Pro/ENGINEER® Wildfire™

Louis Gary Lamit
De Anza College

With technical assistance provided by James Gee

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About the Author

Louis Gary Lamit is currently an instructor at De Anza College in Cupertino, California, where he teaches computer-aided design.

Mr. Lamit has worked as a drafter, designer, numerical control (NC) programmer, technical illustrator, and engineer in the automotive, aircraft, and piping industries. A majority of his work experience is in the area of mechanical and piping design. He started as a drafter in Detroit (as a job shopper) in the automobile industry, doing tooling, dies, jigs and fixture layout, and detailing at Koltanbar Engineering, Tool Engineering, Time Engineering, and Premier Engineering for Chrysler, Ford, AMC, and Fisher Body. Mr. Lamit has worked at Remington Arms and Pratt & Whitney Aircraft as a designer, and at Boeing Aircraft and Kollmorgan Optics as an NC programmer and aircraft engineer. He also owns and operates his own consulting firm, and has been involved with advertising and patent illustrating.

Mr. Lamit received a BS degree from Western Michigan University in 1970 and did Masters' work at Wayne State University and Michigan State University. He has also done graduate work at the University of California at Berkeley and holds an NC programming certificate from Boeing Aircraft.

Since leaving industry, Mr. Lamit has taught at all levels (Melby Junior High School, Warren, Michigan; Carroll County Vocational Technical School, Carrollton, Georgia; Heald Engineering College, San Francisco, California; Cogswell Polytechnical College, San Francisco and Cupertino, California; Mission College, Santa Clara, California; Santa Rosa Junior College, Santa Rosa, California; Northern Kentucky University, Highland Heights, Kentucky; and De Anza College, Cupertino, California).

Textbooks, workbooks, tutorials, and articles by Louis Gary Lamit:

- ***Industrial Model Building*** (1981)
  Engineering Model Associates, Inc.

- ***Piping Drafting and Design*** (1981)
- ***Piping Drafting and Design Workbook*** (1981)
- ***Descriptive Geometry*** (1983)
- ***Descriptive Geometry Workbook*** (1983)
- ***Pipe Fitting and Piping Handbook*** (1984)
  Prentice-Hall

- ***Drafting for Electronics*** (3rd edition, 1998)
- ***Drafting for Electronics Workbook*** (2nd edition, 1992)
- ***CADD*** (1987)
  Charles Merrill (Macmillan-Prentice-Hall)

- ***Technical Drawing and Design*** (1994)
- ***Technical Drawing and Design Worksheets and Problem Sheets*** (1994)
- ***Principles of Engineering Drawing*** (1994)
- ***Fundamentals of Engineering Graphics and Design*** (1997)
- ***Engineering Graphics and Design with Graphical Analysis*** (1997)
- ***Engineering Graphics and Design Worksheets and Problem Sheets*** (1997)
- ***Basic Pro/ENGINEER® in 20 Lessons*** (Revision 18) (1998)
- ***Basic Pro/ENGINEER® (with references to PT/Modeler)*** (Revision 19 and PT/Modeler) (1999)
- ***Pro/ENGINEER 2000i®*** (Revision 2000i) (1999)
- ***Pro/E 2000i²® (includes Pro/NC and Pro/SHEETMETAL)*** (Revision 2000i²) (2000)
  Brooks/Cole

- ***IX Design*** (2001) CD text
  ImpactXoft

- ***Pro/ENGINEER® Wildfire™*** (Revision Wildfire) (2003)
  Brooks/Cole

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Assembly Mode

Drawing Mode

Preface

Pro/ENGINEER is one of the most widely used CAD/CAM software programs in the world today. This book introduces you to the basics of the program and enables you to build on these basic commands to expand your knowledge beyond the scope of the book.

The book does not attempt to cover all available features, but rather to provide an introduction to the software, make you reasonably proficient in its use, and establish a firm basis for exploring and growing with the program as you use it in your career or classroom.

The book covers **Part Mode: Lessons 1-14, Assembly Mode: Lessons 15-16,** and **Drawing Mode: Lessons 17-20**. The basic premise of this book is that the more parts, assemblies, and drawings you create using Pro/ENGINEER Wildfire (Pro/E Wildfire), the better you learn the software. With this in mind, each lesson introduces a new set of commands, building on previous lessons. The parts created in Part Mode are used to create assemblies in Assembly Mode and to generate drawings in Drawing Mode This procedure allows you to work with actual completed parts, assemblies, and drawings in a short-lesson format, instead of building large complex projects where basic commands may be overshadowed and lost in a complicated process.

