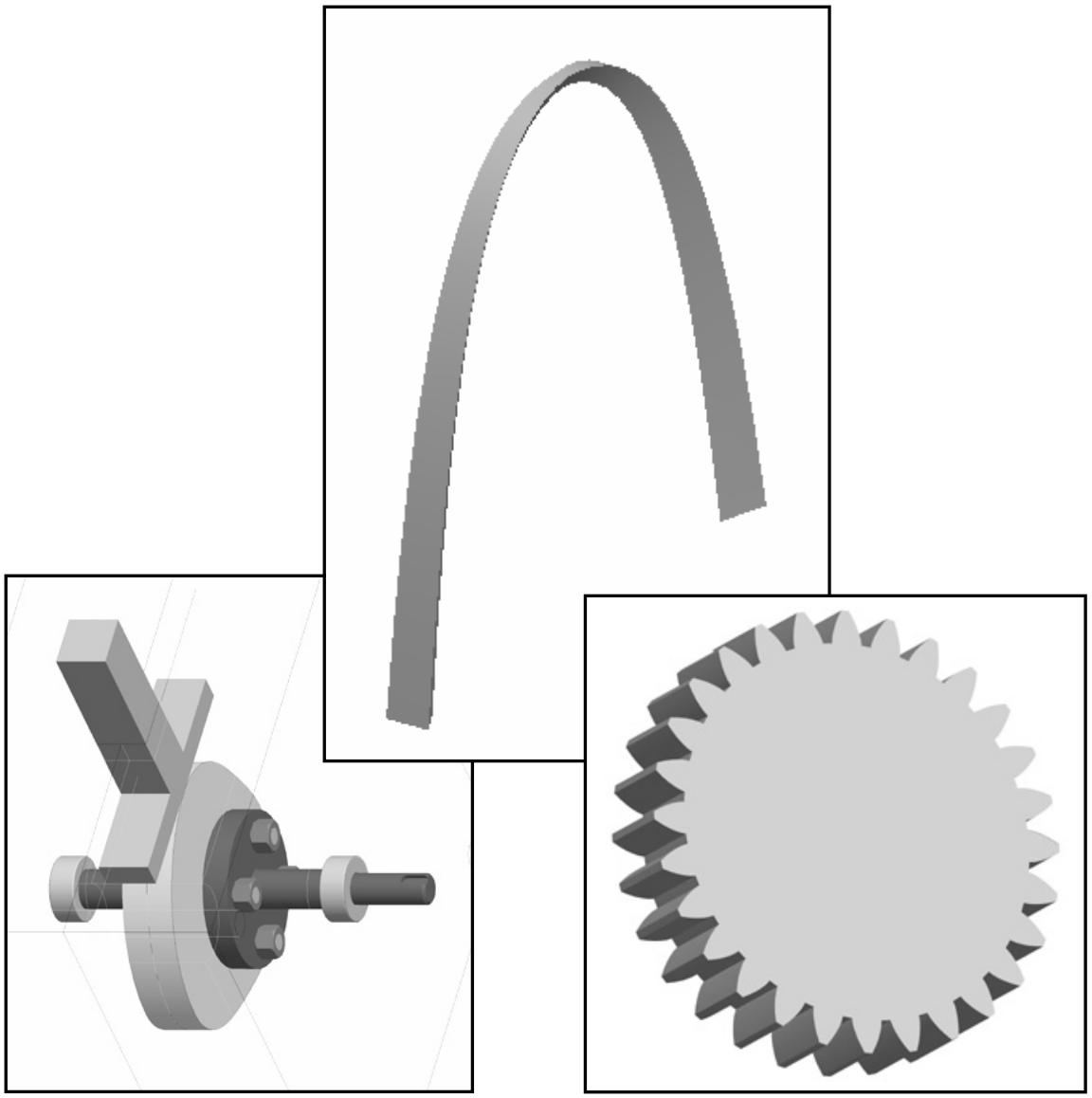
Modeling Using Pro | ENGINEER W I L D F I R E 2.0



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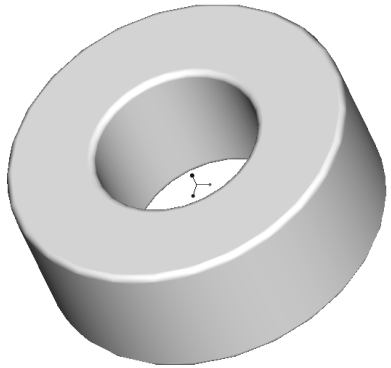
LESSON 2
BEARINGS

Learning Objectives:

- Understand the concept of ***datum planes***.
- Explore the use of ***mouse*** for ***zoom***, ***spin***, and ***pan*** functions.
- Learn ***Extrude*** and ***Round*** features.



Design Information:

Bearings allow relative motion between two components while minimizing frictional losses. For instance, the main bearings in an automobile allow the wheels to rotate relative to the body. Rolling element bearings are often used in machinery. They consist of an outer race and an inner race separated by rolling elements (either balls or cylinders). The rolling elements reduce friction by providing rolling contact. As bearings are purchased items, only the outer profile is modeled. Typically, rolling element bearings are mounted using interference fit. Therefore, the inner and outer diameters of the bearing are critical dimensions. For proper assembly, the edges of the bearing are rounded. The radius of the round is another critical dimension.

Sequence of Steps

Goal I: Experiment with the mouse

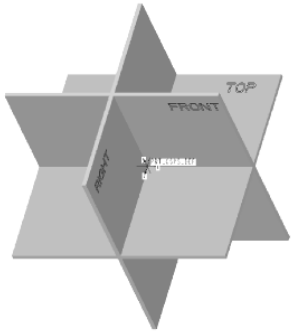
1. Learn spin, zoom and pan functions.

