

Pro|ENGINEER®  
Wildfire 2.0

# Chapter 1

## Introduction to Pro|Engineer Wildfire 2.0

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### 1.0 Content | Chapter 1

- What is Pro/E?
- Concept of Pro/E
- Design modes & Pro/E files
- Interface
  - Navigator | Browser | Graphics
- File management
  - Set working directory
  - Opening existing file
  - Creating new file
  - Save file, backup file & file iterations
  - Deleting files
- Printing
- View control (Mouse setting)
  - Spin, pan and zoom | Spin centre
  - Saved views
  - Display views
- Multiple windows & files in a session
  - Activate window
  - Erase memory content
- Selection (filters | lists)
- Exercise 1.0: Creating your first part using Pro/E
- Exercise 2.0: Creating a 2-feature part

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### 1.1 Pro/E???

- Pro/E is a 3D modeler software plus additional capabilities such as sheet metal and surface modeling, mechanism analysis and simulation, animation, photorender and more.
- Produced by PTC based in US, the first version produced since 1988.

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### 1.2 Concept

- Feature-based modeling
  - The model is built up using blocks called feature.
  - Features are the smallest building blocks in a part model.
  - Each feature are created using 3D operations such as extrude, revolve, hole, round, etc.
  - Other elements such as datum planes, axis are also considered as feature

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### 1.2 Concept

- Associative
  - The whole object are documented by creating part, assembly and drawing files.
  - All these are fully associative, where changes that are done in a part will be reflected in the assembly and drawing

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### 1.2 Concept

- Parametric
  - Features are interrelated or dependence, where modification done in one feature may produce changes in other features as well.
  - Design intent is very important.
  - Normally changes are done to the driving dimensions.
  - Relationship between feature, where one is being referred to another is known as parent/child relationship.

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## 1.2 Basic design modes

- General steps
  - Component parts of an object are created.
  - All the parts are assembled according to their position and orientation.
  - Technical drawings (2D) are created based on the assembly and part model.

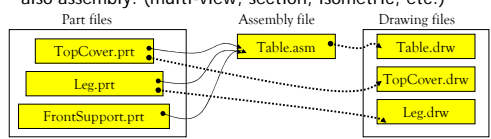
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## 1.3 Pro/E files

- Different file types in Pro/E:
  - PART files: most common & used to produce/model a single part of an object. One single component is drawn in a file.
  - ASSEMBLY files: multiple part files are grouped and arranged in this separate file to produce a complete object.
  - DRAWING files: used to generate 2D drawings of parts and also assembly. (multi-view, section, isometric, etc.)



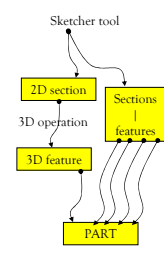
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## 1.3 Pro/E files: part mode

- Most design starts with creation of part
- The file extension is \*.prt
- Each single part are created from features - protrusion & cut of extrude, revolve, sweep, etc.
- Each feature will start with 2D profile or section and assign to 3D operation (extrude, revolve, etc)
- 2D section is created in *Sketcher* tool, similar to 2D drafting software - having tool such as line, circle, trim, etc.
- *Dashboard* is a part of Sketcher allowing access to various tools and commands.



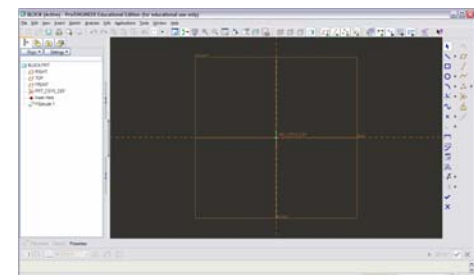
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## 1.3 Pro/E files: part mode: Sketcher

- Need to master *Sketcher* to produce robust model.




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## 1.3 Pro/E files: assembly mode

- After all parts have been created, a new file (assembly) may be created.
- In assembly file, individual parts are loaded up one by one, and being applied constraints.
- Constraints are how each parts are associated to one another.
- Examples of constraint: mate & offset.
- Exploded assembly can be created after all parts have been attached together.
- Also allow top-down design.



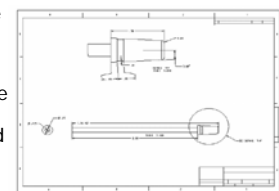
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## 1.3 Pro/E files: drawing mode

- 2D technical drawing is created in Drawing mode.
- New drawing file need to be created and the respective part or assembly is loaded.
- Multi-view, isometric, and section view may be produce in this mode.
- Dimension can be shown and hide easily.
- The drawing produce is associated with the original part file. Any modification done in part file, will be reflected in the drawing file.




