

Lesson 17

Creating a Basic Part

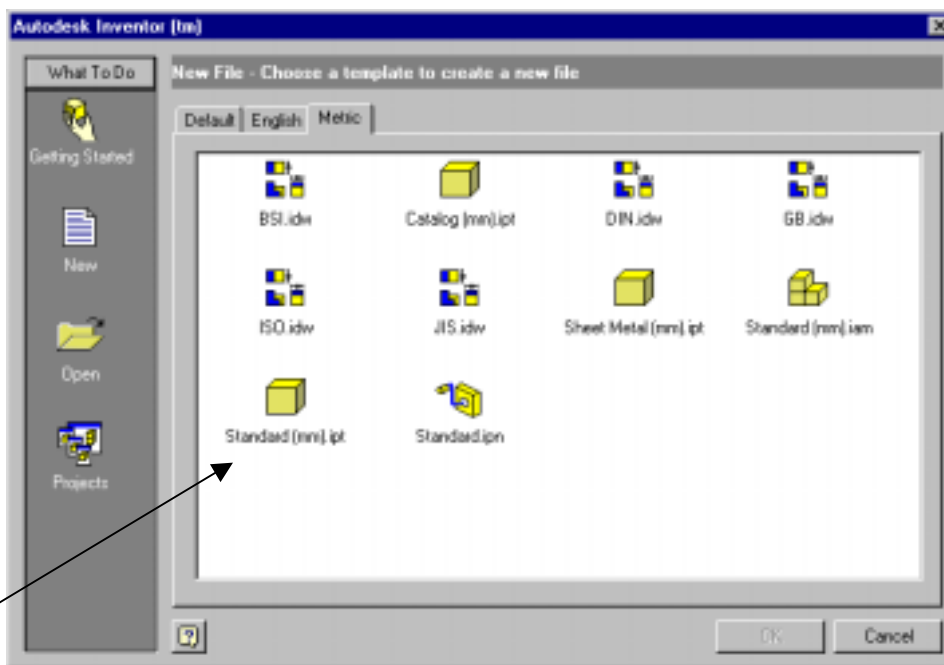
Learning Objectives

This lesson builds the user's modeling skills. The part created will also be used to practice creating auxiliary views and section views. Be sure to save this file for use in those lessons.

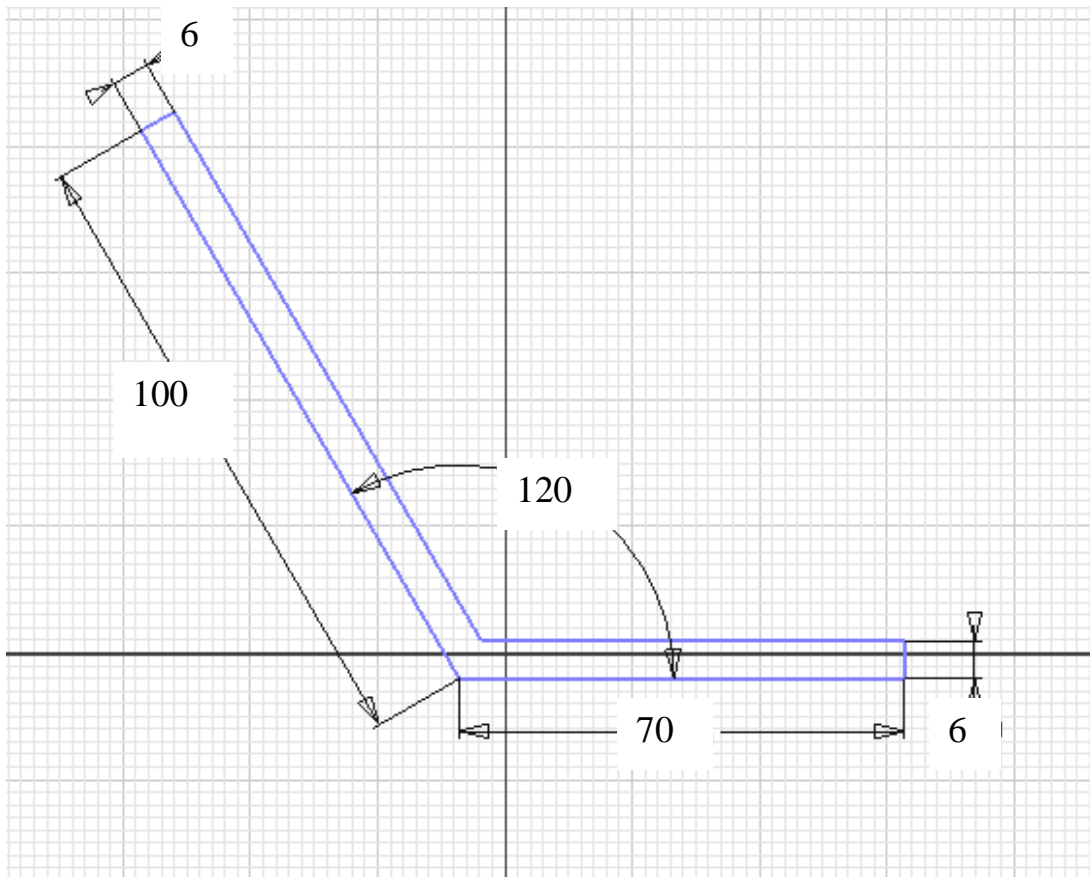
Upon completing this lesson, the user will have gained further mastery of the following tools:

- ◆ Extrude
- ◆ Work Planes
- ◆ Holes
- ◆ Fillet
- ◆ Insert Design Element
- ◆ Rectangular Pattern
- ◆ Rib

Under the menu, select File->New.



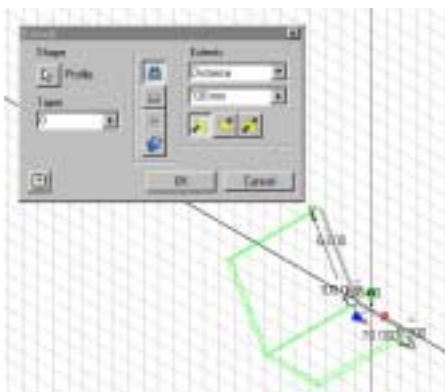
We start by opening a Metric Part file.

