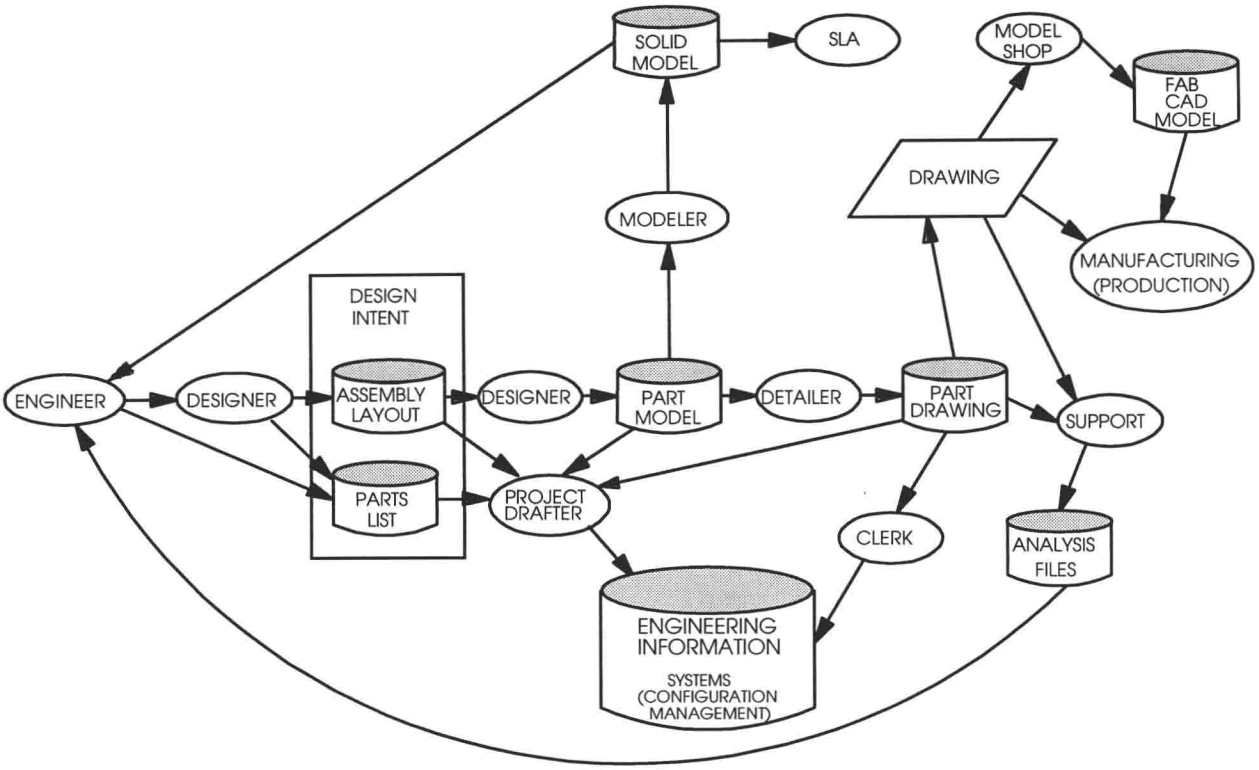


Figure 1-1. Mechanical design process. Underlying problem: multiple design representations.

1. The assembly model database is the master design, which results in approximately 0% of engineering-released drawings having missing or incorrect dimensions.
2. A single product definition database is the basis for all analysis and manufacturing functions, resulting in a 50% to 70% reduction in engineering changes.
3. Concurrent engineering is a team design environment, not a review process, resulting in one drawing cycle per part.
4. Analysis and manufacturing use the design database directly for all process planning operations.

Pro/ENGINEER and Mechanical Design Process Automation Requirements

Pro/ENGINEER fulfills the following critical requirements for an integrated product development outline:

1. Efficient computer hardware/communications network.
2. A design process that creates a product database using the concurrent engineering philosophy.
3. Acceptable CAE/CAD/CAM functionality for each of the critical functional tasks.
4. A design database that is complete, unambiguous, and directly accessible for all engineering, analysis, and manufacturing operations.

