Lesson 1 Pro/ENGINEER Wildfire 3.0 Overview

Figure 1.1 Pin, Fitting, Assembly, Pin Detail, Assembly Drawing

OBJECTIVES

- Create two parts
- Assemble parts to create an assembly
- Create a part drawing
- Create an assembly drawing

Pro/ENGINEER Wildfire 3.0 Overview

This lesson will allow you to experience the part, assembly, and drawing modes (Fig. 1.1) of Pro/ENGINEER Wildfire 3.0 by creating two simple parts (Figs. 1.2 and 1.5), assembling them, and creating drawings.

A bare minimum of explanation is provided here. This lesson will quickly get you up and running on Wildfire 3.0.
**Lesson 1 STEPS**

**Creating the Pin Part**

In this first lesson most commands and picks will be accompanied by the window or dialog box that will open as the command is initiated. After Lesson 1, the commands will not show every window and dialog box, as this would make the text extremely long. Appropriate illustrations will be provided.

*Throughout the text, a box surrounds all commands and menu selections.*

Open **Pro/ENGINEER Wildfire 3.0** using a shortcut icon on your Desktop (or with **WINDOWS**, click: *Start* ⇒ *Programs* ⇒ *proewildfire3.0*) Pro/E will open on your computer.

- **Open Pro/ENGINEER**
- **File** ⇒ **Set Working Directory** ⇒ select the desired working directory or accept the default directory [Fig. 1.3(a)] ⇒ **OK** ⇒ Create a new object from Top Toolchest ⇒ **Part** ⇒ Name PIN ⇒ **Use default template** [Fig. 1.3(b)] ⇒ **OK** [Fig. 1.3(c)]
Figure 1.3(a) Select Working Directory Dialog Box

Figure 1.3(b) New Dialog Box

Figure 1.3(c) Model Tree and Graphics Window (default datum planes and coordinate system)
Click: **Extrude Tool** from Right Toolchest [Fig. 1.4(a)] ⇒ Placement tab ⇒ Define

**Figure 1.4(a)** Extrude Dashboard Opens

**Figure 1.4(b)** Define an Internal Sketch
Select the **FRONT** datum plane from the graphics window (or Model Tree) [Fig. 1.4(c)]. The FRONT datum is the Sketch Plane and the RIGHT datum is automatically selected as the Sketch Orientation Reference [Fig. 1.4(d)].
Click: **Sketch** [Fig. 1.4(e)] ⇒ Create circle by picking the center point and a point on the circle from Right Toolchest [Fig. 1.4(f)]

**Figure 1.4(e)** Activate the Sketcher

**Figure 1.4(f)** Select the Circle Tool
Pick the origin for the circle’s center [Fig. 1.4(g)] ⇒ pick a point on the circle’s edge [Fig. 1.4(h)] ⇒

MMB (Middle Mouse Button) [Figs. 1.4(i-m)] ⇒ double-click on the dimension ⇒ type 1.00 ⇒ Enter

Figure 1.4(g) Pick the Circle’s Center

Figure 1.4(h) Pick a Point on the Circle’s Edge

Figure 1.4(i) Circle and Dimension

Figure 1.4(j) Double-click on the Dimension

Figure 1.4(k) Type 1.00

Figure 1.4(l) Completed Sketch
Click: Continue with the current section from Right Toolchest ⇒ Click in value field [Fig. 1.4(n)] ⇒ type 5.00 [Fig. 1.4(o)] ⇒ Enter ⇒ → ⇒ Standard Orientation [Fig. 1.4(p)]

Figure 1.4(m) Dynamic Preview of Geometry

Figure 1.4(n) Extrude Dashboard Depth Value Field

Figure 1.4(o) New Depth Value

Figure 1.4(p) Standard Orientation
Click: Refit object to fully display it on the screen from Top Toolchest ⇒ hold down the Ctrl key and your MMB, move your mouse downward to zoom in [Fig. 1.4(q)] [if you have three-button mouse with a thumb wheel, simply rotate (not click) the thumb wheel to zoom] ⇒ click MMB [Fig. 1.4(r)] ⇒ File ⇒ Save [Fig. 1.4(s)] ⇒ MMB Model Tree displays the Extrude feature [Fig. 1.4(t)]

You have completed your first Pro/ENGINEER component.
Creating the Plate Part

In this first lesson most commands and picks will be accompanied by the window or dialog box that will open as the command is initiated. After Lesson 1, the commands will not show every window and dialog box, as this would make the text extremely long. Appropriate illustrations will be provided.

