

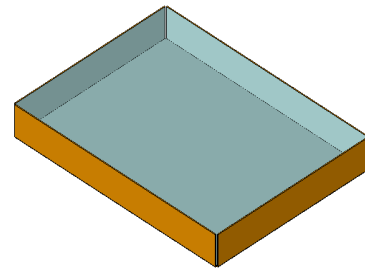


Tips And Tricks

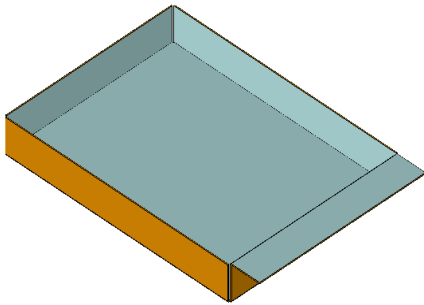


Quick Guide to Keycreator's Sheet Metal Design Tools- Part 2

Welcome back to the Sheet metal Shop! We ended the first part of our sheet metal tips and tricks with the case illustrated to the right.



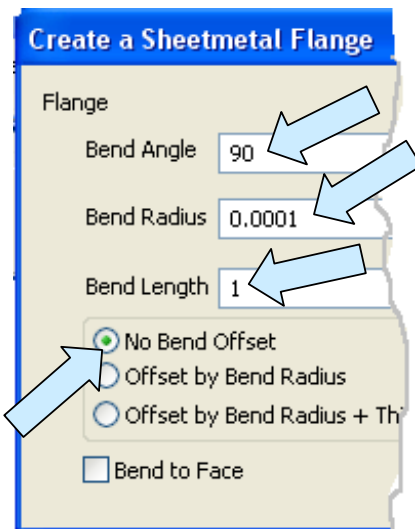
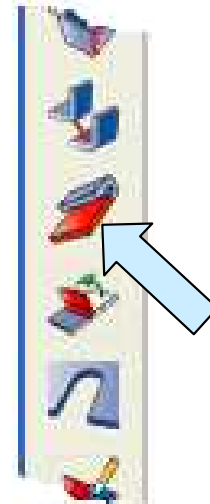
This is available for download as file "metalcas01." You can also load the sheetmetal workspace by clicking on the FILE Pulldown Menu. Then, click on WORKSPACES and then on OPEN WORKSPACES. Select the SheetMetal.wsp in the Dialog Box that appears and click on the OPEN Button.



This will place your sheet metal tools in a vertical toolbar at the right side of the screen.

Now in the beginning of my guide to sheet metal tools-part 1 I showed you a case that looked like this: Let's see how we can quickly create the flange on the right edge of the part.

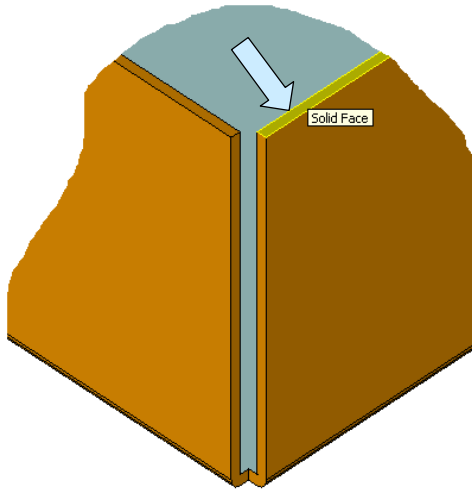
Click on the CREATE FLANGE Icon.



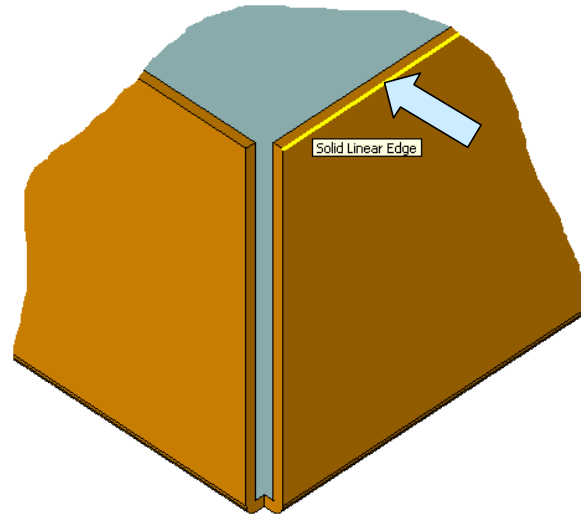
A Dialog Box appears.