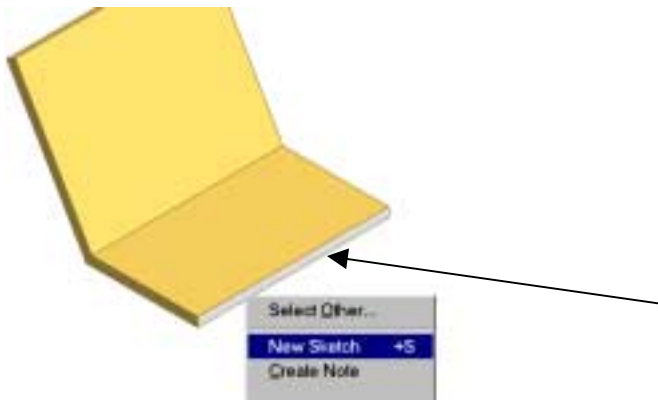


Create the following profile.

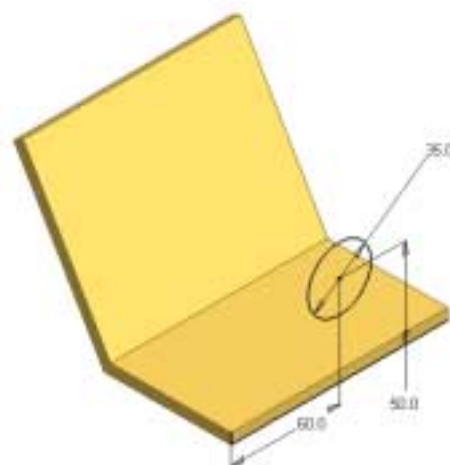
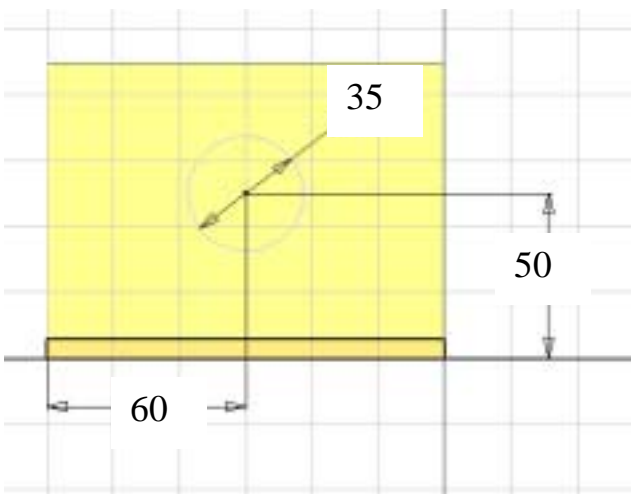
Extruding a Profile

Extrude it as a base feature 120 mm.

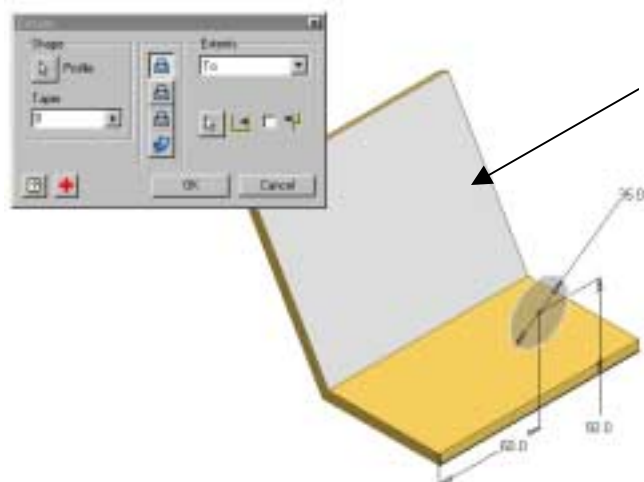




Select the face shown for a New Sketch.



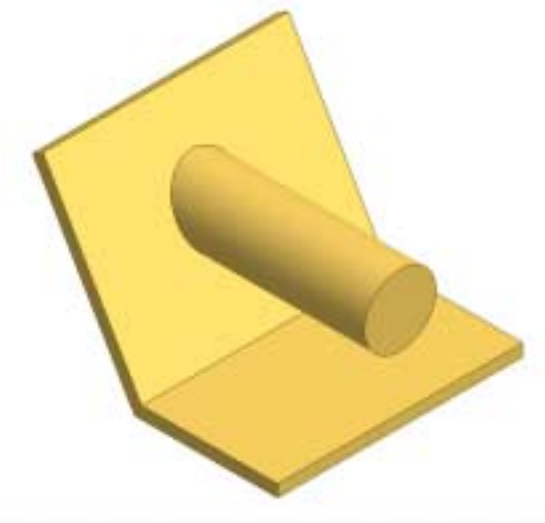
Draw a circle as shown.



