KEYCREATOR®

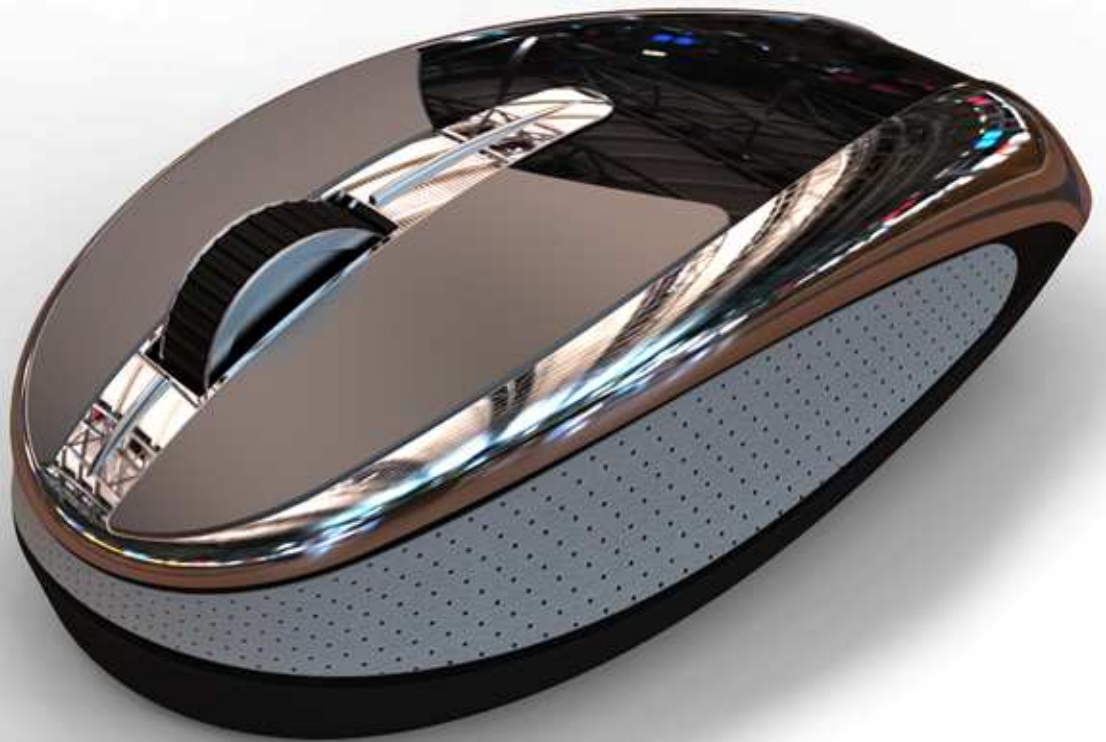# Doctor Walt's™

Tips And Tricks

KUBOTEK®
FREEDOM AND PRECISION

Quick Guide to Keycreator's Sheet Metal Design Tools- Part 1

Many users are not aware of the sheet metal design capabilities in Keycreator. The various tools are located in several different menu locations instead of in one common palette, so I recommend that you create a special sheet metal toolbar to simplify your work. If you are not familiar with creating a custom toolbar, I recommend that you use my Tips and Tricks exercise on creating hot keys and toolbars or one of my books like Doctor Walt's Keycreator Workbook.
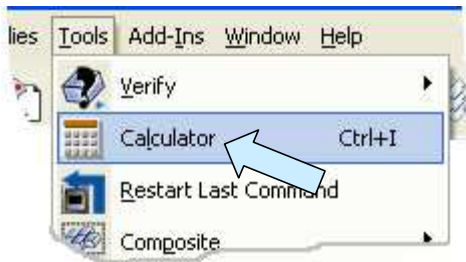
The first tool on the toolbar should be the ONLINE CALCULATOR. You can find this in the TOOLS Pulldown Menu.