Goal II: Understand datum planes

1. FRONT, TOP and RIGHT are the three default datum planes.

2. PRT_CSYS_DEF is the default coordinate system.

3. Spin center (Red, Green and Blue lines) helps in rotating the part.

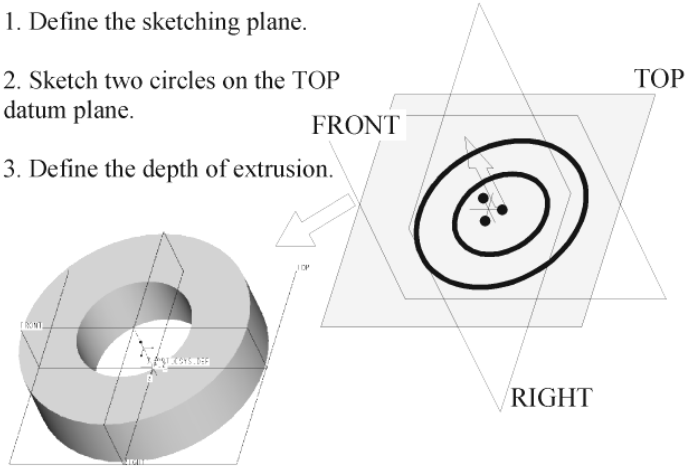


Goal III: Create the base cylinder

1. Define the sketching plane.

2. Sketch two circles on the TOP datum plane.

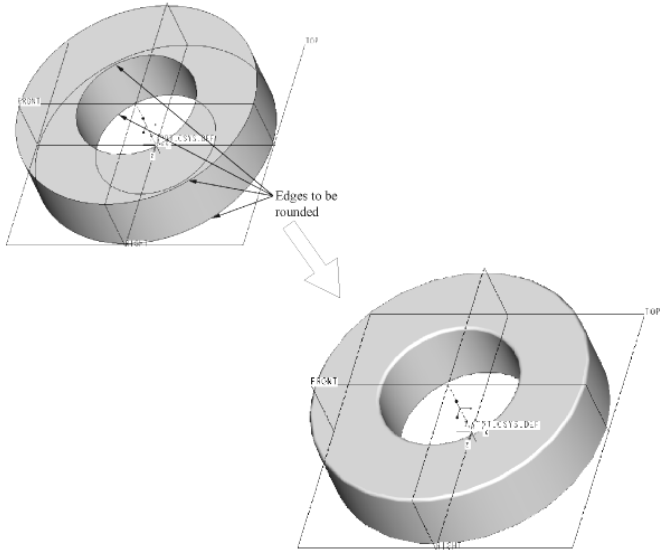
3. Define the depth of extrusion.



Goal IV: Round the edges of the bearing

1. Specify the radius.

2. Select the edges to be rounded.



Goal	Step	Commands
Open a new file for the bearing part	1. Set up the working directory.	**The working directory is where ProE tries to find or save files.** FILE → SET WORKING DIRECTORY → *Select the working directory* → OK
	2. Open a new file.	**We will create the bearing as a solid part.** FILE → NEW → *Part* → *Solid* → bearing → OK **Refer Fig. 2.1.** **ProE displays the three default datum planes – front, top and right - in the graphics window. Also, the default part coordinate system (PRT_CSYS_DEF) is shown at the intersection of these three datum planes.** **Refer Fig. 2.2.**

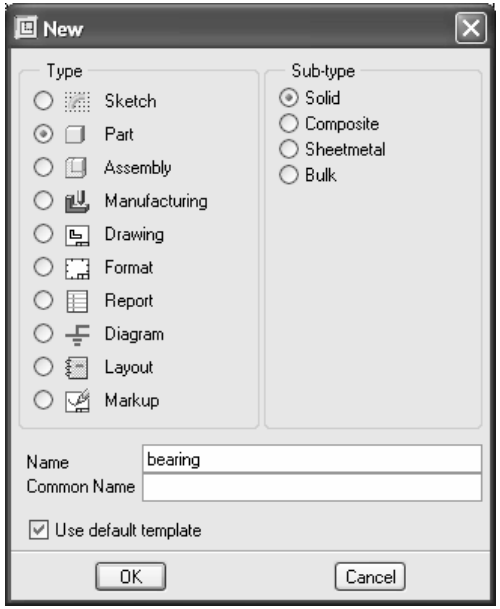


Fig. 2.1.

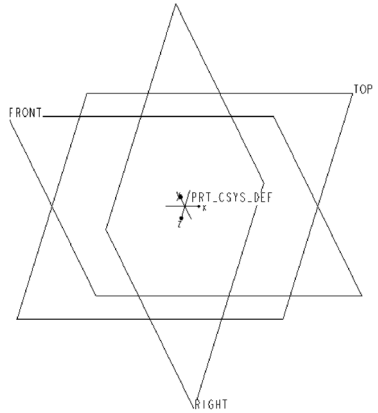


Fig. 2.2.

