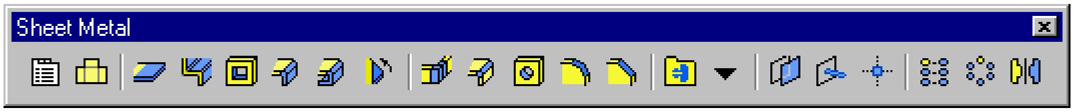
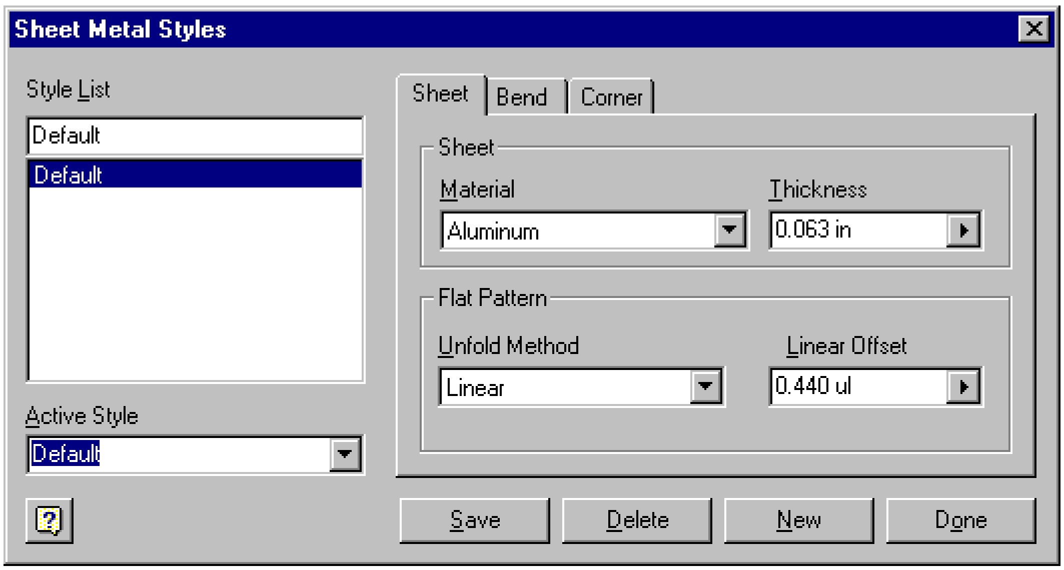
Lesson 9
Sheet Metal Tools



Styles

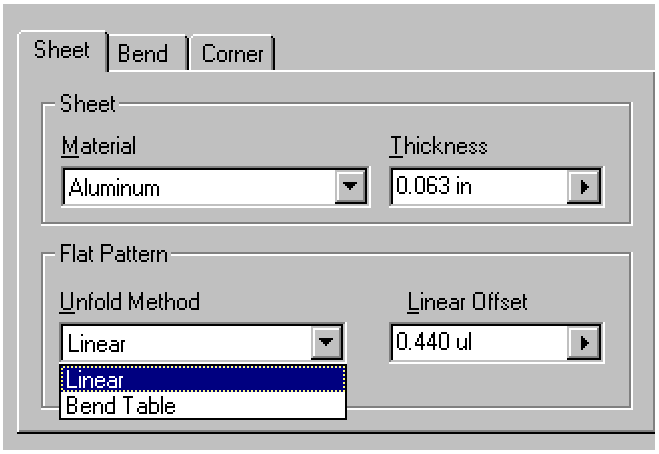
Sheet metal styles specify default parameters for sheet metal parts. Only the bend radius can be modified when creating a sheet metal part; all other settings must be modified on the Sheet Metal Styles dialog box (to change defaults for all sheet metal parts) or in the Parameters dialog box (to change settings for an individual sheet metal part).



Pressing the Styles tool brings up the dialog box shown.

Under the Sheet Tab, we have several options;

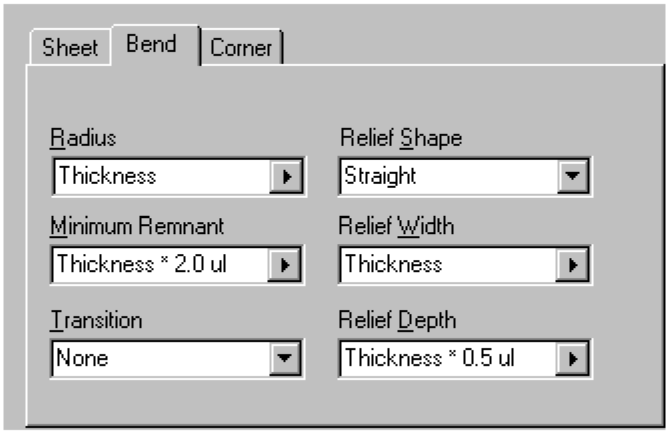
Material	Click the arrow and select a material from the list. This parameter can also be entered on the Physical tab of the Properties dialog box. Select File>Properties>Physical.
Units	Click the arrow and select a system of units from the list. This parameter can also be entered on the Units tab of the Properties dialog box. Select Files>Properties>Units.
Thickness	Sets the sheet metal thickness in decimal units.



On the right side of the Sheet Tab, we set the Unfold Method for the Flat Pattern.

Unfolding Method	Sets the Linear or Bend Table unfolding method. For Linear, enter a value for the linear offset. For Bend Table, the Open dialog box displays so you can browse to the location of a bend table.
Linear Offset	When Linear is selected as the unfolding method, this parameter determines where the bend allowance is calculated. The allowable range is from 0 to 1. The bend allowance is calculated using the following equation: 2*PI*(Bend Radius + Linear Offset*Thickness)*(Bend Angle/360)

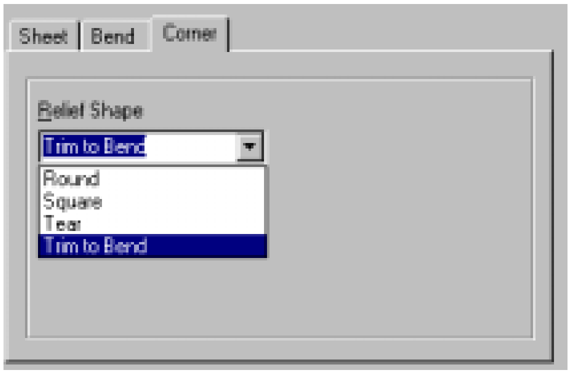
TIP: Users can set up sheet metal styles (similar to dimension styles in AutoCAD) with their favorite sheet metal settings. Create styles by pressing the ‘New’ button in the Styles dialog box.