Let's use a bend angle of 90 degrees, a bend radius of 0.0001 and a bend length of 1 inch.

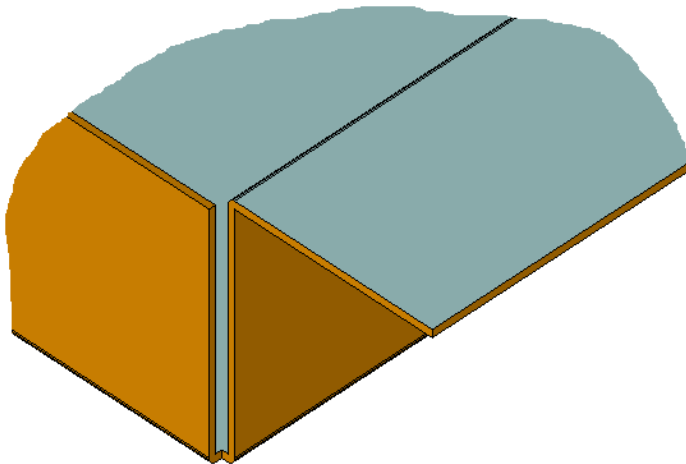
We'll use the No Bend Offset Option. Notice the other possibilities. Click on the OK Button.



Move the cursor over the top edge face on the left side of the part and click on it when it highlights.



Next, select the edge indicated by the arrow in the illustration to the right.



A flange is created like the one illustrated to the left.

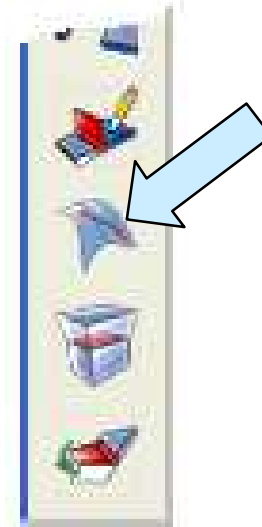
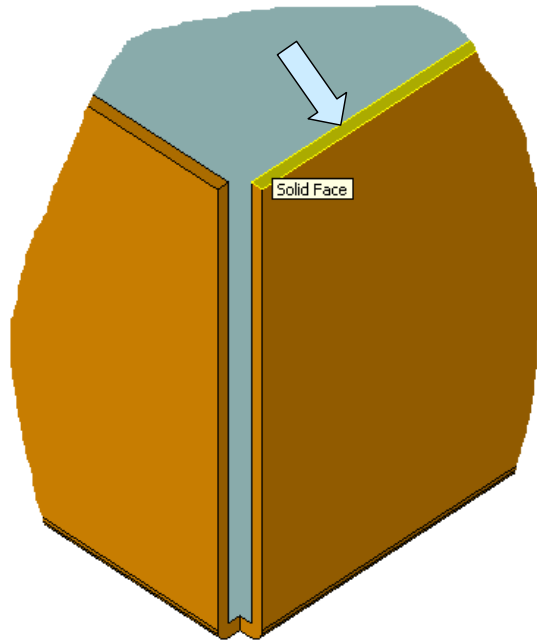
Now, let's suppose that we want to form a partial flange that is centered on the right side of the part.

Click on the UNDO Icon to eliminate the full flange.



To create a partial flange, we have to break the continuous face on the top right edge of the part into three separate faces.

Click on the SCRIBE FACE Icon.