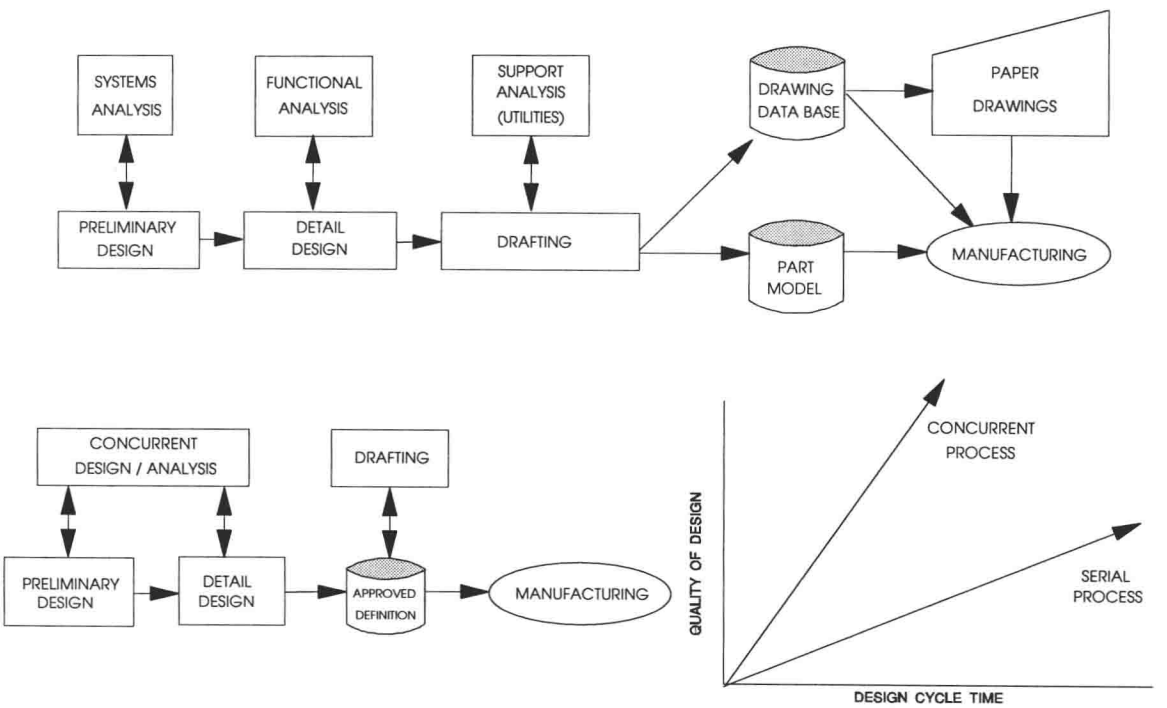


Figure 1-2. Process migration.