Select this face as the 'To' Plane

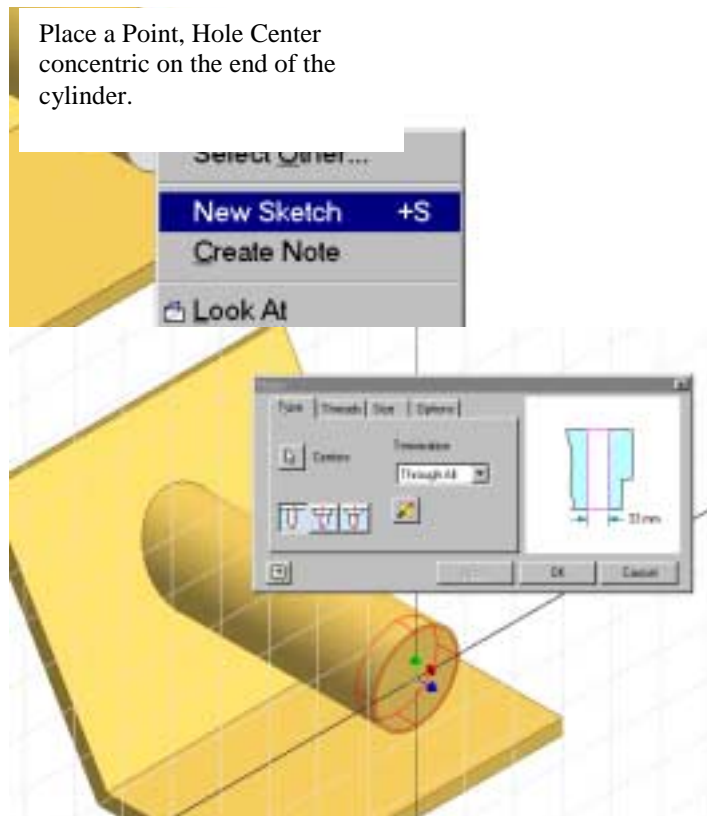
Extrude To a Plane

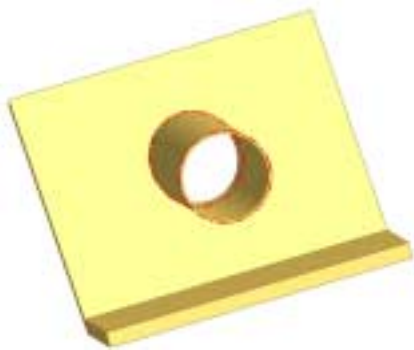
Use Extrude with a 'To' Extents. Select the inclined face as the terminating plane.



Creating a Concentric Hole

Add a 33 mm diameter thru hole concentric to the cylinder.

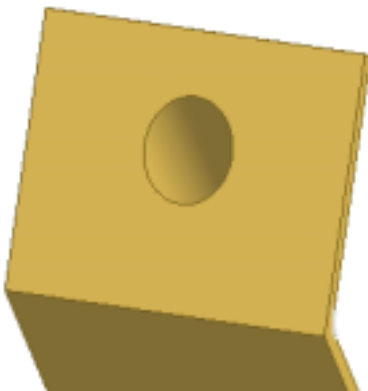
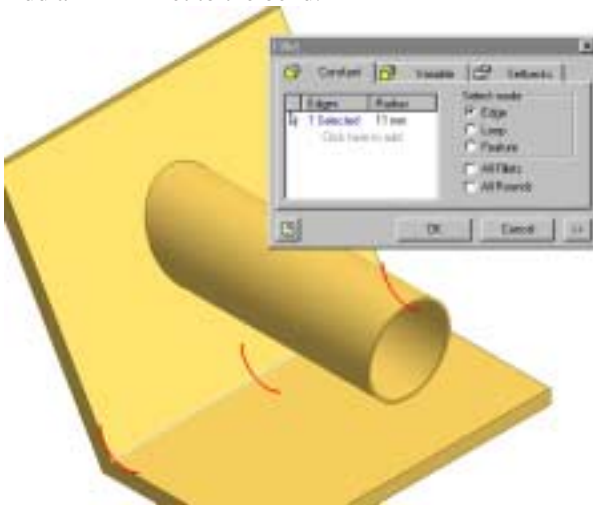




Adding Constant Fillets

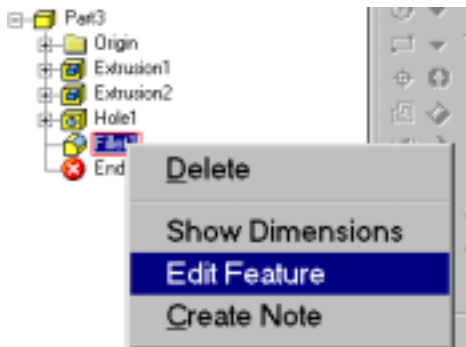


Add an R11 fillet to the bend.

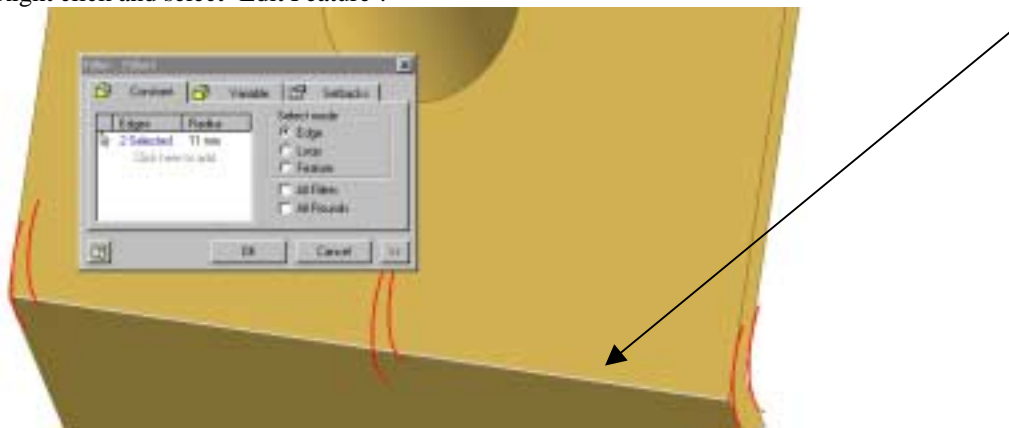


Rotate the part so that you see the back side.

Edit Fillet



Locate the fillet in the browser.
Right click and select 'Edit Feature'.



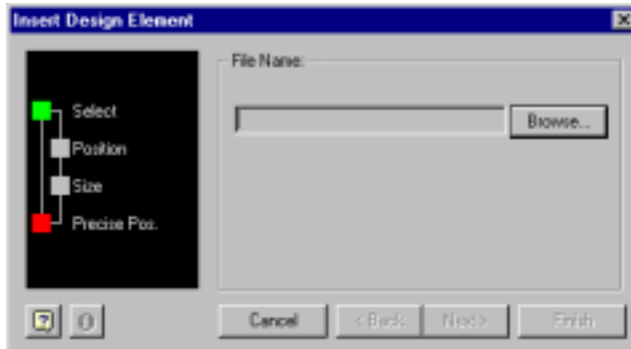
Select the back edge to add to the selection set.
Press 'OK'.

Switch back to an Isometric View.