Goal	Step	Commands
Experiment with the mouse	3. Use the mouse to zoom, spin and pan the model.	**The mouse is a very powerful tool in ProE. We can zoom, spin and pan the model by holding the middle mouse button and CTRL/SHIFT key, and moving the mouse simultaneously.** **Fig. 2.3 illustrates the mouse functions. The center of the zoom occurs at the cursor location. Explore each of these functions.** **The view can be scaled up or down by a factor of 2 by holding either SHIFT or CTRL key, and rotating the middle mouse button.** **To get back to the default view, use the following command:** VIEW → ORIENTATION → STANDARD ORIENTATION. **The default view is typically set as trimetric. However, it can be changed to isometric or user-defined by using the following command:** VIEW → ORIENTATION → REORIENT → (Type) *Preferences* → (Default orientation) Trimetric→ OK

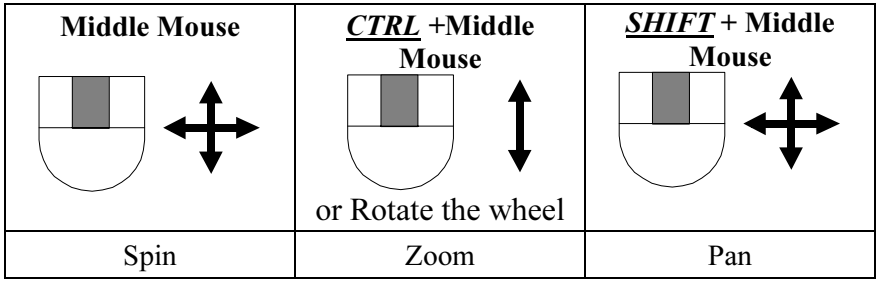


Fig. 2.3.

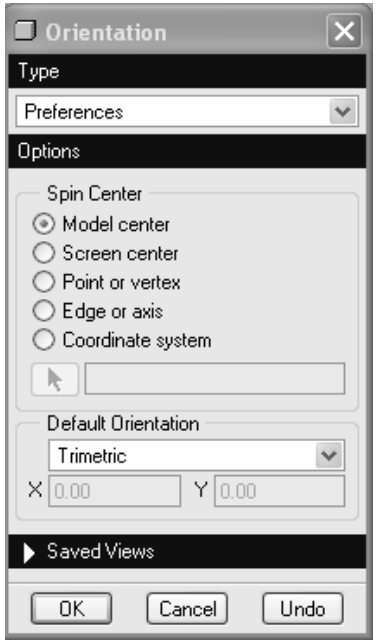


Fig. 2.4.

Goal	Step	Commands
Understand the datum planes	4. Understand the datum planes.	★ **ProE creates three default datum planes named as FRONT, TOP and RIGHT. Each datum plane has two sides marked by two different colors: ORANGE and BLACK. We can visualize the planes better by looking at Fig. 2.5 where the planes are shaded.** **In the default view shown in Fig. 2.2, only the orange sides are visible. We can notice the black color when we rotate the datum planes. The orange side is considered the active side of the datum plane.** **In Figs. 2.2 and 2.5, we can also see the default coordinate system "PRT-CSYS-DEF" at the center. The spin center is shown in Red, Green and Blue (RGB) color lines representing X, Y and Z axes respectively. The spin center helps in rotating the part.**
Create the base cylinder	5. Start "Extrude" feature.	INSERT → EXTRUDE [Or *click*  in the feature toolbar – left side]

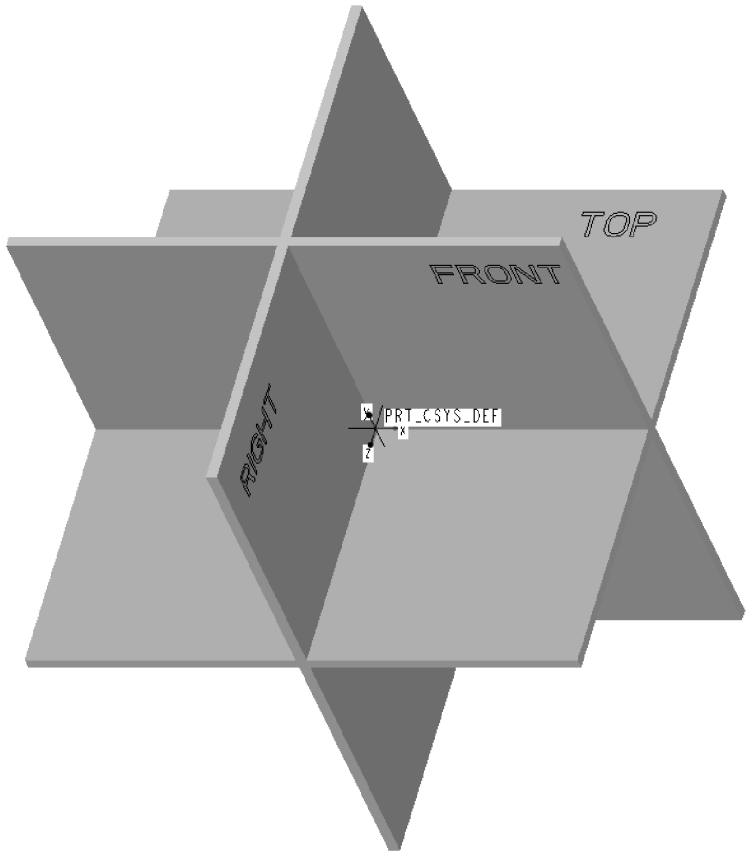


Fig. 2.5.