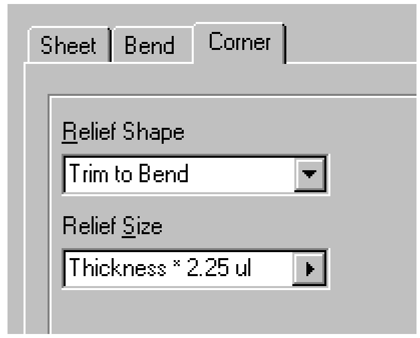


Under the Bend Tab, we have several options:

Bend Radius	Sets value for the default bend radius. This parameter can be overridden on an individual feature.
Bend Relief	Inserts a bend relief if the bend does not extend the full width of a sheet metal face. Select Default Straight (square corners) or Default Round (full radius corners).
Minimum Remnant	Sets the amount of material between the bend relief and the edge of the sheet metal part. If the remnant is less than this value, the width of the bend relief is increased to consume the remnant.
Bend Relief Width	Sets the distance between the edge of the bend and the bend relief. This value is usually determined by the selection of available punches.
Bend Relief Depth	Sets the distance the bend relief extends past the bend zone.

TIP: By using the variable ***Thickness*** multiplied or divided by a value, we can eliminate the amount of editing we need to do. Simply change the value for the Thickness under the Sheet tab and all the values on the Bend tab will automatically update.





The third tab on the Sheet Metal Settings dialog is Corner.

Relief Shape	Inserts a corner relief when a corner seam is applied and three faces come together in the corner. Select None for no corner relief or Default Round for a circular corner relief.
Relief Size	Sets the corner relief size. Specify a value that extends the corner relief past the bend lines on the largest bend.

TIP: To see the correct corner relief, create a flat pattern.

Flat Pattern

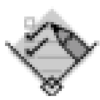
Creates an unfolded representation of a sheet metal part. The flat pattern appears in a separate graphics window from the sheet metal part.

The Flat pattern tool calculates the unfolded state of the 3D model and displays it in a separate graphics window. You can arrange the part window and the flat pattern window so you can view both at the same time. The flat pattern updates automatically when you edit the 3D model, but you cannot edit the part in the flat pattern window.

If you create a model that cannot be unfolded (if the features overlap in the flat pattern, for example), the flat pattern does not update. The flat pattern symbol in the browser is marked.

Features that require material deformation, such as louvers or dimples, cannot be flattened. The flat pattern tool shows the outline of the feature on the flat pattern.

The Drawing Manager uses the flat pattern for the flat pattern view. The flat pattern must be created in the part before you can place a flat pattern view in the drawing



TIP: A flat pattern cannot be created if the part has no bends.
If you delete the flat pattern, the drawing will also lose the flat pattern view.

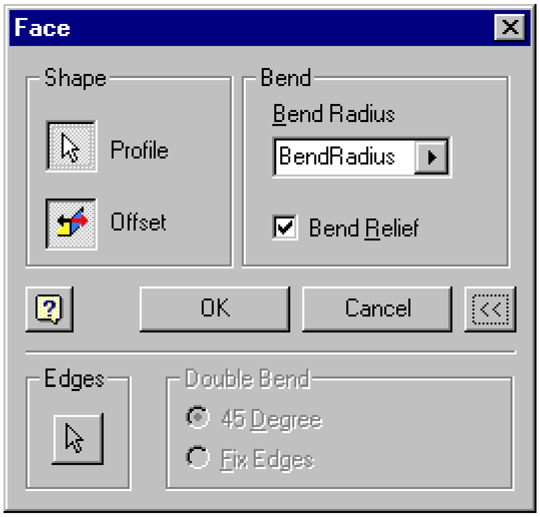
An incorrect flat pattern may be generated if the incorrect face is used as the base face. If this happens, right-click the flat pattern icon in the browser and click Delete. Then, to preselect the face you want to be the base face, click it in the graphics window, and then click the Flat Pattern tool.

Flat patterns are created with MetalBender Solver software from data M Software + Engineering. Their website is http://www.data-m.com/.

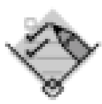
Face

The Face command is similar to the Extrude command.

Creates a sheet metal face by adding depth to a sketched profile. The Feature shape is controlled by the sketch shape and any bends or seams between the new sheet metal face and existing sheet metal faces.



The Shape tool selects a profile to extrude by the sheet metal thickness. If there are multiple profiles, click on one or more profiles in the graphics window. The extrusion of the selected profiles is previewed in the graphics window. Click the Offset button to change the direction of the extrude.

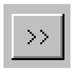


TIP: To select multiple profiles to extrude, position the cursor over the profile and click to select. To unselect, press Ctrl and click profile.

On the right side of the Face Dialog box we specify the bend radius and whether a bend relief is added. If the new sheet metal face is coincident with one existing sheet metal face, a bend is created automatically.

We have two options: Bend Radius and Bend Relief.

The default bend radius is displayed. Enter another value if desired. Select Bend Relief automatically generate bend reliefs. Clear Bend Relief to create a bend without a relief.

Click the More button  to display the Edges button. Select the intersecting model edge on the existing sheet metal face.

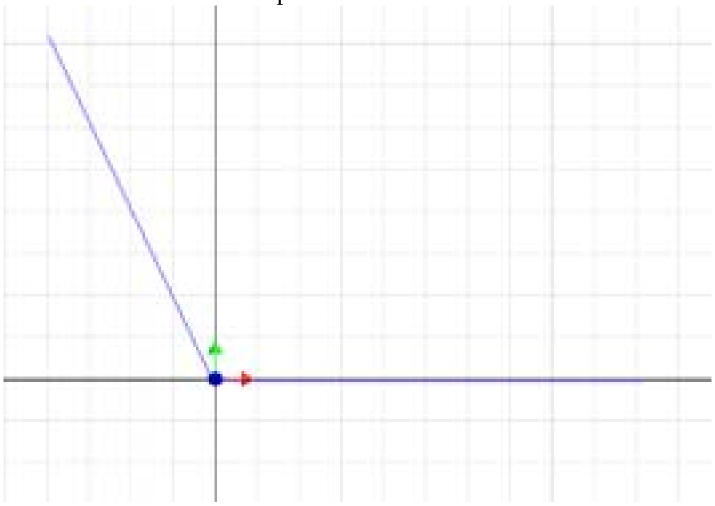
Under the More area we can select a single bend or double bend.

Single bend	For a new sheet metal face, you can create a bend to an existing sheet metal face. Select the sheet metal face for the bend if one of the following applies: • The profile for the new sheet metal face is coincident with multiple existing sheet metal faces. • The profile for the new sheet metal face is not coincident with any sheet metal faces. Autodesk Inventor trims or extends the sheet metal faces as required to create the bends.
Double bend	If sheet metal faces are parallel, but not coplanar, you can create a double bend between the faces. The bends are trimmed so they are tangent or a new sheet metal face is constructed to connect the bends. 45 Degree Sheet metal faces are trimmed or extended as necessary and 45 degree bends are inserted. Fix Edges Equal bends are added to the existing sheet metal edges.