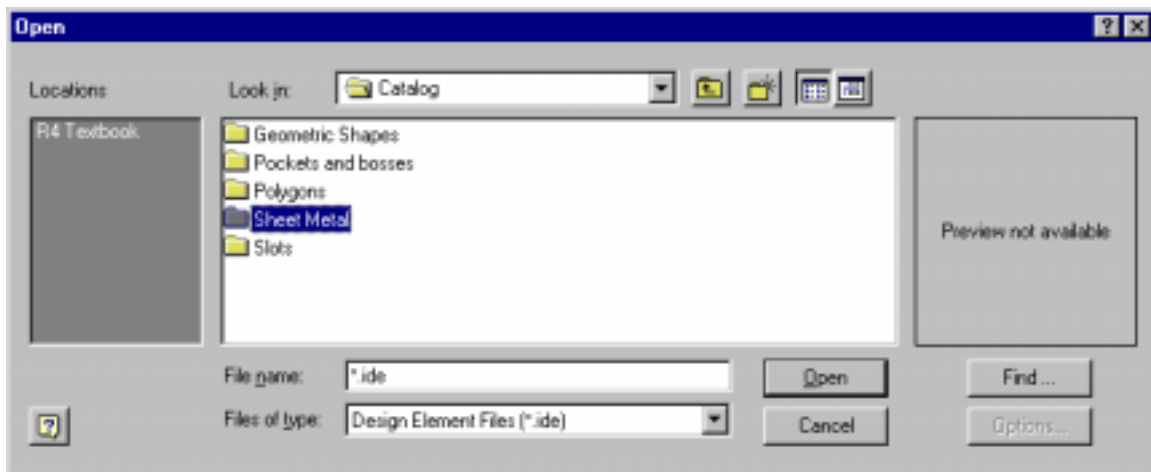


Insert Design Element

Use the Insert Design Element to add a slot to the inclined plane as shown.



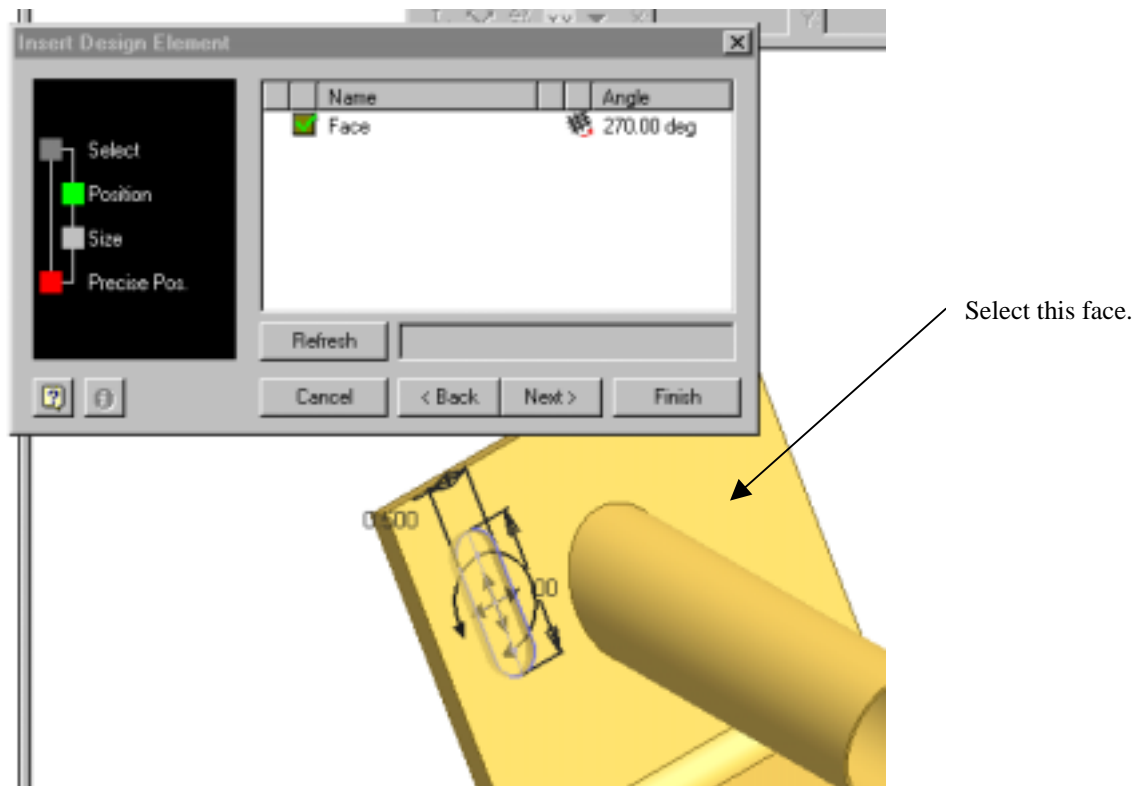
Press the 'Browse' button.



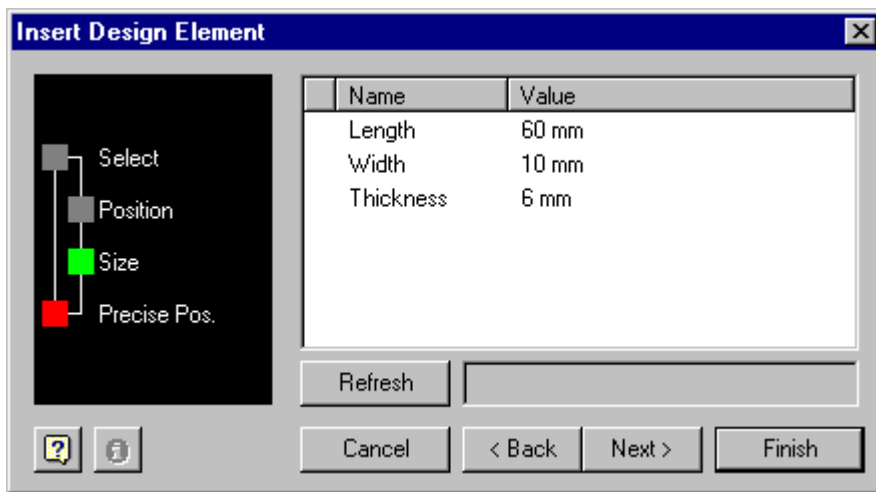
Select the Sheet Metal subdirectory.