Next, I like to place the PRIMITIVE BLOCK Icon, since you can start many sheet metal parts by making a basic plate or volume that can be later shelled out.
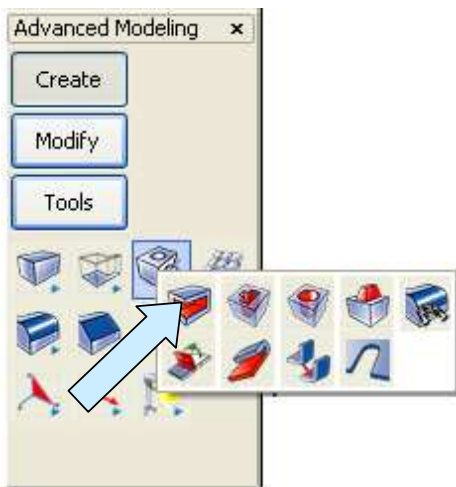
The next useful tool is the SHELL Icon. You'll see in a moment how this comes in useful for making sheet metal parts.

Now, drag the CUT SHEETMETAL CORNER Icon onto your toolbar.
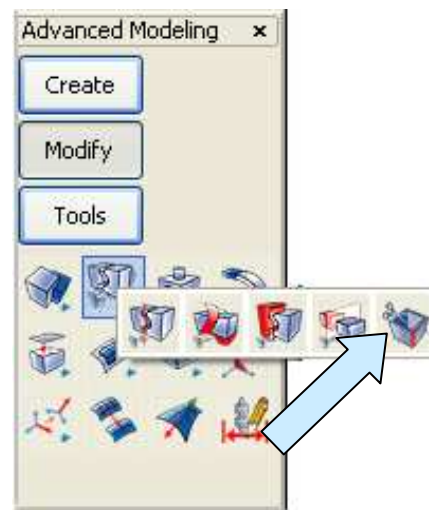
I place the four Icons labeled to the right next on my toolbar, starting with the AUTO BEND CORNER Tool.

Next I place the CREATE FLANGE Icon, then the ADD SHEETMETAL BEND and then the CREATE BEND RELIEF Icon.

CREATE BEND RELIEF

ADD SHEETMETAL BEND

CREATE FLANGE

AUTO BEND CORNER
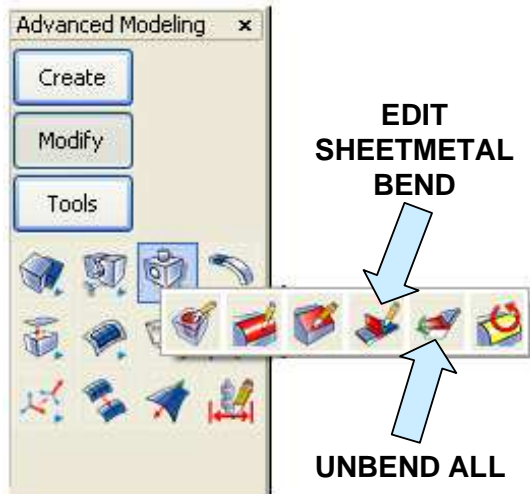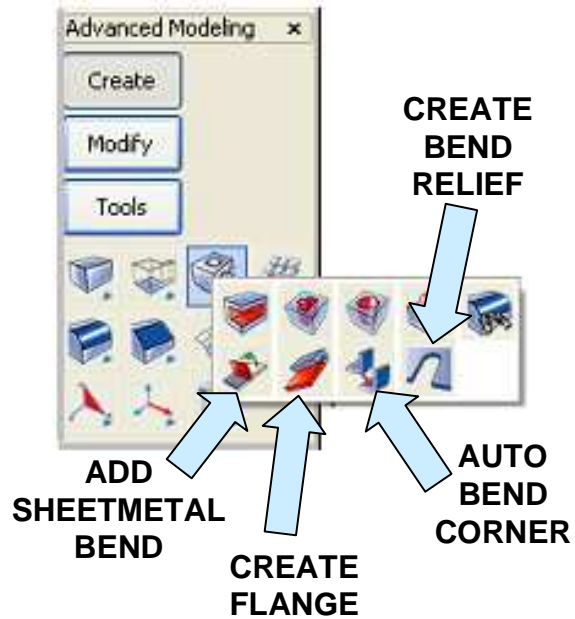
Next, drag the EDIT SHEETMETAL BEND and UNBEND ALL Icons onto the toolbar.

EDIT SHEETMETAL BEND

UNBEND ALL
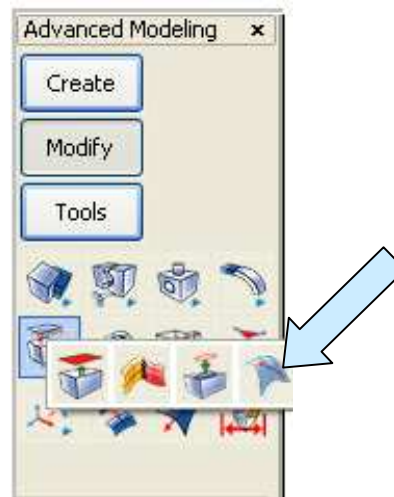
Let's also drag the SCRIBE FACE Icon onto the toolbar.
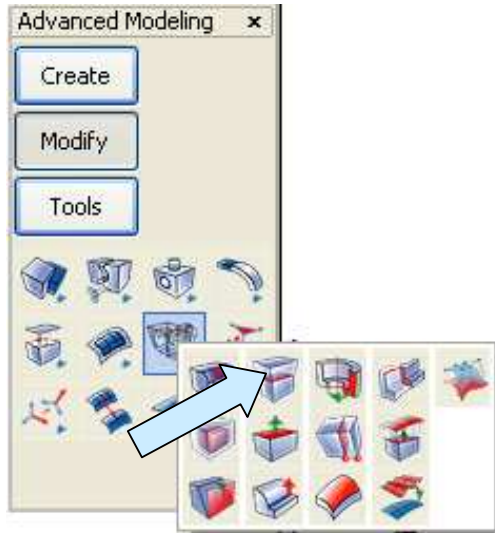
Finally, add the EXTRUDE FACE Icon to complete the set of tools.

Your final toolbar should look like the one illustrated below. You might want to reorient it as a vertical toolbar and dock it at the right side of your screen.

Now you can certainly create your own toolbar with a different collection of Icons. You might argue that functions like the EXTRUDE FACE and ONLINE CALCULATOR don't belong on this toolbar since they are not purely specifically sheet metal tools. My purpose is to give you all the tools you need to do teetotal design work all in one easy to find place. This toolbar is available for download as part of this exercise.

Let's see how this works with a few simple examples.

We'll start with the simple teetotal case shown to the right. The finished part is available as design file "metalcase01."
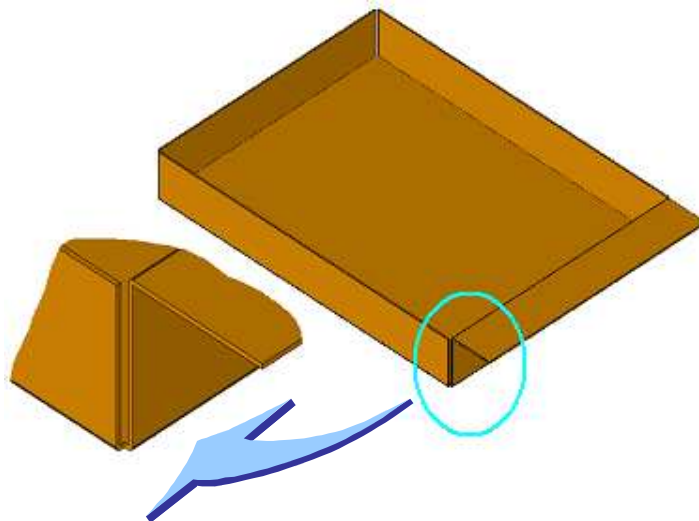
Start with a new file in view 1. Click on the ONLINE CALCULATOR Icon and type "t = 0.032" to assign a value of 0.032 as the sheet metal thickness in this file.
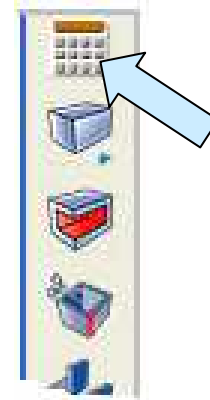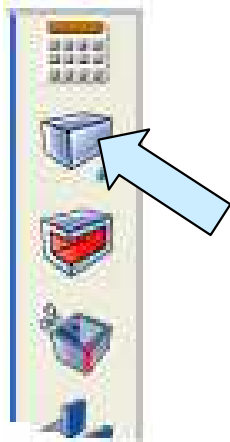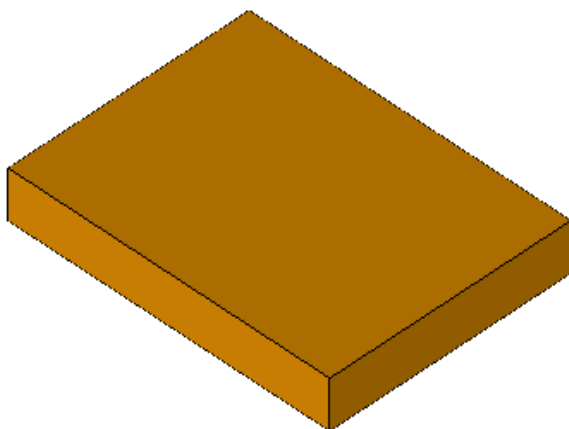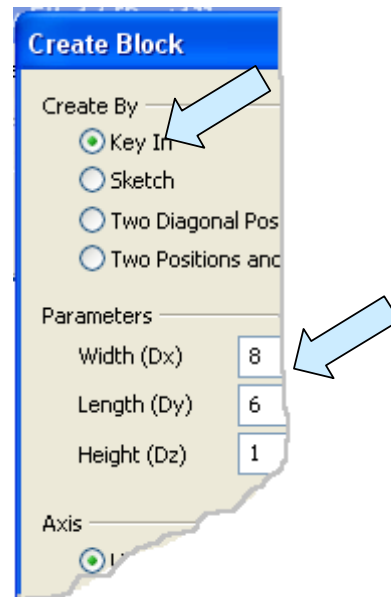
Next, click on the BLOCK Icon.

Create Block

Create By
- ◉ Key In
- ○ Sketch
- ○ Two Diagonal Pos
- ○ Two Positions and

A Dialog Box appears.

Use the Key In Option and type 8 for the width, 6 for the length, and 1 for the height.

Parameters

Width (Dx) 8
Length (Dy) 6
Height (Dz) 1

Use the cursor option to place the file anywhere on the screen.
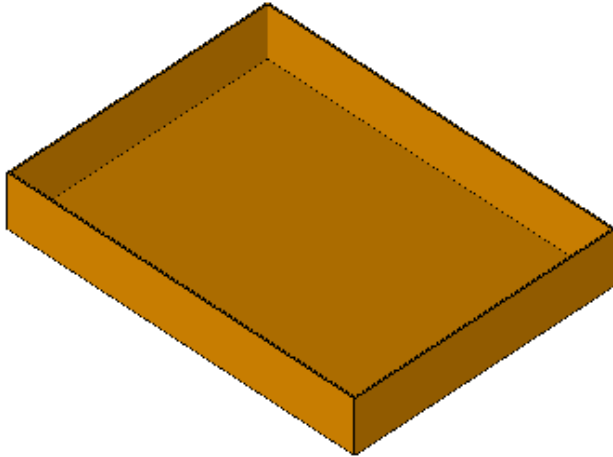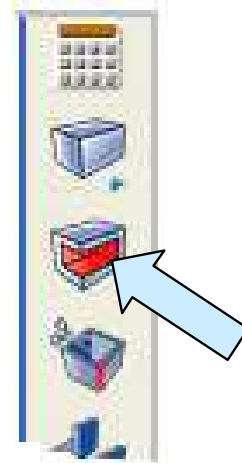
Axis
◉

Now, switch to the Isometric View and Autoscale the display.

Your part should look like this.

Next, click on the SHELL Icon.

When the Dialog Box appears, type "t" for the shell thickness. Use the Select Faces to be Opened Option and click on the OK Button.

Then, select the top surface of the block.

When the shelling operation is completed, your part should look like this:

Now what we have is fine as a conceptual model of the part. However, it does not represent a sheet metal fabricated part for two reasons.

First, the bends at the bottom edges are not sheet metal bends and second, the corners are closed as though it were molded from a piece of plastic.

