An Overview of Pro/ENGINEER

The Foundation of Pro/ENGINEER

What is Pro/ENGINEER?

Pro/ENGINEER is a computer graphics system for modeling various mechanical designs and for performing related design and manufacturing operations. The system uses a 3D solid modeling system as the core, and applies the feature-based, parametric modeling method. In short, Pro/ENGINEER is a *feature-based, parametric solid modeling* system with many *extended design and manufacturing applications*.

How is Pro/ENGINEER different from other CAD Systems?

Pro/ENGINEER is the first commercial CAD system entirely based upon the feature-based design and parametric modeling philosophy. Today many software producers have recognized the advantage of this approach and started to shift their product onto this platform. Nevertheless, the differences between a feature-based, parametric solid modeling CAD system, such as Pro/ENGINEER, and a conventional CAD system include:

Pro/ENGINEER	Conventional CAD Systems
Solid Model	Wireframe and Solid Model
Parametric Model	Fixed-dimension Model
Feature-based Modeling	Primitive-based Modeling
A Single Data Structure and Full Associativity	Function-Oriented Data Structures with Format Interpreters
Subject-oriented Sub-modeling Systems	A Single Geometry-Based System
Manufacturing Information Associated with Features	Texts Attached to Geometry Entities
Generation of an Assembly by Assembling Components	Generation of an Assembly by Positioning Components

An Overview (by Parametric Technology Corp.)

- **Ease of Use:** Pro/ENGINEER was designed to begin where the design engineer begins with features and design criteria. Pro/ENGINEER's cascading menus flow in an intuitive manner, providing logical choices and pre-selecting most common options, in addition to short menu descriptions and full on-line help. This makes it simple to learn and utilize even for the most casual user. Expert users employ Pro/ENGINEER's "map keys" to combine frequently used commands along with customized menus to exponentially increase their speed in use. Because Pro/ENGINEER provides the ability to sketch directly on the solid model, feature placement is simple and accurate.
- **Full Associativity:** Pro/ENGINEER is based on a single data structure, with the ability to make change built into the system. Therefore, when a change is made anywhere in the development process, it is propagated throughout the entire design-through-manufacturing process, ensuring consistency in all engineering deliverables.
- **Parametric, Feature-Based Modeling:** Pro/ENGINEER's features are process plans with imbedded intelligence and are easy to use, while at the same time, powerful enough to fillet, round, and shell even the most complex geometry. These features contain non-geometric information, such as manufacturing processes and associated costs, as well as information about location and relationships. This means that features do not require coordinate systems for placement, and they "know" how they are related to the rest of the model. As a result, changes are made quickly and always adhere to the original design intent.
- **Powerful Assembly Capabilities:** Assembling components is easy with Pro/ENGINEER Simply tell the system to "mate," "insert," or "align" the components and they are assembled, always maintaining the design intent. Also, the components "know" how they are related, so if one changes, either positionally or geometrically, the other will change accordingly. Parts can be designed right in the assembly and defined by other components, so if they move or change size, the part will automatically update to reflect the change.
- **Robustness:** The Pro/ENGINEER family of products is based on a double precision, non-faceted solid modeling core. This provides the engineer with the most accurate representation of geometry, mass properties, and interference checking available.
- **Change Management:** Powerful change capabilities are inherent with Pro/ENGlNEER full associativity, enabling design-through-manufacturing disciplines to execute their functions in parallel. Tools for parametric data management successfully manage these simultaneous processes and promote an organized, controlled workflow.
- **Hardware Independence:** Pro/ENGINEER runs on all of the major UNIX and Windows **NT** platforms, maintaining the same look and feel on every system. Users can select the most economical hardware configuration for their needs, and mix and match any combination of platforms. Information can be easily exchanged from one machine to the other, with Pro/ENGINEER managing any architectural differences.

Pro/ENGINEER Functionality

The basic functionality of Pro/ENGINEER is broken into several areas:

- **Part Design**
  - ◊ Create sketched features including protrusions, cuts, and slots made by either extruding, revolving sweeping along a 2D sketched trajectory, or blending between parallel sections
  - ◊ Create "pick and place" features, such as holes, shafts, chamfers, rounds, shells, regular drafts, flanges, ribs, etc.
  - ◊ Sketch cosmetic features
  - ◊ Reference datum planes, axes, points, curves, coordinate systems, and graphs for creating non-solid reference datum
  - ◊ Modify, delete, suppress, redefine, and reorder features, as well as making features "read-only"
  - ◊ Create table-driven parts by adding dimensions to the family table
  - ◊ Capture design intent by creating relations between part dimensions and parameters
  - ◊ Generate engineering information, including mass properties of parts, model cross sections, and reference dimensions
  - ◊ Create geometric tolerances and surface finishes on models
  - ◊ Assign density, units, material properties or user-specified mass properties to a model
  - ◊ Additional functionality available through Pro/FEATURE.
- **Assembly Design**
  - ◊ Place components and subassemblies using commands like mate, align, and insert to create full product assemblies
  - ◊ Disassemble components from an assembly
  - ◊ Modify assembly placement offsets
  - ◊ Create and modify assembly datum planes, coordinate systems, and cross sections
  - ◊ Modify part dimensions in assembly mode
  - ◊ Generate engineering information, bills of materials, reference dimensions, and assembly mass properties
  - ◊ Additional functionality available through Pro/ASSEMBLY.
- **Design Documentation (Drawings)**
  - ◊ Create numerous types of drawing views, including general, projection, auxiliary, detailed, exploded, partial, area cross-section, and perspective
  - ◊ Perform extensive view modifications, including changing the view scale and the boundaries of partial or detailed views, adding projection and cross-section view arrows, and creating snapshot views
  - ◊ Create drawings with multiple models, delete a model from a drawing, set and highlight the current model of a drawing

- ◊ Use a <u>sketch</u> as a parametric drawing format
- ◊ Manipulate dimensions, including show, erase, switch view, flip arrows, move dimensions, text, or attach points
- ◊ Modify dimension values and number of digits
- ◊ Create, show, move, erase, and switch view for standard notes
- ◊ Include existing geometric tolerances in drawing notes
- ◊ Update the model geometry to incorporate design changes
- ◊ Export a drawing <u>IGES</u> file
- ◊ <u>Markup</u> drawings to indicate changes to be made
- ◊ Additional functionality available through Pro/DETAIL.

- **General Functionality**
  - ◊ Database management commands
  - ◊ Layer control for placing items on a layer and displaying layers
  - ◊ <u>Measuring</u> commands for distance, geometric information angle, clearance, and global interference on parts and assemblies
  - ◊ <u>Viewing</u> capabilities to pan, zoom, spin, shade, and re-orient models and drawings.

The Function Modules of Pro/ENGINEER

The core of Pro/ENGINEER is the *feature-based, parametric solid modeling* system for modeling mechanical *parts*. The part model created by this system can be used to form mechanical *assemblies* and to produce *engineering drawings*. The model can also be used to carry out other related manufacturing activities such as the generation of CNC tool paths and Bills of Material. These extended functions are reflected by the following Pro/ENGINEER modes:

Mode	**Diescription**
Sketcher	Sketch feature sections and parametric drawings. This mode can be accessed directly from the MODE menu as well as from the Part and Assembly modes.
Part	Create the solid model of a part.
Assembly	Form the solid model of an assembly of multiple components.
Drawing	Produce engineering drawings of parts and assemblies created in Pro/ENGINEER. These drawings are fully associative with the 3D solid model. When a dimension in the drawing is changed the dimension of the associated 3D model(s) will be automatically updated, and vice versa.
Manufacture	Define the machining operations that are required to manufacture a part modeled using Pro/ENGINEER.

