An Introduction to Pro/E

25.353 Lecture Series

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Pro/Engineer

- Developed by Parametric Technology Corporation (PTC)
- A computer graphics system for modeling various mechanical designs and
- For performing related design and manufacturing operations
- Pro/ENGINEER is a *feature-based, parametric solid modeling* system comprehensively supporting *design and manufacturing applications*.

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Unique Features of Pro/E

- Associativity
- Parametric, feature-based modeling
- A comprehensive package
- Leading technology
  - behavior modeling
  - concurrent design supported by internet
  - ...

Parametric Technology Corp.

Key industry partners: Airbus Consortium (Aerospatiale, British Aerospace, and Daimler-Benz Aerospace), Rolls-Royce Aerospace Group, BMW, FORD, Boeing, Harris and Mitsubishi and PSA Peugeot Citroen

Revenue: 1.01Billion (1998)

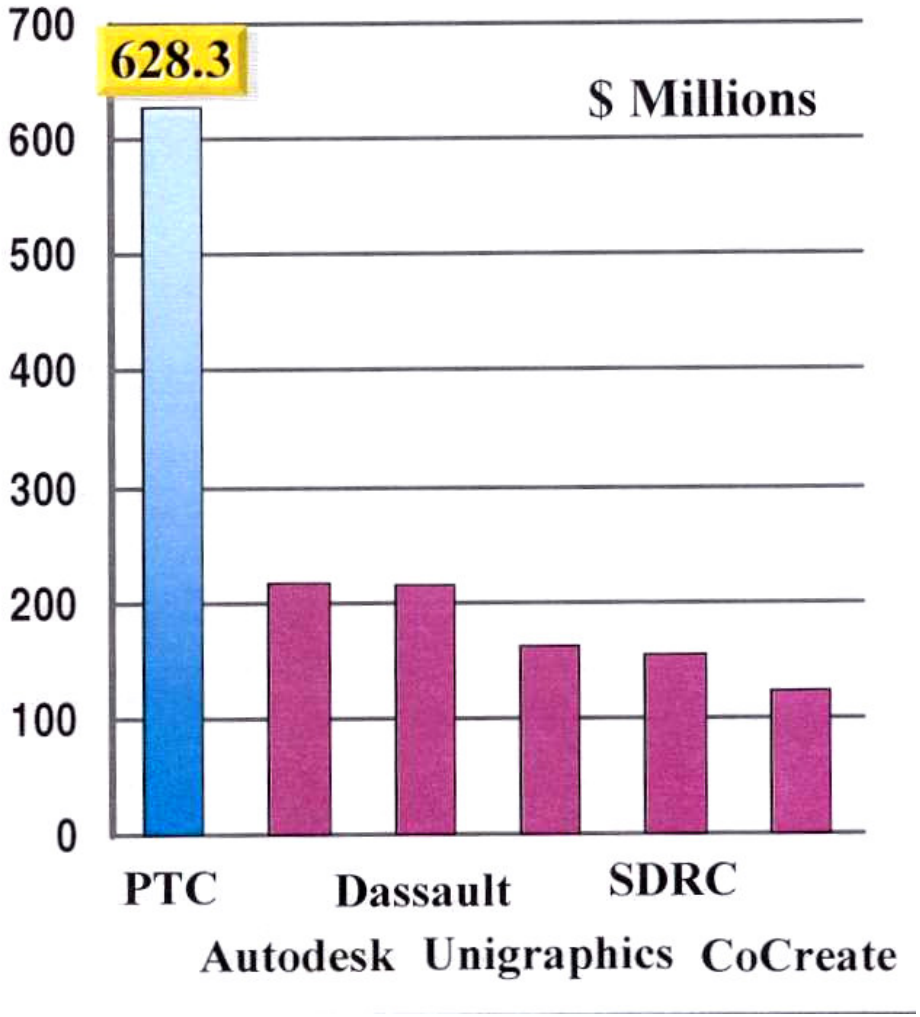
$660 million (2004)

Compare: Dassault $754.8 million (2003)(6 tools combined including CATIA, SolidWorks, ACIS, etc.)

Fact: The world's fifth largest software company (1998)

	Revenue	Business
1. Mircrosoft	$12.8 B	OS and Office Automation
2. Computer Associates	4.5 B	Networking
3. Oracle	4.4 B	Database
4. SAP	2.3 B	ERP
5. PTC	1.01 B	Product Development
6. Novell	930 M	Networking
7. Adobe	912 M	Graphics & Web Publishing
8. Sybase	904 M	Database
9. Cadence	755 M	ECAD
10. Peoplesoft	706 M	Business Applications

1998 New License Revenues



User Interface (Windows, Menus)

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Function Modules of Pro/E

MODE	DESCRIPTION
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.
Analysis	Structural, Motion, Thermal analysis
Manufacture	Define the machining operations that are required to manufacture a part modeled using Pro/ENGINEER.

Core: Part Modeling

Other Pro/E Modules

Sheet Metal	Create solid models of sheet metal parts and develop the NCL data necessary to manufacture them (with Pro/SHEETMETAL).
Mold	Create and analyze molds and moldings (with Pro/MOLDESIGN).
Layout	Create 2-D conceptual assembly sketches (with Pro/NOTEBOOK).
Dieface	Design and analyze the contact surfaces of stamping dies for forming deep-drawn sheet metal parts (with Pro/DIEFACE).

Some Basic & Unique Concepts of Pro/E

1. Modeling vs. drafting
2. Base, Datum, Sketch, Reference features
3. Parent-child relationship

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1. Modeling vs. Drafting

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

2. Datum Features

- Three default datum planes
- Other datum points, lines, and surfaces

3. Parent-Child Relationship

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Quick Questions

- Pro/E creates a drawing first and then constructs a 3-D object from the drawing.
- Datum planes in Pro/E are very useful tools in modeling and assembly.
- Parent-child relationship gives designers the capability to maintain his/her design intent.
- The base feature is the first feature one uses to define other features, it is thus very important.
- Use three short phrases to describe Pro/E
- There are various ways of modeling a component; normally they are equally acceptable.
- In Pro/E, a sketched 2-D section is meaningful only when it is used to construct a 3-D solid.
- The most fundamental component of Pro/E is its drawing tool.

Summary

- Pro/Engineer, PTC
- Features of Pro/Engineer
- Function Modules
- User interface (windows, menus, mouse)
- Modeling vs. Drafting
- Various Features
- Parent-child relationship