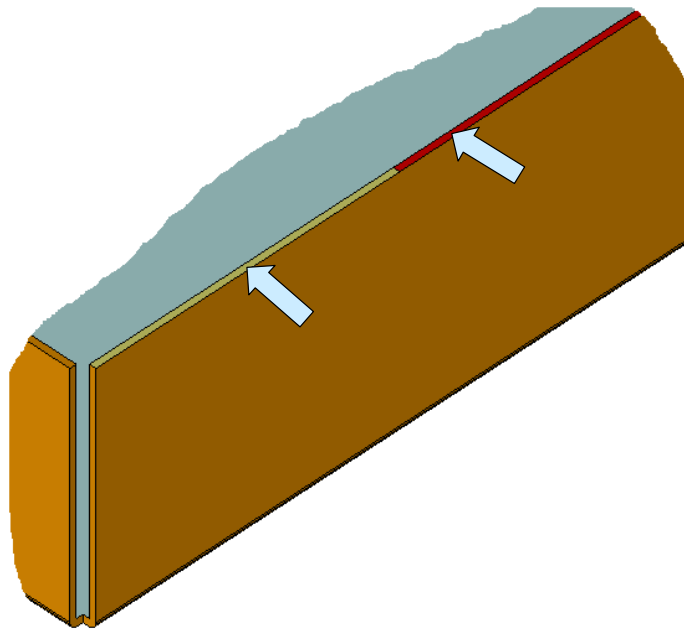


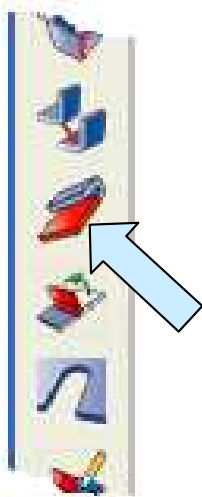
Click on the thin top edge face at the right side of the part.

Now, using the ALONGE Option on the Conversation Bar, click on the front end of the left edge of this face. Then, type 1.5 and hit the ENTER Key.

Repeat, using ALONGE on the right edge of this face with a distance of 1.5 from the front end. Then, click on the DONE Button.

This breaks the face into two new faces. I've highlighted the faces in two different colors in the illustration to the right.

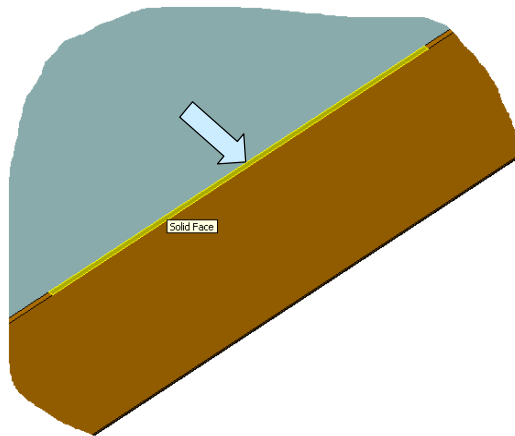
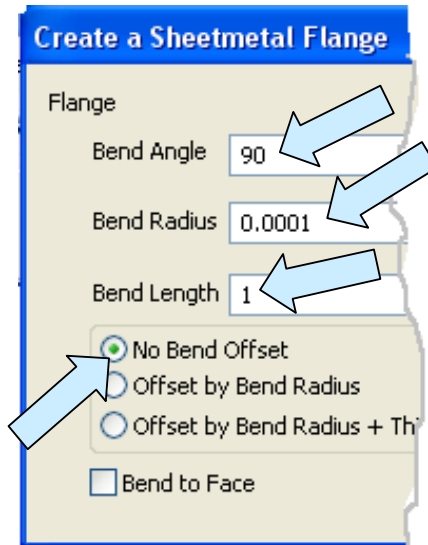




Use the SCRIBE FACE Tool again to create a second scribe line located 1.5 inches from the rear edge of the top, right face. When you are done the top right edge of the part will be broken into three separate faces.

Now, click on the CREATE FLANGE Icon.

A Dialog Box appears. We'll use the same settings that we used before.



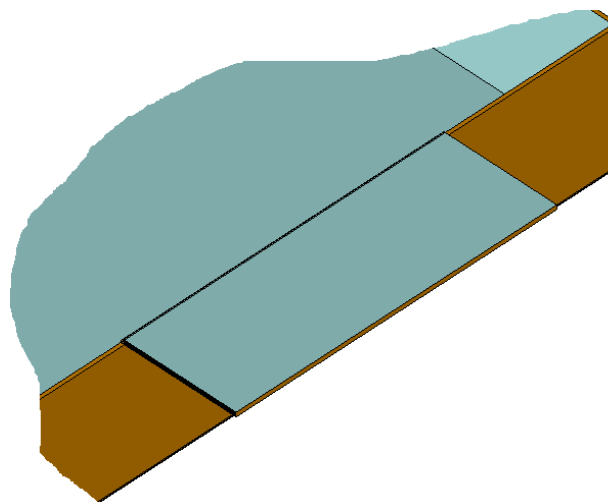
This time, click on the center face on the top, right edge of the part.

