Modeling with Creo™ Parametric 2.0

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LESSON 2
BEARING

Learning Objectives

- Understand the concept of ***datum planes***.
- Explore the use of ***mouse*** for ***zoom***, ***spin***, and ***pan*** functions.
- Learn ***Extrude*** and ***Round*** features.
- Experiment with the use of ***model player*** and ***trail/training files***.

About Creo Parametric files

When modeling, Creo creates several files. Part files have an extension ".prt.X" where X represents the revision number. Each time a user saves a part, Creo creates a new file. For instance, a part, say bearing, is saved for the first time, Creo creates the file - bearing.prt.1. Subsequent saves, it creates "bearing.prt.2", "bearing.prt.3", "bearing.prt.4", and so on. A user can roll back to any previous version of the part by renaming that particular revision file and opening it. For most purposes, the last and latest version is sufficient. The previous versions can be deleted to optimize the disk space by selecting the following list of commands: **FILE → MANAGE FILES → DELETE OLDER VERSIONS**.

File Extensions

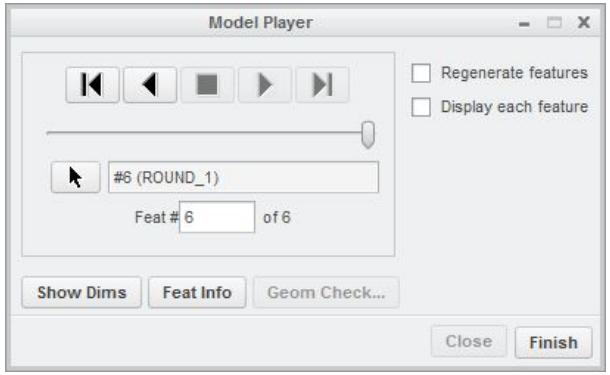
.asm	Assembly
.drw	Drawing
.frm	Format
.iges	Initial Graphics Exchange Specification format
.lay	Notebook
.pro	Configuration file
.prt	Part
.pts	Points
.sec	Section (or sketch)
.stl	Stereolithography file
.txa	Trail file

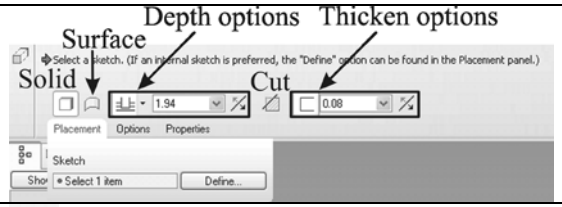
Trail/Training Files

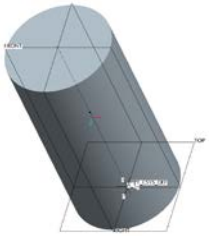
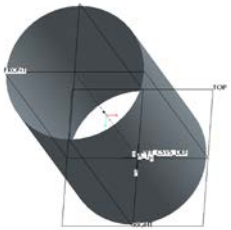
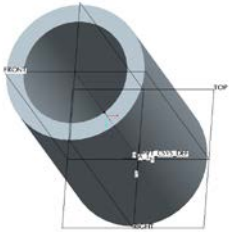
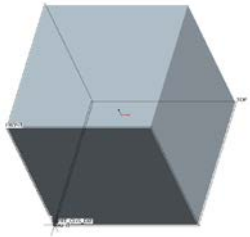
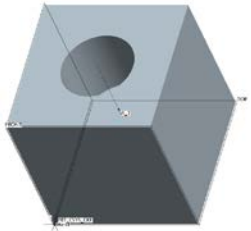
Creo records all the commands, menu selections, and dialog choices in a file called "trail.txt." This file is useful in recreating a session or creating training files. The file can be edited using a text editor. Note that before playing the trail file, the file should be renamed. The following sequence of commands plays the trail file: **FILE → MANAGE SESSION → PLAY TRAIL FILE**.

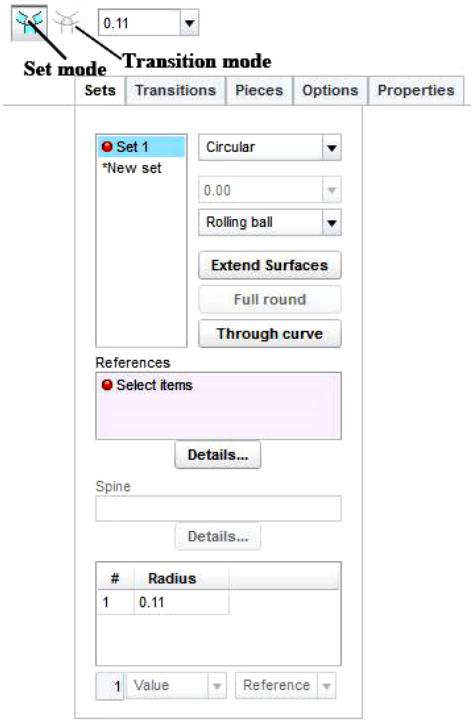
Model Player

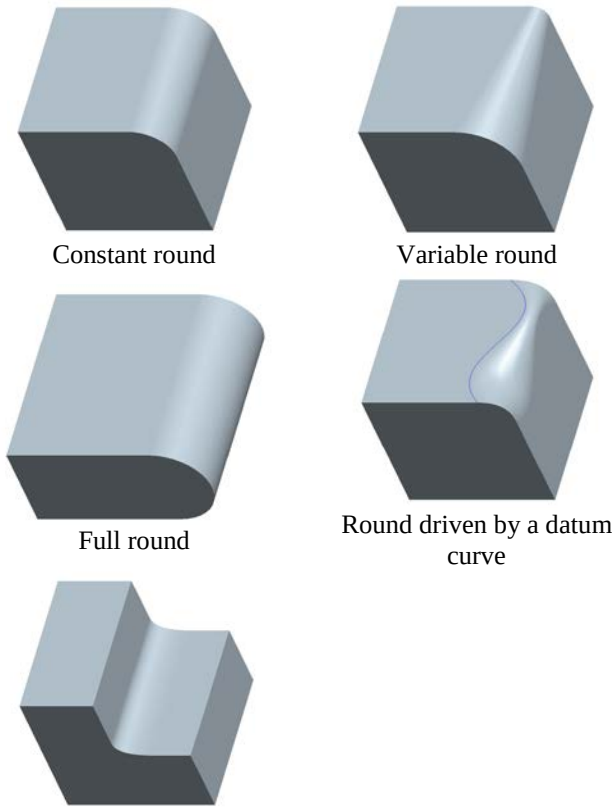
Model player is a useful tool to walk through a part model, and understand the design intent of the original designer. A user can initiate the model player using: **TOOLS → MODEL PLAYER**. Once started, it steps the user through each feature, and provides information about each feature.



Extrude feature	
Useful for creating a solid protrusion, a cut, or a surface.	
Starting the feature	**MODEL →**  Extrude
	
Solid	 creates a solid
Surface	 creates a surface
Depth Options	 specified depth  both sides of the sketch plane by half the specified depth in each direction  up to next surface where the extrusion stops when a surface is encountered  intersect or extend with all surfaces  intersect a specific surface where the user specifies the surface  up to a selected point, curve, plane, or surface  Flips the direction of extrusion
Cut	 Removes the material when the option is highlighted
Thickness Options	 Create a thin feature whose thickness is specified  Adds thickness to the outside/inside/either sides of the sketch
Placement	**Define** to create a new section or redefine the existing one.

Examples	Extruding a circular section with different extrude option.  Extrude – Solid  Extrude - Surface  Extrude - Thin  Block before extrude cut  Extrude – Cut using a circular section

Round feature	
Useful for rounding edges.	
Starting the feature	**MODEL →**  
Mode	 By default, Creo Parametric activates the set mode. In this mode, a user controls a set of round geometry using a single round dimension. When selecting a number of references (edges to be rounded), hold ***CTRL*** to place them in a single set.
	 In the transition mode, a user specifies the filler geometry to transition between round geometries.
Shape of the section	Two basic geometries for round are circular and conic. The conic option provides a better control of the sharpness of the round.
Rolling ball	The round is formed by rolling a spherical ball along the surfaces to which it is tangent. It is the default option.

