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*.

Pro/ENGINEER Packages

		Pro/ENGINEER Foundation XE	Pro/ENGINEER Advanced SE	Pro/ENGINEER Advanced XE	Pro/ENGINEER Enterprise SE	Pro/ENGINEER Enterprise XE
3D Detail Design/ Part Modeling	Design validation with PTC's ModelCHECK™	●	●	●	●	●
	Solid, sheetmetal, and weld modeling	●	●	●	●	●
	Support for 3D drawings and annotations per new ASME Y14.41 standard	●	●	●	●	●
	3D cabling and piping design			○	●	●
Surfacing	Advanced parametric surfacing	●	●	●	●	●
	Global modeling and warp surface modification	●	●	●	●	●
	Interactive surface design for creating complex surfaces, G2 continuity			○	●	●
Assembly Modeling	Embed form, fit and function knowledge with AssemblySense™	●	●	●	●	●
	Flexible models; single BOM entry for multiple geometry states	●	●	●	●	●
	Simplified representations and Shrinkwrap™ which protects intellectual property and reduces file size	●	●	●	●	●
	Advanced assembly with top-down design, process planning, design for manufacturability, and 2D associative process sheets			○	●	●
Interoperability and Data Exchange	Incorporation of Web Services for native Web connectivity	●	●	●	●	●
	Imported data repair	●	●	●	●	●
	Multi-platform support including Windows/Solaris/HP-UX	●	●	●	●	●
	Support for all major standards such as STEP/IGES/DXF/STL/VRML, AutoCAD DWG, DXF (import of 3D with associated 2D), ACIS import/export, Parasolid import/export*	●	●	●	●	●

Pro/ENGINEER Packages

		Pro/ENGINEER Foundation XE	Pro/ENGINEER Advanced SE	Pro/ENGINEER Advanced XE	Pro/ENGINEER Enterprise SE	Pro/ENGINEER Enterprise XE
Simulation	Real-time photorendering	●	●	●	●	●
	Design animation with movies, motion envelopes, and interference checking	●	●	●	●	●
	Mechanism kinematics design with click-and-drag animation	●	●	●	●	●
	Mechanism dynamics simulation of force, velocity, acceleration, torque			○	●	●
	Advanced behavioral modeling for product design and optimization			○	●	●
	Structural and thermal simulation					●
Collaboration and Project Management	A secure project workspace for collaboration with global team members				●	●
	Project management and execution that controls action items, milestones and deliverables				●	●
Digital Product Data Management and Process Control	Pro/ENGINEER CAD data management		●	●	●	●
	Broad enterprise product data management **		●	●	●	●
	Automated change management process**		●	●	●	●
	Configuration management **		●	●	●	●
Engineering Optimization	Engineering calculations (Mathcad)					●
	Tolerance Analysis (Pro/ENGINEER Tolerance Analysis Extension powered by CETOL Technology)					●
	Digital Rights Management (Pro/ENGINEER Rights Management Extension)					●

How is Pro/ENGINEER different from other CAD systems?

Pro/ENGINEER was the first CAD system entirely based upon feature-based design and parametric modeling. Today most software producers have recognized the advantage of this approach and shifted their product onto this platform. Nevertheless, the differences between a feature-based, parametric solid modeling CAD system, and a conventional CAD system include:

Pro/ENGINEER	Conventional CAD Systems
Solid Model	Wireframe and Solid Model
Parametric Model	Fixed Model
Feature-Based Modeling	Primitive-Based Modeling
Single Data Structure and Full Associativity	Function-Oriented Data Structure and 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

Ease of Use:

- Pro/ENGINEER was designed to begin where the design engineer begins with features and design criteria, through cascading menus.
- Expert users employ "map keys" to combine frequently used commands along with customized menus to exponentially increase their speed in use.
- Pro/ENGINEER provides the ability to sketch directly on the solid model, feature placement is thus 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.

Parametric, Feature-Based Modeling:

- Pro/ENGINEER's features 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. They are assembled, always maintaining the design intent.
- 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.

Hardware Independence:

- Pro/ENGINEER runs on all of the major UNIX and Windows 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.

Pro/ENGINEER Functionality

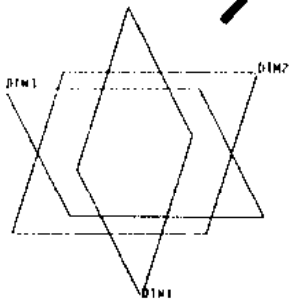
The basic functionality of Pro/ENGINEER is broken into four major areas:

- Part Modeling and Design
- Assembly Modeling and Design
- Design Documentation (Drawing Generation)
- General Functionality

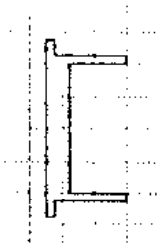
Pro/E Modeling

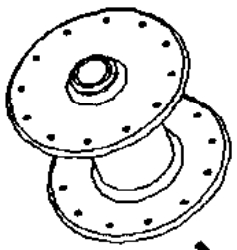
Sketched Features

BASE FEATURE
- Default datum planes.
- Provides assembly and construction references.