Then, click on the outer edge of the face.

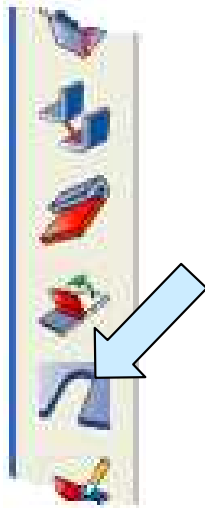
You now have a flange that is centered on the right side of the part.

Now in a typical fabrication we would have to provide relief at the two points where the flange grows out of the right side of the part to prevent metal tearing.

Let's fix that problem using the Bend Relief Tool.

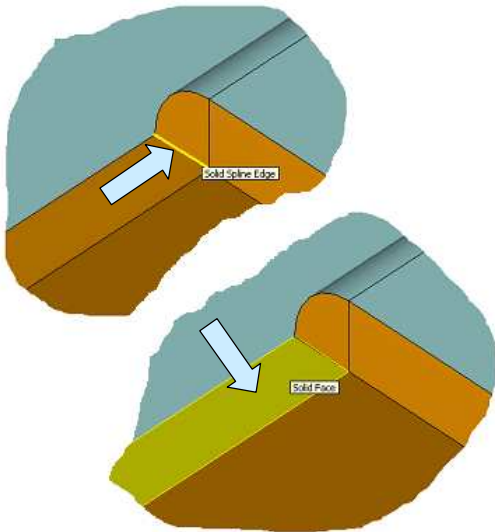
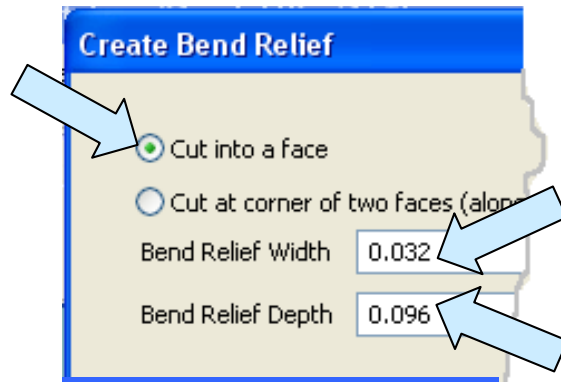


Click on the BEND RELIEF Icon.



A Dialog Box appears.

Click on the Cut Into a Face Option.
Use 0.032 for the Width and 0.096 for the Depth.

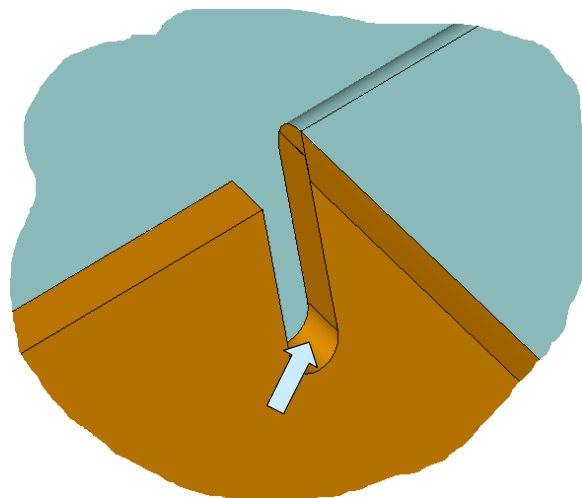


Now, click on the edge indicated by the arrow in the illustration to the left.

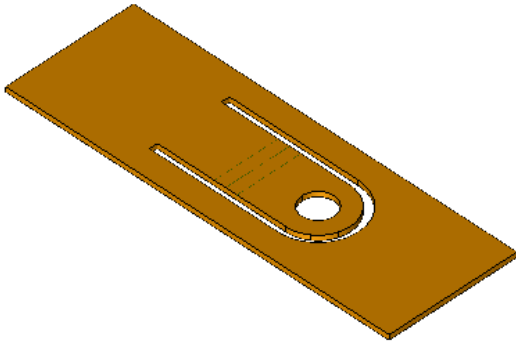
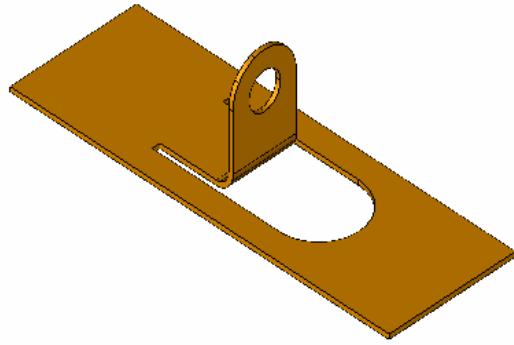
Then, click on the face indicated by the arrow.

