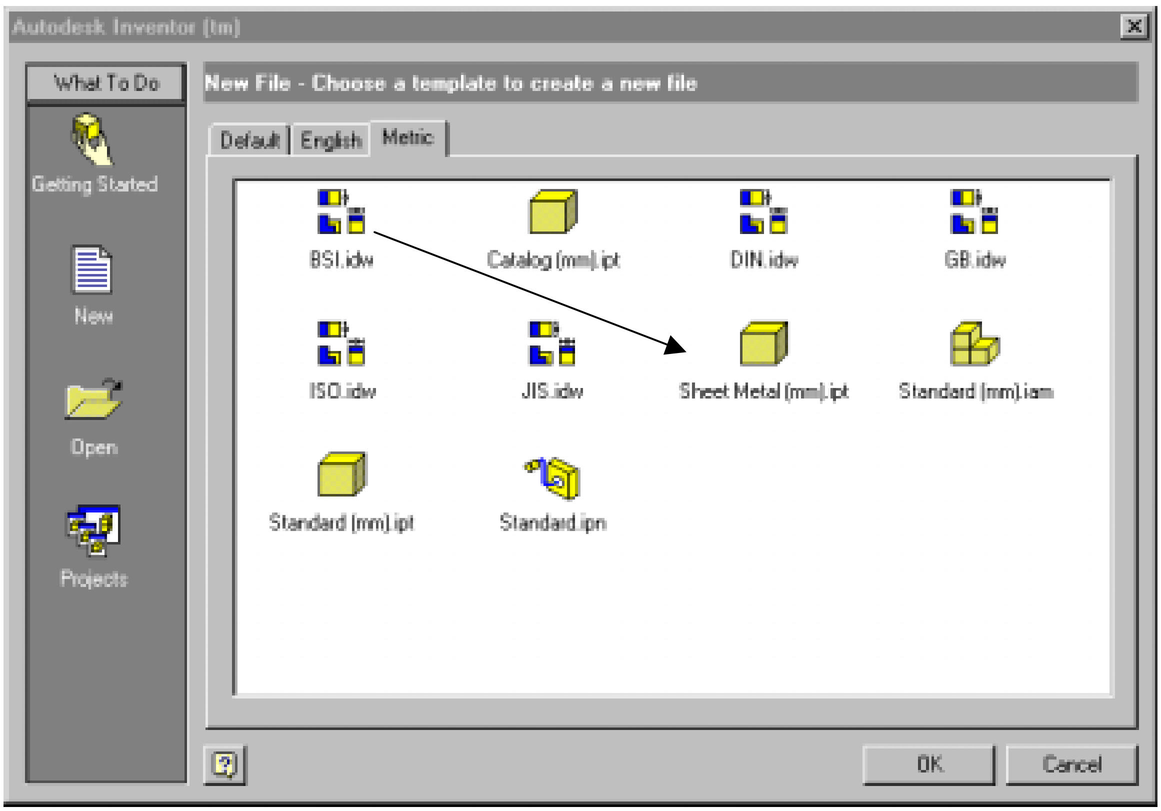
# Lesson 16
# Creating Sheet Metal Parts

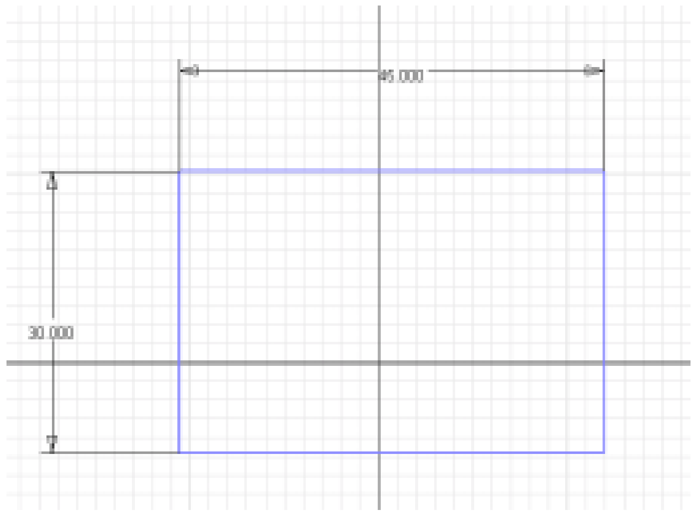
## Learning Objectives

In this lesson, the user will gain further mastery of the sheet metal environment and the following tools:

- Settings
- Face
- Hole
- Rectangular Pattern
- Corner Chamfer

We start by opening a sheet metal part using metric units. Select the metric tab of the dialog box.

