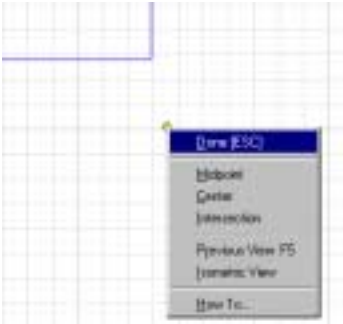


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Step 1:

Create the base feature by sketching a rectangle. Use the rectangle tool from the Sketch toolbar. Select it by picking it with the left mouse button. Locate the rectangle by selecting one corner using the left mouse button and then pick with the left mouse button for the opposite corner.

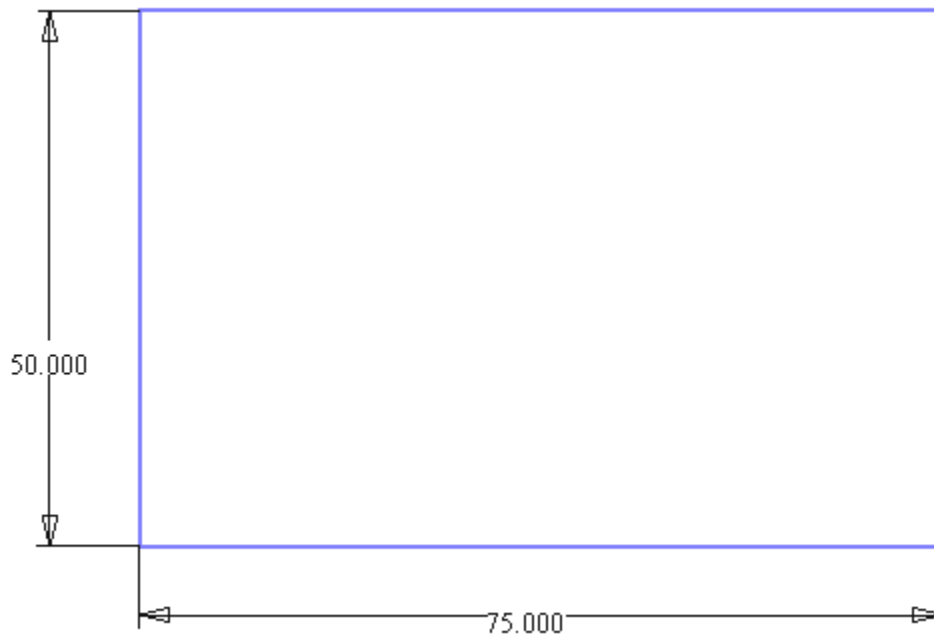


Once the rectangle has been placed, right-click the mouse and select 'Done'.



Next place dimensions using the Auto Dimension tool. Select 'Apply' and 'Done'. Then edit the dimensions as shown.

The rectangle should be 75 units wide and 50 units high as shown.

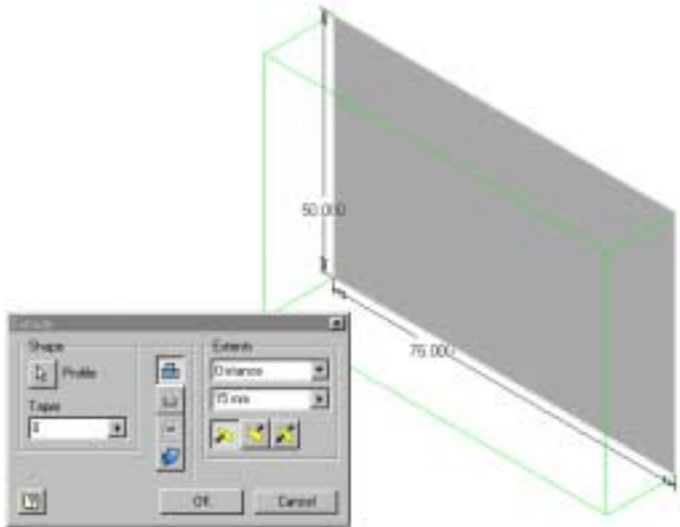


Switch to Isometric View by right-clicking the mouse and selecting 'Isometric View'. Right-click and Select 'Finish Sketch'.

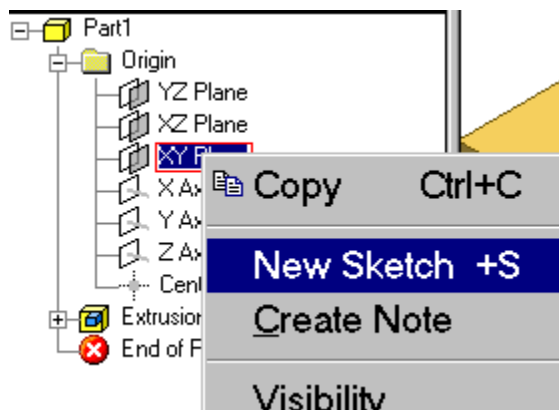


TIP: You can also initiate the Finish Sketch command by pressing *S* on the keyboard.