You will now get a relief notch like the one illustrated to the right.

You can apply the same treatment to the other side of the flange.



Let's work on one more sheet metal design problem before we wrap up this tips and tricks session. I've illustrated a simple sheet metal forming problem to the right. The bent up tab will be punched out of the main part and we need to create a 0.064 relief completely around the tab in the flat layout.

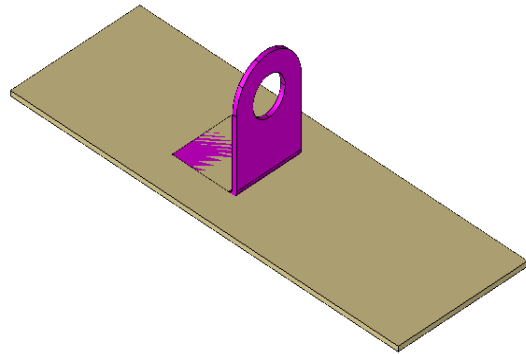


The flat layout will look like this:

How can we quickly do this?

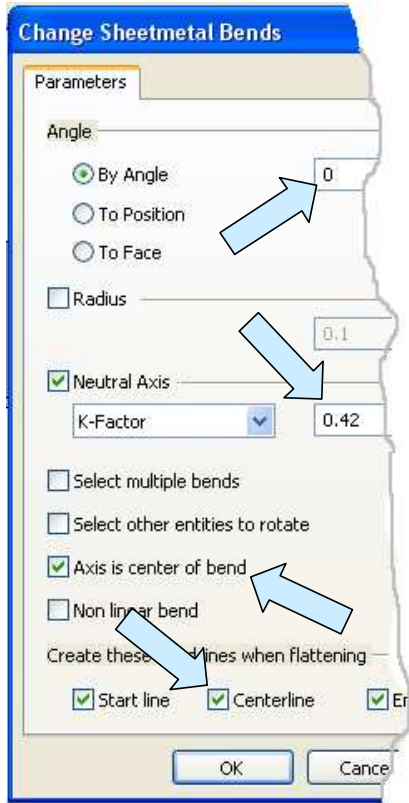
The starting file is provided for download as "SheetmetalTab01."

The thickness is 0.032 and we've created a simple flat sheet with a right angle bend buried in the sheet.



This is a typical start to conceptualizing this type of design. What we now need to do is quickly unfold the angle down into the sheet and then quickly create the relief geometry around the unfolded angle.

Start by clicking on the EDIT SHEET METAL BEND Icon



A Dialog Box appears.

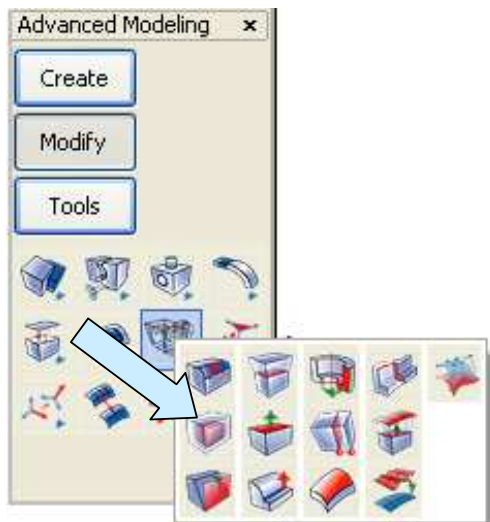
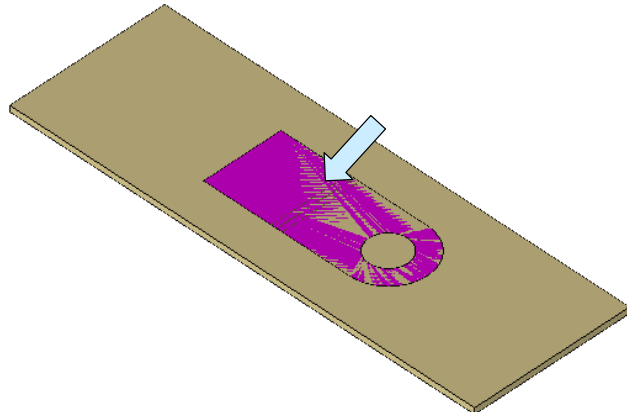
We'll use the By Angle Option with a K-Factor of 0.42

