ZW3D
Sheet Metal Training Guide

ZW3D™ V2014 Sheet Metal Training Guide

Forward

The full version of the ZW3D documentation is available by selecting **Help Browser** from the ZW3D **Help Menu**. Go to the **" Program Folder\doc"** directory on the ZW3D program directory and open the file **"ZW3D.chm".**

This **Sheet Metal Training Guide** is available as a printed manual and in PDF format. To view or print the PDF version, install the Adobe Acrobat Reader and open the corresponding files in the **" Program Folder\PDF"** folder in the program directory. Refer to the **HELP Manual** for information relating to specific tasks.

NOTES:

1. For best results, view and print the PDF version of the documentation using the Adobe Acrobat Reader.
2. Since the **Sheet Metal Training Guide** is only a portion of the complete documentation, it is highly recommended that you utilize the **HELP Manual**. You will find additional topics, more information on command options, input options, additional command notes, and cross-links to related tasks and topics.
3. It should be noted that the HELP Manual will always be more up-to-date than any printed or PDF material. If you find any discrepancies between the two, always refer to the HELP Manual.
4. While you are working with ZW3D, simply press **F1** to view the documentation about the command you select.

To help you customize your CAD/CAM requirements, please contact our Customer Support department on our website http://www.zwsoft.com/ or call at (8620) 38289780 ext 565. You can also contact your local sales representative or application engineer to discuss your requirements.

Thanks for being our customer!
Team ZW3D

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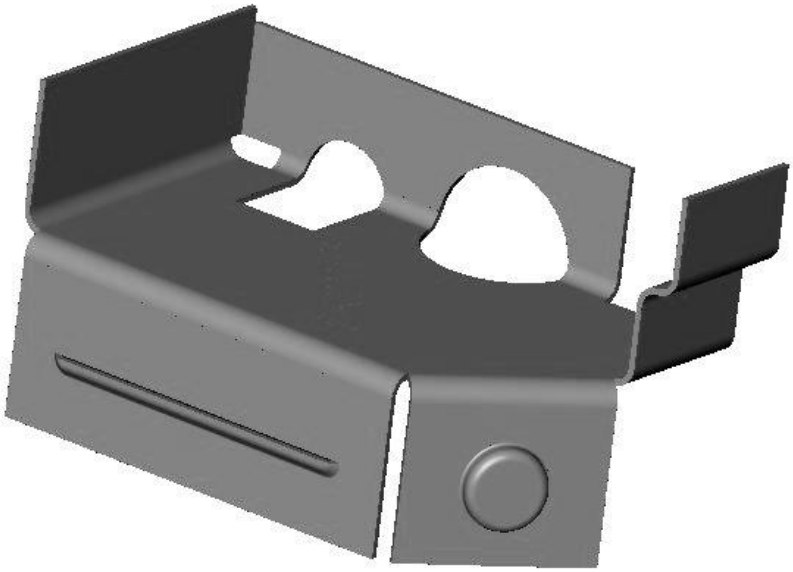


Sheet Metal (FabDesign®)

Offering a comprehensive set of tools for creating standard sheet metal features, FabDesign is an extremely interactive and easy to use solution for the design and documentation of chassis, brackets, housings and other fabricated sheet metal components and assemblies containing sheet metal parts.

Feature List

- Full and partial flange creation with automatic bend radius and relief
- Simplified tab and cut creation
- Corner closing with relief
- Easy S-Bend creation
- Flexible, easy to create louvers and dimples
- Folding and unfolding of entire part or selected bends
- Easy modification of bend angle or radius
- Features can be added in folded or unfolded state
- Tagging of bends for working with non-native parts

Chapter 1 Sheet Metal Attributes

These attributes can be set up prior to creating sheet metal bends but is not necessary. If your company has K-Factor standards and or relief ratios different than these you can open the Part Template file and set these global attributes. Click Sheet Metal Attributes through the Attributes tools. They will then be in effect for every new part you open.

However, they can still be adjusted for each bend created as you will see on the following pages.

Bend Radius is the inside radius of each flange you add.

K-Factor defines the amount the material will stretch when bent. The material as well as its thickness and radius affects K-Factor. There is a default value on the form but you will need to lookup this value in an engineering handbook prior to using it in production.

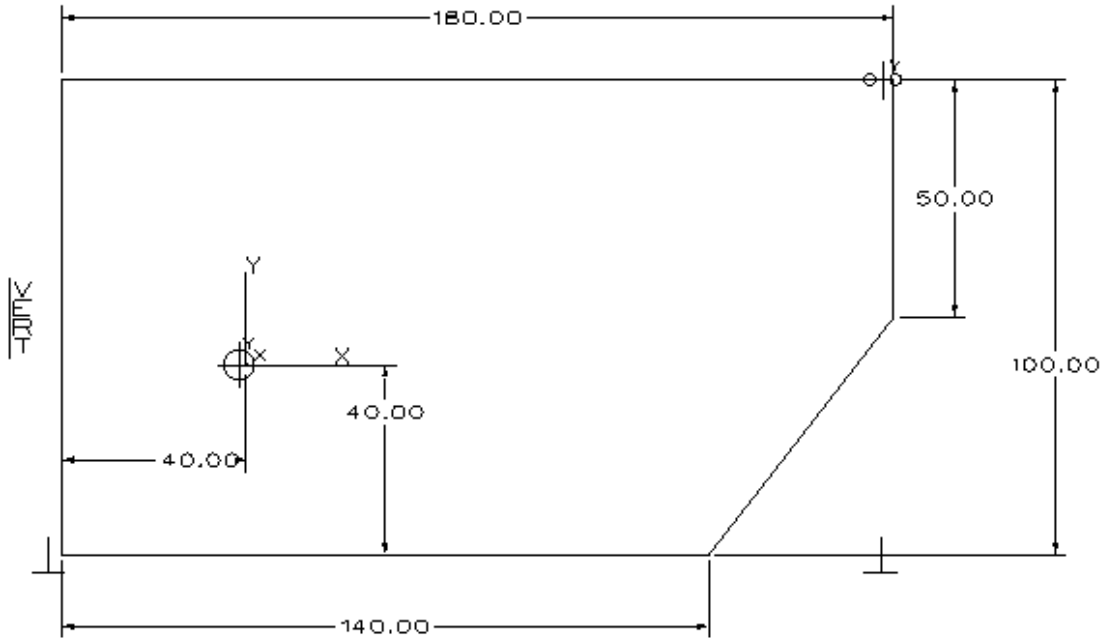
Flat Tolerance Refers to the flat tolerance allowed for sheet metal parts in their unfolded state. The default value is 0.10.

Relief width is either specified as a ratio of the material thickness or a fixed value. This is used for partial Face flanges.

Chapter 2 Creating the Base Geometry

Open a new multi object file, Name it ***Sheet_Metal_V14.Z3***.

Create a new part called ***01_Sheet_Metal.***

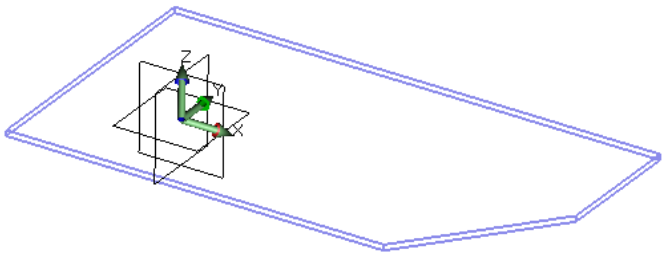


Select the ***Sheet Metal Extruded*** command.

Create a new sketch on the **XY** plane, as shown in the training image above.

To exit the sketch.

For the extrusion use
0 for the **start** and
2 for the **end**.



Chapter 3 Adding a Full Face Flange

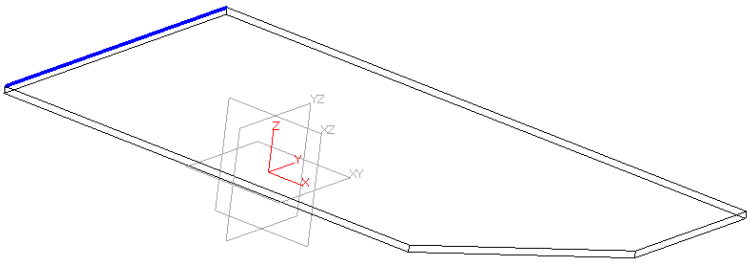
Select the ***Full Face Flange*** command  , and select the upper edge as shown below.

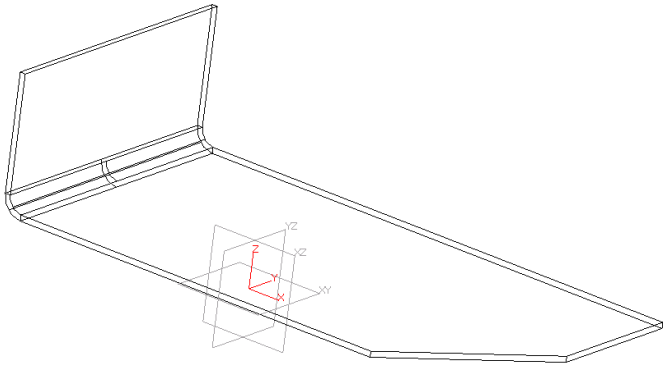
(Move the cursor over both the top and bottom edge before selecting the upper to see a preview of the results.)

Use **90** for the ***Angle*** and **40** for the ***Flange length***.

(You can change the values before hitting **OK** to see the affect each has).

Notice you also have options to change the Bend radius and K-Factor.





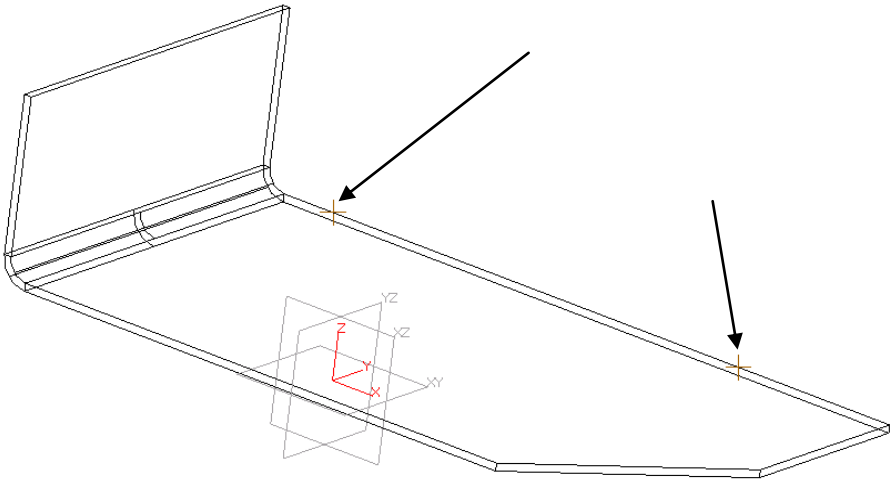
Chapter 4 Adding a Partial Face Flange

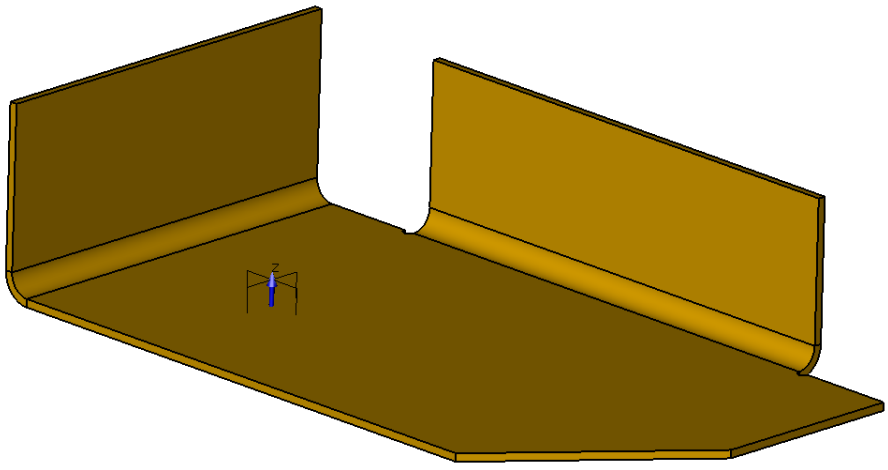
Select the ***Partial Face Flange*** command. 

select the upper edge as shown.

Use the default bend angle of **90**.Use a flange length of **40**.

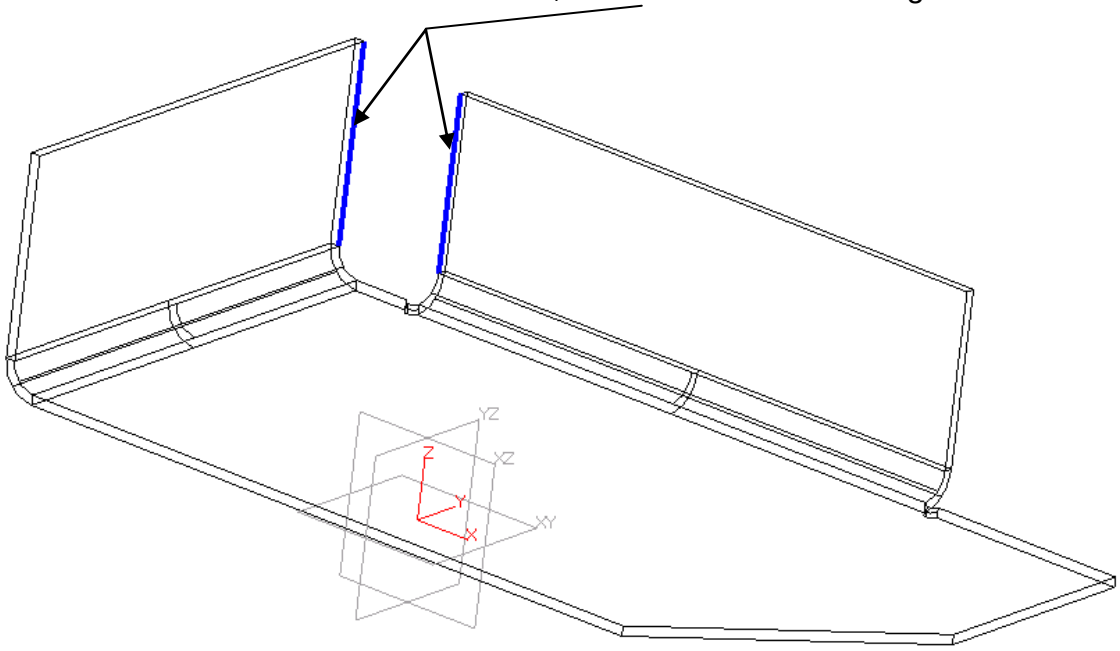
With the partial face flange you can also specify the type and size of the bend relief.

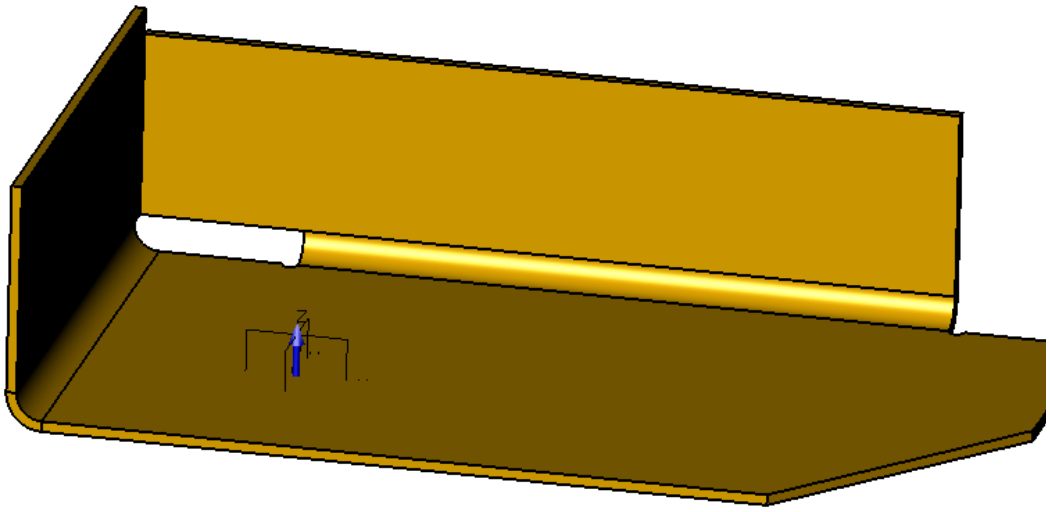




Chapter 5 Creating a Closed Corner

Select the ***Close Corner*** command  , and select the inside edges as shown.

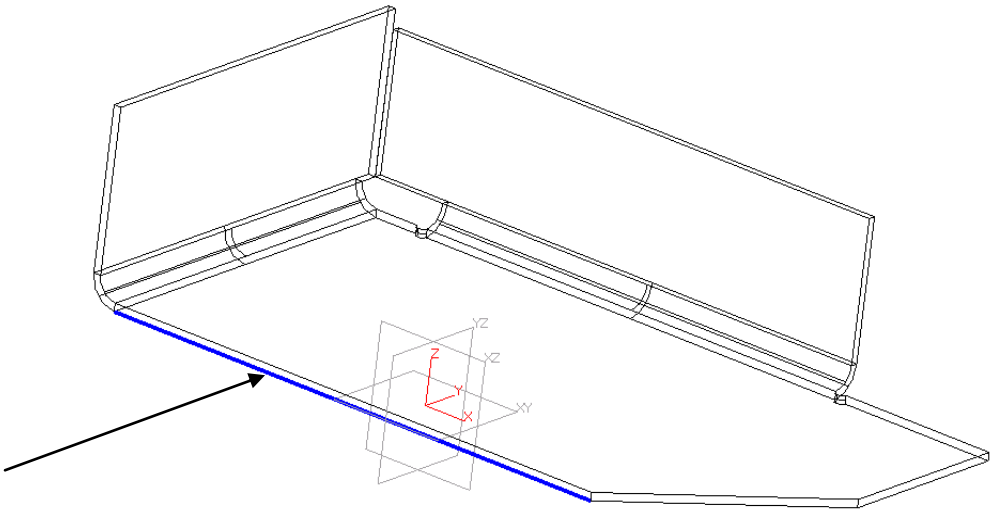


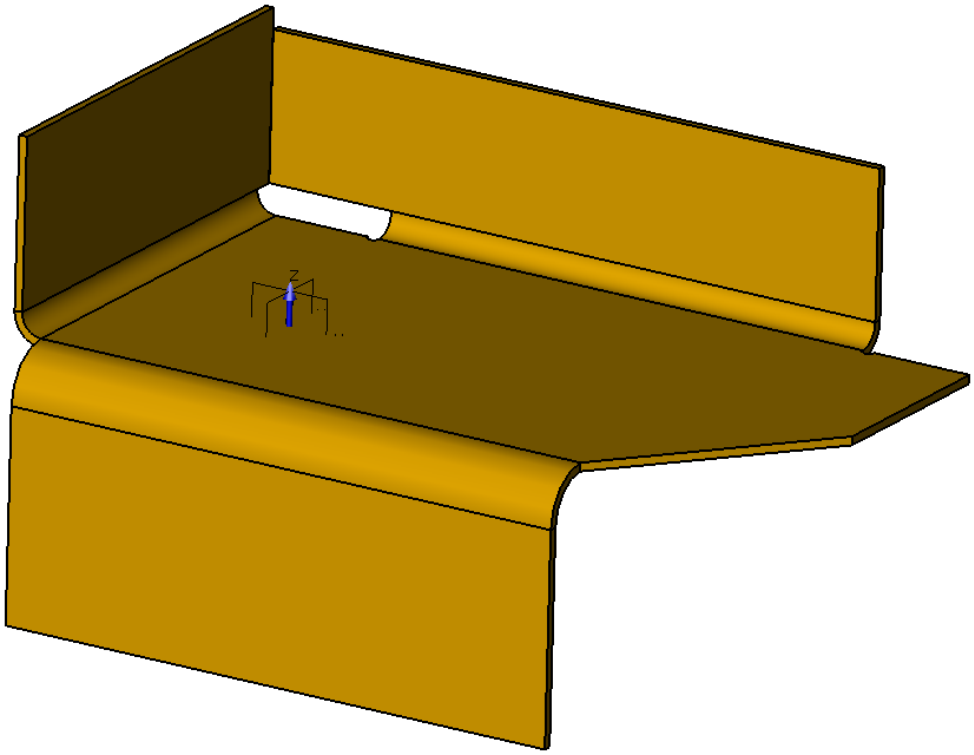


Select the ***Full Face Flange*** command  , and select the lower edge as shown.

Use the defaults for the bend angle. **(Middle-Click)**

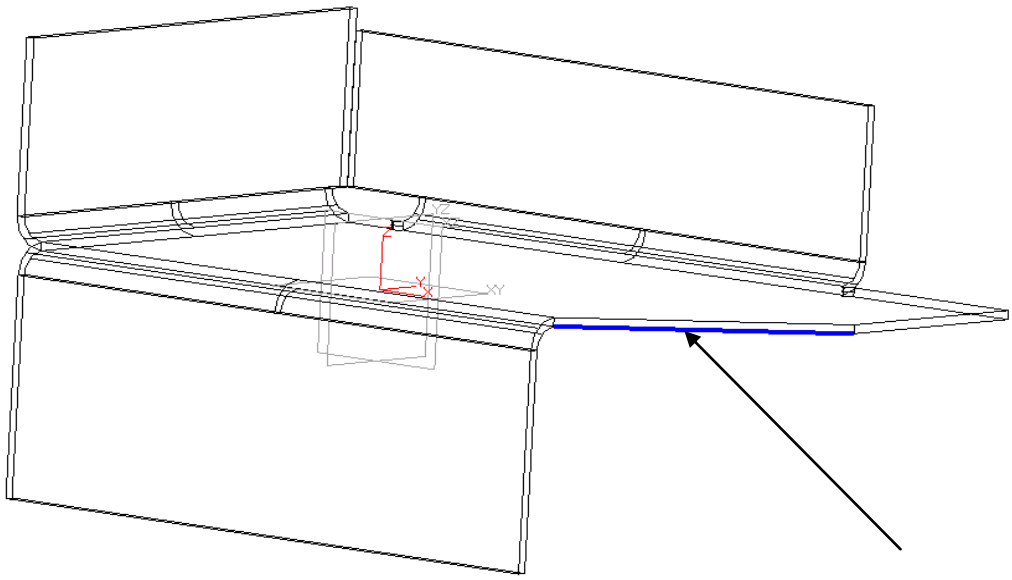
Use **50** for the flange length. Change the bend radius to **10**.

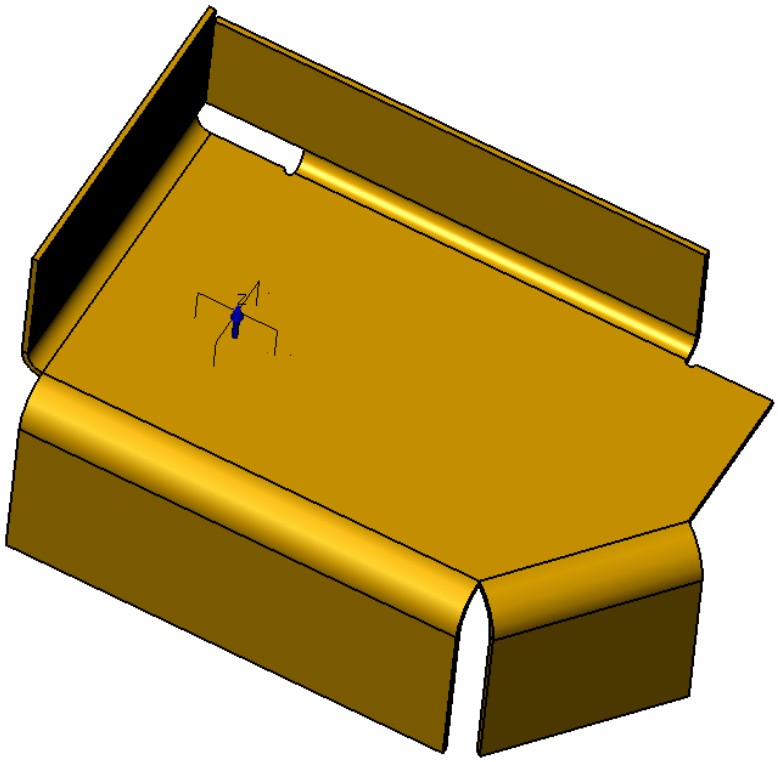




Select the ***Full Face Flange*** command, and select the lower edge as shown.

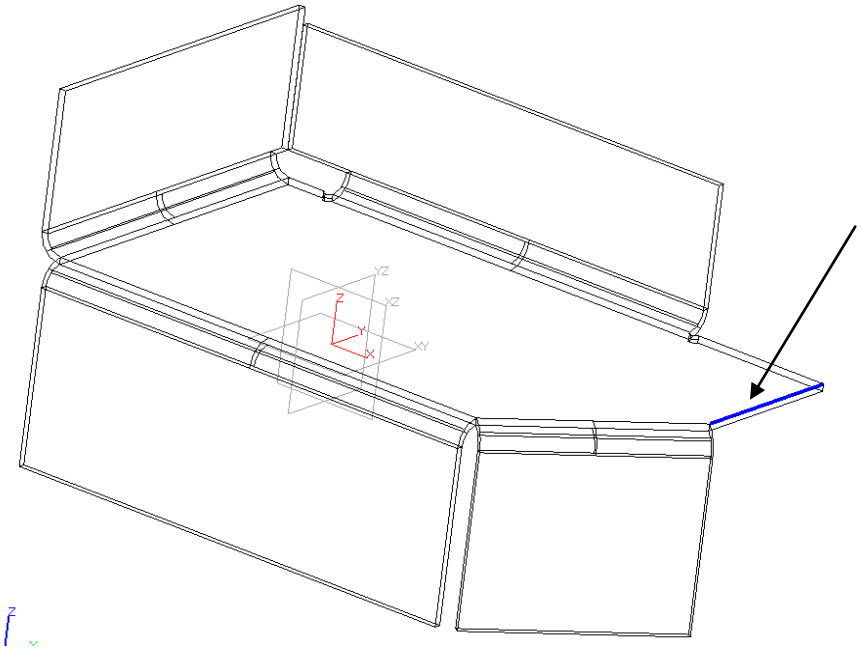
Use the defaults set from the last time.

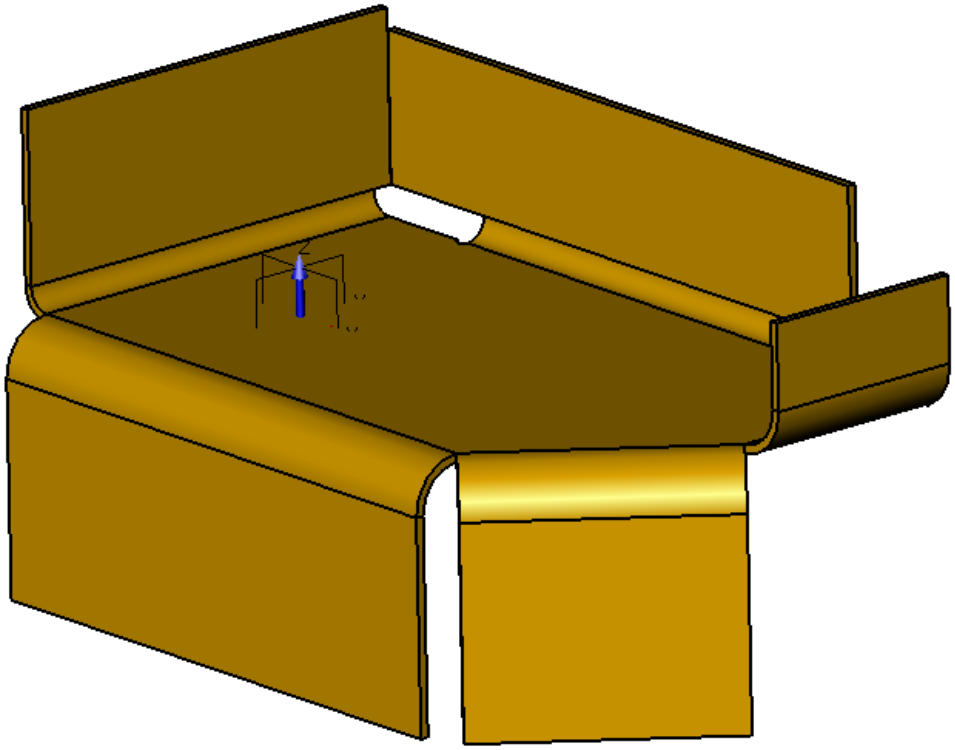




Select the ***Full Face Flange*** command, and select the upper edge as shown.

Change the flange length to **20**.



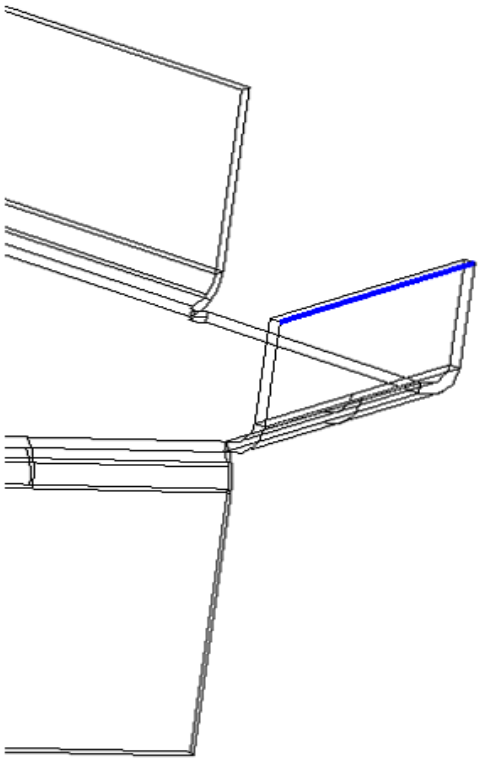


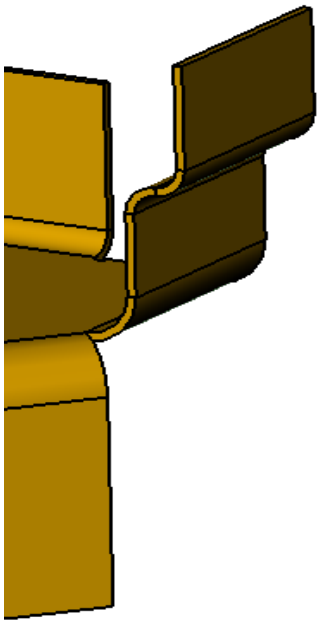
Chapter 6 Adding a Flange with an S-bend

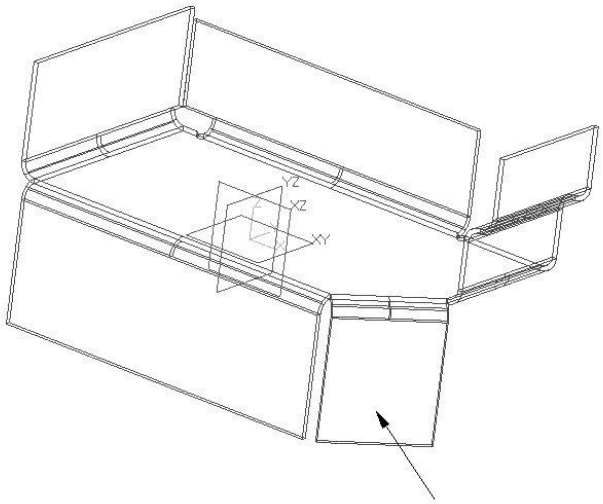
Select the ***Full Face Flange*** command, and select the outer edge as shown.

Select the bend command input option, and choose the **S-bend** option.

Use **25** for the flange length.Use **5** for the bend radius.







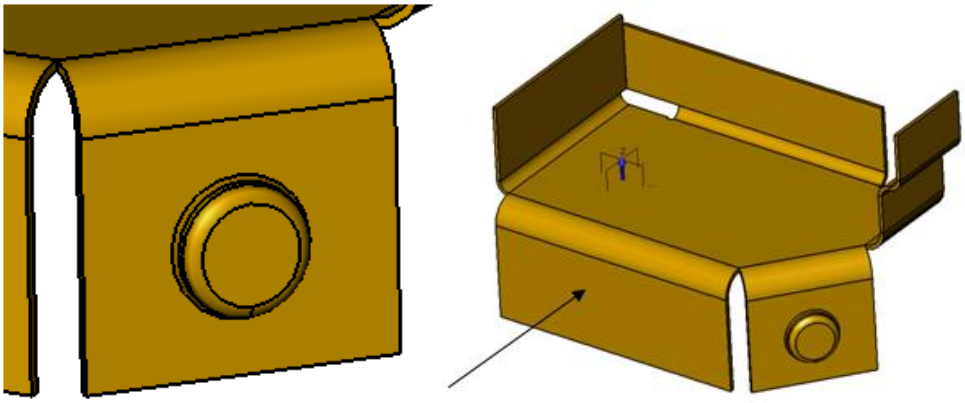
Chapter 7 Adding a Dimple

Select the ***Dimple*** command  . For the flat plane select the outer face indicated above.Middle-click to create the profile.

For the sketch plane select the same face.

Create a circle located at the plane origin (**0,0**) with a radius of 10.

Exit sketch your part should now look like the training image.

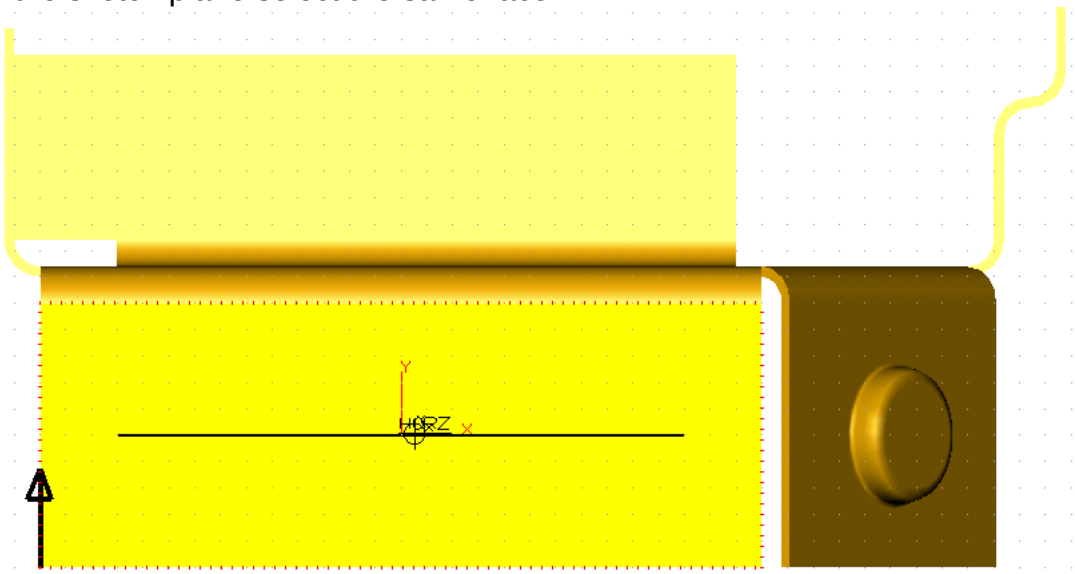


Chapter 8 Adding a Louver

Select the ***Louver*** command. 

For the flat plane select the outer face as indicated above.
Middle-click to create the profile.

Pick the edge indicated for the **Up direction** of the sketch as shown below.
For the sketch plane select the same face.

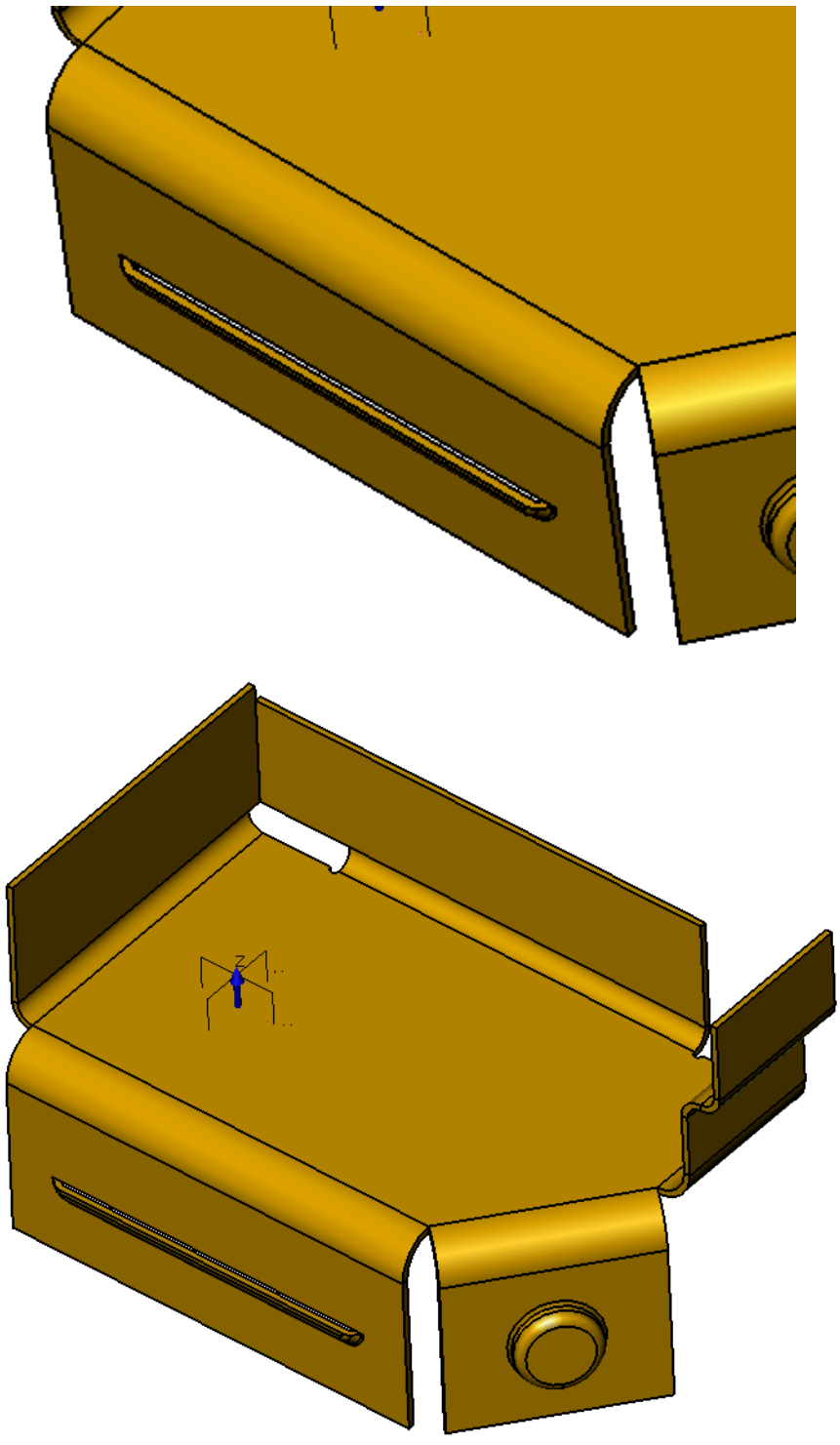


Note – Switching to shaded display mode makes selecting faces easier.

Create a horizontal line that intersects the plane origin.

Pick **OK**.

the sketch.



Chapter 9 Cutting through a Bend while Folded

Now that the basic design is complete, we need to cut two holes in the part.

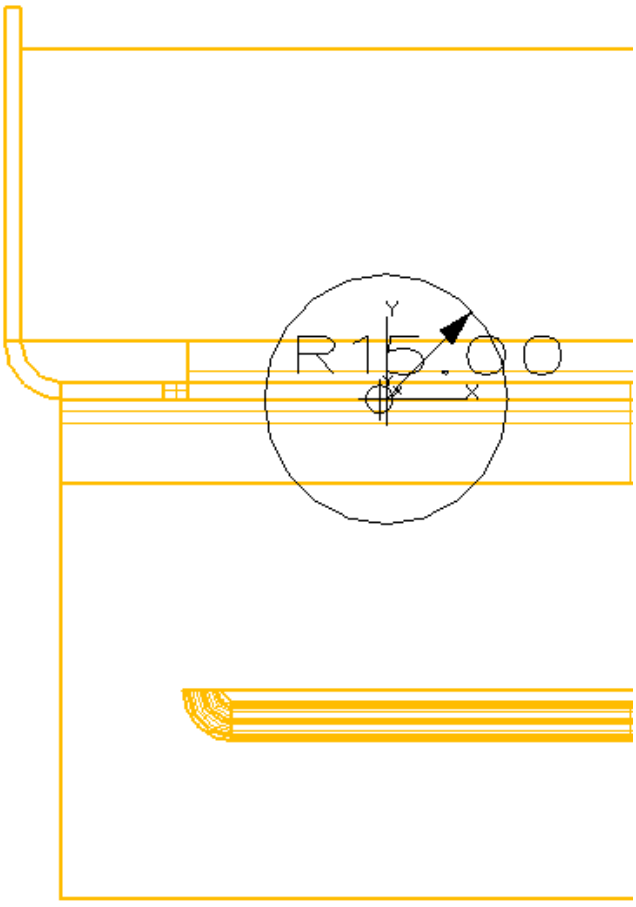
The first cut will be in the folded state. The second while in the flat pattern.

Open the Shape Tab.

Select the **Extrude**  **– Remove**  command,

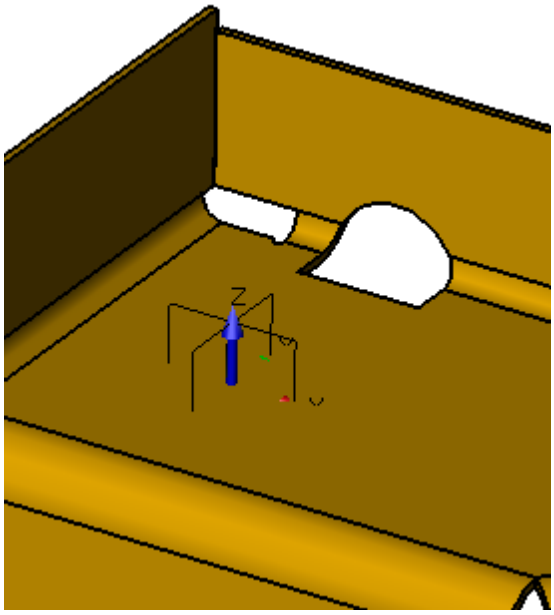
Middle click to create a sketch on the **XZ** plane.

Create a circle as shown below.



Exit the sketch.

Complete the Extruded cut using **-50** for the start point, and **-100** for the end point.Your part should now look like the training image.



When we created the partial face flange, we just estimated to position of the flange. If yours is too close to the cutout then you may want to edit the history.

Right-Click on the "**AddFlange2"** command, Pick **Redefine**.Or Edit->History Operation-> **Redefine**

Pick the button next to **Edge**.

Left-Click and pick **Unpick all**.

then repick the start and end of the flange.

Yours should now look like the image above.

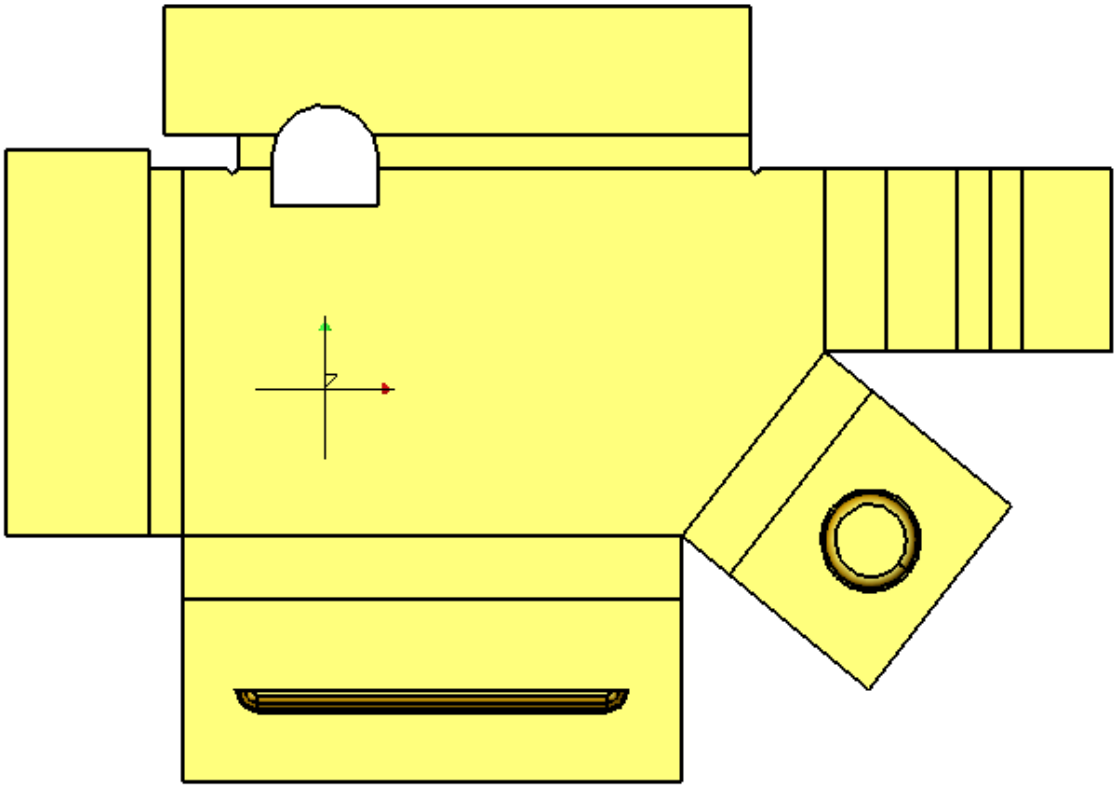
Chapter 10 Unfold Part

Select the **Unfold** command. 

For the shape to unfold, select anywhere on the part.

Pick up the underside as **Stationary.**

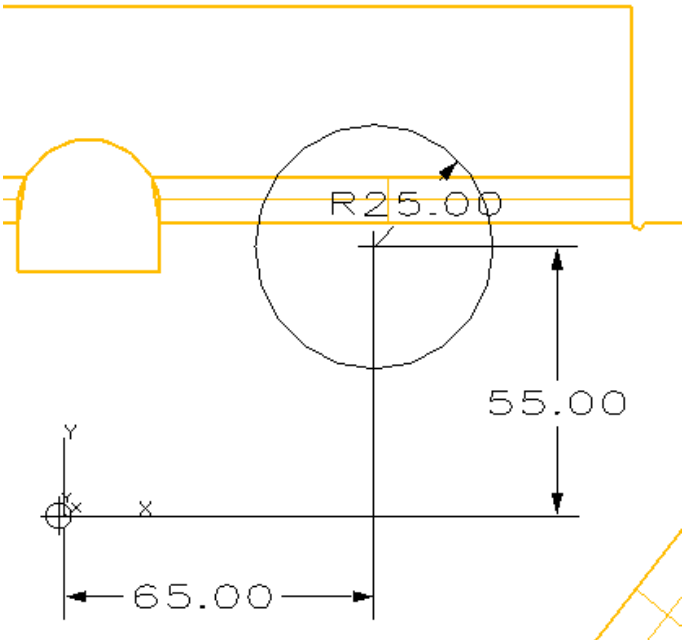
Pick **OK** to accept the defaults for the rest of the command.

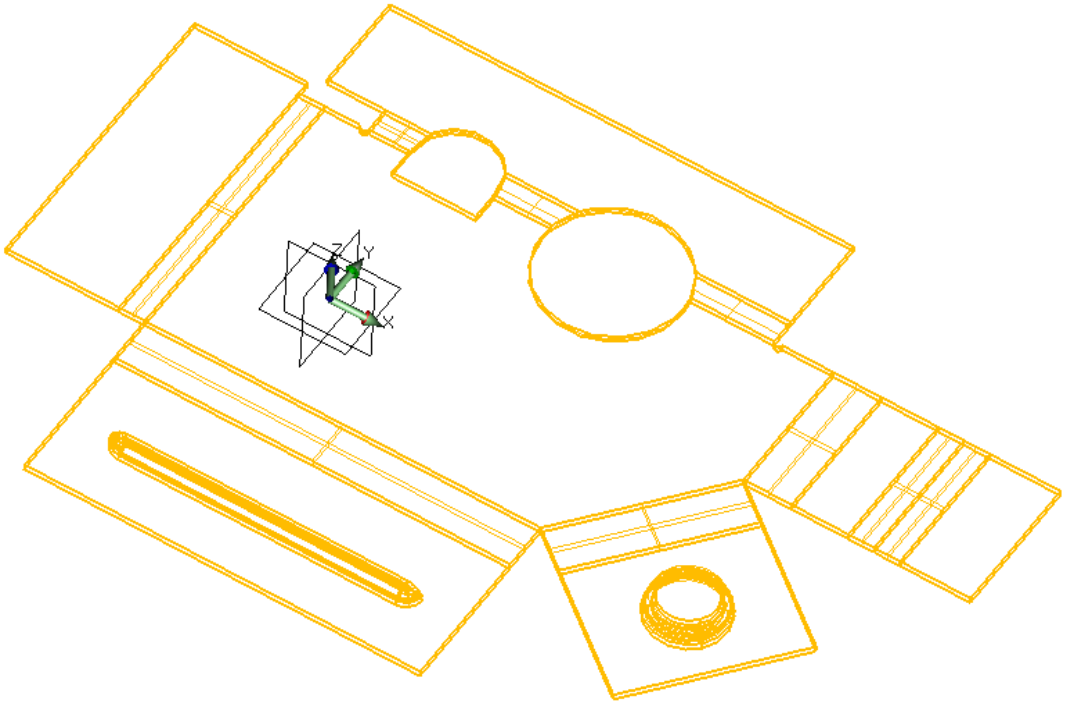


Your flattened part should now look like the training image.

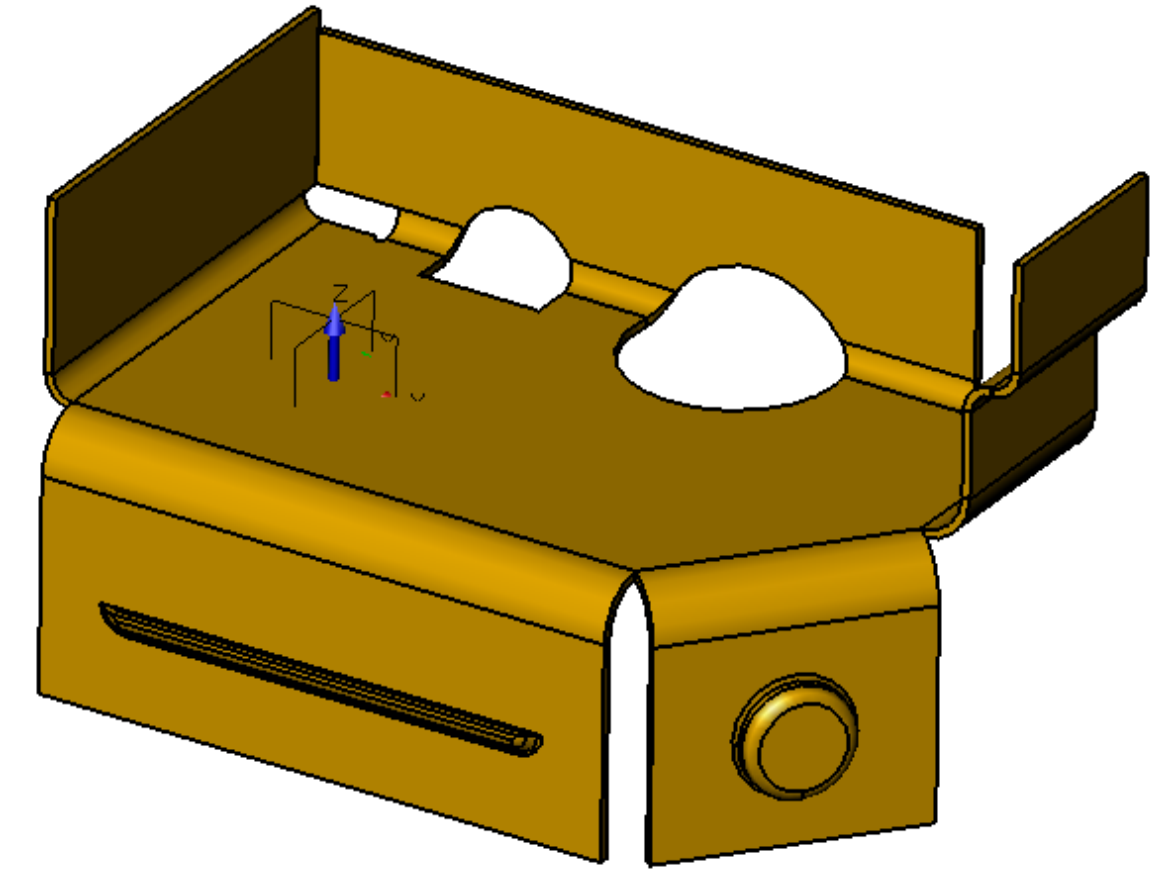
Chapter 11 Cutting through a Bend while Flat

Still in the **unfolded** state, select the **Extrude - Remove** command with the sketch on the **XY** plane.Create a circle as shown below. Complete the Extruded cut





Chapter 12 Fold Part



Your completed part should now look like the training image.

Chapter 13 Editing Bend information

NOTE: The history saves the bend radius attribute with each flange you add.

See the set of commands behind the **Mark Bend** command if you need to edit existing bend values such as:

Bend Angle
Bend Radius
The stationary face during fold and unfold operations.

Mark bend is a powerful tool that allows you to identify bends on imported data. The radius is derived from the geometry. All you have to do is **Mark** the inside faces as a bend radius and unfold.

On a ZW3D created part you can also edit the history.

Right-click on **AddFlange2** and pick **OPEN/CLOSE**

You can **Right-Click EDIT** the
Angle,
Length,
Position: 0= Inner, 1= Outer, 2= Natural
Relief: 0 = Rectangular, 1 = Obround, 2 = none
Radius
K-Factor
Flat Tolerance is not used at this time.

Let's change the relief to rectangular.
Change the relief value as 0.

When you hit OK the rest of the history will play to the end.

Chapter 14 Displaying stages of folding on a drawing

The ZW3D software not only allows you to fold and unfold the entire model but you can specify which flanges to fold and save the current state of the model to be displayed on a drawing.

This will allow you to define a process plan for your shop.

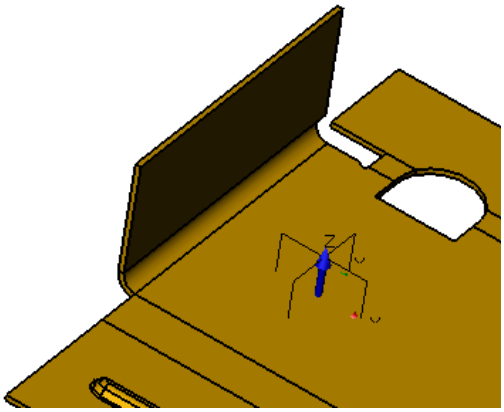
Sheet metal is generally made by folding each flange one at a time. So let's start defining the process plan.

Unfold the part.

Right-Click any blank in history window,then click Record State. Or **Insert > History Operation > Record State** from the pulldown.

You will add a **Record State feature** after each bend.

Fold the part and define the bend face as shown.



Add another **Record State** feature.

Continue doing this for each view you wish to display on the drawing.

You do not have to add a **Record State feature** at the end of the history.

Each **Record State** feature makes a copy of the geometry at that state. On the drawing sheet we can create a view displaying that state of the part.

A shortcut to creating a drawing sheet for the current object is to simply **Right-click** and select **2D Sheet** from the menu.

Starting in version 14.01 you will be prompted to **Select a template**.

There are some templates in the Select a template .For example,[no template] , A0(H)、A1(H)、A2(H)、A3 (H)、A40(H) and so on.

Here we select A3(H) template.

The drawing sheet will be automatically named the same as the part with **_2D** appended to the end.

Or from the Object level create a new drawing sheet with the appropriate title block template.

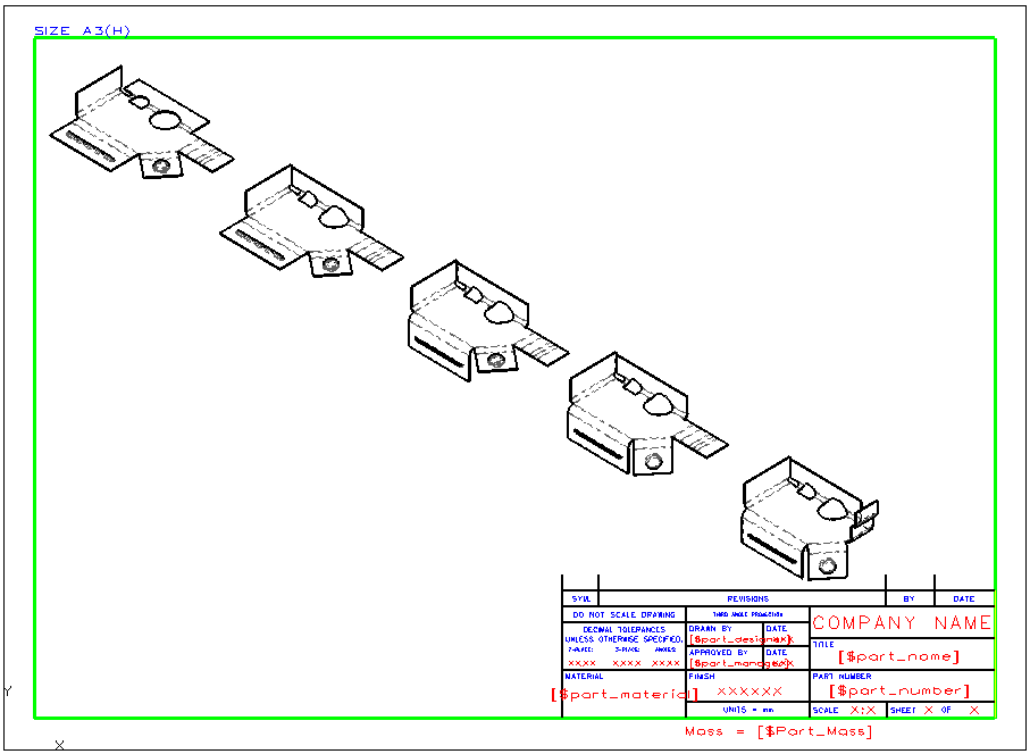
Cancel the **standard layout** command that comes up automatically.

Create a standard view  . Pick the part. Pick an orientation like Isometric.
Use custom scale x/y=1:5

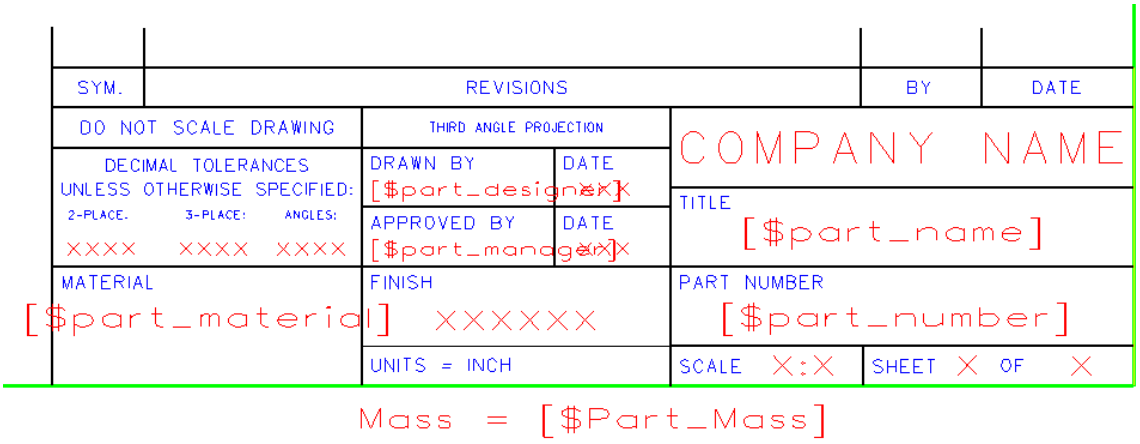
Pick the Record State from the **Advanced** tab.

Repeat this procedure creating a view for each **Record State feature** you defined.

Remember there is no **Record State feature** at the end of the history so for your last view you do not need to select the Advanced tab.



If you are using the default ZW3D Template file you will notice the special formatted text in the title block. Example: [$part_number] can be defined at the part level in the **Part Attributes**.



Right-click on one of the views and pick **Edit Part**  .

Then select **Attributes > Part**

The Part Attribute form is used for a number of functions. Some of the entries are linked to the variables on the drawing sheet template. Others are used when creating a BOM (Bill of Materials)

If you look back at the title block you will notice that in addition to the fields filled in above, there is a variable for material.

Use the **Attribute > Material** command to assign this value.

Click on the arrow below **Bundles.Z3** for a predefined list of materials.

Click **Apply** and click on the sheet metal part on the screen.

Middle-click when done picking.Click “OK”

One other variable that you will see below the title block is **Mass**.

Now that you have selected a material you will notice that the density of the material is defined.

Click on **Inquire > Mass Properties**

Pick the part.

Pick the "**Record Inquired Information**" in the popup dialog. Pick the Ok and close the dialog.

You can view the variables just created by selecting **Tools > Variable Browser**

On the Variable Bbrowser, you can see these messages as follows:

Part_Area=9.3383.2

Part_Volumen=90043.1

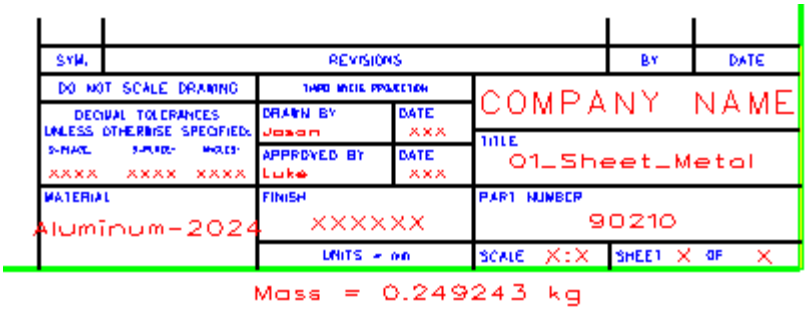
Part_mass=0.249243

You are now ready to go back to the drawing sheet.

ZW3D knows that changes have been made that affect the drawing. You are asked if you want to regen the views.

Click "Yes" .This regen not only updates the views but if you had tables such as BOM or Hole Tables they would also be regenerated.

Embedded text variable are also regenerated. See the title block.



Part_name

Part_number

Drawn by

Approved by

Material … are all populated with the information provided in the **Attributes**.

CONCLUSION:

ZW3D hopes that this lesson gave you a good understanding of our Sheet Metal package.

LIMITATION:

Currently, your part must be designed in the FOLDED or BENT state.