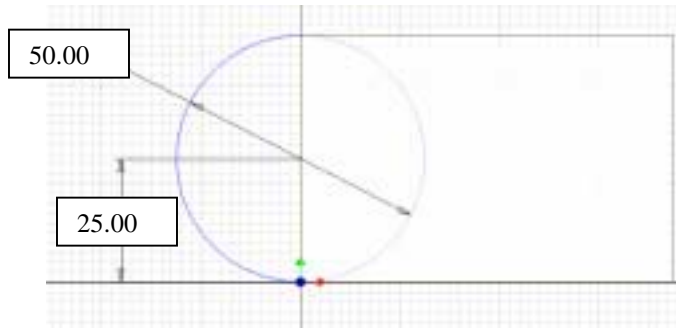
In the Features toolbar, select the Extrude icon (or type *E*)
In the Extrude popup window, enter 15 as the extrusion distance.



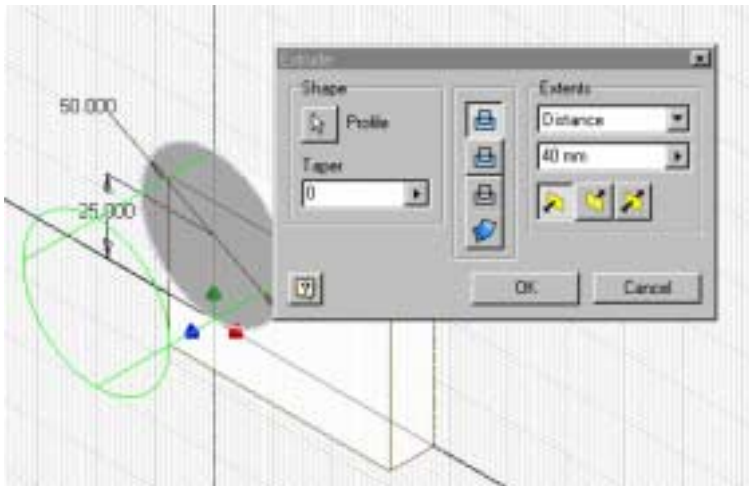
Press 'OK'.



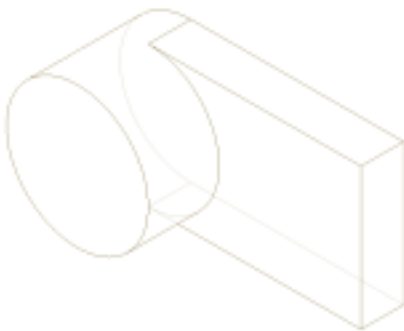
Highlight the XY plane in the Browser. Right click and select 'New Sketch'.



Add a circle using the dimensions shown.
Switch to 'Isometric View'.



Select the 'Extrude' icon.
Use the 'Join' option (the top button in the middle of the Extrude dialog box).
Set the Extents to 'Distance'.
Enter a value of '40'.
The dialog box should appear as shown.
Click on 'OK' to proceed with the Join operation.



Creating a Hole

Next we will add a hole concentric to the circular feature just made.

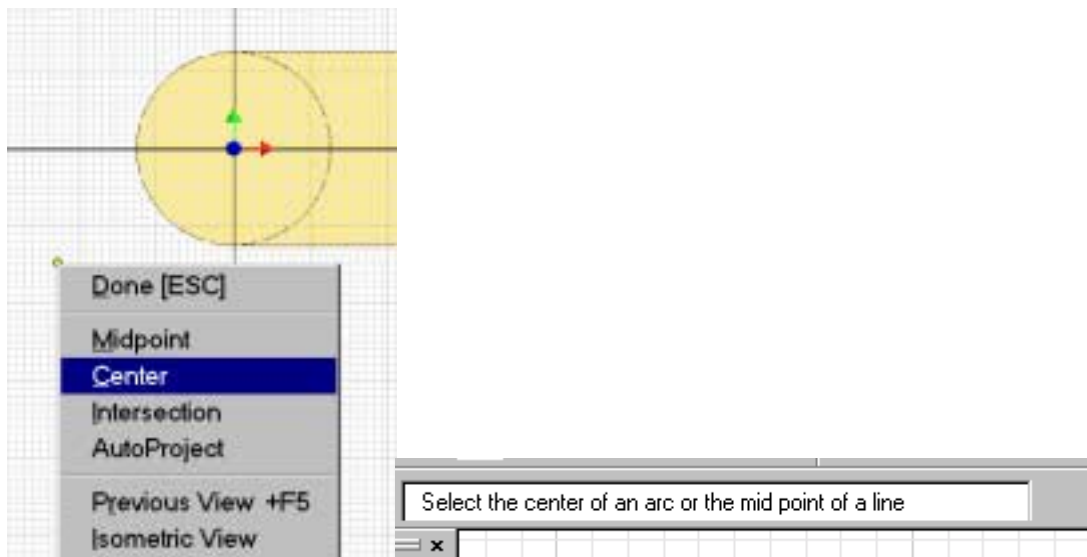


Pick the top of the cylinder with the left mouse so that it highlights. Then right-click and select 'New Sketch' from the popup menu.

