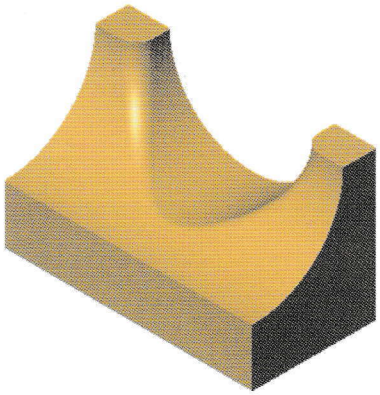
Exercise 11: Working with Fractions

Create this part using the information and fractional dimensions provided. Sketch and extrude profiles to create the part. This lab reinforces the following skills:



- Entering and displaying dimensions as fractions.
- Bosses.
- Cuts.
- Fillets.
- Blind and Through All end conditions.

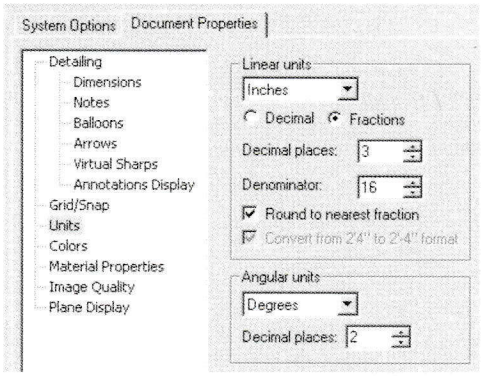
Fractions

There are two things to consider when working with dimensions that are given in fractions:

1. Setting the document units to fractional inches.
2. Entering dimension values as fractions.

Document Units

On the **Tools, Options** dialog, click the **Document Properties** tab and select **Units**. The two types of linear units that support **Fractions** are:



- **Inches**
- **Feet & Inches**

When you choose **Fractions**, you should specify the default **Denominator**. Dimensions that are evenly divisible by this denominator are displayed as fractions. How dimensions are displayed that are *not* evenly divisible depends on whether you select the **Round to nearest fraction** option.

For example, if the **Denominator** is set to 16 and you enter a value of 3⁄64 the value will display as 1⁄16 if **Round to nearest fraction** is selected. It will display as 0.047 if it is not selected.

Entering Dimensions

You can enter dimensions as fractions regardless of whether the document units are set to fractions. To enter a value such as 1 7⁄8", type 1, press the Spacebar, then type 7/8, and press Enter.

Design Intent

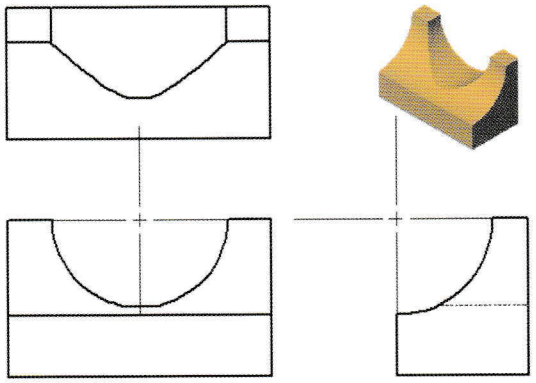
The design intent for this part is as follows:

1. The side to side cut is centered on the corner.
2. The front to back cut is centered at the midpoint of the edge.

Use the `Part_IN` template.

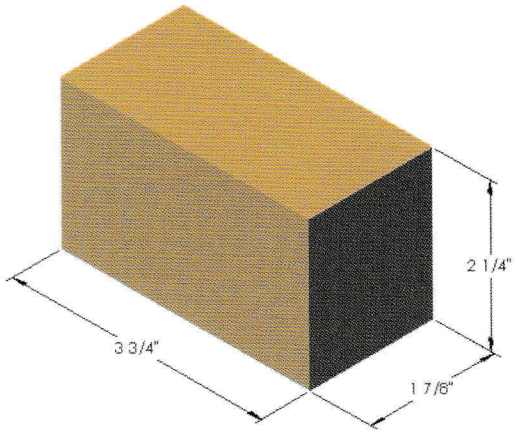
Views

Use the following graphics to help visualize the part.

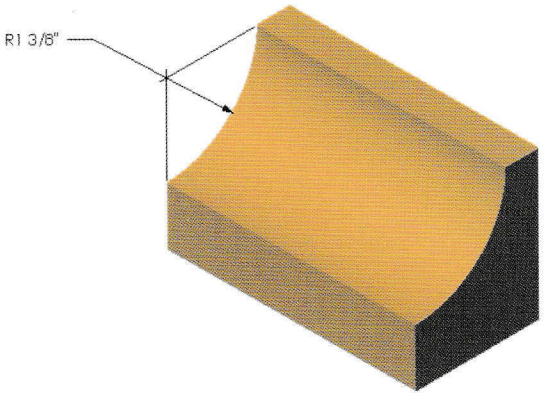


1 **New part.**
Open a new part using the `Part_IN` template.

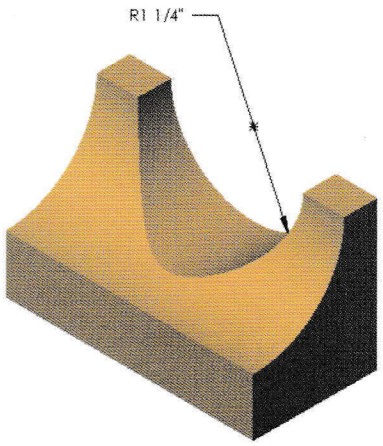
2 **First feature.**
Create the first feature using a sketch and an extruded boss. You can use either the `Front`, `Top` or `Right` reference plane for sketching.



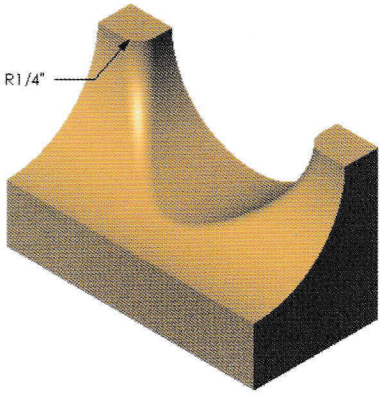
3 **Cut Feature.**
Using a sketch containing a circle, create an extruded cut feature. The circle is centered on the corner (vertex) of the first feature.



4 Second cut feature.
Create a second extruded cut using a circle. This circle should be centered at the midpoint (halfway along the edge).



5 Fillet/Round.
Using the edge created by the cuts, create a fillet/round feature.



6 Save and close the part.