If you are continuing from the last section, then skip this set of commands:

Open **Pro/ENGINEER Wildfire 3.0** using a shortcut icon on your Desktop ⇒ File ⇒ Set Working Directory ⇒ select the working directory or accept the default directory (make sure this working directory is the same directory where the PIN was saved) ⇒ OK
Click: Create a new object ⇒ Part ⇒ Name PLATE ⇒ OK ⇒ click Window from menu bar [Fig. 1.6(b)] (note: you now have two parts “in session”) ⇒ Activate ⇒ Save the active object (Hint: Pro/E does NOT save for you ever! Nor does it prompt you to save ever! So save often) ⇒ OK (or MMB) ⇒ [Fig. 1.6(c)]
Click **Tools** from menu bar [Fig. 1.6(d)] ⇒ **Environment** [Fig. 1.6(e)] ⇒ (your system may have this already set as the default) ⇒ **Apply** ⇒ **OK**
Pick/select the **FRONT** datum plane [Fig. 1.7(a)] ⇒ Extrude Tool ⇒ RMB (Right Mouse Button) ⇒ Define Internal Sketch [Fig. 1.7(b)] ⇒ Sketch [Fig. 1.7(c)]

**Figure 1.7(a)** Plate

**Figure 1.7(b)** Plate

**Figure 1.7(c)** Sketch Dialog Box
Click: **Insert foreign data from Palette into active object** [Fig. 1.7(d)] ⇒ scroll down to see the octagon ⇒ double-click **Octagon** [Figs. 1.7(e-f)]

**Figure 1.7(d)** Insert foreign data from Palette into active object

**Figure 1.7(e)** Sketcher Palette

**Figure 1.7(f)** Polygons: Octagon
Place the octagon on the sketch by picking a position [Fig. 1.7(g)] \(\Rightarrow\) with the LMB, drag and drop the center of the octagon at the origin [Fig. 1.7(h)] \(\Rightarrow\) modify Scale value to 2 \(\Rightarrow\) Enter \(\Rightarrow\) from the Scale Rotate dialog box.

**Figure 1.7(g)** Place Octagon anywhere on the Sketch by picking a Position

**Figure 1.7(h)** Drag and Drop the Octagon at the Origin of the Part (PRT_CSYS_DEF)
Click: **Close** the Sketcher Palette ⇒ [Fig. 1.7(i)] ⇒ from Right Toolchest [Fig. 1.7(j)] ⇒ modify the depth in the dashboard to **2.00** [Fig. 1.7(k)] ⇒ Enter ⇒ [Fig. 1.7(l)] ⇒ **Standard Orientation** from Top Toolchest [Fig. 1.7(i)] ⇒ from dashboard [Fig. 1.7(m)] ⇒ LMB to deselect

**Figure 1.7(i)** Octagon Sketch

**Figure 1.7(j)** Octagon Dynamic Preview

**Figure 1.7(k)** Modify Depth to **2.00**

**Figure 1.7(l)** Orientation

**Figure 1.7(m)** Extruded Octagon
Click: **Hole Tool** from Right Toolchest ⇒ pick on the top surface [Fig. 1.8(a)] ⇒ **Hidden Line** from Top Toolchest [Fig. 1.8(b)]
Click: RMB [Fig. 1.8(c)] ⇒ **Secondary References Collector** ⇒ press and hold the **Ctrl key** ⇒ pick the **TOP** datum plane from the Model Tree ⇒ with the **Ctrl key** still pressed select the **RIGHT** datum plane from the Model Tree [Fig. 1.8(d)] ⇒ **Placement** tab from the dashboard [Fig. 1.8(e)]

**Figure 1.8(c)** RMB ⇒ **Secondary References Collector**

**Figure 1.8(d)** Select the **RIGHT** and the **TOP** Datum Planes as Secondary References

**Figure 1.8(e)** Click Placement
In the Secondary references collector, click on **TOP** [Fig. 1.8(f)] ⇒ click on **Offset** ⇒ **Align** [Fig. 1.8(g)] ⇒ click on **RIGHT** [Fig. 1.8(h)] ⇒ click on **Offset** ⇒ **Align** [Figs. 1.8(i-j)]

**Figure 1.8(f)** Click on **TOP** ⇒ **Offset**  
**Figure 1.8(g)** Click **⇒** **Align**  
**Figure 1.8(h)** Click on **RIGHT** ⇒ **Offset**  
**Figure 1.8(i)** Click **⇒** **Align**  