Goal	Step	Commands
Create the base cylinder (Continued)	6. Define the sketching plane.	**To select the sketching plane,** *Placement* (in the dashboard - lower part of the screen) → Define **Refer Fig. 2.6.** **ProE brings up the section window where we define the placement of the section.** **Refer Fig. 2.7.** **We are going to sketch the section on the TOP datum plane. Note that ProE highlights different planes as we move the mouse over the planes.** *Select the TOP datum plane in the graphics window or in the model tree by clicking on "TOP"* → **Refer Fig. 2.8.** **The arrow points to the view direction. We can reverse the viewing direction by clicking "Flip" in the section window. ProE automatically orients the sketching plane.** 

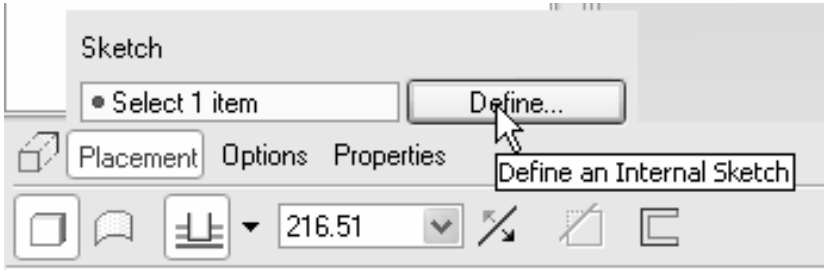


Fig. 2.6.

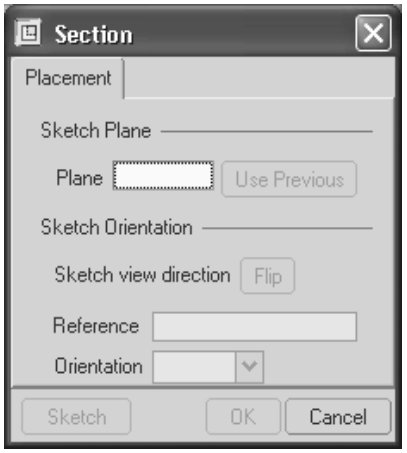


Fig. 2.7.

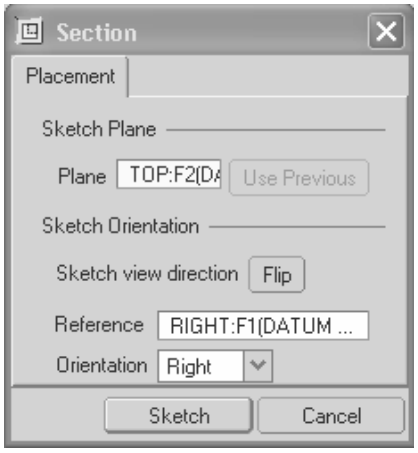


Fig. 2.8.

Goal	Step	Commands
Create the base cylinder (Continued)	7. Understand the orientation of the sketcher.	**The screen changes to the sketcher mode. The "References" popup window has two references: F1(RIGHT) and F3(FRONT).** **Refer Fig. 2.9.** **If this window is not visible, select** SKETCH → REFERENCES **All dimensions are placed with respect to the references. If necessary, additional references can be added to this list. It is advisable to select the references before sketching.** Close **Let us rotate the model to understand where we are sketching.** *Move the mouse holding Middle Mouse* → **To get back to the previous view, use the following command:** VIEW → SKETCH VIEW (Or )

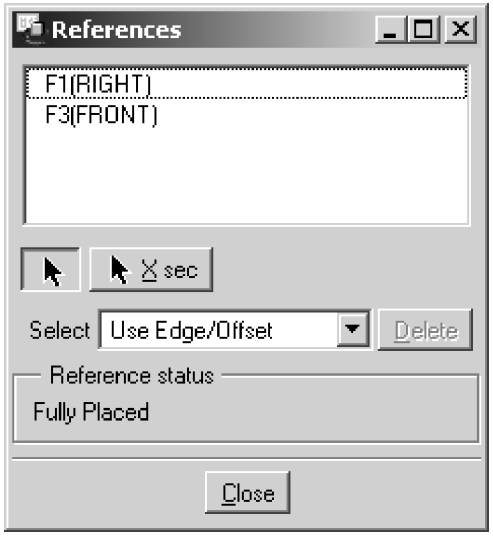


Fig. 2.9.

Goal	Step	Commands
Create the base cylinder (Continued)	8. Draw an outer circle.	→ *Select the center of the circle as the intersection of the FRONT and RIGHT datum planes* → **Refer Fig. 2.10.** **The cursor snaps onto the intersection.** *Select a point to define the outer edge of the circle* **ProE automatically puts the dimension for the circle.** **Refer Fig. 2.10.**
	9. Create an inner circle.	→ *Select the center of the circle as the intersection of the FRONT and RIGHT datum planes* → *Select a point to define the inner circle* **Refer Fig. 2.11.**
	10. Modify the dimensions.	**It is a good practice to modify smaller dimensions first.** → *Double click the inner circle dimension* → 1 → ***ENTER*** → *Double click the outer circle dimension* → 2 → ***ENTER*** **ProE automatically regenerates the section.**

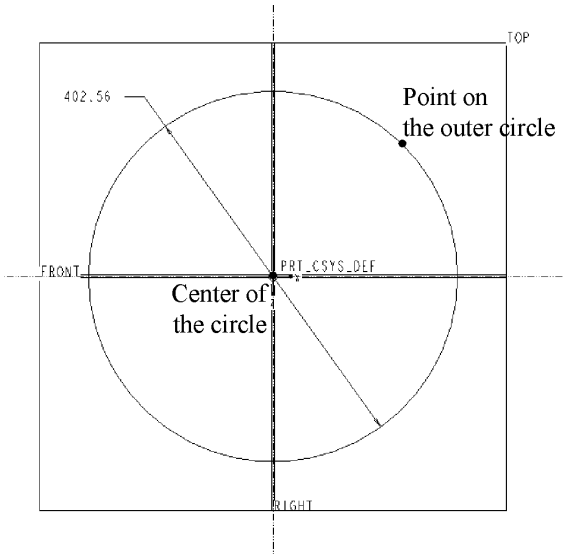


Fig. 2.10.

