

Now, click on the SHELL Icon.

A Dialog Box appears.

You are prompted to enter a Shell Thickness.

Remember that we started this exercise by assigning a value of 0.032 to the variable "t" in the Online Calculator.

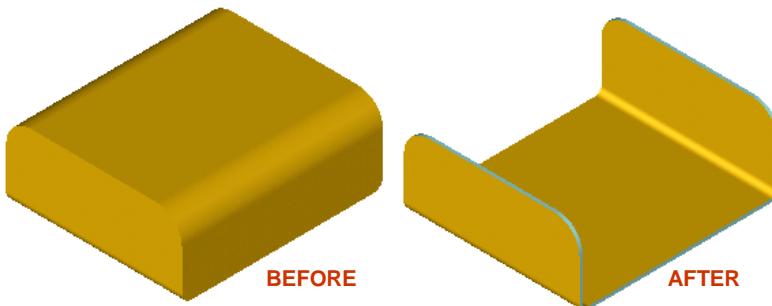
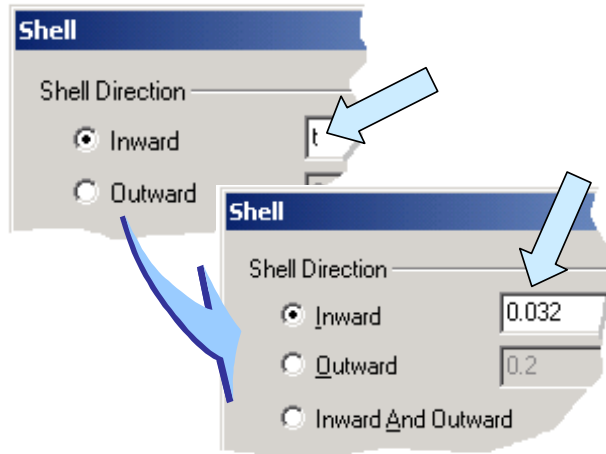
We just used the expression $3*t$ in our Blending Operation.

For the Shelling Operation, type "t" in the Shell Thickness Field and hit the ENTER Key.

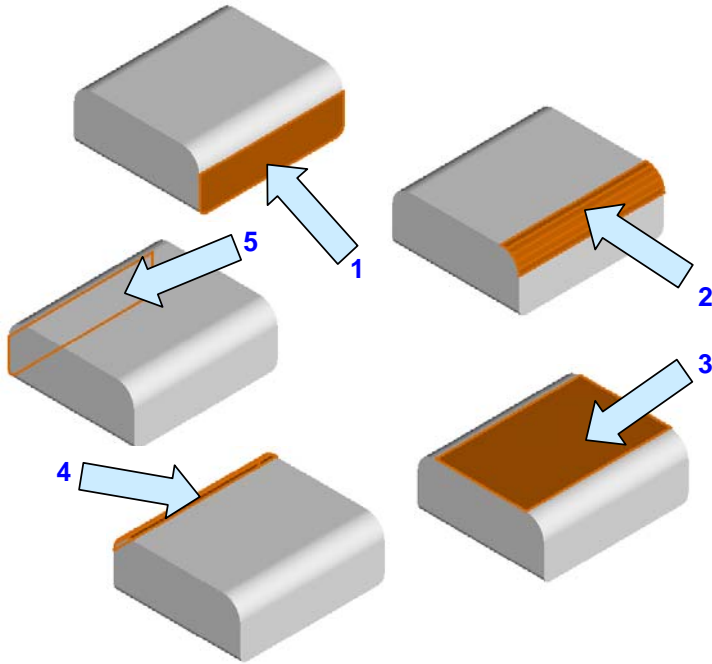
Notice that the value 0.032 is echoed back.

Click on the OK Button.

You are prompted to select the faces to be opened. Now, let's work on the larger block first.



Remember, we only want to select faces that will change on the part. Look at the primitive shape to the left and then at the desired final shape. You can clearly see the faces that will change.