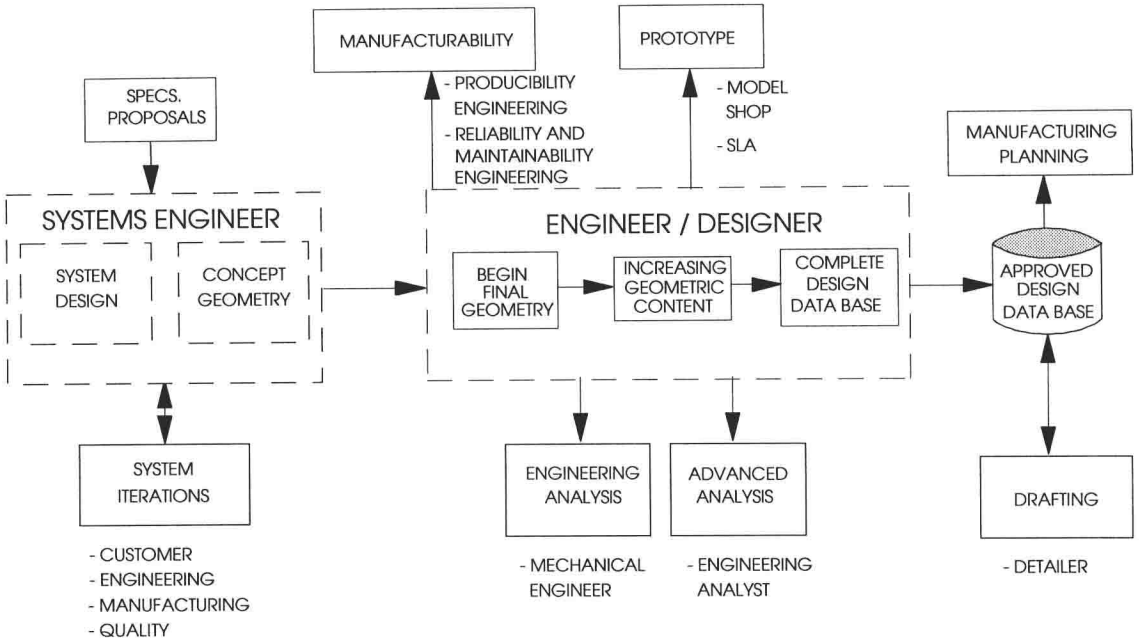


Figure 1-3. Product database management system. Concurrent engineering workstation integration requirements.

5. A fully associative design/drawing database.
6. A full range of engineering analysis capability, including geometric (weight, center of gravity), finite element model, and tolerance.
7. A product data management system to manage the flow of data within Pro/ENGINEER for the concurrent process.

In the following chapters, you will learn how to use Pro/ENGINEER to accomplish the concurrent engineering process.

CHAPTER 2
Inside Parametric's Engine

To understand how Pro/ENGINEER constructs a part, you must have a good understanding of how Pro/ENGINEER creates each feature. You also need to be aware of the techniques used in creating these features.

In this chapter we cover the following topics:

- What a feature is
- How feature information and parts are stored
- The Sketcher environment
- Rules and assumptions for regenerating section sketches
- Unsuccessful regeneration of a sketch
- Setting Sketcher accuracy
- Special geometry in Sketcher

What Is a Feature?

Pro/ENGINEER is a *feature-based* solid modeler. This means that every part is made up of a series of features. A *feature* in Pro/ENGINEER is any hole, shaft, round, chamfer, slot, cut, protrusion, neck flange, rib, shell, pipe, or tweak such as a draft. *Datums* (planes, axes, curves, points, coordinate systems, graphs) are features, but are used as references in geometry construction and placement.

Each feature changes the appearance of and adds information to the model. When a feature is created, all the information that defines the feature is recorded and filed in memory. It can be changed or modified as the design progresses.

Take for instance a "D-Shaped" cut created in a simple block as in Figure 2-1. To create this cut, you would select the following menu items: *Feature, Create,*

Solid/Cut, Extrude/Solid/Done, One Side/Done. You would then select a Sketching plane, set the Direction arrow, and select a Sketcher Reference plane (Top, Bottom, Left, or Right). After this, you would sketch the cut, dimension, and regenerate it. To complete the process, you would modify the dimensions, select *Regenerate, Done, Okay,* go to the DEFAULT menu to check the direction of creation, and then select *ThruAll/Done, Done.* In this process much information is given to Pro/ENGINEER. This information will remain with the cut feature forever unless the cut is deleted.