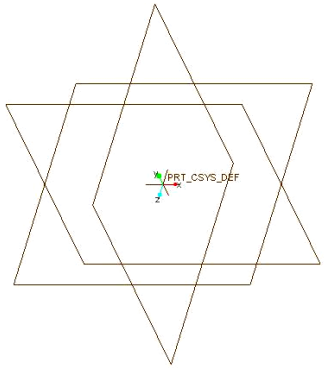
Normal to spine	The round is created by sweeping a conic section normal to a spine.
Examples	

Working Directory

The working directory is a designated area for Creo to save its files. Creo looks for files in the working directory. Note that if you retrieve a file from another directory and use **FILE → SAVE**, Creo saves the file in the original directory, and not in the working directory. Use **SAVE AS** command to save it in the working directory.

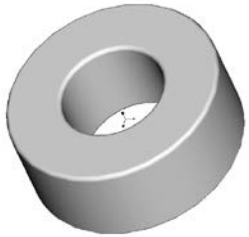
Datum Planes

Creo creates three default datum planes - FRONT, TOP and RIGHT - as the initial features to start the modeling process. Each datum plane has two sides marked by brown and gray colors. In the standard orientation, only the brown sides are visible. The gray color appears when the model is rotated. The brown side is considered to be the active side of the datum plane. The default part coordinate system "PRT-CSYS-DEF" is located at the intersection of the three datum planes. The spin center shown in Red, Green and Blue (RGB) colors helps in rotating the model.



Background Information:

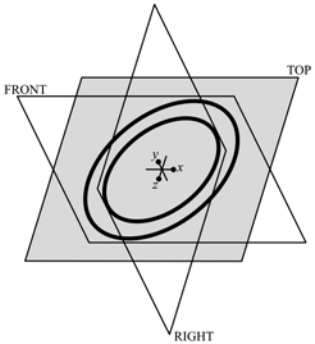
Bearings allow relative motion between two components while minimizing frictional losses. For instance, the main bearings in automobile allow the wheels to rotate relative to the axle. A rolling element bearing, one of the widely-used bearings, consist of an outer race and an inner race separated by 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. As the rolling element bearings are typically mounted using an interference fit, the inner and outer diameters of the bearing are critical. For a proper assembly, the edges of the bearing are rounded, and therefore, the radius of the round is another critical dimension.

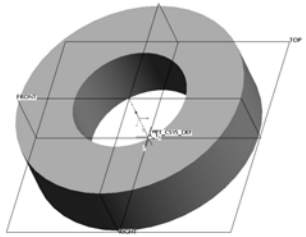


SEQUENCE OF STEPS

Step I - Extrude the base cylinder

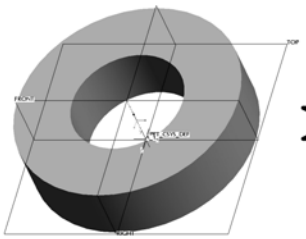
1. Use "Extrude" feature.
2. Define the sketch plane
3. Sketch two circular sections
4. Define the depth of extrusion

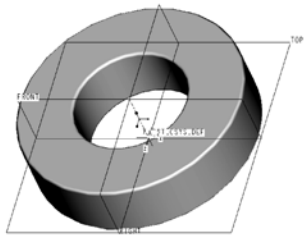




Step II - Round the edges of the base cylinder

1. Use "Round" feature
2. Specify the radius
3. 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 a designated area for Creo for opening as well as saving files. We recommend creating a folder for each project.  → ***Select the working directory*** → **OK** Alternatively, use **FILE →MANAGE SESSION →** ***SELECT 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 to Fig. 2.1. In the model tree window, Creo displays the three default datum planes (RIGHT, TOP, and FRONT) and the default part coordinate system (PRT_CSYS_DEF) at the intersection of the three datum planes. Refer to Fig. 2.2.

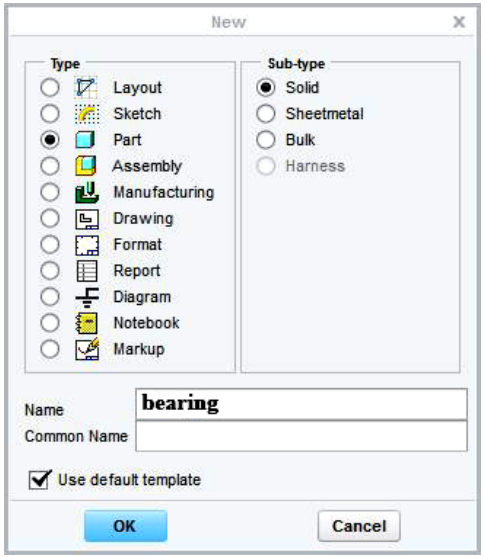


Fig. 2.1.

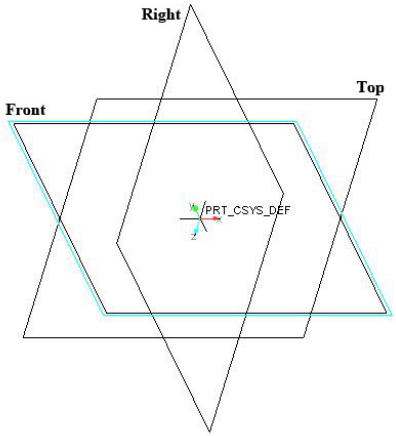


Fig. 2.2a.

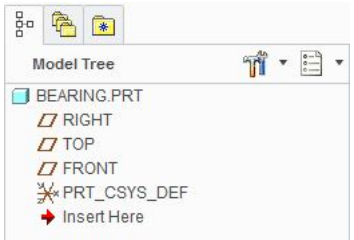


Fig. 2.2.b

Goal	Step	Commands
Experiment with the mouse	3. Use the mouse to zoom, spin, and pan the model.	The user can spin, zoom, and pan the model by moving the mouse while holding middle mouse button, middle mouse and ***CTRL*** key, and middle mouse and ***SHIFT*** key respectively. Fig. 2.3 illustrates the mouse functions. The center of the zoom is always at the cursor location. The view can be scaled by a factor of 2 by holding ***SHIFT*** or ***CTRL*** key, and rotating the middle mouse button. Explore each of these functions. To get back to the default view, use the following command: **VIEW → STANDARD ORIENTATION** (or  **→ STANDARD ORIENTATION**) Refer to Fig. 2.4. The default view is typically set as trimetric. However, it can be changed to isometric or user-defined by using the following command:  **VIEW →** **→ REORIENT → (Type)** ***Preferences*** **→ (Default orientation) Trimetric→ OK** Refer to Fig. 2.5.

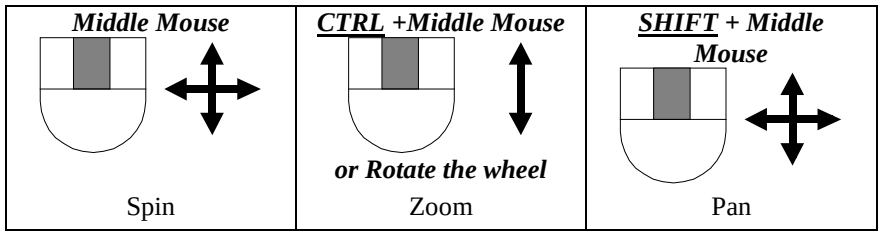


Fig. 2.3.

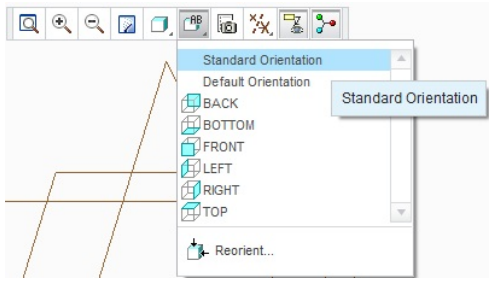


Fig. 2.4.

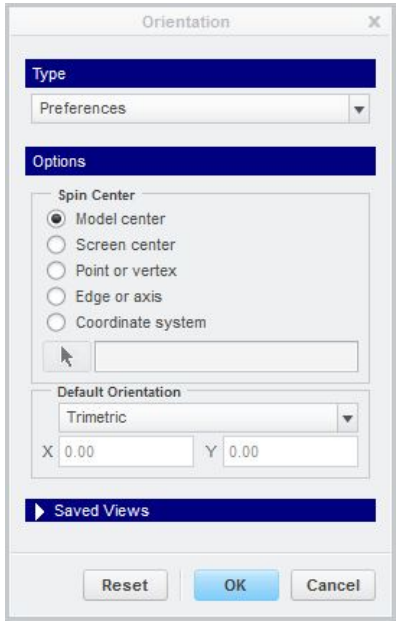


Fig. 2.5.