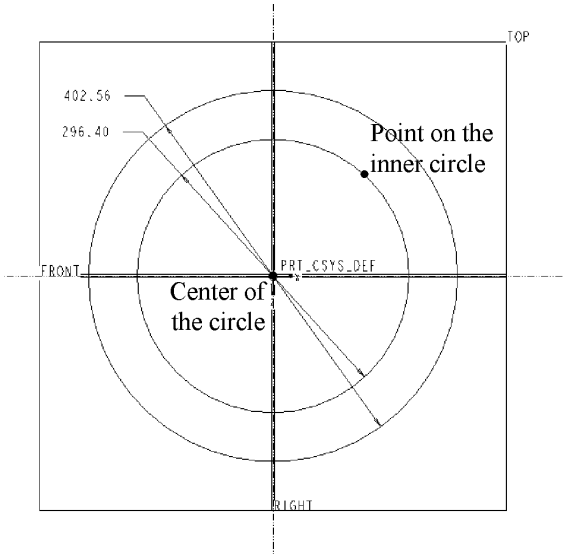


Fig. 2.11.

Goal	Step	Commands
Create the base cylinder (Continued)	11. Exit sketcher.	✓
	12. Define the depth.	**The depth dimension is shown in two places – in the dash and on the part.** **Refer Figs. 2.12 and 2.13.** *Select the depth dimension* → 0.5 → ***ENTER***
	13. Accept the feature creation after previewing.	VIEW → ORIENTATION → STANDARD ORIENTATION → → **Refer Fig. 2.14.**

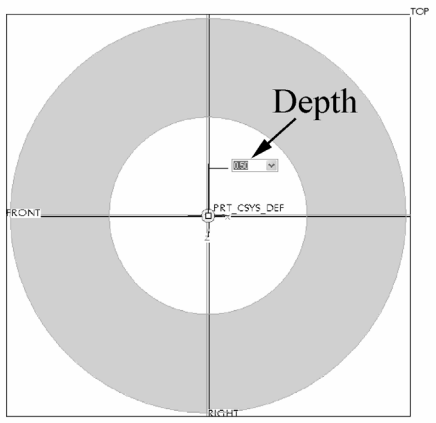


Fig. 2.12.

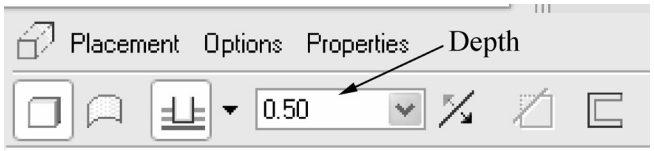


Fig. 2.13.

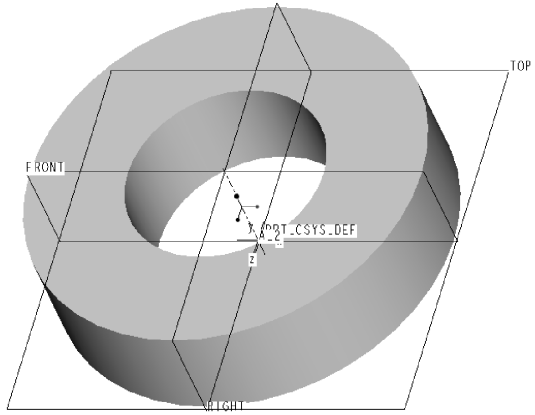


Fig. 2.14.

Goal	Step	Commands
Round the edges	14. Round the four edges of the bearing.	INSERT → ROUND (or ) → **We will specify the radius of all rounds to be 0.025.** 0.025 → ***ENTER*** → **Refer Fig. 2.15.** *Select the four edges to be rounded* → **Refer Fig. 2.16.**  **Refer Fig. 2.17.**

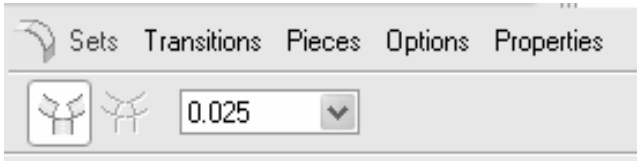


Fig. 2.15.

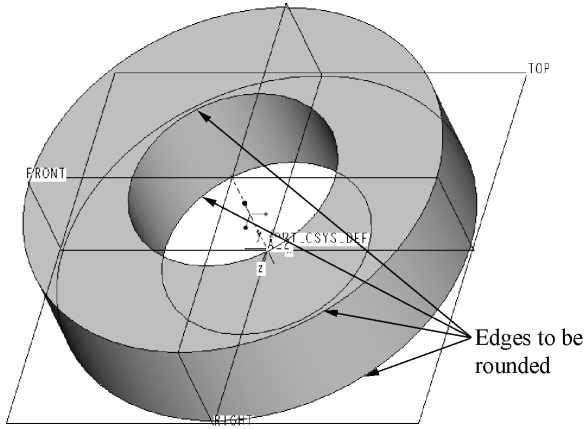


Fig. 2.16.

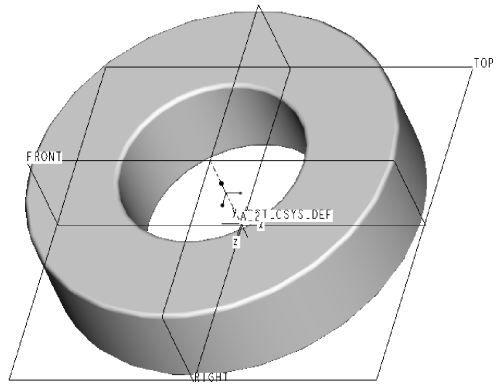


Fig. 2.17.

