INTRODUCTION TO
PRO/SHEETMETAL

WILDFIRE 4.0
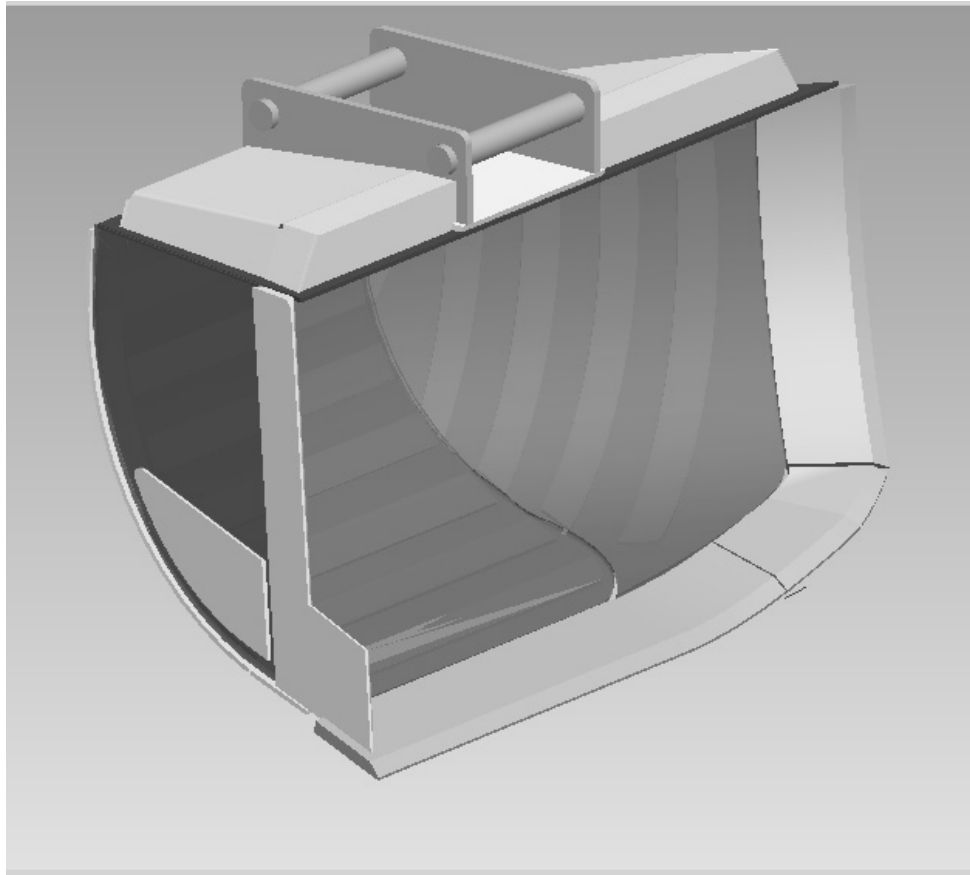
Sheetmetal Assembly of a 'Bucket'

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> **Estimated time: ½ hour**

Exercise 1a

First Wall Features

Objectives

At the end of this exercise, the user should be able to:

1. Create a new file using sheet metal type;
2. Create first wall features of sheet metal parts using the following options:
   - ➤ Extruded wall
   - ➤ Flat wall
3. Convert a solid model to a sheet metal model.

Designing in Sheetmetal Mode

Introduction

In my numerous years of CAD and MCAD experience, I have yet to see software as sophisticated as Pro/ENGINEER Wildfire Sheetmetal mode. The modeling features available to version 3.0 are simply amazing. Finally, a software package that thinks as an engineer or designer would. I am writing this manual for two reasons. Apart from earning some royalties, I have yet to see a decent manual that is as practical as I have been. The second reason is that there is no decent manual available in Wildfire 4.0.