Goal	Step	Commands
Understand the datum planes	4. Understand the datum planes.	★ Creo creates three default datum planes - FRONT, TOP and RIGHT. Each datum plane has two sides marked by brown and gray colors. These planes can be visualized by looking at Fig. 2.6 where the planes are shaded. In the standard orientation (shown in Figs. 2.2a and 2.6), only the brown sides are visible. The gray color appears when the model is rotated. The brown side is considered to be the active side of the datum plane. In Figs. 2.2a and 2.6, the default part coordinate system "PRT-CSYS-DEF" is located at the intersection of three datum planes. The spin center shown in Red, Green and Blue (RGB) color lines helps in rotating the model.
Create the base cylinder	5. Start "Extrude" feature.	**MODEL →** 

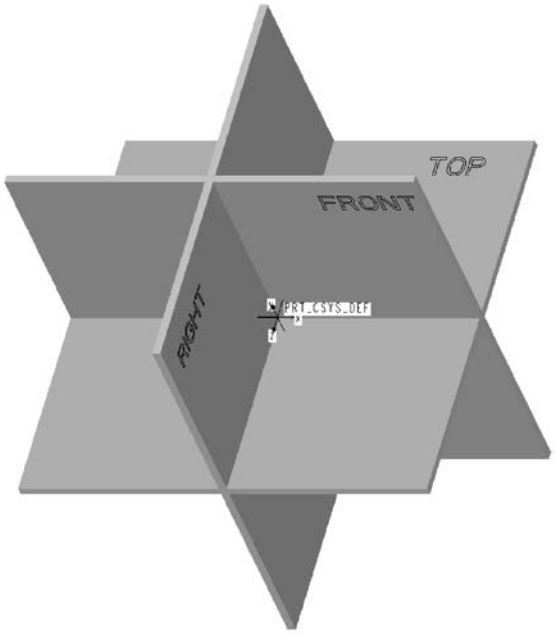


Fig. 2.6.

Goal	Step	Commands
Create the base cylinder (continued)	6. Define the sketch plane.	To select the sketch plane, ***click Placement*** **(highlighted in red) →** **Define** Refer to Fig. 2.7. Creo brings up "Sketch" window where we define the sketch plane. Refer to Fig. 2.8. We are going to sketch the section on the TOP datum plane. Creo highlights different planes as we move the mouse over them. ***Select the TOP datum plane in the graphics window or in the model tree by clicking on "TOP" →*** Refer to Fig. 2.9. The red arrow in the graphics window points to the view direction. Clicking "Flip" in the sketch window reverses the view direction. Creo automatically orients the sketch plane by aligning the outward normal from the reference (the right datum plane) in the right direction. Refer to Fig. 2.9. **Sketch**

Fig. 2.7.

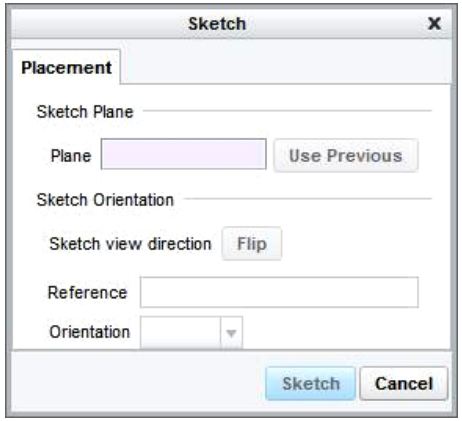


Fig. 2.8.

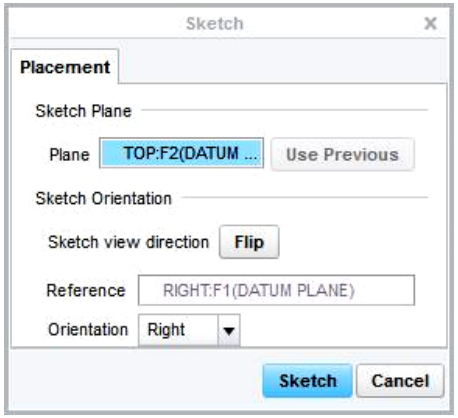


Fig. 2.9.

Goal	Step	Commands
Create the base cylinder (continued)	7. Identify and select references.	The screen changes to the sketcher mode. Activate “References” window by selecting:  (located in Setup group of icons) The “References” window shows two references: F1(RIGHT) and F3(FRONT). Refer to Fig. 2.10. All dimensions are placed with respect to these two references. If necessary, additional references can be added to this list. It is advisable to select the references before sketching. **Close**
	8. Understand the orientation of the sketcher.	Holding the middle mouse button and moving the mouse rotates the model. ***Move the mouse holding Middle Mouse →*** Activate the sketch view by clicking  Sketch view orients the sketch plane parallel to the screen.

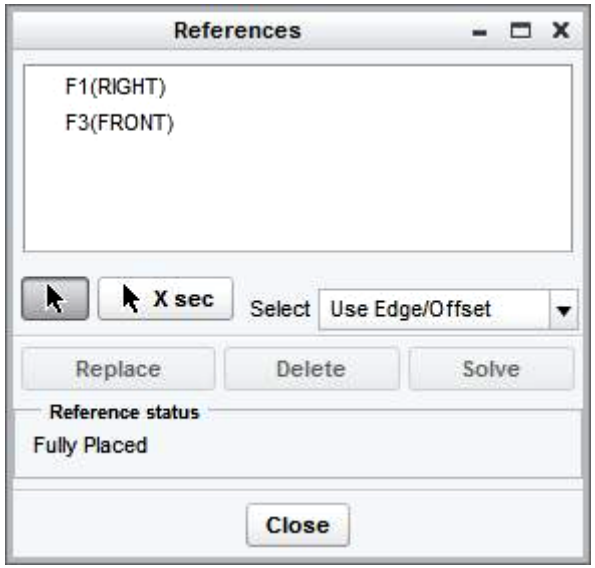


Fig. 2.10.

Goal	Step	Commands
Create the base cylinder (continued)	9. Draw an outer circle.	Circle → ***Select the center of the circle as the intersection of the FRONT and RIGHT datum planes*** → Refer to Fig. 2.11. The cursor snaps onto the intersection. ***Select a point to define the outer edge of the circle*** Refer to Fig. 2.11.
	10. Create an inner circle.	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 to Fig. 2.11.
	11. Modify the dimensions.	Creo automatically places dimensions for the circles. A good practice is to modify smaller dimensions first. → ***Double click the inner diameter dimension*** → **1**→ ***ENTER*** → ***Double click the outer diameter dimension*** → **2** → ***ENTER*** Creo automatically regenerates the section. Refer to Fig. 2.12.
	12. Exit sketcher.	OK

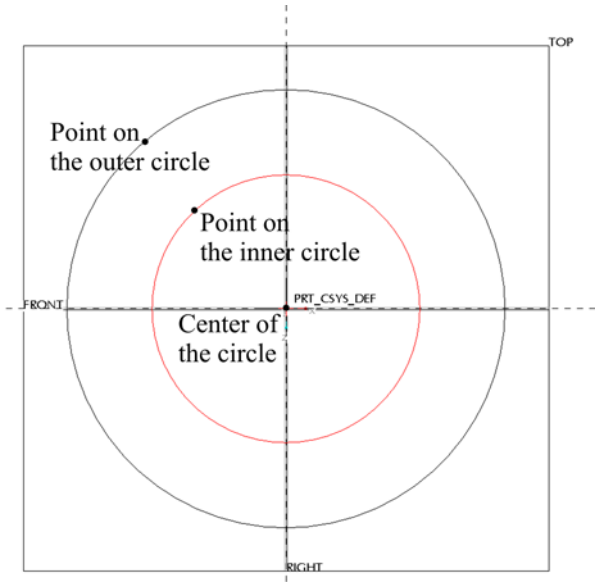


Fig. 2.11.

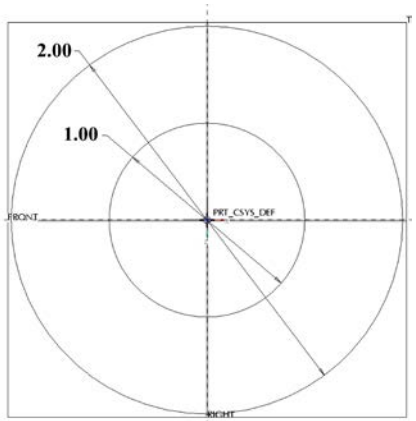


Fig. 2.12.