Contour Flange

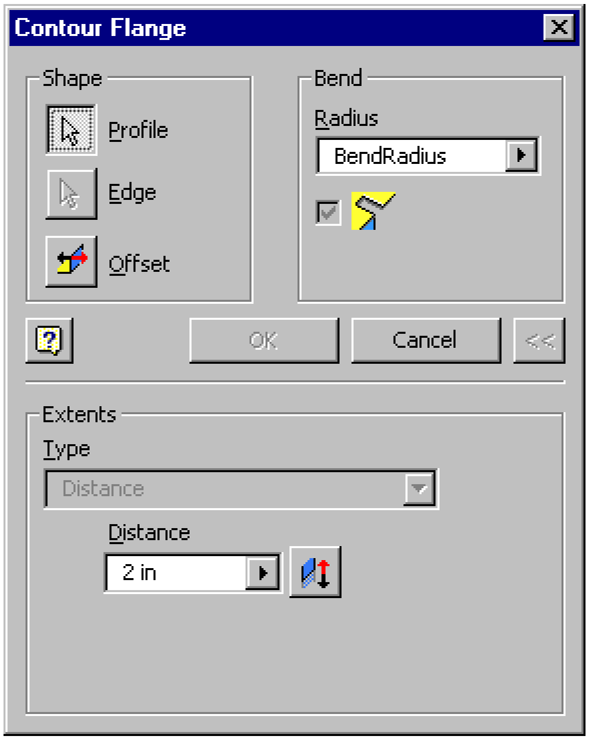
A contour flange is created using an open profile.

Create a new sheet metal part file.

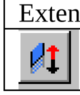


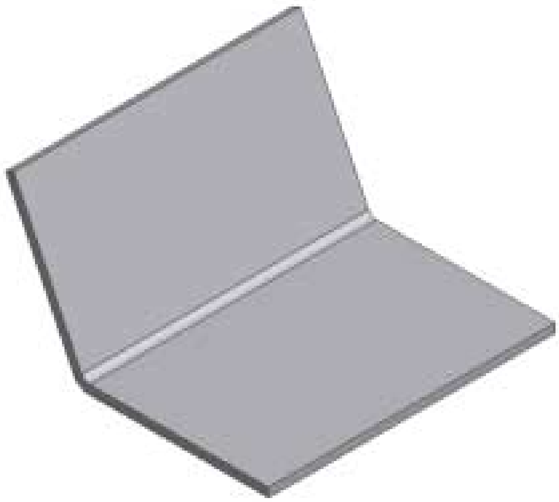
Draw two lines as shown: one horizontal and one at an angle.

Select the Contour Flange tool.



The Contour Flange dialog box comes up.

Shape	
Profile	Select an open profile to be used
Edge	Select the existing edge of a part as the contour
Offset	Select the direction for the offset
Bend	Defaults to Bend Radius defined by Styles or allows the user to enter a value
Extents	If Contour Flange is base feature requires a finite distance.
	Determines the direction of the extrusion

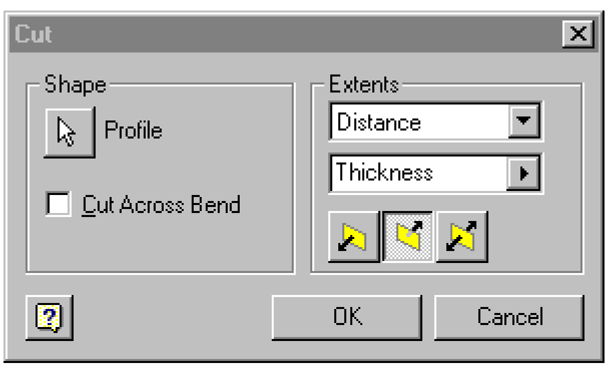


Our contour flange.

Cut

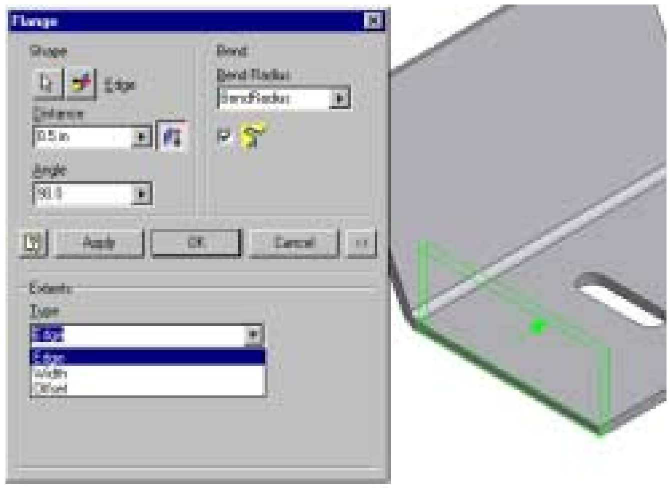
A cut removes material from a sheet metal face. You sketch a profile on a sheet metal face and then cut through one or more faces. You can use Design Elements to create a library of punch shapes. Cut features can be used with the Design Element, Mirror, Rectangular Array, and Circular Array tools.

You can specify a distance for a cut or it can terminate on a face or workplane. In an assembly, the terminating face or workplane can be on another part.



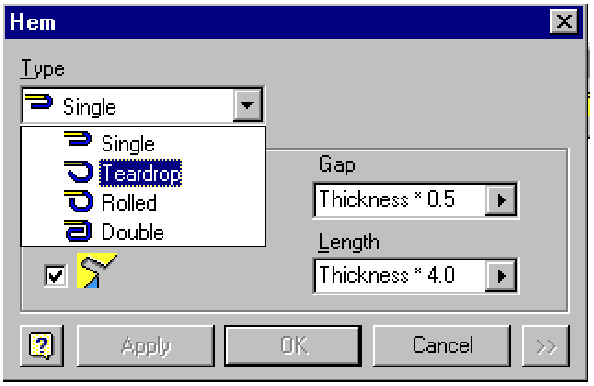
Flange

Creates a flange by adding a sheet metal face and a bend to an existing face. The flange is created the full width of the existing face.