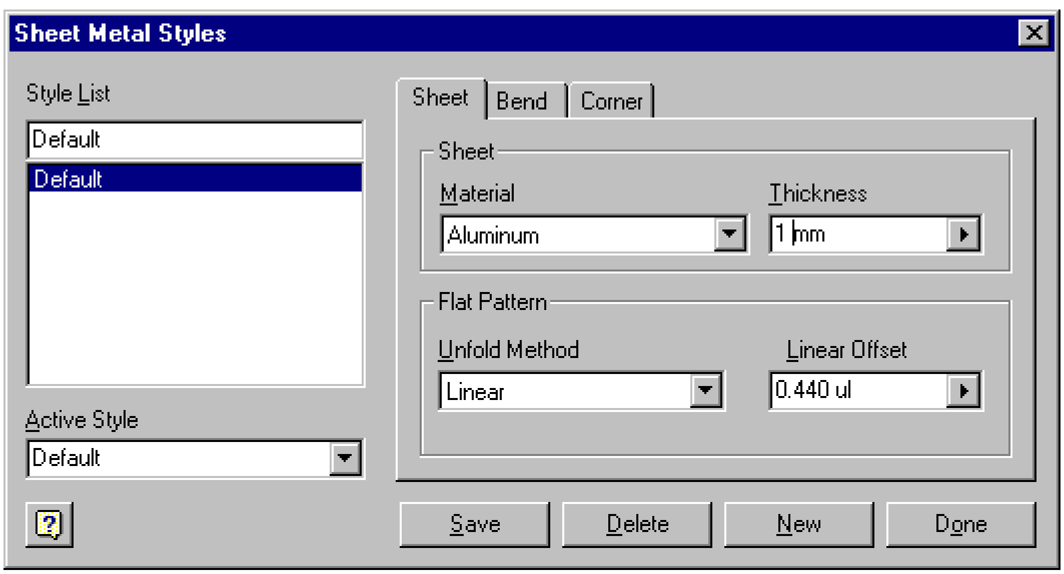




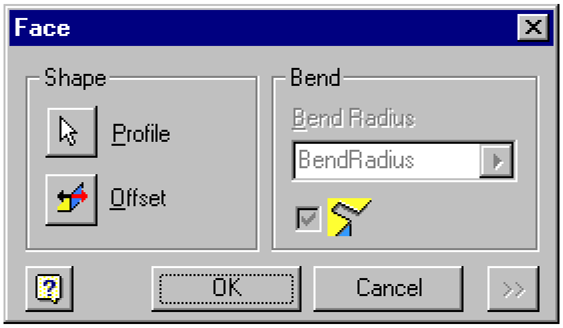

We create our first profile using the Rectangle tool and dimensioning 45 mm wide by 30 mm high.

## Sheet Metal Styles

Access Styles.




Set to Aluminum with a Thickness of 1 mm.
Press ‘Done’ and then ‘OK’.

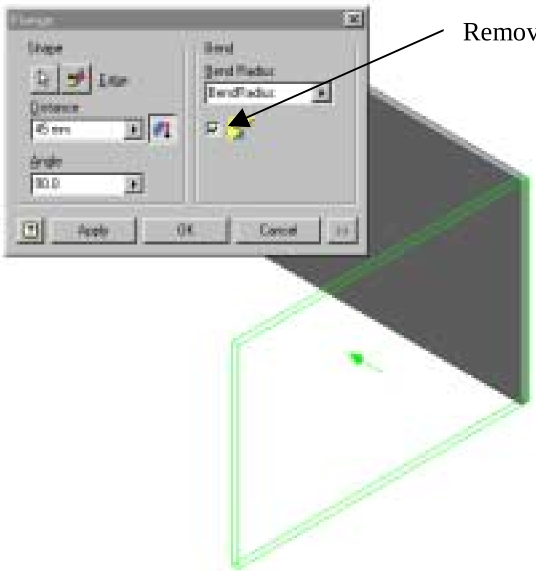




## Creating a Face

Switch to an isometric view.

Select the Face tool. The profile we created in automatically selected. Press 'OK'.

**TIP:** The face tool works differently than the Extrude tool. We only need to select a profile and a direction. Inventor automatically uses the thickness defined in Settings.