Goal	Step	Commands
Create the base cylinder (continued)	13. Define the depth.	The depth dimension appears in the dashboard and on the part. Modify the depth at one of these two places. Refer to Figs. 2.13 and 2.14. ***Select the depth dimension*** → **<u>0.5</u>** → ***<u>ENTER</u>***
	14. Accept the feature creation.	→ → **STANDARD ORIENTATION** Refer to Fig. 2.15.
Round the four edges	15. Round the four edges of the bearing.	Round → Specify the radius of the rounds to be 0.025. **<u>0.025</u>** → ***<u>ENTER</u>*** → Refer to Fig. 2.16. ***Select the four edges to be rounded while holding <u>CTRL</u>*** → The four edges are rounded regardless of the ***<u>CTRL</u>*** key. Holding ***<u>CTRL</u>*** places the four edge rounds in one round set. Therefore, one parameter, the radius of the round, controls the geometry of all the four rounds. Refer to Fig. 2.17a. Refer to Fig. 2.17b.

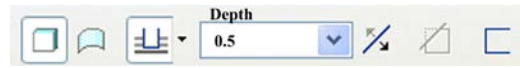


Fig. 2.13.

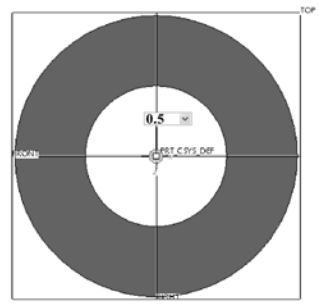


Fig. 2.14.

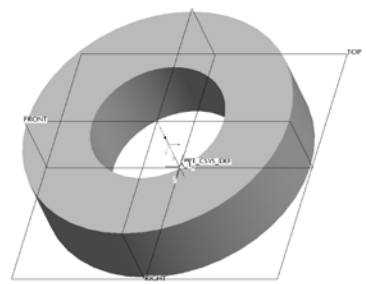


Fig. 2.15.

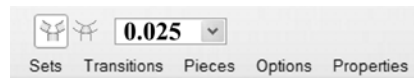


Fig. 2.16.

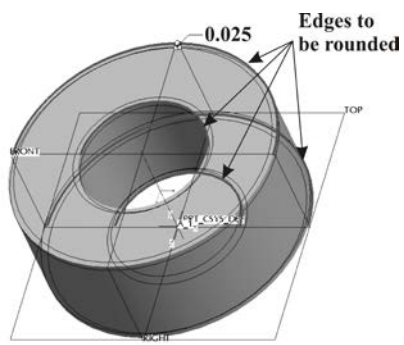


Fig. 2.17a.

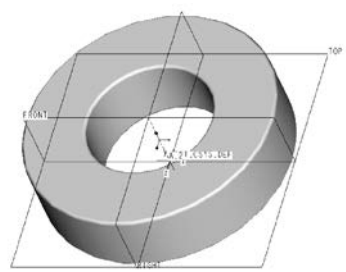


Fig. 2.17b.

Goal	Step	Commands
View the model	16. Turn the datum planes off.	**VIEW →** ***Click the following icons to switch off the datum planes, axes, points, and default coordinate system.***  These icons turn the datum planes, axes, points, coordinate system, and spin center on/off. Refer to Fig. 2.18. Modifying the display helps in visualizing the model better. The six model display options can be selected by clicking on the corresponding icons. Refer to Fig. 2.19. Fig. 2.20 shows the model in the six display types. **VIEW →** ***Click the following icons to switch on the datum planes, axes, points, and default coordinate system.*** 

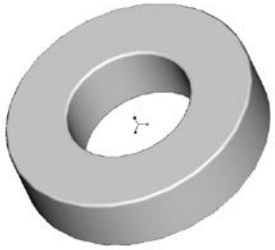


Fig. 2.18.

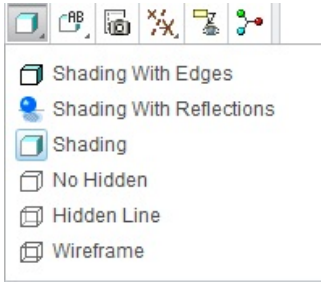


Fig. 2.19.

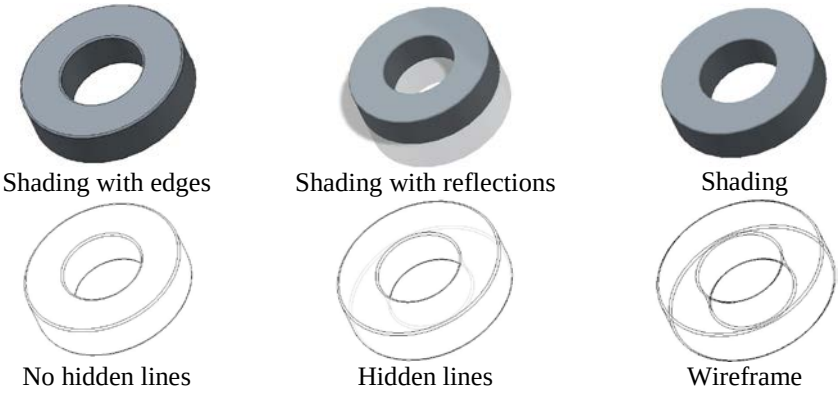


Fig. 2.20.

Goal	Step	Commands
Modify dimensions	17. Modify the dimensions using Edit feature.	***Select the extrusion feature in the graphics window or from the model tree*** → ***Right Mouse*** → **Edit** → (May need to hold down the right mouse button to for it to take effect.) Refer to Fig. 2.21. ***Select the 1.0 dimension*** → **<u>0.6</u>** → ***<u>ENTER</u>*** The model automatically changes to the new dimensions. Refer to Fig. 2.22.
	18. Modify the dimensions using dynamic edit feature.	**MODEL** →***Expand***  ***menu***→  → ***Select the extrusion feature in the graphics window or from the model tree*** → ***Right Mouse*** → **Edit** → ***Select the 2.0 dimension*** → **<u>1.25</u>** → ***<u>ENTER</u>*** → Note that the part did not change in size.  The modifications take effect after regeneration. Refer to Fig. 2.23.

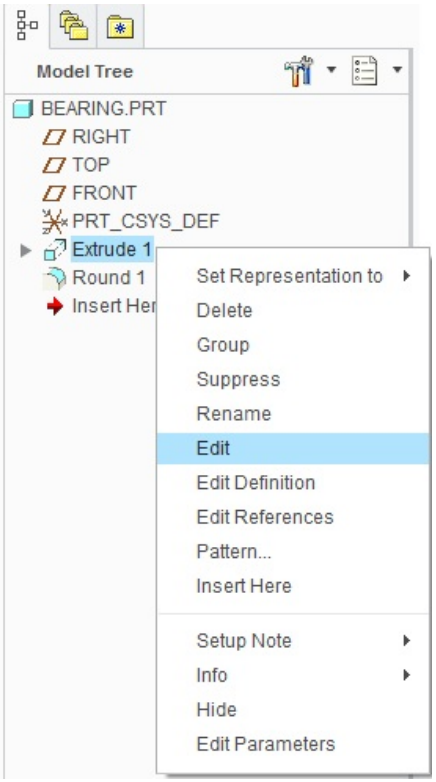


Fig. 2.21.

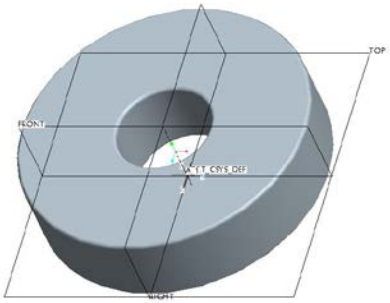


Fig. 2.22.

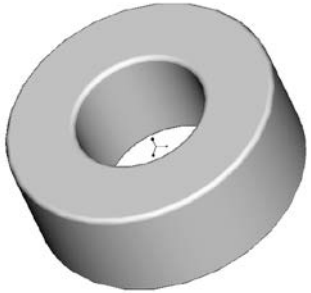


Fig. 2.23.