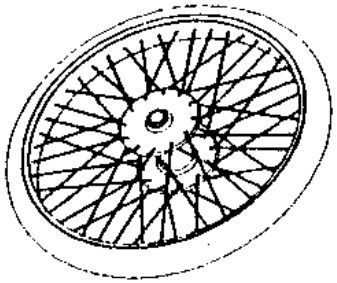
SKETCHED FEATURE
- Revolved protrusion.
- Symmetry captured in sketch.
- Dimensioned for easy modification.



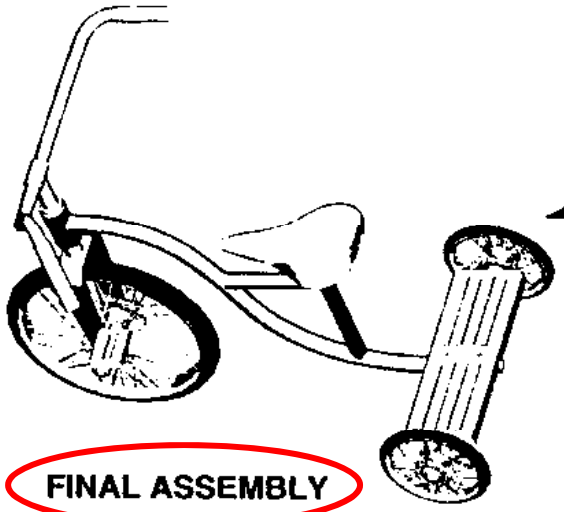


FINISHED PART
- Composed of several features.
- Patterned spoke holes.
- Relation maintains spacing of spoke holes.

Virtual & Build in Features



SUBASSEMBLY
- Composed of several parts.
- Tire created from rim.
- Spokes reference hub hole patterns.
- Family tables of hub and rim parts for rear wheels.



FINAL ASSEMBLY

Part Design and Modeling

Defining Geometry - Feature-Based Design

- Create **sketched features** including protrusions, cuts, and slots made by either extruding, revolving, or 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

Manipulating Geometry and Parametric Modeling

- 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 (Model → 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 bound-aries of partial or detailed views, adding projection and cross-section view arrows, & creating snapshot views
- Create drawings with multiple models, delete a model from a drawing, set and high-light the current model of a drawing
- Use a sketch as a parametric drawing format
- **Manipulate dimensions**, including show, erase, switch view, flip arrows, move dimen-sions, 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
- **Markup** drawings to indicate changes to be made
- Export a drawing **IGES file**
- 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 **many other** related *analysis, simulation, planning and manufacturing* activities such as the generation of CNC tool paths and Bills of Material. These extended functions are reflected by the following example Pro/ENGINEER **modes**.

BASIC MODES

Sketcher Define the 2D cross-section (or section) of an object model for sweeping.

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.

These are frequently used Pro/ENGINEER modes.

Pro/ENGINEER Wildfire Modes

Pro/ENGINEER Detailed Design (CAD)

Pro/ENGINEER Flex3C
Pro/ENGINEER Foundation Advantage
Pro/ENGINEER **Advanced Assembly**
Pro/ENGINEER **API Toolkit**
Pro/ENGINEER **Design Collaboration**
Pro/ENGINEER Expert Framework
Pro/ENGINEER Student Edition

Pro/ENGINEER Simulation (Analysis/Result Display) (CAE)

Pro/ENGINEER **Advanced Structural and Thermal**
Pro/ENGINEER **Structural and Thermal**
Pro/ENGINEER Fatigue Advisor
Pro/ENGINEER **Mechanism Dynamics**
Pro/ENGINEER Behavioral Modeling

Pro/ENGINEER Wildfire Modes

Pro/ENGINEER Production (CAM)

Pro/ENGINEER Complete Mold Design
Pro/ENGINEER **Complete Machining**
Pro/ENGINEER **Computer-Aided Verification**
Pro/ENGINEER Expert Moldbase
Pro/ENGINEER NC Sheetmetal
Pro/ENGINEER Plastic Advisor
Pro/ENGINEER **Prismatic and Multi-surface Milling**
Pro/ENGINEER Production Machining
Pro/ENGINEER Progressive Die
Pro/ENGINEER Tool Design

Pro/ENGINEER Routed Systems

Pro/ENGINEER Routed Systems Designer
Pro/ENGINEER Piping Design
Pro/ENGINEER Cabling Design