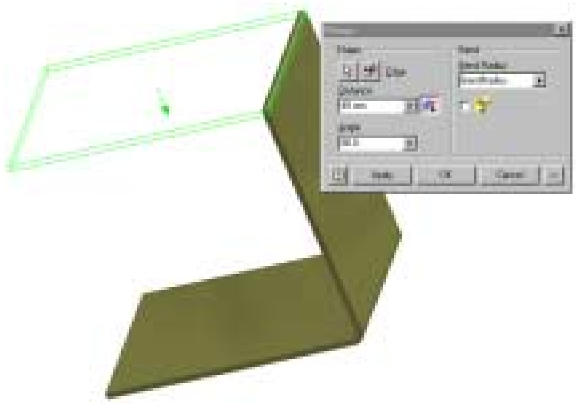


## Creating a Flange

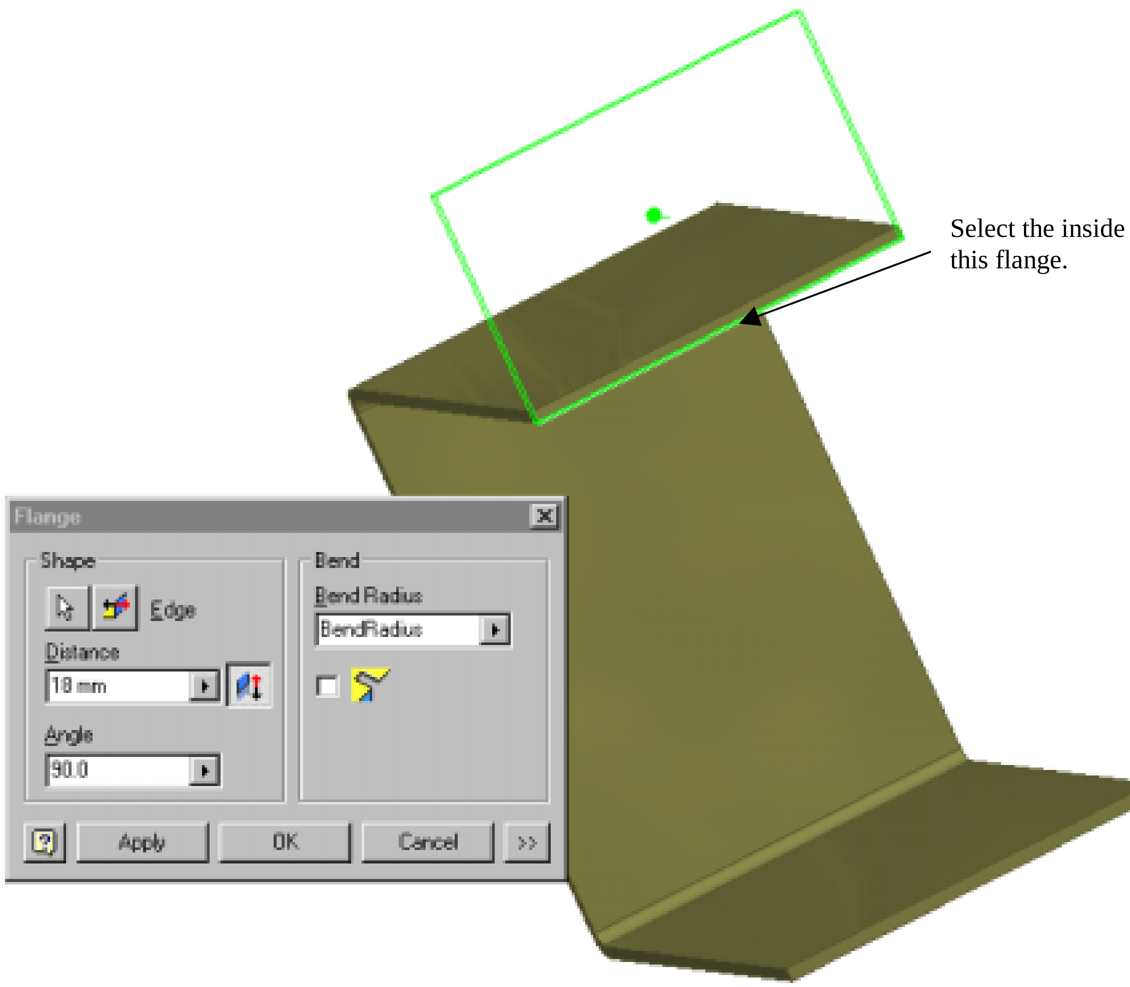
Select the Flange tool to add a Flange. Set the Distance to 45 mm. Disable the Bend Relief box. Press 'Apply'.

**TIP:** To add the flange, we only need to select the edge. We do not need to draw a profile.

Add a Flange to the opposite side using the same settings as shown.

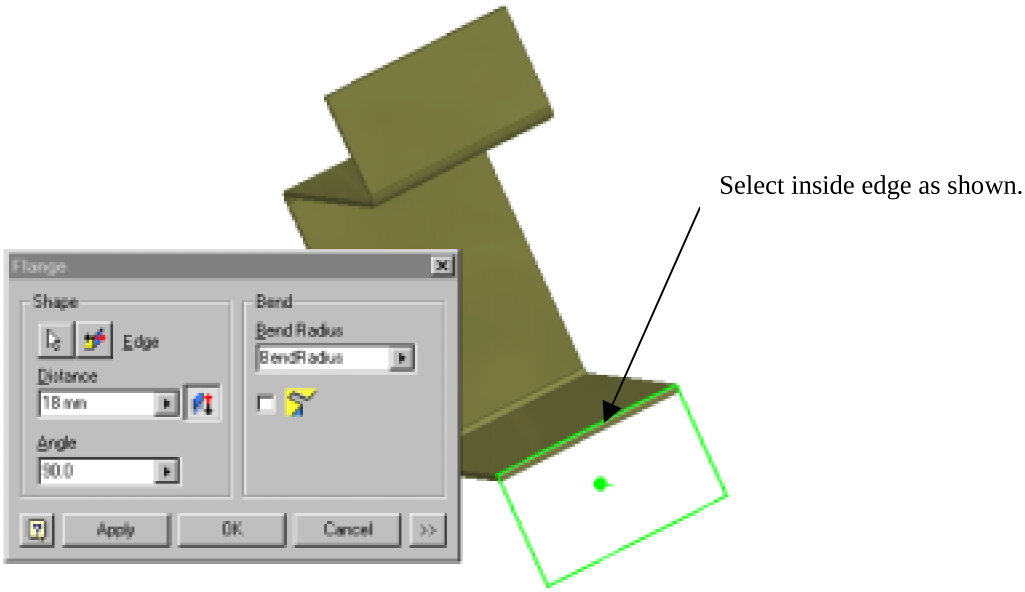


Press ‘Apply’ to add.



