



Walt Disney World Swan and Dolphin Resort
Orlando, Florida

Sheet Metal Design: Beyond Bends and Flanges

Anthony B. Rodriguez - KETIV Technologies

MA31-2 Develop sheet metal parts within the assembly to ensure fit and function. Understand all of the sheet metal style settings. Implement standards within your team that follow manufacturing rules. Create and unfold advanced sheet metal shapes. With the ability to deliver an accurate design and flat pattern, you will reduce fabrication lead times and costs.

About the Speaker:

Anthony is responsible for conducting business audits and Autodesk Inventor implementations for KETIV Technologies, one of the top mechanical resellers. He was recently awarded the Inventor Channel Excellence award for his work. He is a Certified Inventor Expert and a Certified Inventor Implementation Expert. Anthony is also one of the founders of the Southern California Autodesk Inventor User Group. Prior to joining KETIV, he worked as a product R&D manager in the telecommunications industry for 5 years. Anthony is certified as an ISO9001 auditor.

anthony@ketivtech.com

Section I – Sheet Metal Templates

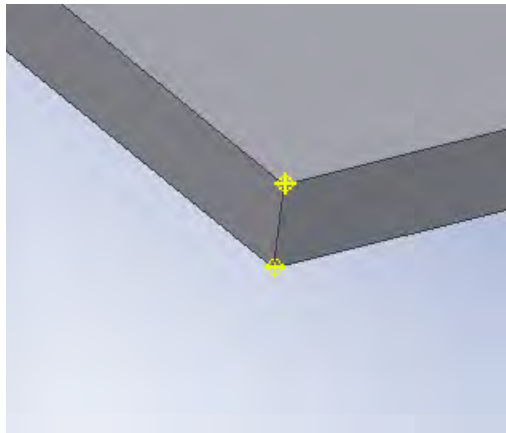
When creating your sheet metal template you will want to save the following within the template file:

- Sheet Metal Styles
- Materials
- Starting plate with workpoints

Sheet Metal Styles – This will allow you to save predefined materials along with manufacturing specifications and rules for each material.

Materials – Save accurate material specifications. This is necessary when generating weight calculations of finished parts.

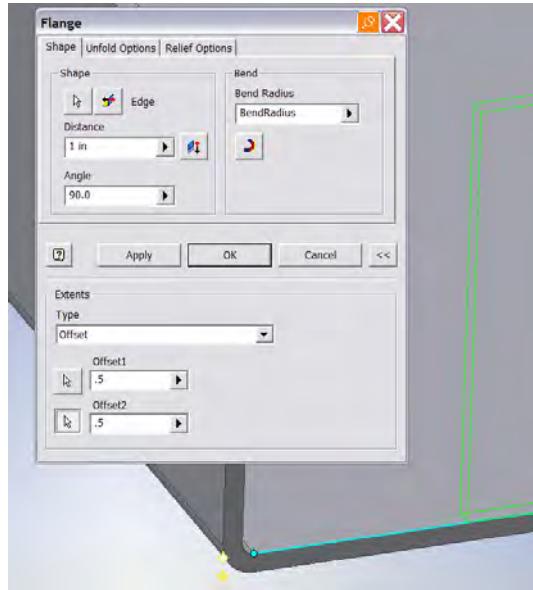
Starting plate with workpoints – When creating rectangular sheet metal parts it is recommended to start with a flat plate along with workpoints on all corners of the plate. Eliminate these respective steps each time you make a new part by saving a rectangular face along with the workpoints in your template as shown below.



Now when starting a new part you can change the rectangular sketch and you are off. There are two benefits to having the workpoints on all corners of the initial face.

1. You can now dimension all holes and cuts to the workpoints. This allows you to easily delete features later in the development of your part without affecting the cuts and holes which otherwise may have been dependant on the deleted feature causing error messages.

- When creating an offset flange you can now offset from the original corner or outer face of your part as shown in the image.