These are frequently used Pro/ENGINEER modes. Their functions in modeling a mechanical design are illustrated in Figure 1. Other Pro/ENGINEER modes include:

Mode	**Diescription**
Cabling	Accessed from within Assembly mode, it is used to route cables between connectors and other electrical terminators (with Pro/CABLING).
Cast	Design die assemblies and prepare castings for manufacturing (with Pro/CASTING).
Composite	Create and document parts made of composite materials (with Pro/COMPOSITE).
Diagram	Create 2-D schematic representations of electrical, piping, power, heating and ventilation assemblies (with Pro/DIAGRAM).
Dieface	Design and analyze the contact surfaces of stamping dies for forming deep-drawn sheet metal parts (with Pro/DIEFACE).
Format	Create and modify drawing formats used by other Pro/ENGINEER products (with Pro/DETAIL).
Interchange	Create an object called an "interchange group", providing the ability either to automatically exchange functionally-equivalent members in an assembly or to substitute simplified versions of members in an assembly.
Layout	Create 2-D conceptual assembly sketches (with Pro/NOTEBOOK).
Legacy	Import 3D data and 2D drawings into Pro/ENGINEER from other CAD products and update these using optimized tools to work with wireframe, surface, and 2D data (with Pro/LEGACY).
Markup	Mark up a drawing, part, or assembly without changing the object itself (with basic Pro/ENGINEER).
Mold	Create and analyze molds and moldings (with Pro/MOLDESIGN).
PProcessor	Set up CL Data Post Processor
Process	Create or modify process assemblies
Report	Create custom reports for assembly Bills of Material and Project Engineering Change Orders (with Pro/REPORT).
Scan Model	Create or dynamically modify surfaces using an array of scanned point data (with Pro/SCAN-TOOLS).
Sheet Metal	Create solid models of sheet metal parts and develop the NCL data necessary to manufacture them (with Pro/SHEETMETAL).
Verify	Compare scanned model data to the design model.

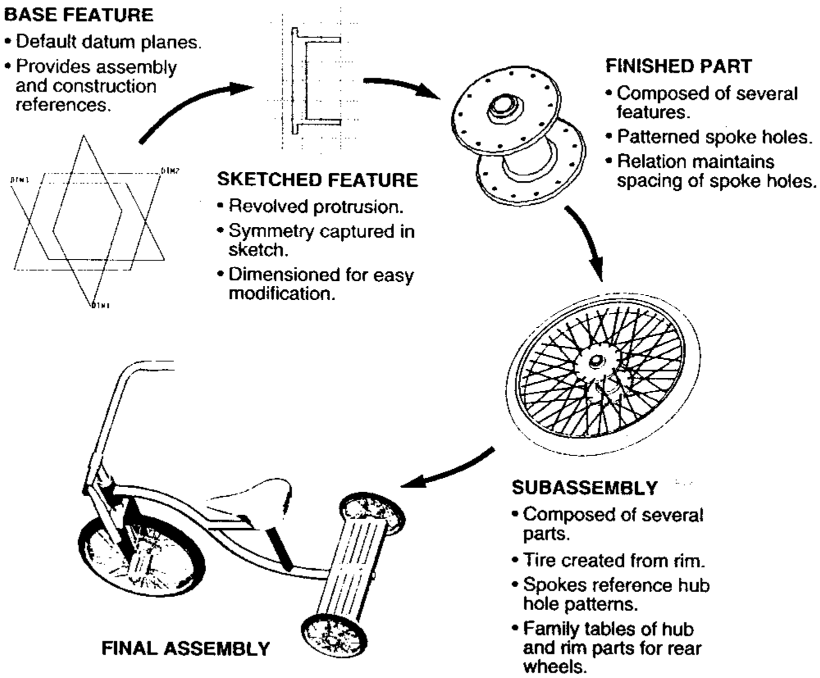


Figure 1. Commonly Used Function Modes of Pro/ENGINEER

Documentation and On-line Help

- Pro/ENGINEER Manual
  A set of Pro/ENGINEER manual can be found in the CAD/CAM Laboratory, ELW B119.

- The same manual was put on-line within Pro/ENGINEER. To read a manual item one needs to point the mouse cursor to the item and to press the right mouse button. At present this function on several Windows NT stations is not working properly.

- Pro/ENGINERR On-line Tutorials
  The web addresses of a number of excellent sites are listed. Our Pro/ENGINEER cite is located at: http://www.me.uvic.ca/mech410

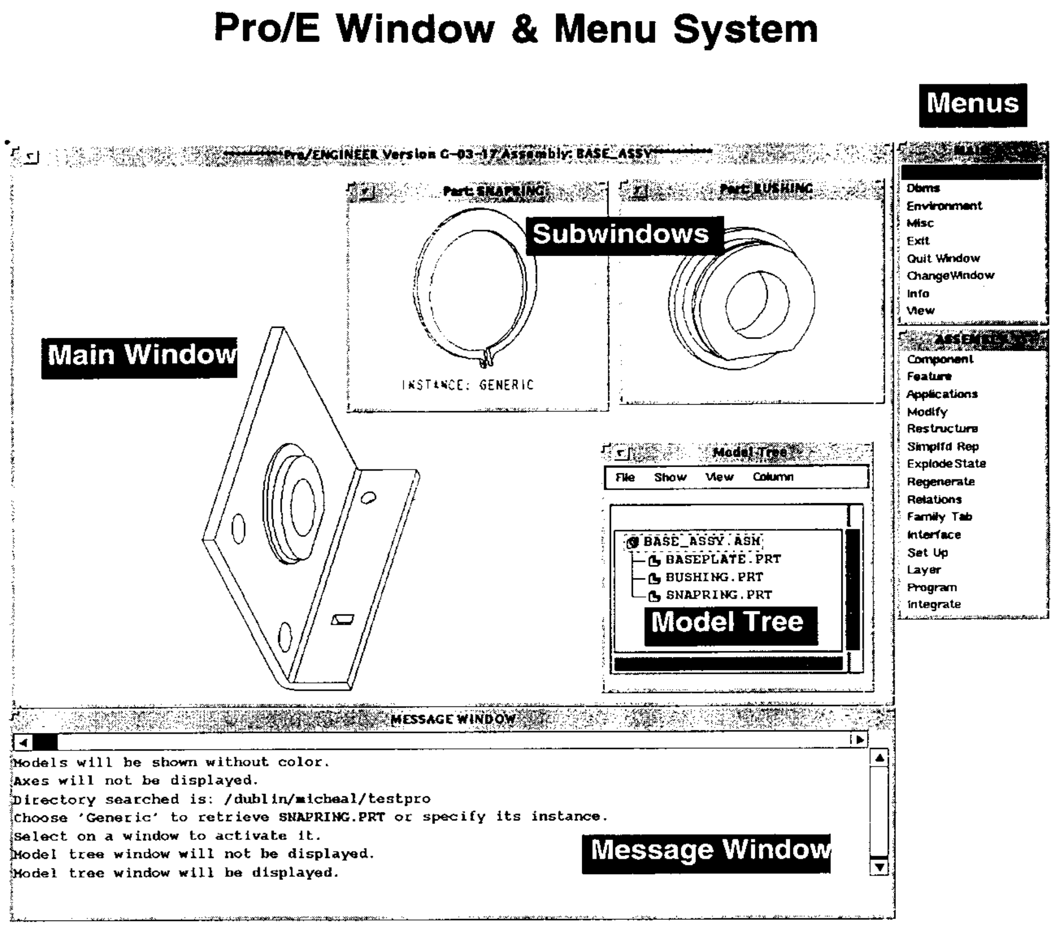
- Related web page from *Parametric Technology*

