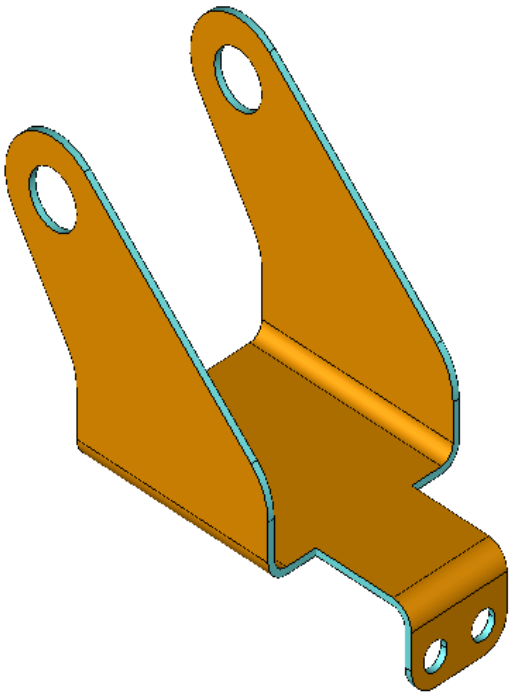
KeyCreator Lesson KC2111

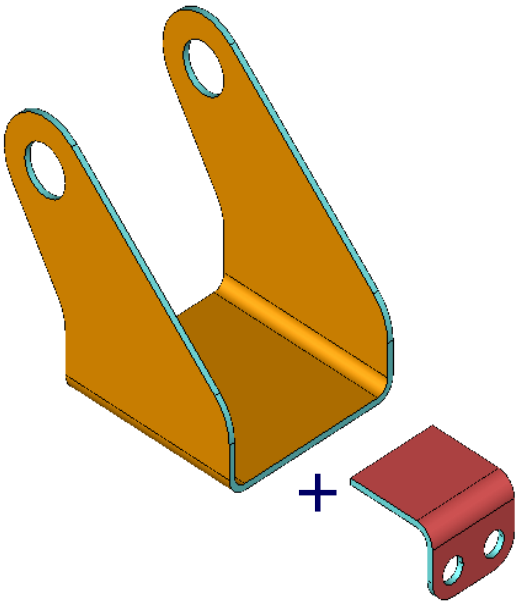
Using Fast Shell to Create a Sheet Metal Part



Any time you want to create a “U-shaped” or “L-shaped” part, you’ll want to consider using the Fast Shell approach to save time!

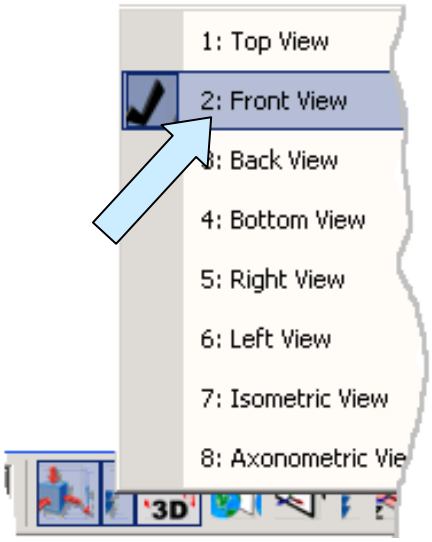
The fast shell approach really pays big dividends when you need to construct a class 2 part. (I define a class 2 part as one containing two or more basic extruded shapes.)

We’re going to build the sheet metal part illustrated to the right. This is a typical example of a class 2 part.



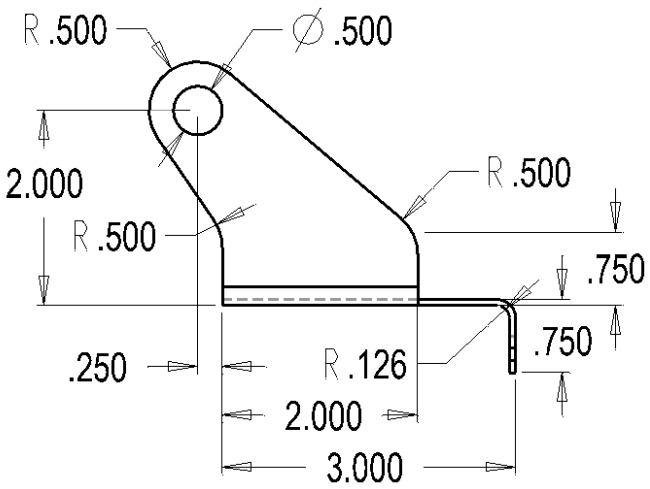
If you look closely at the part, you’ll see that it is really just a combination of a fancy “U-shaped” part and a simpler “L-shaped” part. The drilled holes are just secondary features.

Let’s get started with a new file in View 2. (The Front View.)



For reference purposes, I've illustrated a front view of the proposed part below with key dimensions.

Some of my intrepid readers might be tempted to jump right in and begin constructing geometry.



Let's step back for a moment and look at the big picture before we take the plunge.

First, we'll plan on breaking down the modeling task into two separate tasks.

- Construct a "U-shaped" part.
- Construct an "L-shaped" part.

Let's assume that the part will be made out of 0.063 thick material and that the internal bend radii will be 2 times the material thickness.

Take a moment to access the Online Calculator Function by using the Hot Key CTRL + I. Type "t=0.063" in the Online Calculator Field and hit the ENTER Key. Then, exit the calculator.

This assigns the value 0.063 to the variable "t" during this session of KEYCREATOR. We'll see in a moment how valuable this can be.

Now, let's get started with the construction. I'll assume that you are using the default starting color GREEN.

We'll begin by clicking on the CREATE RECTANGLE WIDTH HEIGHT Icon.

This time, let's click on the BotLeft Anchoring Option on the Conversation Bar.

Type 2 for the dXC value and 0.75 for the dYC value.

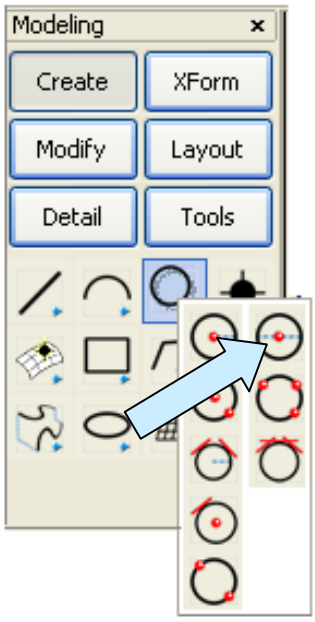
Then, click on the KEYIN Option and hit the ENTER Key three times.

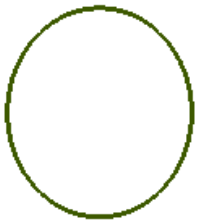
This places the rectangle with the bottom, left corner positioned at the origin of the file.

Next, click on the CREATE CIRCLE DIAMETER Icon.

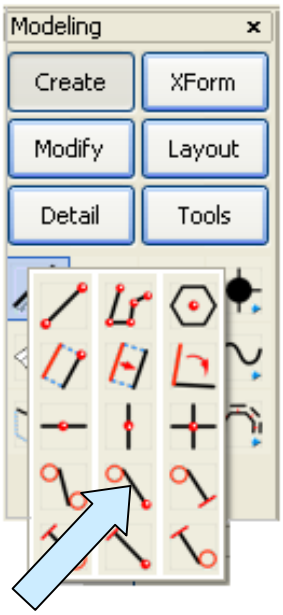
Type 1 for the Diameter.

Using the KEYIN Option, type -.25 for the XC value, 2 for the YC value, and 0 for the ZC value.





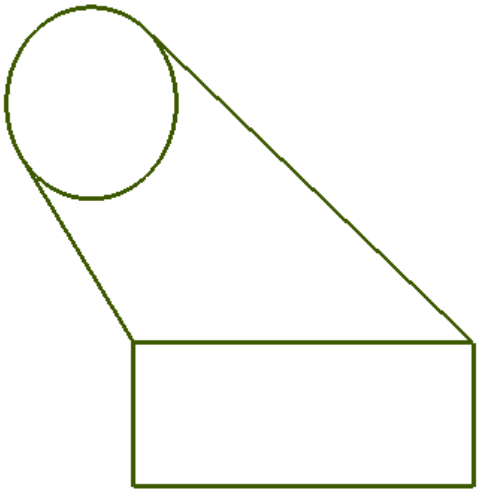
Your screen should look like this:



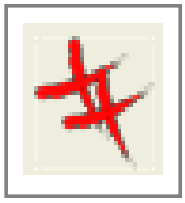
Next, click on the CREATE LINE TANGENT THRU POSITION Icon.

Click on the circle near the 8 O'Clock position. Then, using the EndEnt Option, click on the top, left corner of the rectangle.

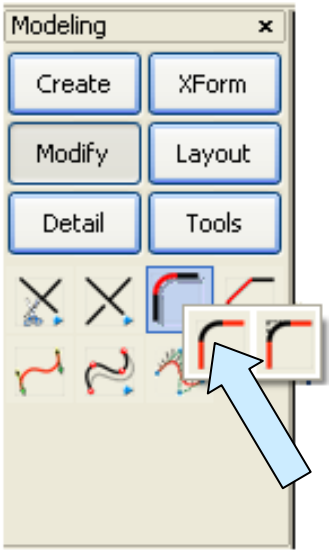
Click on the BACKUP Button. Next, click on the circle near the 2 O'Clock position. Then, using the EndEnt Option, click on the top, right corner of the rectangle.



Your construction should now look like this:



Click on the DELETE MULTIPLE Icon. Now, click on the top edge of the rectangle and then on the ACCEPT Button.

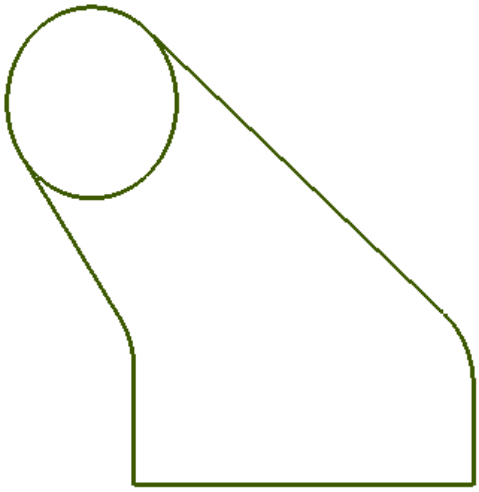


Next, click on the FILLET WITH TRIM Icon.

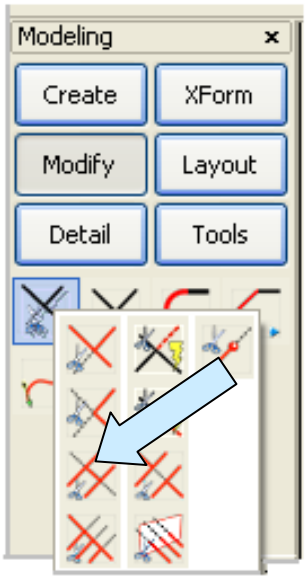
Type 0.5 for the Radius.

Now, click on the left, oblique line and then on the left, vertical line.

Next, click on the right, oblique line and then on the right, vertical line.



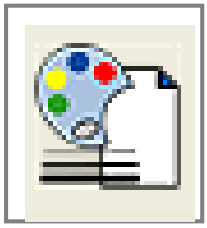
Your construction should now look like this:



Next, click on the TRIM DOUBLE Icon.

Click on the top, left portion of the circle.

Then click on each of the oblique lines. This eliminates the lower, right portion of the circle.



Now, click on the SET ATTRIBUTE Icon.

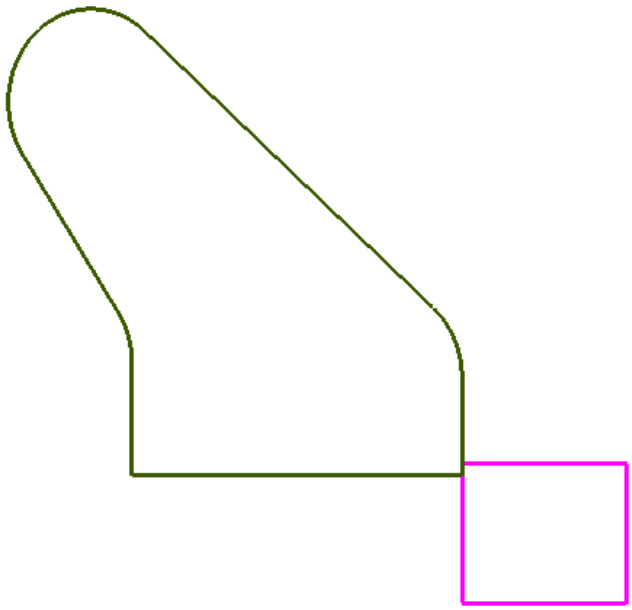
A Dialog Box appears. Select a new construction color. (I'm going to use MAGENTA.) Then, click on the OK Button.

Now, click on the CREATE RECTANGLE WIDTH HEIGHT Icon.

Use the TopLeft anchoring option.
Type 1 for the dXC value and 0.75 for the dYC value.

Click on the AlongE Option.
Click on the bottom end of the right, vertical line.

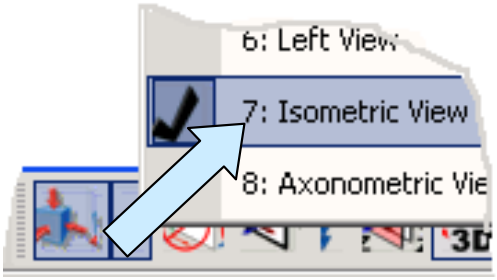
You are prompted to enter a distance. Type "t" and hit the ENTER Key. Notice that the 0.063 value that you entered earlier appears in the entry field. Hit the ENTER Key to accept the value.



Your screen should look like this:

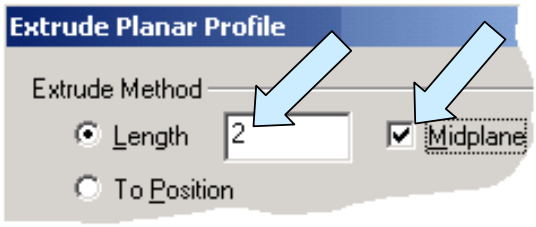
Notice that the top edge of the new magenta rectangle lies exactly 0.063 inches above the bottom edge of the green profile.

Now, switch to the Isometric View.



Click on the EXTRUDE Icon.

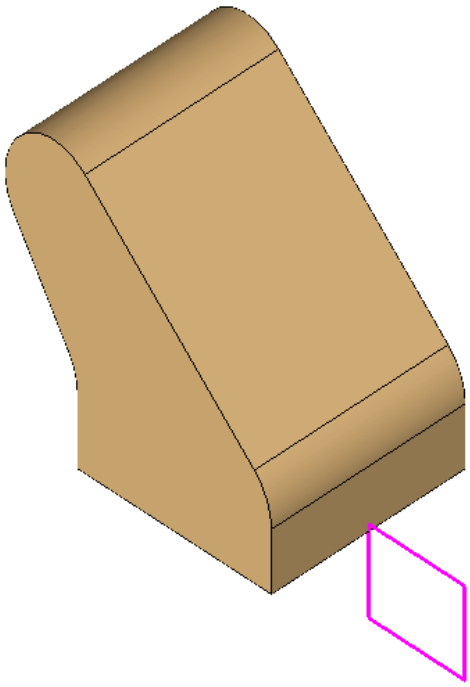
A Dialog Box appears.



Click on the Midplane Option. Type 2 for the Length and click on the OK Button.

Click on the ALL DSP Option and then on the BY TYPE Option.

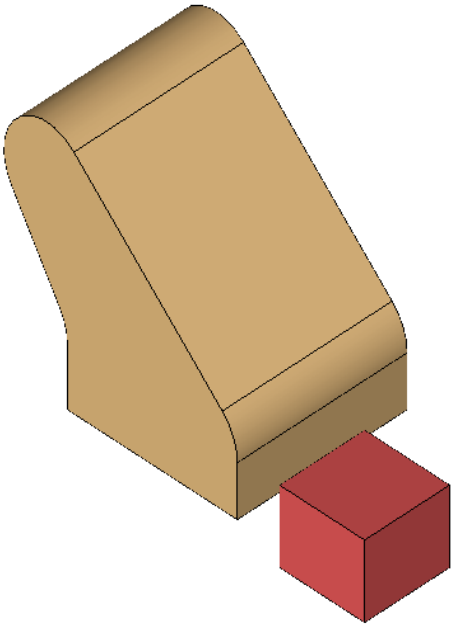
A Dialog Box appears. Click on the GREEN Option and then on the OK Button.



Your construction should now look like this: (Note: I've changed the color of the green part to gold for a better image.)

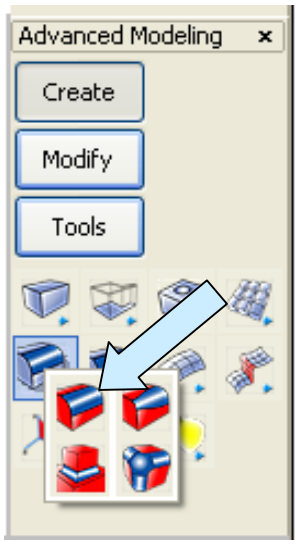
Let's use the EXTRUDE Function again.

This time, type 1 for the Length and use the Midplane Option. Click on the OK Button.



Click on the ALL DSP Option and then on the BY TYPE Option. Now, click on the MAGENTA Option and then on the OK Button.

Your screen should look like this:

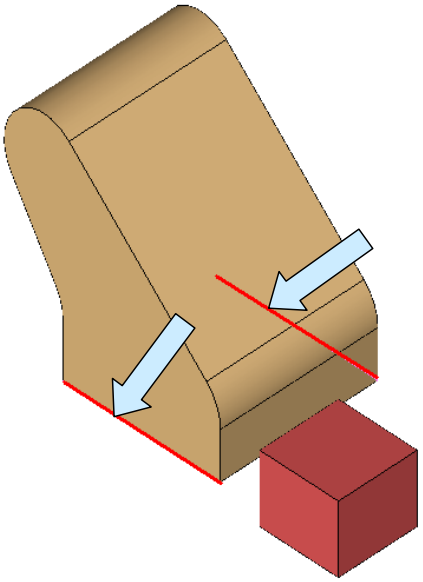


Next, click on the BLEND CONSTANT RADIUS Icon. A Dialog Box appears.

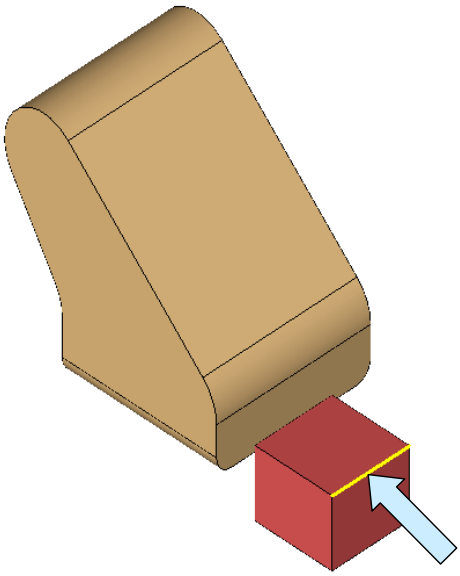
Now, remember that we entered a value of 0.063 for "t" which represents the thickness of the material.

We want the inside bend radius on the sheet metal bends to be two times the material thickness. This means that the outside radius on these bends would be three times the material thickness. So type in the Radius Field "3*t" and hit the ENTER Key.

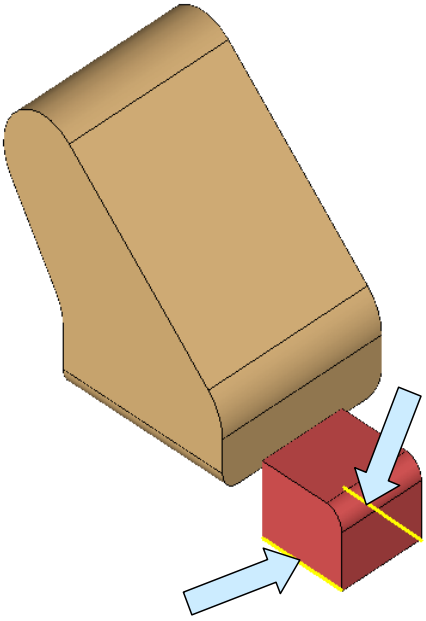
You'll see that a value of 0.189 appears. Click on the OK Button.



Now, click on the bottom two edges of the larger part indicated by arrows in the illustration to the left. Then, click on the ACCEPT Button.

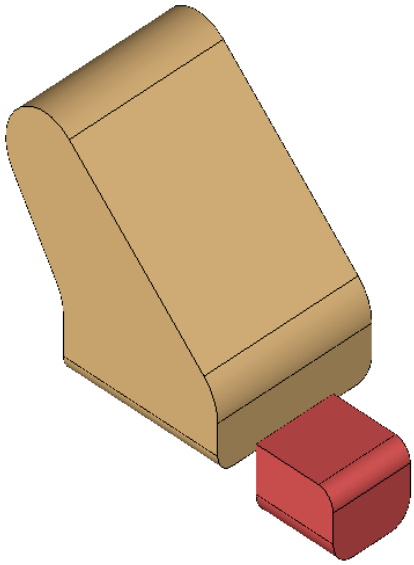


Next, click on the top right edge of the small solid indicated by the arrow in the illustration to the right. Then, click on the ACCEPT Button.

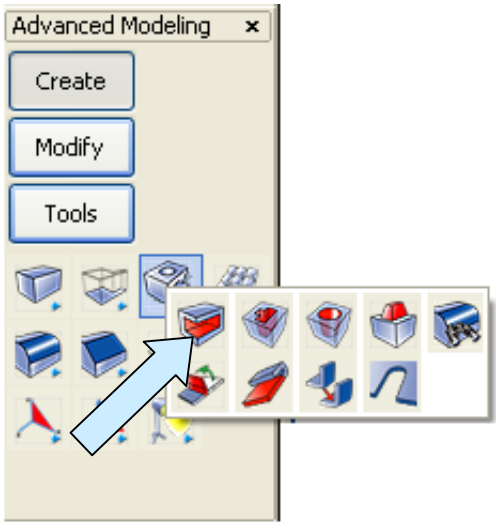


Click on the BACKUP Button. Type 0.25 for the Radius and then click on the OK Button.

Now, click on the two lower edges of the small solid indicated by arrows in the illustration to the left and then on the ACCEPT Button.



Your screen should now look like this:



Next, click on the SHELL Icon.

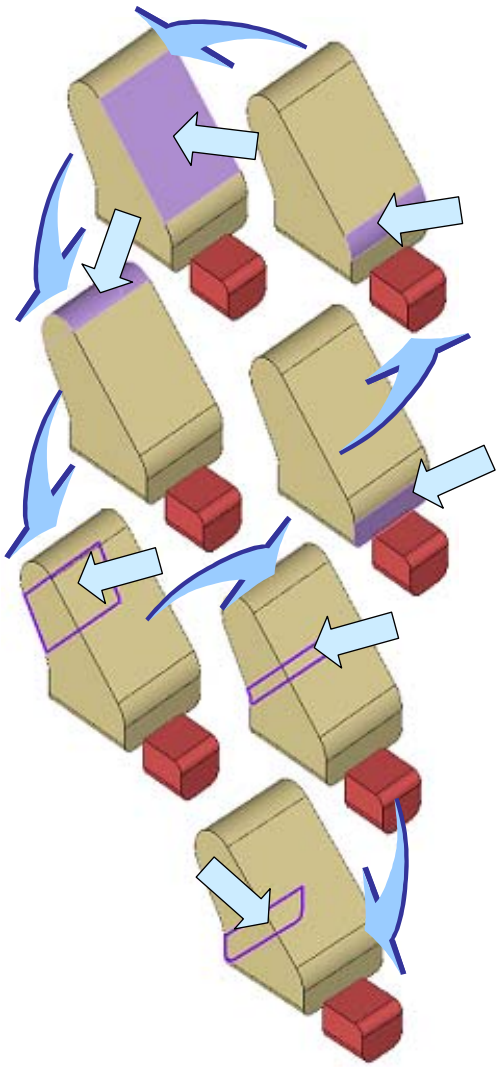
A Dialog Box appears. Type “t” for the Shell Thickness and click on the OK Button.

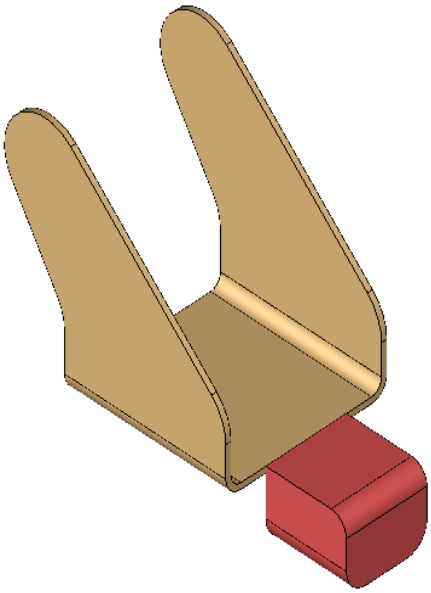
The predefined 0.063 value appears. Click again on the OK Button to confirm the value.

Now, let’s use the Fast Shell approach to make each of the portions of our desired part.

Select the faces on the larger part indicated by arrows in the illustration to the right. (Use the SPACE BAR or TAB Key to access the hidden faces on the left side of the part.)

When you have selected all seven faces, click on the ACCEPT Button.

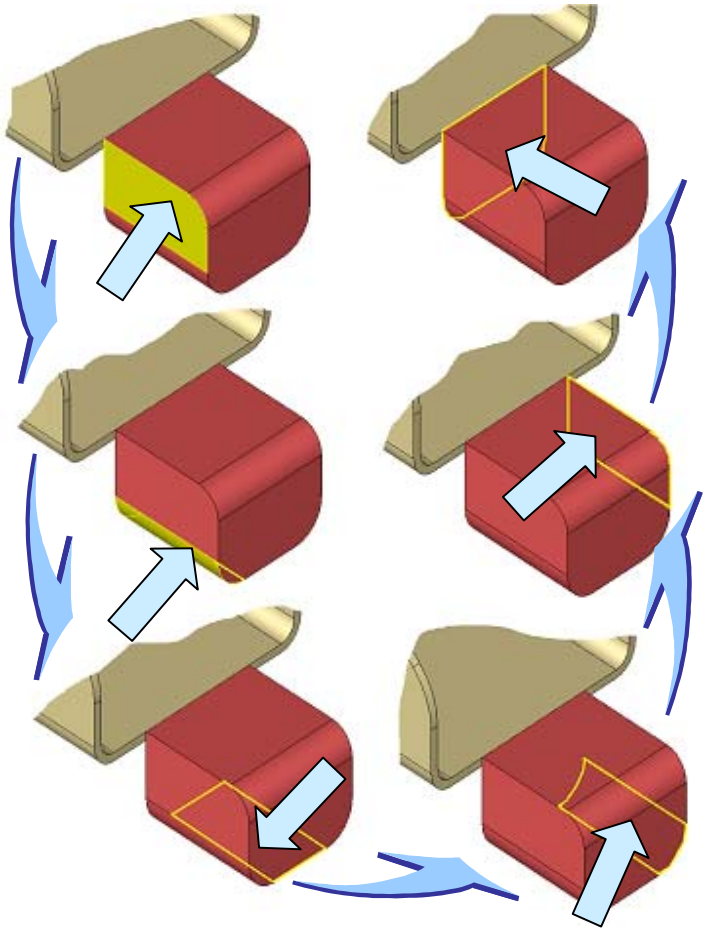




Your large part should now look like this:

Let's continue using the SHELL Function to complete the smaller part.

On this part, the top face, the top, right blend surface, and the right face don't change as we convert from the dense part to the sheet metal part. Therefore, we don not want to select these surfaces.



Click on the front face.

Now, click on the bottom, front blend surface.

Next, click on the bottom surface. (This is hidden, so you have to use the SPACE BAR or TAB Key.)

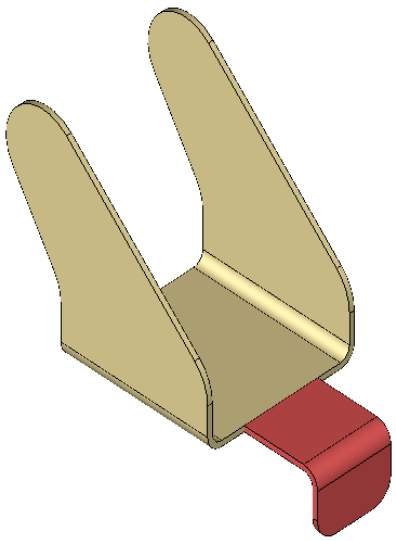
Then, click on the bottom, rear blend surface. (Also hidden, so use the SPACE BAR or TAB Key.)

Next, click on the rear surface. (Also hidden, so use the SPACE BAR or TAB Key.)

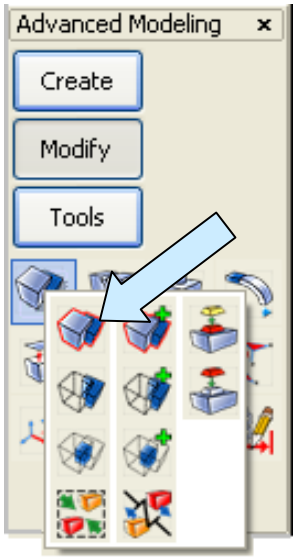
Then click on the left surface. (Also hidden, so use the SPACE BAR or TAB Key.)

When you have selected the faces, click on the ACCEPT Button.

Your screen should now look like this:



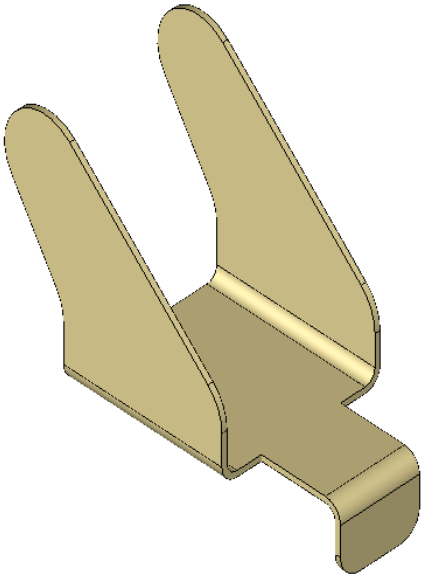
Now that we have the two pieces of our elephant built, let's merge them into one animal!



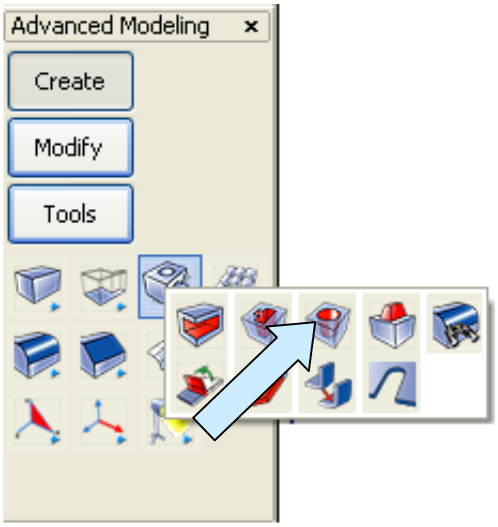
To do this, click on the BOOLEAN UNION Icon.

Click on the green part and then on the magenta part.

Click on the ACCEPT Button to confirm the selection.



You will now have one solid on the screen.



Our final task is to create the holes in the part.

Click on the DRILL Icon.

A Dialog Box appears.
We'll use the Through Hole Option.

Type 0.5 for the Diameter and click on the OK Button.

Position the cursor over the broad, front face of the part and click on it when it highlights.

Then, using the CtrMid Option, click on the radiused edge at the top of the face.

This creates two matching 0.5 inch diameter holes in the upright legs of the part.

Next, click on the BACKUP Button twice. This returns you to the Dialog Box.

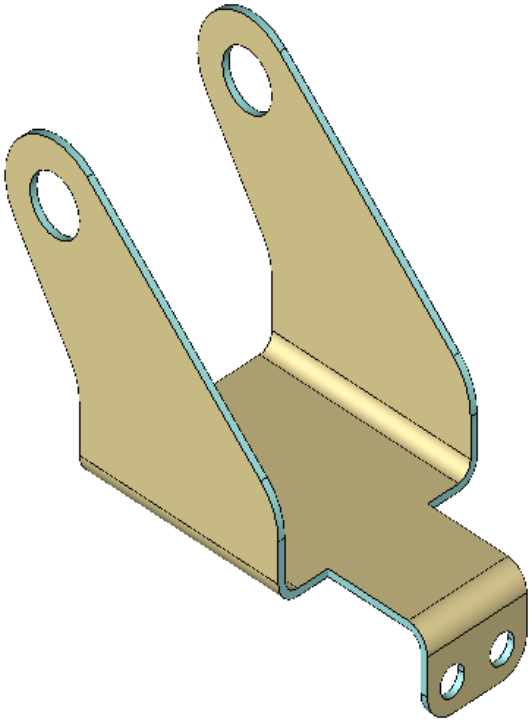
Type 0.25 for the Diameter and click on the OK Button.

Now, move the cursor over the right side face of the “L-shaped” flange on the right end of the part.

Click on this face when it highlights.

Using the CtrMid Option, click on the radiused edge at the front end of this face and then on the radiused edge at the back end of this face.

Your finished part should look like this:

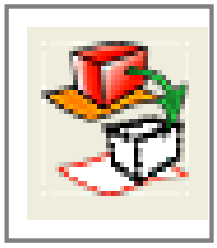


Unfolding the Sheet Metal Part

Most sheet metal part designers need to create a flat development of a formed sheet metal part for manufacturing purposes.

So let’s take a moment to do that with this part.

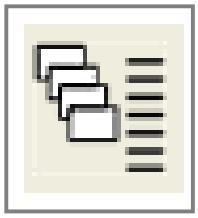
I like to create a duplicate of my original part that I’ll leave in the finished form. I then unfold the original part and place the two parts on unique levels. I can then quickly view either version in model space. When creating an associated layout drawing, I can also call up either the flat development or the finished part in any instance by manipulating the levels.



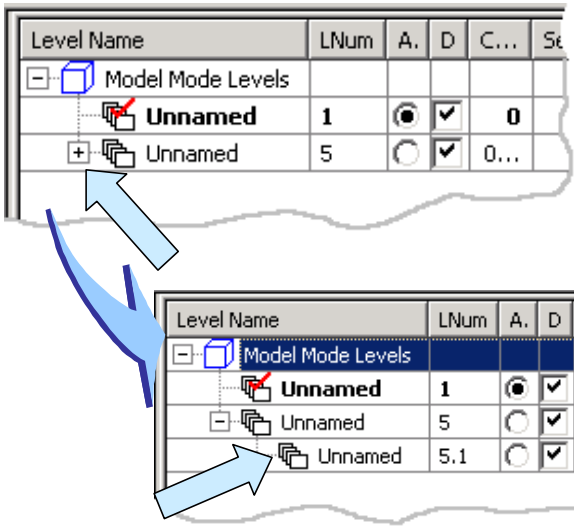
First, we’ll click on the MOVE ENTITIES Icon. (If you don’t have this Icon on one of your toolbars, you’ll want to add it. You’ll find the function in the VIEW Pulldown Menu under the LEVELS Option.)

Click on the SELECT Option.
Click on the part and then on the ACCEPT Button.

You are prompted to indicate a destination level. Type 5.1 in the entry field.

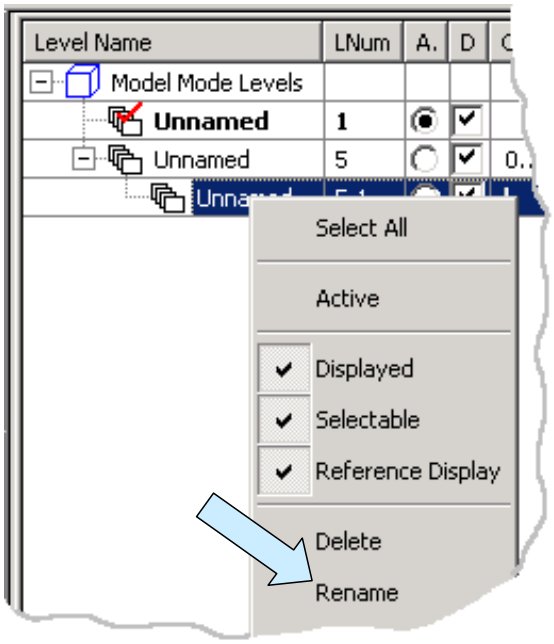


Now, click on the TOGGLE SPLITTER Icon.



Notice that a level 5 has been added to the level list with a "+" sign in front of it.

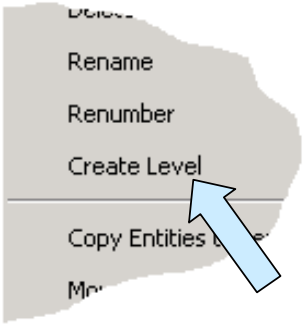
Click on the "+" sign to expand the level and you will see the 5.1 level appear. You can click on the checkmark in the Display column for this level at any time to remove the part from the display.



RIGHT MOUSE CLICK on the 5.1 level row.

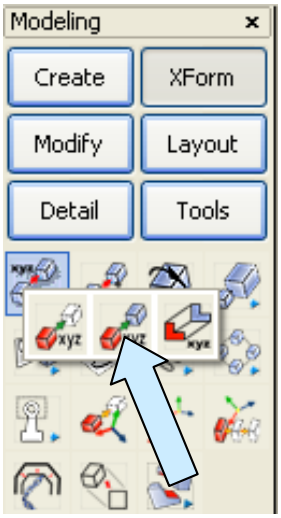
Then, click on the Rename Option in the Menu that appears.

Type "Finished Part" for the new name and hit the ENTER Key.



Next, RIGHT MOUSE CLICK on the Level 5 row in the level list.

Then, click on the Create Level Option in the Menu that appears. Notice that a level 5.2 appears and it is now checked as active.



Now, click on the XFORM DELTA COPY Icon. Click on the part and then on the ACCEPT Button.

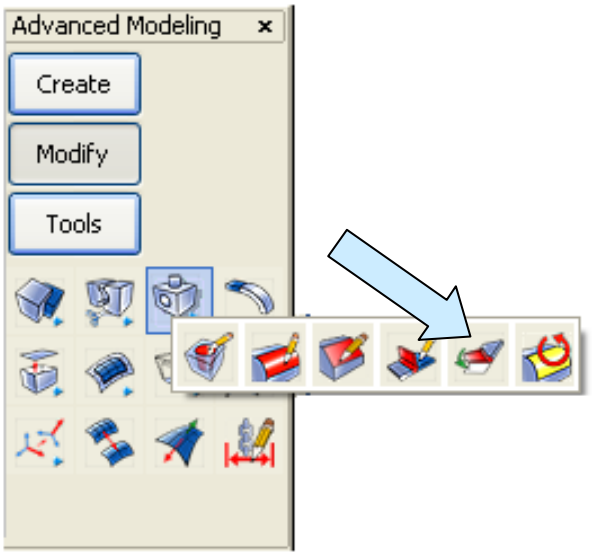
Type 1 for the Number of Copies. Hit the ENTER Key three times and then hit the ESC Key.

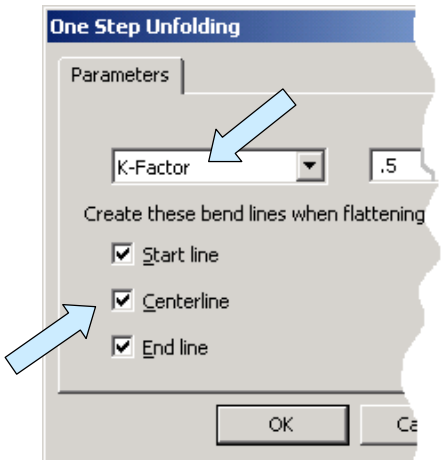
This creates a copy of the part in the same location as the original part.

Now, click on the checkmark on level 5.1 in the Display Column to remove the original part from the display. (Nothing appears to happen on the display since the copied part occupies the same space!)

Now, click on the UNBEND ALL Icon. A Dialog Box appears.

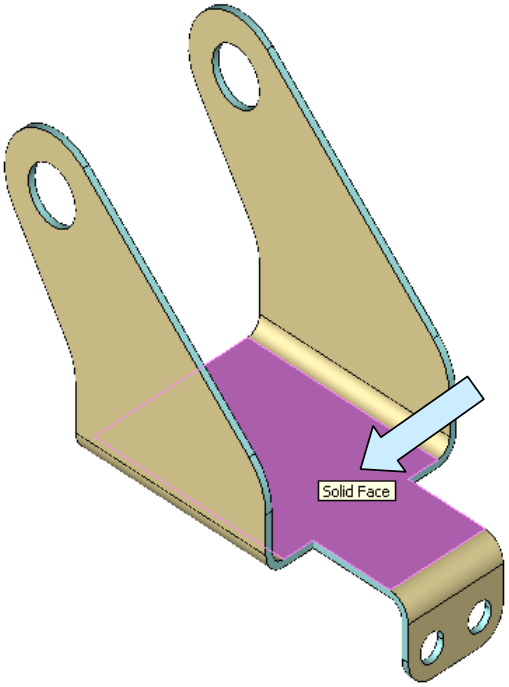
For this exercise, we'll use the K-Factor Option and the default value of 0.5. (Depending on the material and particular bend that you are making, you would adjust the value of the K-Factor to allow for the right amount of extra material needed in the blank to get the final finished part dimensions.)





We'll check all of the line options.

Click on the OK Button.



You are prompted to select a face to unfold from.

Click on the horizontal top face of the part indicated by the arrow in the illustration to the left.

Now, watch as the part automatically unfolds to a full flat development.

Because the flat development and the original folded part are on different levels, you can display either one by manipulating the level list.