Section II – K Factor & Bend Allowances

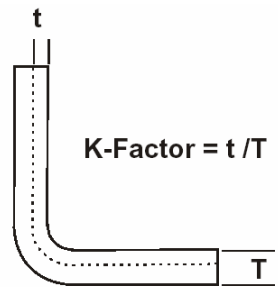
When sheet metal is bent, the inside surface of the bend is compressed and the outer surface of the bend is stretched. Within the thickness of the metal, lies its Neutral Axis, which is a line in the metal that is neither compressed nor stretched.

What this means in practical terms is that if we want a work piece with a 90 degree bend in which one leg measures A, and the other measures B, then the total length of the flat piece is NOT A + B as one might first assume. To work out what the length of the flat piece of metal needs to be, we need to calculate the **Bend Allowance** or **Bend Deduction**. This will tell us how much we need to add or subtract to our leg lengths (A & B) to get exactly what we want.

$L_t = A + B + BA$	$L_t = A + B - BD$
<p>where: L_t is the total flat length A and B are shown in the illustration BA is the bend allowance value</p>	<p>where: L_t is the total flat length A and B are shown in the illustration BD is the bend deduction value</p>

The location of the neutral line can be different depending on the material itself, the radius of the bend, the ambient temperature, direction of material grain, and the method by which it is being bent, etc. The location of this line is what is referred to as the “**Kfactor**”.

K-factor is a ratio that represents the location of the neutral with respect to the thickness of the sheet metal part.



sheet

The only truly effective way of working out the correct bend allowance is to reverse engineer a sheet metal part. a measured strip of material, bending it, and measuring it you the correct bend allowance. These bend allowance can be measured for many materials and scenarios and then be used in an Inventor bend table.

Taking
will give

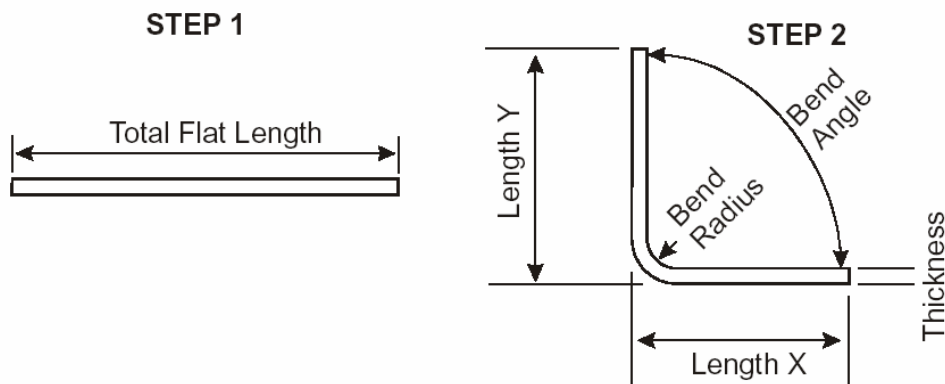
Reverse Engineering the K-Factor

First, cut a strip of material and measure its length and thickness as accurately as possible. The width of the strip is not that critical but generally somewhere around 4 inches or so will work.

Then, bend the strip to 90 degrees, and measure its Length X and Length Y as shown in the diagram below.



Note: This is very important to bend the sample piece in exactly the same manner as you plan to bend your real pieces, so that whatever you measure now becomes reproducible later.



The correct K-factor to use in Autodesk Inventor can now be calculated as follows:

$$\text{BendDeduction} = \text{Total Flat Length X} - \text{Length Y}$$

$$\text{K-Factor} = (-\text{BendRadius} - (\text{BendDeduction} / (\pi * \text{BendAngle} / 180))) / \text{thickness}$$

The other option Inventor has is a Bend Table option to control flat patterns which will be discussed next.