Goal	Step	Commands
View the model	15. Turn the datum planes off.	*Click on the following icons to switch off the datums, axis, datum points and default coordinate system.*  **These icons help in turning the datum planes, axis, datum points and coordinate system on and off.** **Refer Fig. 2.18.** **Modifying the display may help in visualizing the model better. One of the four model display options can be selected by clicking on the corresponding icon:** **Wire-frame -**  **Hidden line -**  **No hidden line -**  **Shaded -**  **Fig. 2.19 shows the model in the four display types.** **Spin center can be turned on/off by clicking**  **.**

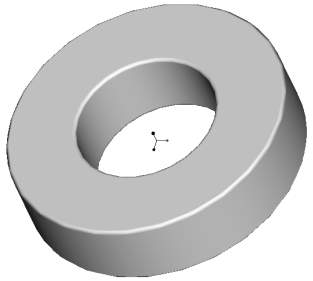
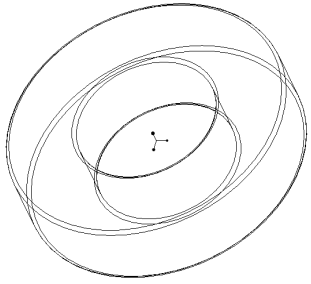
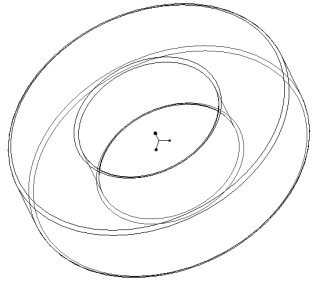


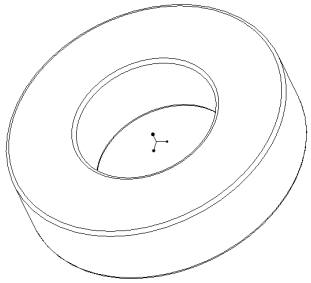
Fig. 2.18.



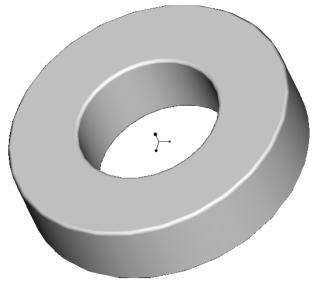
a. Wire-frame



b. Hidden line



c. No hidden line



d. Shaded

Fig. 2.19.

Goal	Step	Commands
Modify dimensions	16. Modify the dimensions.	*Select the extrusion feature by clicking on the bearing in the graphics window or from the model tree → Right Mouse →* Edit → **Refer Fig. 2.20.** *Select the 2.0 dimension →* 1.25 → ***ENTER*** → *Select the 1.0 dimension →* 0.6 → ***ENTER*** → EDIT → REGENERATE (or ) **Modifications take affect after regeneration.** **Refer Fig. 2.21.**
Save the file and exit ProE	17. Save the file and exit ProE.	FILE → SAVE → BEARING.PRT → OK → FILE → EXIT → Yes

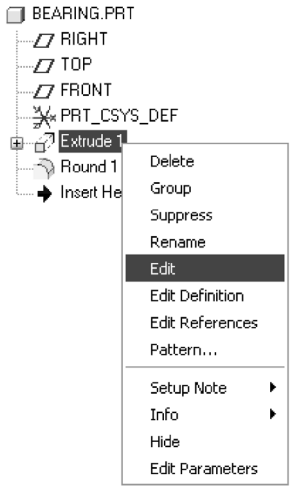


Fig. 2.20.

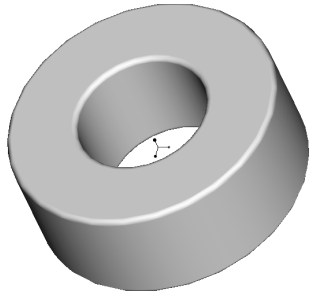


Fig. 2.21.