Pro/ENGINEER Workgroup Data Management

Pro/INTRALINK
Windchill Pro/ENGINEER Extension

Pro/ENGINEER Wildfire Modes

Pro/ENGINEER Conceptual and Industrial Design

- Pro/CONCEPT
- Pro/ENGINEER **Advanced Rendering**
- Pro/ENGINEER **Interactive Surface Design**
- Pro/ENGINEER **Reverse Engineering**

eDrawings for Pro/ENGINEER

Windchill

- Windchill PDMLink
- Windchill ProjectLink
- Windchill PartsLink
- Windchill Integrations
- Windchill MCAD & ECAD Integrations
- Windchill Enterprise Systems Integrations
- Windchill DynamicDesignLink
- Windchill ProductView

Pro/ENGINEER Wildfire Modes

Pro/MECHANICA (FEA)

Pro/DESKTOP

DIVISION

- DIVISION Mockup
- DIVISION ProductView
- DIVISION Reality

CADDS 5i

- CADDS 5i Modeling Foundation
- CADDS 5i Mechanical
- CADDS 5i **Shipbuilding**
- CADDS 5i Data Exchange
- CADDS 5i Optegra

Pro/ENGINEER Wildfire Modes

DIMENSION III

- Granite Interoperability Kernel
- Harmony
- InterComm
  - InterComm Expert
  - InterComm EDAcompare
  - InterComm EDAconduit

PLM Solutions

- Product Development System
- Product Lifecycle Management (PLM)
- Product First

References and Tutorials

- Pro/ENGINEER Tutorials at the Course Home page:
  http://www.me.uvic.ca/~mech410/proe_tutorials.html
- Pro/ENGINEER Manual - the manual can be 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.
- **Schroff Development Corp. (SDC) Professional Bookstore**

  http://www.schroff.com/

  Books for Pro/ENGINEER Wildfire
  - Introduction to Pro/ENGINEER Wildfire
  - Design Modeling with Pro/ENGINEER
  - Modeling with Pro/ENGINEER
  - Mechanical Engineering Design with Pro/ENGINEER
  - Parametric Modeling with Pro/ENGINEER
  - Pro/ENGINEER Tutorial & MultiMedia CD
  - Pro/ENGINEER Advanced Tutorial
  - Design Process Management using Pro/INTRALINK
  - Design for Manufacturing with Pro/MANUFACTURING
  - Pro/MANUFACTURING Tutorial
  - Pro/MECHANICA Structure Tutorial
  - Pro/MECHANICA Structure: Elements and Applications
  - Pro/MECHANICA Structure: Elements and Applications - Part 2
  - Pro/MECHANICA Motion - Mechanism Design and Analysis
  - An Introduction to Pro/SHEETMETAL
  - Applications in Sheet Metal: Using Pro/SHEETMETAL and Pro/ENGINEER

Pro/ENGINEER Wildfire Resource Center

The Resource Center appears each time when one starts Pro/ENGINEER Wildfire. It can be accessed at:

http://www.ptc.com/community/resource_center/proengineer/index.htm

The web site provides many useful Pro/ENGINEER Tools and Tutorials.

Three Key Sources of Pro/E Information:

1) Course website: http://www.me.uvic.ca/~mech410/
2) Pro/ENGINEER Wildfire Resource Center (above address)
   - Pro/ENGINEER Tools and Tutorials (next page).
3) Schroff Development Corp. (SDC) Professional Bookstore

Web address: http://www.ptc.com/community/resource_center/proengineer/index.htm#

Pro/ENGINEER Tools and Tutorials (1)

- Getting Started with Pro/ENGINEER Wildfire
- A tutorial-based introduction to creating parts, assemblies, and drawings in Pro/ENGINEER.
- Quick Reference Card - Pro/ENGINEER Wildfire 4.0
- Full color quick reference card as a handy reference for toolbars, selection and controls.
- Pro/ENGINEER Wildfire Menu Mapper
- Learn the menu paths of different versions.
- Personal Productivity Tools
- Tips to become more productive with Pro/ENGINEER.
- Pro/ENGINEER Wildfire 4.0 Tutorial for New Users
- Learn solid object modeling in a CAD environment.

Pro/ENGINEER Tools and Tutorials (2)

- …
- Parts Library
- View parts libraries, use free Pro/ENGINEER drawings and CAD models from leading suppliers and search for manufacturers.
- Demos, Tools & Tutorials
- These additional demos and tutorials were developed by product experts.
- What's New in Pro/ENGINEER
- Find out what's new in the latest release of Pro/E.

Discussion Forum

PTC University

Introduction to Pro/ENGINEER Wildfire 4.0 Tutorial

- Website: On-line Tutorial

  http://www.ptc.com/products/tutorials/

  Added to course web page.

- **Paperback Book:**
  Pro/Engineer Wildfire 4.0: Tutorial And Multimedia CD

Author: Roger Toogood
Format: Paperback, **384** pages
Publication Date: 2008
Publisher: Schroff Development Corp
http://www.sdcpublications.com/978-1-58503-415-4.htm
ISBN: 978-1-58503-415-4