Goal	Step	Commands
Use the model player	19. Use the model player to view the feature creation sequence.	**TOOLS →** Model Player **→** Refer to Fig. 2.24. **→ → → → → → (till you see Feat # 5 of 6) →** **Show Dims →** Note that Creo shows the final dimensions. Refer to Fig. 2.25. **FINISH**
Save the file and exit Creo	20. Save the file and exit Creo.	**FILE → SAVE → BEARING.PRT → OK → FILE → EXIT → Yes**

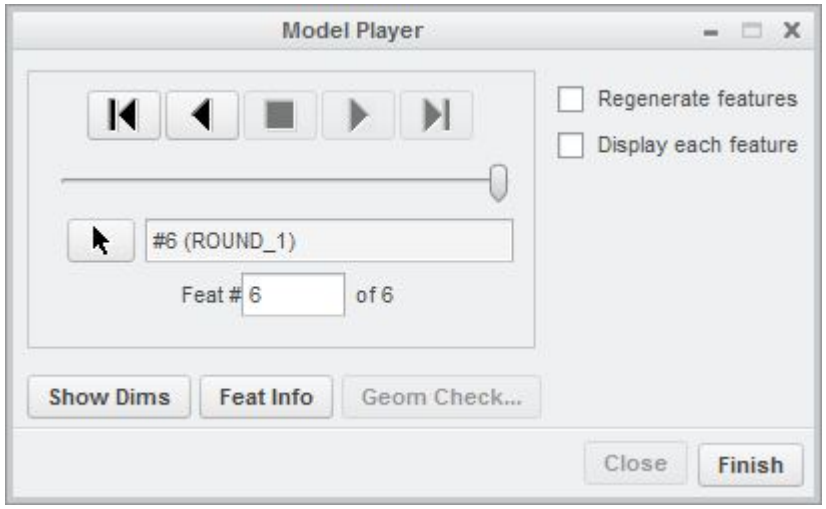


Fig. 2.24.

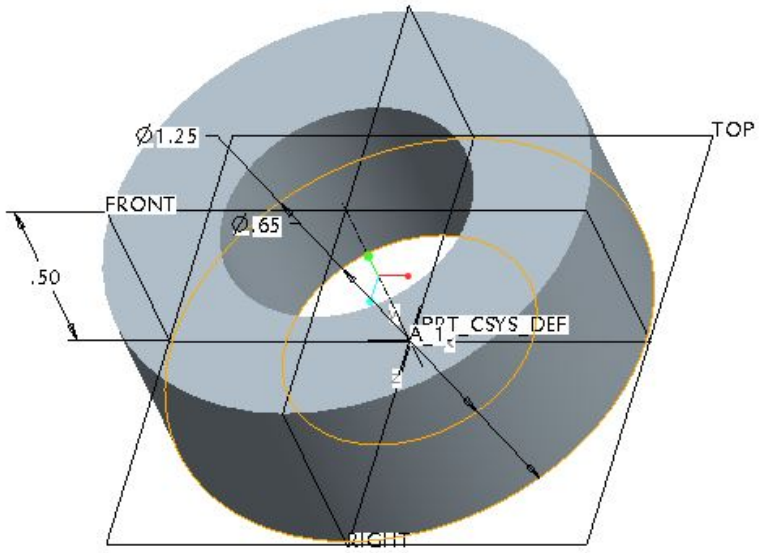


Fig. 2.25.

Goal	Step	Commands
Use trail file to recreate the session	21. Trail.txt file location depends on the configuration. Find its location by searching for trail.txt. Note that there can be several trail.txt files. Identify the correct one by checking the time it was created.	
	22. Rename the trail.txt file as bearing.txt.	
	23. Start notepad and open bearing.txt file.	
	24. Search for word “bearing” and replace it as “new_bearing”. Now, when the trail file is played, it creates a part - new_bearing.	
	25. Delete the highlighted portion in the trail.txt file.	
	26. Use save as and write the file name in quotes - “bearing.txa”. Now, we are changing the file extension to eliminate version number. Extension .txa is for training file. Exit notepad.	
	27. Open Creo.	
	28. Play trail file.	**TOOLS → MANAGE SESSIONS →PLAY TRAIL FILE → Select “bearing.txa” file → OPEN** Creo recreates the session. By deleting the highlighted section, Creo does not exit at the end of the trail file.
Exit Creo	29. Exit Creo.	**FILE → EXIT → Yes**

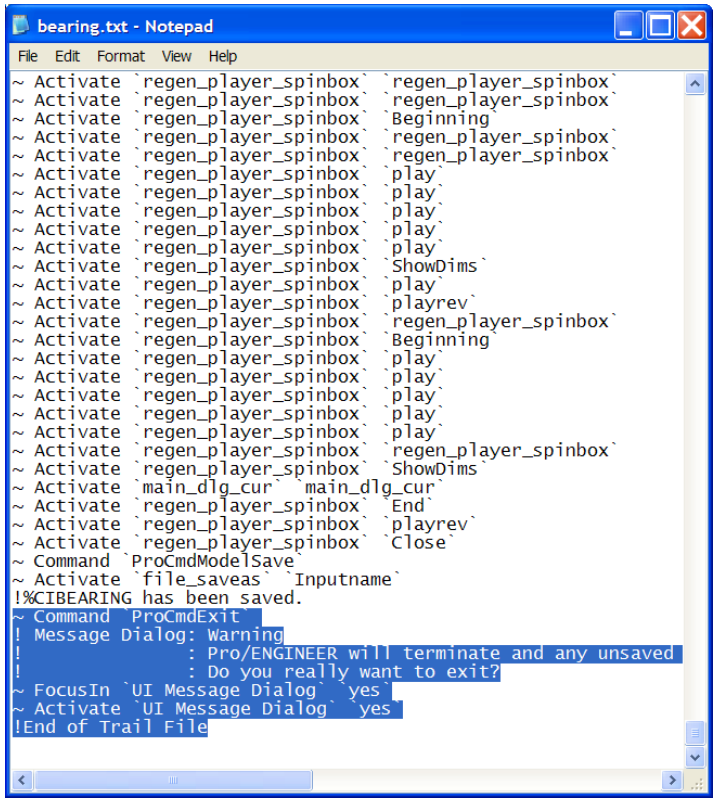
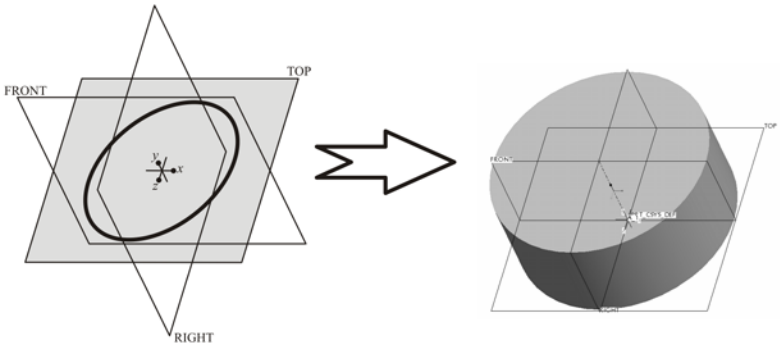
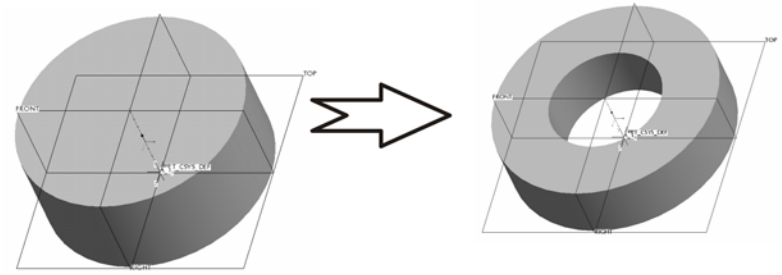


Fig. 2.26.