Check on the Axis is center of bend and make sure that the Centerline Option is checked for lines to be created when flattening. The other lines can be selected or not selected.

Then, click on the OK Button.

Now, select the bend in the angle part and then the vertical leg. **NOTE: Select the bend by moving the cursor in from the left side so it "sees" the inside radius of the bend first. There is currently a bug in Keycreator that may give you an error message if you select the bend from the right side.**

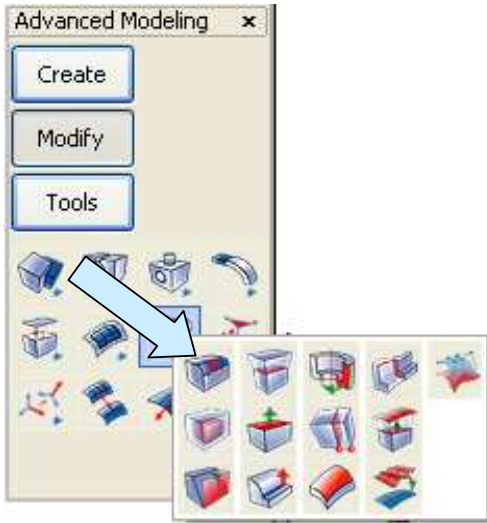
The part will unbend into the larger solid and your screen should look like the illustration to the right.



Now, click on the OFFSET SOLID Icon.

A Dialog Box appears. Click on the Create New Offset Solid option and type 0.064 for the Distance.

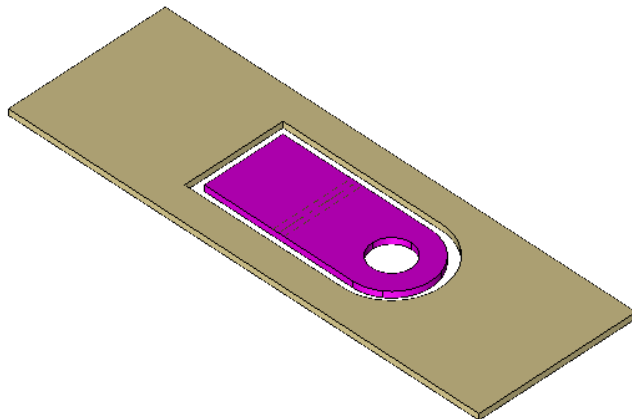
Then, click on the unfolded angle and hit the ENTER Key.



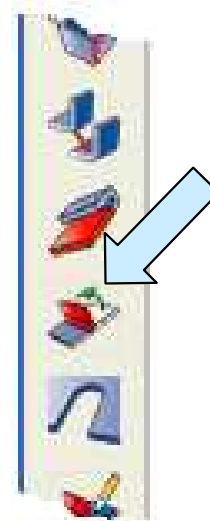
You will now have a solid that completely surrounds the unfolded angle and is 0.064 bigger in all dimensions.

Click on the REMOVE FEATURE Icon and select the hole in this bigger solid. Hit the ENTER Key to confirm the deletion.

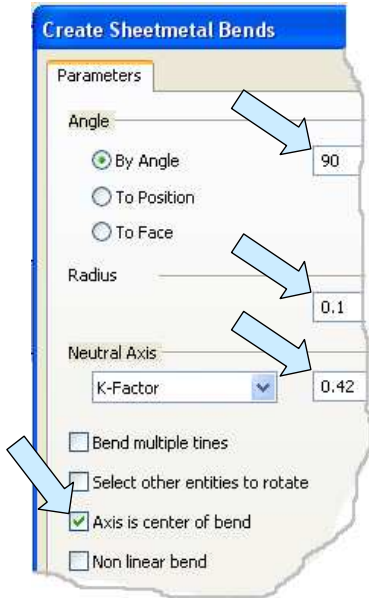
Now, click on the BOOLEAN DIFFERENCE Icon. Select the large plate solid and then the solid that you created with the Offset Function.