Every lesson introduces a new set of commands and concepts that are applied to a *part*, an *assembly*, or a *drawing*, depending on where in the book you are working.

Lessons involve creating a new part, an assembly, or a drawing, using a set of Pro/E commands that walk you through the process step by step. Each lesson starts with a list of objectives and ends with a lesson project. The lesson project consists of a part, assembly, or drawing that incorporates the lesson's new material and uses and expands on previously introduced material from other lessons.

COAch for Pro/ENGINEER®, has been referenced in the book's figures with the authorized use of illustrations. COAch is one of the best ways available for teaching and learning CAD/CAM software. ***A sampler CD can be requested from CADTRAIN®.*** This CD contains a sample of products offered in *CADTRAIN's COAch for Pro/ENGINEER®*.

For a small handling and shipping fee, a CD (with all Pro/E files used in this text) is available from the author for ***instructors*** who adopt this text.

A 100-page booklet on Pro/SHEETMETAL® (formerly Lesson 23) is available and includes a CD. To order, please go to the WEB site listed below and follow the instructions.

If you wish to contact the author concerning orders, questions, changes, additions, suggestions, comments, and so on, please send an email to one of the following:

Louis Gary Lamit & Lamit and Associates

Web Site: www.cad-resources.com

Email: lgl@cad-resources.com

Dedication

This book is dedicated to my daughter Corina and her husband Michael.

Om Mani Padme Hum

Acknowledgments

I want to thank the following people and organizations for the support and materials granted the author:

Ken Page, **Nick Maly** and **Larry Fire**	Parametric Technology Corporation
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In addition, I would like to thank the following for assistance in checking and editing the text: Gary Mahany, Tom Modrzejowski, Tracey Jones, Erika M. Shapiro and Dean Collins**.**

Resources

A variety of information, books, online products, and job opportunities (www.pejn.com) are available. We have listed some of the more useful and important resources. You can also search on the Web with **PTC, Pro/E,** and **Pro/ENGINEER** as keywords.

Parametric Technology Corporation www.ptc.com

Pro/ENGINEER Job Network www.pejn.com

A variety of services are available over the Internet: **Employers:** Announce your job openings on the Web site. Job seekers will fax, mail, or email their resumes directly to you. **Job Seekers:** Look over the job listings on the Web site, free of charge. Fax, mail, or email your resume directly.

CADTRAIN's COAch® for Pro/ENGINEER www.cadtrain.com

COAch® is a computer-based training (CBT) product designed to provide a comprehensive and affordable training program for Pro/ENGINEER users in their actual CAD environment. This self-paced onscreen training tool enables engineers, designers, drafters, and NC programmers to customize their training experience by following the learning sequence best suited to their individual needs.

FroTime® online tutorials for Pro/ENGINEER www.frotime.com

FroTime® is the leader in providing Online Web-Based Pro/ENGINEER training tutorials on a variety of subjects, including: Modeling, Detailing, Surfacing, Sheet Metal, Data Management, etc. In addition to training, FroTime® also provides several custom software applications to maximize efficiency for Pro/ENGINEER users.