About ProE files

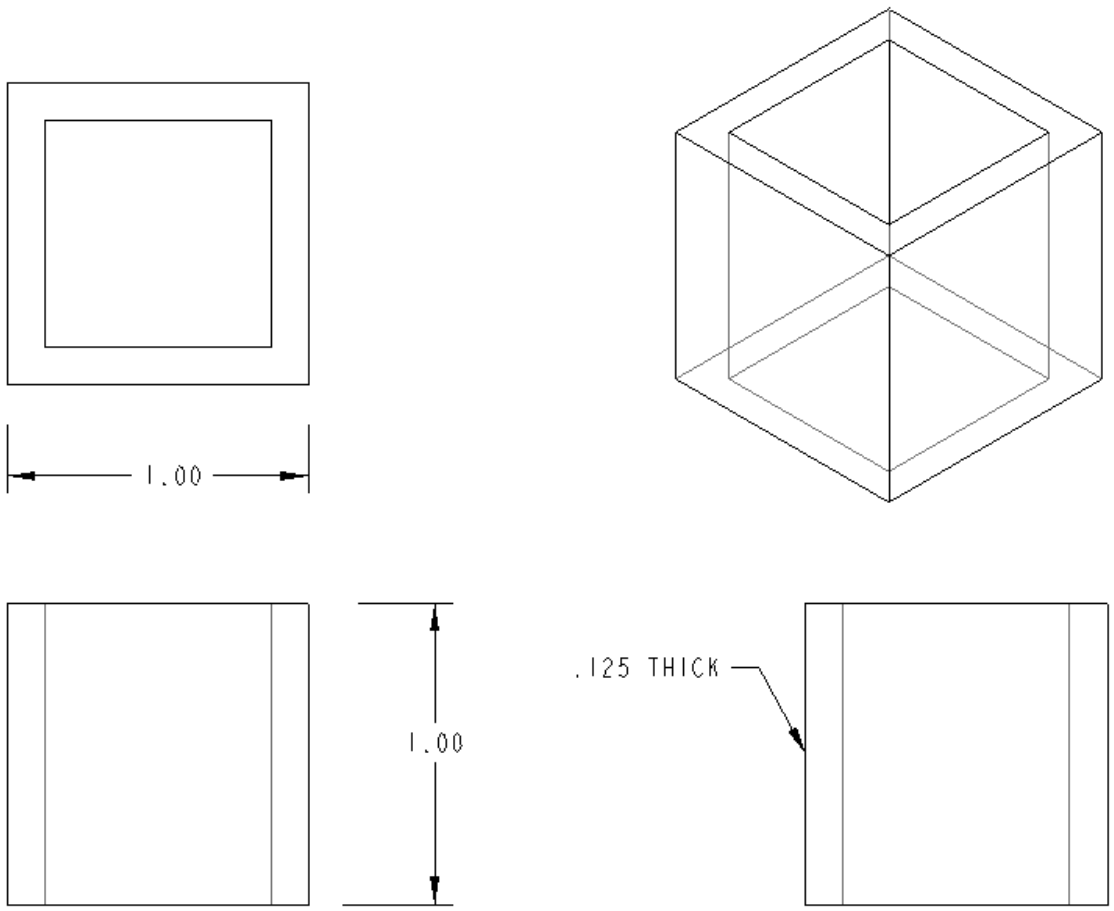
ProE creates several files. Part files with .prt.X extension. Each time we save the part, ProE creates another file. For instance, the first time we save the bearing part, ProE creates a file "bearing.prt.1." Next time we save the same part, it creates another file "bearing.prt.2." We really need the last file for most purposes. However, if we want to roll back, we can look at any previous saved versions of the part. The previous versions can be deleted to save disk space by selecting following list of commands: FILE → DELETE → OLDER VERSIONS.

ProE records all the commands, menu selections used, and dialog choices in a file called "trail.txt." This file can be used to either re-create a session or create training files. Note that the file should be renamed before opening it in ProE. This file can be edited in a text editor. A trail file can be played in ProE by the following command: UTILITIES → PLAY TRAIL/TRAINING FILES

Exercise

Create the following parts.

Problem 1

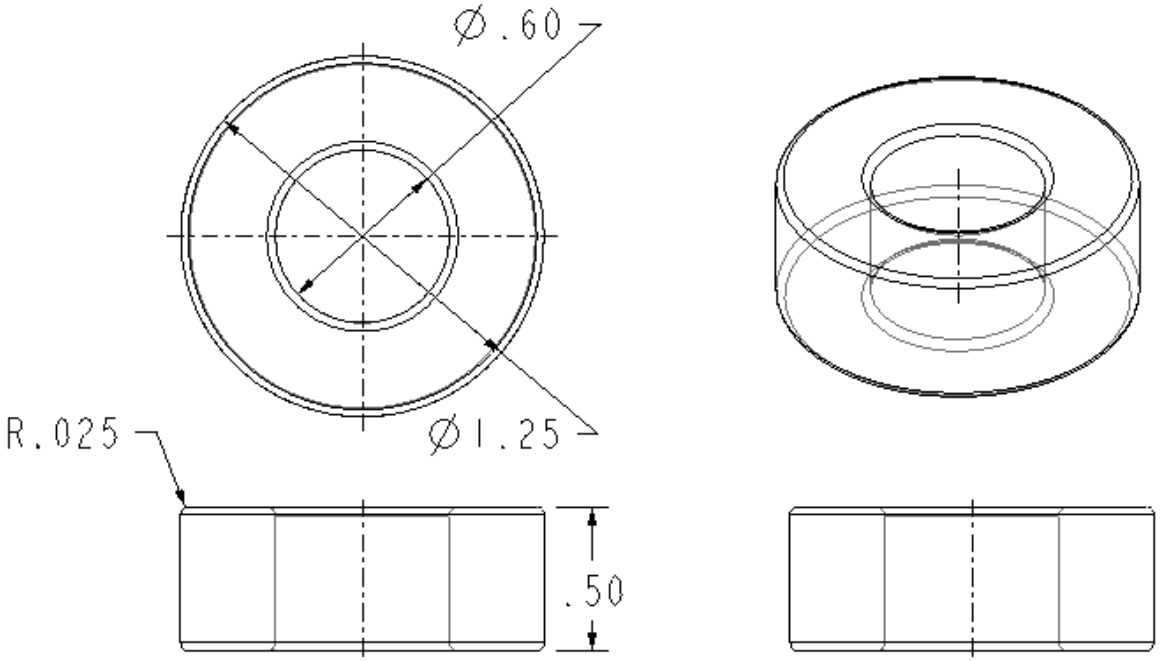


Hints:

1. Start the extrude feature.
2. Create a rectangle (1×1) using the create rectangle tool in the sketcher.
3. Select thin option and define the thickness as 0.125. Sample dash is shown in the figure below.