Section III – Bend Tables

When you select Bend Table as the unfolding method in the Sheet Metal Styles dialog box, the flat pattern analyzer uses the bend table to create the flat pattern. A bend table contains a bend allowance for the specified material thickness at specific bend radii and bending angles.

Autodesk Inventor provides sample Imperial and Metric bend tables as spreadsheet files. You can use them to create a table for each material type and store multiple bend tables in one file.

Using the sample bend tables, you specify units of measure, and tolerances.

1. Browse to the sample bend table spreadsheets in the \Autodesk\Inventor\Samples\Bend Tables folder and double-click one of the .xls files to open.
2. Enter units of measure. Scroll to the line that starts with **/U** and enter units of measure. The default is cm.
3. Enter tolerances. Scroll to the lines that starts with **/T** and enter the following:
 - o On line **/T1**, enter the +/- tolerance for sheet thickness.
 - o On line **/T2**, enter the equal minimum and maximum bending radius.
 - o On line **/T3**, enter the tolerance for equal minimum and maximum opening angle.



Note: Tolerances 2 and 3 are used only under special conditions, as noted in the table.

4. Select File>Save As, give the spreadsheet a new name, and click OK.

Use the bend table file with your units of measure and tolerances to create a bend table for each material thickness.

To begin, browse to the bend table file and double-click to open.

1. Enter sheet thickness. Scroll to the line that starts with **/S** and enter thickness.
2. Enter bend radii. Scroll to the line that starts with **/R** and enter radii.
3. Edit the equation used to calculate the bend allowance, as necessary.

Note: All cells in the sample bend table use the same equation, but you can customize the equation for individual cells.

4. Highlight the table, then press Ctrl+C to copy. Scroll down to an empty area of the spreadsheet and then press Ctrl+V to paste.
5. Rename the table and reset the material thickness, as needed.
6. Continue to copy and paste the table, renaming it and setting a new material thickness, as desired.
7. Select File>Save As, select .txt file type, and then click OK.



Note: It is important to save the spreadsheet in a text (.txt) file format. The analyzer uses the text format to calculate bends.



Note: Once the text file is selected to be used as the unfold method in the sheet metal styles dialog box it is embedded into the file. It is not associated to the text file.

Section IV – Sheet Metal Styles

Modify List – When the Modify List button is selected, the Unfold Method List dialog box opens. It contains a list of the k factors you have specified and the bend tables you have selected. You can add a new k factor or bend table entry using the New button. After the entry has been added to the list, you can change the type from linear to bend table, change the name, select a text file that includes the bend table values, or enter a value for the k factor.

Any entry can be removed from the list except the default k factor value (the first item in the list).

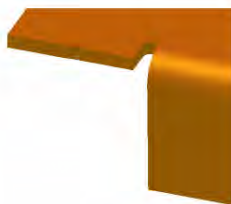
Radius – Sets the value for the default bend radius. This parameter can be overridden on an individual feature.

Relief Shape – Inserts a bend relief if the bend does not extend the full width of a sheet metal face. Select None (no relief), Round (full radius corners), or Square (square corners).

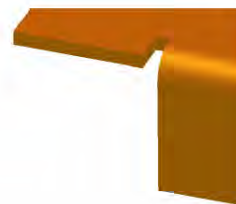
None



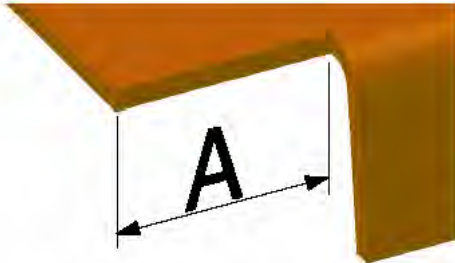
Round



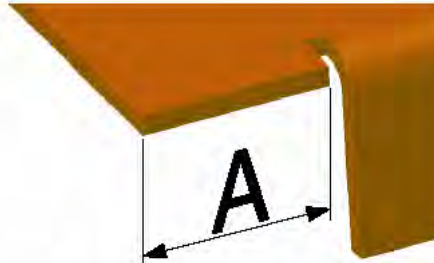
Square



Minimum Remnant – Specifies the minimum remnant to leave after a bend (the amount of material between the bend relief and the edge of the sheet metal part). The remnant and the bend relief are removed if the distance between the bend relief and the side of the part is smaller than this value, unless the bend relief type is None. Use the default or specify a value.