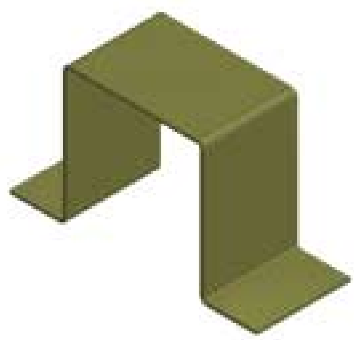

Select the inside edge to add this flange.

Add a flange as shown. Distance should be set to 18 mm. Bend Relief should NOT be checked. Press ‘Apply’.

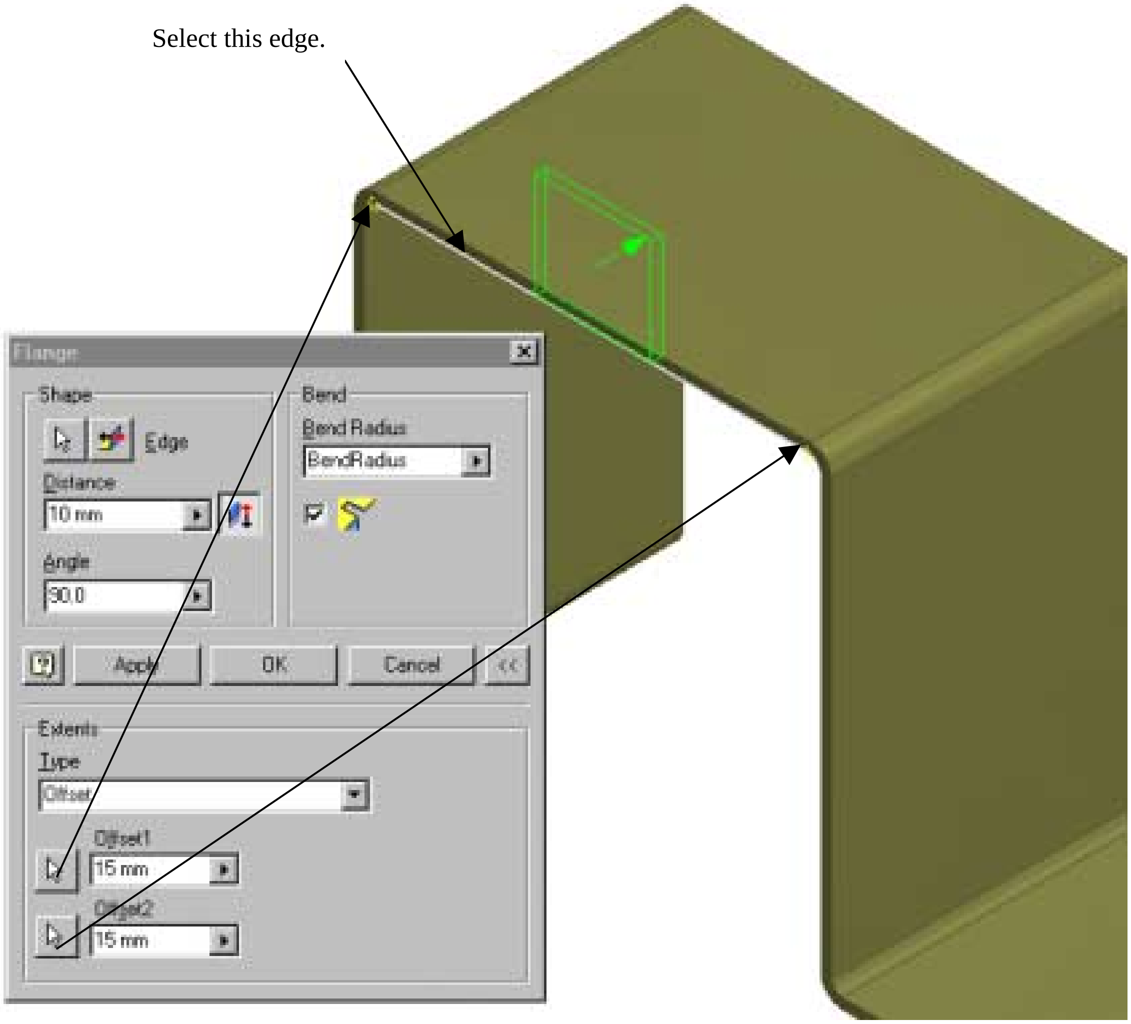



Add another flange as shown.  Distance should be 18 mm.
Bend Relief should NOT be checked.
Press ‘Apply’.