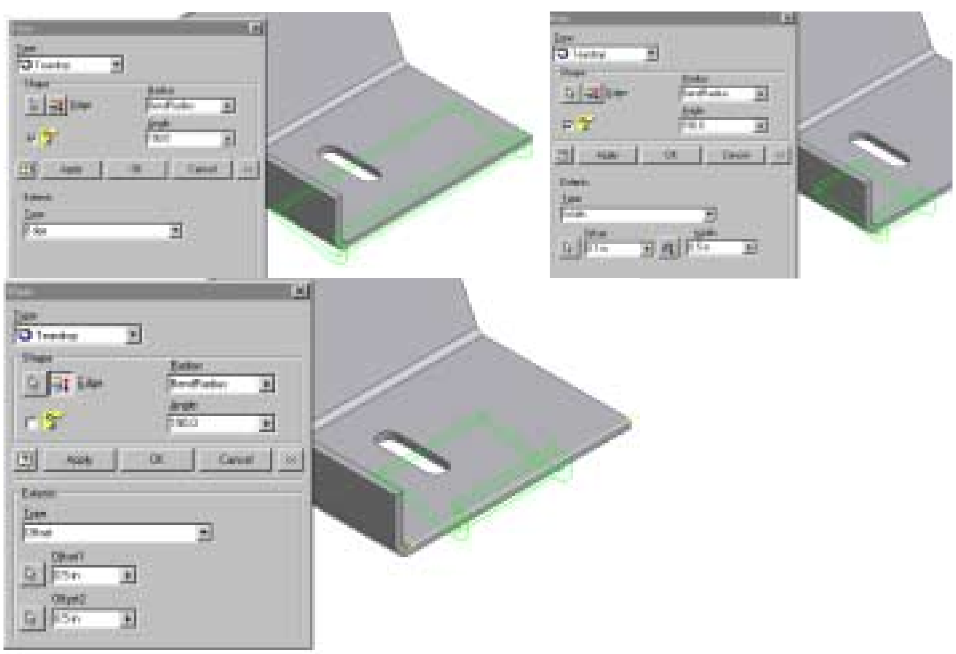


Distance	Enter the distance for the depth of the flange. The distance is measured from the model edge you select for the flange.
Angle	Enter the angle for the flange. The angle must be less than 150 degrees.
Flip Offset	Creates the flange on the inside or the outside of the face. Click to change the position of the flange as needed.
Flip Direction	Changes the side of the face the flange is created on. Click to change direction as needed.
Edge	Selects a model edge for the flange.
Bend radius	Displays the default Bend Radius. Enter another value if desired.
Bend relief	Specifies Bend Relief to automatically generate bend reliefs. Clear the Bend Relief check box to create a bend without a relief.
Apply button	Places specified flange and allows you to continue defining and adding flanges.
OK button	Places specified flange and closes the dialog box.

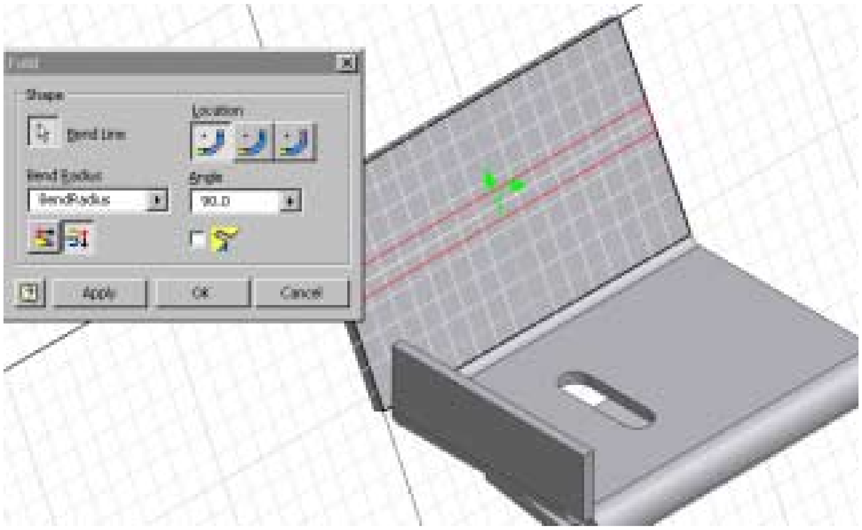
Hem



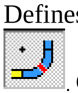
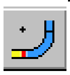
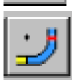
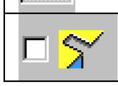
Type	
Hem Style	User can select Single, Teardrop, Rolled, or Double
Shape	Selects profile or edge to be used
Edge	Determines direction of Hem
Radius	Displays the default Bend Radius. Enter another value if desired.
Angle	Sets the angle for the bend; must be greater than 180 degrees
Extents	Edge – Uses the entire selected edge Width – Allows user to set an offset from a point and a width value for the hem Offset – Allows user to offset the hem from two points

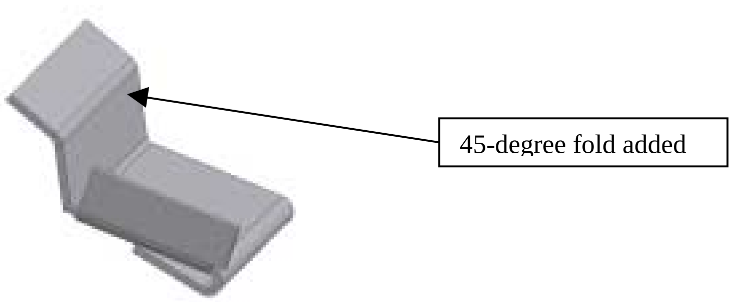


Fold



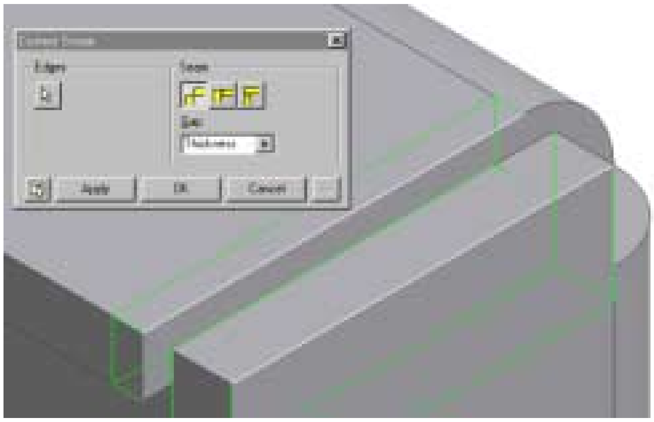
To add a fold to a sheet metal part, select a plane and create a 'New Sketch'. Then draw a line where the fold should be placed. Add dimensions as needed. Select the Fold tool.

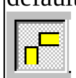
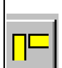
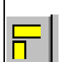
Bend line	Sketched line used to locate the fold
Location	Defines how the line selected is to be used:  Centerline of Bend  Start of Bend  End of Bend
BendRadius	Displays the default Bend Radius. Enter another value if desired.
Angle	User defines angle of fold
	Flip Side
	Flip Direction
	Enable/Disable Bend Relief – uses Bend Relief value defined in Style



Corner Seam

Use the Corner Seam tool on the Sheet Metal toolbar to add a seam to two sheet metal faces.