Minimum Remnant > A



Minimum Remnant < A

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.

Transition – Where no bend relief has been specified, Transition specifies the type of transition for the bend in an unfolded state.

None



Intersection



Straight Line



Arc



Trim to Bend



Relief Depth – Sets the distance the bend relief extends past the bend zone. If the relief type is Round, the relief depth must be greater than half of the relief width.

Corner Relief Shape – Inserts a corner relief when a corner seam is applied and three faces come together in the corner.

- Trim to Bend for no corner relief.
- Round for a circular corner relief.
- Square for a square corner relief.
- Tear for a torn corner relief.

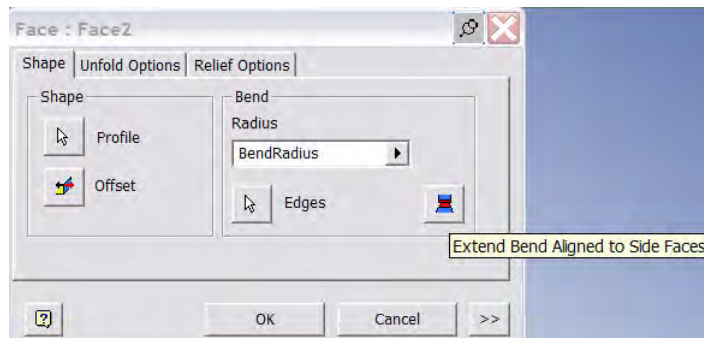


Relief Size – Sets the corner relief size. Specify a value that extends the corner relief past the bend lines on the largest bend.

Section V – Modeling Features

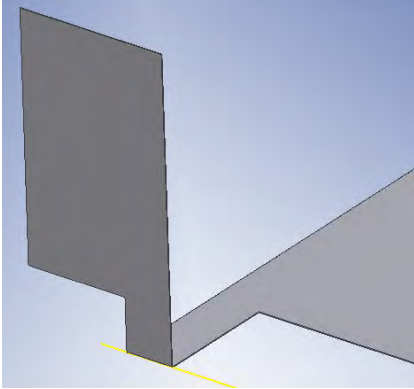
Face – In the face dialog box there is an option to “Extend Bend Aligned to Side Faces” as shown in the dialog box below.

This option: Extends material along the faces on the side of the edges connected by the bend instead of perpendicularly to the bend axis.

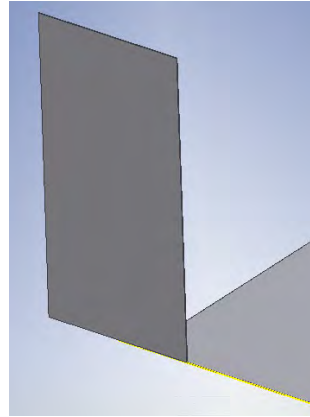


See Examples below:

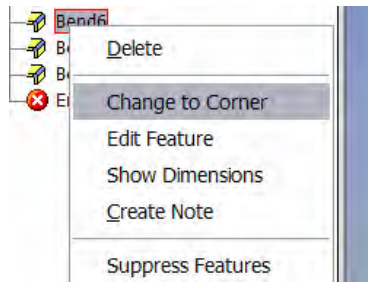
Not selected



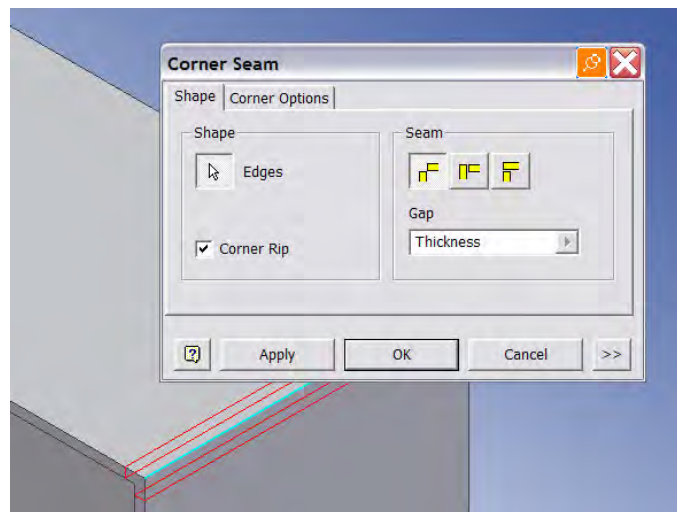
Selected



Bend – Once a bend or corner seam has been applied to a sheet metal part you can easily switch the feature back and forth between a corner seam and bend. This allows you to quickly change the manufacturing approach for the part.



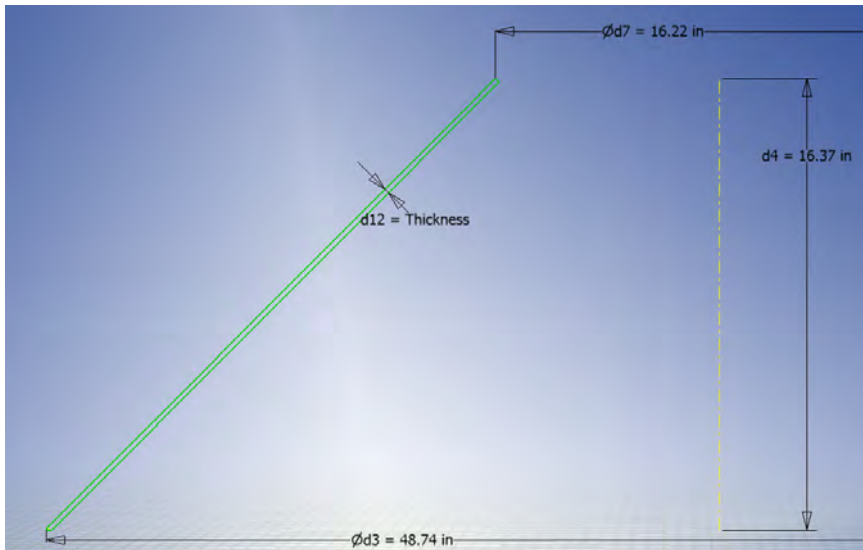
Corner Seam – In the corner seam dialog box you have the ability to use the “corner rip” option. This allows you to quickly convert non-sheet metal parts to sheet metal parts with proper seams for unfolding.



Section VI – Cones and Cylinders

The flat pattern tool in Inventor will unfold cones and cylinders. The parts must be built with a seam allowing the part to unfold. Other shapes like square to round transitions and ducting can be created in Inventor however not unfolded. In order to unfold parts like these you can use AutoPOL www.fccsoftware.com an Inventor certified 3rd party product.

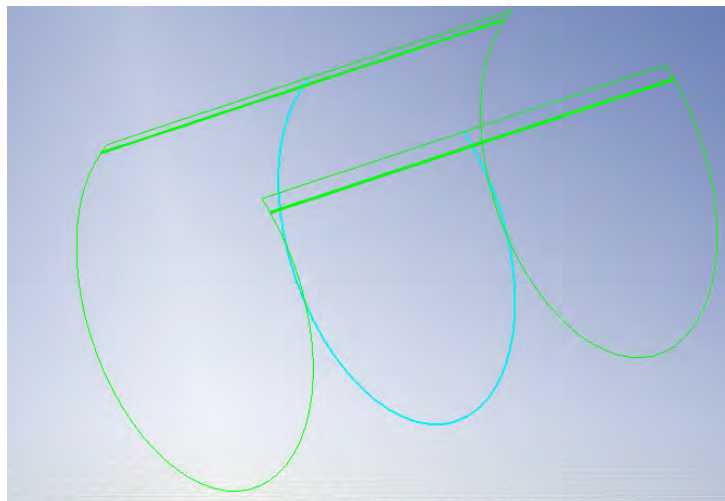
Cones – The most common approach for creating sheet metal cones is to use the revolve tool with a sketch as shown below.