Inventor will automatically activate the Sketch toolbar.



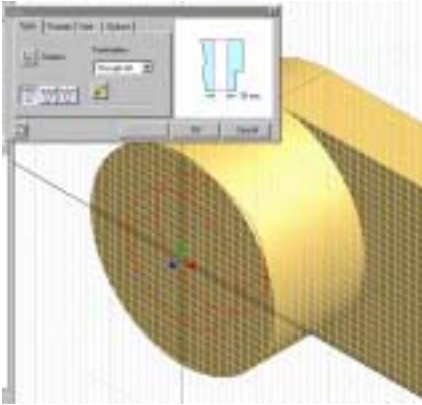
Select the hole point tool. Right click and select the Center option to place the hole point at the center of the cylinder. In the message box, we are prompted to select the center of an arc or the midpoint of a line. Select the center of the cylinder.



Switch to an 'Isometric View' by right-clicking the mouse and selecting 'Isometric View'.

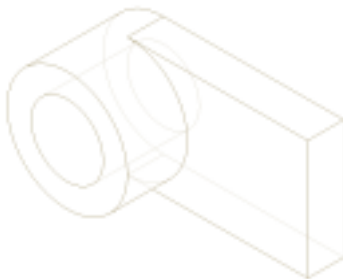


Add a hole using the Hole tool on the Features toolbar.

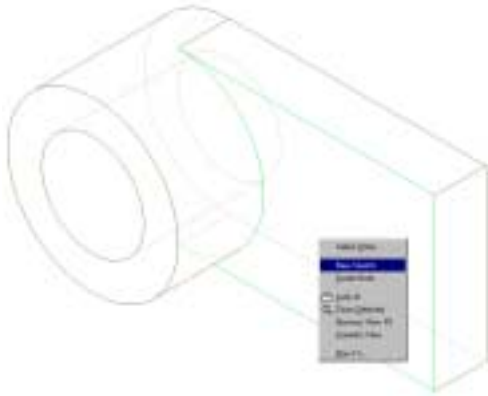


In the Holes dialog box, select a 'Through All' termination and set the diameter to '30'. The diameter can be changed by left-picking on the diameter value and entering a new value.

Press 'OK'.