Edges	Selects a model edge on each face.
Seam	Specifies the overlap (no overlap or one of the faces overlap) condition of the edges. The overlap condition impacts manufacturing. If one face overlaps the other, it must be formed first. Enter a value for the Gap if it is different than the default specified in the Sheet Metal Settings dialog box.  No overlap  Overlap  Reverse Overlap
More button	Click the More button to specify adjacent edges for an Aligned Seam. If the faces are coplanar, you can specify these overlap options: Miter joint requires two faces joined by a corner seam before the miter can be applied. Butt joint specifies one face overlapping the other face.
Apply button	Places specified corner seam and allows you to continue defining and adding corner seams.
OK button	Places specified corner seam and closes the dialog box.

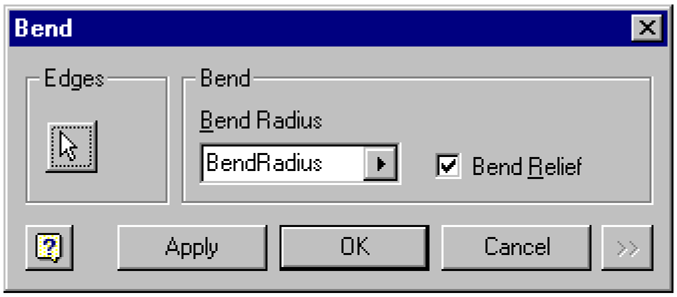
1. Click the Corner Seam tool.
2. Select a model edge on each sheet metal face.
3. Under Seam, choose the overlap method.
4. Enter a value for the seam gap. The default is the thickness of the sheet metal.

5. To align the seam gap for miter seams, click the More button and select edges.

Click Apply to continue to add seams or click OK to close the dialog box.

Bend

Adds a bend between two sheet metal faces.

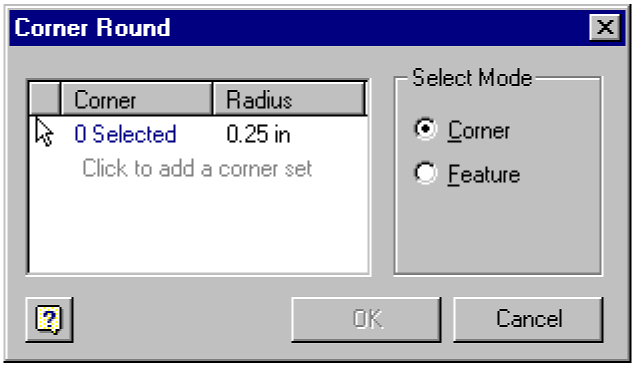


Edges	Selects a model edge on each face. The bend is previewed. The sheet metal faces are trimmed or extended as necessary to create the bend.
Bend	Specifies the radius for the bend. Clear the check mark in the Bend Relief box if you do not want a relief inserted.
More Button	Click the More button to display the Double Bend options. If the sheet metal faces are parallel, but not coplanar, you can create a double bend between the faces. The bends are trimmed so they are tangent or a new sheet metal face is constructed to connect the bends. 45 Degree Connection The sheet metal faces are trimmed or extended as necessary and 45 degree bends are inserted. Fix Edges Equal bends are added to the existing sheet metal edges.
Apply button	Places specified bend and allows you to continue defining and adding corner bends.
OK button	Places specified bend and closes the dialog box.

Hole

Refer to Lesson 5 regarding the Hole tool.

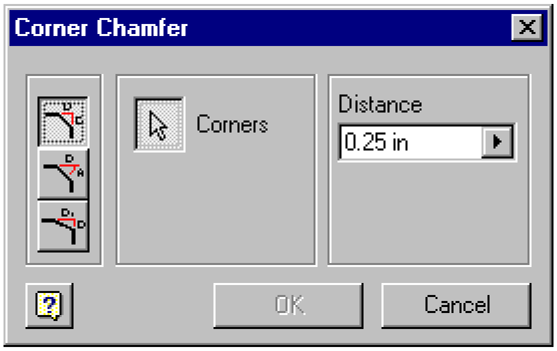
Corner Round



Adds fillets or rounds to one or more corners of a sheet metal part. You can create fillets or rounds of different sizes in a single operation. All fillets and rounds created in a single operation are one feature.

Corner	Defines corners for fillets or rounds. To add corners, select the set from the Corner box, and then select the corners by clicking the model edges in the graphics window. (To remove corners, press Ctrl as you click.) To add another corner set, click the prompt in the last row of the Corner box. Use Select Mode to simplify the selection of edges.
Cut	Specifies the radius for the selected set of corners. To change the radius, click the radius value, then enter the new radius.
Select Mode	Changes the selection method for adding or removing corners from a corner set. Click to select the mode from the list. Corner selects or removes single corners. Feature selects or removes all corners of a feature that do not result from intersections between the feature and other sheet metal faces.

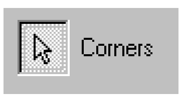
Corner Chamfer



Adds a chamfer on one or more sheet metal corners. You specify the corner appearance and select corners individually. All chamfers created in a single operation are one feature.

The three left buttons determine the method the chamfer is created. The top button is Equal Distance. The middle button is Distance and Angle. The bottom button is Two Distances.

Distance	Creates a chamfer with the same offset distance from the corner on both sheet metal edges. Select a single corner or multiple corners.
Distance and Angle	Creates a chamfer defined by an offset from a sheet metal edge and an angle from that sheet metal edge to the offset. One or both corners of a selected sheet metal edge may be chamfered in one operation.
Two Distances	Creates a chamfer on a single corner with a specified distance for each sheet metal edge. Edges may be chained together.



The Corners button located in the center of the dialog box selects one or more corners or edges.

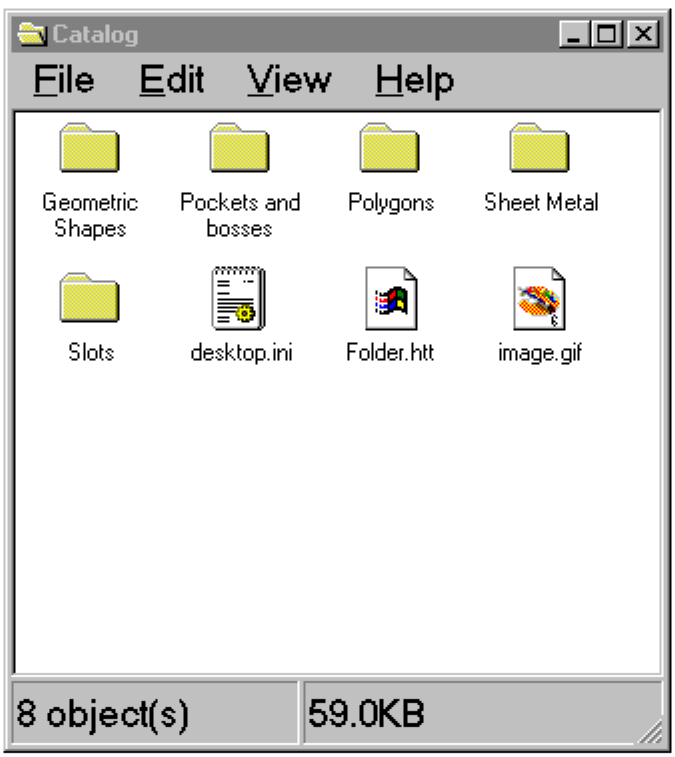
Edges	Selects individual corners to chamfer and previews default distance.
Face	For chamfers defined by a distance and angle, selects the affected sheet metal edge.
Flip	For chamfers defined by two distances, flips the direction of the chamfer distances.