Homepage	*http://www.ptc.com/*
Products	*http://www.ptc.com/products/index.htm*
Mid-Range Solution Products	http://www.ptproducts.com/PTProducts/products.stm

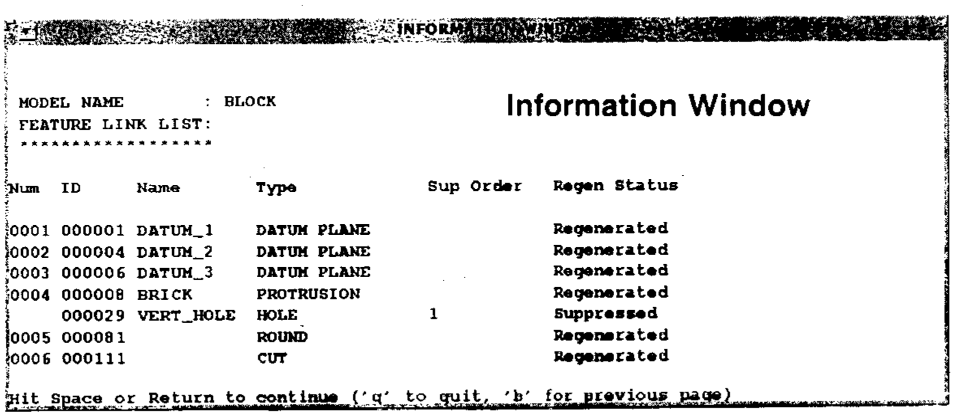
- All Pro/ENGINEER manuals can be read by executing the command on UNIX
  proguide

The User Interface of Pro/ENGINEER

To use Pro/ENGINEER, one needs first to get familiar with its graphical user interface. These include the menu system, the windows and the functions of different mouse buttons. Pro/ENGINEER screen consists of several windows as illustrated in the figure. The largest window you see when you first start Pro/ENGINEER is the *Main Window*.



A Multiple Window Environment



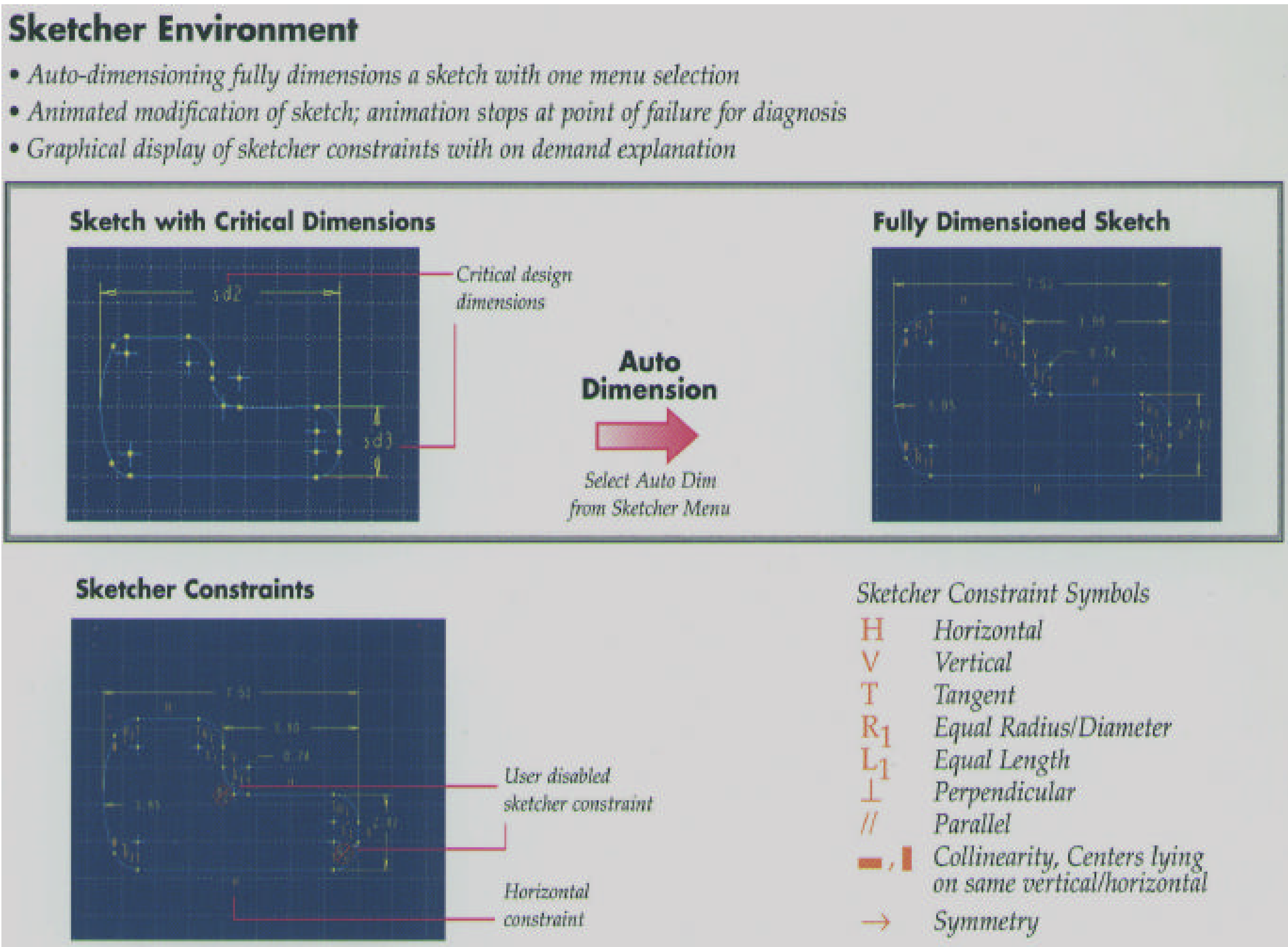
MODEL NAME : BLOCK
FEATURE LINK LIST:
****

Num	ID	Name	Type	Sup Order	Regen Status
0001	000001	DATUM_1	DATUM PLANE		Regenerated
0002	000004	DATUM_2	DATUM PLANE		Regenerated
0003	000006	DATUM_3	DATUM PLANE		Regenerated
0004	000008	BRICK	PROTRUSION		Regenerated
	000029	VERT_HOLE	HOLE	1	Suppressed
0005	000081		ROUND		Regenerated
0006	000111		CUT		Regenerated

Hit Space or Return to continue ('q' to quit, 'b' for previous page)

Typical Information Window

Pro/ENGINEER Sketcher



Pro/ENGINEER Menu System

Screen *Menus* are the primary means of navigating through Pro/ENGINEER. When a menu option is highlighted, a one-line explanation of that option is displayed in the message window at the bottom of the screen. To choose a menu option, move the pointer to the option and press the left button of the mouse. The mouse provides most input to Pro/ENGINEER. Occasionally, you need to input data through keyboard, such as naming a part or a file. Pro/ENGINEER program can be terminated by choosing *Exit* from *MAIN* menu.