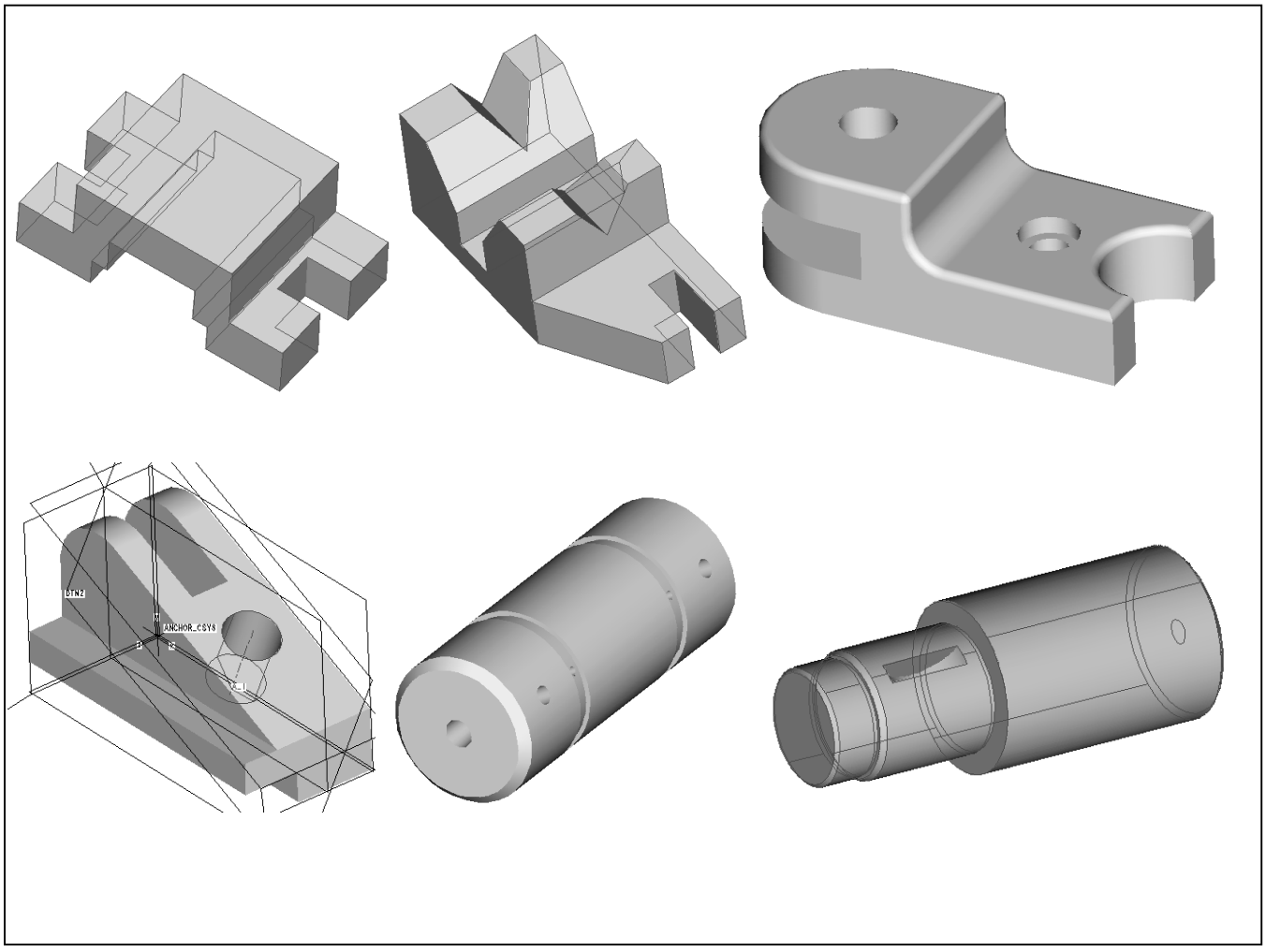
Lamit and Associates (CAD-RESOURCES) tutorials for Pro/ENGINEER

www.cad-resources.com lgl@cad-resources.com

Booklets are now available on **Pro/SHEETMETAL** (100 pages, with CD). To order please go to the Web site and follow the ordering instructions.