**Figure 1.8(j)** Hole Preview
Click: **Placement** tab to close the Placement panel ⇒ [Fig. 1.8(k)] Drill to intersect with all surfaces ⇒ modify the hole’s diameter ⇒ Enter [Fig. 1.8(l)] ⇒ [Fig. 1.8(m)] Ctrl+S to save the part ⇒ OK

![Figure 1.8(k) Drill to intersect with all surfaces](image)

![Figure 1.8(l) Diameter 1.00](image)

![Figure 1.8(m) Completed Hole](image)
Click: Shading ⇒ View from menu bar [Fig. 1.9(a)] ⇒ Color and Appearance ⇒ ptc-metallic-gold ⇒ Apply [Fig. 1.9(b)] ⇒ Close the dialog box
You have completed your second Pro/ENGINEER component. In the next section you will put the parts together to create an assembly.
Creating the Assembly

Using the two simple parts just created you will now create an assembly (Fig. 1.11). Just as you can combine features into parts, you can also combine parts into assemblies. The Assembly mode in Pro/E enables you to place component parts and subassemblies together to form assemblies.

As with a part, an assembly starts with default datum planes and a default coordinate system. To create a subassembly or an assembly, you must first create datum features. You then create or assemble additional components to the existing component(s) and datum features.

If you are continuing from the last section then skip this set of commands:

Open Pro/ENGINEER Wildfire 3.0 using a shortcut icon on your Desktop ⇒ File ⇒ Set Working Directory ⇒ select the working directory or accept the default directory (make sure you set the same working directory as where your parts were saved) ⇒ OK
Click: [Create a new object] ⇒ [Assembly] ⇒ [Design] ⇒ [Name CONNECTOR] ⇒ [OK] ⇒ [Settings from Navigator] ⇒ [Tree Filters] ⇒ [toggle all on] ⇒ [Apply] ⇒ [OK] ⇒ [File] ⇒ [Save] ⇒ [OK]

Figure 1.12(a) New Dialog Box, Assembly

Figure 1.12(b) Tree Filters

Figure 1.12(c) Model Tree Items Dialog Box
Figure 1.12(d) Assembly Datums and Coordinate System

Click: Add components to the assembly from Right Toolchest ⇒ plate.prt [Fig. 1.13(a)] ⇒ Open

Figure 1.13(a) Open Dialog Box
Click: from Top Toolchest [Fig. 1.13(b)] ⇒ Auto ⇒ Default [Fig. 1.13(c)] ⇒

**Figure 1.13(b) Assembly Dashboard**

**Figure 1.13(c) Default Constraint**
Click: from dashboard [Fig. 1.13(d)] ⇒ Ctrl+S ⇒ OK ⇒ in the Model Tree expand ⇒ expand ⇒ expand [Fig. 1.13(e)]

**Figure 1.13(d) Fully Constrained Component**

**Figure 1.13(e) CONNECTOR Assembly**
Click: **Add components to the assembly** from Right Toolchest ⇒ **pin.prt** ⇒ **Preview** >>> [Fig. 1.14(a)] ⇒ **Open** [Fig. 1.14(b)]

**Figure 1.14(a) Previewed Pin**

**Figure 1.14(b) Component Shown in Assembly Window with No Constraints**
Pick on the Pin’s cylindrical surface [Fig. 1.14(c)] ⇒ pick on the Plate’s cylindrical hole surface [Fig. 1.14(d)] ⇒ **Placement** tab from dashboard ⇒ **New Constraint** [Fig. 1.14(e)]
Figure 1.14(e) New Constraint
Pick on the Plate’s front surface [Fig. 1.14(f)]
Pick on the Pin’s end face surface [Fig. 1.14(g)]

**Figure 1.14(g)** Pick on the Pin’s End Face Surface
Click: [Fig. 1.14(h)] ⇒ [Fig. 1.14(i)] ⇒ type 0.50 as the offset distance

Figure 1.14(h) Status: Fully Constrained

Figure 1.14(i) Changing Offset Option

Figure 1.14(j) Offset 0.50
You have completed your first Pro/ENGINEER assembly. In the next section you will create a set of drawings for the Plate component and the Connector assembly.
Creating Drawings

Using the parts and the assembly just created, you will now create a detail drawing of the Plate and an assembly drawing of the Connector (Fig. 1.15).

The Drawing mode in Pro/E enables you to create and manipulate engineering drawings that use the 3D model (part or assembly) as a geometry source. You can pass dimensions, notes, and other elements of design between the 3D model and its views on a drawing.