Main Menu

The MAIN menu is the menu that remains on the screen and available throughout every Pro/ENGINEER session. The options of this menu is described in the following section. Each submenu carries out a specific function.

- Mode

  Choosing the Mode option from the MAIN menu brings up the MODE menu. Pro/ENGINEER is actually made up of several sub-products, called modes, each of which carries out a separate function. These modes are accessed through the MODE menu. The basic MODE menu options include:

  - ◊ Sketcher - for generating the 2D sections of the 3D model.
  - ◊ Part - for creating 3D solid part models.
  - ◊ Assembly - for assembling multiple components.
  - ◊ Drawing - for generating an engineering drawing from 3D part or assembly models.
  - ◊ Manufacture - for defining the machining sequence necessary to manufacture a part.

- View

  The View option is used to alter the way a model in the current working window is displayed. Using the VIEW menu, you can rotate, pan, and zoom the display, as well as color and shade a model.

- Dbms

  The Dbms option brings up the Data Base Management System (DBMS) menu. You can store, copy, rename, delete and erase objects using this menu at any time.

- Environment

  The Environment option is used to customize Pro/ENGINEER's operating environment.

- Exit

  The Exit option in the MAIN menu will end your Pro/ENGINEER session. Please note that Exit will not automatically save your work.

- Quit Window

  The Quit Window option will close the current model. If the current model is within the large display window, it will be cleared for the next object. If the current model is in its own smaller window, it is removed from the screen.

- ChangeWindow

  The ChangeWindow option will let you choose which window you want as the current working window.

Pro/E Menus

MAIN
- Mode
- Dbms
- Environment
- Misc
- Exit
- Quit Window
- ChangeWindow
- Info
- View

DBMS
- • Save
- Save As
- Backup
- Rename
- Erase
- EraseNotDisp
- Purge
- Inst Dbms
- Delete All
- Done-Return

ENVIRONMENT
- ✓Bell
- ✓Display Tol
- Disp DtmPln
- Disp Points
- DispPntTags
- ✓Disp Axes
- Disp Csys
- Spin Center
- Save Disp
- • Grid Snap
- SketStart2D
- Ortho Snap
- Zone Labels
- ✓Zone Grids
- ✓RegenBackup
- KeepInfoDtm
- Model Tree

- Colors
- LODs
- Texture

- ✓DispSnapLine
- SnapLineSnap

- • Wireframe
- Hidden line
- No hidden
- • Shading

- Tan Solid
- No Disp Tan
- Tan Ctrln
- Tan Phantom
- Tan Dimmed

- Isometric
- Trimetric
- User def.

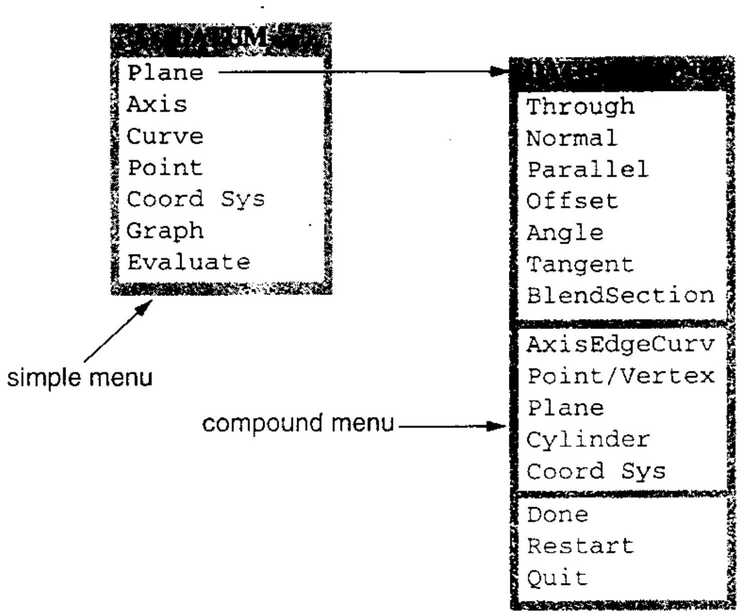
- Thick Cables
- Center Line

- Done-Return

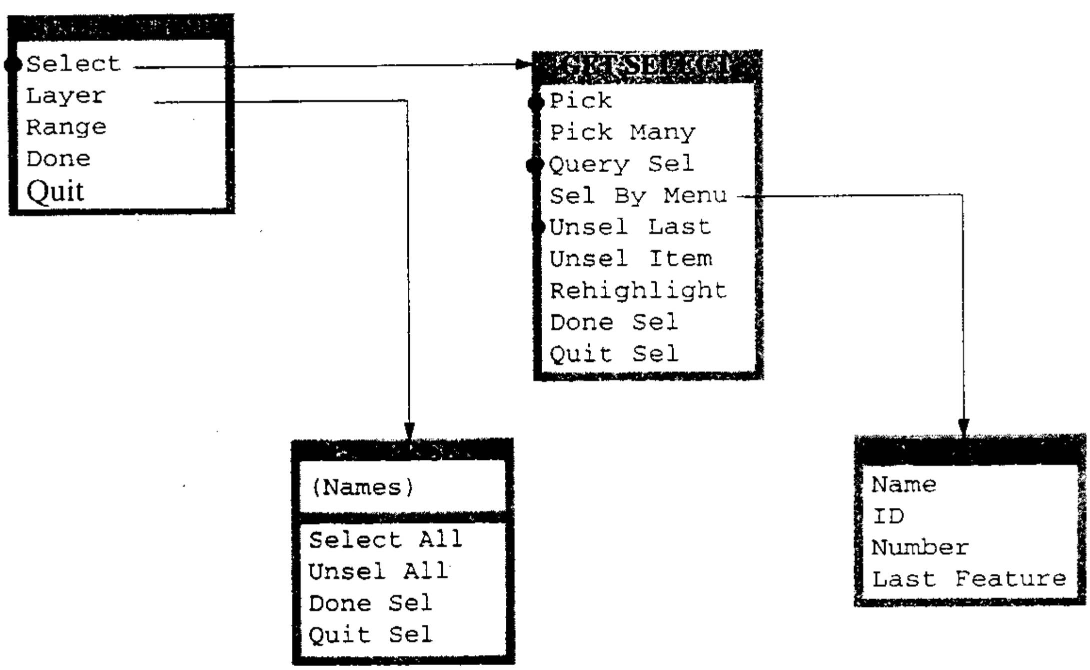
MODE
- • Sketcher
- • Part
- Sheet Metal
- Composite
- Scan Model
- • Assembly
- • Drawing
- Manufacture
- Mold
- Cast
- Dieface
- Layout
- Format
- Report
- Markup
- Diagram
- Interchange
- Legacy
- Verify
- PProcessor
- Process

Misc
- List Dir
- Show Dir
- Change Dir
- System
- Load Config
- Edit Config
- Trail
- Train
- SystemColors
- FloatOptions
- Picture
- CompareImage
- Time
- Aux App Mgmt
- Support Info
- Mapkey
- Done-Return

Pro/E Menus



Example of Simple and Compound Menus



Menu Structure for Selecting Geometry.