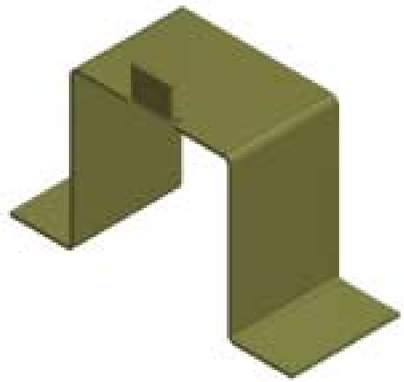
Close the Flange dialog box by pressing the X located in the upper right corner.



Our bracket so far.

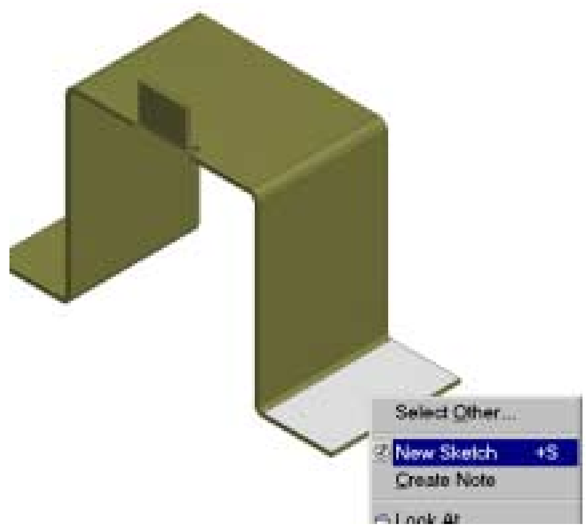



Select the Flange tool again.
Select the bottom edge as shown.
Set the distance to 10 mm.
Press the ‘More’ button.
Set Type to Offset.
Select the left point as shown.
Set the Offset to 15 mm.
Select the right point as shown.
Set the Offset to 15 mm.
Enable the Bend Radius as shown.
Press ‘Apply’.

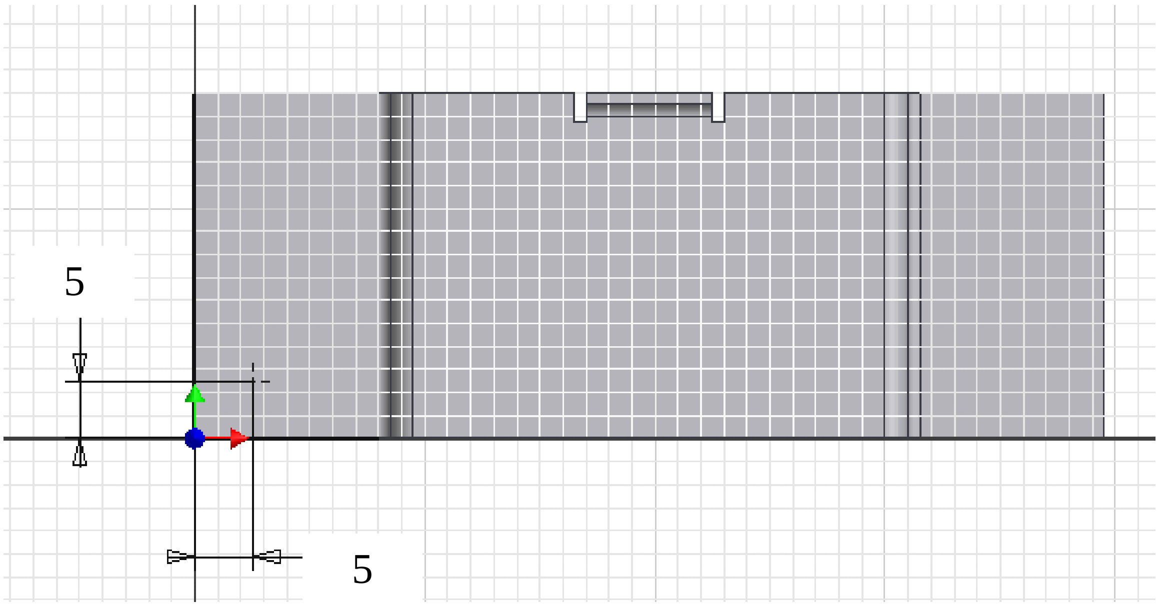


Our model so far.
Note that a bend relief was automatically added using the values in the Settings dialog box. This occurred because we enabled the Bend Relief option.