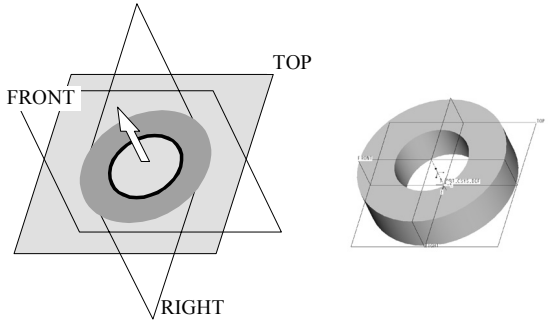
Problem 2

Create the bearing part using "Extrude – Thin" option.



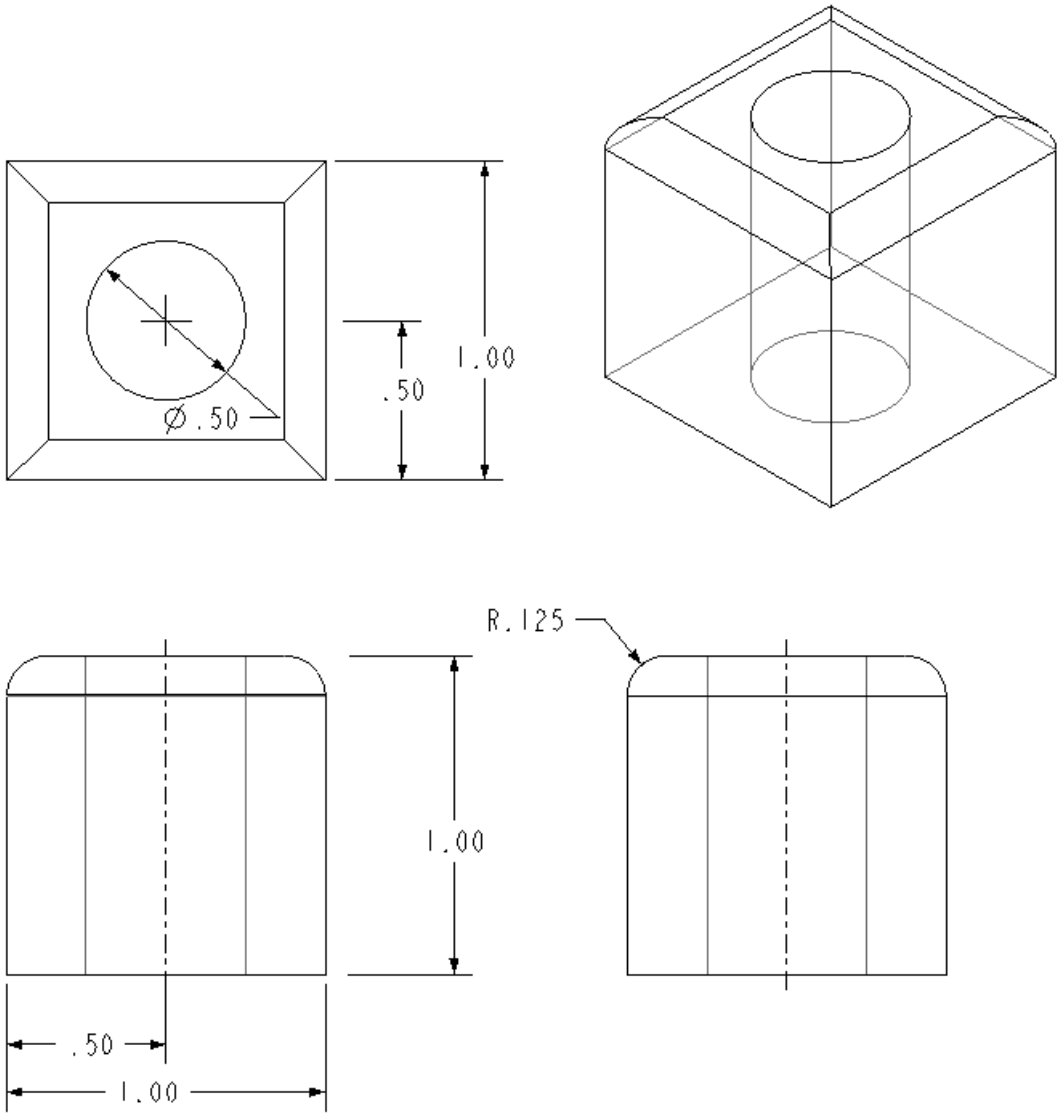
Hints:

1. Start the extrude feature.
2. Select the top datum plane as the sketching plane.
3. In the sketcher, create a circle of 0.6″ diameter.



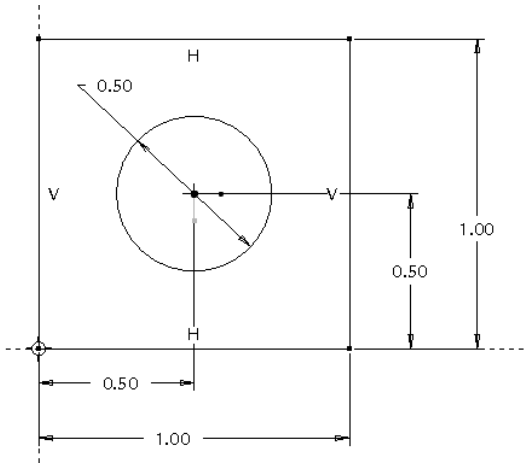
4. Select thin option and define the thickness as 0.325″. Sample dash is shown in the figure below.

Problem 3



Hints:

1. The sketch is shown in the figure below.



Notes: