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:

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<tr>
<th>Pro/ENGINEER</th>
<th>Conventional CAD Systems</th>
</tr>
</thead>
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<tr>
<td>Solid Model</td>
<td>Wireframe and Solid Model</td>
</tr>
<tr>
<td>Parametric Model</td>
<td>Fixed-dimension Model</td>
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<tr>
<td>Feature-based Modeling</td>
<td>Primitive-based Modeling</td>
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<td>A Single Data Structure and Full</td>
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<td>Associativity</td>
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<td>Subject-oriented Sub-modeling Systems</td>
<td>A Single Geometry-Based System</td>
</tr>
<tr>
<td>Manufacturing Information</td>
<td>Texts Attached to Geometry Entities</td>
</tr>
<tr>
<td>Associated with Features</td>
<td></td>
</tr>
<tr>
<td>Generation of an Assembly by</td>
<td>Generation of an Assembly by</td>
</tr>
<tr>
<td>Assembling Components</td>
<td>Positioning Components</td>
</tr>
</tbody>
</table>
An Overview

- **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/ENGINEER 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 **sketch** 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 **IGES** file
◊ **Markup** 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
  ◇ **Measuring** commands for distance, geometric information angle, clearance, and global interference on parts and assemblies
  ◇ **Viewing** 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:

<table>
<thead>
<tr>
<th><strong>Mode</strong></th>
<th><strong>Description</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Sketcher</td>
<td>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.</td>
</tr>
<tr>
<td>Part</td>
<td>Create the solid model of a part.</td>
</tr>
<tr>
<td>Assembly</td>
<td>Form the solid model of an assembly of multiple components.</td>
</tr>
<tr>
<td>Drawing</td>
<td>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.</td>
</tr>
<tr>
<td>Manufacture</td>
<td>Define the machining operations that are required to manufacture a part modeled using Pro/ENGINEER.</td>
</tr>
</tbody>
</table>
These are frequently used Pro/ENGINEER modes. Their functions in modeling a mechanical design are illustrated in Figure 1. Other Pro/ENGINEER modes include:

<table>
<thead>
<tr>
<th>Mode</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cabling</td>
<td>Accessed from within Assembly mode, it is used to route cables between connectors and other electrical terminators (with Pro/CABLING).</td>
</tr>
<tr>
<td>Cast</td>
<td>Design die assemblies and prepare castings for manufacturing (with Pro/CASTING).</td>
</tr>
<tr>
<td>Composite</td>
<td>Create and document parts made of composite materials (with Pro/COMPOSITE).</td>
</tr>
<tr>
<td>Diagram</td>
<td>Create 2-D schematic representations of electrical, piping, power, heating and ventilation assemblies (with Pro/DIAGRAM).</td>
</tr>
<tr>
<td>Dieface</td>
<td>Design and analyze the contact surfaces of stamping dies for forming deep-drawn sheet metal parts (with Pro/DIEFACE).</td>
</tr>
<tr>
<td>Format</td>
<td>Create and modify drawing formats used by other Pro/ENGINEER products (with Pro/DETAIL).</td>
</tr>
<tr>
<td>Interchange</td>
<td>Create an object called an &quot;interchange group&quot;, providing the ability either to automatically exchange functionally-equivalent members in an assembly or to substitute simplified versions of members in an assembly.</td>
</tr>
<tr>
<td>Layout</td>
<td>Create 2-D conceptual assembly sketches (with Pro/NOTEBOOK).</td>
</tr>
<tr>
<td>Legacy</td>
<td>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).</td>
</tr>
<tr>
<td>Markup</td>
<td>Mark up a drawing, part, or assembly without changing the object itself (with basic Pro/ENGINEER).</td>
</tr>
<tr>
<td>Mold</td>
<td>Create and analyze molds and moldings (with Pro/MOLDESIGN).</td>
</tr>
<tr>
<td>Processor</td>
<td>Set up CL Data Post Processor</td>
</tr>
<tr>
<td>Process</td>
<td>Create or modify process assemblies</td>
</tr>
<tr>
<td>Scan Model</td>
<td>Create or dynamically modify surfaces using an array of scanned point data (with Pro/SCAN-TOOLS).</td>
</tr>
<tr>
<td>Sheet Metal</td>
<td>Create solid models of sheet metal parts and develop the NCL data necessary to manufacture them (with Pro/SHEETMETAL).</td>
</tr>
<tr>
<td>Verify</td>
<td>Compare scanned model data to the design model.</td>
</tr>
</tbody>
</table>
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/ENGINEER 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](http://www.me.uvic.ca/mech410)

- **Related web page from Parametric Technology**

- **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*.

---

**Pro/E Window & Menu System**

A Multiple Window Environment

---

**Information Window**

<table>
<thead>
<tr>
<th>Num</th>
<th>ID</th>
<th>Name</th>
<th>Type</th>
<th>Sup Order</th>
<th>Regen Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>0001</td>
<td>000001</td>
<td>DATUM_1</td>
<td>DATUM PLANE</td>
<td></td>
<td>Regenerated</td>
</tr>
<tr>
<td>0002</td>
<td>000004</td>
<td>DATUM_2</td>
<td>DATUM PLANE</td>
<td></td>
<td>Regenerated</td>
</tr>
<tr>
<td>0003</td>
<td>000006</td>
<td>DATUM_3</td>
<td>DATUM PLANE</td>
<td></td>
<td>Regenerated</td>
</tr>
<tr>
<td>0004</td>
<td>000008</td>
<td>BRICK</td>
<td>PROTRUSION</td>
<td></td>
<td>Regenerated</td>
</tr>
<tr>
<td>0005</td>
<td>000001</td>
<td>HOLE</td>
<td>HOLE</td>
<td>1</td>
<td>Suppressed</td>
</tr>
<tr>
<td>0006</td>
<td>000111</td>
<td>CUT</td>
<td>CUT</td>
<td></td>
<td>Regenerated</td>
</tr>
</tbody>
</table>

Hit Space or Return to continue ('g' to quit, 'b' for previous page)
Pro/ENGINEER Sketcher

**Sketcher Environment**

- Auto-dimensioning fully dimensions a sketch with one menu selection
- Animated modification of sketch; animation stops at point of failure for diagnosis
- Graphical display of sketcher constraints with on demand explanation

**Sketch with Critical Dimensions**

- Critical design dimensions

**Fully Dimensioned Sketch**

- Select Auto Dim from Sketcher Menu

**Sketcher Constraints**

- User disabled sketcher constraint
- Horizontal constraint

**Sketcher Constraint Symbols**

- **H**: Horizontal
- **V**: Vertical
- **T**: Tangent
- **R**: Equal Radius/Diameter
- **L**: Equal Length
- **⊥**: Perpendicular
- **//**: Parallel
- **↔, ↔**: Collinearity, Centers lying on same vertical/horizontal
- **→**: Symmetry
Pro/ENGINEER Menu System

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
- Change Window
- 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

**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/ENGINEER Operating Functions
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:

<table>
<thead>
<tr>
<th>Mouse Buttons</th>
<th>Left</th>
<th>Middle</th>
<th>Right</th>
</tr>
</thead>
<tbody>
<tr>
<td>Regular</td>
<td>Pick</td>
<td>Done Select</td>
<td>Query Select</td>
</tr>
<tr>
<td>Dynamic View Control</td>
<td>(drag)</td>
<td>(drag)</td>
<td>(drag)</td>
</tr>
<tr>
<td>Dynamic View Control (press CTRL plus...)</td>
<td>Zoom In/Out</td>
<td>3D Spin</td>
<td>Pan</td>
</tr>
<tr>
<td>Zoom Window</td>
<td>Click opposite corners of zoom box</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Query Select</td>
<td>Pick</td>
<td>Accept</td>
<td>Next</td>
</tr>
<tr>
<td>Mouse Sketch - Draw Entity</td>
<td>Line</td>
<td>Circle</td>
<td>Tangent Arc</td>
</tr>
<tr>
<td>Mouse Sketch - Line mode</td>
<td>Abort/End</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mouse Sketch - Circle mode</td>
<td>Abort/End</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mouse Sketch - Tangent arc mode</td>
<td>Abort/End</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sketcher Dimension - Linear</td>
<td>Pick entity</td>
<td>Place Dimension</td>
<td></td>
</tr>
<tr>
<td>Sketcher Dimension - Radius</td>
<td>Pick arc/circle</td>
<td>Place Dimension</td>
<td></td>
</tr>
<tr>
<td>Sketcher Dimension - Diameter</td>
<td>Double pick arc/circle</td>
<td>Place Dimension</td>
<td></td>
</tr>
</tbody>
</table>
Modeling with Pro/ENGINEER

Feature Based Modeling
Model solid parts by combining intuitive engineering features

Build parts by combining features relative to each other and in a logical order; pattern features within a part; create family tables of related parts.

Create new parts based on assembly dimensions and clearances.

Capturing Design Intent
Incorporate engineering knowledge into the solid model with implied and explicit relationships, patterns and family tables.

Assemble parts relative to each other and in a logical order; pattern parts within an assembly; create family tables of related assemblies.

Creating Assemblies
Combine parts into assemblies.

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 models are constructed are called features. Features generally fall into one of 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](image)

![Default Coordinate System](image)

![Sketched Protrusion](image)

**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.](image)

![Revolve: Hub created by revolving section.](image)

![Blend: Fork created by blending several cross sections.](image)

![Sweep: Frame created by sweeping cross section along shown trajectory.](image)

**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.

Base Feature: Revolved protrusion from sketched cross-section.

Datum Features: Datum plane created at zero offset normal to z-axis of default coordinate system.

Sketched Feature: Extruded protrusion sketched on datum plane with center aligned to top of base feature.

Referenced Features: Hole drilled coaxially through top protrusion, rounds created along sharp edges.

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.
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.

![Diagram](Image)

**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.

![Diagram](Image)

**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.

![Diagram showing parts in assembly mode]

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.

![Diagram of a wheel with different values of P0](image)

Relation: \( d_{14} = \frac{360}{P_0} \)

Where: \( d_{14} = \) angular separation between pattern instances  
\( P_0 = \) 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.

![Diagram of wheel rims with varying diameters](image)

<table>
<thead>
<tr>
<th>Name</th>
<th>Diameter</th>
<th>Width</th>
</tr>
</thead>
<tbody>
<tr>
<td>MOUNTAIN</td>
<td>24.00</td>
<td>1.25</td>
</tr>
<tr>
<td>ROAD</td>
<td>26.00</td>
<td>0.50</td>
</tr>
<tr>
<td>DIRT</td>
<td>18.00</td>
<td>1.00</td>
</tr>
</tbody>
</table>

**Figure 2-15**  
Family Tables