Sheet Metal - Basic

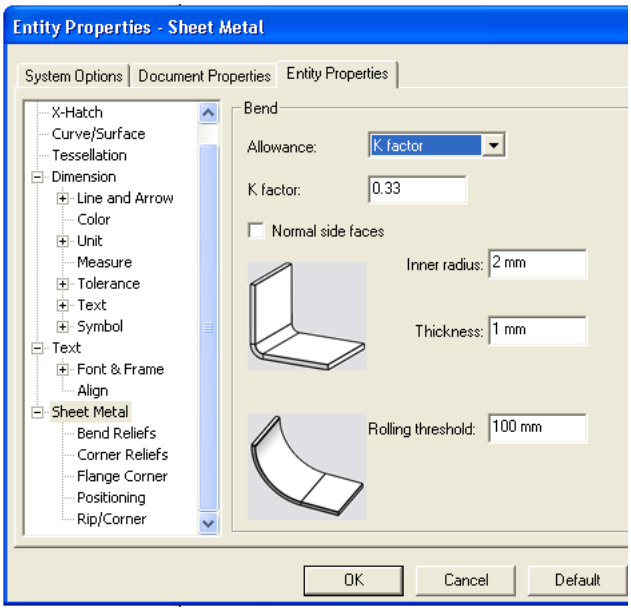
ThinkDesign do not force you to follow a single predefined design methodology. Depending on the requirements of your project, it is possible to change the approach to design - starting from scratch or starting from an existing model. This rule also applies to Sheet Metal application. You can mix these approaches and take advantage of the set of specific features and of an enlarged and powerful library of Sheet Metal-specific Smart Objects. The following exercise will demonstrate how to use Sheet Metal tools to create a casing for a motor. Let's get started!

Table of Contents

1. Sheet Metal Parameters

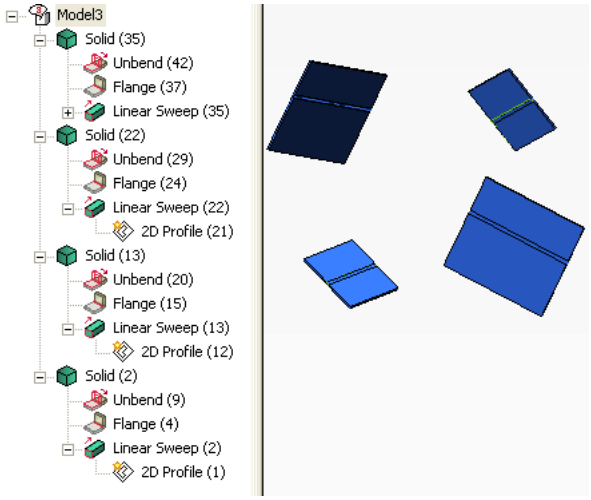
ThinkDesign provide a set of global parameters for each session. Right-click on the Graphics Area and click **Options/Properties**. Click the Sheet Metal category in the Entity Properties tab to see these parameters, shown in the image below.

The first step in designing a Sheet Metal part from scratch involves one of the following two methods:



- Sketching a closed profile (i.e: a rectangle) and extruding it. In this case the Thickness equals the extrusion depth.
- Sketching an open profile (i.e: a L-shaped profile) and extruding it with thickness option on ☑Thickness. In this case also the Thickness equals the extrusion depth.

Alternately, you can use the **Solid Flange** command to extrude an open profile, then use the **Unbend** command to unbend the solids

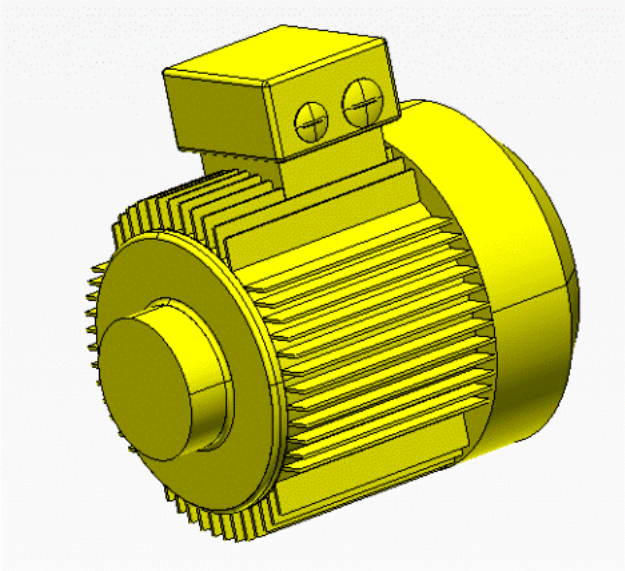


Now let's start a task that will help us understand the dynamics of working with ThinkDesign Sheet Metal commands.

2. Step 1: The Initial Casing Plate

In this step we will demonstrate how the new sheet metal functionalities are integrated into ThinkDesign 3D environment.

Open the file motor.e3 from the Webtask installation folder. (This file may have already been opened when you downloaded and started the "Sheet Metal Basic" executable from the think3 Webtask page.)



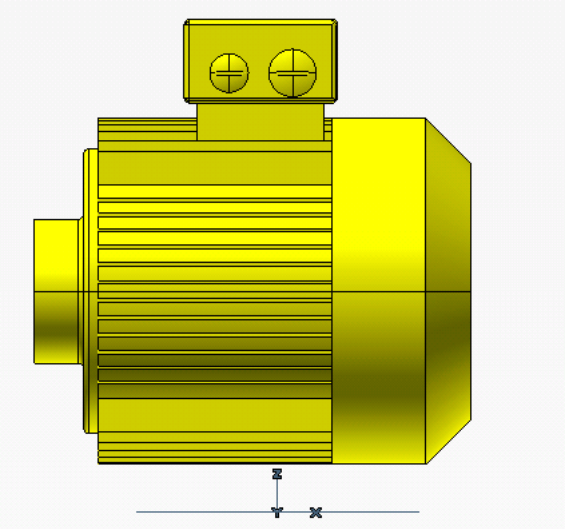
The object here is to create a Sheet Metal casing for this Motor and in the process use various ThinkDesign Sheet Metal functions.

Let's start building the Casing!!

You'll need a base profile that is large enough to encompass the Motor body.

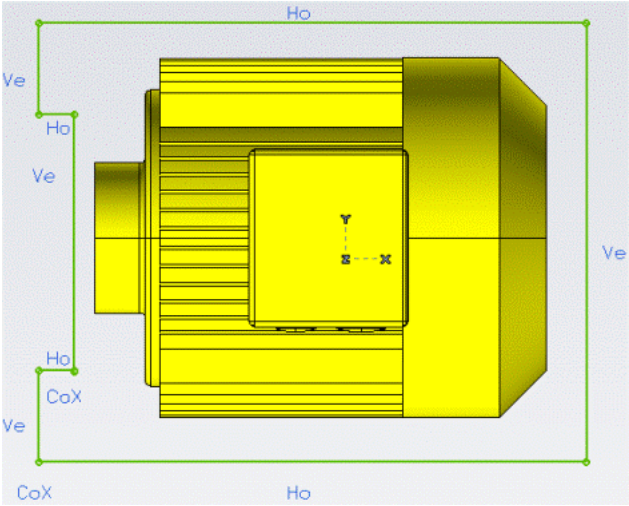
Before you sketch the profile, ensure that your Work Plane is oriented correctly for this exercise.

The Work Plane and the model must look like the image below. Note that the XY Plane is perpendicular to this view.