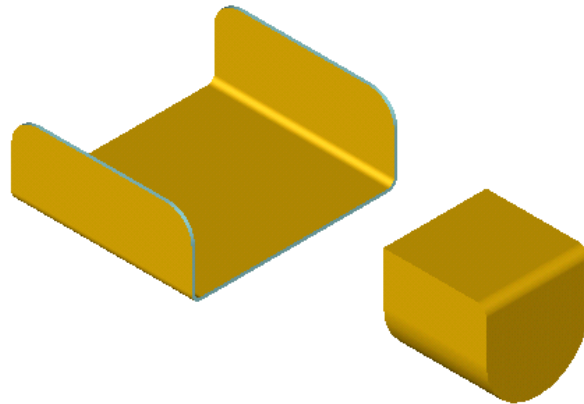


Click on each of the faces. I've indicated the picks with numbered arrows in the illustration to the left.

To access the hidden, left face, move the cursor into the area of the face and touch the TAB Key to index the Face Selector to the face.

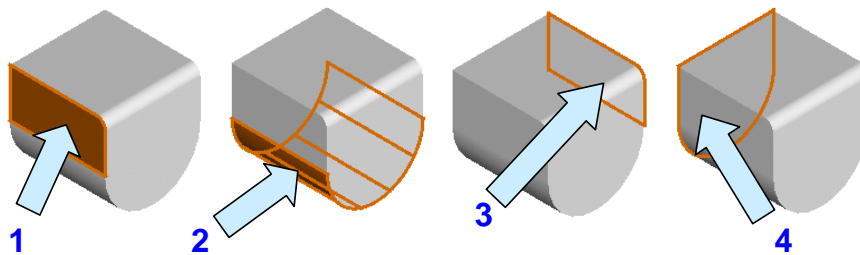
Then, click when the face is highlighted. When all of the faces are selected, click on the ACCEPT Button.

Your screen should now look like this:



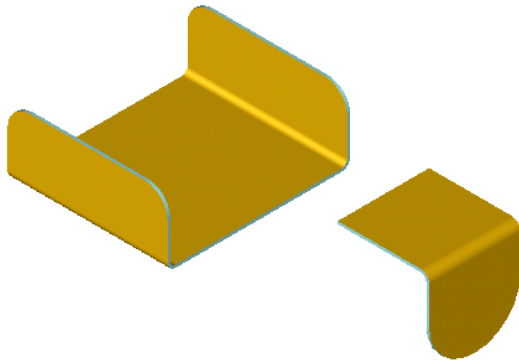
Now, we're going to continue using the SHELL Function to modify the smaller block.

Select the four faces indicated in the illustration below:



Use the TAB Key to reach the hidden faces.

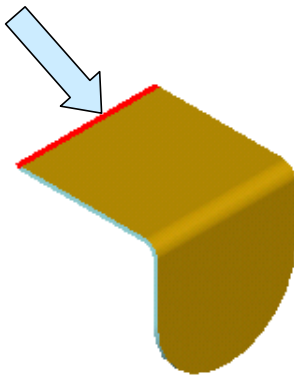
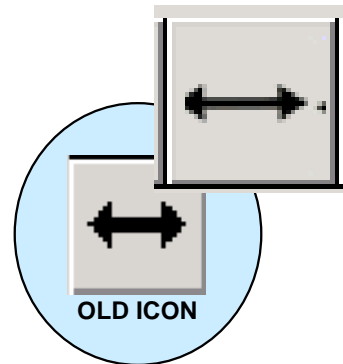
Click on the ACCEPT Button at the end.



Your screen should now look like this:

Our next task is to position the small, “L-shaped” solid with respect to the larger, “U-shaped” solid.