Pro/ENGINEER Modes

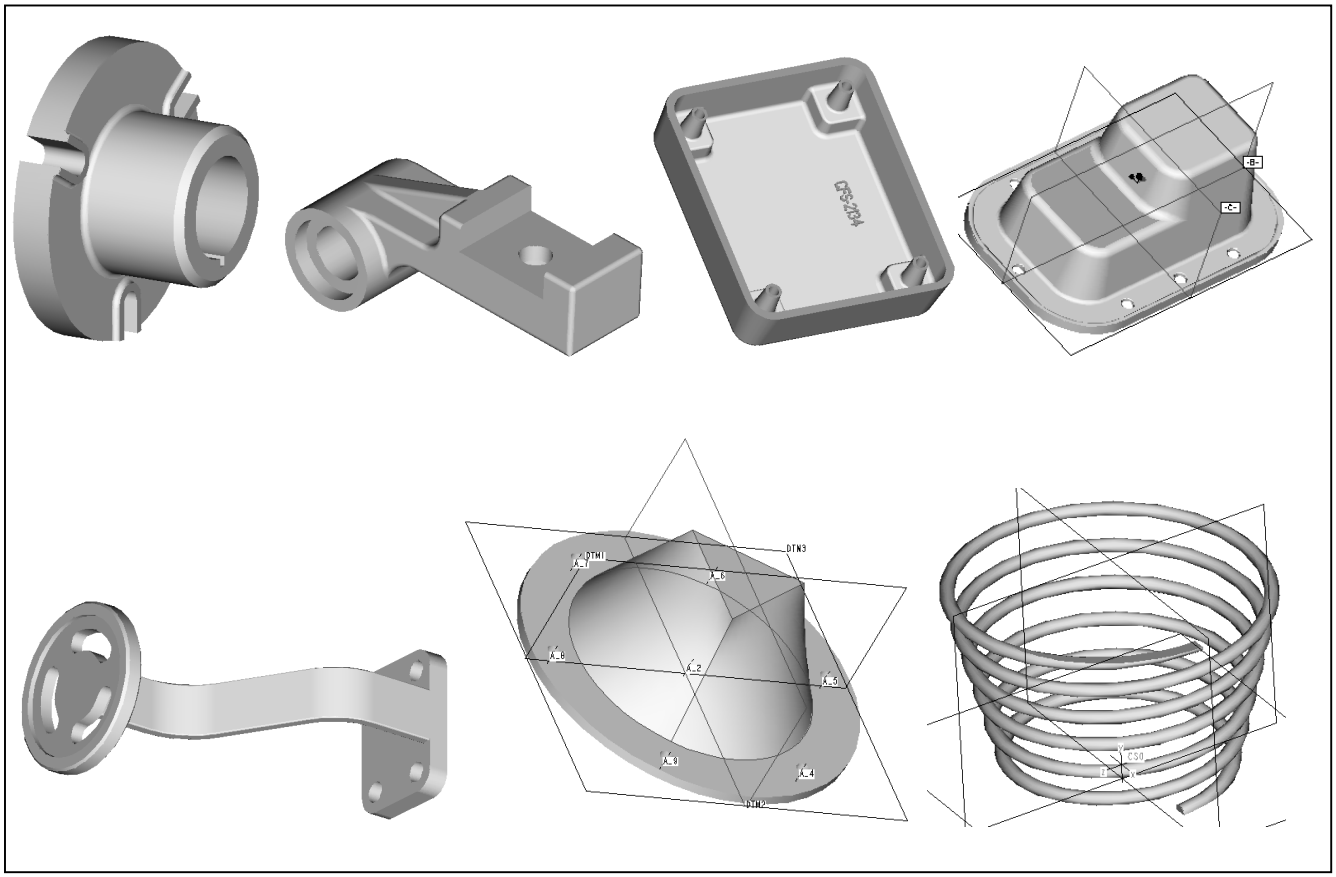
Part Mode



Lesson Parts 2 through 7

PARTS

The *design intent* of a feature, a part, or an assembly (and even a drawing) should be established before any work is done using Pro/ENGINEER. Skipping this step in the design process is a recipe for disaster.



Lesson Parts 8 through 14

In industry, there are thousands of stories of how a designer created a *graphically correct* part (or assembly) that *"looked"* visually precise. Upon closer examination, the part (or assembly) had too many or too few datum planes, parent-child relationships that were glaring examples of the designer's incorrect use of Pro/E, and massive feature failures that resulted when minor ECOs were introduced after the original design was complete. Pro/E is only as good as the drafter, designer, or engineer using it.

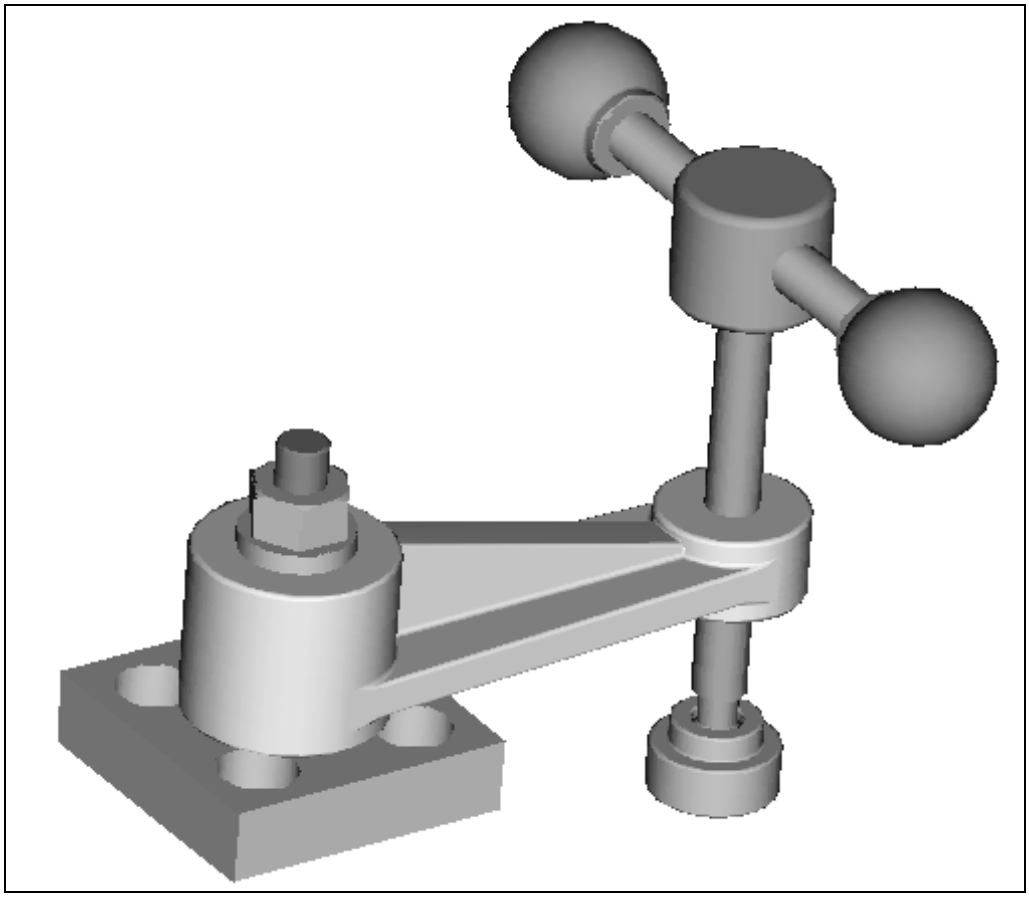
Without proper process planning, organization, and well-defined design intent, the part model is useless. In most cases, such poor design habits result in the parts being remodeled, because it would take more time to reorder, modify, redefine, and reroute. In fact, most poor designs can't be fixed.