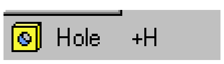
## Adding a Hole Feature

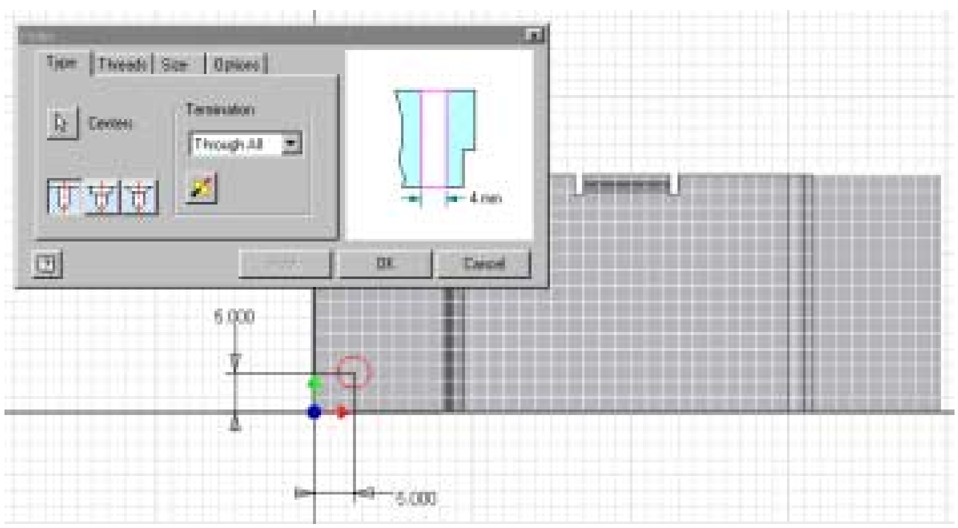



Select the foot as shown and 'New Sketch'.




Place a Point, Hole Center as shown.