Your screen should now look like this:



Click on the ADD SHEET METAL BEND Icon.

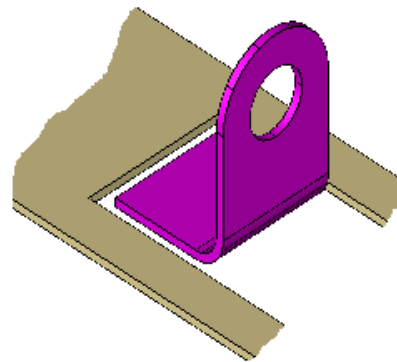


A Dialog Box appears. We'll use the By Angle Option at 90 degrees.

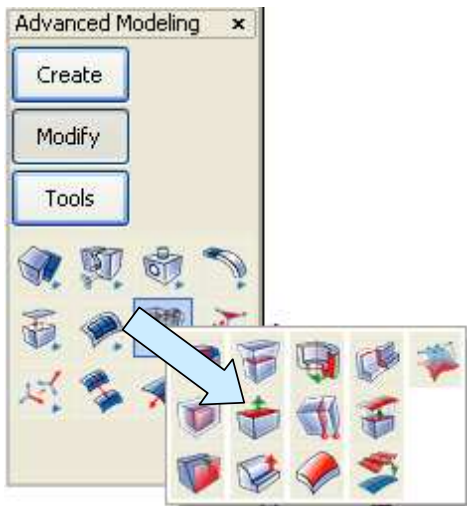
Use 0.1 for the Radius. (This was the inside radius of the original angle.)

Use a K-Factor of 0.42 and select the Axis is Center of Bend Option. Then, click on the OK Button.

Click on the center bend line and then on the rounded end of the tab that has the hole.



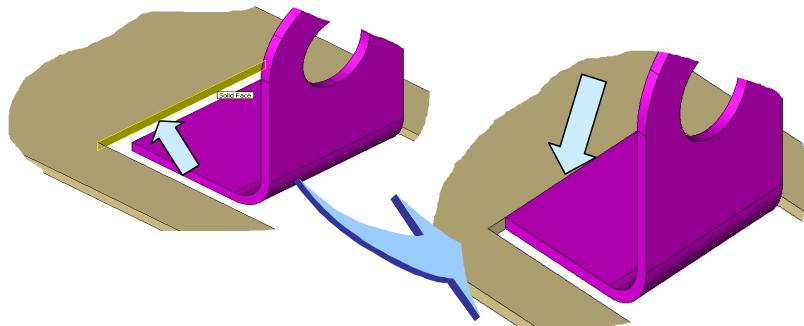
The tab bends back to the original shape.



Now, click on the OFFSET FACE Icon.

Type 0.064 (This is the same amount that you used for the Offset Body Operation.) and click on the face indicated by the arrow in the illustration below.

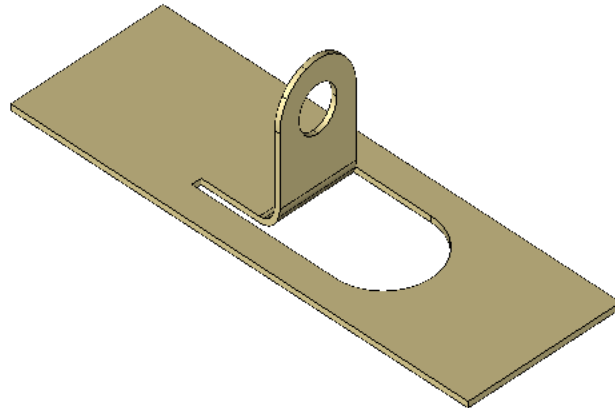
Now the horizontal leg of the angle solid touches the wall of the flat plate.





Our final task is to click on the BOOLEAN UNION Icon.

Select both pieces and hit the ENTER Key.



You will now have a finished piece.

Using the OFFSET SOLID Tool combined with the EDIT SHEET METAL BEND and ADD SHEET METAL BEND Tools provides a quick way to make perfect lance clearance geometry for almost any bent tab configuration that you can imagine.

Once you practice this a few times you will find that it works nicely with the natural creative process used to develop parts.