Pro/ENGINEER Windows

- Main Window

  The Main Window displays the solid object models created by Pro/ENGINEER.
- Subwindows

  Subwindows are created by the users to display multiple object models and multiple views. The Sketcher will also create a 2D subwindow.
- Model Tree Window

  The window displays the hierarchical structure of the created assembly model, as well as the composing elements of an object model. Assembly and topological relations of modeled objects can be edited within this window.
- Message Window

  Underneath the Main Window, the Message Window feeds back to the user and acquires additional information from the user.
- Information Window

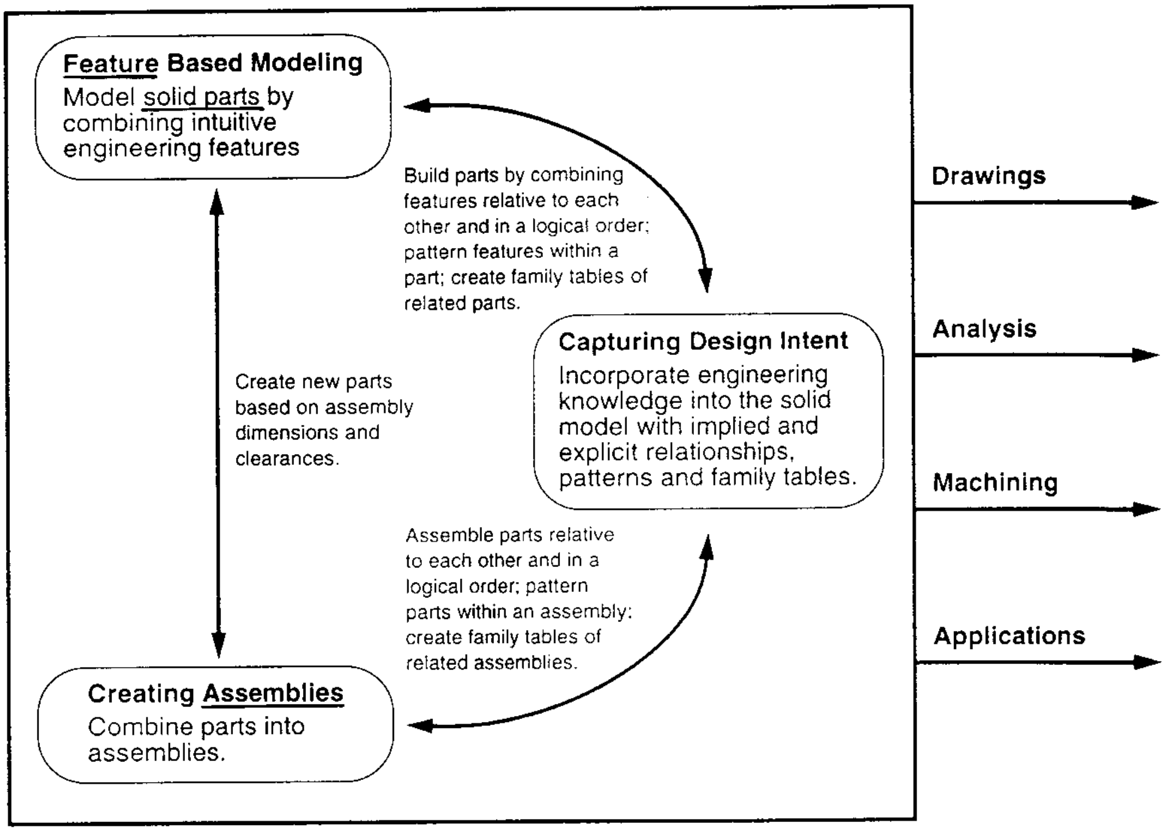
  The window displays all related data of the model. The window tells a user whether all information has been completely specified to form a part model.

Functions of Different Mouse Buttons

The three mouse buttons provide different functions and short cuts to menu items. These have been summarized by Prof. Roger TooGood in his Pro/E Tutorial book:

Mouse Buttons	**Left**	**Middle**	**Right**
Regular	Pick	Done Select	Query Select
Dynamic View Control (press CTRL plus...)	(drag) Zoom In/Out	(drag) 3D Spin	(drag) Pan
Zoom Window (press CTRL plus...)	Click opposite comers of zoom box		
Query Select	Pick	Accept	Next
Mouse Sketch - Draw Entity	Line	Circle	Tangent Arc
Mouse Sketch - Line mode		Abort/End	
Mouse Sketch - Circle mode	Abort/End		
Mouse Sketch - Tangent arc mode		Abort/End	
Sketcher Dimension - Linear	Pick entity	Place Dimension	
Sketcher Dimension - Radius	Pick arc/circle	Place Dimension	
Sketcher Dimension - Diameter	Double pick arc/circle	Place Dimension	

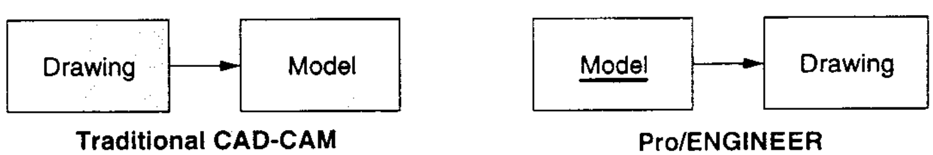
Modeling with Pro/ENGINEER



Parametric Solid Model

Modeling vs. Drafting

A primary and essential difference between Pro/ENGINEER and traditional computer aided drafting systems is that Pro/ENGINEER models are three dimensional. In Pro/ENGINEER, drawings are produced as views of the model, rather than the other way around. Pro/ENGINEER models are not *drawn* so much as *sculpted* from solid volumes of material.

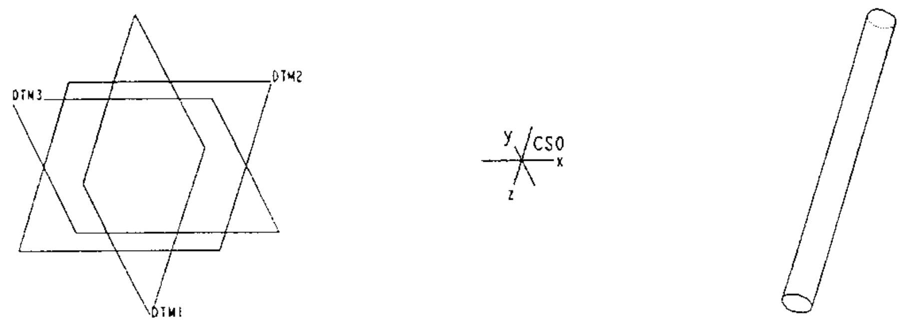


Comparison of Traditional CAD-CAM and Pro/ENGINEER

FEATURE BASED MODELING

The "chunks" of solid material from which Pro/ENGINEER mode are constructed are called *features*. Features generally fall into one the following categories:

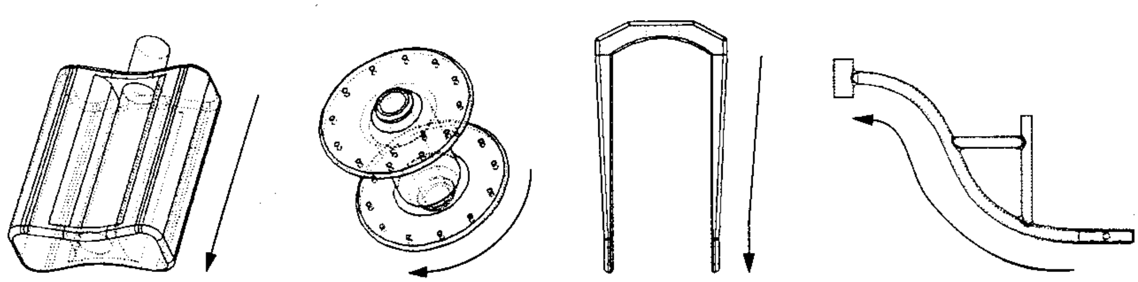
- **Base Feature** - The base feature may be either a sketched feature or datum plane(s) referencing the default coordinate system. The base feature is important because all future model geometry will reference this feature directly or indirectly; it becomes the *root feature*. Changes to the base feature will affect the geometry of the entire model.



Default Datum Planes | Default Coordinate System | Sketched Protrusion

Base Features

- **Sketched Features** - In general, sketched features are created by extruding, revolving, blending, or sweeping a sketched cross section. Material may be added or removed by protruding or cutting the feature from the existing model.



Extrude: Pedal created by extruding bow-shaped section.

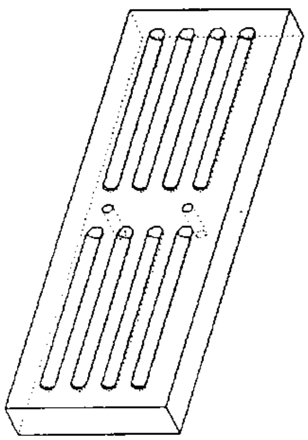
Revolve: Hub created by revolving section.

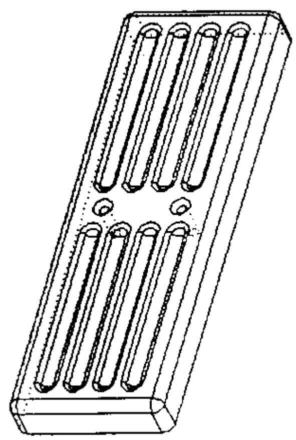
Blend: Fork created by blending several cross sections.

Sweep: Frame created by sweeping cross section along shown trajectory.

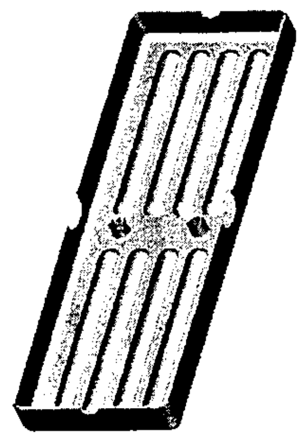
Sketched Features

- **Referenced Features** - Referenced features reference existing geometry and employ an inherent form; they do not need to be sketched. Some examples of referenced features are rounds, drilled holes, and shells.





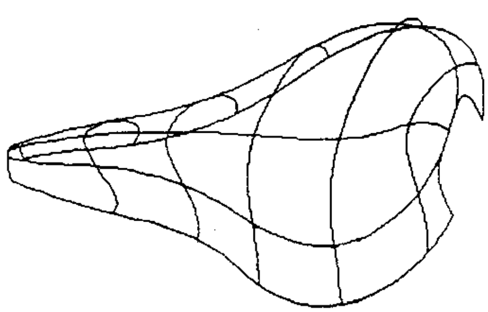
Rounds reference feature edges and surfaces, removing material to a specified radius.

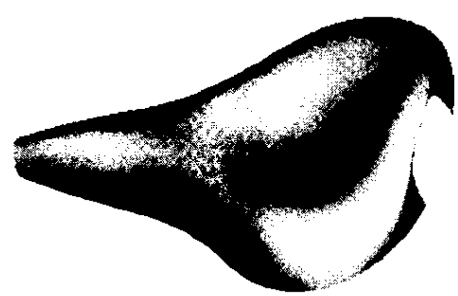


Shell feature references outer surfaces, reducing thickness to specified value.

Referenced Features

- **Datum Features** - Datum features, such as planes, axes, curves, and points, are generally used to provide sketching planes and contour references for sketched and referenced features. Datum features do not have physical volume or mass, and may be visually hidden without affecting solid geometry.



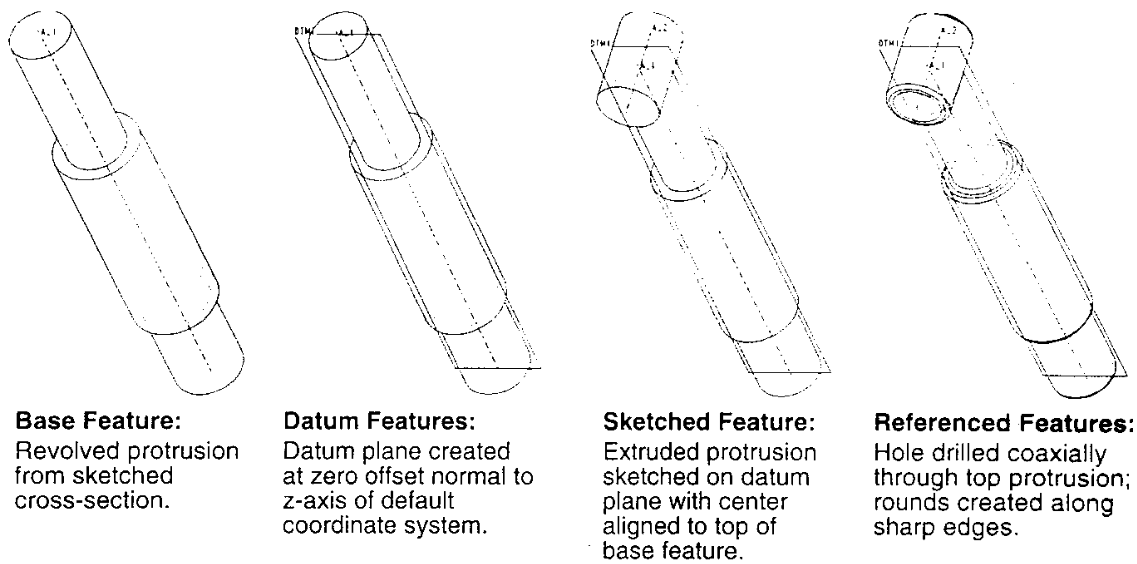


Web of datum curves used to control surface contour. Seat created by enclosing volume with additional surfaces and filling with solid material.

Use of Datum Curves to Control Surface Contours

Combining Features into Parts

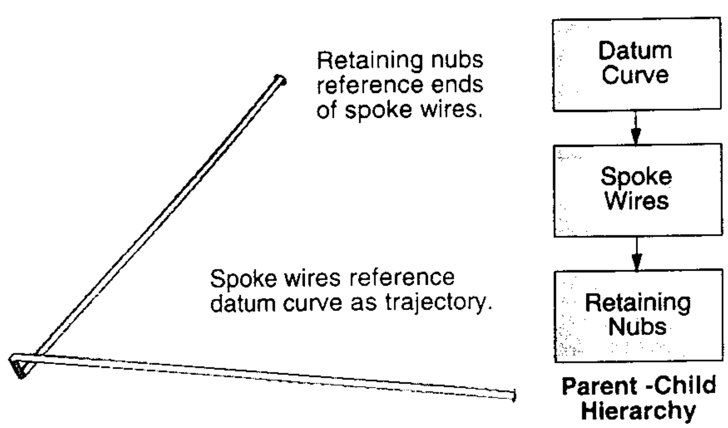
The various types of features are used as building blocks in the progressive creation of solid parts. Figure 2-7 demonstrates this process.



Combining Features into Parts

PARENT-CHILD RELATIONSHIPS

Because solid modeling in Pro/ENGINEER is a cumulative process, certain features must, by necessity, precede others. Those that follow must rely on previously defined features for dimensional and geometric references. The relationships between features and those that reference them are termed *parent-child relationships*. Because children reference parents, features can exist without children, but children cannot exist without their parents.



Parent-Child Relationships

Parametric Modifications

The parent-child relationship is one of the most powerful aspects of Pro/ENGINEER; when a parent feature is modified, its children are automatically revised to reflect the changes in the parent feature's geometry. It is therefore essential to reference feature dimensions so that design modifications are correctly propagated through the model. In the above illustration, for example, it is best to reference the nubs to the tips of the spoke wires so that should the spoke wires change length, the nubs will remain at their ends. The neck piece works the same way. Figure 2-9 illustrates how a modification to the length of the neck is automatically propagated through the part.

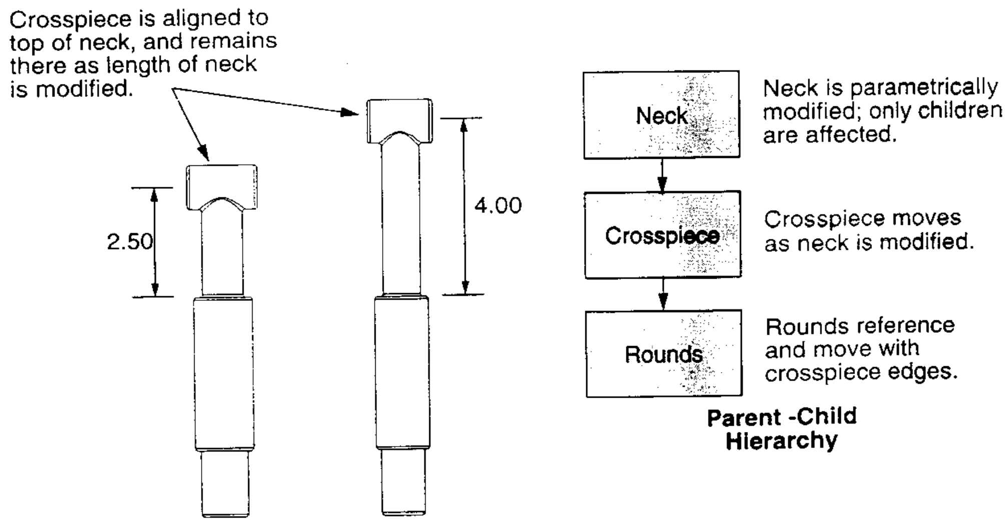


Figure 2-9
Parametric Modifications

CREATING ASSEMBLIES

Just as parts are created from related features, so assemblies are created from related parts. As shown in Figure 2-10, the progressive assembly of parts and features into an assembly creates parent-child relationships based on the references used to assemble each component.

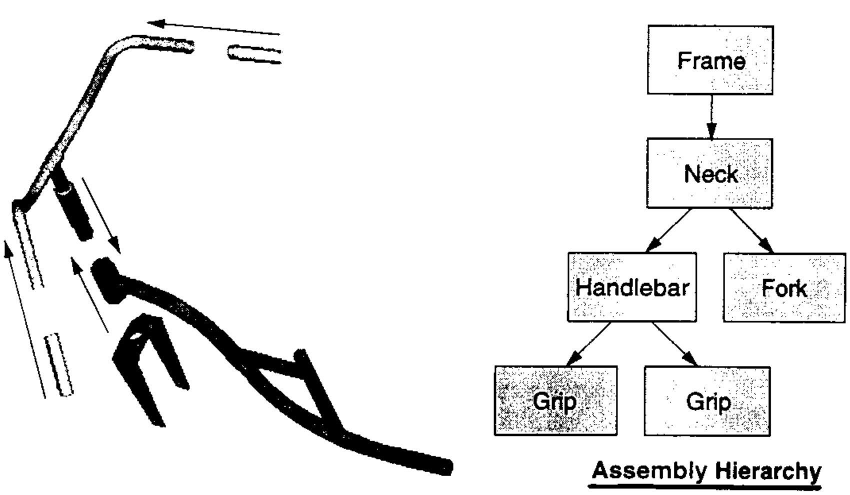


Figure 2-10
Creating Assemblies

Creating Parts in Assembly Mode

Similarly, as features can reference part geometry, Pro/ENGINEER also allows creation of parts referencing assembly geometry. Assembly mode allows the designer to both fit parts together and to design parts based on how they *should* fit together.

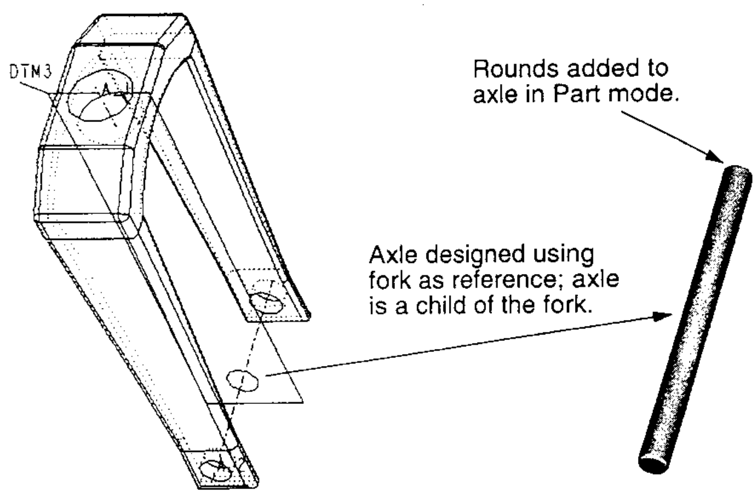


Figure 2-11
Creating Parts in Assembly Mode

CAPTURING DESIGN INTENT

A valuable aspect of any design tool is its ability to not only *render* the design, but to capture its *intent*. The concept of capturing design intent is based on incorporating engineering knowledge into a model by establishing and preserving certain geometrical relationships. The wall thickness of a pressure vessel, for example, should be proportional to its surface area, and should remain so even as its size changes. Pro/ENGINEER captures these relationships in several ways:

- **Implicit Relationships** - Implicit relationships occur when new model geometry is sketched and dimensioned relative to existing features and parts. An implicit relationship is established, for instance, when the section sketch of a tire uses rim edges for reference.