*If you are continuing from the last section then skip this set of commands:*

Open **Pro/ENGINEER Wildfire 3.0** using a shortcut icon on your Desktop ⇒ **File** ⇒ **Set Working Directory** ⇒ select the working directory or accept the default directory (make sure you set the same working directory as where your parts and assembly models were saved) ⇒ **OK** ⇒ **File** ⇒ **Open** ⇒ **Pin** ⇒ **File** ⇒ **Open** ⇒ **Plate** ⇒ **File** ⇒ **Open** ⇒ **Connector** [you now have all three objects (the two parts and the assembly) “in session”]
Click: Create a new object ⇒ Drawing ⇒ Name CONNECTOR_ASM ⇒ [Fig. 1.16(a)] ⇒ OK [Fig. 1.16(b)]
(Note: if the Default Model is not the CONNECTOR.ASM, then click: Browse ⇒ Preview >>> ⇒ connector.asm [Fig. 1.16(c)] ⇒ Open)

Figure 1.16(a) New Dialog Box, Drawing

Figure 1.16(b) New Drawing Dialog Box

Figure 1.16(c) Connector Assembly Preview
Click: OK with the default model set as [Fig. 1.16(d)] ⇒ RMB ⇒ Page Setup [Fig. 1.16(e)]
Page Setup dialog box displays [Fig. 1.16(f)] ⇒ click in the C Size field [Fig. 1.16(g)] ⇒ to see options [Fig. 1.16(h)] ⇒ Browse [Fig. 1.16(i)] ⇒ c.frm [Fig. 1.16(j)] ⇒ Open [Fig. 1.16(k)] ⇒ OK

Figure 1.16(f) Page Setup Dialog Box

Figure 1.16(g) C Size Field

Figure 1.16(h) Format Options

Figure 1.16(i) Browse

Figure 1.16(j) c.frm from the System Formats Library

Figure 1.16(k) Format C
off ⇒ RMB ⇒ Insert General View [Fig. 1.16(l)] ⇒ DEFAULT ALL [Fig. 1.16(m)] ⇒ OK ⇒ pick a position for the new view [Fig. 1.16(n)] ⇒ OK from the Drawing View Dialog Box [Fig. 1.16(o)] ⇒ Ctrl+S ⇒ OK the drawing is now complete [Fig. 1.16(p)] ⇒ LMB

Figure 1.16(l) Insert General View

Figure 1.16(m) Select Combined State

Figure 1.16(n) Pick a Position for the New View
Figure 1.16(o) Drawing View Dialog Box

Figure 1.16(p) Completed Assembly Drawing
Click: Create a new object ⇒ .Drawing ⇒ Name PLATE ⇒ Use default template [Fig. 1.17(a)] ⇒ OK [Fig. 1.17(b)] ⇒ Default Model Browse ⇒ plate.prt ⇒ Preview >>> [Fig. 1.17(c)] ⇒ Open

**Figure 1.17(a)** New Dialog Box, Drawing

**Figure 1.17(b)** Browse

**Figure 1.17(c)** Plate Preview
OK [Fig. 1.17(d)] ⇒ drawing opens ⇒ RMB ⇒ Page Setup [Figs. 1.17(e-f)] ⇒ click in the C Size field [Fig. 1.17(g)] ⇒ to see options [Fig. 1.17(h)] ⇒ Browse [Fig. 1.17(i)] ⇒ c.frm [Fig. 1.17(j)] ⇒ Open ⇒ OK formatted drawing is displayed [Fig. 1.17(k)]
Click: RMB ⇒ Insert General View [Fig. 1.17(l)] ⇒ pick a position for the new view ⇒ OK from the Drawing View Dialog Box [Fig. 1.17(m)]

Figure 1.17(l) Insert General View

Figure 1.17(m) Drawing View Dialog Box
Click: LMB ⇒ [Fig. 1.18(a)] ⇒ Open the Show/Erase dialog box ⇒ Show ⇒ [Fig. 1.18(b)] ⇒ Show all items ⇒ Yes [Fig. 1.18(c)] ⇒ Accept All [Fig. 1.18(d)] ⇒ Close ⇒ LMB ⇒ [Fig. 1.18(f)] ⇒ MMB

Figure 1.18(a) Show/Erase

Figure 1.18(b) Type: Axis

Figure 1.18(c) Show All

Figure 1.18(d) Confirm Dialog Box: Yes

Figure 1.18(e) Accept All
This concludes the overview of Pro/ENGINEER Wildfire 3.0.

Spend some time exploring the website www.cad-resources.com for downloads, materials, and other CAD related information. To get on the CAD-Resources email list for updates and lessons, send an email to cad@cad-resources.com.