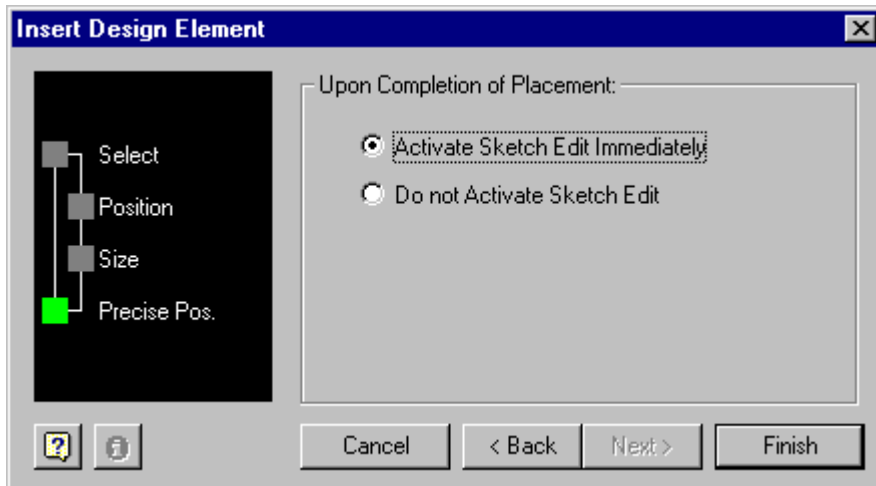
Locate the Obround.ide file under Sheet Metal.



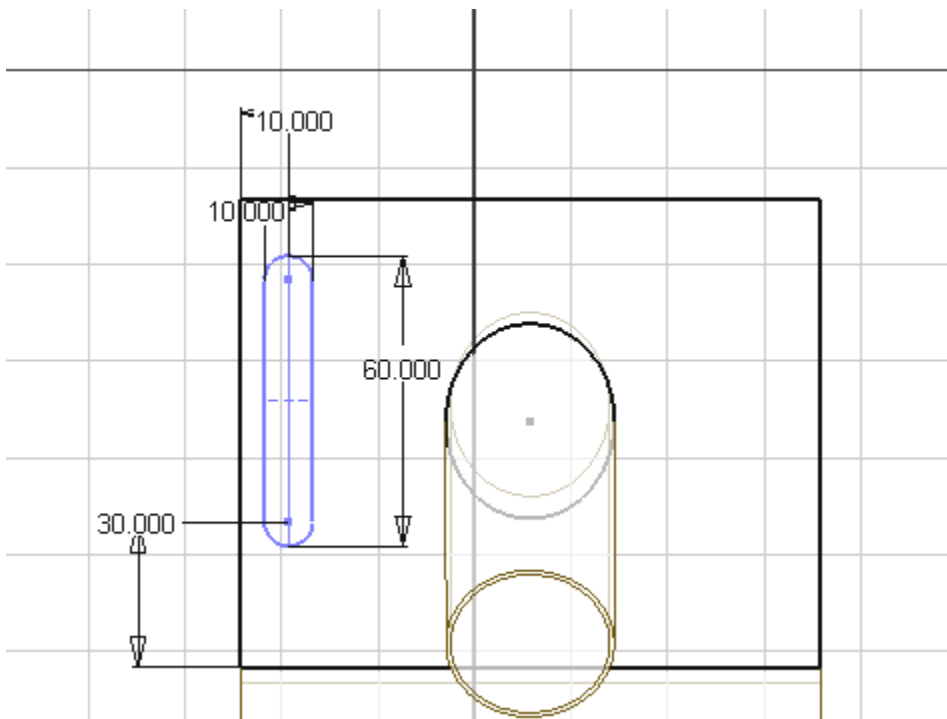
Locate the slot as shown.



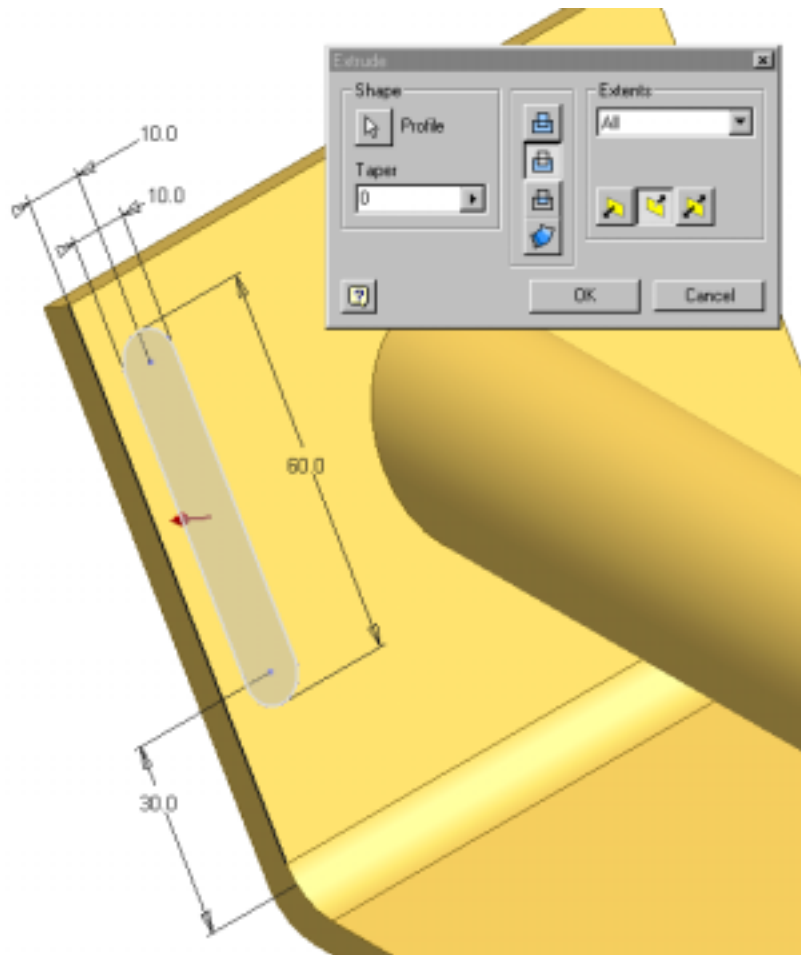
Modify the parameters as shown. Note that we can assign metric values even though the default is inches.