The **design intent** of a project must be understood before modeling geometry is started. Sketch and analyze your part before modeling.

The *dimensioning scheme* will establish the dimensions that are critical for the design: What dimensions on the part might be modified during an *ECO*? What dimensions are required for *manufacturing* the part economically and to the correct *tolerances*? Are there any dimensional *relationships* that must be established and maintained? Will the part be a member of a *family of similar parts?* How does the part relate to other *parts in the assembly?*

Assembly Mode

Lesson 15 Assemblies
Lesson 16 Exploded Assemblies



Swing Clamp Assembly

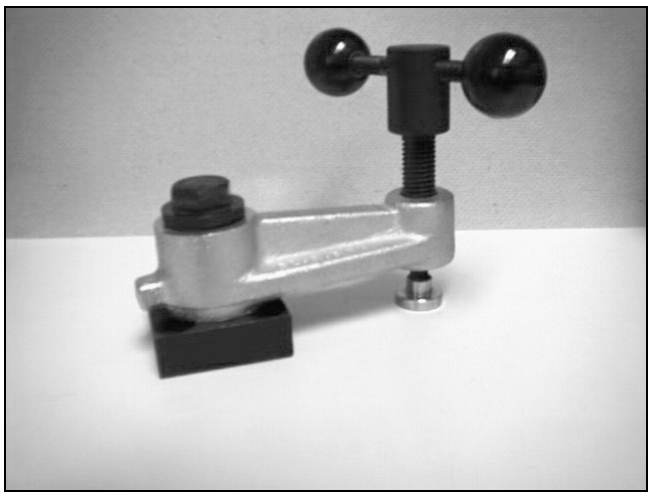
ASSEMBLIES

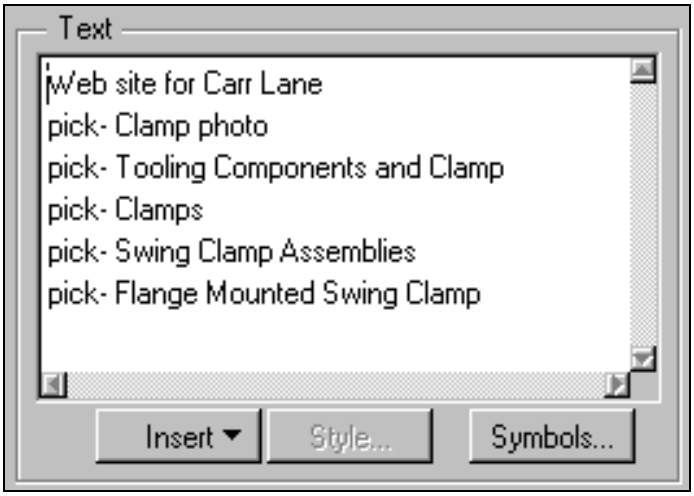
The **Assembly mode** allows you to place together component parts and subassemblies to form assemblies. Assemblies can then be modified, reoriented, analyzed, and documented. Assembly mode is used for the following functions:

- Placing components into assemblies (*bottom-up* assembly design)
- Altering the display settings for individual components
- Designing in Assembly mode (*top-down* assembly design)
- Part modification, including feature construction
- Analysis of assemblies
- Assemble component parts and subassemblies to form assemblies
- Delete or replace assembly components
- Modify assembly placement offsets, and create and modify assembly datum planes, coordinate systems, and sectional views
- Modify parts directly in Assembly mode
- Get assembly engineering information, perform viewing and layer operations
- Exploding views of assemblies