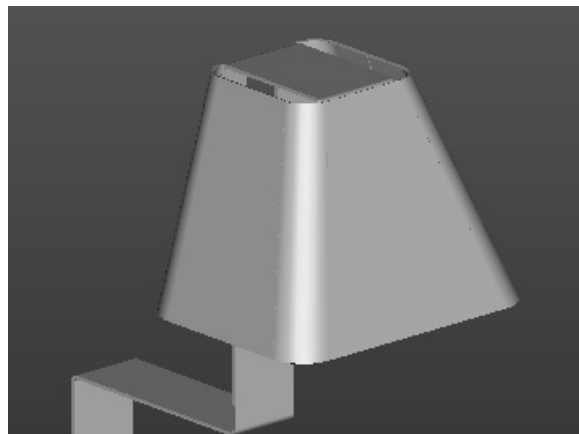Sheetmetal mode allows a designer to perform many sheet metal features. Examples would be:
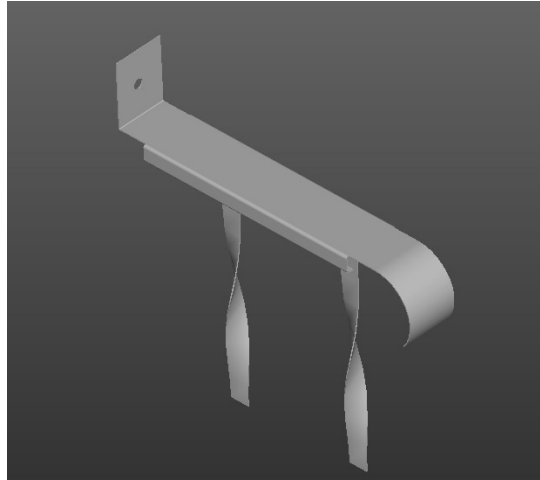
➢ Simple Wall Features (Extruded, Flat);
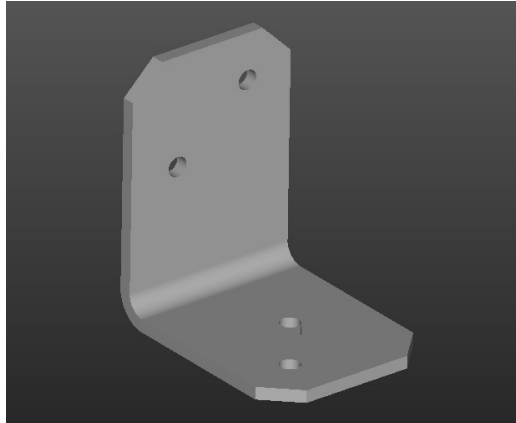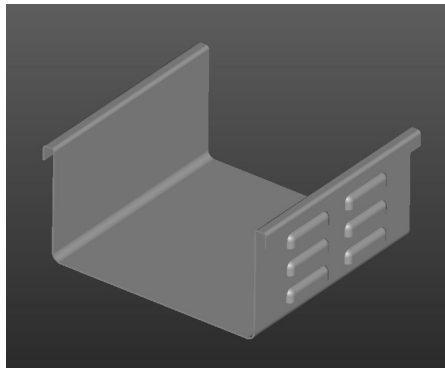

➢ Flat forms (unbend and bend back features);

➢ Advanced Wall Features (Revolve and blend);

> ➢ Bends (Angles, Rolls) and twists and rolls

> ➢ Louver and Forms

And not forgetting:

> ➢ Bend Tables and Bend Order Tables;
> ➢ Solid to Sheet Metal Conversion Features.

Introduction to Pro/ENGINEER Wildfire Sheetmetal

Feature creation in Sheetmetal mode is quite different from Part mode. While some of
the Part mode features (described as solid features) can be accessed from the Sheetmetal
mode menu, other features can be created, and all are related to sheet metal design.

The model building philosophy can also be quite different. Again, while some aspects of
Part mode are integrated within the Sheetmetal environment, one thing is always true: "*If
you don't put any thinking into your first (base) feature, you will run into trouble at some
point in your modeling process.*" Needless to say, sheet metal parts should be well
thought of ahead of time before the first feature (first wall) is created.

There are numerous ways in which a sheet metal user can create a first wall option in Sheetmetal mode. The options are:

1. In Sheetmetal mode, by creating an extruded section;

2. In Sheetmetal mode, by creating a flat wall;

3. In solid type Part mode, by creating a thin section, then converting it to sheet metal.

The following table gives a review and description of the basic wall feature types available in Pro/ENGINEER Wildfire 4.0.

TABLE 1: WALL FEATURE TYPES

Wall Feature Type	Description
Extruded	A sketched 2D profile with an offset wall thickness. Bend angles can be built into the 2D profile as in Project 1 (similar to Pro/ENGINEER's Thin Wall Protrusion)
Flat	A sketched boundary in a 2D plane (flat) at a specified bend angle (Similar to Pro/ENGINEER's Extrude Protrusion)
Extended	Creates an extension of a wall surface to another wall surface, usually **at a corner**.
Revolved	The cross-section of a revolved wall is sketched and rotated about an axis (similar to Solid mode Revolved Protrusion).
Blended	The cross-section of the blended wall is sketched, with other options such as parallel, rotational and general.
Offset	An offset sheet metal wall is created by offsetting from an existing solid surface.

While extruded and flat are the most common first wall features used, the other features prove to be quite useful in many applications.

Procedure

We need to first get organized before beginning the training files. A working directory needs to be created for locating all training files created in this manual. Most, if not all, files will be created from scratch, and will be used for more than one exercise. Using your browser, **create a folder** named '**sheetmetal_1**' in the location of your choice.