By revolving the sketch you will generate your conical shape. The key is to revolve at an angle. This allows you to create a seam. An example would be to revolve at 359 degrees to create a real small seam.

Cylinders – To create a cylinder you can use multiple approaches.

One approach would be to sketch the cross section of the cylinder shape and extrude.



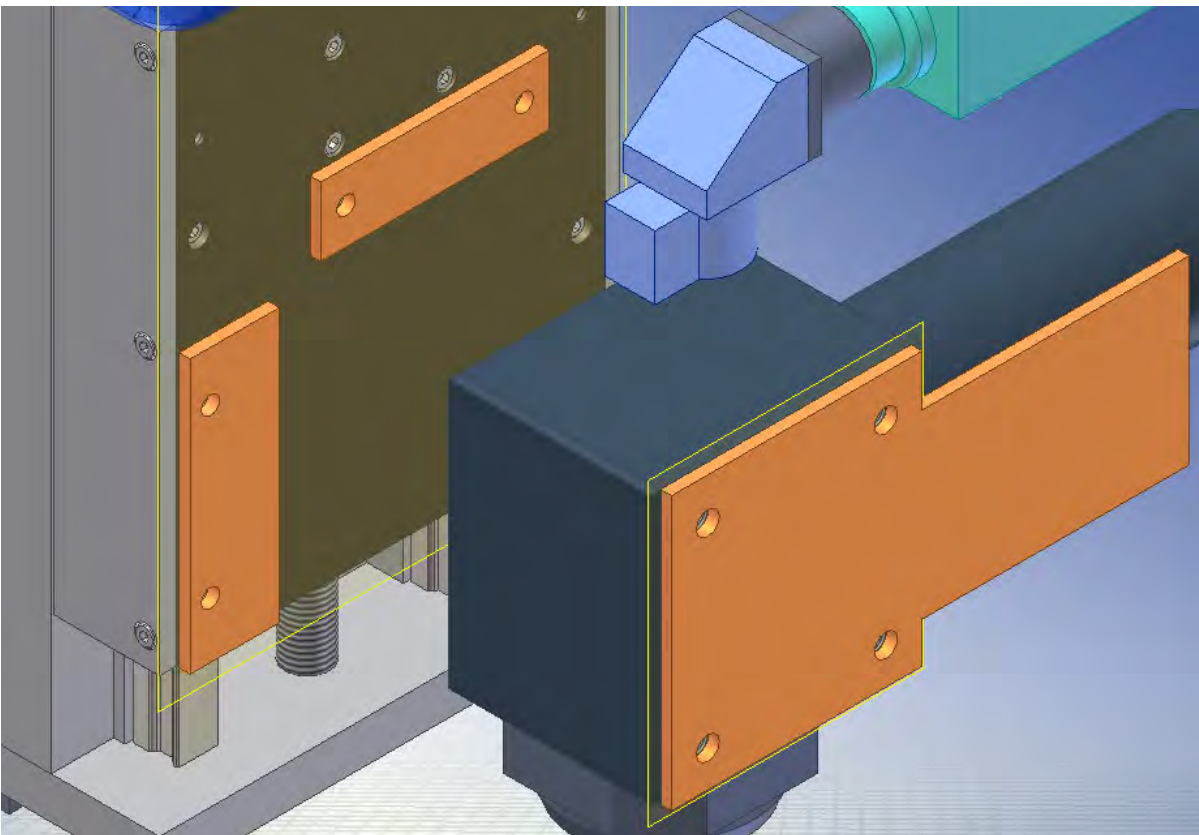
Another example would be to use the revolve option shown above in the Cone example.