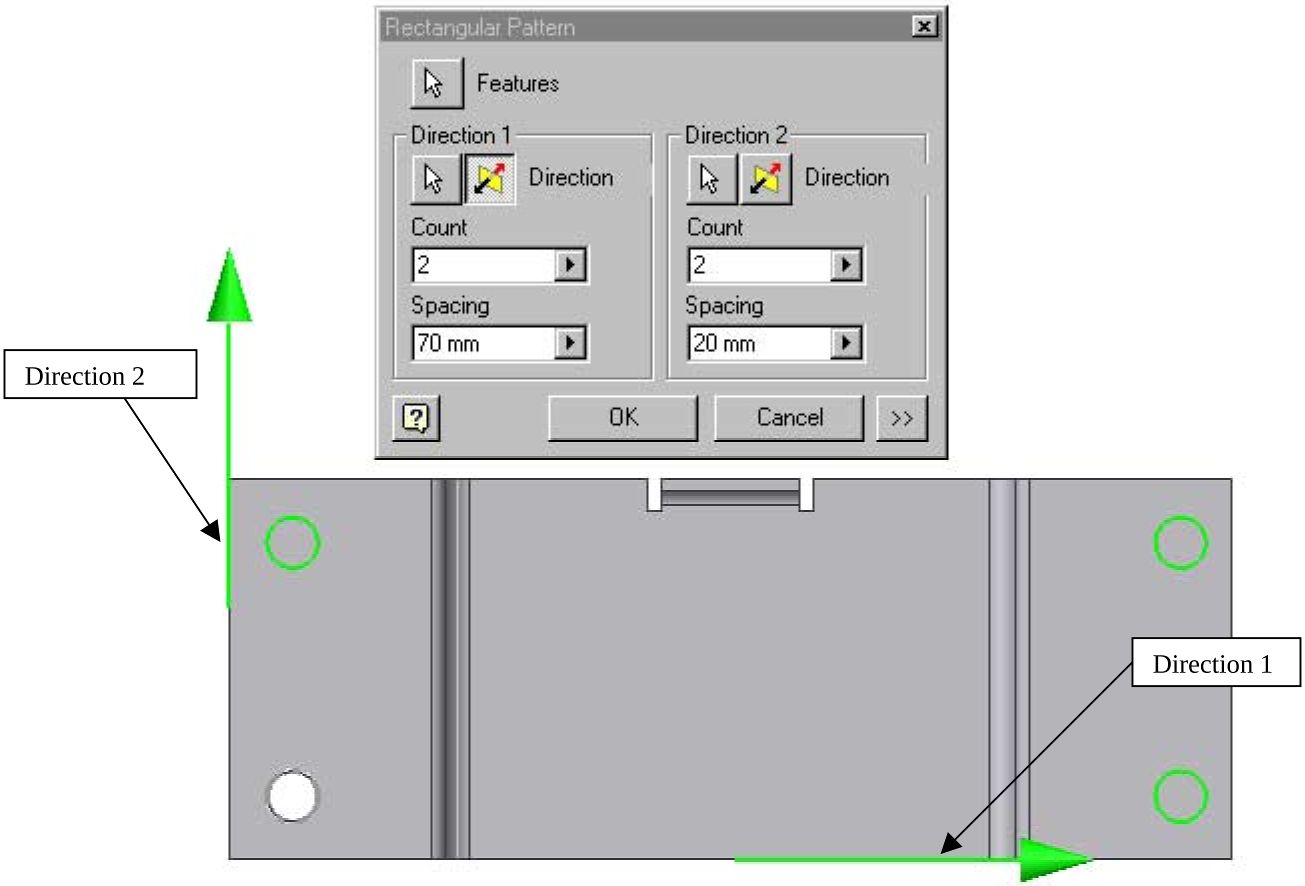


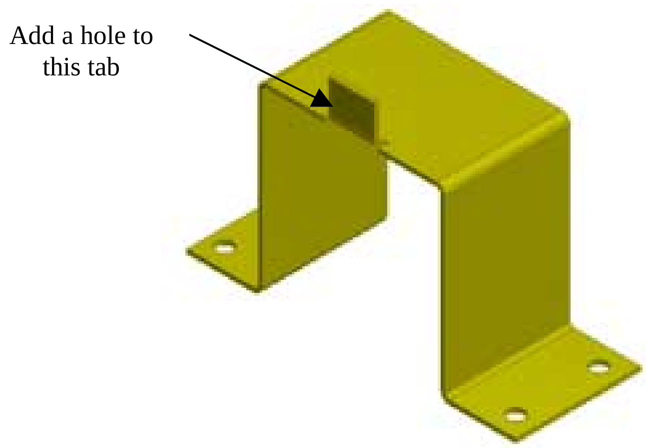

Create the hole as shown with Diameter of 4 mm and Termination of Through All.

## Adding a Pattern

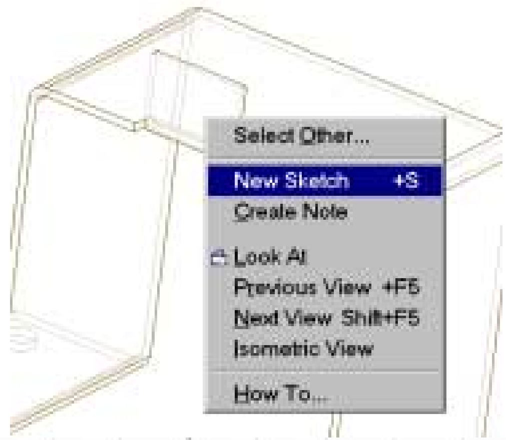

Select the Rectangular Pattern tool.
Select the hole from the browser to be patterned.

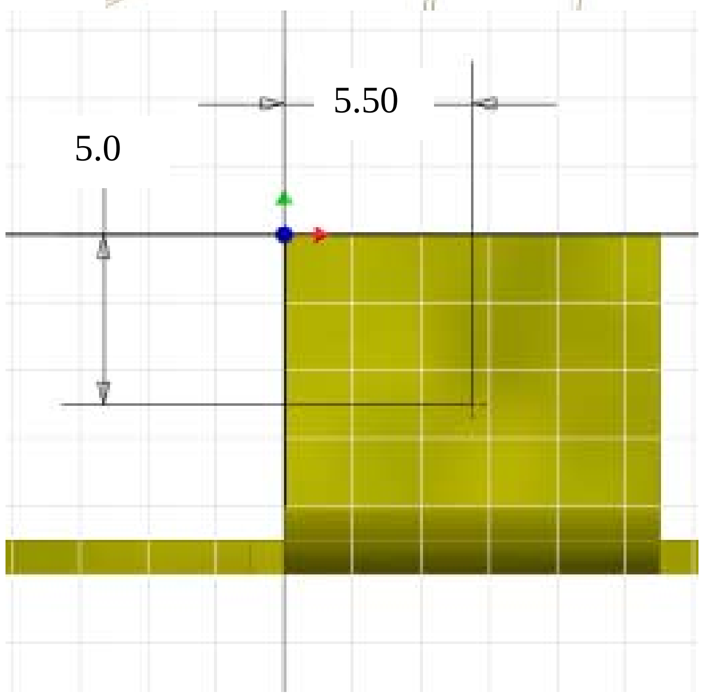



Set the pattern as shown. Under Direction 1, set Spacing of 70 mm. Under Direction 2, set spacing of 20 mm.

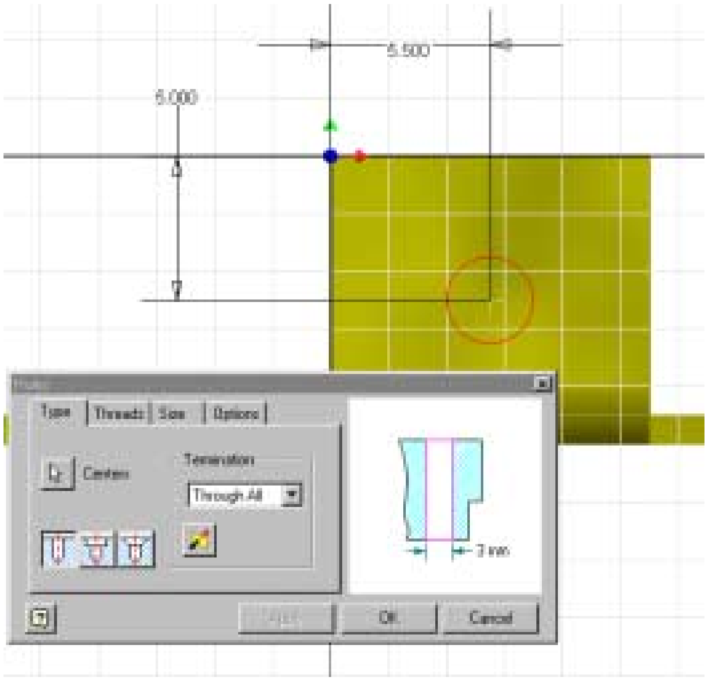



Add a hole to the small tab as shown.
Select the front plane of the tab.
Right click and select ‘New Sketch’.
Place a point as shown.