The Distance edit box specifies the extent of selected chamfer method.

Distance	Specifies offset distance of chamfer from selected sheet metal edge. For chamfers defined by two distances, specifies both offsets.
Angle	For chamfers defined by a distance and angle, specifies the angle of the chamfer.

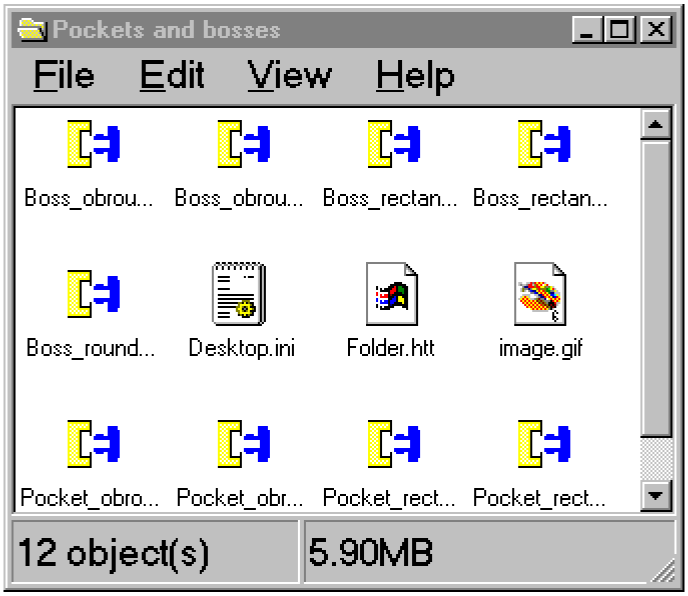
View Catalog

The Catalog is similar to AutoCAD Design Center. The user can browse a library of parts and blocks that can be inserted into the current file.



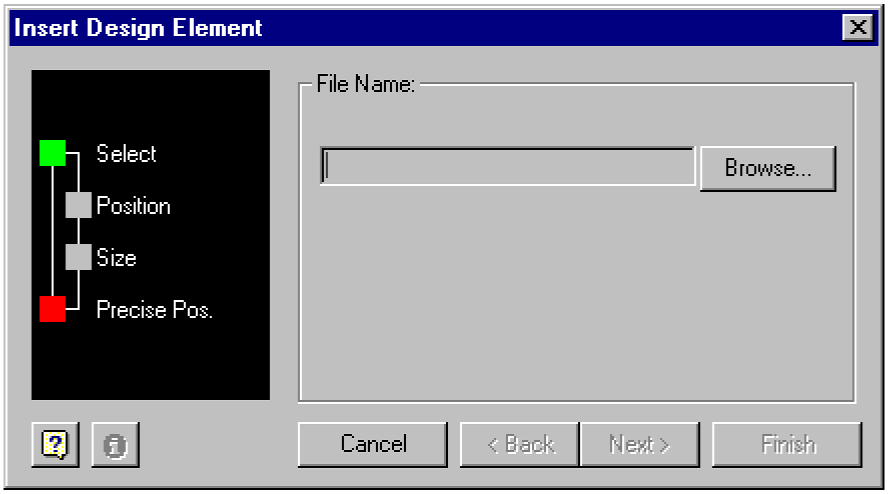
Inventor includes a set of features ready for use. Users can explore the pre-existing library of sketches to save time when constructing their models.

Design elements are stored in feature files with the **.ide** extension. You can open a feature file to view and edit the design element.

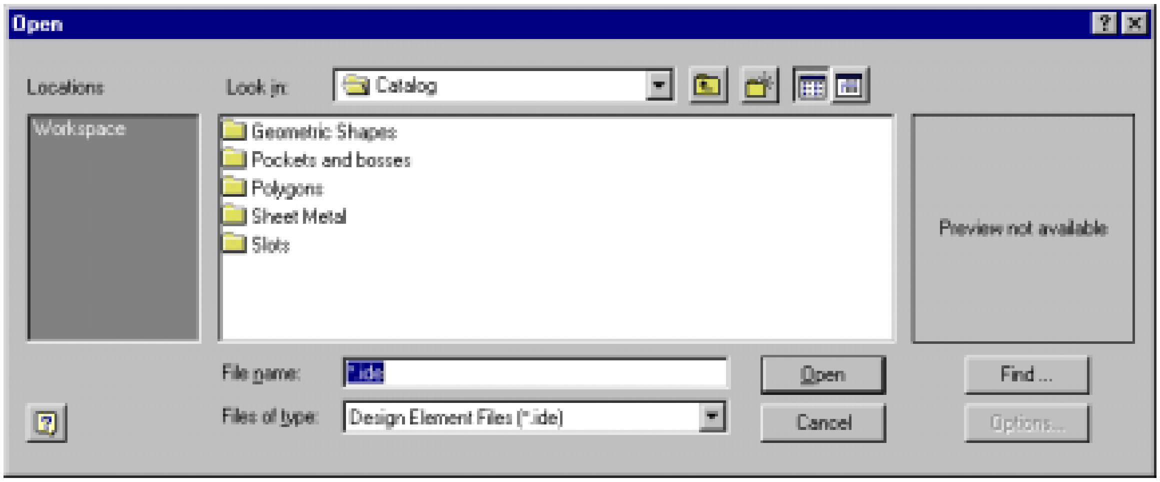


The library features can be previewed using the Insert tool.