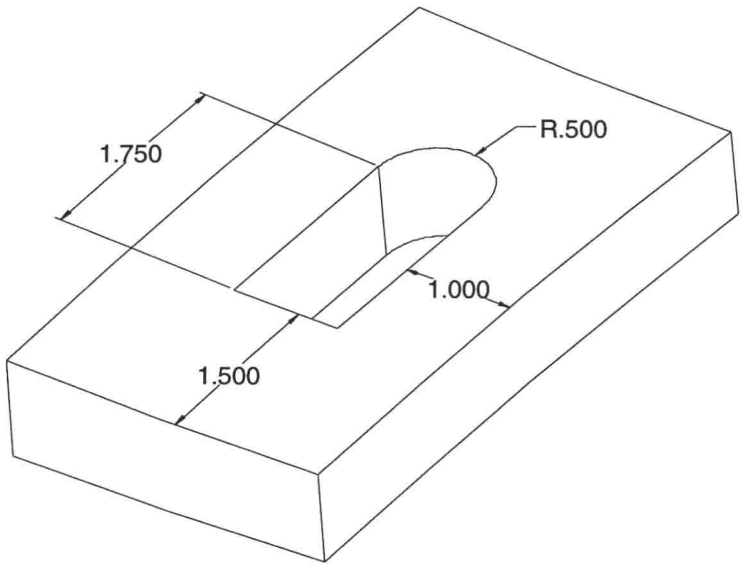


Figure 2-1. A D-shaped cut.

In review, we just created a cut *Thru All* the part extruded in one direction from the sketching plane.

How Feature Information and Parts Are Stored

Feature Information

Now that we have created a cut feature, let's see how feature information is stored within Pro/ENGINEER. Figure 2-2 shows a listing of the information stored for this D-shaped cut. Each of these items will be discussed in detail at a later time.

```
PART NAME = D-CUT
FEATURE NUMBER   2
INTERNAL FEATURE ID   52
PARENTS = 33(#1)
TYPE = CUT
FORM = EXTRUDED
SECTION NAME = S2D0010
DEPTH = FROM SKETCH TO ALL
FEATURE'S DIMENSIONS:
d3 = 1.750
d4 = .500R
d6 = 1.000
```

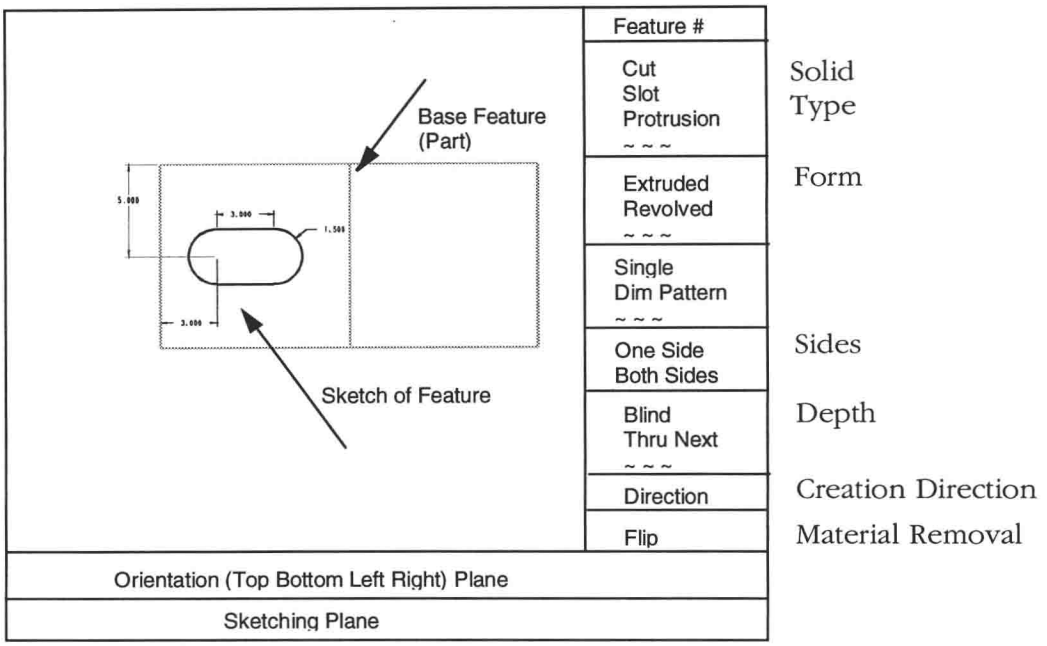
Figure 2-2. Feature Information

The feature form, Figure 2-3, helps us visualize graphically what this information is. For example:

CUT is a TYPE of solid feature.

EXTRUDED is the FORM used to create this feature.

DEPTH is an ATTRIBUTE of the feature.