Launch Pro/ENGINEER Wildfire. Once it opens, a default browser is displayed on start up (as shown below). The default URL directs you to a PTC site that provides many Wildfire tools.
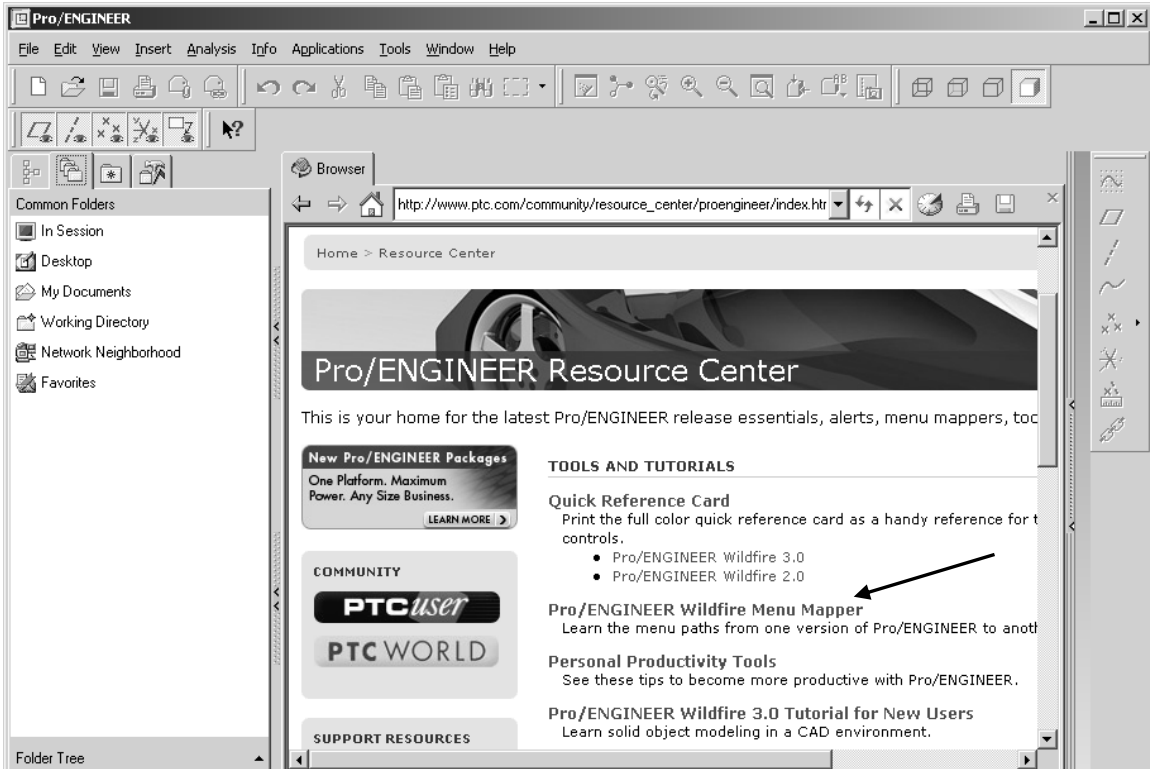
Figure 1

Select the Menu Mapper link as shown above. Menu Mapper provides you with a tool to find where Release 2001 menu options can be found in Wildfire. Using the Mapper, browse and find the location for Wildfire 4.0 to create a sheet metal wall (**select SheetMetal** under Release Wildfire 2.0 to Wildfire 3.0), then select **Insert > Sheet Metal Cut > Solid**, as shown below.
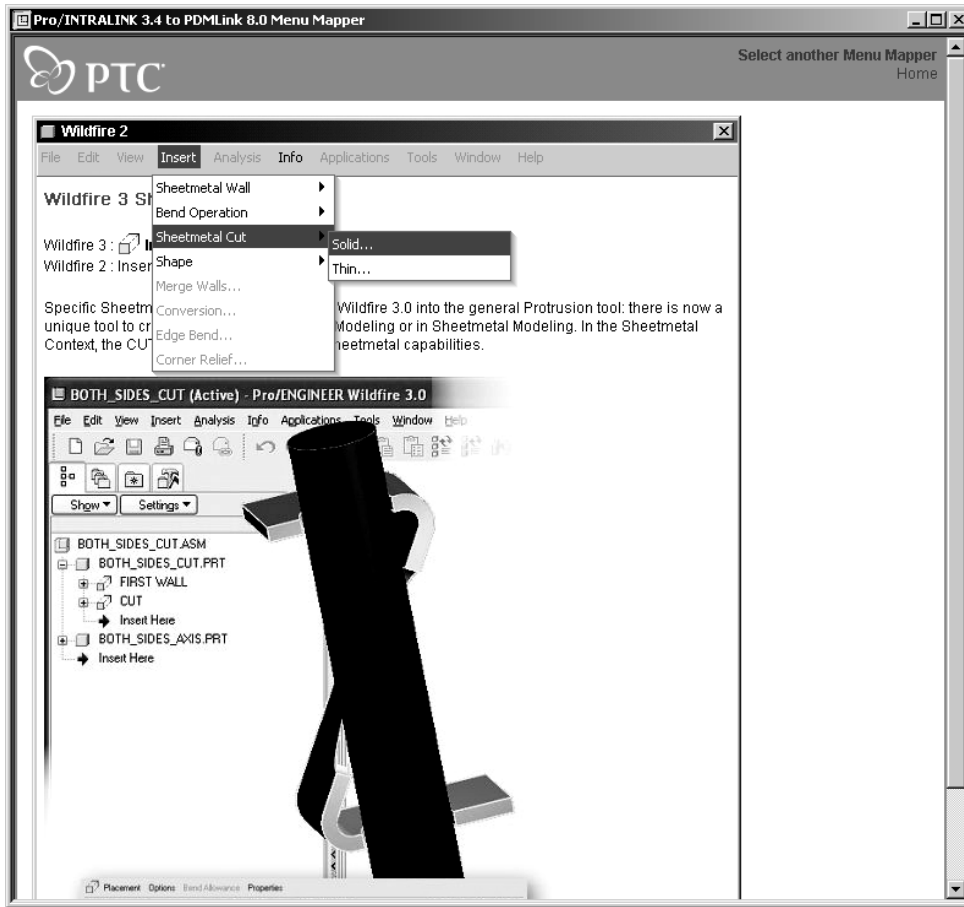
Figure 2