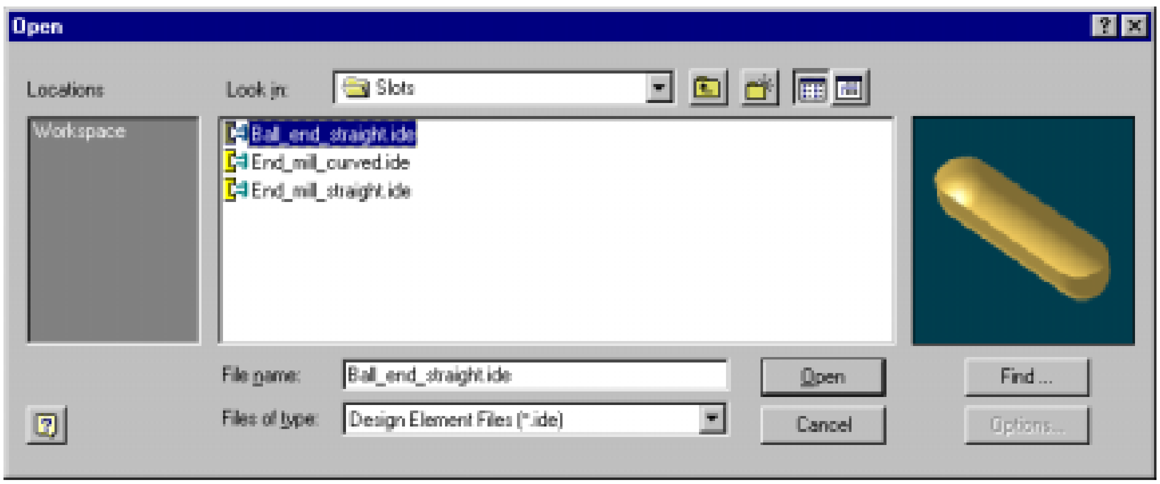
Insert Design Element



Selecting this tool brings up the dialog box shown. Select the ‘Browse’ button to locate the element to be added to the model.

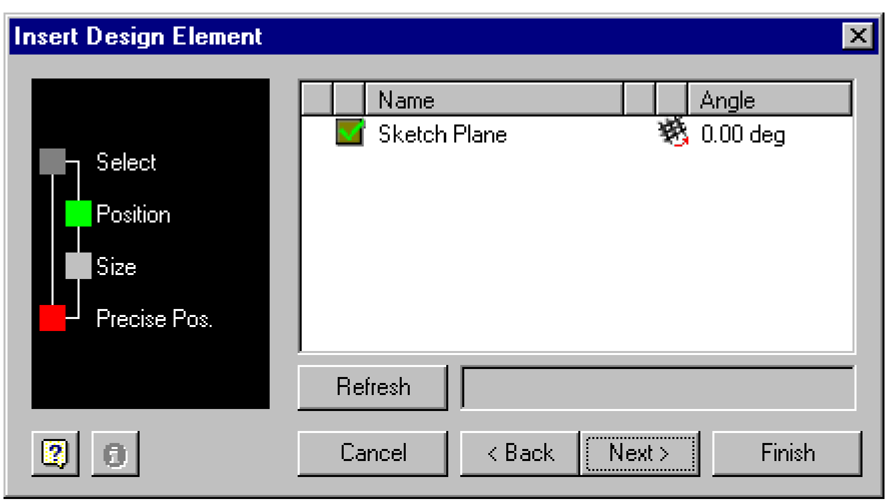


The user can then select the element category.

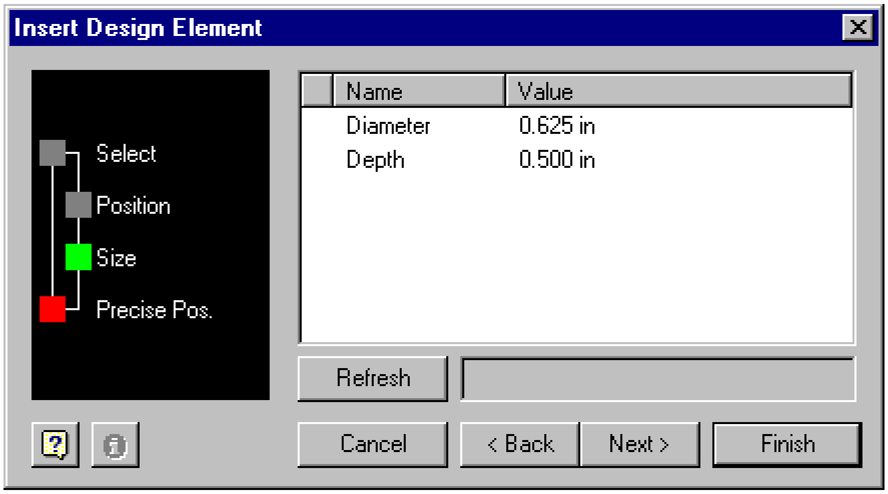


Once we get into the category area, we can preview each element to determine which one we want.

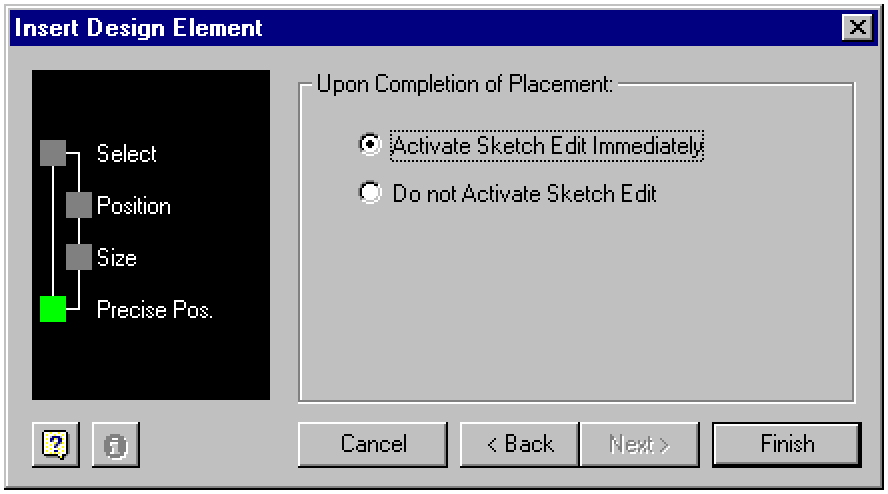
Select the desired element and press ‘Open’.



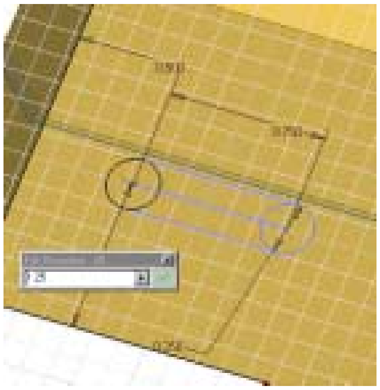
Once the element is selected, the next dialog appears. The user is required to select the plane on the existing model to place the element.



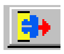
The user is then prompted for the values for the element to be placed. At this time, the user can specify his specifications for the design element.



The user is then allowed to set whether to enter Sketch Edit mode immediately upon placement. Entering Sketch Mode allows the user to reposition the element or make further modifications to the basic element.



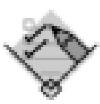
Modifying the inserted element is exactly the same as if we had created the sketch from scratch.



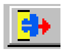
Create Design Element

Use the Create Design Element tool to extract and save a sketched feature in a catalog for future use. If the selected feature has geometrically dependent features, they are automatically selected, but you may delete them using the Create Design Element dialog box. To extract all the features in a part, select the base feature.

