UG Exercise #5

Start a new file and name it “exercise5”. Set the APPLICATION to MODELING.

Use the INSERT > CURVE > RECTANGLE command to create a rectangle in the XY plane. For the purposes of this exercise, the exact dimensions of the rectangle are not critical.

Use the RMB menu to REPLACE your current display orientation with TFR-ISO.

Select INSERT > FORM FEATURE > EXTRUSION

Select as the Section String the four curves of the rectangle.

Select the EXTRUSION METHOD of DISTANCE & DIRECTION.

The Vector Constructor dialogue will open and an arrow will appear indicating the current direction in which the extrusion will be constructed. The default setting for the extrusion direction is normal to the plane of the Section String. Select OK in the Vector Constructor dialogue to accept this default direction.

Set the START DISTANCE at zero. This will begin the extrusion at the Section String. Set the END DISTANCE to a value of 4. Press the MMB to accept this input.
The solid feature will be constructed.

DISPLAY COMMENTS: When dealing with solid and surface geometries, you can set the display parameters to various settings to make your work easier. Using the RMB menu option, HIDDEN EDGES, you can set how the hidden edges in a wireframe display will appear. These hidden edges can be shown as NARROW GREY LINES, DASHED LINE, or not shown at all. You will note that when these settings are made, they will NOT change dynamically with view orientation. Rather you must select RMB menu option UPDATE DISPLAY.

Select the WCS > ORIGIN command and place the coordinate system origin at the front, left-hand corner of the solid. Select the WCS > ROTATE command and rotate the current coordinate system orientation 90 degrees about the positive X-axis.

Use the Quick Screen Menu (RMB), to set Replace the current display orientation with FRONT.

Use the INSERT > CURVE > BASIC CURVES > LINE command to create the shape shown. Make sure that the entire shape lies within the face of the first solid created.

Use the Quick Screen Menu (RMB), to set Replace the current display orientation with TFR_ISO.

Select INSERT > FORM FEATURE > EXTRUSION
Select as the Section String the curves just created.
Select the EXTRUSION METHOD of TRIM TO FACE/PLANE.

Set the extrusion direction as indicated by the arrow in the figure. If this is the default, simply select OK. If this direction is 180 degrees opposite of the default direction, select the dialogue option CYCLE VECTOR DIRECTION.

Set the TRIMMING FACE option to DO NOT EXTEND TRIM FACE.

Select as the Trim Face the face on the backside of the object, in the direction of the extrusion. To select a solid face, pick within the area of the face, not on the boundary. NOTE: You will not be able to directly select this face since it is hidden in the current view. Pick within its area and the selection window will open allowing you to cycle between the faces within whose areas you have picked.

Leave the EXTRUDED BODY PARAMETERS at their default settings and select OK.

Select BOOLEAN OPERATION > SUBTRACT. The new solid feature will be created.

Use the RMB menu option ROTATE to change the view orientation to approximately what is shown in the figure.

Use the WCS > ORIGIN command to move the coordinate system origin to the corner as shown in the figure.

Use the WCS > ROTATE command to change the coordinate system orientation. Rotate 90 degrees about the negative Z-axis and then 90 degrees about the positive X-axis. This should place the XY plane on the left face of your object.

Use the RMB to REPLACE DISPLAY with LEFT
Use the INSERT > CURVE > BASIC CURVE > CIRCLE command to create a circle on the left face of the solid as shown in the figure.

Use the Quick Screen Menu (RMB), to set Replace the current display orientation with TFR_ISO.

Select INSERT > FORM FEATURE > EXTRUSION

Select as the Section String the curves just created. Select the EXTRUSION METHOD of TRIM TO FACE/PLANE.

Set the extrusion direction as indicated by the arrow in the figure.

Set the TRIMMING FACE option to DO NOT EXTEND TRIM FACE.

Select as the Trim Face the face on the left face of the through slot. This will be the first face that the extrusion intersects.

Leave the EXTRUDED BODY PARAMETERS at their default settings and select OK.

Select BOOLEAN OPERATION > SUBTRACT. The new solid feature will be created.