The Menu Mapper reports that the Wildfire menu is **Insert > Extrude** as shown below.

Figure 3

Close the Menu Mapper windows. Next, Select **Tools > Options**. The window shown in Figure 4 comes up.
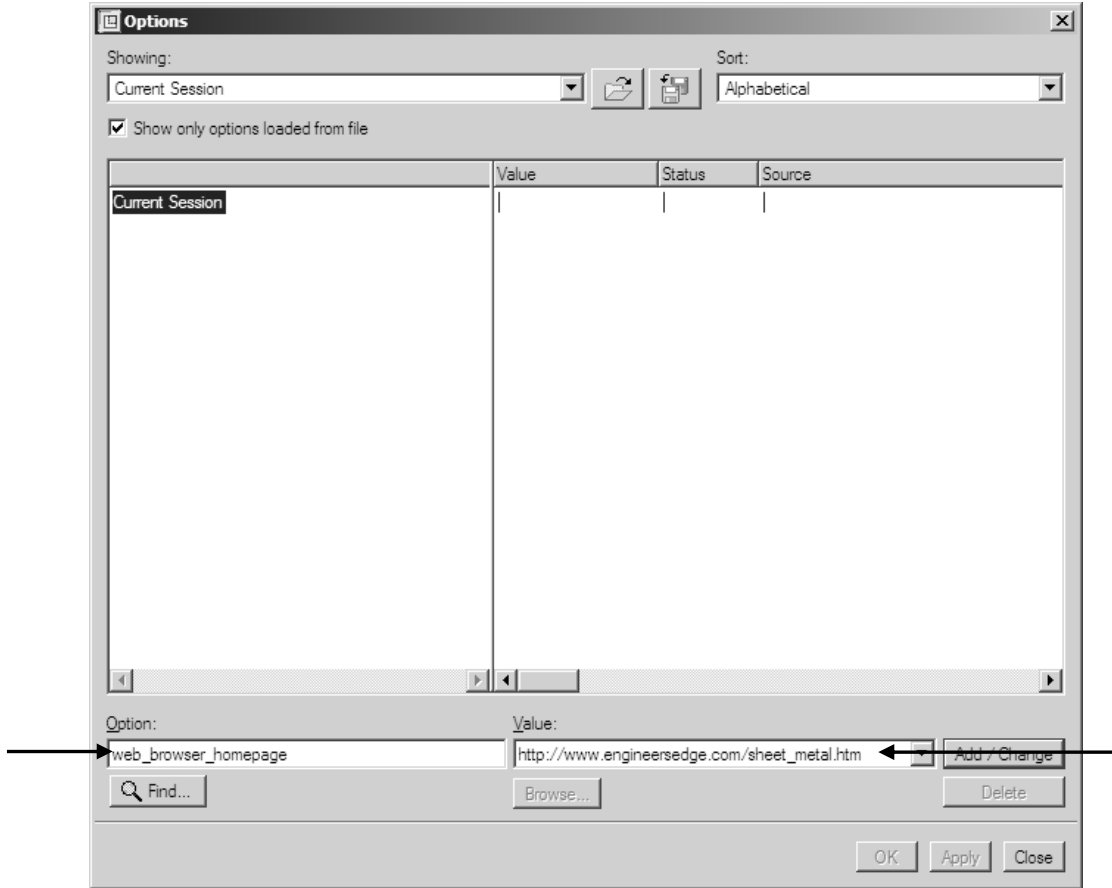
Figure 4

Enter the Option and Value as shown above. The website listed is a great website for sheet metal design research. If you have some time, browse through it for future reference.

Click **Apply – Close**. Next, select the home icon (it looks like a house) at the top of the browser window to access the home page that you previously gave. You can **compress the browser** by selecting the arrows along the right-hand edge of the browser as shown in Figure 5.