The key to creating the flat pattern of these shapes is to select one of the cylindrical/conical faces prior to selecting the flat pattern tool.

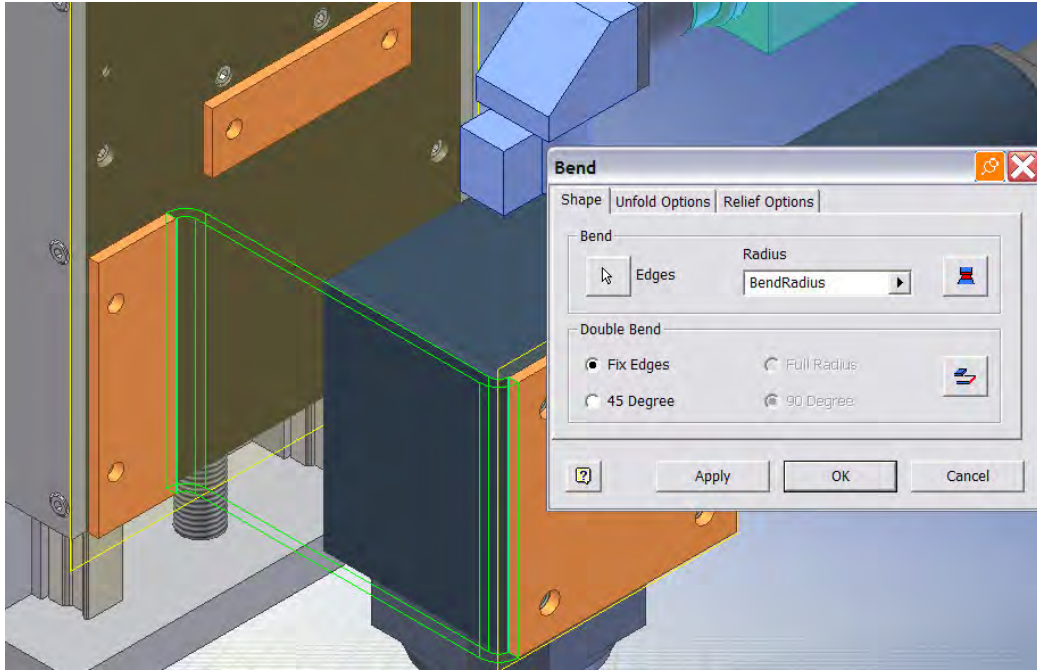
Section VII – Design in the Assembly

There are several techniques we can use in Inventor to create intelligent sheet metal parts in the assembly. One of the approaches is the unique ability to build your sheet metal parts dis-jointed and then connect or join the dis-jointed faces at the end of the design.

In this picture the orange faces are a single part. I was able to build the mounting locations for the part early in the development without having to worry about connecting the faces. This allows me to quickly meet the design requirements and let Inventor calculate how to best connect the faces to complete the part.



By using the bend tool next, I will be able to easily connect the faces. This will enforce the Sheet Metal Style settings when building the remaining geometry.