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### 1.3 Pro/E files

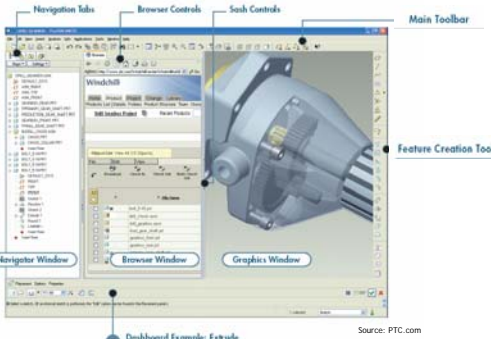
- Other file types: (screenshot)
  - FORMAT: normally used to create drawing layout.
  - SKETCH: to create 2D sketch file.
  - MANUFACTURING: to produce CNC codes.
  - REPORT
  - DIAGRAM
  - LAYOUT
  - MARKUP



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### 1.4 Interface




Dashboard Example: Extrude  
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### 1.4 Interface: Navigator

- Initially, it will show the folder in Windows Explorer style.
- Folder can be selected and files can be opened in this stage.
- Working directory can also be set by clicking the required folder and right click to choose Set Working Directory.
- When a file is opened, by default the navigator will show model tree.
- Model tree is where all the features of the part will be shown.
- Layer tree and others may be toggled and shown.



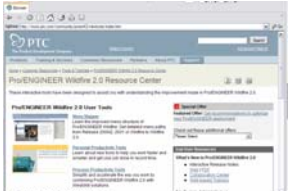
Source: PTC.com

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### 1.4 Interface: Browser

- Pro/E has an embedded browser.
- When it is first started, browser will show PTC website (or local files) in the main area of the screen.
- If any folder in navigator is clicked, the browser will show the content of that folder.

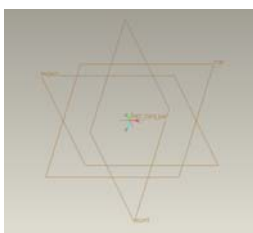


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### 1.4 Interface: Graphic window

- This is the main modeling window.
- Just after a new file is started, it will show 3 datum planes.
- Model are created here, and you can use spin, zoom or pan in this area.




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### 1.4 Interface: Toolbars

- Main toolbar is located at the top of the window - mainly consist of file operations tool and view controls.



Source: PTC.com

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## 1.4 Interface: Toolbars

- Drawing toolbar will appear underneath the main toolbar in Drawing Mode.

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## 1.4 Interface: Toolbars

- Feature creation toolbar is located at the right hand side. Consists of the tool to create feature such as extrude, revolve, datum plane, etc.
- The feature creation toolbar will be exchanged with Sketcher toolbar during sketching of 2D section.
- This toolbar will also be different when different types of files are opened. (assembly, drawing, etc.)

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## 1.4 Interface: Toolbars

- Feature creation toolbar
  - Datum
  - Assembly (in assembly file)
  - Pick/Place ()
  - Base
  - Editing

Source: PTC.com

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## 1.4 Interface: Toolbars

- Sketcher toolbar
- Will be displayed in drawing mode

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## 1.4 Interface: Dashboard

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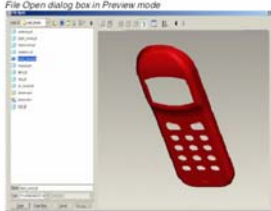
## 1.5 File management

- Set working directory
  - When you save or open a file, Pro/E will look in the default folder which is called *working directory*.
  - It is advisable that you change the working directory to your preferred folder *every time you start a session*.
  - To change the working directory, either
    - Under file menu: File > Set Working Directory... and choose the preferred folder, or
    - In the navigator window: Right click on the preferred folder and select Set Working Directory.
  - It is important to set one working directory when you work with multiple part files, with assemble and drawings as they are being referred to one another.

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## 1.5 File management

- Opening files
  - From file menu: File > Open
  - Click the open file icon in the main toolbar
  - You may preview the file before opening



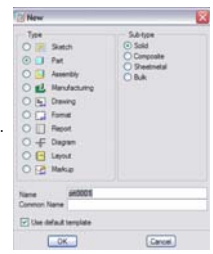
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## 1.5 File management

- Creating new file
  - Under file menu: File > New..
  - Click on the new file icon on the main toolbar.
  - You need to select application type and sub-type if necessary.
  - Type in the filename and click OK button.