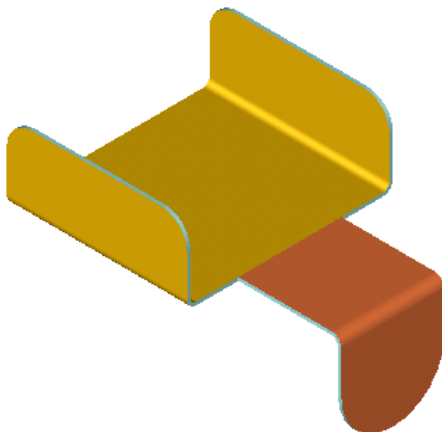
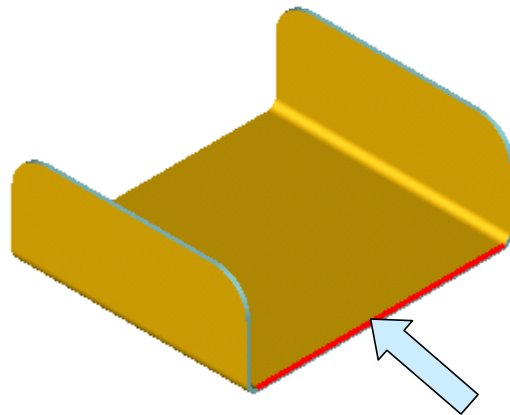
Click on the GENERIC MOVE Icon.
Now, click on the small solid.



You are prompted to select a Base Position.
Using the CtrMid Option, click on the top, left edge of the small solid.

Now, as you move your cursor, the small solid follows.

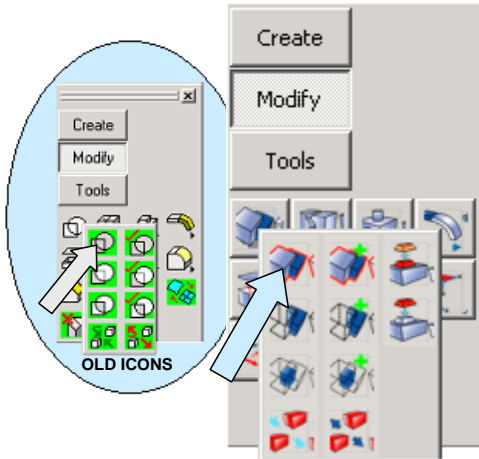
Using the CtrMid Option, click on the top, right edge of the “U-shaped” solid to complete the move.



Your screen should now look like this:

(Note: I’ve changed the color of the “L-shaped” part to emphasize the fact that you still have two separate solid objects.)

We need to combine them into one solid.



To do this, click on the BOOLEAN UNION Icon.

Click on the “U-shaped” solid and then on the “L-shaped” solid.

Then, click on the ACCEPT Button.

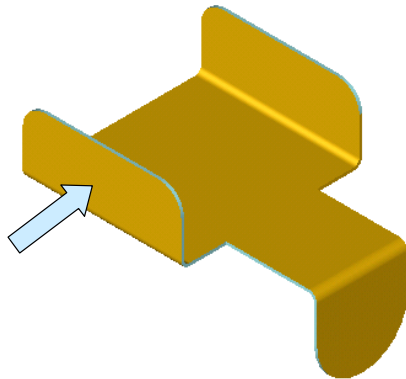
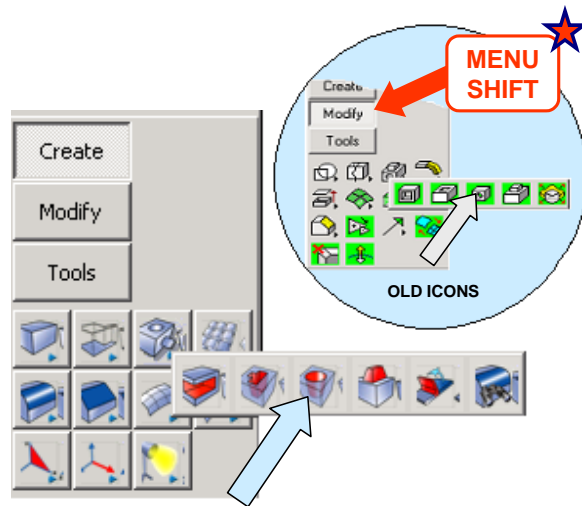
You now have one solid object.

Next, let’s create the holes in the part.

Click on the DRILL Icon.
A Dialog Box appears.