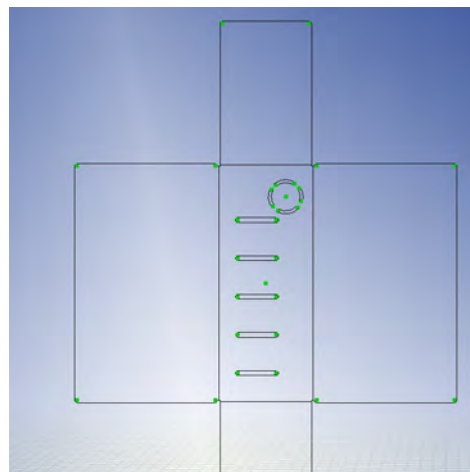


Section VIII – Sheet Metal Import

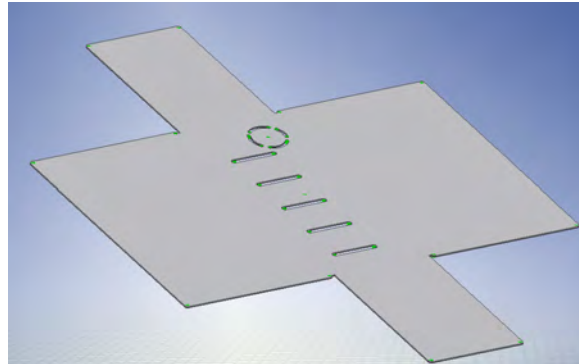
Autodesk Inventor allows you to import files from other software packages and still re-use them as Inventor sheet metal parts.

Re-Using AutoCAD flat patterns – If you have flat patterns that have been drawn in AutoCAD you can re-use them to reverse engineer a folded Inventor part.

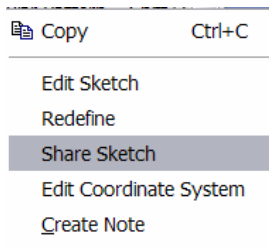
While active in a sketch use the AutoCAD import option. This will bring in the AutoCAD flat pattern and bend lines.



The next step is to thicken the entire flat pattern with the face tool. As shown below:



From here you can now reuse the original sketch that came in from AutoCAD by making it a shared sketch. Right click on the sketch and select the Shared Sketch option.

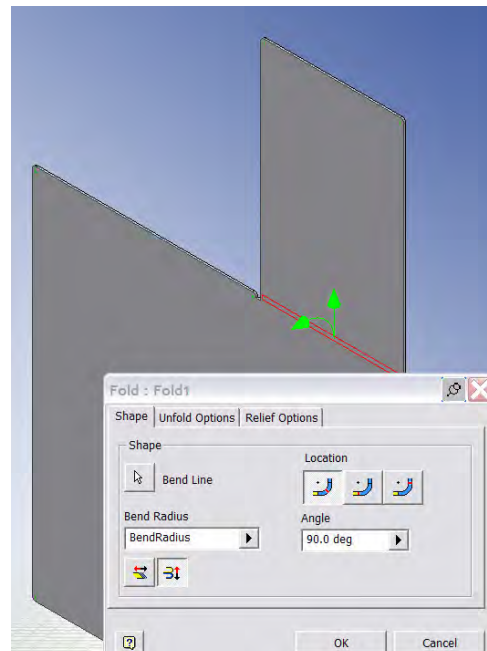


Next you will use the fold tool to fold the sheet along the bend lines. This will create your folded model.

The fold dialog box requires you to pick the bend line. You can then enter the bend direction and angle.

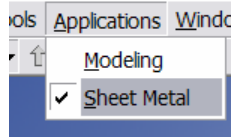
The bend line can also represent the start, center or end of the bend.

Repeat this step for each bend line.



Step files – Files that are imported into Inventor via STEP can also be unfolded. AutoCAD solids, ProEngineer and all other modeling products that export to STEP can be imported into Inventor and treated as sheet metal parts.

Once the parts are imported you will have to change the setting in the Application menu to Sheet Metal.



You then have to make sure that your sheet metal style “Thickness” is equal to the thickness of the part. You can use the measure tool if you do not know the thickness.

Once these two items are set you can unfold the part and add flanges, etc.



Note: If the sheet metal part is brought in and there are no bends on the corners you can use the bend tool to add them. This is necessary in order to unfold.

Thank you for attending the class. Please feel free to contact me with any questions.

anthony@ketivtech.com

For Tips and Tricks – www.ketivtech.com

