Pro/Engineer

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“A **feature** based, **parametric**, solid 3D geometric modeling CAD/CAM package that enables a user to develop all aspects of a product design in a concurrent manner.”
Feature

- Parts are created as combination of engineering features
- Features are specified based on setting the values and attributes of elements
- Features have vertices, edges, surfaces, volume and mass properties
  - Protrusion, holes, cuts …
Parametric

- The physical shape of the part or assembly is driven by the value assigned to the attributes (primarily dimensions) of its features.
- May change dimensions or attributes at any time
- Relations
Solid Modeling

- Model has volume and can have mass if density is specified
- Surfaces are automatically created when removing material
- May only create realizable objects
Pro/E System Organization

Pro/E system is divided into a number of specialized modes including:
- Part - 3D
- Assembly
- Drawing
- Manufacturing
- Sketcher – 2D
Example Assembly
Pro/E Modes

- Part
- Drawing
- Manufacturing
- Assembly
Datum Planes in Pro/E

- Datum planes are used as references for constructing the model.
- Datum planes are not geometry features but act as reference when you sketch a feature, orient the model, assemble components, etc.
- 2 sides: Yellow + Red, depending on which side is facing the screen.
- Arrow Direction: ◯ ◯
Parent/Child Relations

- A child feature is one that references a previously created parent feature

- A change in a parent feature may potentially affect a child feature
Pro/E Storage

- Storage during a session
  - Set Working Directory – CAE etc.
- Trail files - Keeps track of everything you do during a session. If you exit w/o saving, you can play a trail file to get back what you lost
- Delete Old Versions
Pro/E Tutorial

- Spin, Pan, Zoom
- Part Creation
  - Units
  - Sketching
  - Dimensioning
  - Extruding
  - Cutting
  - Revolving
  - Editing
- Saving
  - .prt
  - .igs