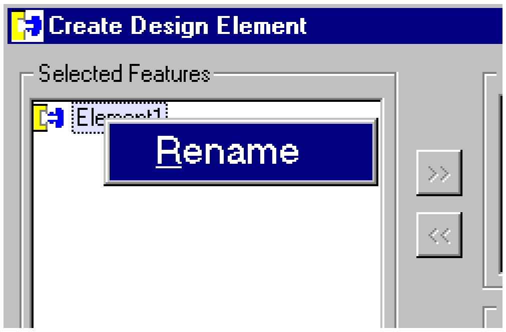
1. Select the Create Design Element tool from the pop-up menu.
2. On the model or in the browser, select one or more features to extract.
3. Save the design element with a unique name.



TIP: The sketch must be turned into a feature using Extrude, Revolve, or Sweep BEFORE it can be turned into a design element.



The Create Design Element tool may also be used to RENAME a design element.



In the Create Design Element dialog box, the feature and the geometry used to create it are listed in the Selected Features tree.

1. Right-click the top level in the feature tree.

2. In the edit box, enter a new feature name.

The new name appears in the browser when you place the design element, but does not change its saved file name. To make the design element easier to use, give similar names to the design element file (.ide) and the design element feature.

TIP: Do not use spaces and other special characters in the feature name. It is included in the parameter name and may be used in equations when placing the design element.

Create Work Plane

Use the Work Plane tool on the Feature toolbar to define a work plane using unconsumed sketch geometry, feature vertices, edges, faces, or other work features. Work planes can also be created in-line when a work feature command requires you to select a plane. Use one or more of the following relationships to define a work plane:

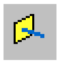
- On geometry (on three points, for example)
- Normal to geometry
- Parallel to geometry
- At an angle to geometry (on a plane and an axis)

1. Click the Work Plane tool.
2. Select appropriate vertices, edges, or faces to define a work plane.

   For offset work planes, drag the work plane to desired location and enter a distance or angle in the Offset dialog box. Click the check mark in the dialog box to accept the preview and create the offset work plane.

If desired, you can create multiple work planes offset from one another at a specific distance or angle. Follow the steps above, selecting the last created work plane as the sketch plane, then drag the new work plane to the desired offset distance.

TIP: If more than one solution is possible, a selection box appears. Click the forward or reverse arrows in the selection box, then click the check mark when the correct solution is previewed.

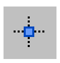


Work Axis

Use the Work Axis tool on the Feature toolbar to designate unconsumed sketch geometry, points, or a part edge as a work axis. Work axes can also be created in-line as input to other work feature commands.

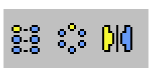
1. Click the Work Axis tool.
2. Select one of four methods:
    - Select a revolved feature to create a work axis along its axis of revolution.
    - Select two valid points to create a work axis through them.
    - Select a work point and a plane (or face) to create a work axis normal to the plane (or face) and through the point.

Select any two non-parallel planes to create a work axis at their intersection.



Work Point

Create a work point to constrain other sketch geometry. Work points can be placed anywhere on the active sketch plane.



The last three tools in the Sheet Metal toolbar are the same as the ones used in the Features toolbar. They are Rectangular Pattern, Circular Pattern, and Mirror Feature. Refer to Lesson 5 for more information on these tools.

Sheet Metal Tools

Button	Function	Settings/Options	Special Instructions
	Styles	Sets sheet metals styles	Use the Variable Thickness to specify values under the Bend tab to boost productivity.
	Flat Pattern	Creates a flat pattern of the sheet metal part.	
	Face	Creates a sheet metal face.	Similar to Extrude.
	Contour Flange	Uses an open profile to create a flange	
	Cut	Removes a profile from a sheet metal face.	
	Flange	Creates a flange on a sheet metal edge	
	Hem	Creates several different styles of hems	
	Fold	Creates a fold	Uses a single sketched line
	Corner Seam	Creates a corner seam between two sheet metal faces	
	Bend	Creates a bend between two sheet metal faces	
	Hole	Creates a Hole	This is the same as the Features tool
	Corner Round	Creates a fillet or round on a corner	
	Corner Chamfer	Creates a chamfer on a corner	
	View Catalog	Open a Catalog of Design Elements	
	Insert Design Element	Add a Design Element	
	Create a Design Element	Create a Design Element from an Existing Feature	
	Work Plane	Create a work plane	
	Work Axis	Create a work axis	
	Work Point	Create a work point	
	Rectangular Pattern	Create a rectangular pattern of one or more features	
	Circular Pattern	Create a circular pattern of one or more features	
	Mirror Feature	Mirror one or more features along a plane	

Review Questions

1. The three tabs on the Sheet Metal Styles dialog box are:

    A. Sheet, Bend, Corner
    B. Material, Bend, Corner
    C. Material, Thickness, Bend
    D. Sheet, Material, Properties

2. The Flat Pattern is used as:

    A. A method to determine problems with the design
    B. A view in a drawing file
    C. A method for designing sheet metal parts
    D. A silverware pattern

3. True-False
   If you delete the flat pattern in a sheet metal file, the flat pattern view will also be deleted in the corresponding drawing file.

4. The file extension for a sheet metal part file is:

    A. spt
    B. iam
    C. idw
    D. ipt

5. The Face command is similar to the command:

    A. Work Plane
    B. New Sketch Plane
    C. Extrude
    D. Extend

6. When adding a bend to a part:

    A. Inventor will automatically trim or extend the perpendicular faces to form the bend
    B. The user must trim or extend the perpendicular faces to form the bend
    C. Inventor will automatically extend perpendicular faces to form a bend, but can not automatically trim.
    D. Inventor will automatically trim perpendicular faces to form a bend, but can not automatically extend.

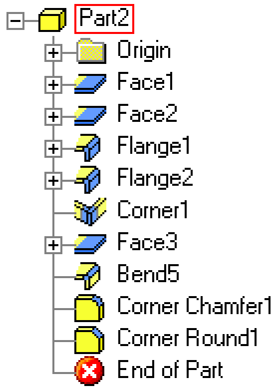
7. To create a cut in a sheet metal part:

    A. Draw a profile and then Extrude
    B. Draw a profile and then Cut
    C. Draw a profile and then Subtract
    D. Draw a profile and then Intersect

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

8. To add a flange to a sheet metal part:

    A. Draw the profile, extrude, and then add a bend
    B. Draw the profile, create a face, and then add a bend
    C. Draw the profile, and then use flange (flange will automatically create the bend and bend relief)
    D. Draw the profile, use flange and then bend



9. A corner seam is indicated in the browser by the prefix:

    A. Face
    B. Flange
    C. Corner Chamfer
    D. Corner

10. The Design Elements tools are available on the Sheet Metal toolbar. Another toolbar where we see these tools is:

    A. Sketch
    B. Feature
    C. Solids
    D. Assembly