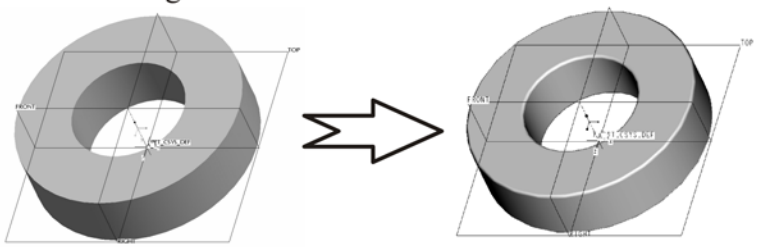
Step I - Extrude the base cylinder
1. Use “Extrude” feature.
2. Define the sketch plane
3. Sketch a circular section
4. Define the depth of extrusion



Step II - Create the circular cut
1. Use “Extrude-Cut” feature
2. Define the sketch plane
3. Sketch the inner circle
4. Define the depth of extrusion



Step III - Round the edges of the base cylinder
1. Use “Round” feature
2. Specify the radius
3. Select the edges to be rounded



SEQUENCE OF STEPS

Goal	Step	Commands
Open a new file for the bearing part	1. Set up the 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* → bearing1 → OK** Refer to Fig. 2.27. In the graphics window, Creo displays the three default datum planes (FRONT, TOP, and RIGHT), and the default part coordinate system (PRT_CSYS_DEF) at the intersection of the three datum planes. Refer to Fig. 2.28.
Create the base cylinder	3. Start "Extrude" feature.	**MODEL →** 

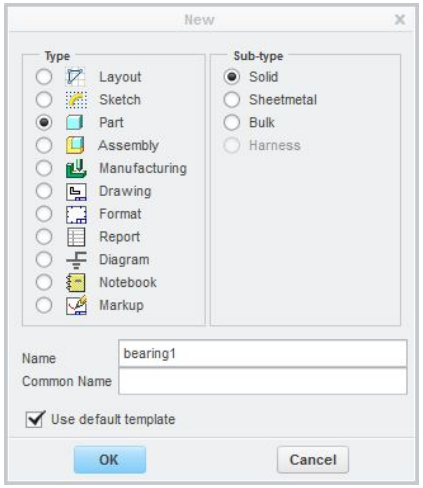


Fig. 2.27.

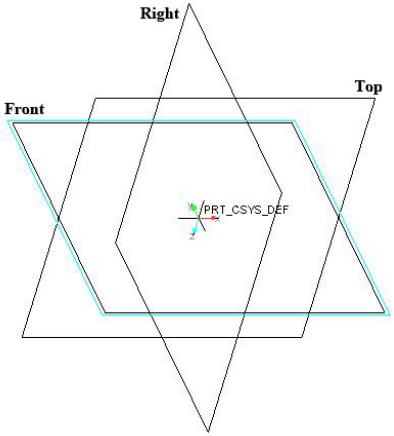


Fig. 2.28.

Goal	Step	Commands
Create the base cylinder (continued)	4. Define the sketch plane.	To select the sketch plane, ***click Placement*** **(in the dashboard) →** **Define** Refer to Fig. 2.29. Creo brings up the "Sketch" window where we define the sketch plane. Refer to Fig. 2.30. We are going to sketch the section on the TOP datum plane. ***Select the TOP datum plane in the graphics window or in the model tree by clicking on "TOP" →*** Refer to Fig. 2.3. **Sketch**
	5. Draw an outer circle.	Circle → ***Select the center of the circle as the intersection of the FRONT and RIGHT datum planes →*** Refer to Fig. 2.32. The cursor snaps onto the intersection. ***Select a point to define the outer edge of the circle*** Refer to Fig. 2.32.

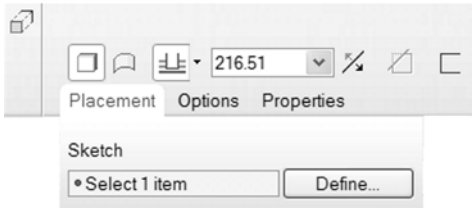


Fig. 2.29.

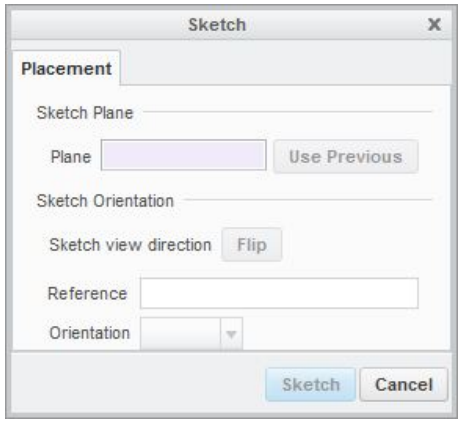


Fig. 2.30.

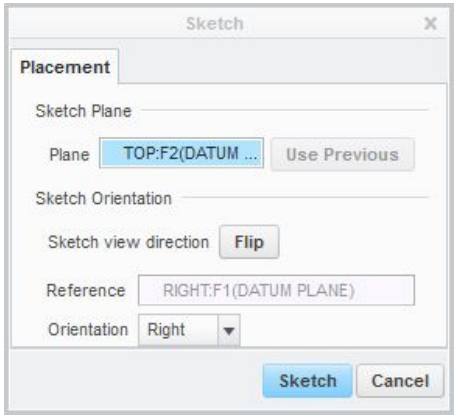


Fig. 2.31.

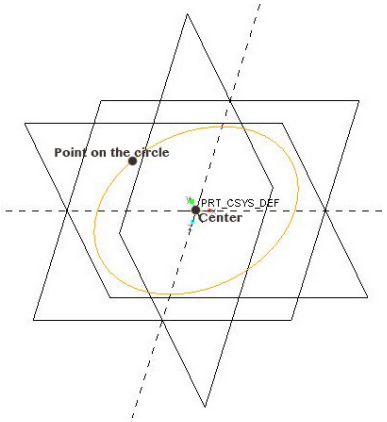


Fig. 2.32.

Goal	Step	Commands
Create the base cylinder (continued)	6. Modify the dimension.	Creo automatically places dimensions for the circles. → ***Double click the diameter dimension*** → **<u>1.25</u>** → ***<u>ENTER</u>*** Creo automatically regenerates the section. Refer to Fig. 2.33.
	7. Exit sketcher.	OK
	8. Define the depth.	***Select the depth dimension*** → **<u>0.5</u>** → ***<u>ENTER</u>*** Refer to Fig. 2.34.
	9. Accept the feature creation.	→ **VIEW** → **STANDARD ORIENTATION** Refer to Fig. 2.35.
Create the central hole	10. Start "Extrude – Cut" feature.	Extrude → Refer to Fig. 2.36.
	11. Define the sketch plane.	To select the sketch plane, ***click Placement*** **(in the dashboard)** → **Define** Creo brings up the "Sketch" window. Refer to Fig. 2.35. Sketch the section on the previous sketch plane - the TOP datum plane. **Use Previous**

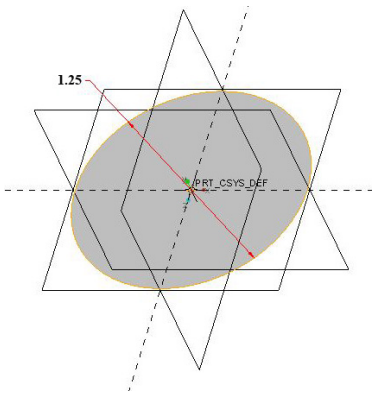


Fig. 2.33.

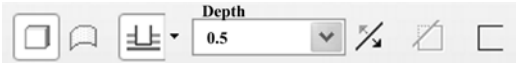


Fig. 2.34.

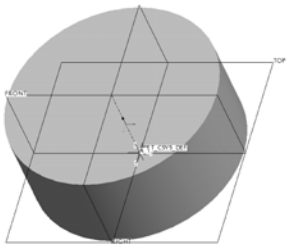


Fig. 2.35.

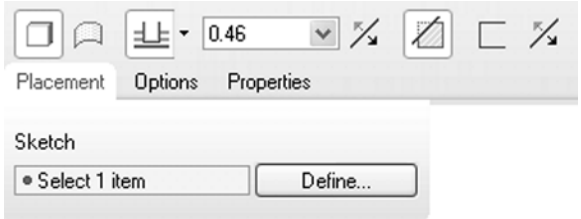


Fig. 2.36.

Goal	Step	Commands
Create the central hole (continued)	12. Draw an inner circle.	→ Circle → ***Select the center of the circle as the intersection of the FRONT and RIGHT datum planes*** → Refer to Fig. 2.37. The cursor snaps onto the intersection. ***Select a point to define the outer edge of the circle*** Refer to Fig. 2.37.
	13. Modify the dimension.	→ ***Double click the diameter dimension*** → **<u>0.60</u>** → ***<u>ENTER</u>*** Creo automatically regenerates the section.
	14. Exit sketcher.	OK

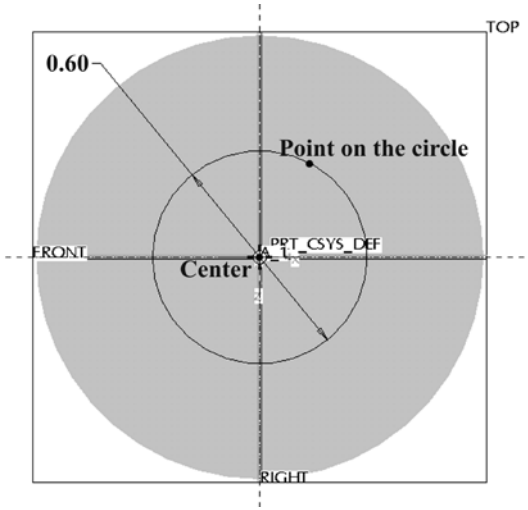


Fig. 2.37.