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## 1.5 File management

- Save file, backup file & file iterations
  - Use File > Save to save changes to the file
  - Use File > Save a copy.. to create a backup file or the file in another name.
  - Using Save a copy is different from other "Save As.." command.
  - Whenever you use Save command, a new iteration of the file will be created (it would not overlap) e.g.
    - Block.prt.1 and Block.prt.2 and Block.prt.3 and so on..
  - When you open the file, it will open the latest version (with the biggest iteration number at the back)

Source: PTC.com

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## 1.5 File management

- Deleting files
  - Use File > Delete to permanently removes file from disk.
  - Use File > Delete > Old versions to permanently removes the older iterations of your files and only leaves the most recent file.

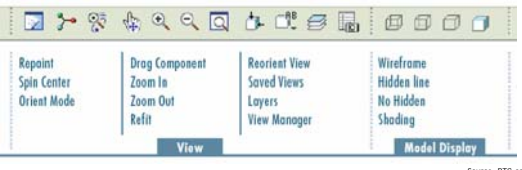
Source: PTC.com

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## 1.6 View control

- View control (Mouse setting)
  - Spin, pan and zoom | Spin centre
  - Saved views
  - Display views



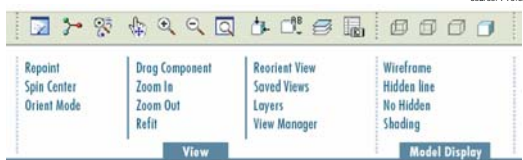
Source: PTC.com

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## 1.6 View control

- Spin: press middle mouse button & move mouse.
- Pan: press middle mouse button + SHIFT key & move mouse.
- Zoom: press middle mouse button + CTRL key & move mouse (or use mouse scroll).
- CTRL+D: will reorient the part to default view.
- p/s You MUST use 3-button mouse, otherwise it will be very difficult to use Pro/E*





Source: PTC.com

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## 1.6 View control

- Zoom in & Zoom out icon may also be used.
- Clicking the  will enable you to define the window to be zoomed.
- 'Zoom all' can be achieved by clicking the Refit icon.
- This will make all the part be fitted in one window.

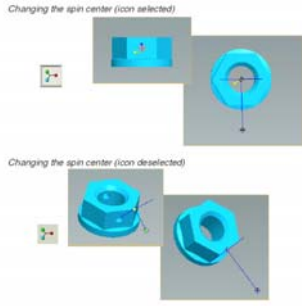


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## 1.6 View control

- Spin centre



Source: PTC.com

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## 1.6 View control

- Display view




Source: PTC.com

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## 1.7 Multiple windows & files









- If you open more than one files (or windows) in one session, you can only work in one at a time (the active window).
- When you switch window, you need to activate the new window.
- This is done by selecting Window Menu and select Activate.
- To close a file (or window)
  - Use *File > Close* to close the active window
  - Use *File > Erase > Current* to close window and remove it from memory
  - Use *File > Erase > Not displayed* to show the list of file in memory

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## 1.8 Making Selection

Making Selections

Mouse Controls		
Highlight Geometry		Over Geometry
Query to Next Item		Until Highlighted
Select Highlighted Geometry		
Add or Remove Items from Selection	 + 	
Construct Chains of Surface Sets	 + 	
Clear Selection		On Background




Source: PTC.com

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## 1.8 Colour assignment

System Color Assignments

Cyan		<b>Preselection Highlight</b> Item will be added to or removed from the set of selected items
Red		<b>Selected Geometry</b> Items currently selected
Yellow		<b>Preview Geometry</b> Results of the current operation when complete

Source: PTC.com

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## 1.8 Using filters

The screenshot shows a selection list with the following items: Smart, Parts, Features, Geometry, Datums, Quilts, Annotation, and Smart. A '1 selected' indicator is shown at the top of the list. A 'Smart Filter' dialog box is open, showing a 'Smart' filter selected. The dialog box has a green checkmark icon and a dropdown menu. A 'Filters' section is also visible, with a 'Smart' filter selected. An 'Active Filter' section is also present.

**Smart Filter**  
(2-level filter)  
EXAMPLE: Select a **Feature** first, then select **Geometry** (Surface/Edge/Vertex) from the Feature

**Filters**  
Limit the scope of Selection

**Active Filter**  
TIP: Double-click to view items in Selection list

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# End of Chapter 1

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