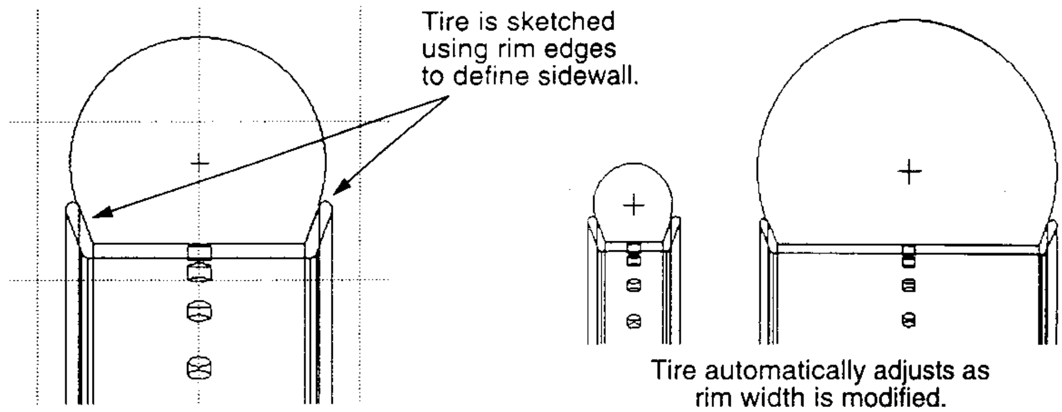


Figure 2-12
Implicit Relationships

- **Patterns** - Design features often follow a geometrically predictable pattern. Features and parts are patterned in Pro/ENGINEER by referencing either construction dimensions or existing patterns. One example of patterning is a wheel hub with spokes. First, the spoke holes are radially patterned. The spokes can then be strung by referencing this pattern.

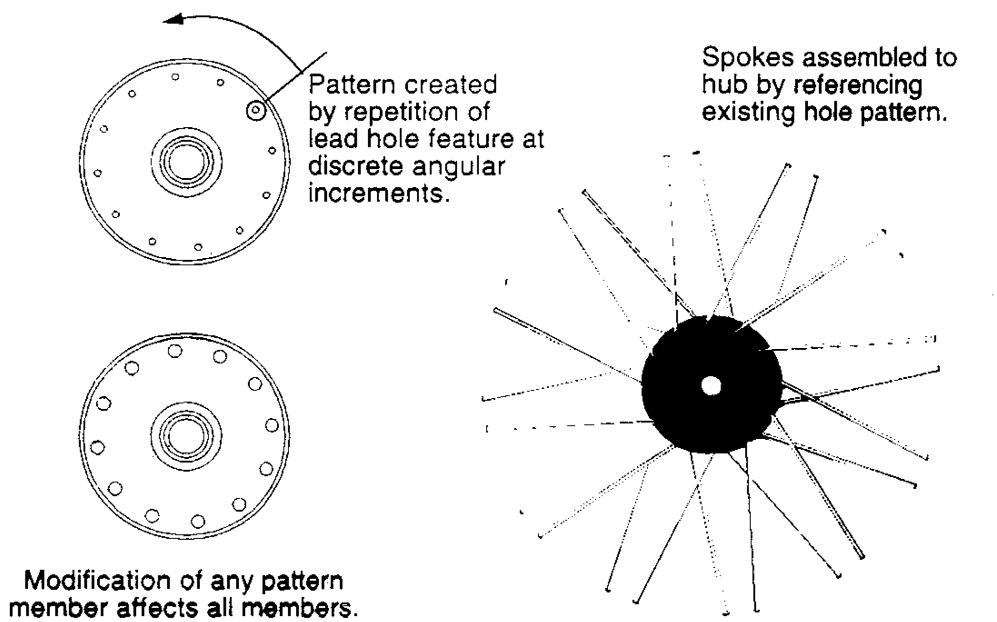
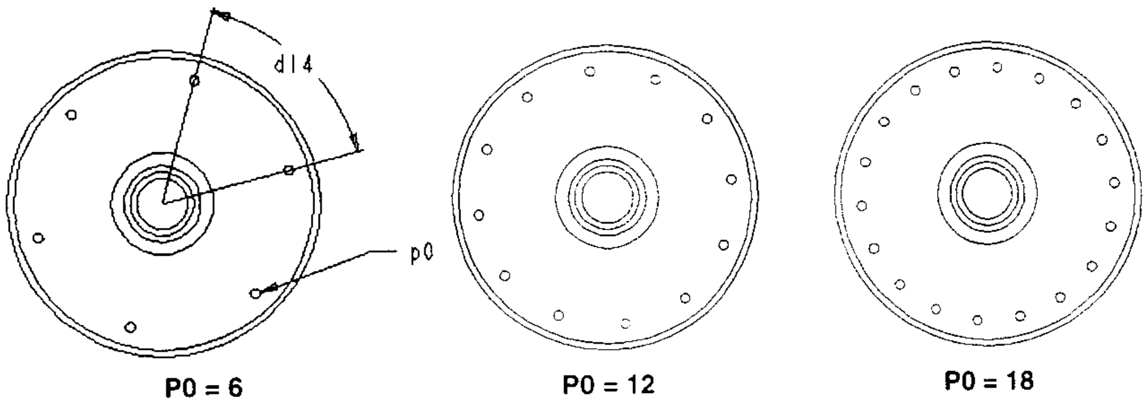


Figure 2-13
Patterns

Note also that any modification of a pattern member affects all members of that pattern. This helps capture design intent by preserving the duplicate geometry of pattern members.

- **Explicit Relationships** - While implicit relationships are implied by the feature creation method, an explicit relation is mathematically entered by the user. This equation is used to relate part and feature dimensions in the desired manner. An explicit relation might be used, for example, to ensure that any number of spoke holes will be evenly spaced around the hub.

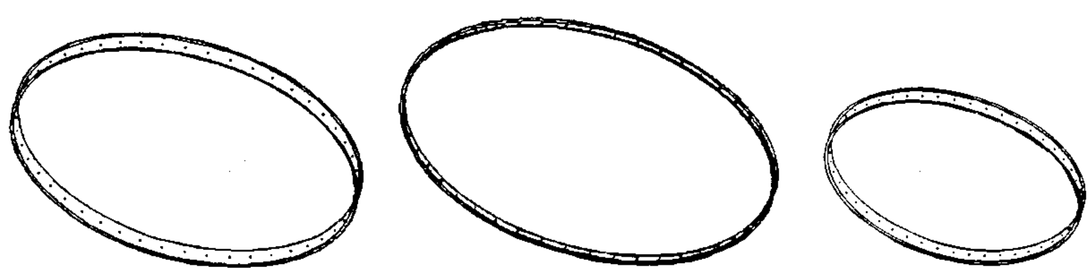


Relation: `d14=360/p0`

Where: `d14` = angular separation between pattern instances
`p0` = number of pattern instances

Figure 2-14
Explicit Relationships

- **Family Tables** - Family tables are used to create part families from generic models by tabulating dimensions or the presence of certain features or parts. A family table might be used, for example, to catalog a series of wheel rims with varying widths and diameters.



Name	Diameter	Width
MOUNTAIN	24.00	1.25
ROAD	26.00	0.50
DIRT	18.00	1.00

Figure 2-15
Family Tables