PTC CREO Parametric
(Formerly named Pro/Engineer)
Introduction
MAE-205
Pro/ENGINEER Wildfire Basic Modeling Process

The Basic Modeling Process can be summarized in four high-level steps:

1. Preparing for Part Model Design
2. Creating a New Part Model
3. Creating a New Assembly by Assembling the Part Models
4. Creating a Drawing of the New Part Model
Pro/ENGINEER Wildfire enables you to create solid model representations of your part and assembly models. Solid Models:

- Are realistic visual representation of designs.
- Contain properties such as mass, volume, and center of gravity.
- Can also be used to check for interferences in an assembly.
PTC’s CREO Parametric Fundamentals

Concepts
- Solid Modeling
- Feature-Based
- Parametric
- Parent / Child Relationships
- Associative
- Model Centric
Pro/ENGINEER Wildfire Fundamentals (cont.)

**Solid Modeling**
- Contain properties such as mass, volume, and center of gravity.
- Can also check for interferences in an assembly.

**Feature-based**
- Build one simple feature at a time.
- Each new feature can reference previous features.
PTC’s CREO Parametric Fundamentals (cont.)

Parametric
- Model geometry is controlled by parameters & dimensions.
- Modifying values updates geometry.

Parent/Child Relationships
- Features referenced during creation become parents.
- If parents change then child updates.
PTC’s CREO Parametric Fundamentals (cont.)

**Associative**
- Change in one mode will be reflected in the other modes automatically.

**Model-Centric**
- The model is the central source of design information.
- Part models are used to create assemblies.
- Drawings are quickly created from models.
In CREO Parametric the model is the center of all downstream deliverables.

Model-Centric

- Assemblies reference the models being assembled.
- The drawing references the model being documented.
- The FEM model references the model being meshed.
- The mold tool references the model being molded.
Recognizing File Extensions

Each CREO Parametric object type has a unique file extension used to identify it.

The following are common CREO file extensions:
- .prt - Part File
- .asm - Assembly File
- .drw - Drawing File
Understanding the Main Interface

The Main Interface includes the following areas:

- Graphics Window
- Main Menu
- Toolbars
- Dashboard
- Message Window
- Dialog Boxes
- Menu Manager
PTC’s CREO Parametric Interface (cont.)

Navigator
- Model Tree

Web Browser
- (Minimized)

Dashboard
- Tabs
- Icons

Menu Manager

Dialog Box
Understanding the Folder Browser

The Folder Browser is a pane in the Navigator that enables you to browse the folders on your computer and network.

The Folder Browser is divided into:
- Common Folders
- Folder Tree

The Folder Browser enables you to:
- Browse folders.
- View In Session objects.
- View contents of your Desktop, My Documents, and Network Neighborhood.
- Browse directly to the Working Directory.
- Resize the width by dragging the sash control.
- Click the sash arrows to close the Navigator.
Understanding the Window Menu

The Window Menu contains commands for activating, opening, closing, and re-sizing Pro/ENGINEER widows. You can also switch between open windows.

- A window must be active to use all applicable Pro/ENGINEER features.
- The word Active appears on the title bar of the active window next to the model name.
- The active model has a black dot next to its name in the Window menu.

Switching Between Open Windows

An Active Window
Setting the Working Directory and Opening and Saving Files

The Working Directory is the location for opening files from and saving new files to.

- CREO Parametric is started in default working directory.
- Different working directories can be set.
- New working directory locations are not saved upon exiting.

Set Working Directory in Folder Tree

Working Directory Common Folder
CREO Parametric is a memory-based system.

- Opening & Saving
  - Working Directory
  - In Session memory (RAM)
  - My Documents
- Erasing Memory (RAM)
  - Erase - Current
  - Erase - Not Displayed
- Deleting Models
  - Delete - All Versions
  - Delete - Old Versions
- File Extensions
  - *.PRT, *.ASM, *.DRW
  - Part.1 Part.2 Part.3
- Renaming Models
  - Rename On Disk and In Session
  - Rename In Session
Understanding Basic Display Options

Display is controlled independently for the following datum options:
- Datum Planes
- Datum Axes
- Datum Points
- Coordinate Systems

There are four different model display options:
- Shaded
- No Hidden
- Hidden Line
- Wireframe

Repaint redraws the screen.
Orienting Design Models

Manipulate the 3-D orientation of your design models in the CREO Parametric graphics window.

- Spin – Middle mouse button.
- Pan – SHIFT + middle mouse button.
- Zoom – CTRL + middle mouse button.
- Turn – CTRL + middle mouse button (move horizontally).

Orient Modes

- Anchored, Delayed, and Velocity

Quick Reference Card

- Provides definitions and shortcuts for the following points of interest.
  - Toolbar icons and functions
  - Dashboard icons and functions
  - Mouse and keyboard combinations for orienting design models
Understanding the View Manager

The view manager is a powerful content-sensitive dialog box that enables you to edit how a model displays in the graphics window.

Use the view manager to create and manage:
- View orientations
- Style states
- Cross-sections
- Explode states
The graphics window provides you with color-based feedback to mouse and keyboard inputs.

System Color Assignments in the graphics window:
- Cyan — Preselection Highlight
- Red — Selected Geometry
- Yellow — Preview Geometry

Keyboard and mouse actions perform different functions:
- Cursor Over (Preselection Highlight)
- Query to Next Item
- Select
- Clear Selection
Using Drag Handles

Drag handles are graphical objects used to manipulate geometry or components during creation or redefinition in real time.

Drag handles are used to:
- Resize geometry.
- Reorient geometry.
- Move geometry.
- Reference geometry.
- Adjust component offset.
- Access context-sensitive right mouse button options.
Editing Features and Regenerating

Edit enables you to alter dimensions of a selected feature or component. Edit Definition enables you to modify feature type, size, shape, location, references, or options.

Edit:
- Enter a value directly on the model.
- Use the Most Recently Used option.

Edit Definition using:
- Dashboard.
  - Preview Feature
  - Pause Feature
  - Resume Feature
- Drag handles.
- Context-sensitive right mouse button options.

Regenerate to update the model.

Undo / Redo capability.
Reviewing Sketcher Theory

A sketch is a 2-D entity that graphically captures an idea with lines, constraints, and dimensions.

2-D sketches are:
- Placed on a 3-D model.
- Used to create solid features.

2-D Sketch

Sketches are Used to Create Solid Features

Sketch Placed on a 3-D Model
Understanding Design Intent

Design Intent in Sketcher is to create, constrain, and dimension a sketch in a manner that will cause it to update predictably if modified.

Design intent is captured in sketches by:
- How it is constrained.
- How it is dimensioned.

Capture design intent by using the Intent Manager to:
- Maintain fully defined sketches at all times.
- Maintain weak/strong items.
- The following display options are available in Sketcher:
  - Dimensions
  - Constraints
  - Grid
  - Section vertices

Sketch Orientation reorients parallel to the screen.