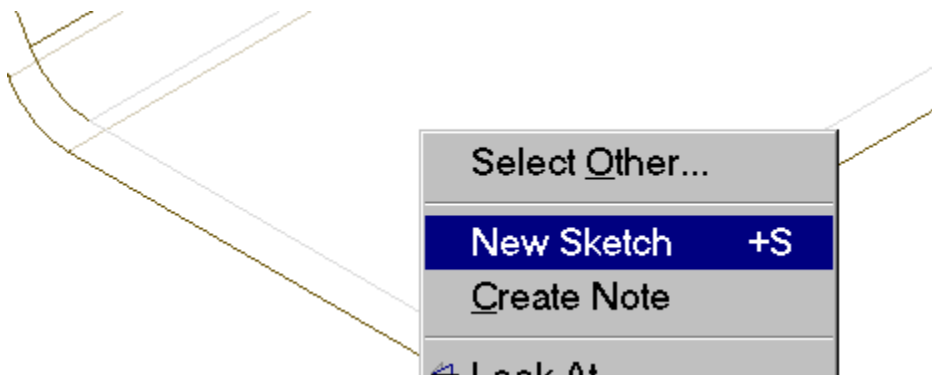
Activate Sketch Mode so we can locate the sketch properly.



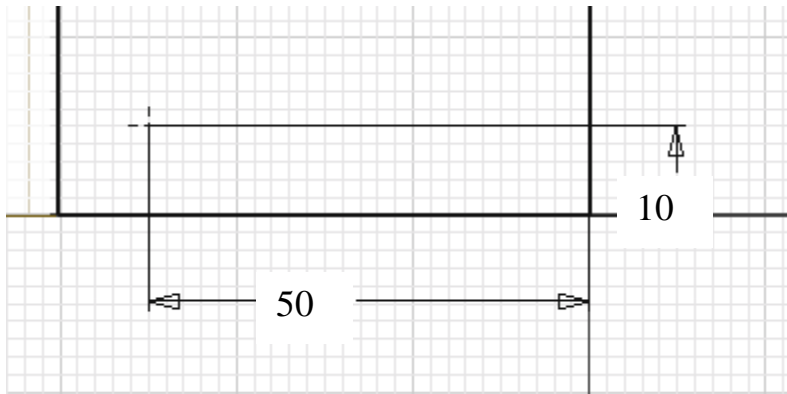
Locate the slot as shown.



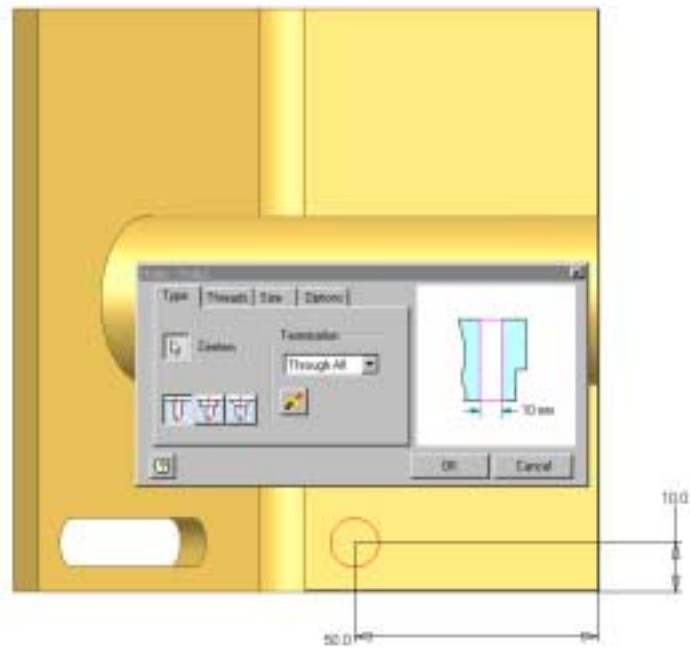
Extrude the sketch as a Cut through the part.



Select the bottom face for a New Sketch.



Place a Point, Hole Center as shown.



Place a 10 mm through hole as shown.