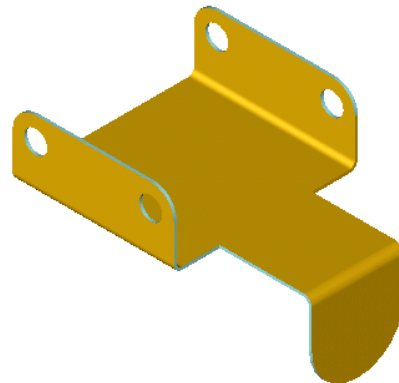
We’ll use the Through Hole Option.
Type 0.25 for the Hole Diameter.

Then, click on the OK Button.



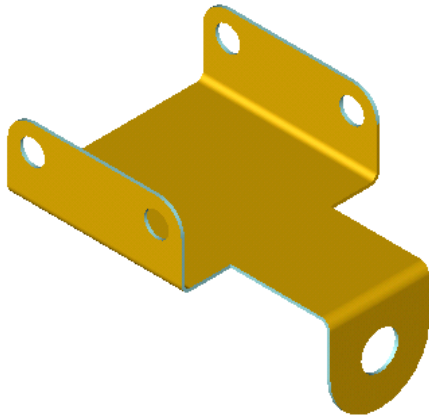
Touch on the front face of the front flange on the “U-shape.” Then, using the CtrMid Option, click on the top, left radius of the flange and then on the top, right radius of the flange. (You can select either the front edge or rear edge radius. You can even position by selecting the radii on the rear flange!)

Your part should now look like this:



Click on the BACKUP Button. Type 0.375 for the Diameter and then click on the OK Button.

Click on the right face of the “L-shaped” section.
Then, using the CtrMid Option, click on the circular edge of this face to position the hole.



Your part should now look like this:

Our last task is to create the rectangular cutout in the broad face of the “U-shaped” section.

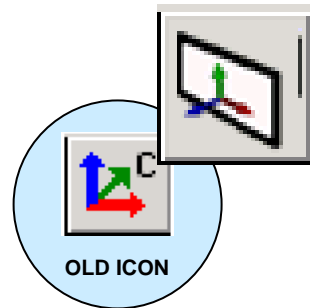
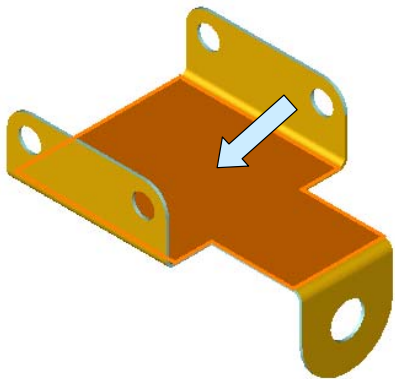
We are going to create a rectangular cutout with radiused corners that is centered on the face.

Now I typically see users undertake an elaborate construction ritual to construct this type of feature. Centerlines are constructed, parallel lines are then created offset from these lines, with fillets and trims proliferating until the final cutout geometry is developed!

Now I’ve got to tell you that they don’t hand out medals for the most number of mouse clicks! There is an extremely easy way to position a rectangular cutout at any time on any surface of a solid! Let’s use this technique on the part in question.

The rectangular cutout will be 1 inch by 0.75 inches with 0.125 corner fillets.

First, let’s click on the Construction Plane (Cplane) Icon.



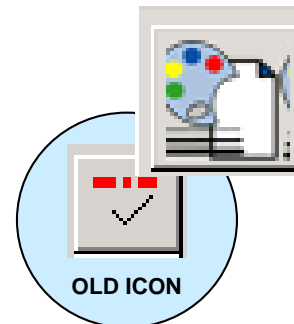
Move the cursor over the broad face of the part and click on it when it highlights.

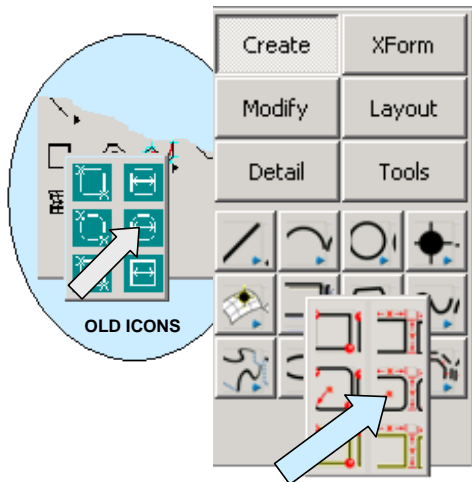
You will now have a Construction Plane established on this face.