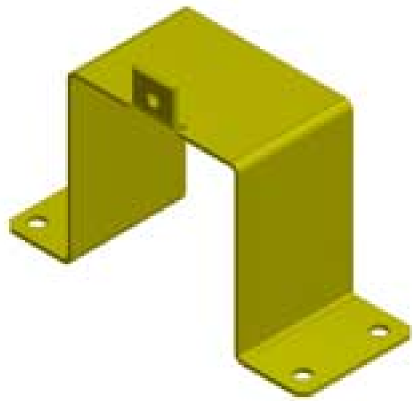

Set the hole to 3 mm in Diameter and Termination of Through All.




## Adding Corner Chamfers

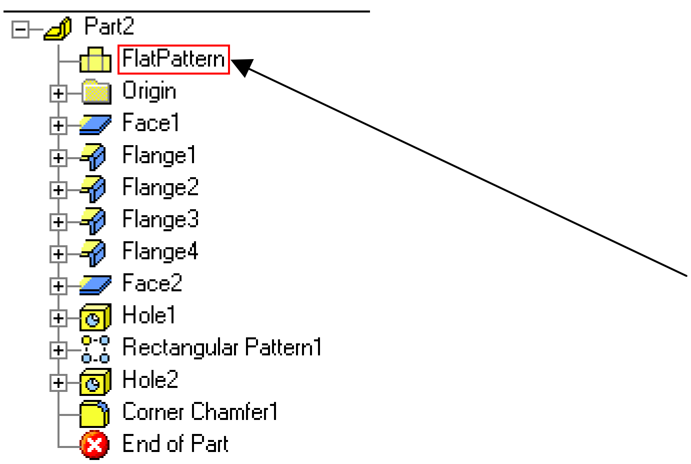

Add Corner Chamfers to the edges of the feet as shown.


Use the Equal Distance option (top left button). Select all four corners as shown. (Use Rotate to aid in selection.) Set the Distance to 2 mm.
Press OK.

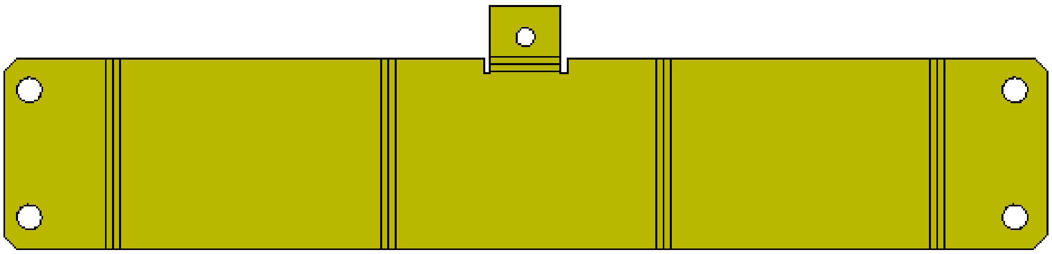


## Creating a Flat Pattern


Select the Flat Pattern tool.

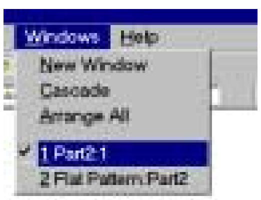



A Flat Pattern icon is automatically added to the Browser.

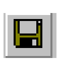


Our model is viewed as a Flat Pattern.

**TIP:** The Flat Pattern will automatically update with any changes we make to the model.




The Flat Pattern is actually viewed in a separate window that was instantly created when we selected the Flat Pattern tool. To switch back to our 3D model, go to Windows under the Menu and select our Part window.



Save our file as ‘sheet-metal.ipt’.

## Review Questions

1. The first feature in a sheet metal part is created using:

    A. FACE
    B. FLANGE
    C. CORNER SEAM
    D. EXTRUDE

2. To create a face 90 degrees in relation to an existing face that is the same width, use this tool:

    A. FACE
    B. EXTRUDE
    C. FLANGE
    D. BEND

3. To select the type of material used in a sheet metal part, use:

    A. Options
    B. Styles
    C. Properties
    D. Physical

4. To automatically apply a bend relief when adding a flange or a face:

    A. Enable Bend Relief in the Settings Dialog box
    B. Enable Bend Relief in the Flange/Face dialog box
    C. Use the bend relief tool
    D. Enable Bend Relief in the Options dialog box.

5. When we use the Flat Pattern tool, this happens:

    A. A flat pattern view is created to be used as a drawing view
    B. The 3D model is transformed into a flat piece of sheet metal
    C. The 3D model is patterned in a single row
    D. A text file is created to be used by a CNC operator

6. When we first open up a sheet metal part file, this toolbar is active in the panel bar:

    A. Sheet Metal
    B. Solids
    C. Features
    D. Sketch

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