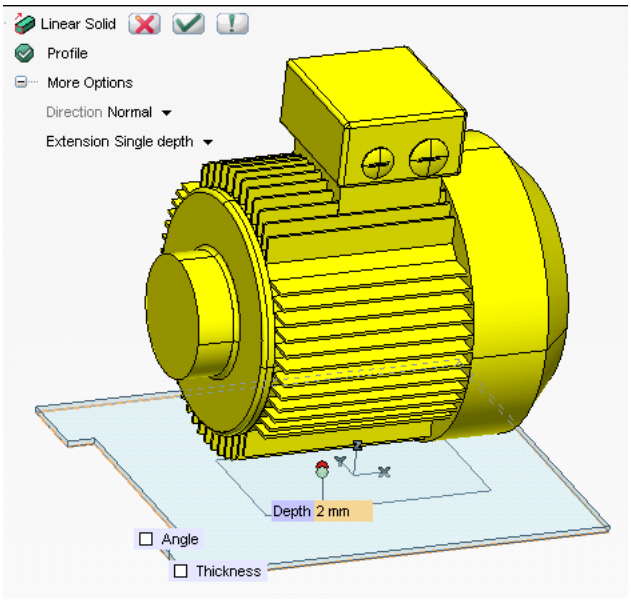
- Click on the 2D Profile tab at the bottom of the graphics area to create a Profile. If you do not see the 2D/3D Profile tabs, Check the option Show tabs Model, 2D Profile and 3D Profile fromOptions/Properties > System Options > General > Advanced Starting from ThinkDesign 2008.1, We have an option of creating both 2D as well as 3D Profiles by clicking on Insert > Profile > 2D or 3D.

- Start the **Polyline** command and sketch the Profile shown below. It does not have to be exact -- some general dimensions are: a horizontal size of 150mm and a vertical size of 140mm. Just make sure the Profile encompasses the Motor and that the notch is sketched on the left side.



- Start the **Linear Solid** command.

- Select the Profile you just sketched (if it is not selected already) and Set a extrusion depth of 2 mm in the mini dialog.



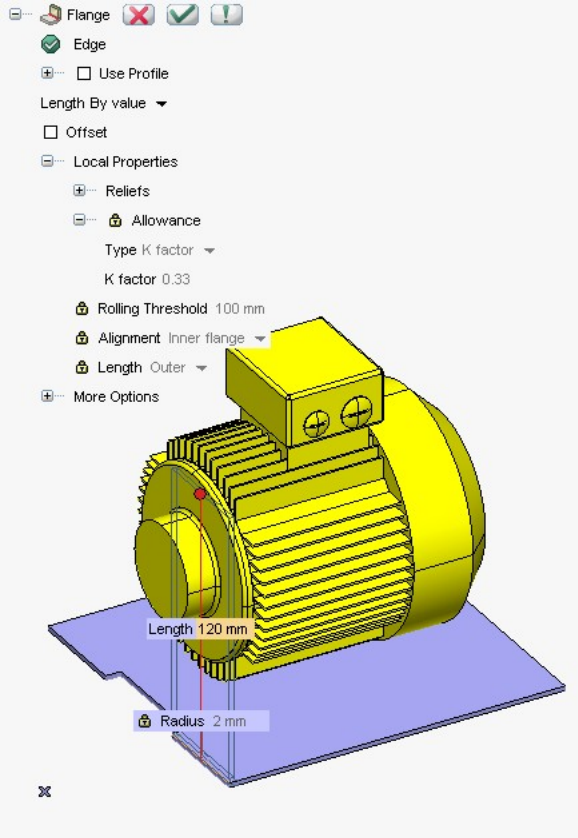
- Click OK to create the first face of the Sheet Metal casing.

The base portion of the casing is ready. Now let's go about covering the motor from its sides.

3. Step 2: Build the Flanges

Let's continue building the Motor's casing by creating the side flanges...

- Start the **Flange** command from ThinkDesign Sheet Metal toolbar.
- Select the Edge as shown below and pull the Flange handle away from the edge...then set the Mini-dialog as Length120.

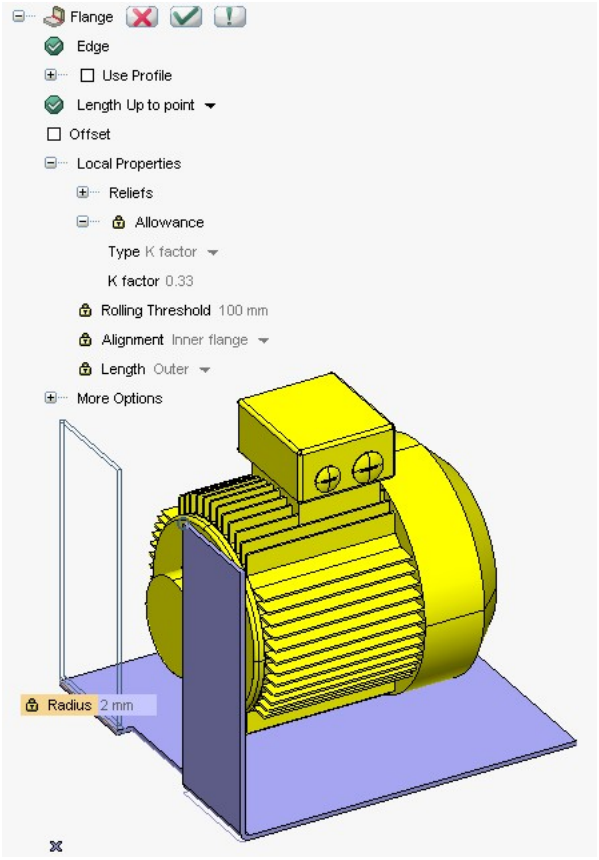


Keep the default values and settings for the other parameters in the Selection List as shown in the image.

- Click Apply to create this first side flange.

Now let's create a twin flange to the previous one, but on the adjacent edge of the base plate.

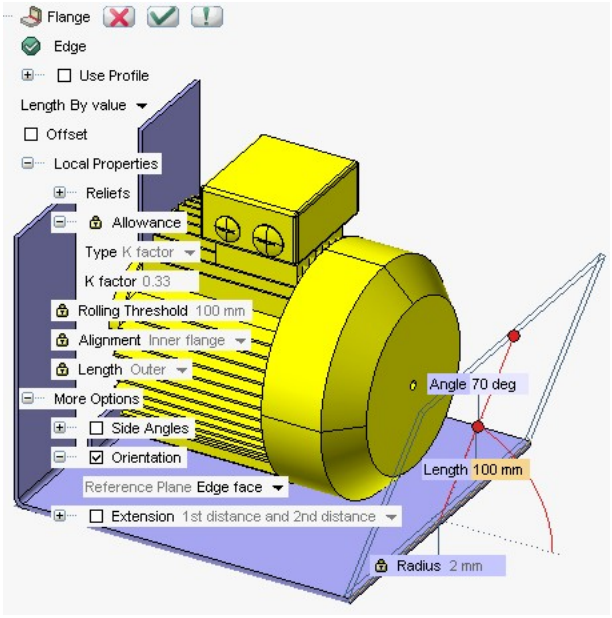
- Continue with the **Flange** command.
- Select the Edge as shown and pull the Flange handle and control the height in a parametric way with Length Up to Point option.



- Click OK to finish.

Now let's create another **Flange** using the edge of the base on the opposite side.

- Start the **Flange** command.
- Select the Edge as shown in the image below.

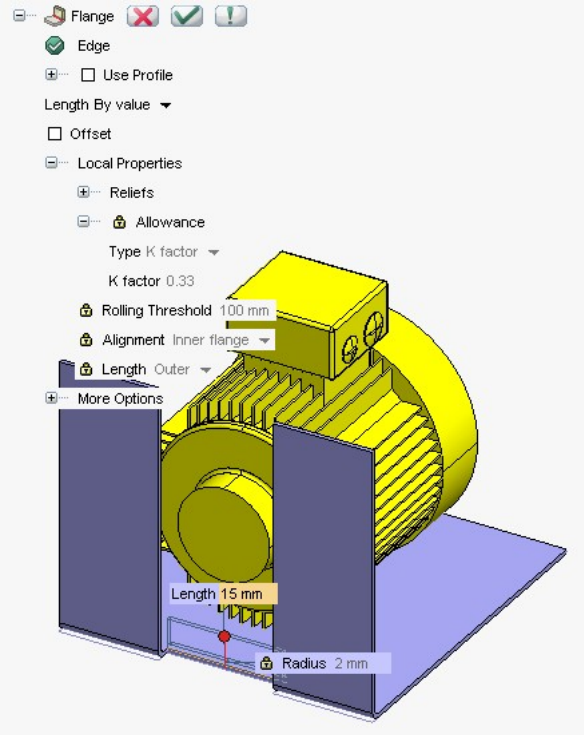


In the Selection List:

- Expand More Options and check the Orientation option
- Set an Orientation of 70
- Click OK.

Lets now make a **Flange** at the center, between the first two flanges that we have created.

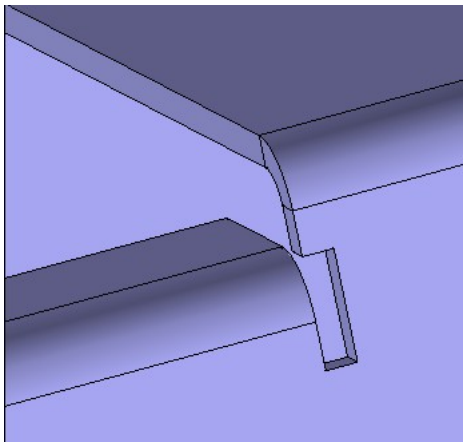
- Start the **Flange** command.
- Select the Edge as shown below.



In the Selection List:

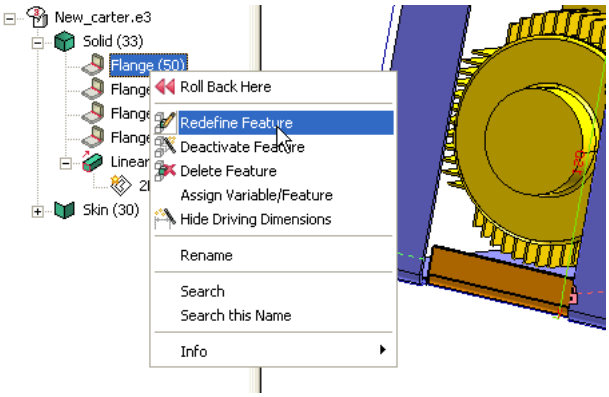
- Click on Alignment and set the alignment to Inner flange.
- Click OK.

Notice that the **Flange** has a Rectangular relief.



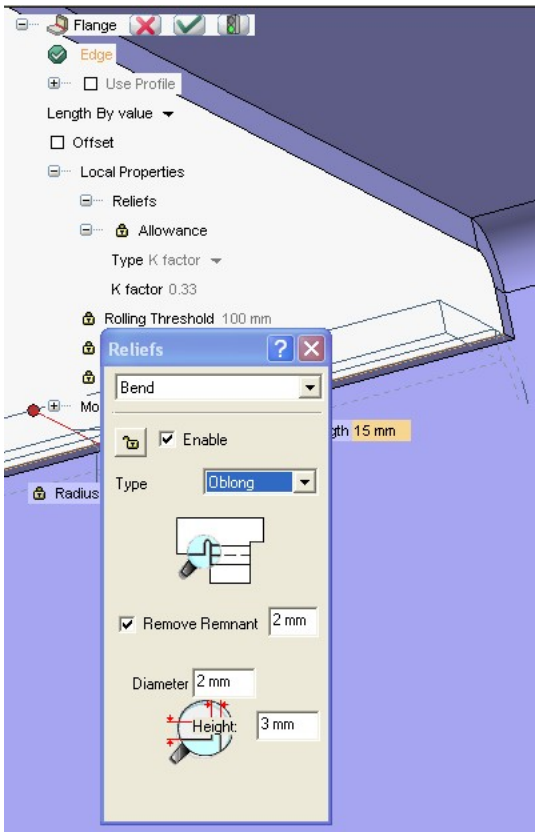
Let's now see how we can change the relief to an Oblong type.

- Right click on the **Flange** in the History Tree and click **Redefine Feature**.



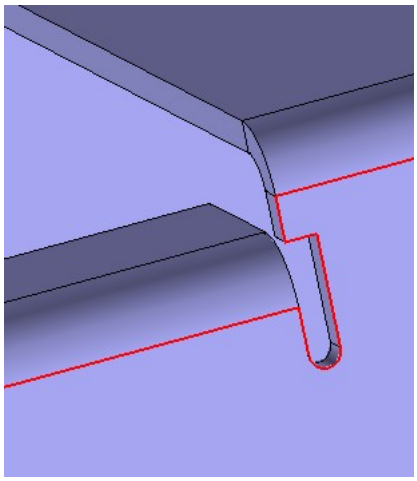
- Expand Reliefs in the Selection List.

- In the Reliefs dialog that pops up click the Bend button, then click on Type to unlock it.
- Select Oblong from the pull-down menu.



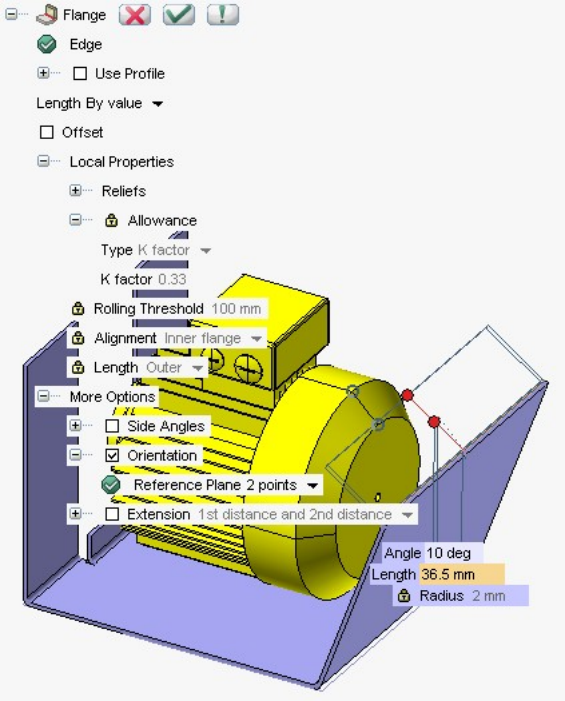
- Close the the Reliefs form by clicking the X...
- ...then Rebuild the model

You can see the change in the shape of the Relief.



Let's now cover one more side of the Motor with a **Flange**.

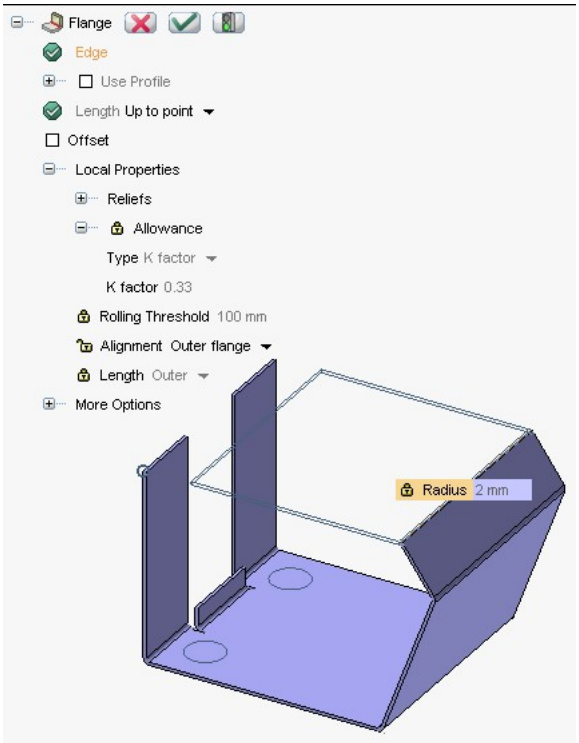
- Start the **Flange** command.
- Select the Edge as shown below.
- Click on More Options and check Orientation with reference plane as 2 points as shown with the values with an angle of 10



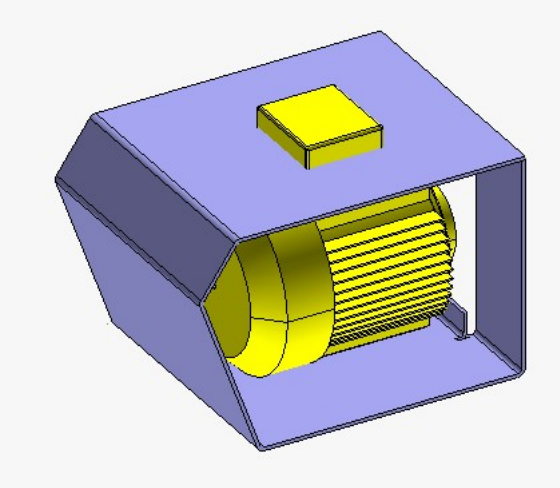
- Click OK

Let's add another **Flange** to the remaining side.

- Start the **Flange** command.
- Select the Edge as shown below.
- Create a flange by selecting length up to point and selecting end point on opposite flange as shown below.
- Click OK.



- Click OK when finished.

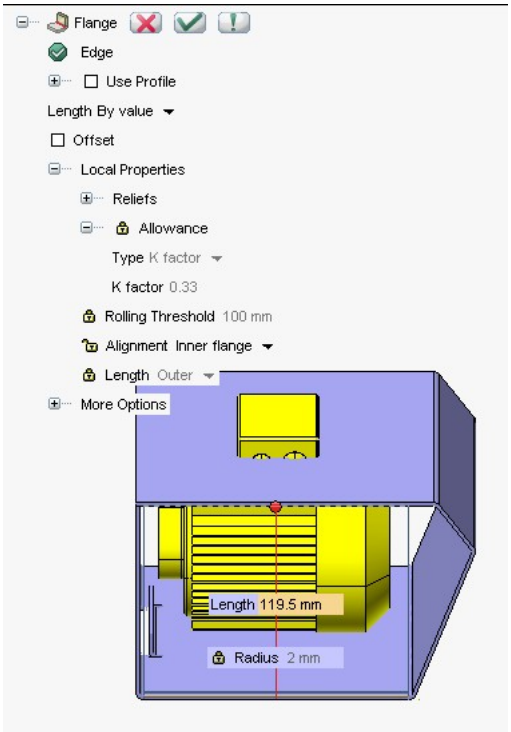


Let's see how easy it is to work with traditional Solid commands on Sheet Metal parts. On to the next step!

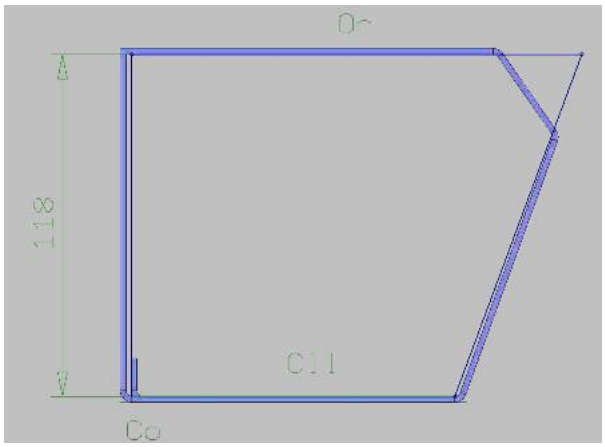
4. Step 3: Sketching Sheet Metal Features, and Easy Creation of Drawings

Now Lets add some side flanges to the casing.

- Start the **Flange** command again and select the Edge as shown
- Set the length to 119.5 and set the alignment to inner.



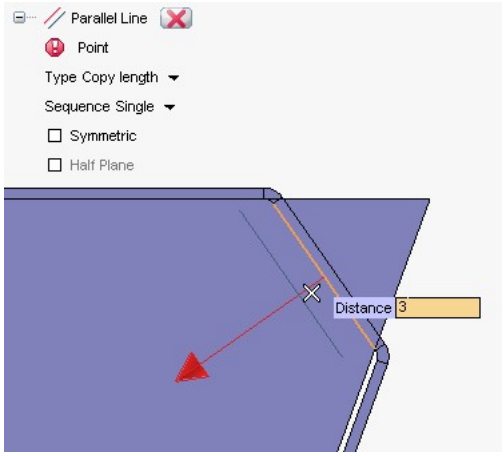
- Now check the Use Profile and edit the profile as shown .
- Redefine a new circular/rectangular relief on corner in flange local properties with dia8.
- Set a Collinear constraint for the base horizontal line on its edge and a horizontal constraint on top line. Also, add a coincident constraint as shown.
- Insert vertical dimension of 118



- Repeat the same to create the flange on the other side of the casing.

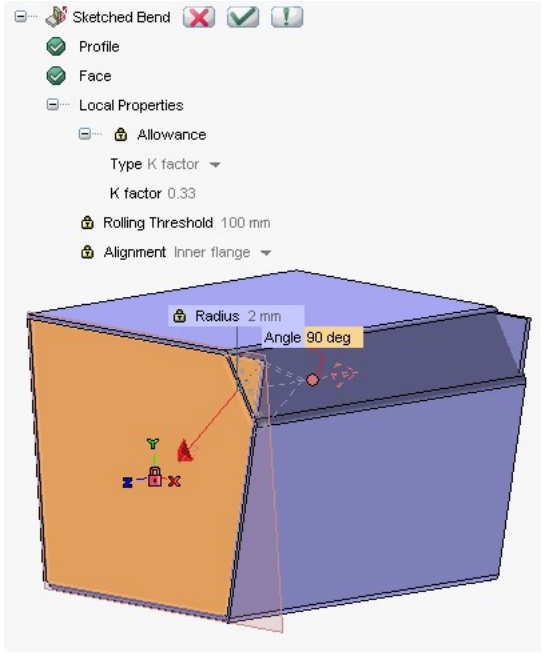
We will use the **Sketched Bend** command to bend a sheet along a predefined Profile.

- Place the Work Plane on the last created flange .
- Create a parallel line selecting the edge as shown using copy length mode, distance 3

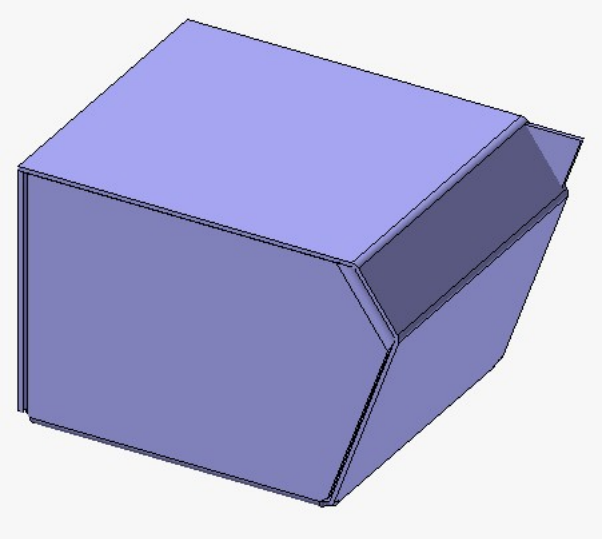


- Start the **Sketched Bend** command.
- Select the line we have drawn just now.

The Face gets selected automatically and the preview shows the direction of bend creation.

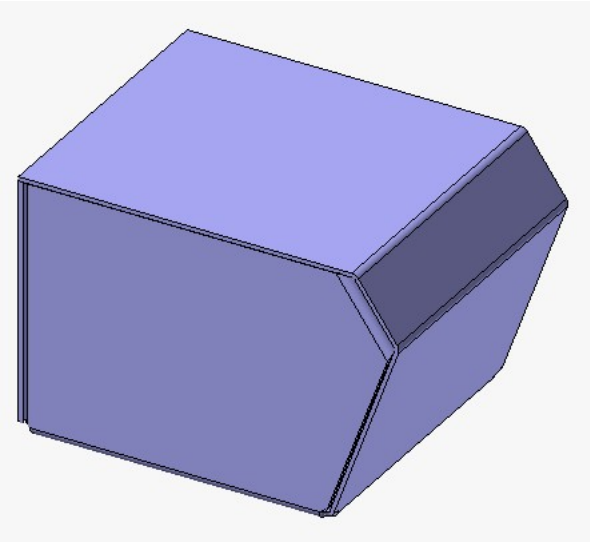


- If necessary, double click on the red arrow to invert the direction of bend, and...
- ...if necessary, double click on the other red arrow to select the smaller corner portion of the face to be bent.
- Set the bend angle in the Mini-dialog to Angle 90 .
- Click Apply from the Selection List to apply the command.



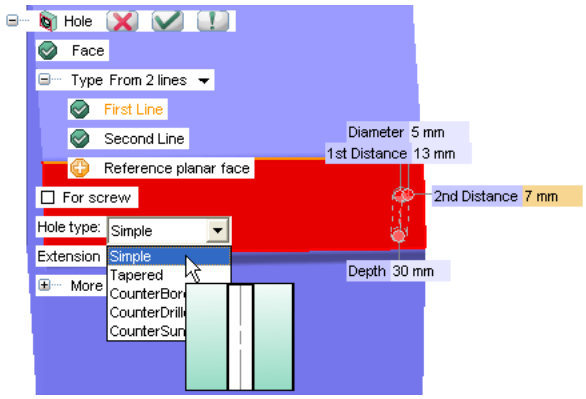
The first bend is created! Now go for the second one..

- Repeat the same procedure and create the bend on the other flange of the casing.



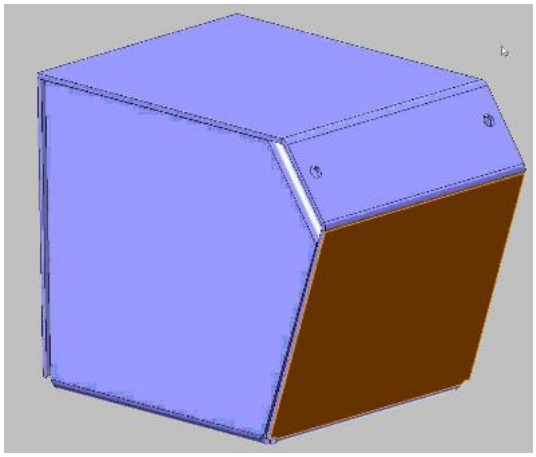
Sketched bends are easy, aren't they? Now let's examine another command: **Unbend**.

- Insert two holes of Dia5 as shown below.
- Set the first distance as 13 and second distance as 7.
- Set the depth of the hole as 30

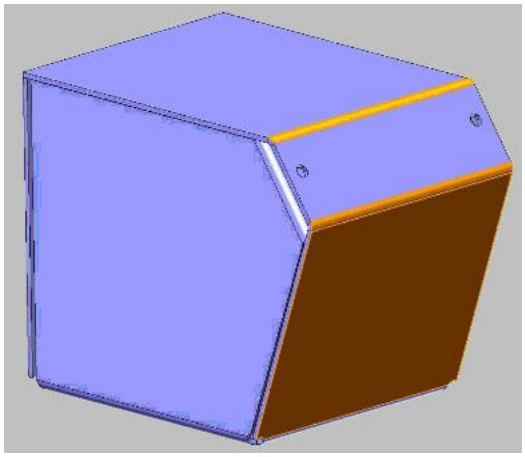


We will now **Unbend** the flanges locally selecting a fixed face.

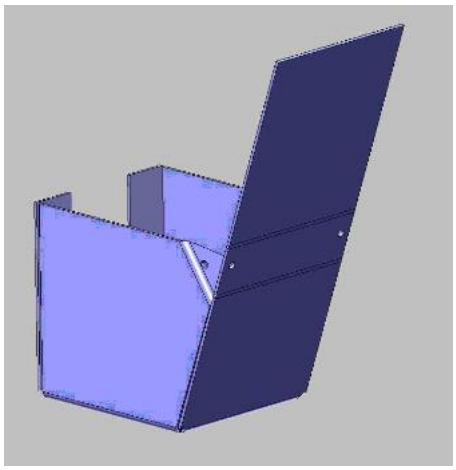
- Start the **Unbend** command.
- Click on the flange as shown to specify the Fixed Face.
- Set Type Local in the Selection List.
- Select the bend as shown to specify the portion of the material to be "unbent."



Click Apply.

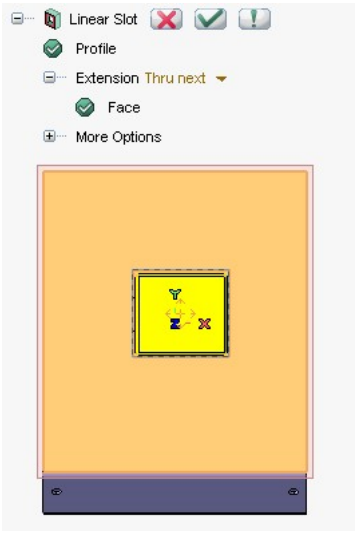


You can see from the image below that the **Unbend** occurs only at the specified portion of the Motor casing. This is a "local" unbend.

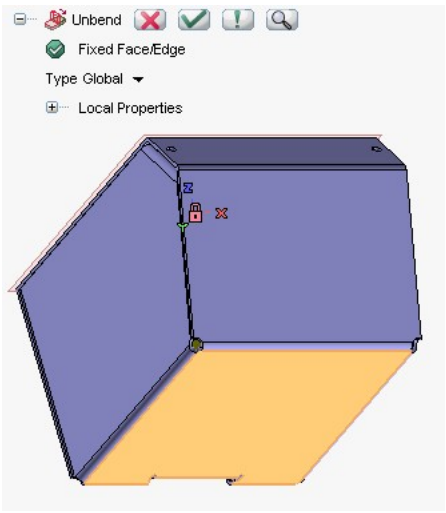


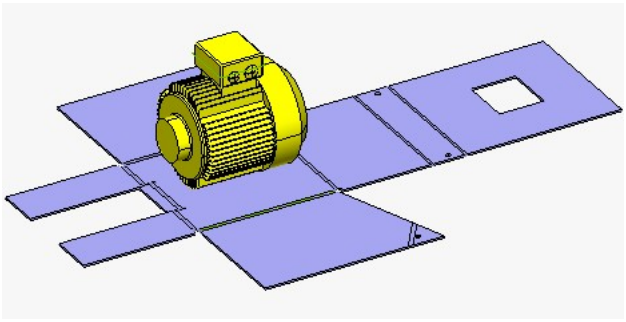
We will not need this particular **Unbend** feature for this task -- so **Delete** it from the history tree or use the **Undo** command to undo it.

- Now move the Work Plane to the top flange.
- Sketch a rectangle in order to make a blind slot of depth 10 to avoid interference between the electric connectors panel and the top flange of the casing.

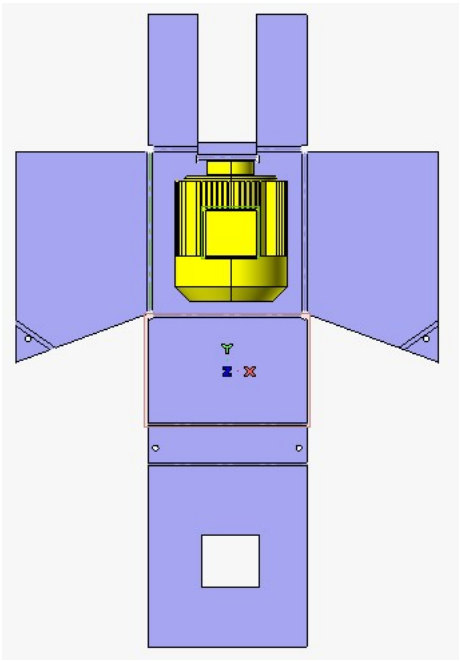


- Now use Unbend command to make a global unfold of the sheetmetal casing selecting the bottom face as the fixed face.

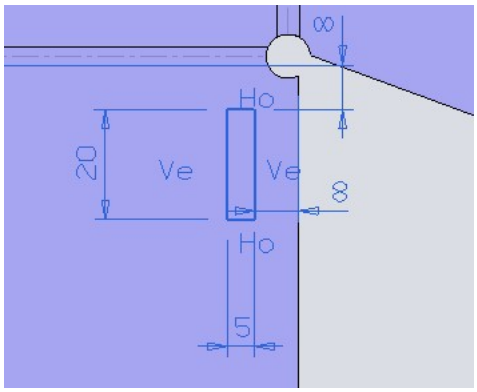




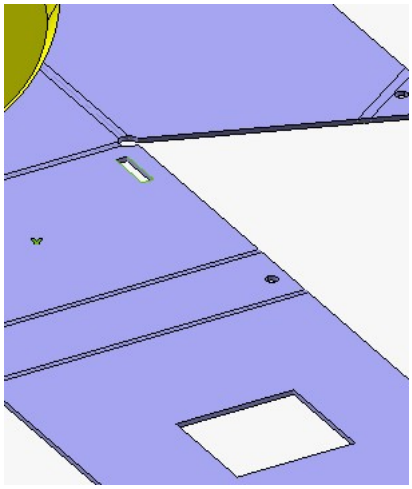
- Now move the Work Plane to inner side of the rear flange as shown



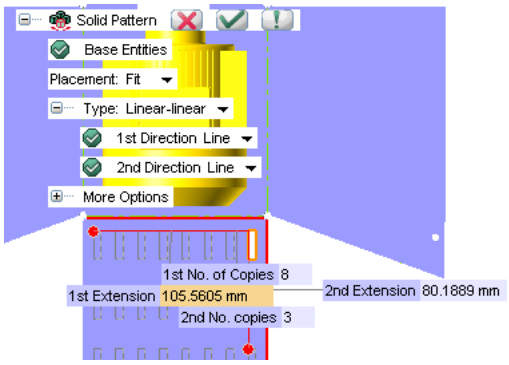
- Now insert a profile as shown.

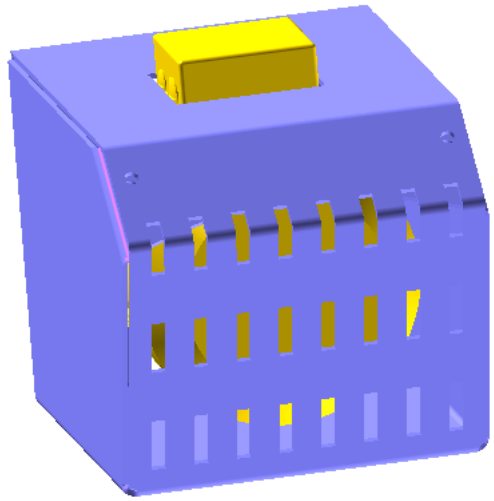


- Make a slot with the profile and make it Thru all as shown

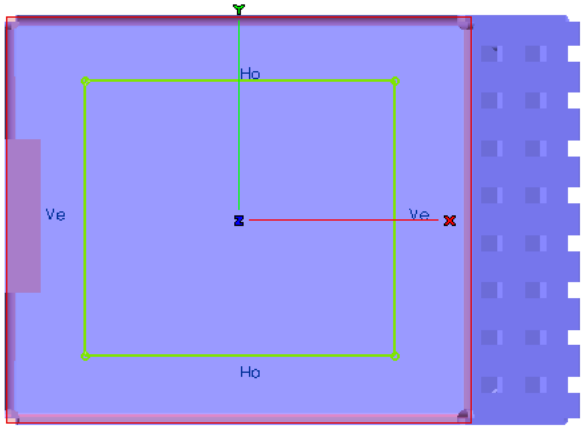


- Pattern the slot with a Linear-Linear feature with the parameters as 80mm x 8 and 75mm x 3.
- Use Rebend command with respect to the same fixed face we have used to unbend.

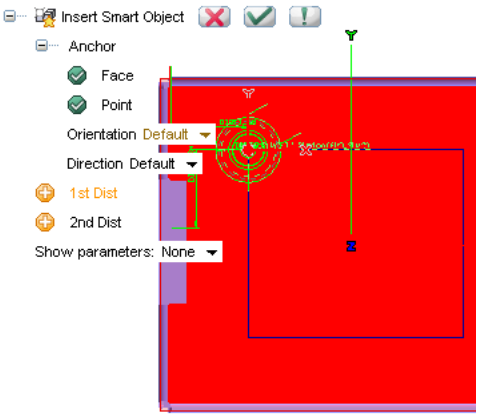


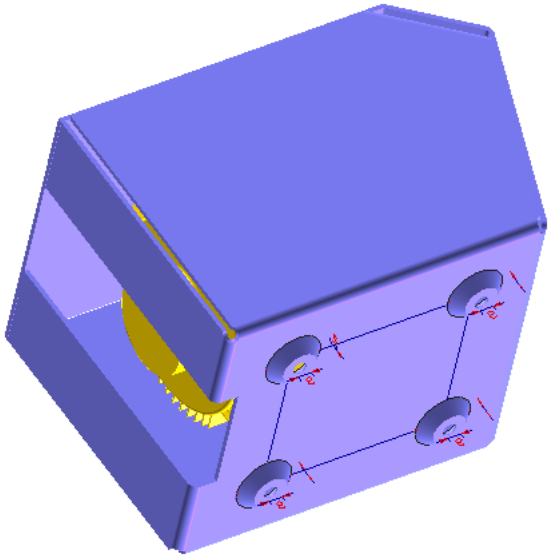


- Move the Work Plane on to the external side of the bottom face.
- Insert a simple rectangular profile offsetting the external edges as shown.

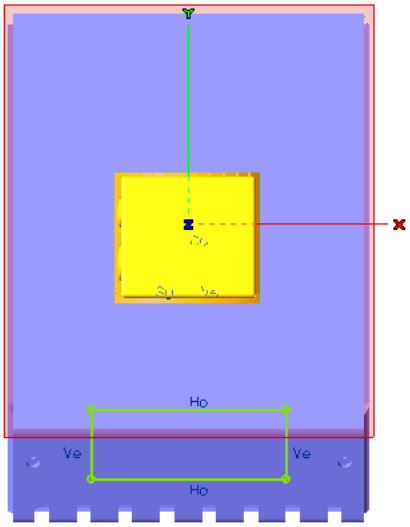


- Click on Smart Object Library and click on Insert Smart object and select PUNCH CONIC selecting rectangular profile corners as placing points.

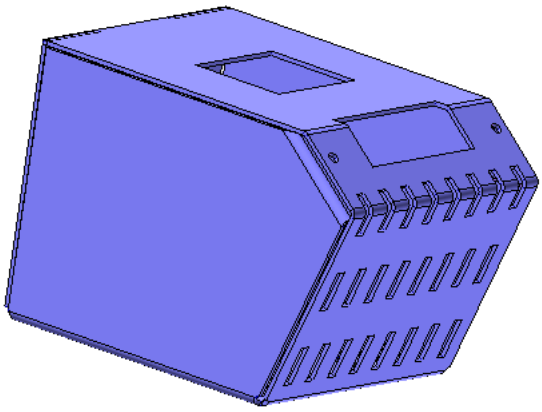




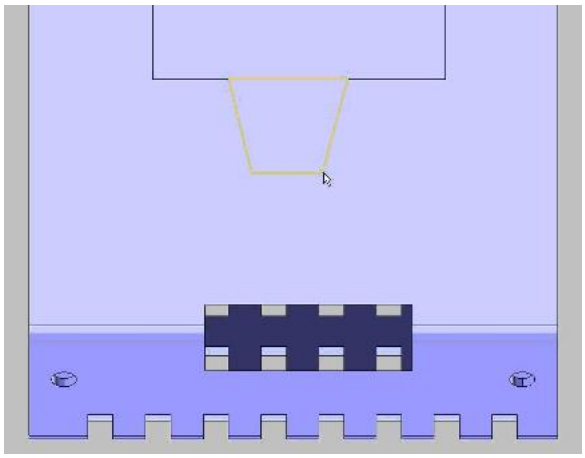
- Now Create a new Profile on the top flange as shown.



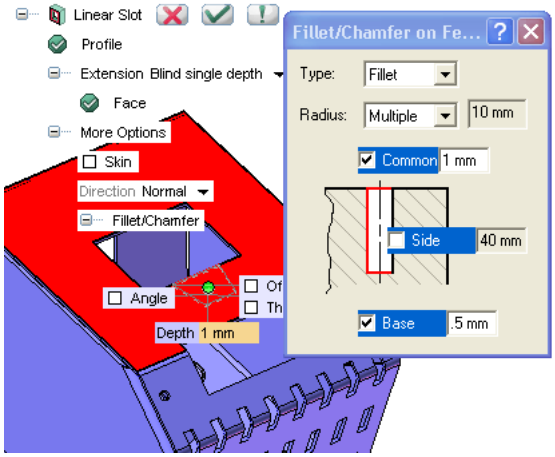
- Insert a new blind slot and set the depth to 20mm



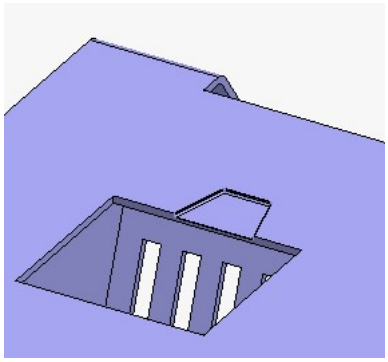
- Move the Work Plane on to the top flange and add one more profile on the flange as shown.



- Insert a new blind slot of Depth 1mm, Radius 0.5 on side and base and 0.1 on ../../Common as shown.

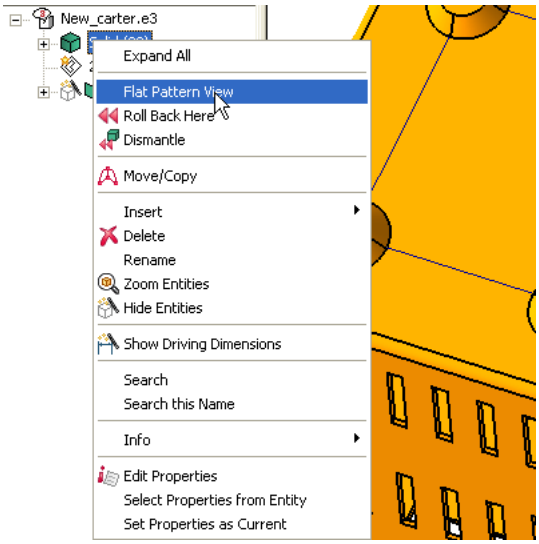


- and the result is.....

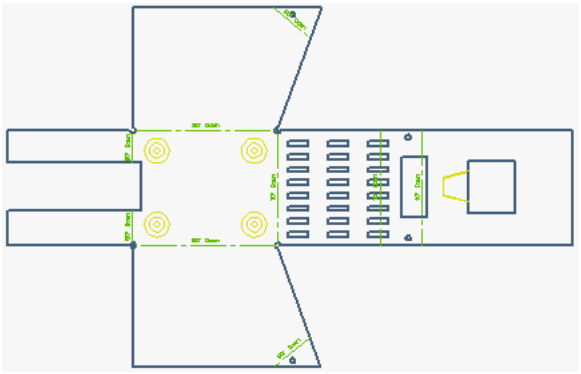


Sheet Metal designs are usually presented in their 2D "flat-panel" or "flat pattern" view on a drawing. ThinkDesign has the feature **Flat Pattern View** (or **Insert→Drawing View→Flat Pattern View**) to achieve this from the 3D environment. Let's check it out!

- Right click on the Solid in the History Tree and click on **Flat Pattern View**.

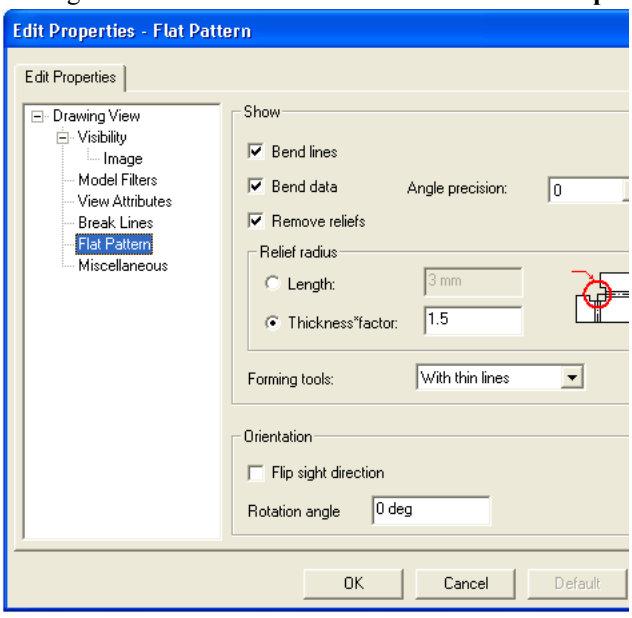


ThinkDesign will automatically open a **New Drawing** with the "flat pattern" View of the selected Sheet Metal part. The image below shows our casing in its **Flat Pattern View**.



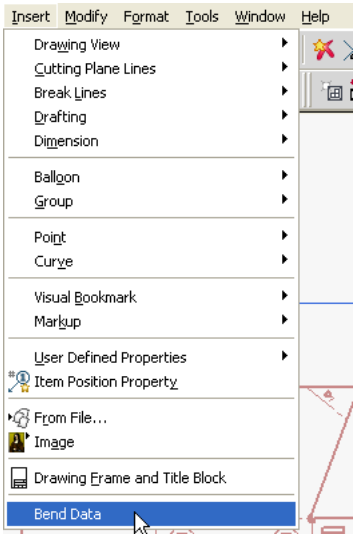
You can see that all technical details like the bend lines, bend data, etc. are readily available.

- Right click on the **Flat Pattern View** and click **Properties**.

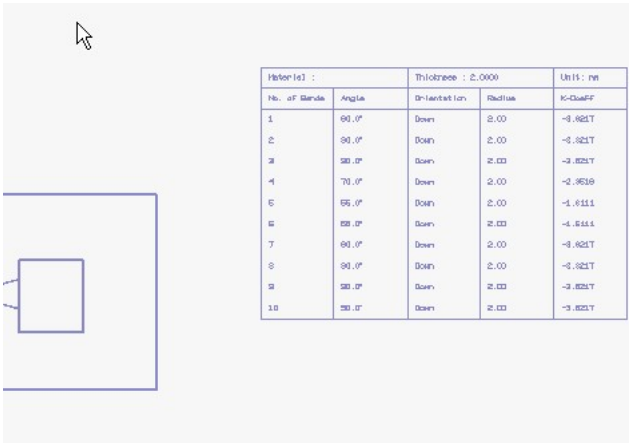


Under the Flat Pattern tab you can control the visibility of the Bend lines and Bend data by checking/unchecking the boxes.

- Now you can insert bend data on to the layout as shown.



Bend data are associative. Go back to the model and change the value of one or more bends. Go back to the drawing and regenerate for upto date data..

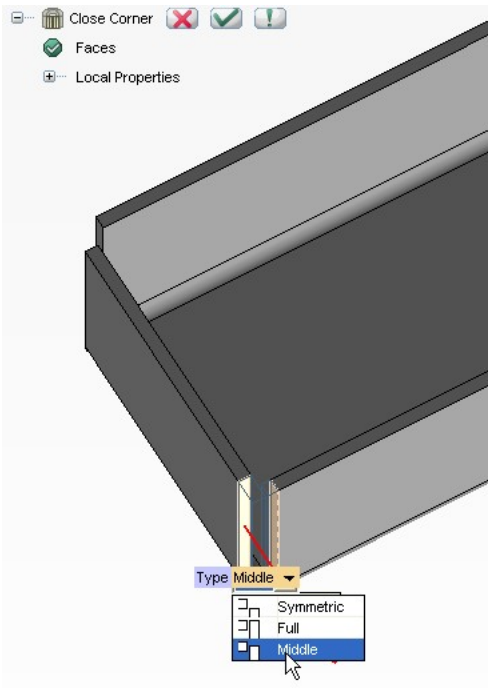


- Congratulations... Job well done!.

Lets explore some more features of Sheet metal in ThinkDesign

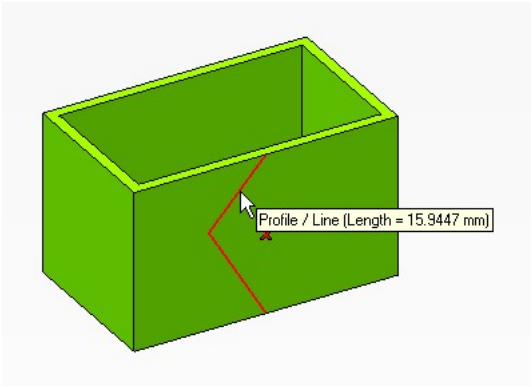
Some times, when we create flanges, we get to see some gaps at the corners. Lets see how we can close these gaps using **Close Corner** command

- Create a sheetmetal component as shown and add flanges. Now click on **Close Corner**
- Observe the relults by changing Type to Full, Middle and Symmetric.

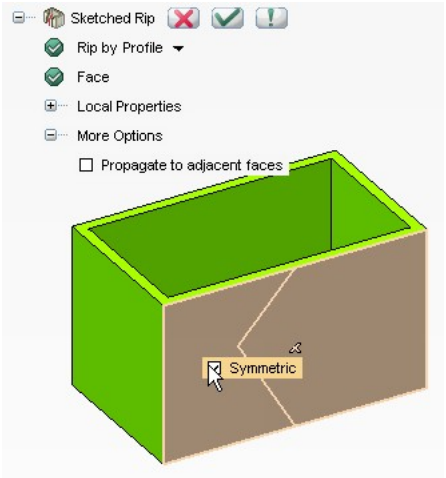


Let's now see a new command **Sketched Rip**

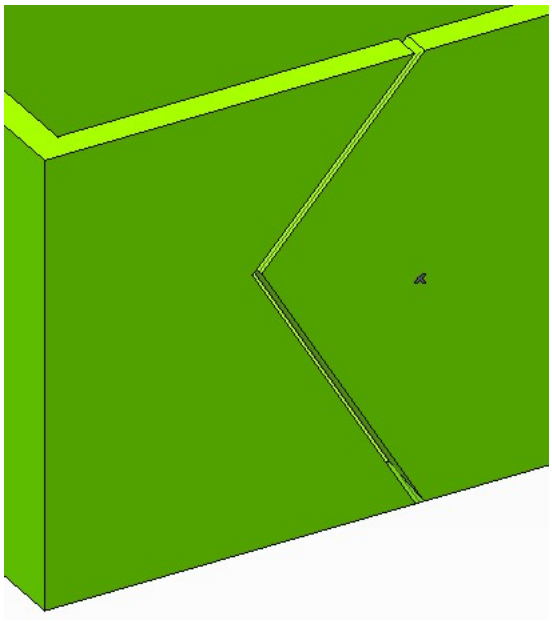
- Create a solid as shown and create a profile on one of its faces.



- Start the **Sketched Rip** command by selecting the profile you have created and the face of the solid.



Now see the result. The face of the solid has been ripped.



This brings us to the end of this task. Congratulations!! We have covered some of the basic concepts of ThinkDesign Sheet Metal functionalities:

- Sheet Metal Properties
- Flange creation and modification
- Bend/Unbend (locally and globally)
- Flat Pattern View creation
- Solid operations on Sheet Metal parts
- ...and much much more!

Now you are ready to create your own personal designs with ThinkDesign Sheet Metal. Good luck!

A.

- Sheet Metal workflow in ThinkDesign.