Sketching and Orientation Planes

Figure 2-3. The Feature Form.

It is important to remember that a feature is not just the sketch but is all the information in all the fields, including the sketch itself.

Part Information

A Pro/ENGINEER part is a collection of one or more features. Parts are made by sequentially creating one feature after another. This construction sequence gives rise to the Feature Number and the Internal Feature ID.

Parts are stored as a series of feature forms, not unlike pages in a book. To bring a model into memory, Pro/ENGINEER simply pulls up feature number one, regenerates it, pulls up the next feature, places it on the first, regenerates it, and continues this process until there are no more features to be retrieved.

Exercise 2-1: Using the feature information just covered, circle the information on the feature form, Figure 2-5, that applies to the cut illustrated in Figure 2-4.

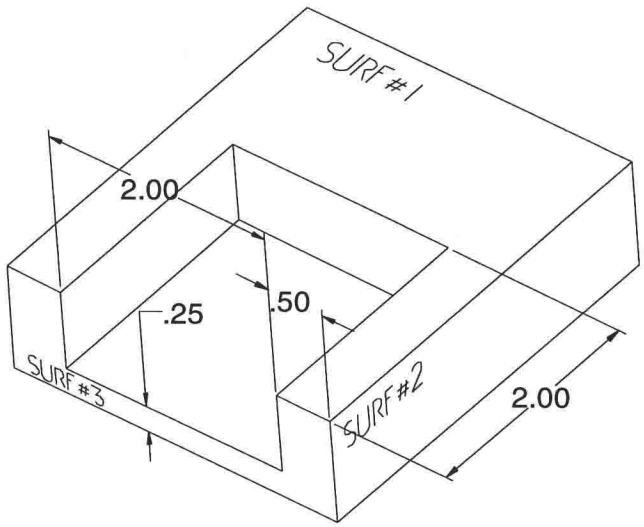
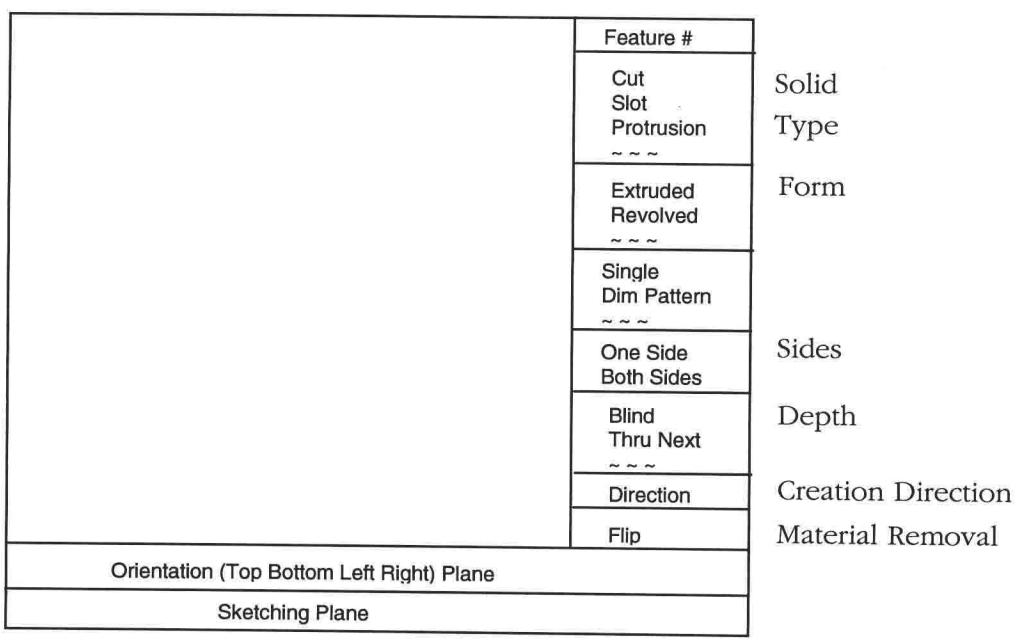


Figure 2-4.



Sketching and Orientation Planes

Figure 2-5.