Goal	Step	Commands
Create the central hole (continued)	15. Define the depth.	→ **STANDARD ORIENTATION** → → (Wireframe icon) →***Click on Extrude tab*** Notice the cut (red arrow) point away from the TOP datum plane. Refer to Fig. 2.38. Change the depth direction to the other side of the sketch. (before the cut icon) Select the depth option as extrude to intersect with all surfaces. Refer to Fig. 2.39.
	16. Accept the feature creation.	→ → (shading) Refer to Fig. 2.40.

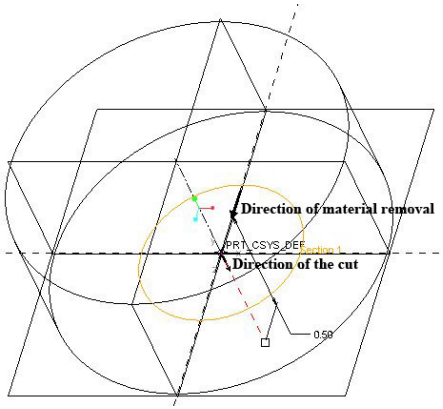


Fig. 2.38.

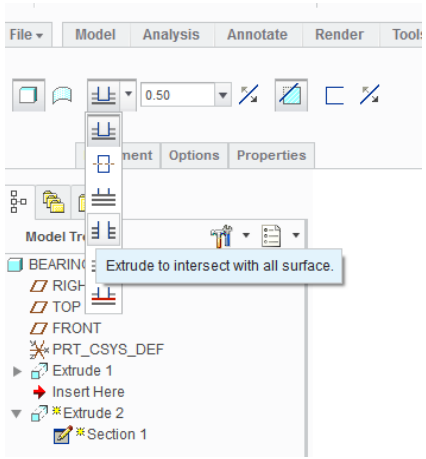


Fig. 2.39.

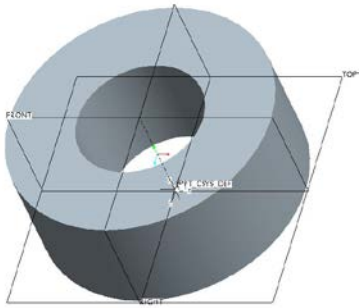


Fig. 2.40.

Goal	Step	Commands
Round the four edges	17. Round the four edges of the bearing.	 → Specify the radius of the rounds to be 0.025. **<u>0.025</u> → *<u>ENTER</u>* →** Refer to Fig. 2.41. ***Select the four edges to be rounded while holding <u>CTRL</u>* →** Refer to Fig. 2.42.  Refer to Fig. 2.43.
Save the file and exit Creo	18. Save the file and exit Creo.	**FILE → SAVE → <u>BEARING1.PRT</u> → OK → FILE → EXIT → Yes**

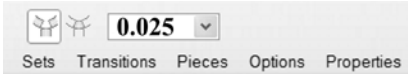


Fig. 2.41.

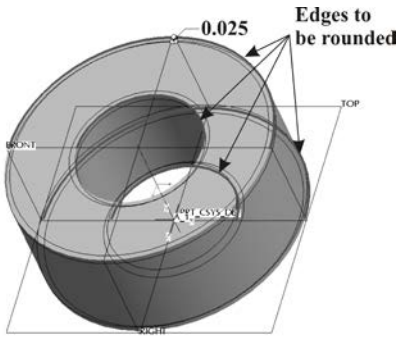


Fig. 2.42.

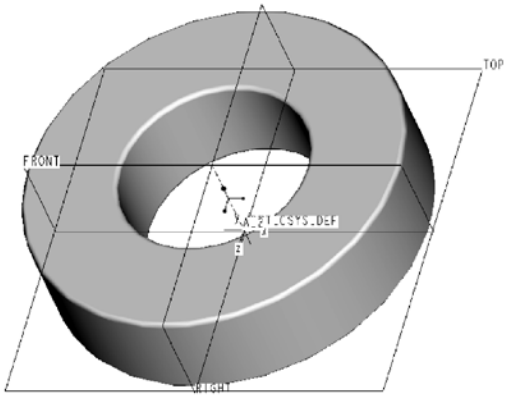
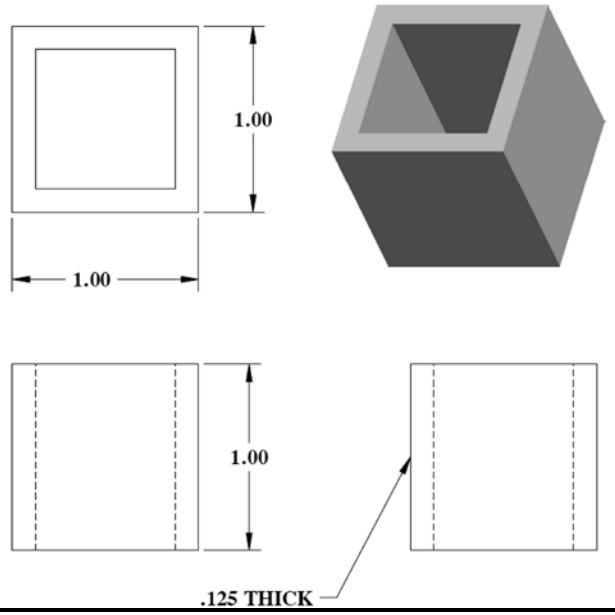


Fig. 2. 43.

Exercises

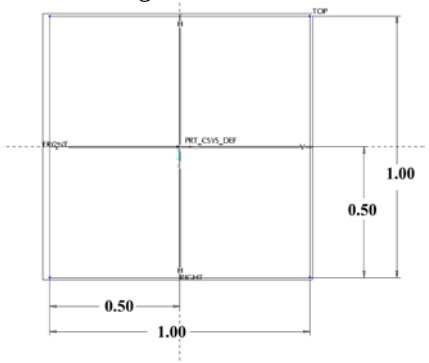
Problem 1



Hints:

1. Start the extrude feature. Create a rectangle (1×1) using the create rectangle tool 

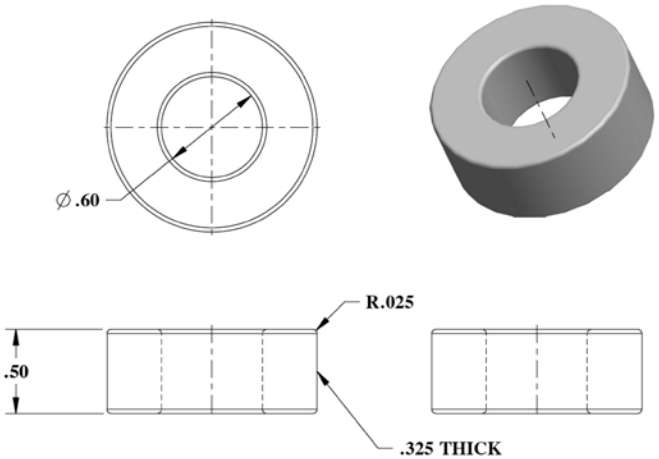
in the sketcher.



2. Select thicken option and define the thickness as 0.125. The dash is shown in the figure below.

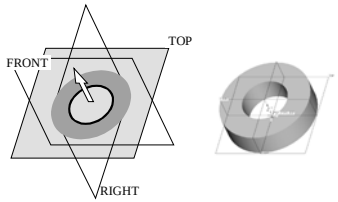
Experiment with the two  icons and discover what they do.