Next, click on the SET ATTRIBUTE Icon.

Select a different color than that used for your model. (I’ll use CYAN.)

Hint: It is always a good idea to use a different color for things like cutout profiles. When you have to select the cutout profile during the operation, you can do so by using the BY TYPE Option and then simply picking on the unique color of the profile geometry.





This is an example of good entity management. With a simple cutout like a sharp cornered rectangle you are tempted to ignore this, since you just have to pick four lines manually. As the cutout geometry becomes more complex, the task of selecting increases in complexity also!

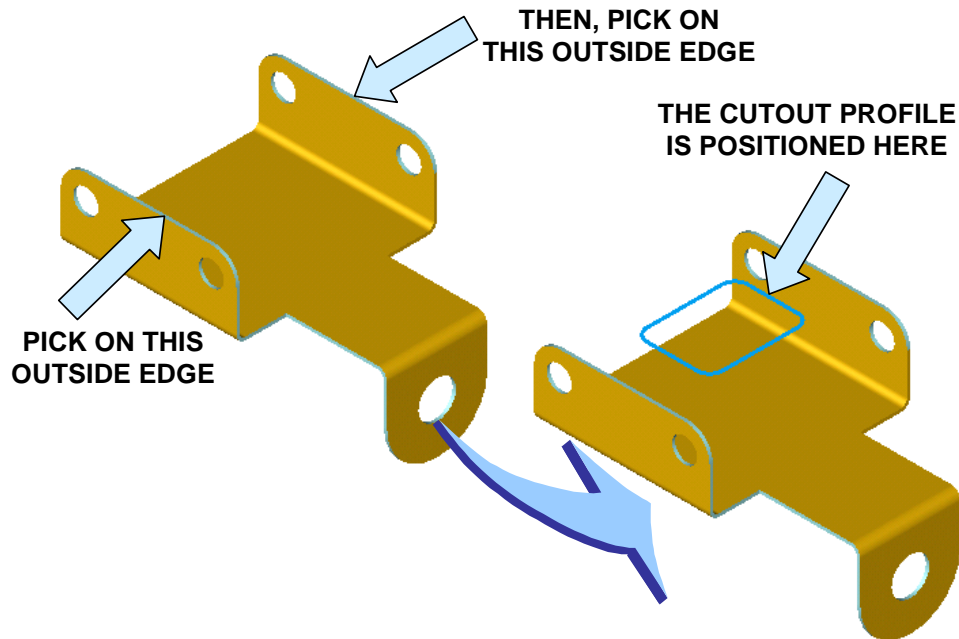
Now, click on the CREATE ROUNDED RECTANGLE WIDTH/HEIGHT Icon.

Type 0.125 for the Corner Radius.
Click on the MidCtr Anchoring Option.

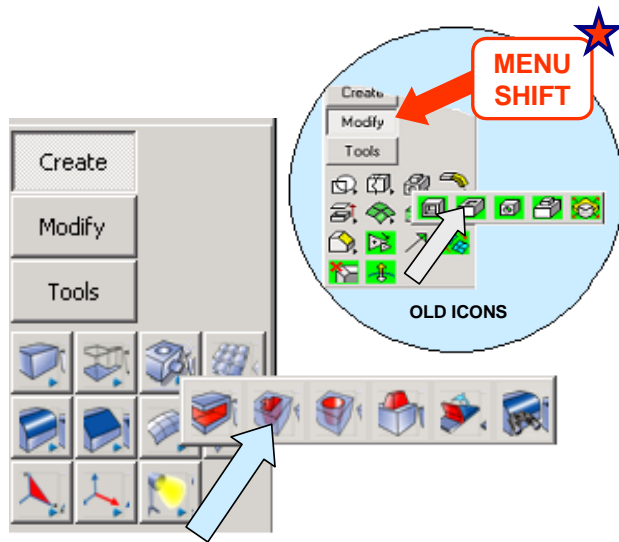
Type 0.75 for the dXC value.
Type 1 for the dYC value.

Now, click on the TwoPos Option on the Conversation Bar.

This powerful positioning feature gives you the ability to position things at a point that lies at the exact center of two selected positions.

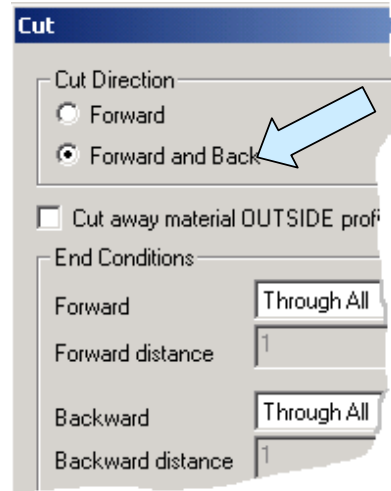


For our model, we'll use the CtrMid Option and then proceed to select the top, outside edge of the front flange and then the top, outside edge of the rear flange.



Now, click on the CUT AWAY Icon.

A Dialog Box appears.



Click on the Forward and Back Option.

We'll use the default Through All Options.
Click on the OK Button.

Now, click on the solid Body.

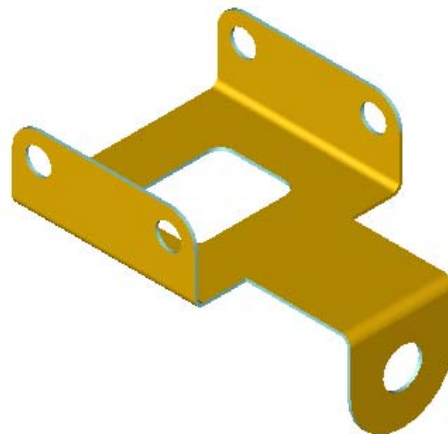
You are then prompted to select the cutting profile. Since we constructed this in a different color, simply click on the ALL DSP Option.

Then, click on the BY TYPE Option.

The Masking By Type Dialog Box appears.

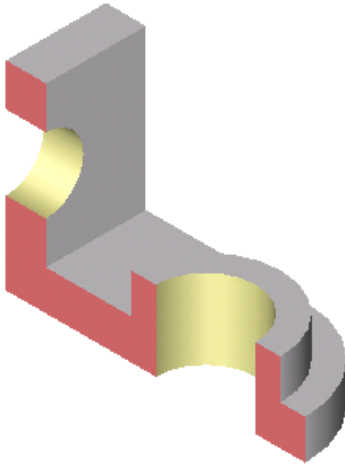
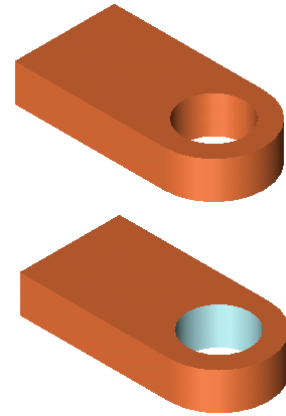
Click on the color of your profile and then on the OK Button. Now, click on the ACCEPT Button.

Your completed part should look like this:



Enhancing Edges and Cores

Throughout this book, you'll see many examples of enhanced hole geometry. We've talked about using the FACE COLOR Tool to change the color of a hole surface so that the hole contrasts against the main part color. This is illustrated to the right.



We've also seen how using a different face color on a solid section accentuates the cross-sectional shape of the part. Since documentation should communicate concepts as clearly as possible, this is an obvious tool that you want to use to advantage.

You might have noticed that I've enhanced the thin edges on our current sheet metal part to enhance the images.

Now while this technique is dramatic, in the old days it was tedious since there were one heck of a lot of faces to select.

One of the new selection tools in KEYCREATOR 5 now makes this a breeze!

Let's walk you through the procedure so that you can see for yourself how easy it is to do!