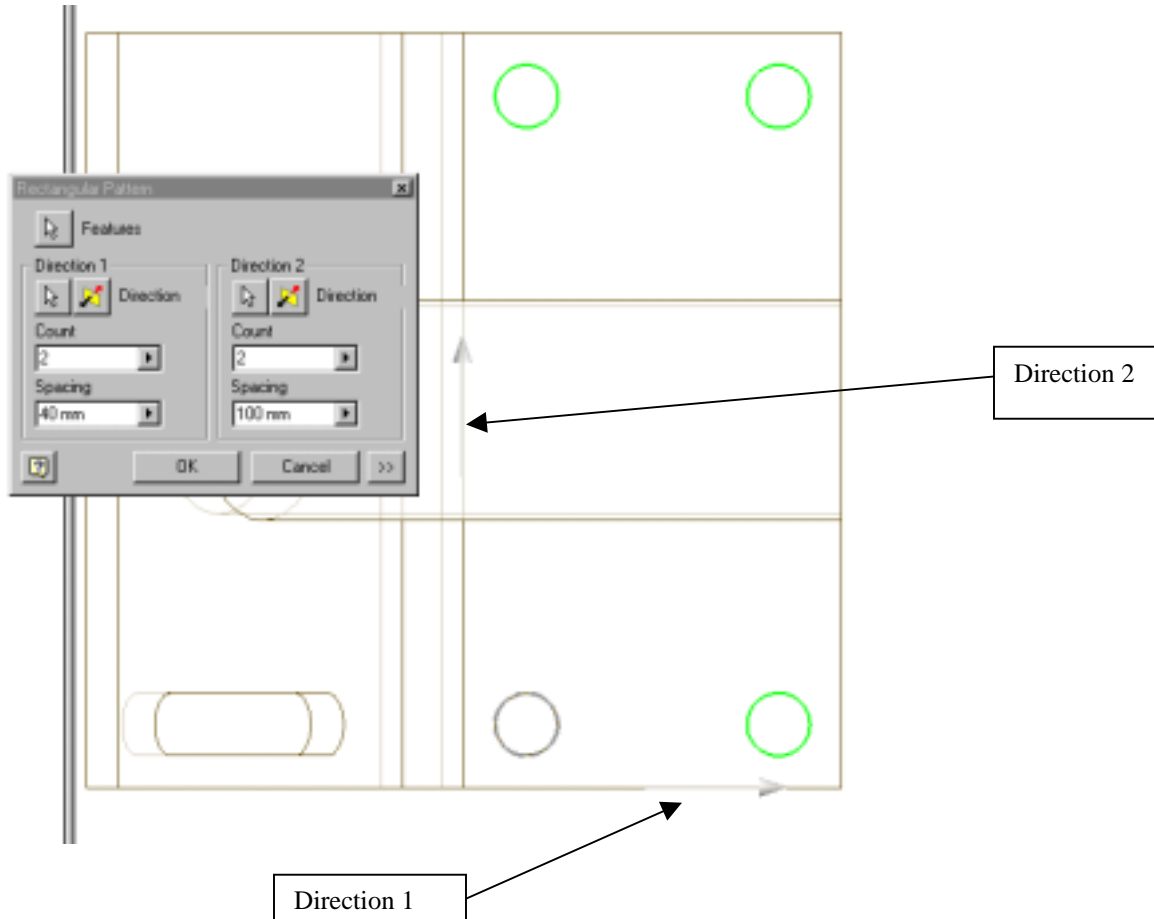


Create a Rectangular Pattern

Place a Rectangular Pattern of four holes as shown.

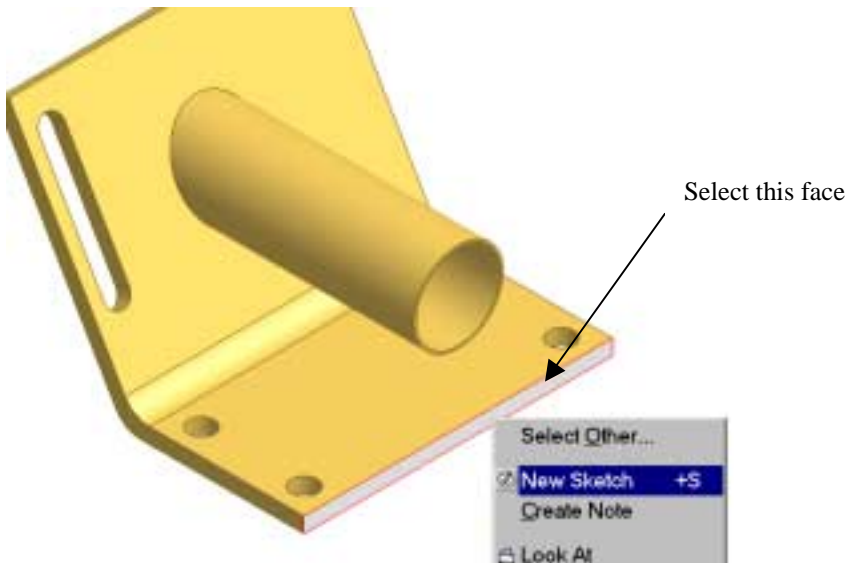
Set the Spacing to 40 mm for Direction 1 and the Count to 2.

Set the Spacing to 100 mm for Direction 2 and the Count to 2.

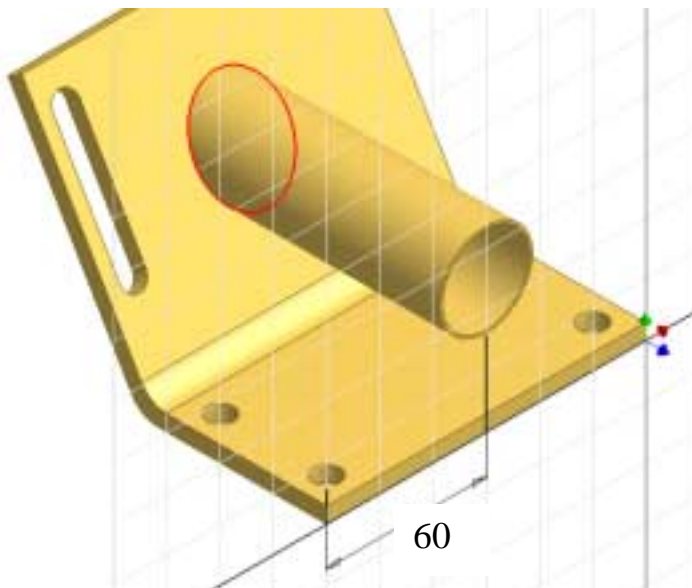




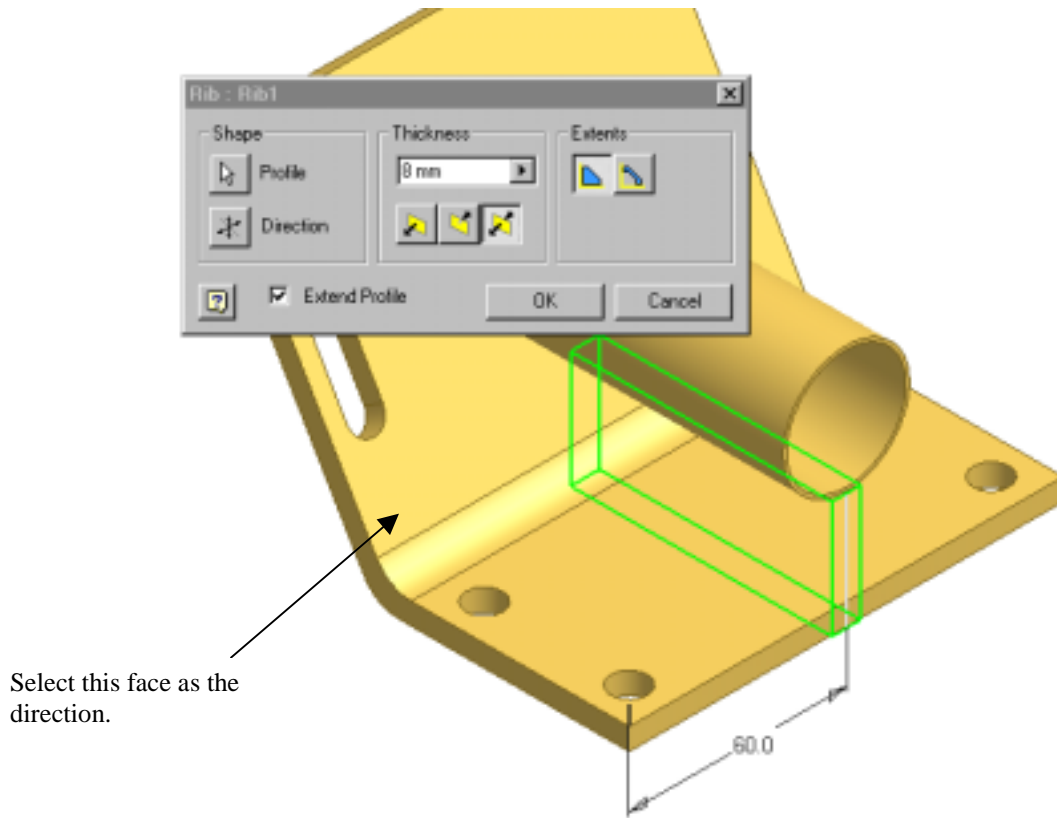
Creating a Rib



Select the front face as shown for a new sketch.