Problem 2

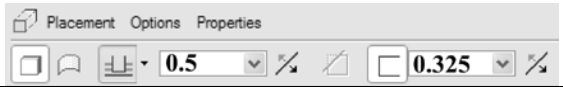


Hints:

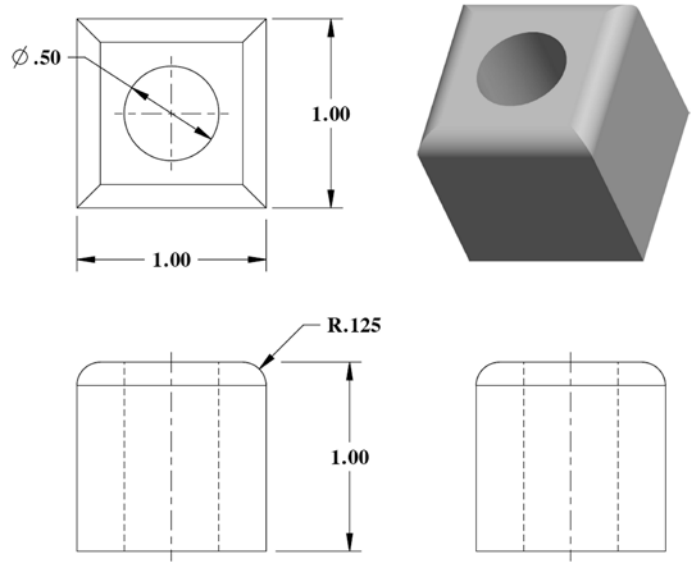
1. Start the extrude feature.
2. Select the TOP plane as the sketch plane. Sketch a 0.6″ diameter circle. Exit sketcher.



3. Define the depth as 0.5″.
4. Select thicken option and define the thickness as 0.325. Flip the direction of material addition (inside, outside, and both sides of the circle) by clicking the second  . The dash is shown below.

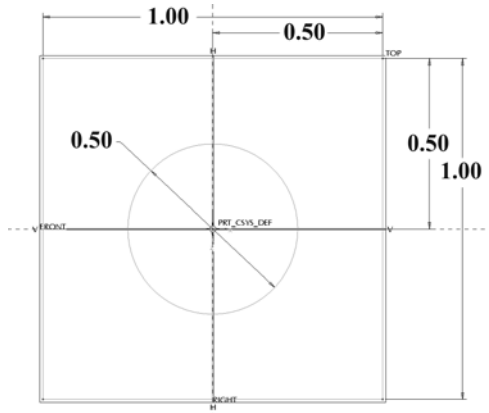


Problem 3



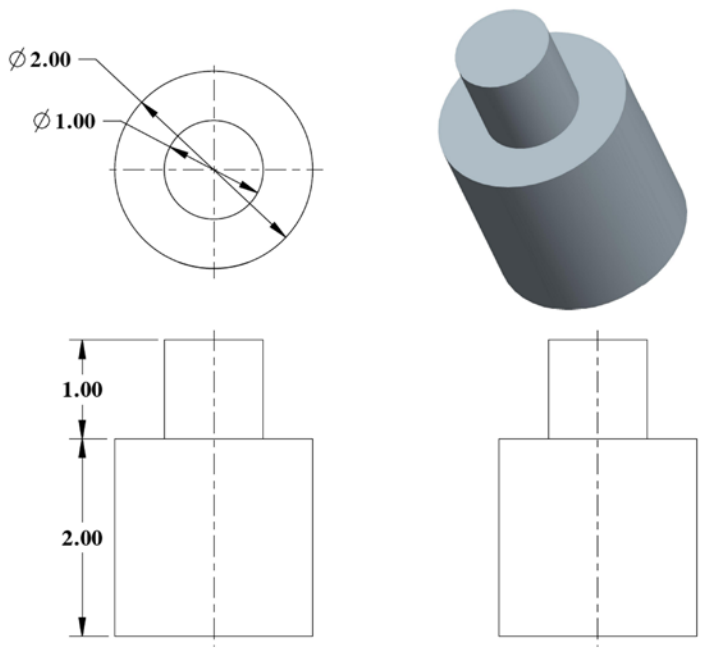
Hints:

1. Use extrude feature. The sketch is shown in the figure below.



2. Use round feature.

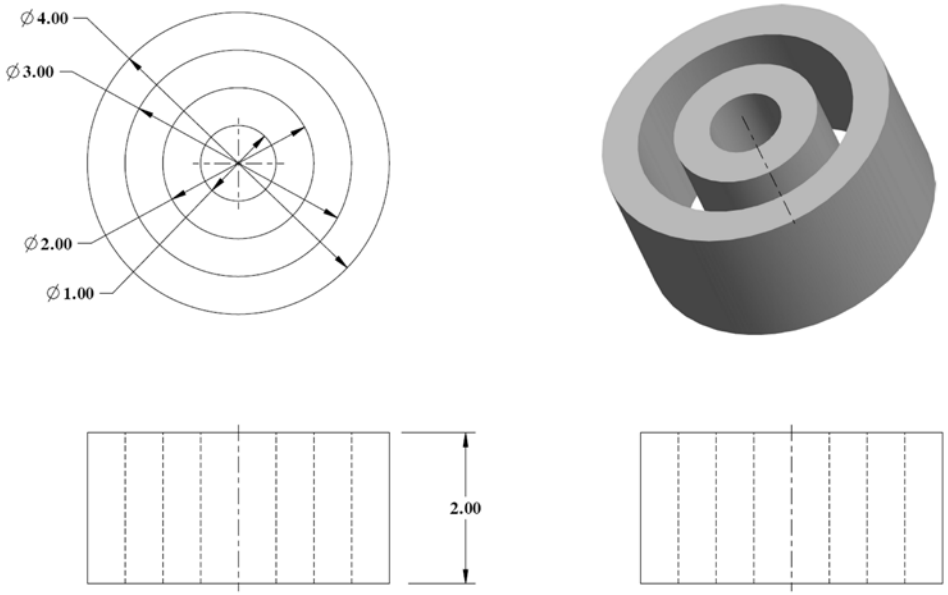
Problem 4



Hints:

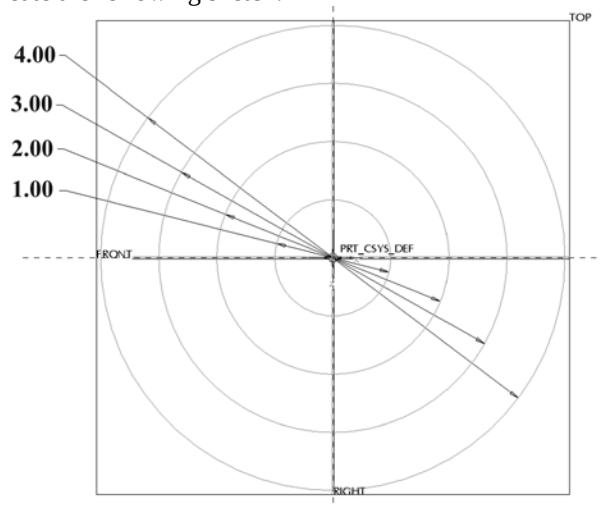
1. Create the first cylinder.
2. Select the top of the first cylinder as the sketch plane for creating the second cylinder.

Problem 5

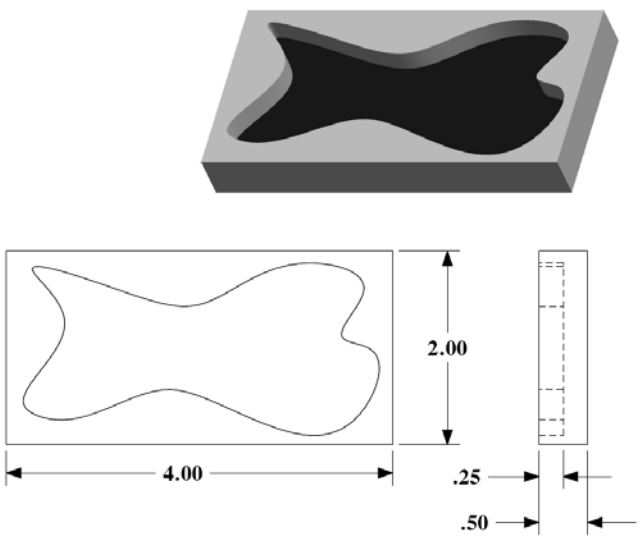


Hints:

1. Start extrude feature.
2. Create the following sketch:



Problem 6



Hints:

1. Create a rectangular block.
2. Create an extrude cut. Use the spline feature (

) in the sketcher to draw the cut profile.

OPEN-ENDED DESIGN – Explore the sketcher & create your own logo.

Hints:

1. Explore the sketcher tools.
2. Make sure that you read the message window when creating any sketches.
3. You may create several extrusions one at a time. Remember that you cannot extrude intersecting geometric entities.
4. The palette 

has several sections that can be imported directly. Double click the section and then, click in the graphics window to drop the section.