Figure 5

Your screen should now have a gray background. Next, click **Tools > Customize Screen**. Select the **Browser** tab and **de-select** "*Expand Browser by default while loading ProE*" as shown in Figure 6.
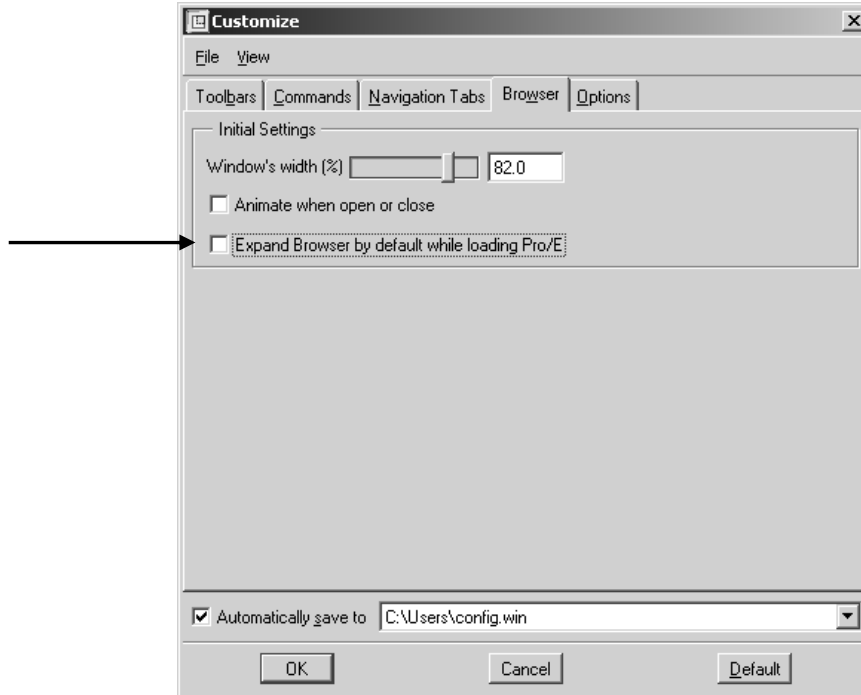
Figure 6

Select **OK**. Next, click on the working directory folder/icon on the left hand side of the window, then set **sheetmetal_1** (the folder you created earlier) as your working directory for the session, as shown below using the Pro/ENGINEER Wildfire Navigator.
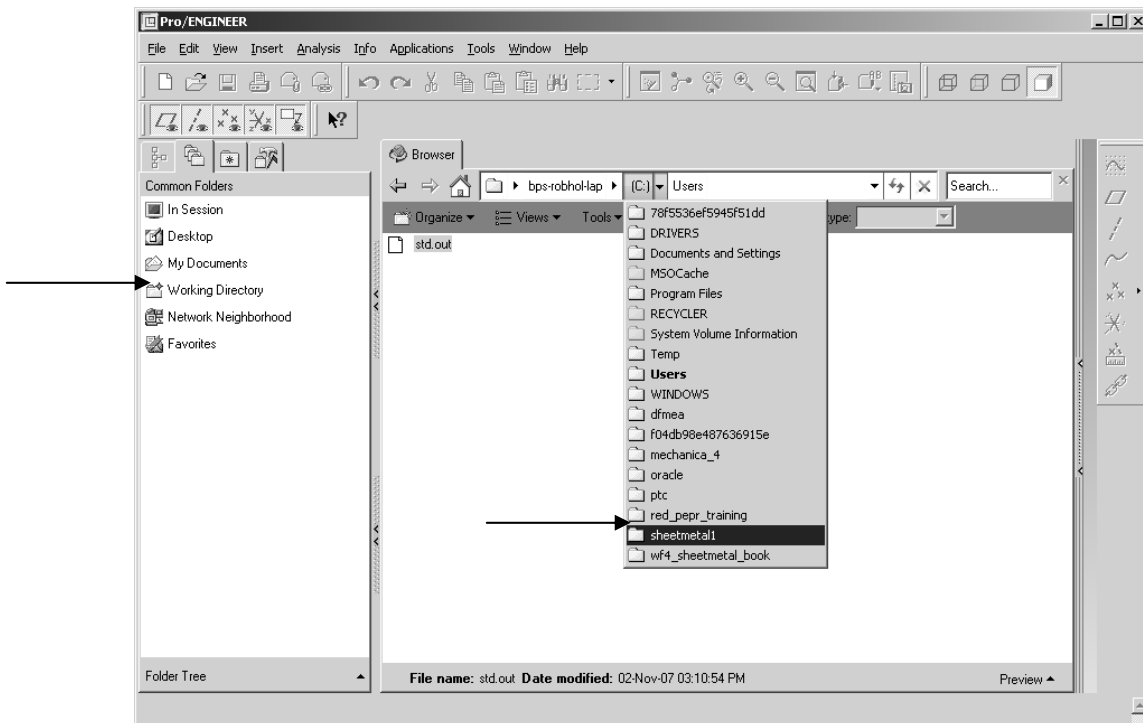

Figure 7

1. Creating a Wall Feature Using the Extruded Option

One of the ways you can create sheet metal parts is by creating the first feature representing the part in a formed state. This type of wall feature is used when the user wants to build one or more bends into a single feature. An example of a formed part created using this method is shown below.
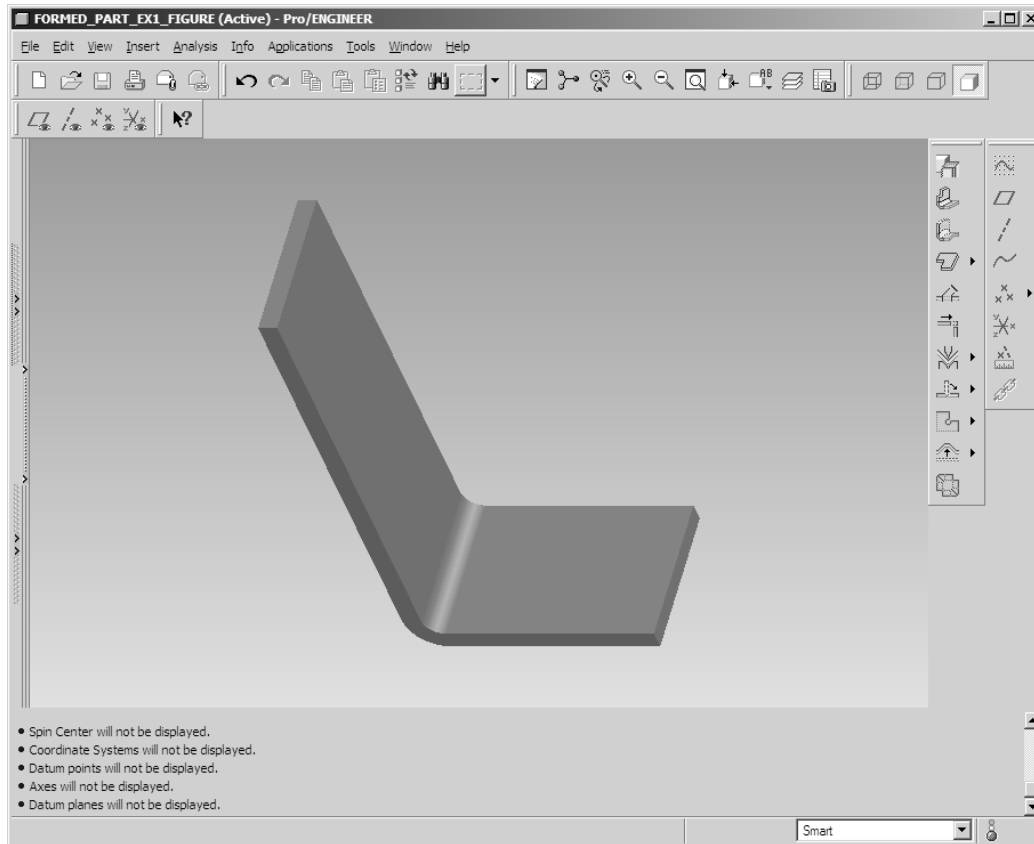

Figure 8

An extruded wall extends from an edge into space. You can sketch the side section of the wall and project it out a specified length. The primary wall is the first wall in your design, or a secondary (additional) wall dependent on the primary wall.

The base feature for the part we will be creating in this exercise uses the unattached wall extruded option. Keep the following points in mind when using this option:
   ➢ The sketch is an open loop (for simplicity and dimension control);
   ➢ The thickness is added using the appropriate sketch option 'thicken';
   ➢ Apply dimensioning intent to the sketch (i.e., apply the bend radius to the inside curve.

Create a **new** part file and select the **Sheetmetal** option under *Sub-type* in the NEW dialog box, as shown in the Figure 9.
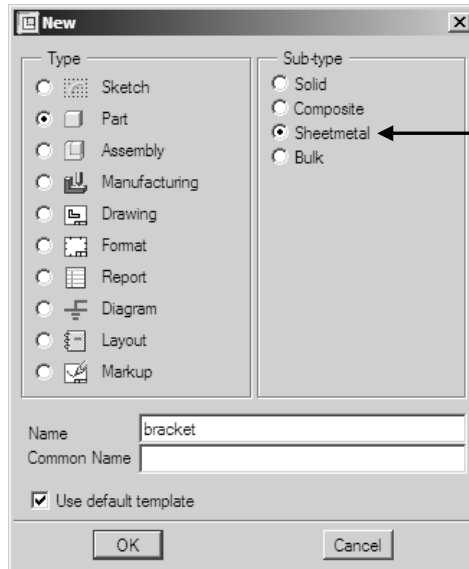
Figure 9

Ensure that the **Use default template** option is selected. Enter **bracket** as the part name, then select **OK**.

Select **Insert > Extrude**. Select **Placement > Define**. Select the **FRONT** datum plane as the *sketch plane.* Use the default direction for the feature creation, then select the **TOP** plane reference to face the *top direction.* **Click Sketch**.

 Sketch the open loop shown in the figure below with appropriate dimensions. Sketch dimensions shown below are: 8 in. wide × 4 in. high (radius is .75 in).
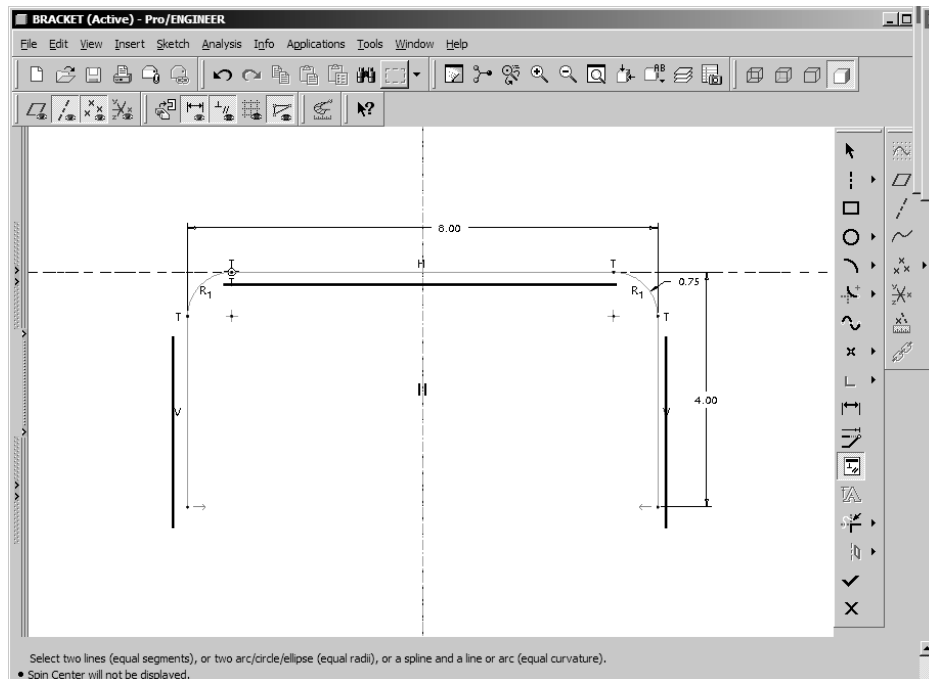

Figure 10

Once the sketch is completed, select **Sketch > Feature Tools > Thicken**. Ensure that the arrow points *toward the inside* of the loop sketch as shown in the figure below (select **Flip**, if necessary).
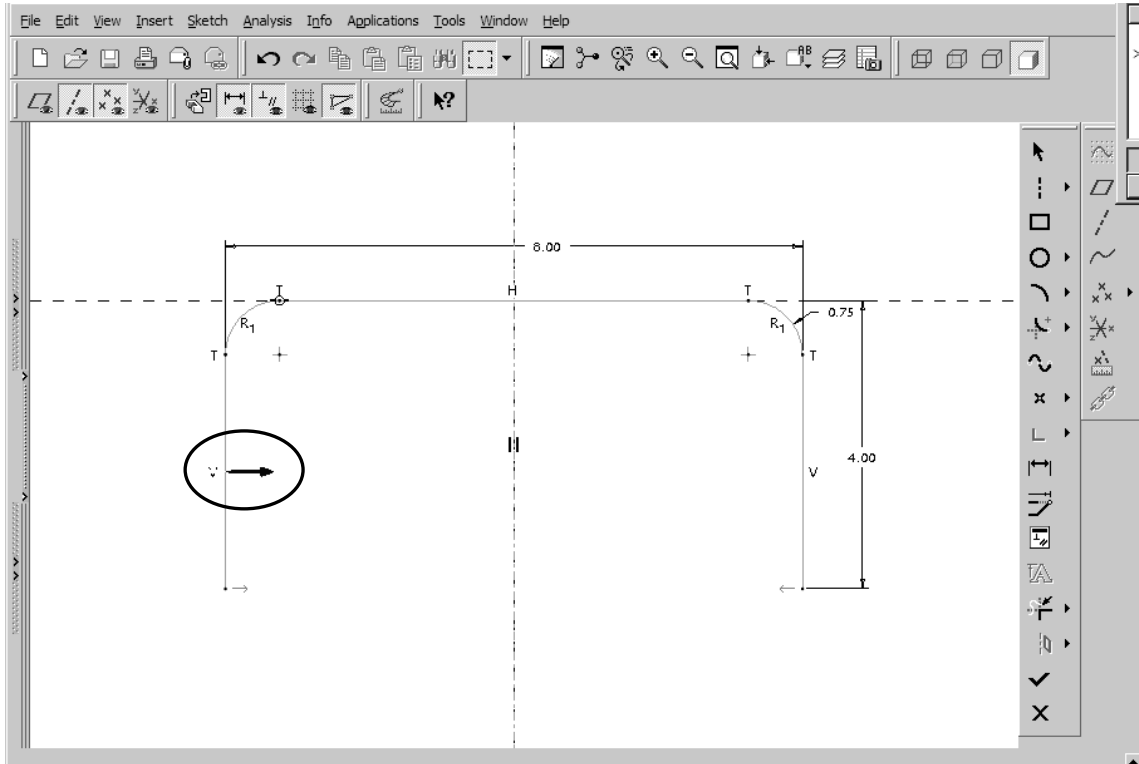

Figure 11

Then select **OK**. Enter **.135** (corresponding to a thickness of 10 gauge steel) for the material thickness. Other gauge steel thicknesses are given in Table 2 below for your reference.