Draw a vertical line as shown.



Select this face as the direction.

Select the Rib tool.
Select the inclined face for the direction.
Set the thickness to 8 mm.

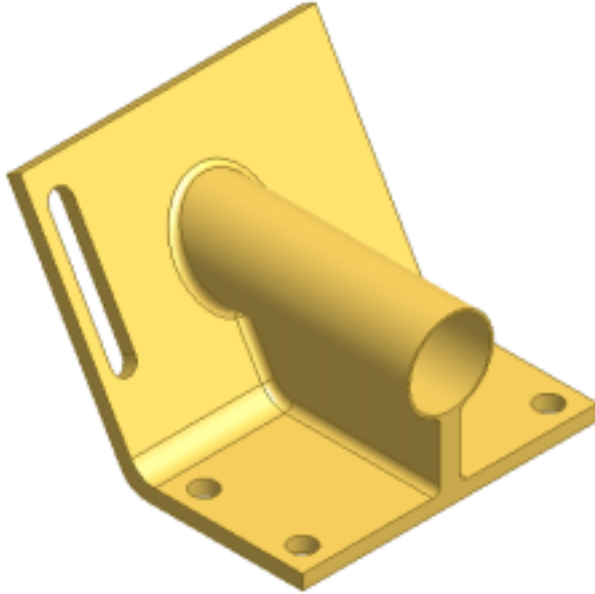
Set the extrusion direction as Mid-Plane.
Set the Extents as Next Face.
Press 'OK'.





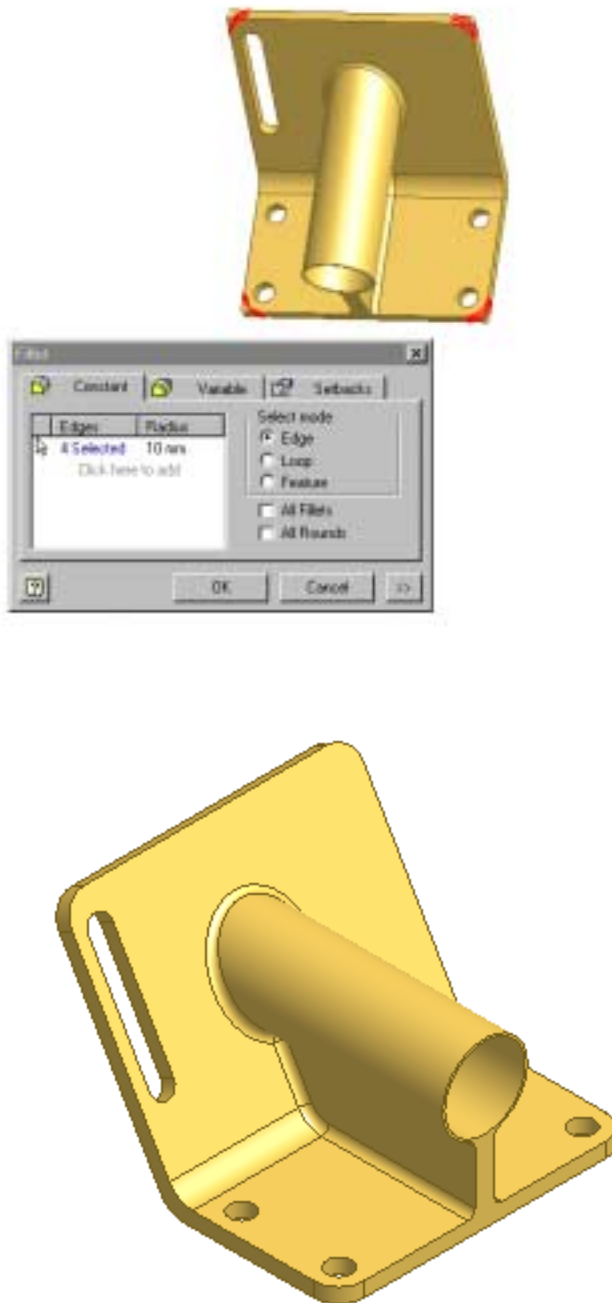
Adding Fillets to a Rib

Add R3 fillets along all the rib's edges.



HINT: If you get an error message when adding the fillets. Try adding a few fillets at a time instead of all at one instance.

Add an R10 fillet to the four corners of the bracket.



Save the completed bracket as 'lesson17.ipt'.

Review Questions



1. In this step, we selected the front edge as our sketch plane but drew our sketch above and away from the front edge. We were able to do this because:
 - A. When we select the front edge, we are actually selecting a planar surface
 - B. Inventor automatically created a work plane for our use
 - C. It's magic
 - D. None of the above

2. In the same step we selected the inclined plane as the 'To' Extents. Why did we do this and not specify a distance?
 - A. Specifying a distance would require us to measure
 - B. If we specified a distance the cylinder would not automatically create an angle when it intersects the inclined face.
 - C. Specifying a distance is more work
 - D. All of the above

3. When we inserted the Design Element, we were able to modify the parameters. The parameters for the Design Element were in inches even though we were creating a metric part. Why?
 - A. Because most machine shops in the United States use standard units.
 - B. Because when we set up Inventor we selected ANSI as our default units
 - C. Because the design element is an ANSI part
 - D. The units for the Design Element are determined when the Design Element is created and saved. The Design Element selected was created using inches.

Match the toolbar icons with their functions:



- A. RECTANGULAR PATTERN
- B. RIB
- C. INSERT DESIGN ELEMENT
- D. EXTRUDE
- E. HOLE

ANSWERS: 1) A; 2) D; 3) D; 4) D; 5) E; 6) B; 7) C; 8) A