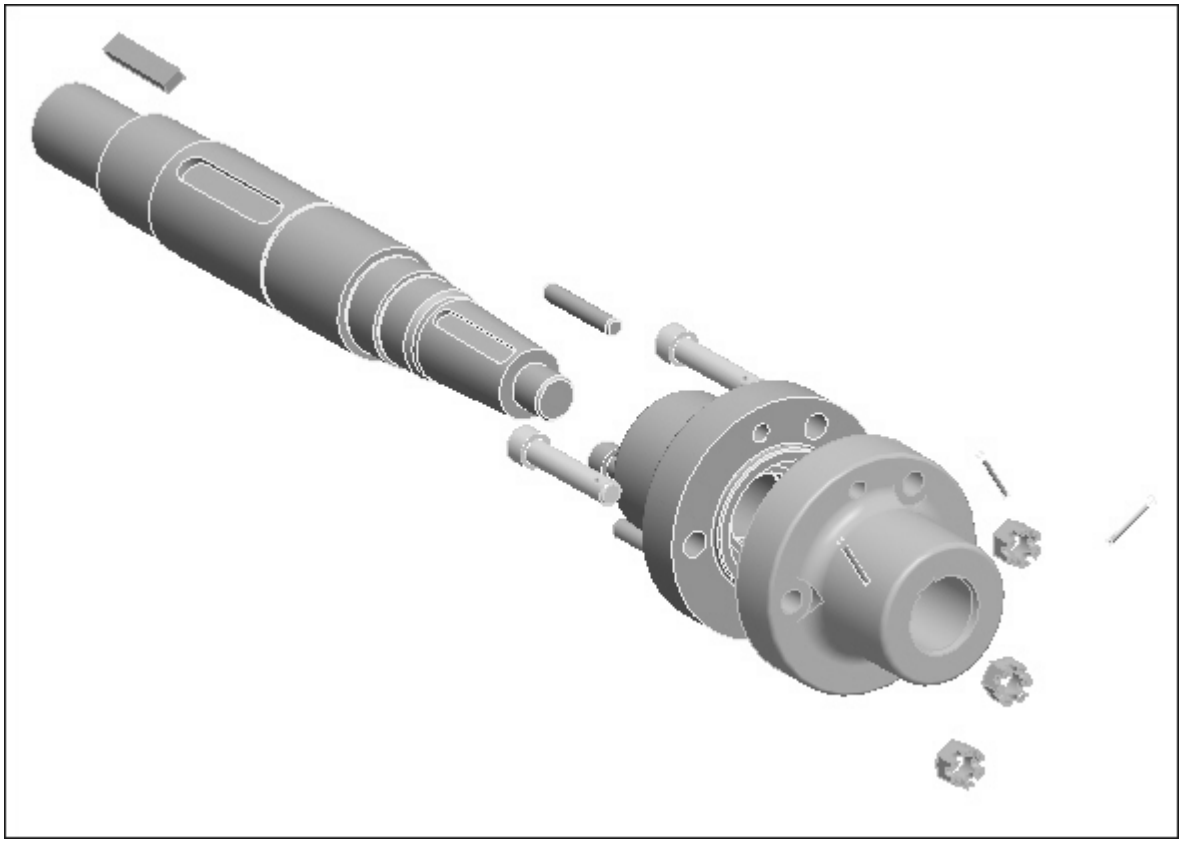
The process of creating an assembly is accomplished by adding models (parts/subassemblies) to a base component (parent part/subassembly) using a variety of constraints. A placement constraint specifies the relative position of a pair of surfaces on two components. The Mate, Align, and Insert commands and their variations are used to accomplish this task.

For approximately $60.00 US, the Swing Clamp can be purchased from CARRLANE at www.carrlane.com.





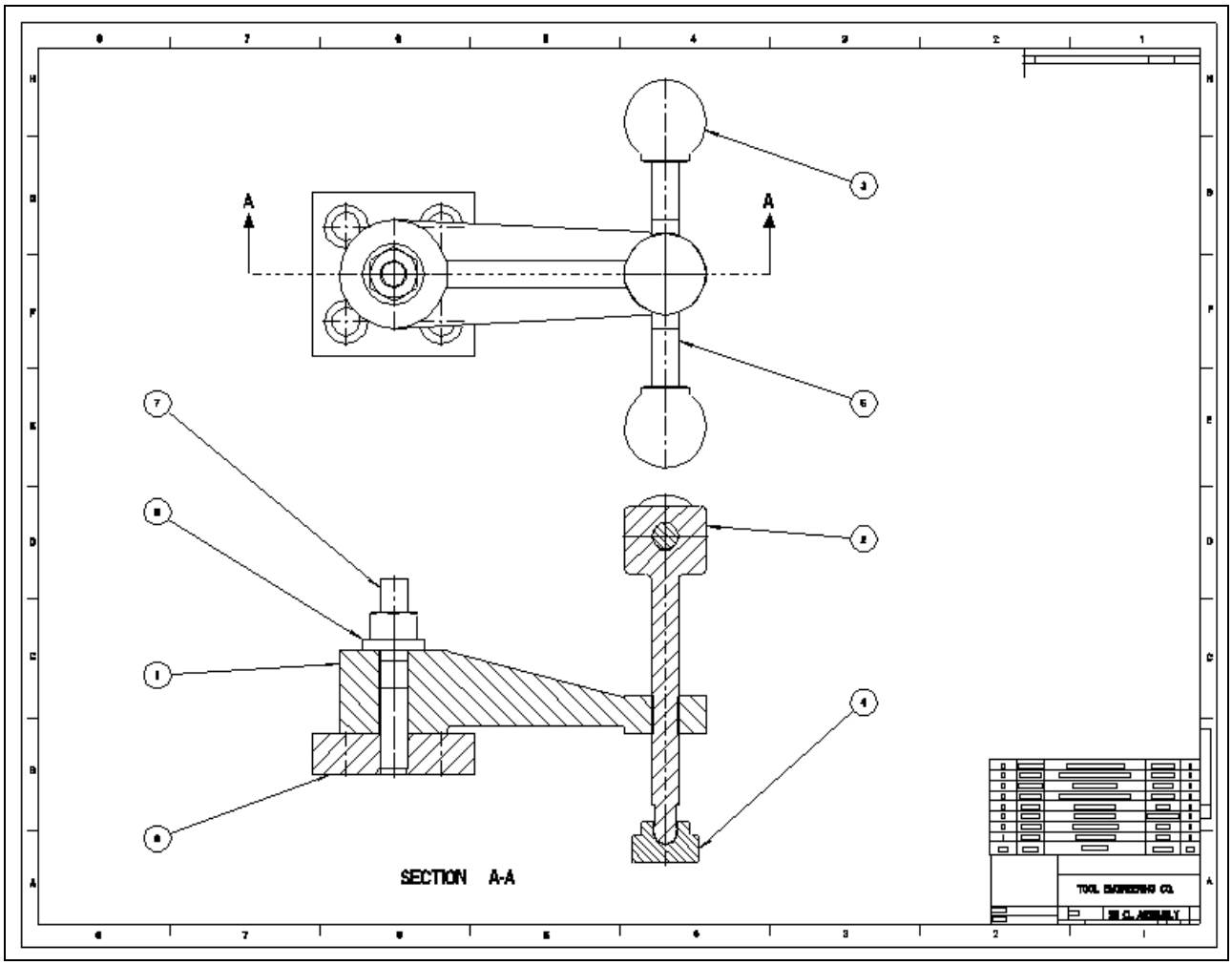
CARRLANE Swing Clamp http://www.carrlane.com/oncatfrm.html



Exploded Coupling Assembly

Drawing Mode

Lesson 17 Formats, Title Blocks, and Views
Lesson 18 Detailing
Lesson 19 Sections and Auxiliary Views
Lesson 20 Assembly Drawings



Assembly Drawing

DRAWINGS

The drawing functionality in Pro/E is used to create annotated drawings of parts and assemblies and has a variety of options, including:

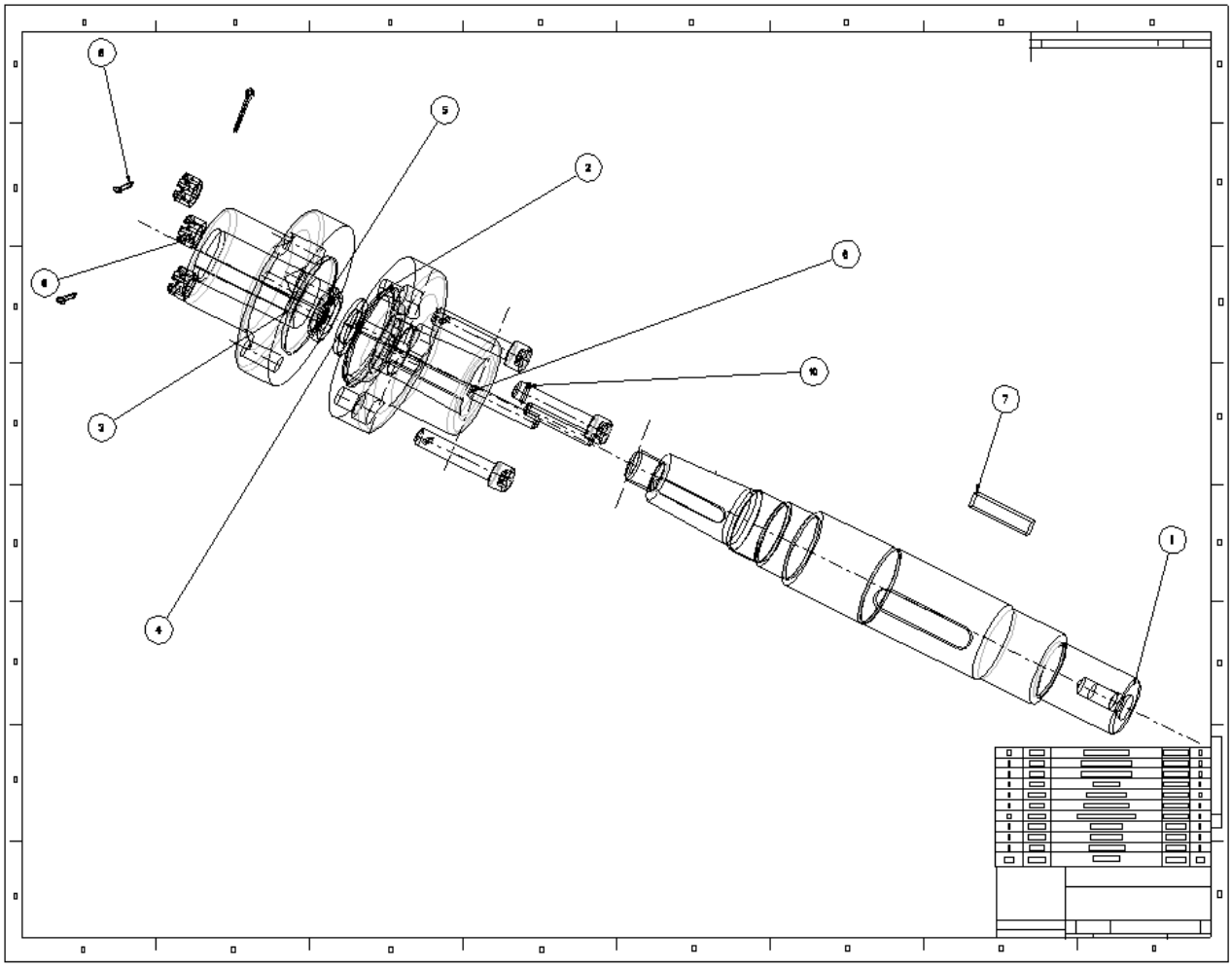
- Use default views created on template drawings
- Add views of the part or assembly to the drawing
- Display existing design dimensions
- Create additional driven or reference dimensions
- Create and insert notes on the drawing
- Add views of additional parts or assemblies
- Add multiple sheets to the drawing
- Create a BOM and balloon the assembly
- Add draft entities to the drawing

Drawings can be created of parts and assemblies. Drawings can be multiview or pictorial and can include section, auxiliary, detailed, exploded, and broken views. With Pro/DETAIL, ANSI, ISO, DIN, or JIS, standard drawings can be created.

The **Drawing Mode** is designed to allow you to create drawings, add views, dimension, and document the part or assembly. Pro/E offers several methods for creating views on the drawing. All methods are based on the rules of orthographic projection. When you create a Drawing, Pro/E creates a new file to hold your drawing. Drawing files all have a .drw file extension. The new drawing is displayed in its own graphics window.

The Drawing Mode is very parameter-intensive. There are separate parameters, which control the display of drafting annotation, views, formats, etc. The user interface would become too cumbersome if all of these parameters were controlled by menu options, and because virtually all of these parameters are actually company standards (that you do not change constantly), the parameters are defined and modified in the Drawing Options.

All drawings have a Name, a Size (Height by Length), a Scale, a Projection Angle, and a Drawing Unit. The Name is the name of the file containing the drawing. The size defines, in Drawing Units, how much space is available on the drawing to place views and annotation. The size is also used to plot the drawing. The Scale establishes the default size at which views are placed on the drawing. This scale does not affect the size of annotation in the drawing. The Projection Angle controls how views are projected. Views are projected using one of two conventions: First-Angle or Third-Angle. The Projection Angle is controlled by a Drawing Options parameter called *projection_type*. The default value of this parameter is *third_angle.*



Coupling Exploded View Assembly Drawing