TABLE 2: IMPERIAL SHEET METAL GAUGES

Gauge Number	Decimal Equivalent (inch)
1/4	.25
3/16	.188
7 ga.	.179
8	.164
9	.149
10	.134
11	.119
12	.104
13	.089
14	.074
15	.067
16	.059
17	.053
18	.048
19	.041
20	.036
22	.030
24	.024
26	.018
28	.015
29	.014
30	.012

In order to dimension for design intent, we need to delete the current radius dimension and re-dimension the radius by selecting the dotted arc previously created. Enter **.375** as the new inner radius value. Complete the sketch (select check mark) and enter a symmetrical **Blind Depth** of **8** when prompted. Select **Done** from the Dashboard. The completed model should now appear on your screen as shown in Figure 12.
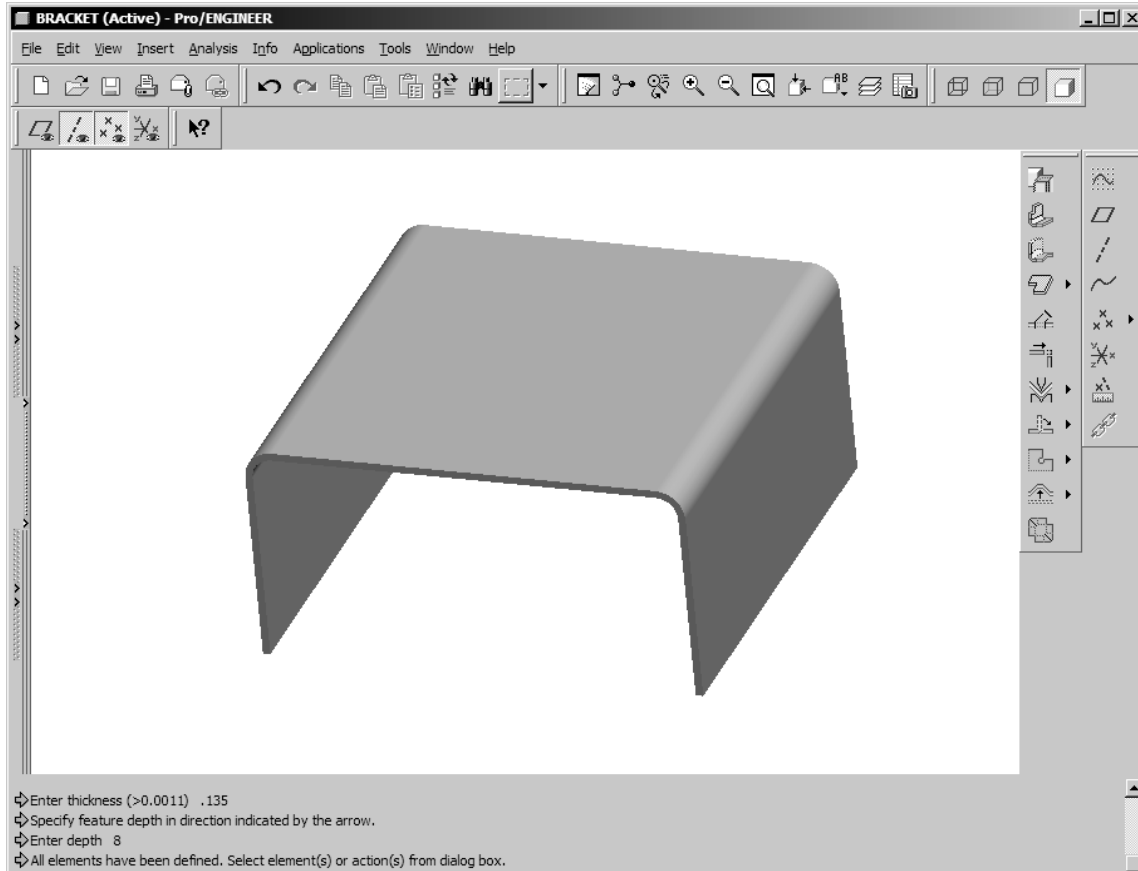
Figure 12

Set the display to **Wireframe.** Have a look at the different color schemes on the model. Note that the geometry created in Sheetmetal mode has a distinctive color scheme. One side of the model is green, and the other side white (or as per the user specified color displayed). When a sheet metal part is modeled, its wall thickness is constant over the entire model. The software uses the green side as the 'driving side.' The white surface is offset by a distance equal to the material thickness. In practice, you want to use the green side for your sketching reference for creating geometry and other sheet metal features.

This completes the first wall creation and this exercise. **Save** the part for future use and **erase** it from memory.

End of Exercise