I've illustrated what we want to achieve in the enlarged view to the right. (Note: In actual practice we use a much smaller corner rip. I've made this one large so you can see what is going on.)
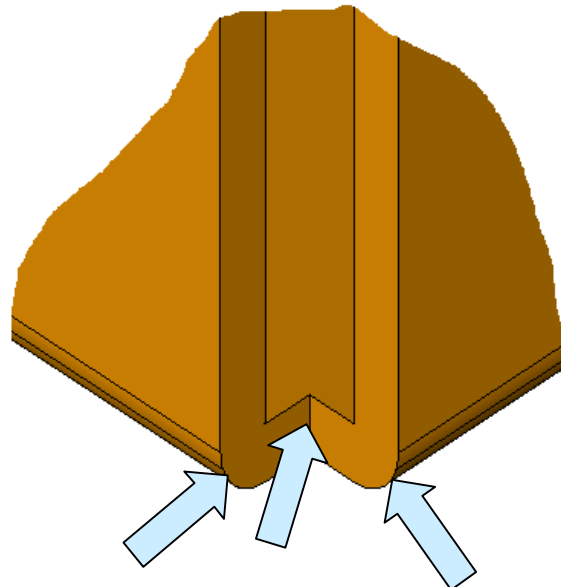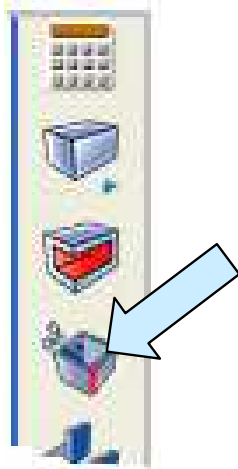
Let's rip the four vertical corners first. To do this, click on the CUT SHEETMETAL CORNER Icon.

A Dialog Box appears. Click on the Cut Away Both Sides Option.

Then, type "2 * t" in the Cut Width Field. Keycreator will automatically calculate 2 times the sheet metal thickness that you enter in step one of this exercise.

Cut Sheetmetal Corner

○ Cut one side

○ Cut away both sides (by diagonal cut wi

◉ Cut away both sides (by side cut width)

Cut Width   2*t
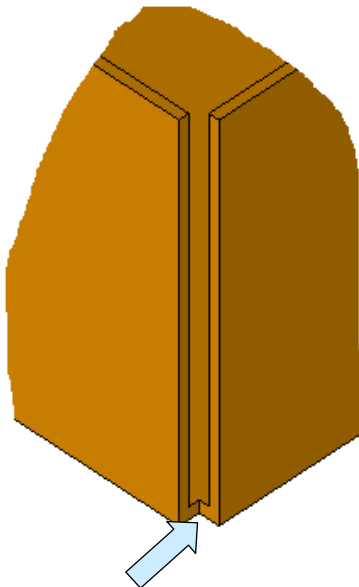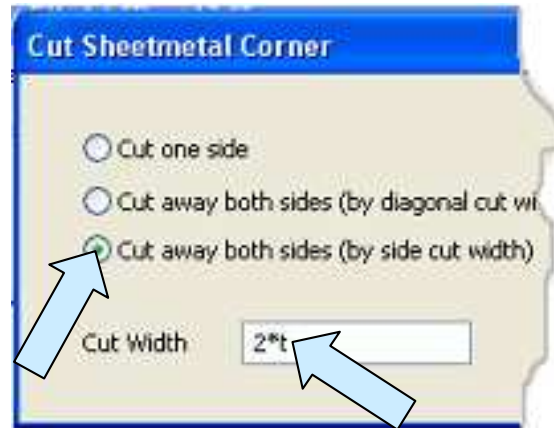
Now, click on each of the vertical corners. (Don't worry about whether you are picking the inner or outer corner. It doesn't matter!)

Your part will now have ripped corners that look like this:

Our next step is to fix the four bottom bends on the part. You'll notice that the inside and outside corners are both sharp. For Keycreator to handle unbending this part, these four corners must be replaced with true sheet metal corners.

Although many sheet metal designers work with nominal zero radius bends on the inside corners of parts, Keycreator requires just a small radius here to make everything work. We'll use 0.0001 for our exercise.
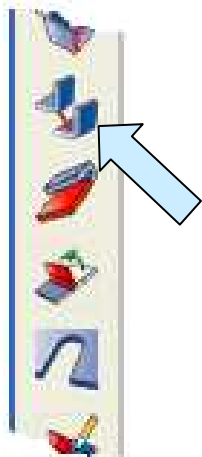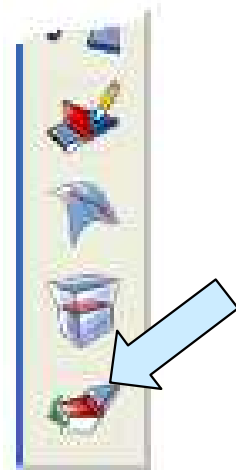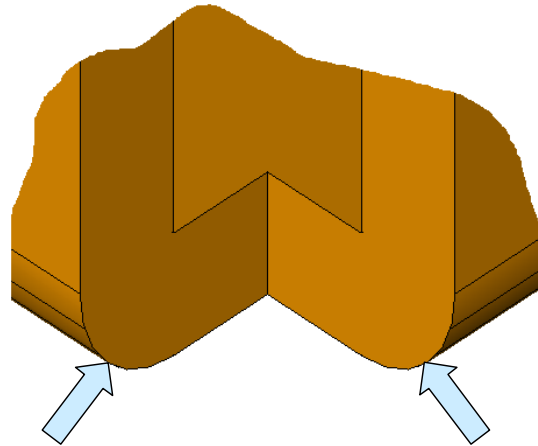
Click on the AUTO BEND CORNER Icon.

You are prompted for a radius value. Type 0.0001 and hit the ENTER Key.

Now, click on each of the bottom corners on the part. (Again, don't worry about inner versus outer edge. Keycreator is smart enough to know what you want!)
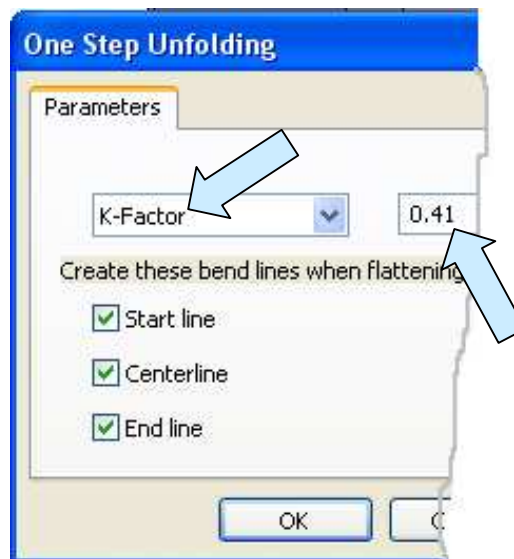
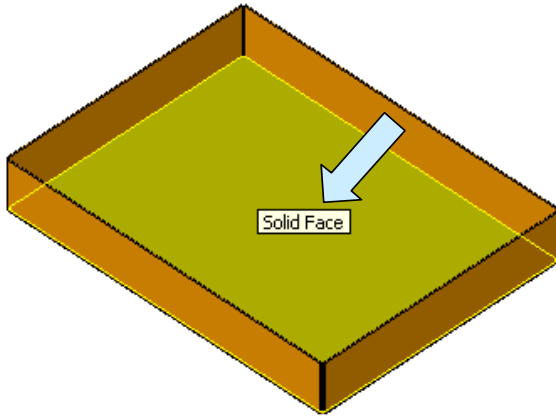The bottom of your part will now have true sheet metal bends.

Now, to create the flat development for the part, click on the UNBEND ALL Icon.

One Step Unfolding

Parameters

K-Factor   0.41

Create these bend lines when flattening

☑ Start line

☑ Centerline

☑ End line

OK

A Dialog Box appears. Select the K-Factor Option and type 0.41 for the value.
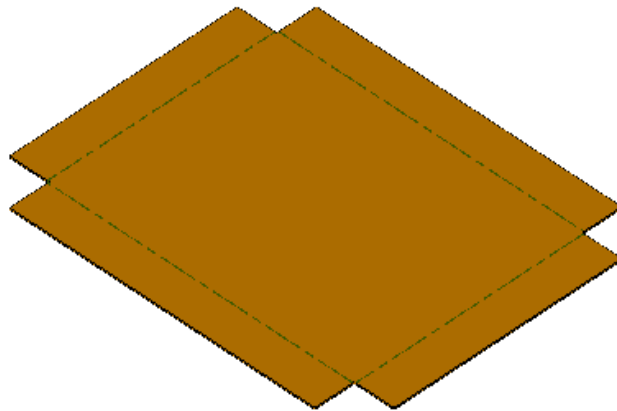
Then, click on the OK Button.

You are prompted to select a surface to unfold from.

Click on the broad, inside surface of the part.

The flat development is created on the screen.

You can click on the UNDO Icon to revert to the folded part.

This ends Part 1 of our exercise. In an upcoming Tips and Tricks we'll explore more neat sheet metal operations, so save your sheet metal toolbar for future use!