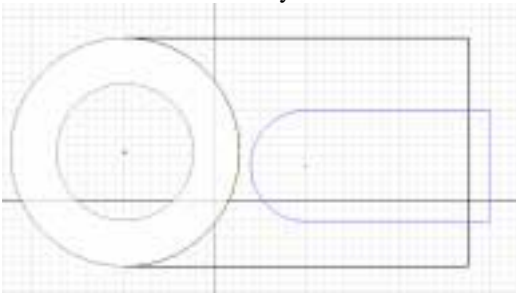


Adding a Cut Feature

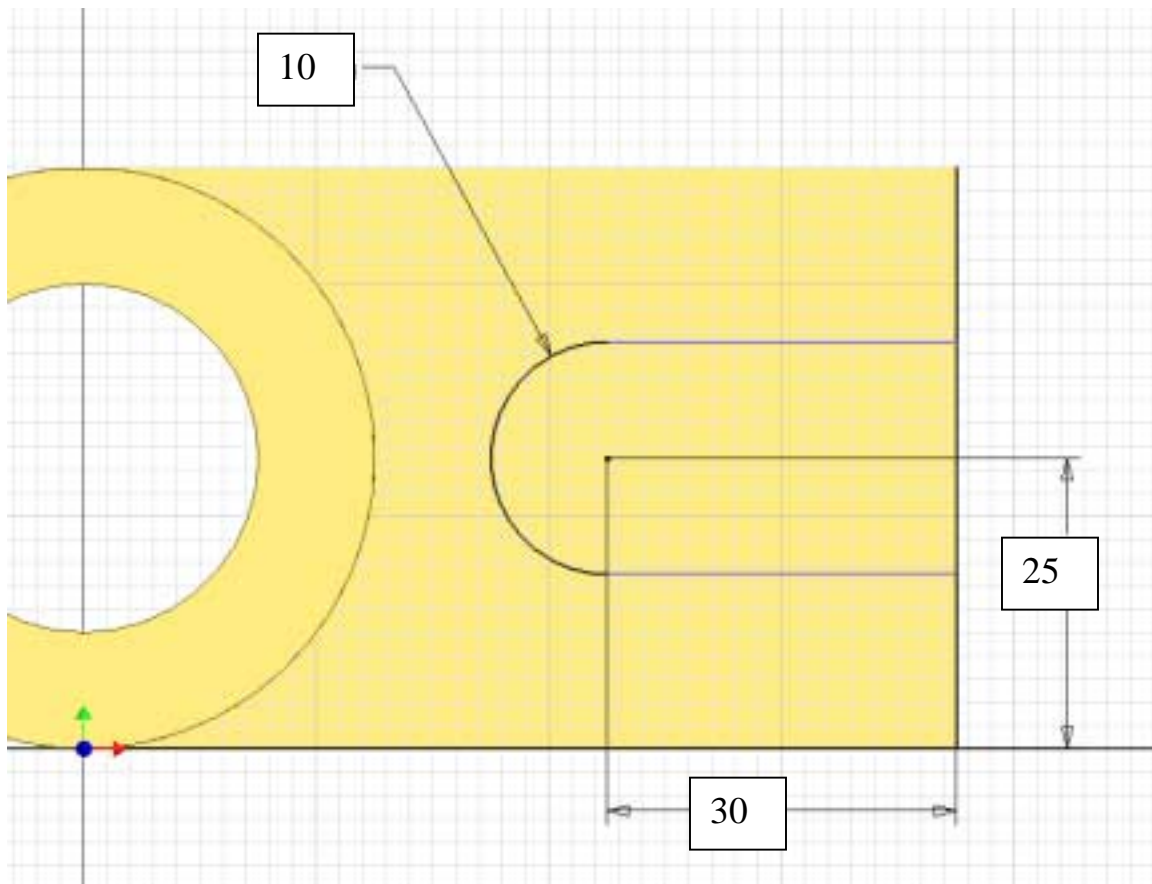


Select the plane shown so that it highlights. Right-click and select 'New Sketch'.

Inventor will automatically activate the Sketch toolbar.



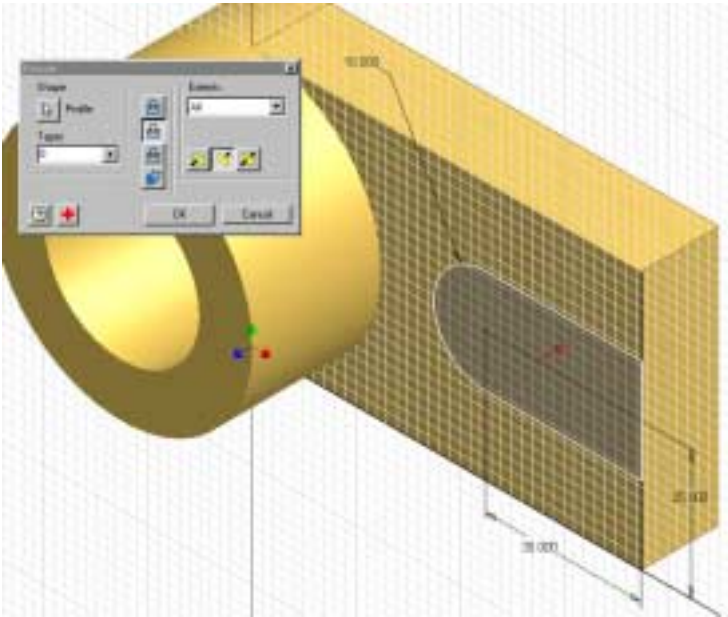
Create the sketch shown. To switch to 'arc' mode, press down the left mouse button and move the mouse in the correct direction to form the arc. To close the figure, right-click and select 'Done'.



Dimension the sketch as shown. Apply geometric constraints as necessary.

For example, a collinear constraint between the right vertical line and the edge of the bracket., Tangent constraints between the arc and the two horizontal lines, and vertical/horizontal constraints.

Select 'Done' and switch to Isometric View.



Select the 'Extrude' icon.

The correct profile should be highlighted. If it is not, select the arrow next to Profile and pick the correct profile.

Activate the 'Cut' option. This is the middle button in the Extrude dialog box.

Extents should be set to 'All'.

The direction arrow should indicate that the cut will be going through the part.

Press 'OK'.



Some users are going to be annoyed by this isometric view of the part.

The reason it looks like this is because the base sketch was drawn on the XY or Front Plane.

We could use the 'Reattach Sketch' tool, but we would have to redefine all our sketches to the correct orientating planes. Another method would be to use the 'Redefine Isometric View' under 3D Common Space Rotation.

Save the completed model as 'lesson11.ipt'.

Notes: