ZW3D from Entry to Master Tutorial

Sheet Metal

ZW3D™ V2023 From Entry to Master Sheet Metal

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Foreword

In this tutorial, we provide various case studies, which are from easy to difficult and combine theory with practice. We hope to improve users' 3D CAD/CAM skills and techniques with ZW3D.

The tutorial bases on our technical engineers' years of experience in the industry and ZW3D, which is the fruit of a lot of efforts and wisdom. We sincerely hope that the tutorial will do help to you, and your precious advice on it is highly welcomed.

There are three series for this tutorial: ***Primary Tutorial***, ***From Entry to Master Tutorial***, and ***Advanced Tutorial***. From easy to difficult, they offer a step-by-step learning process that can meet different user needs.

Primary Tutorial series is for users who have little or no prior 3D CAD/CAM experience. If you are green hands of 3D CAD/CAM software, or if you are a new user of ZW3D, we recommend that you get started with this tutorial. Here you can learn the basic knowledge and concepts of ZW3D, rapidly master the simple operations and workflows of ZW3D, and practice simple cases.

From Entry to Master Tutorial series is for users with basic know-how of 3D CAD/CAM software. If you have experience in 3D CAD/CAM software and want to master common functions of ZW3D, we suggest that you start with this series. Here you can dig deeper into the functions and master more operations of ZW3D.

Advanced Tutorial series is for users with practical experience in 3D CAD/CAM software. If you hope to have a comprehensive command of ZW3D and get the complicated operations done independently, you can choose to learn this series. Here you can learn to use the software more flexibly and get rich experience to increase your efficiency.

What you are learning is ***ZW3D From Entry to Master Sheet Metal***, a master tutorial.

Thanks for being our user!

The ZW3D Team

Contents

1 ZW3D Sheet Metal General Introduction

Key Points:

- ✧ Familiar with ZW3D sheet metal functions
- ✧ Create sheet metal base, flange, corner, forming feature, etc.
- ✧ Covert non-sheet metal part
- ✧ Fold / unfold a sheet metal part

ZW3D Sheet metal module provides users a full set of tools to finish various sheet metal designs.

Besides ZW3D sheet metal also offers convert tools to convert non-sheet metal part into sheet metal part and supports unfolding it correctly. It can also create the corresponding drawing after unfolding the sheet metal part with bend information for workshop as well.

Following is the tool bar for sheet metal tools in ZW3D:

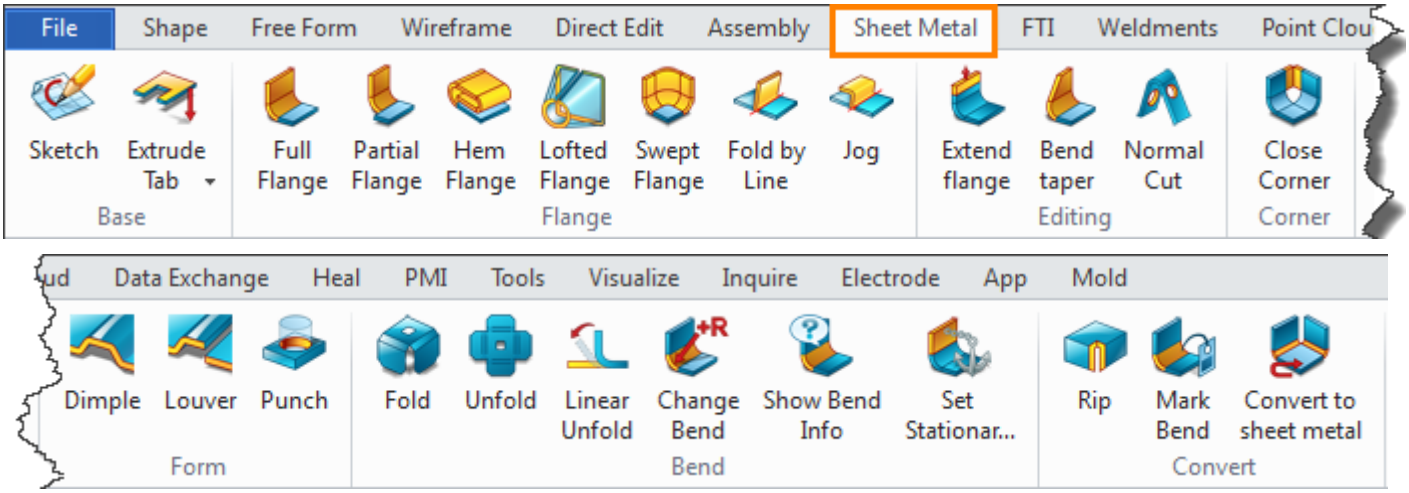


Figure 1 ZW3D Sheet Metal Module

Next let's take a look at the details about each command in this module and related functions.

1.1 Set up Sheet Metal Attribute

It is necessary to set up a default attribute for sheet metal such as bend radius, K-factor etc. as follows:

Tool ribbon tab->Attributes->Sheet Metal

STEP 01 Choose the Sheet metal attribute button

STEP 02 Set up the parameters as follows

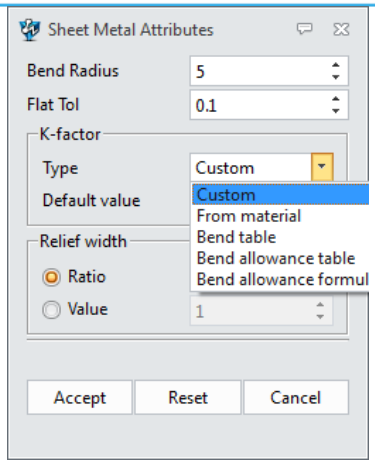


Figure 2 Sheet Metal Attribute

For K-factor, it is allowed to gain the value in different ways:

From the material table

From the Bend table

From Bend Allowance table

From Bend Allowance formula

For details on how to customize K-factor table, please refer to the table in installing path. (Such as ***C:\Program Files\ZWSOFT\ZW3D 2017 Eng (x64)\SMD_K_FACTOR_TABLE***). As shown in Fig3.

Open any of them, then you can get the details on how to customize the corresponding table.

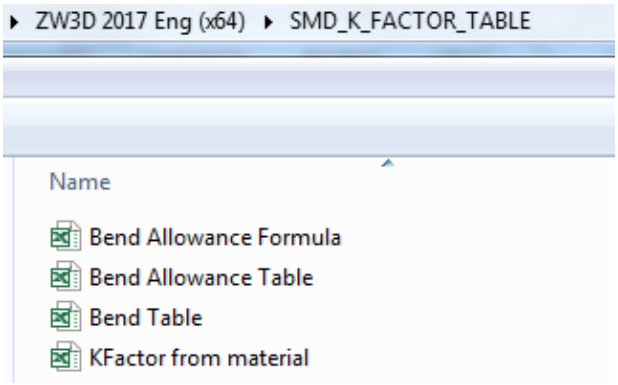


Figure 3 K-Factor Table

1.2 Create Sheet Metal Base

Base Flange includes 2 different commands: Extrude Tab and Extrude Flange command.

1.2.1 Extrude Tab

Sheet Metal ribbon tab->Base->Extrude Tab

This command can be used to create both base and tab.

Extrude Base

STEP 01 Create a sketch as follows:

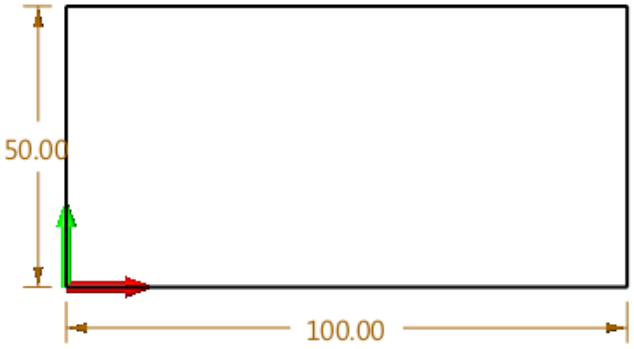


Figure 4 Sketch for extrude base

STEP 02 Pick ***Extrude Tab*** command and then choose the sketch as input profile, after setting up the thickness as follows:

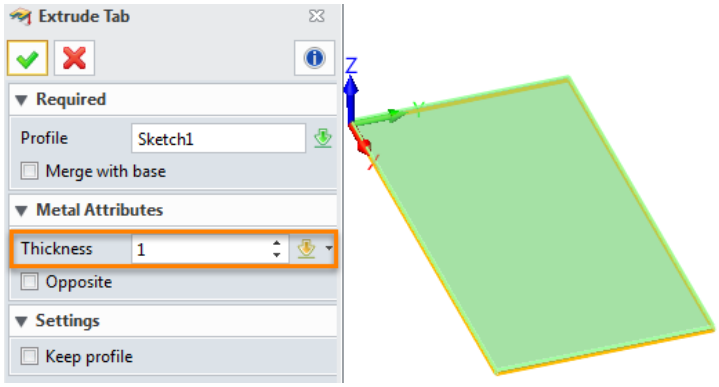


Figure 5 Extrude base

Then we can add one more tab based on it.

Add a tab

STEP 01 Create a sketch as shown in Figure 6. And then exit the sketch.

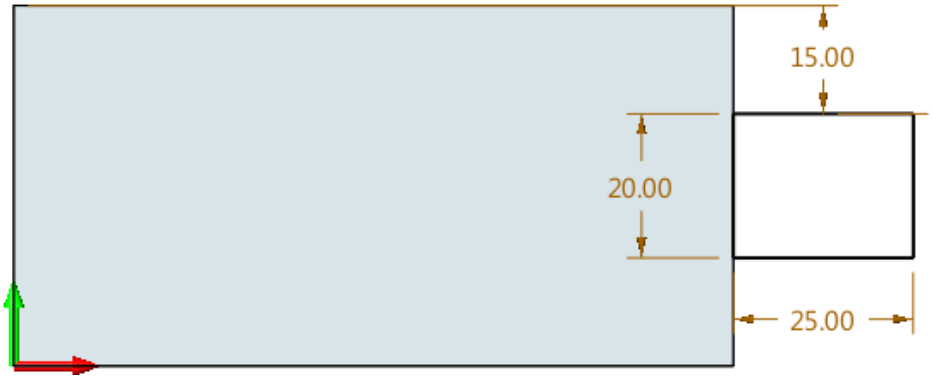


Figure 6 Sketch for merged tab

STEP 02 Repeat the ***Extrude Tab*** command to merge a new tab into the base as shown in Figure 7.

STEP 03 Save it as ***01_Tab_Merge***.

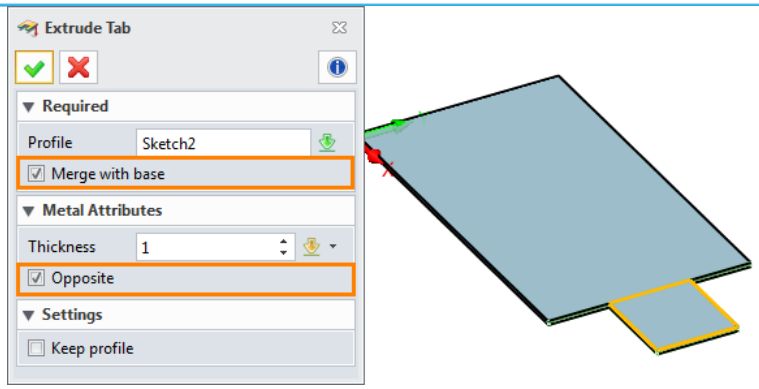


Figure 7 Merge Extruded Base

1.2.2 Extrude Flange

Sheet Metal ribbon tab->Base->Extrude Flange

It is used to create a flange by extruding sketch. (Support both open and enclosed sketch).

STEP 01 Create an open sketch as shown in Figure 8.

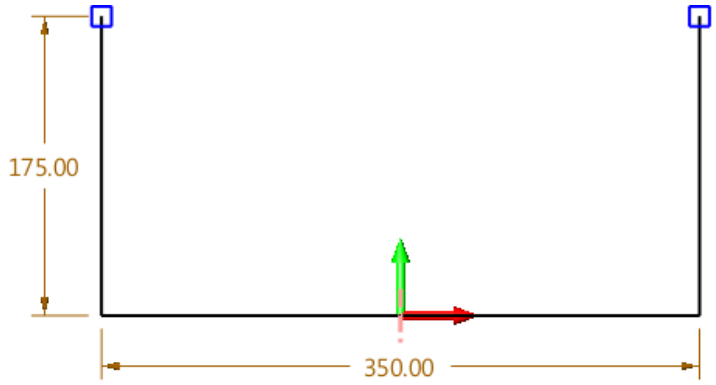


Figure 8 Sketch with Open Profile

STEP 02 Extrude flange as shown in Figure 9.

STEP 03 Save it as ***02_Extrude Flange Open***.

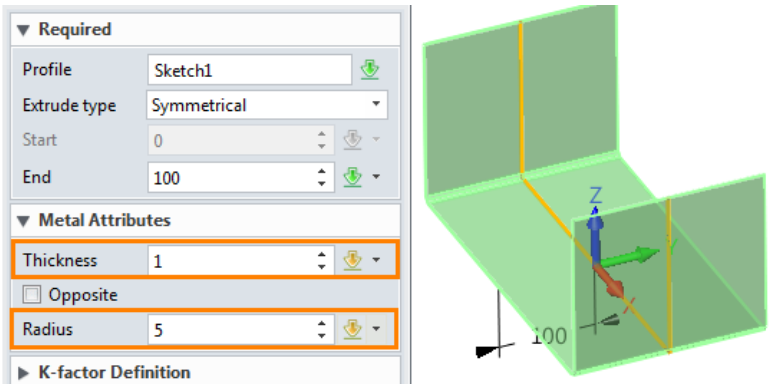


Figure 9 Extrude Flange

Notes : *The Radius option will stop working if the sketch has radius in the corner.*

Next let's design a new flange with a closed sketch profile.

STEP 01 Create a sketch---closed profile as follows:

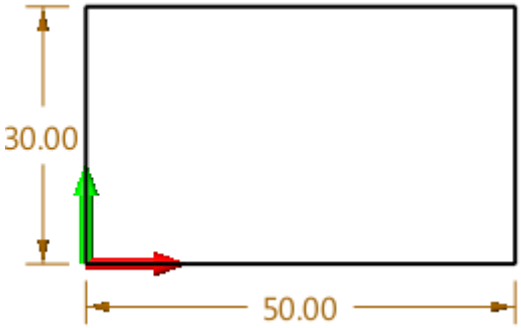


Figure 10 Sketch with Closed profile

STEP 02 Extrude the closed sketch by ***Extrude Flange*** command as follows.

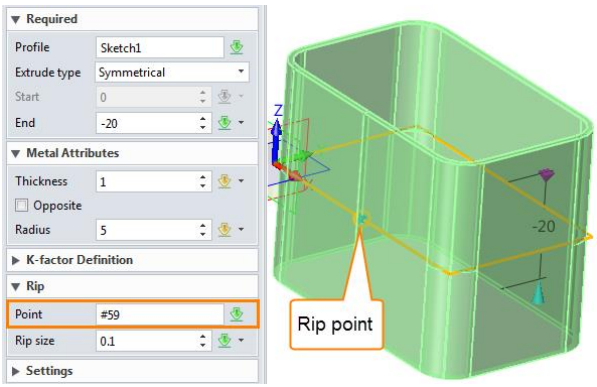


Figure 11 Extrude flange by closed sketch

Notes: *If the Rip option is blank, then it will create a closed shape that cannot be unfolded.*

STEP 03 Unfold this part by the unfold command as follows.

Sheet Metal ribbon tab->Bend->Unfold

Select the shape and define the stationary face to unfold the part. The unfold result is shown in Figure 13. After that, save it as ***03_Extrude Flange_Closed***.

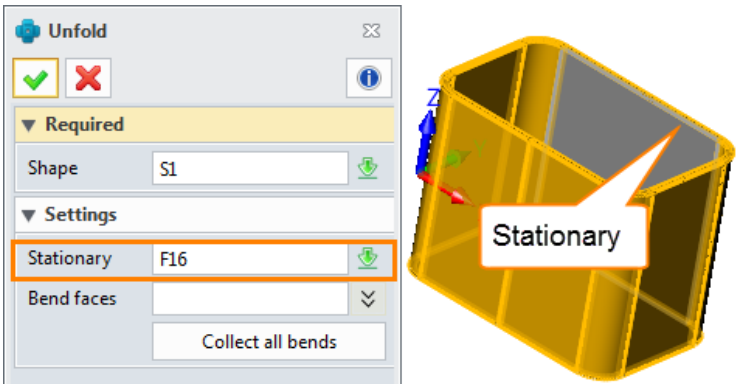


Figure 12 Unfold the Part

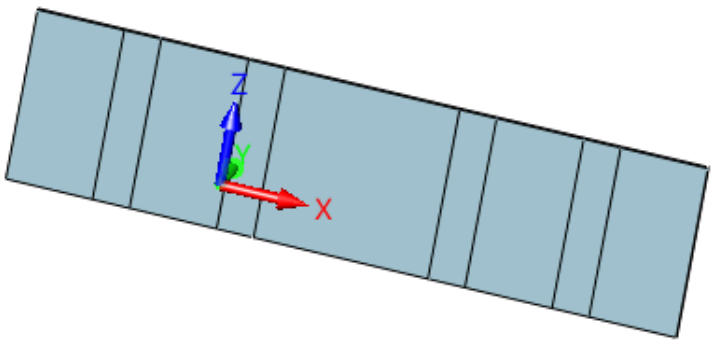


Figure 13 Unfold Result

1.3 Flange Creation

1.3.1 Full Flange

Sheet Metal ribbon tab->Flange->Full Flange

This command can create full flange and S type flange.

STEP 01 Create a base flange as follows.

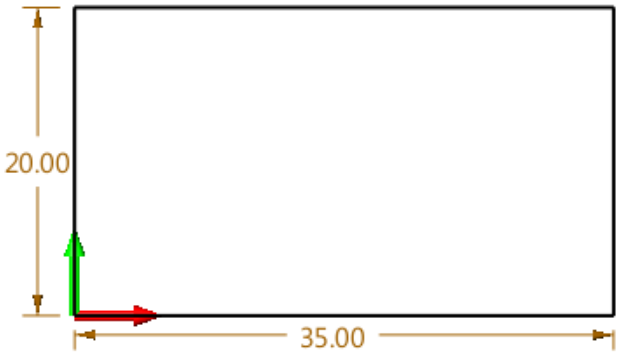


Figure 14 Create sketch for base

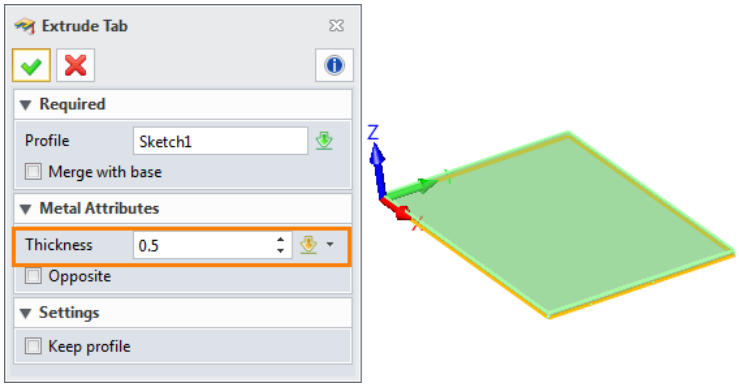


Figure 15 Create base

STEP 02 Create a full flange at an edge as follows.

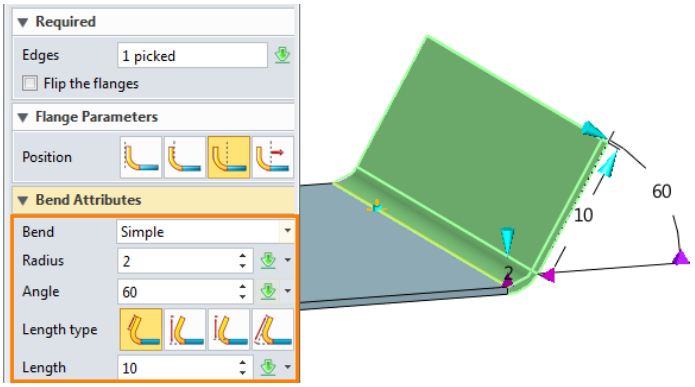


Figure 16 Create full flange

STEP 03 Change the bend type into S bend as follows.

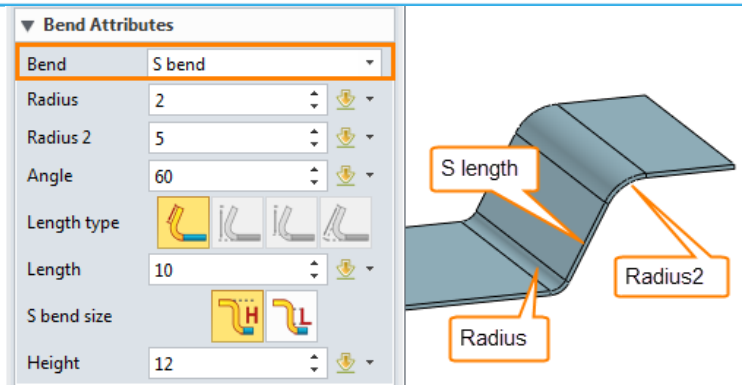


Figure 17 Create S bend

STEP 04 Add flanges on other edges, meanwhile set the ***Close corner*** option as follows.

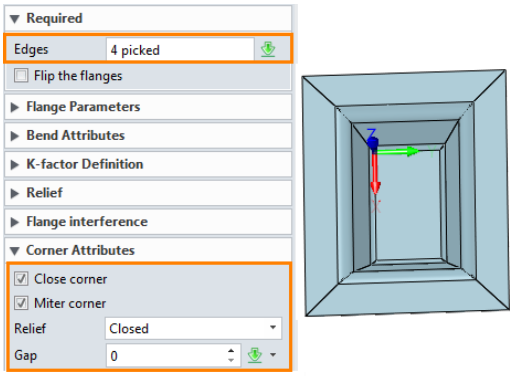


Figure 18 Close corner

STEP 05 Add gap in corner.

STEP 06 Save this file as ***04_Full Flange_S bend***.

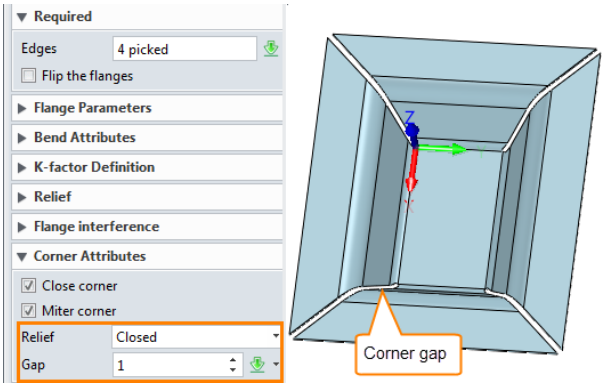


Figure 19 Add gap for corner

1.3.2 Flange with profile

Sheet Metal ribbon tab->Flange->Flange with Profile

STEP 01 Create a basic tab first, then select the edge that you want to create a non-standard shape flange, then modify the parameter as follow, and click the ***Edit profile*** button.

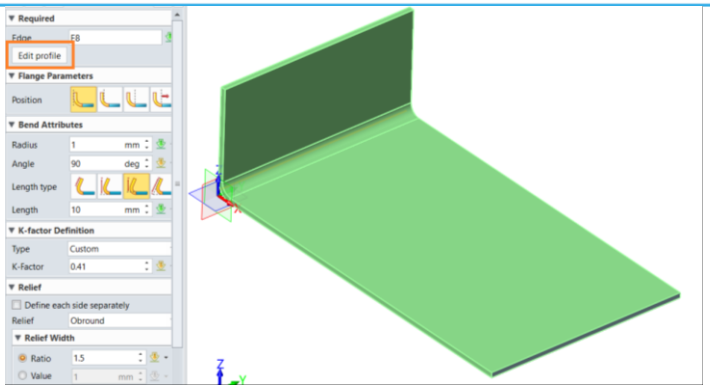


Figure 20 create basic flange shape

STEP 02 Modify the sketch shape as follow, then exit the sketch editor.

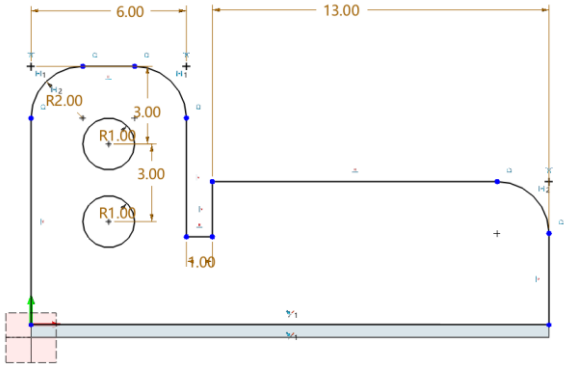


Figure 21 Modify the flange shape

STEP 03 You will get the result as follow:

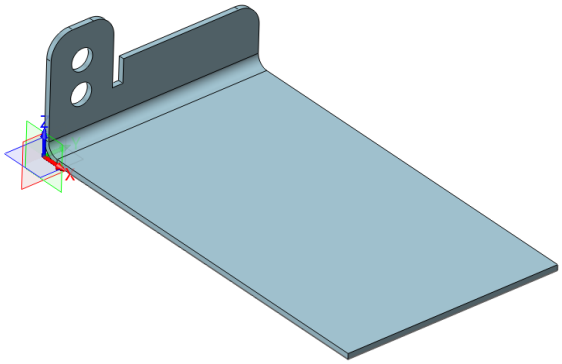


Figure 22 Result

STEP 04 Save this file as ***05_Flange with Profile***.

1.3.3 Partial Flange

Sheet Metal ribbon tab->Flange->Partial Flange

STEP 01 Create a base as the one created in full flange.

STEP 02 Create a partial flange on one of edges as shown in Figure 23.

STEP 03 Sav this file as ***06_Partial_ Flange***.

Width type:

1) Start-Width :

Start : set up how far the flange's start point away from the edge's start point

Width: set up the partial flange's width which starts from the start point

2) Start-End:

Start: set up how far the flange's start point away from the edge's start point

End: set up how far the flange's end point away from the edge's end point

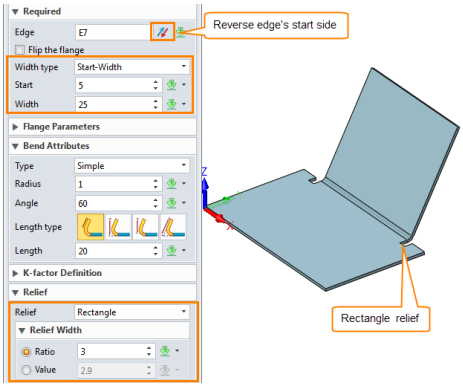


Figure 23 Create Partial Flange

1.3.4 Hem Flange

Sheet Metal ribbon tab->Flange-> Hem Flange

STEP 01 Open the file ***Sheetmetal Function.Z3->07_Hem***

STEP 02 Choose Hem Flange to create hem flange as shown in Figure 24.

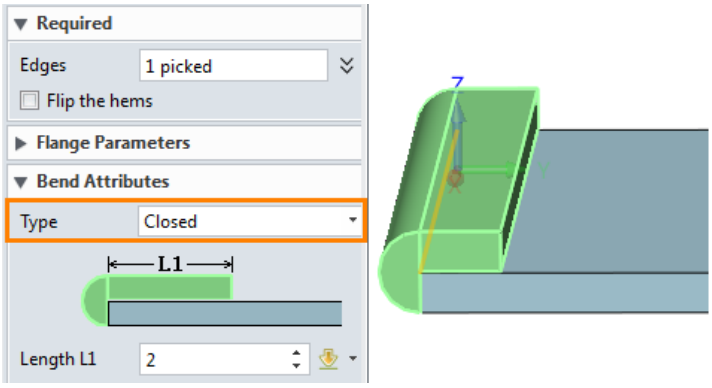


Figure 24 Hem Flange

STEP 03 Try other bend types of Hem.

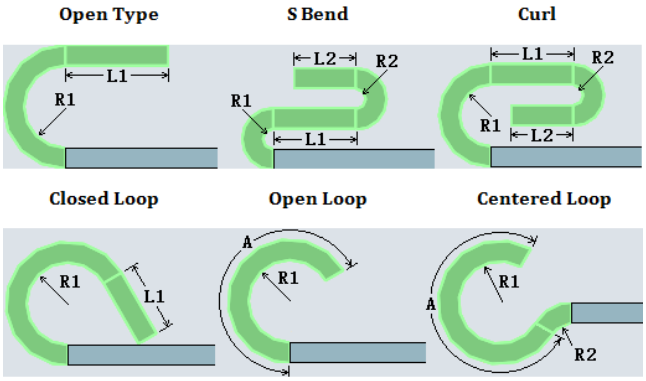


Figure 25 Hem Flange Type

STEP 04 Create miter hem flange as follows.

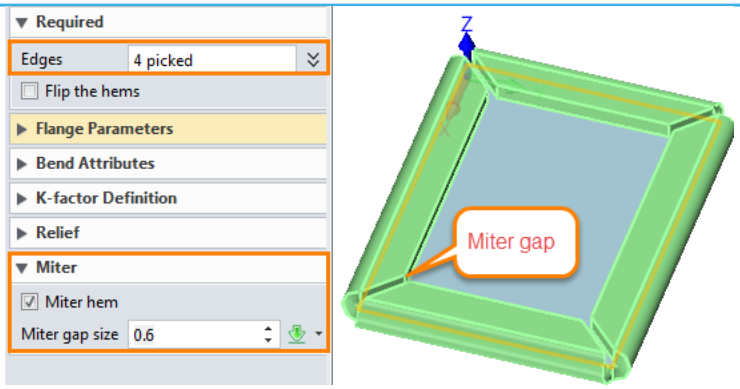


Figure 26 Miter Hem Flange

1.3.5 Loft Flange

Sheet Metal ribbon tab->Flange->Lofted Flange

Supports to create circle to circle, circle to rectangle, rectangle to rectangle type of flanges.

STEP 01 Create 2 sketches with open gap as follows:

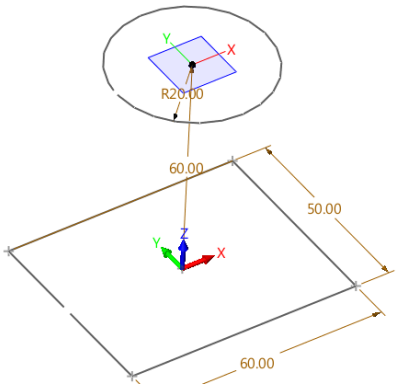


Figure 27 Sketch for Loft flange

Notes: 1) *Make sure the profile only consists of line, arc, if other types of geometry are used, we can convert them into line or arc by command "Convert to Arc/Lines".*

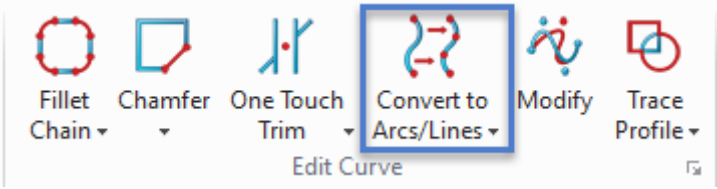


Figure 28 Convert to Arc/Lines

2) *Make sure the profile is open.*

STEP 02 Select ***Loft Flange*** command and then set up parameters, as follows.

Notes: *Make sure the profiles has the same direction as the picture shown.*

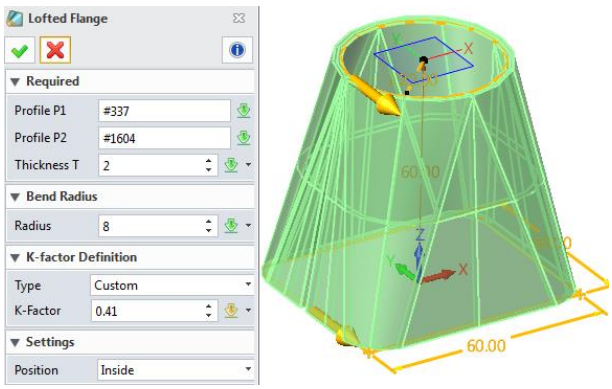


Figure 29 Loft Flange

STEP 03 Unfold it.

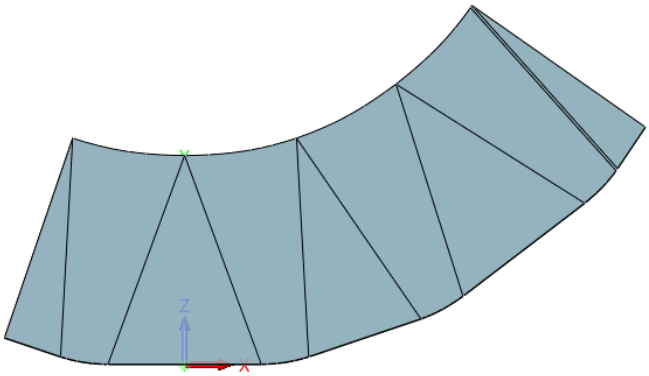


Figure 30 Unfold loft flange

1.3.6 Swept Flange

Sheet Metal ribbon tab->Flange->Swept Flange

Use this command to create swept flange.

STEP 01 Create a base flange as follows.

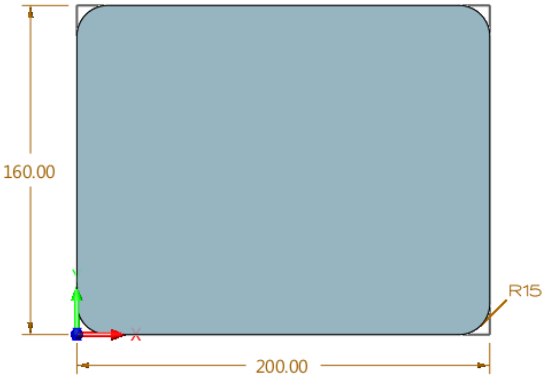
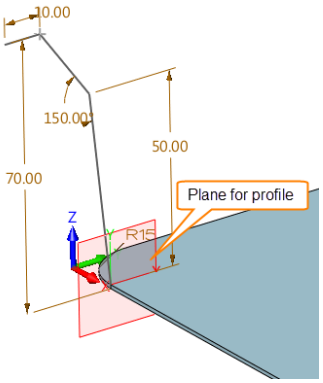


Figure 31 Base for Swept flange

STEP 02 Create a new datum at the end point and then create a new sketch as sweeping profile on it as follows:



Profile for Sweeping

Notes: *The profile's locating plane must be coincident with the end point of the path.*

STEP 03 Choose Swept flange command and then set up parameter as follows.

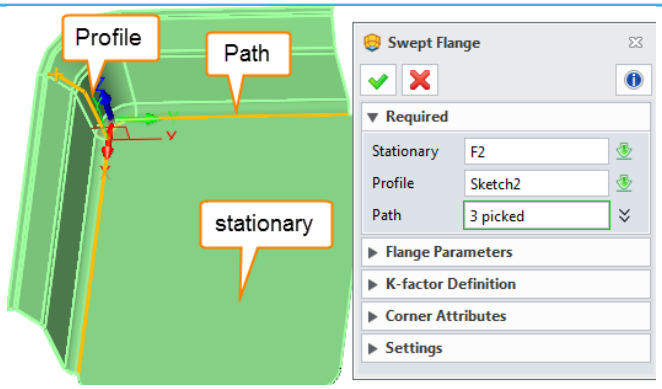


Figure 32 Swept Flange

Tips: *It can pick all the continuous edges in one click by setting the filter as follows.*

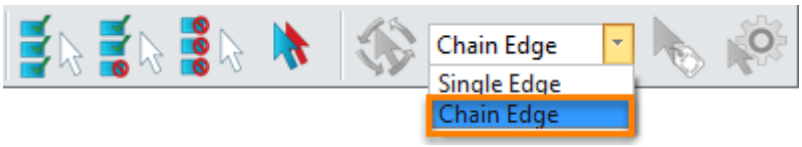


Figure 33 Filter on Pick Tools Mini Bar

STEP 04 Select all of the paths tangent, then we can get the following result.

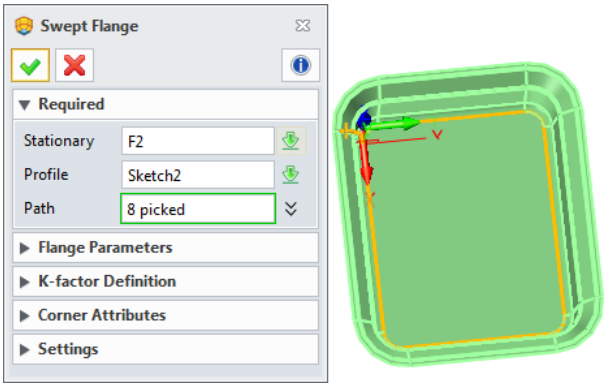


Figure 34 Swept Flange with closed path

STEP 05 Unfold the swept flange.

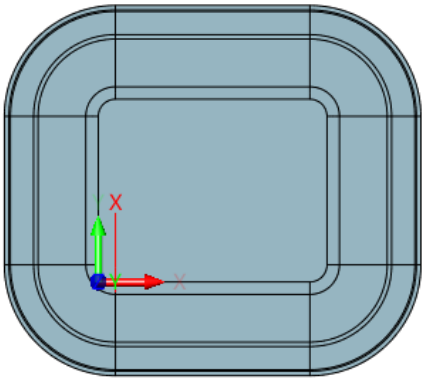


Figure 35 Unfold the Swept Flange

Even if the path is not tangent, it can create swept flange by selecting all of the paths as follows:

STEP 01 Create base as follows:

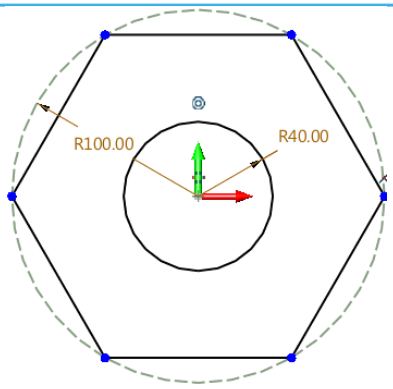


Figure 36 Sketch for Swept Flange Base

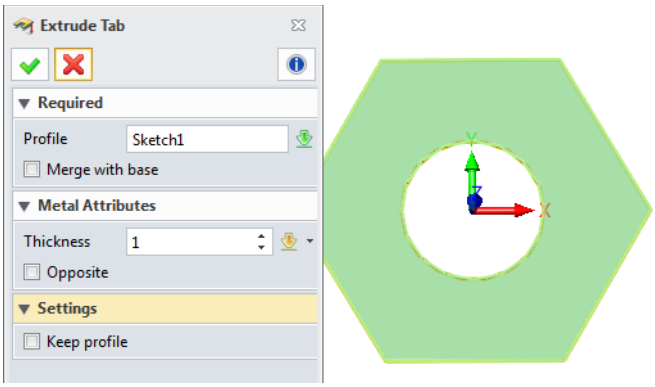


Figure 37 Base for Swept flange

STEP 02 Create profile on assigned datum, as shown in Figure 38.

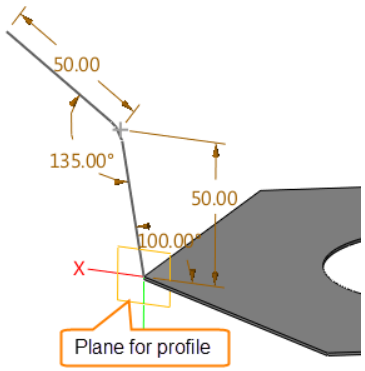


Figure 38 Profile for Swept Flange along discontinuous Path

STEP 03 Create swept flange as follows.

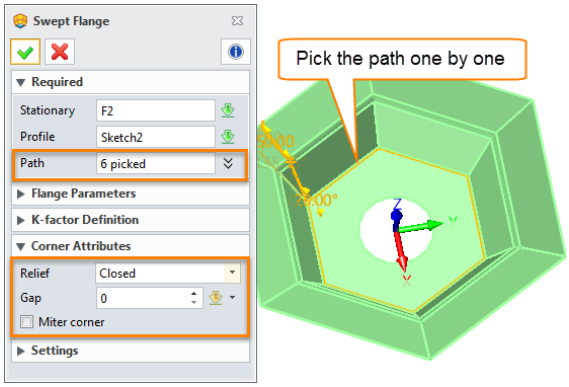


Figure 39 Swept Flange along discontinuous Path

STEP 04 Unfold the swept flange.

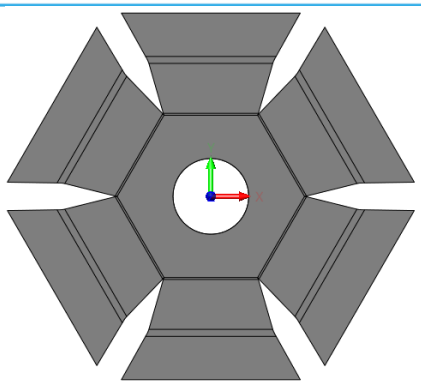


Figure 40 Unfold Swept Flange on discontinuous path

1.3.7 Fold by Line

Sheet Metal ribbon tab->Flange->Fold by Line

Use this command to create bends along a line.

STEP 01 Open the file ***Sheetmetal Functions.Z3->10_Fold by line*** as follows.

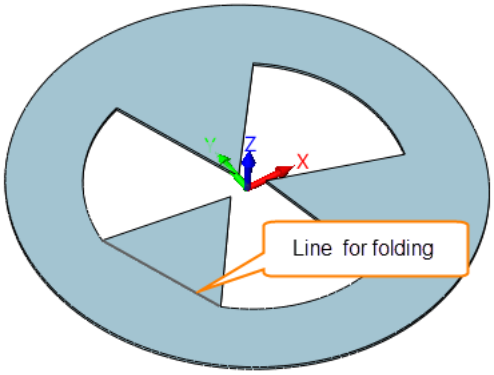


Figure 41 Open File “Fold by line”

Notes: *Current* ***Fold by line*** *command just only supports a line, but non-linear curve.*

STEP 02 Choose the command fold by line and then choose the curve as the fold line as follows.

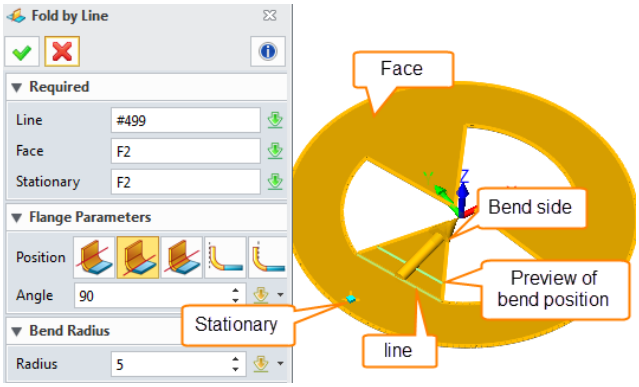


Figure 42 Parameter for Fold by Line

STEP 03 Fold result is shown in Figure 43., then do the same process for other parts.

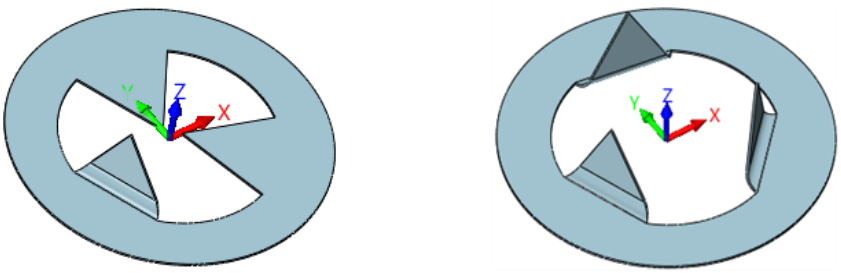


Figure 43 Fold by line Result

1.3.8 Jog

Sheet Metal ribbon tab->Flange->Jog

Use this command to create two bends and two flanges along a line.

STEP 01 Open the file named ***Sheetmetal Functions.Z3->11_Jog***.

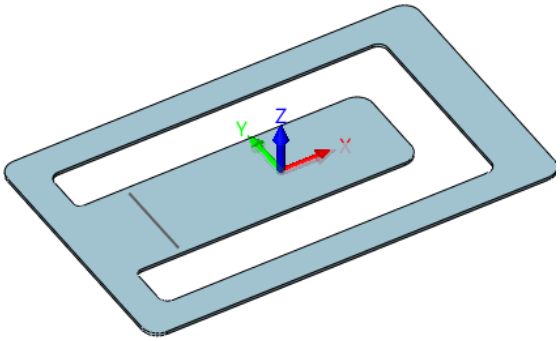


Figure 44 Jog part

STEP 02 Click Jog command, then select the curve to create a jog as follows.

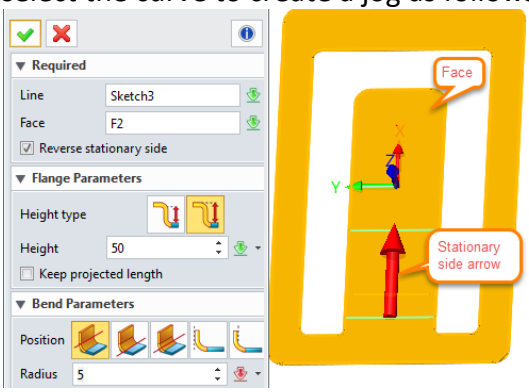


Figure 45 Create Jog

STEP 03 Confirm it and then get the following result.

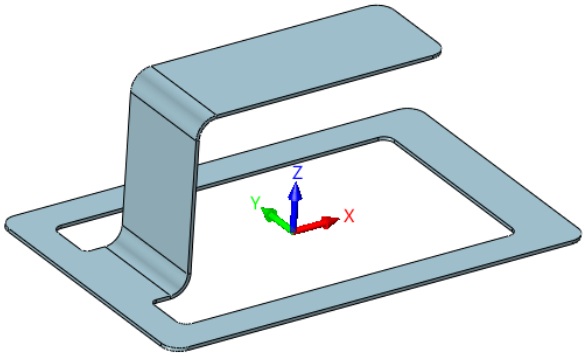


Figure 46 Jog shape

1.4 Editing Tools

1.4.1 Extend Flange

Sheet Metal ribbon tab->Editing->Extend Flange

It is used to stretch an existing tab or flange from picked edges. It would be helpful if you don't want to redefine a flange or an Extrusion tab feature to change the size, or you want to work on an imported/converted sheet metal part that doesn't have any redefine-able features.

STEP 01 Open the file named ***Sheetmetal Functions.Z3->12_Extend flange*** as follows.

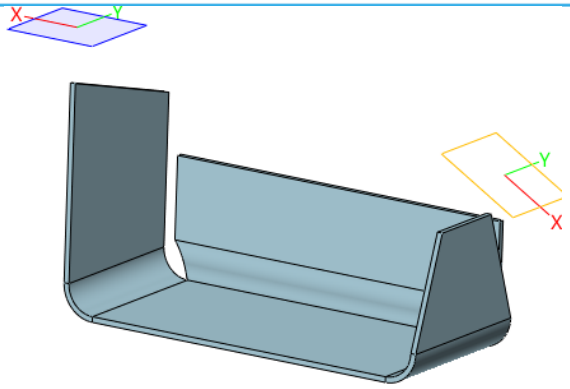


Figure 47 Extend flange file

STEP 02 Choose Extend flange command and then select the edge on the left flange as follows.

a. Extend by distance

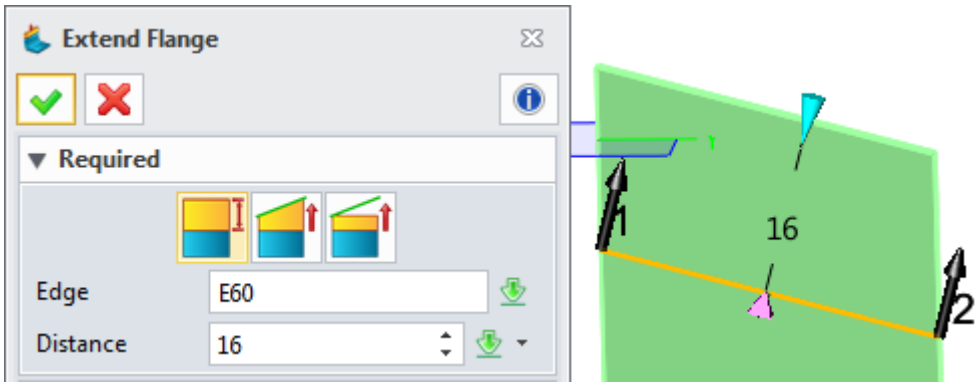


Figure 48 Extend Flange by Distance

b. Extend flange through until intersection with plane

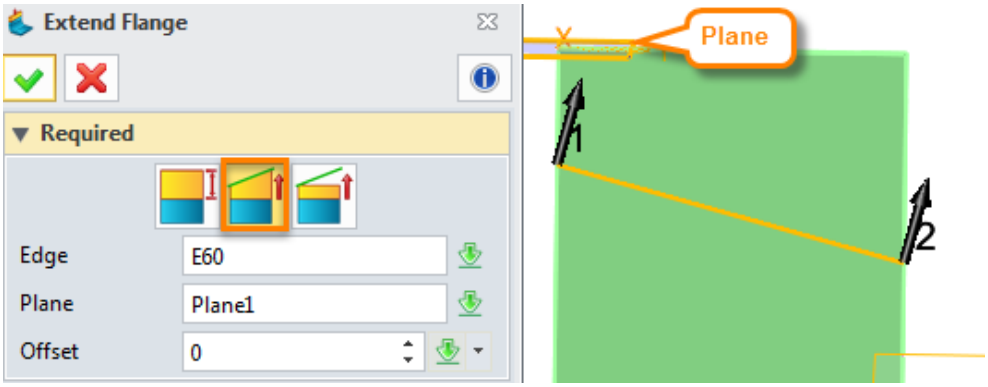


Figure 49 Extend Flange through until Intersection with Plane

c. Extend flange up to selected plane

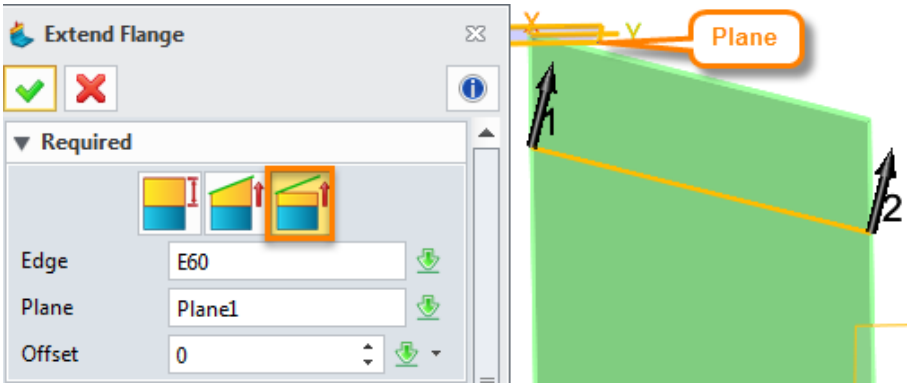


Figure 50 Extend Flange up to Selected Plane

Then let's take a look at the **Extension** option as follows.

1) Along Boundary Edge: Extend the selected edge along the boundary edge

2) Normal to Extended Edge: Extend the flange normal to the picked edge

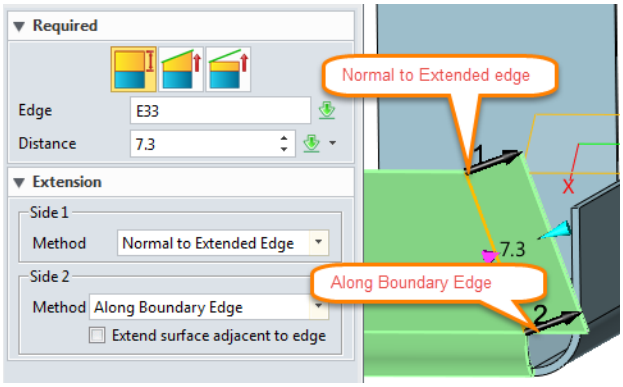


Figure 51 Different Extension Method

Check the option ***Extend surface adjacent to the edge*** to get the following result.

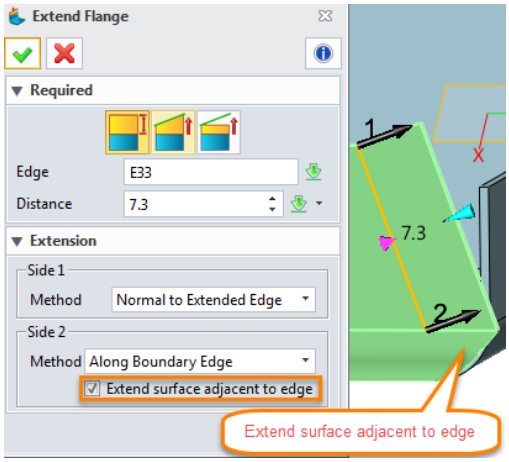


Figure 52 Extend surface adjacent to the edge

1.4.2 Bend Taper

Sheet Metal ribbon tab->Editing->Bend Taper

It is used to miter the flange to change its profile and avoid interfering with other flanges, or make some gap, especially on the imported sheet metal which has no flange features.

STEP 01 Open the file named ***Sheetmetal Functions.Z3->13_Bend Taper*** as follows.

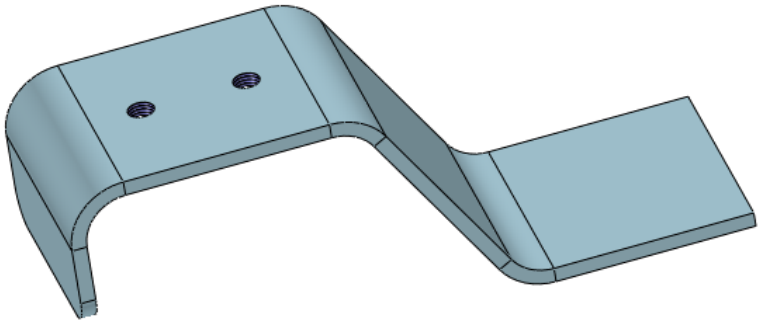


Figure 53 Bend Taper part

STEP 02 Choose the Bend Taper command and then choose the bend face to taper as follows.

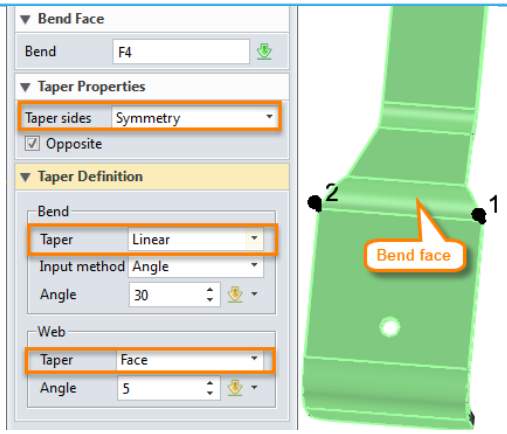


Figure 54 Bend Taper

Bend Taper definition

There are two methods (Linear/Tangent) to define bend taper, as shown in Figure 55. The edge of the bend is tangent to the web when the tangent method is used.

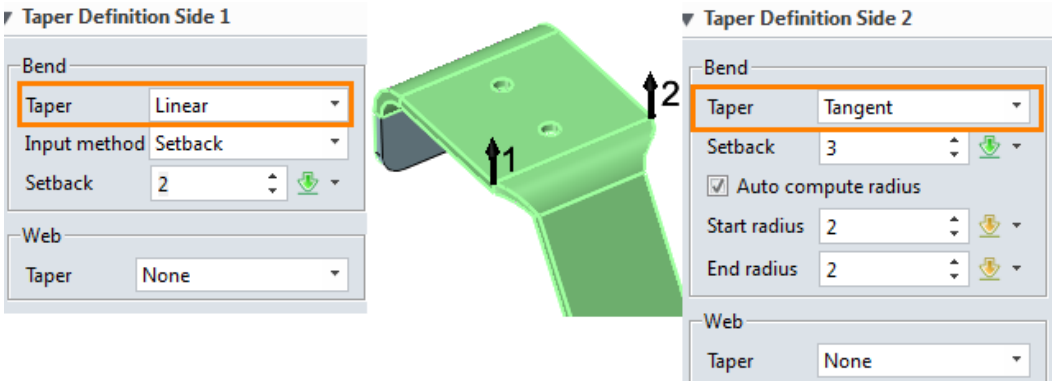


Figure 55 Bend Taper

Web Taper definition

None --- Only the selected bend is tapered.

Face --- The selected bend and the adjacent web is tapered.

Face chain --- The selected bend and all the webs in the chain are tapered.

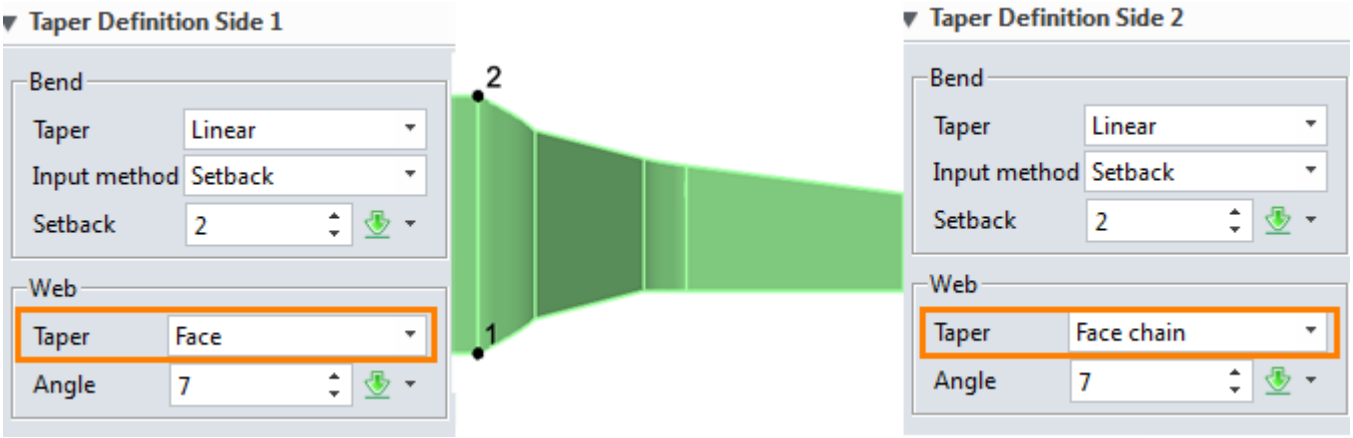


Figure 56 Web Taper

1.4.3 Normal Cut

Sheet Metal ribbon tab->Editing->Normal Cut

It is used to cut material by projecting a sketch onto the shape and then cutting perpendicular to the faces intersected by the projection.

STEP 01 Open the file named ***Sheetmetal Functions.Z3->14_NormalCut*** as follows.

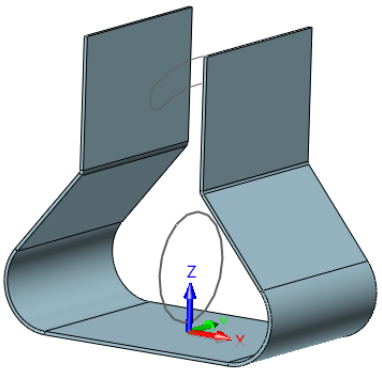


Figure 57 Normal Cut File

STEP 02 Choose Normal Cut command and then select the sketch in this file to cut the material as follows.

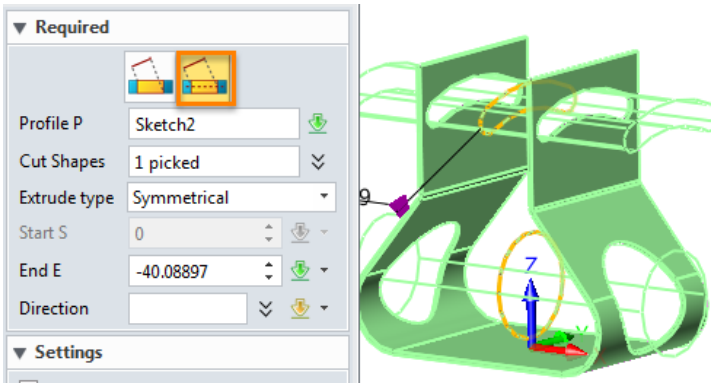


Figure 58 Normal Cut to Remove Material

If you check the option of ***Flip the side to cut*** to keep the opposite side as follows.

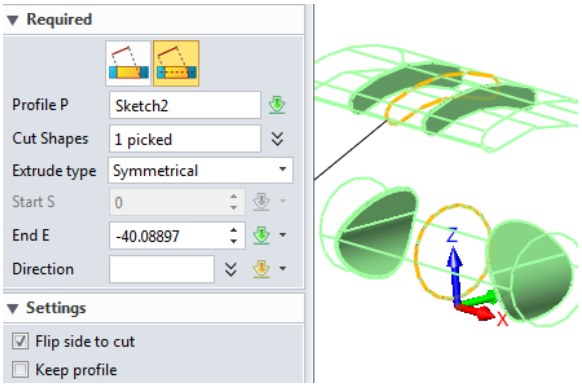


Figure 59 Normal cut to fall the material

Here don't check the option ***Flip side to cut***, and then click ok to get following result.

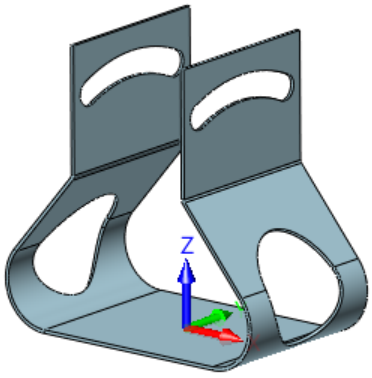


Figure 60 Normal Cut Result

STEP 03 Unfold it to check the cut result as follows:

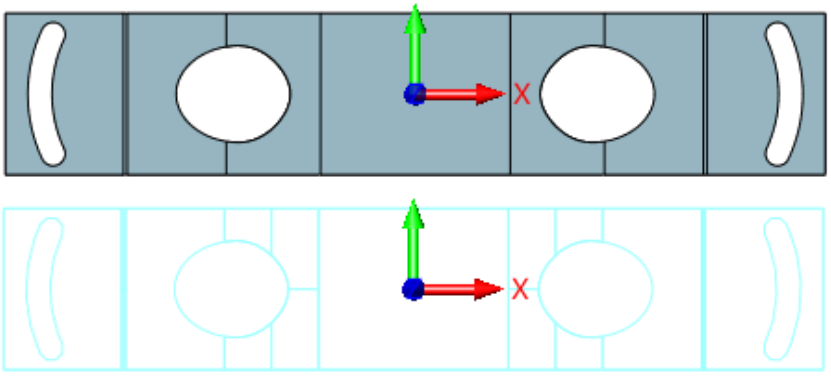


Figure 61 Unfold the Normal Cut Result

Options: 

1) Normal to both side  **:** The profile is projected onto both side faces of the sheet metal part, and then projection areas are merged and used to remove the material on the thickness direction as follows.

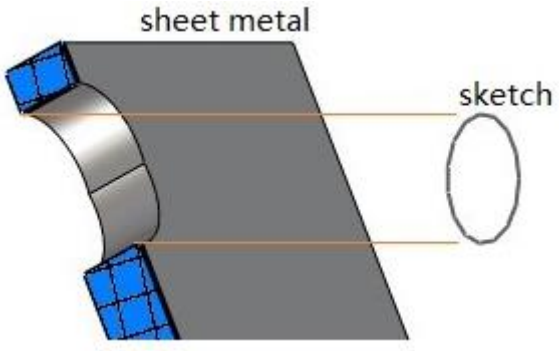


Figure 62 Normal to Both Sides

2) Normal to middle  **:** ZW3D projects the sketch on mid-plane between the near and far side of sheet metal part, and then removes material towards both sides.

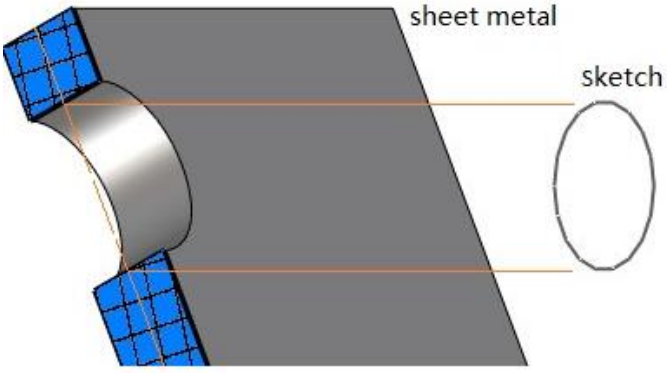


Figure 63 Normal to Middle

1.5 Corner

1.5.1 Close Corner

Sheet Metal ribbon tab->Corner->Close Corner

It is used to extend sheet metal flanges and bend to form a closed corner.

STEP 01 Open the file named ***Sheetmetal Functions.Z3->15_Close Corner*** and activate ***Single Level*** part configuration.

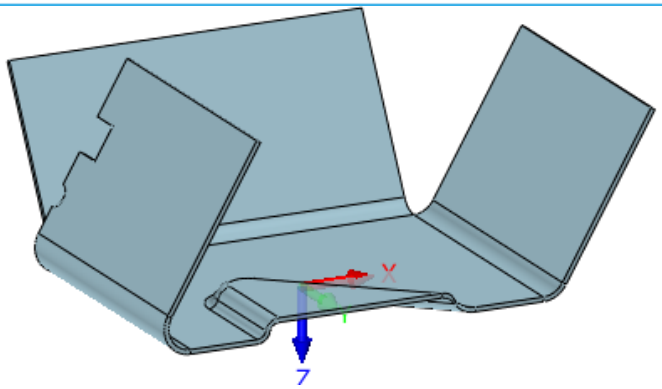


Figure 64 Close Corner Part

STEP 02 Choose Close Corner command and apply the first type-Edge, as shown in Figure 65.

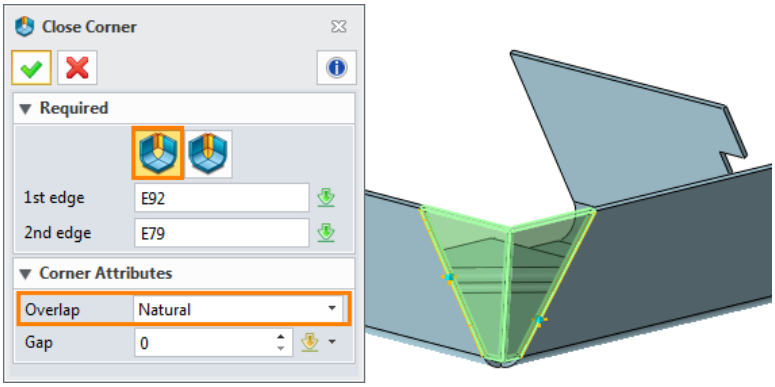


Figure 65 Close Corner by Edge

Notes: *For the 1st edge and 2nd edge, there is no sequence for them. 1st and 2nd edge just affected by the overlap type.*

Corner Attribute

Underlap: The 1st edge will be the inner edge.

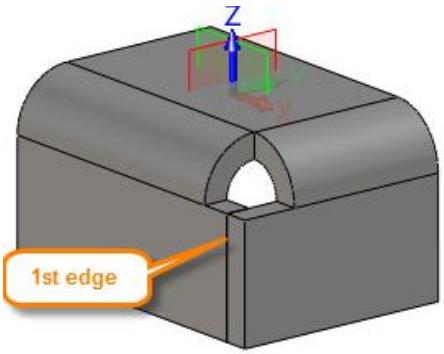


Figure 66 Underlap Flange Corner

Overlap: The 1st edge will be the outer edge.

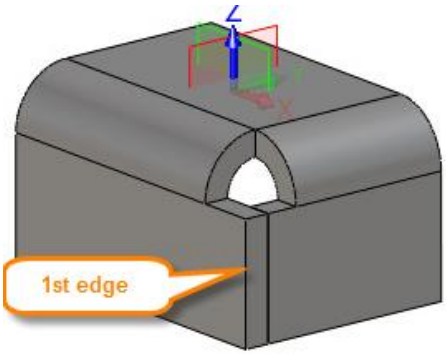


Figure 67 Overlap Flange Corner

Natural: Both edges will meet naturally.

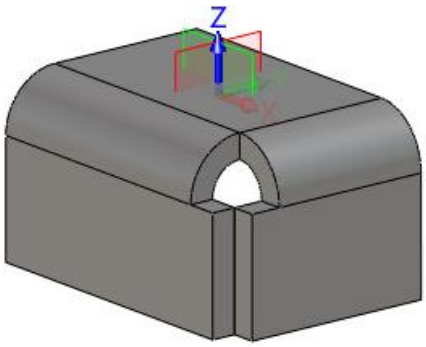


Figure 68 Natural Closed Flange Corner

STEP 03 Try the send type—Bend. Select two bend faces to close corner as follows.

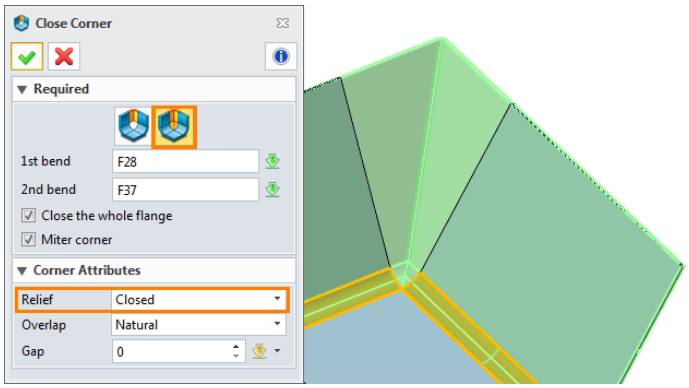


Figure 69 Close the Corner by Bend Face

The 1st and 2nd bend doesn't require the sequence, and they are just affected by the Overlap type. The overlap type is the same meaning as the overlap type in Edge option.

Options:

1) **Close the whole Flange: it is used to close the whole multiple levels flange as follows.**

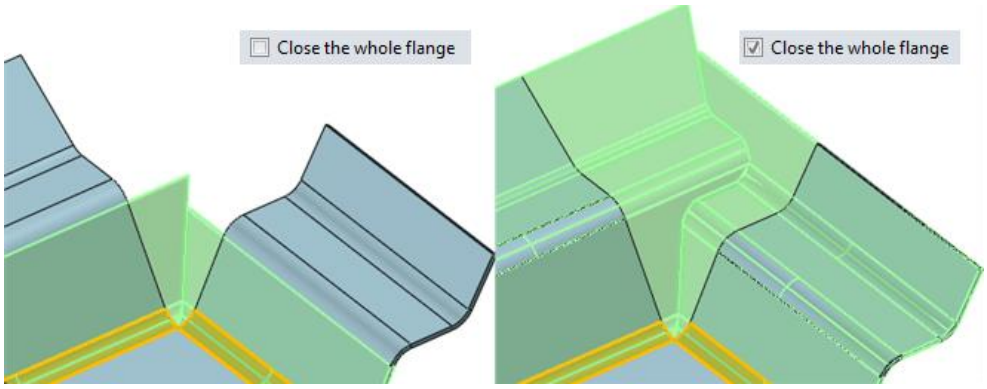


Figure 70 Close the Whole Flange

2) **Milter corner:**

Uncheck this option , the corner's gap shape will be linear. Unfold shape is linear and simple so it is easy to manufacture,as shown in Figure 71.

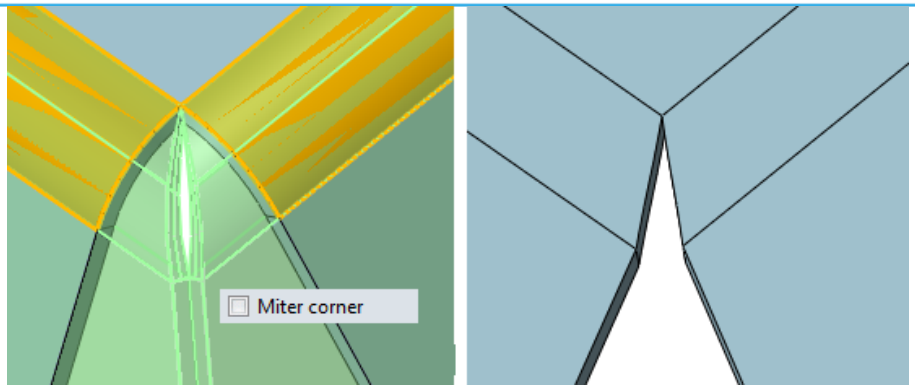


Figure 71 Unfolded Un-mitered Corner

Check this option, the corner will be close to meet the gap shape. Then the unfold shape is not linear as shown in Figure 72.

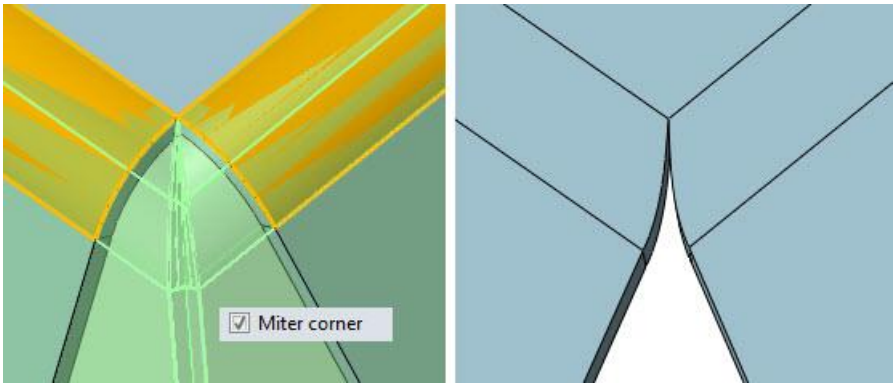


Figure 72 Unfolded Mitered Corner

1.6 Forming Tools

1.6.1 Dimple

Sheet Metal ribbon tab->Form->Dimple

It creates both plain dimple and flare dimple feature by selecting closed tangent connected sketch.

STEP 01 Open the file named ***Sheetmetal Functions.Z3->16_Forming Tool***.

STEP 02 Choose Dimple command, and then go to choose the circle inputted as follows.

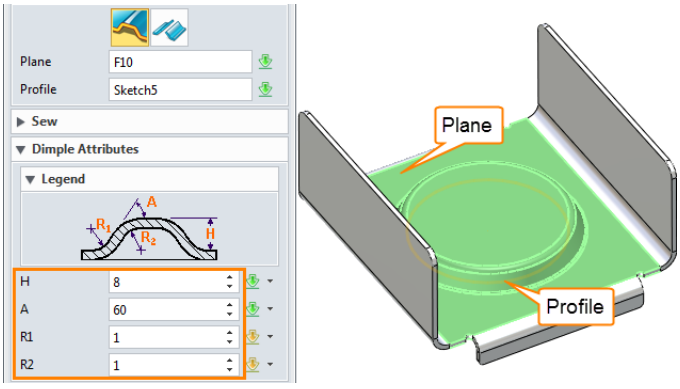


Figure 73 Create Plain Dimple

Notes: *The sketch must be connected tangently and closed.*

Here we use the "Plain Dimple" mode and switch to flare dimple as follows.

STEP 03 Create a sketch in the dimple top face as shown in Figure 74.

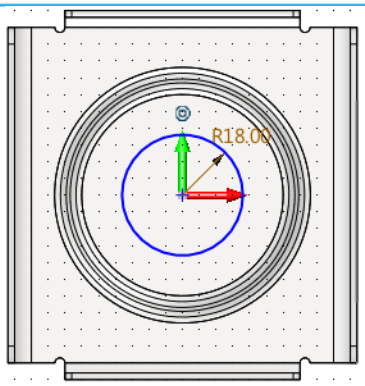


Figure 74 Sketch on Top Face of Plain Dimple

STEP 04 Create dimple again and choose Flare dimple option as follows.

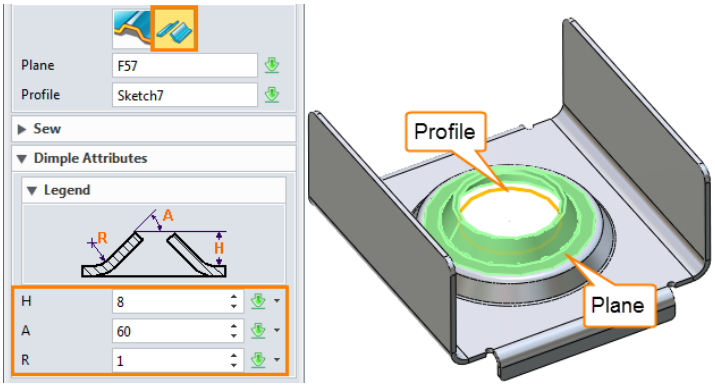


Figure 75 Flare Dimple

1.6.2 Louver

Sheet Metal ribbon tab->Forming-> Louver

It creates louver feature by only selecting sketch line.

Based on the Dimple shape finished in last step to create louver.

STEP 01 Select the plane and a sketch as the louver profile. Set the suitable louver parameters, as shown in Figure 76.

STEP 02 Check the option***Flip louver 180 degrees***, the louver direction will be changed.

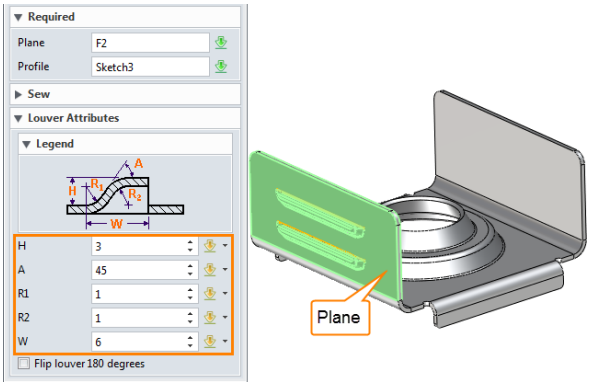


Figure 76 Create Louver

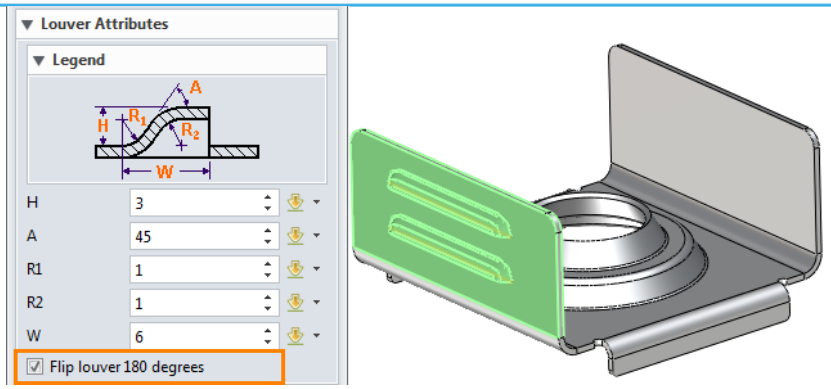


Figure 77 Flip Louver 180 Degrees

STEP 03 Finish the louver in another side as follows.

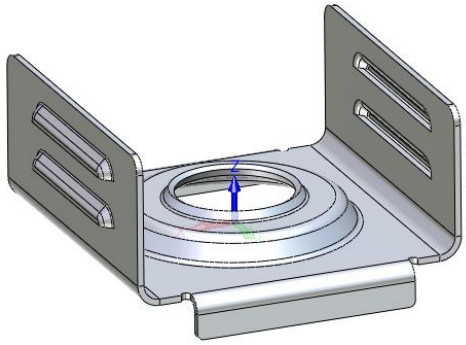


Figure 78 Louver in two Sides

1.6.3 Punch

Sheet Metal ribbon tab->Forming->Punch

It performs a punch operation between two shapes (solids or open shapes) to create a shelled feature.

STEP 01 Open file named ***Sheetmetal Functions.Z3->17_Punch*** as follows.

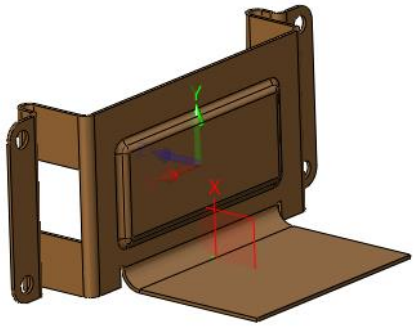


Figure 79 Punch Part

STEP 02 Create a sketch which will be used to create a puncher to punch on the datum plane.

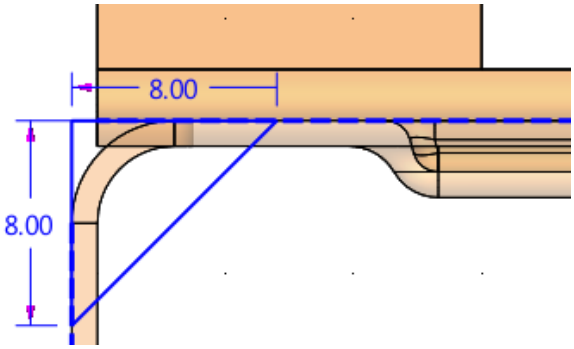


Figure 80 Sketch for Puncher

STEP 03 Extrude the sketch as the puncher.

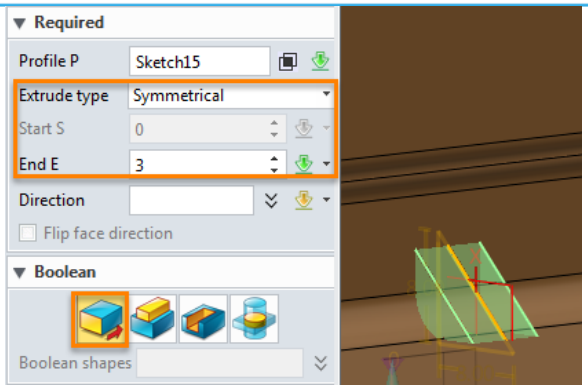


Figure 81 Extrude the Sketch as Puncher

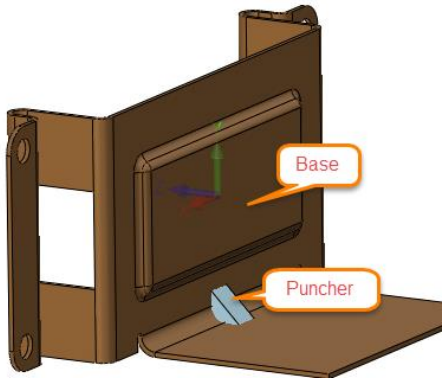


Figure 82 Puncher and Base

STEP 04 Punch a stiffener as shown in Figure 83.

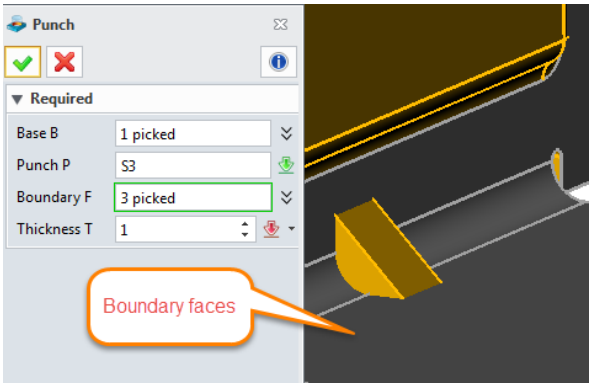


Figure 83 Punch a Stiffener

STEP 05 Confirm it to get the following result.

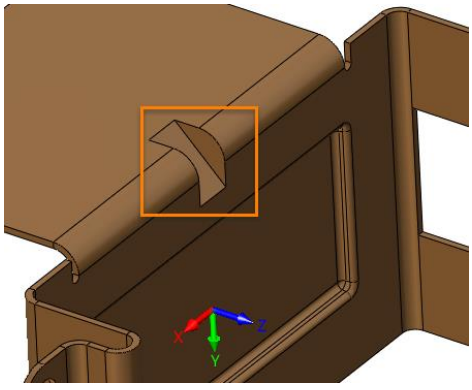


Figure 84 Punch Result

STEP 06 Add fillet on the boundary of the stiffener as follows. Then save the file.

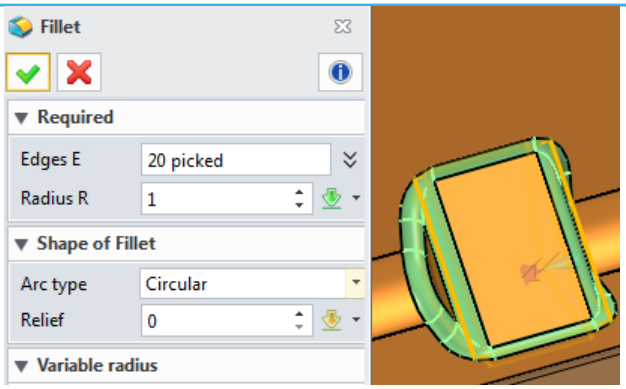


Figure 85 Fillet for the Stiffener

1.7 Bend Function

1.7.1 Fold and Unfold

Sheet Metal ribbon tab->Bend-> Fold/Unfold

It is used to fold the unfolded flange and unfold the folded flange. These 2 commands are a pair of commands to create the folded status of sheet metal part and recover the unfolded status. Taking unfold as example to explain:

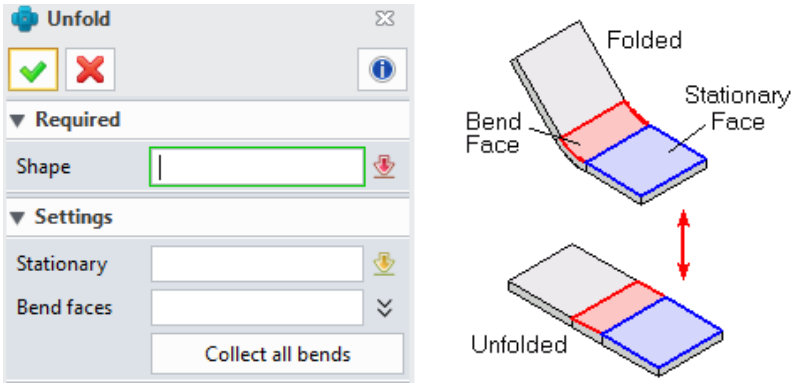


Figure 86 Unfold

Shape: Choose sheet metal shape to unfold or fold.

Stationary face: choose a face as the stationary face, which will determine the unfolded shape's location plane. This option is not mandatory, if this option is blank, it will use the default base flange's top face as the stationary face.

Bend faces: choose the bend face to determine whether bend all or not; if this option is blank then it will go to unfold all bend face.

Collect all bends: It can collect all bend feature created by ZW3D sheet metal or the bend marked by Mark Bend command automatically

Because the fold command is the same usage as the unfold command.

1.7.2 Linear Unfold

Sheet Metal ribbon tab->Bend->Linear Unfold

It is used to unfold a bend flange partially to the specified unfolded angle. It can show the process of forming the bend, especially for the sheet metal part with big thickness which couldn't be formed with one bend.

STEP 01 Open the file named ***Sheetmetal Functions.Z3->18_Linear Unfold.***

STEP 02 Choose Linear Unfold command to unfold it by 45 degrees as follows.

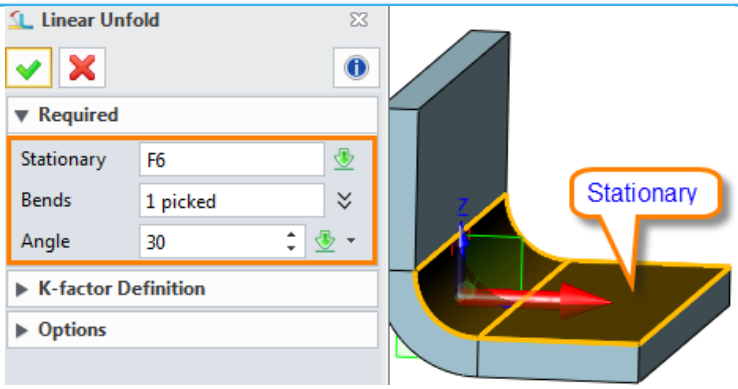


Figure 87 Linear unfold

Then confirm it and we will get the result as follows:

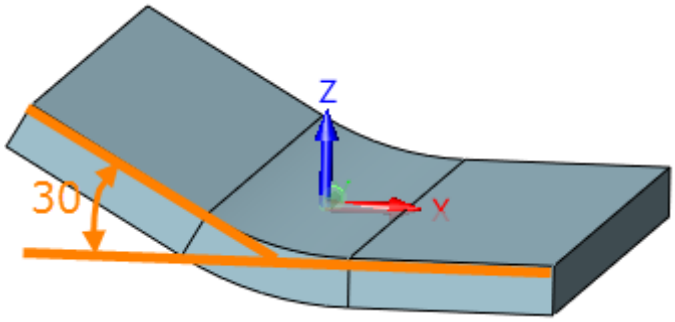


Figure 88 Linear unfolded shape

STEP 03 Next, let's go to redefine this command again and then pull down the ***Options***:

Check the box of ***Add a new forming status*** and set the parameter for new forming status as follows:

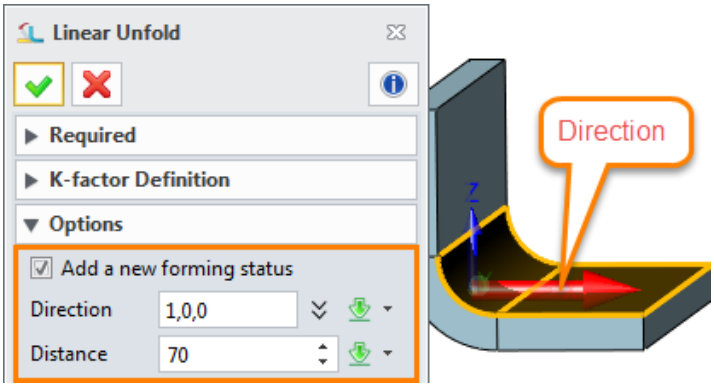


Figure 89 Add a new forming status

After confirming it we can get the following result:

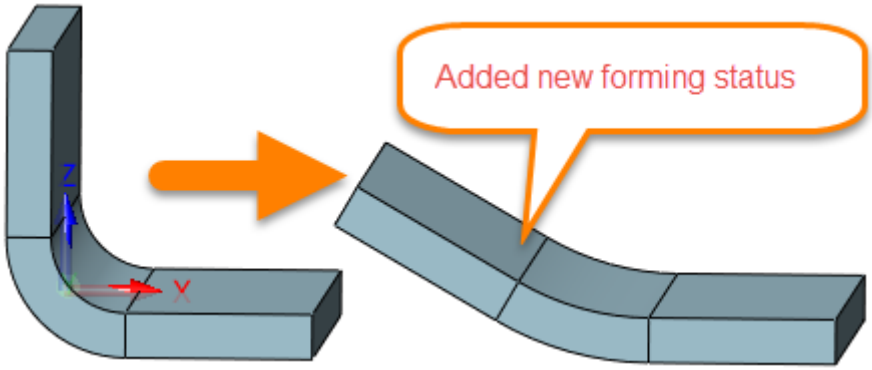


Figure 90 Added new forming status

Notes: *The added new fomring status is associated with the original part.*

1.7.3 Change Bend

Sheet Metal ribbon tab->Bend->Change Bend

It is used to change the finished bend's information such as bend radius, bend angle, K-factor, etc.

STEP 01 Open the file named ***Sheetmetal Functions.Z3->19_Change Bend***, in this file we have already created 2 configures for Fold and Unfold status. Here let's go to activate the Fold status to check as follows.

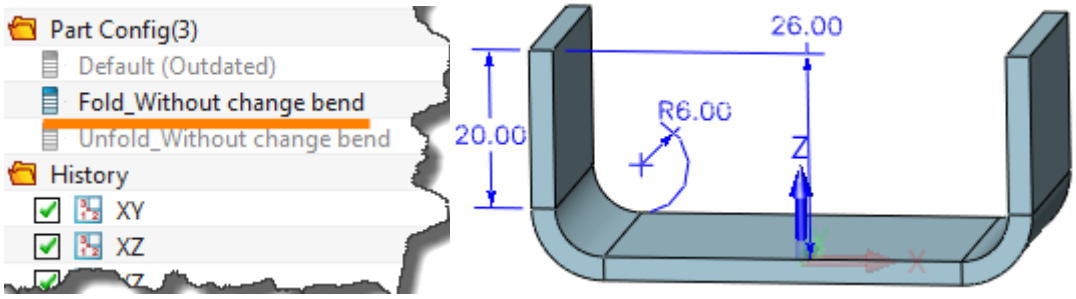


Figure 91 Fold status without change bend

Notes: *These 3D dimensions are created by PMI function.*

And then we can go to check the unfold status without changing bend as follows.

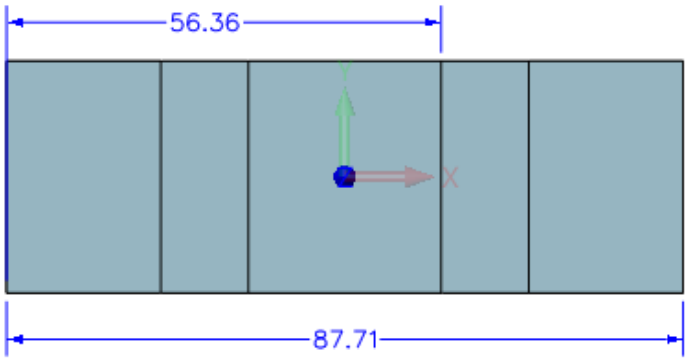


Figure 92 Unfold status without change bend

STEP 02 After recovering the fold status , we can change its left bend as follows.

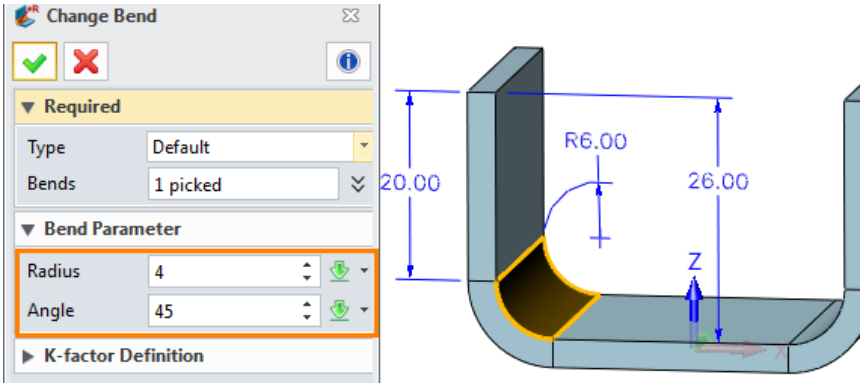


Figure 93 Default way to change bend

Confirm it and the result will be as follows.

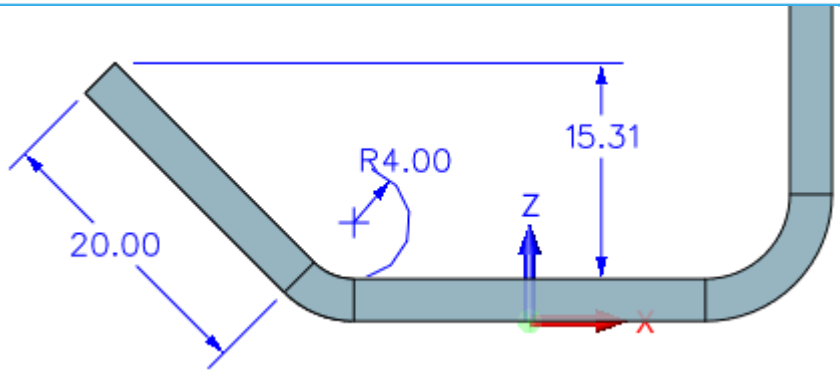


Figure 94 Default way changed bend result

STEP 03 Unfold it to get the following result.

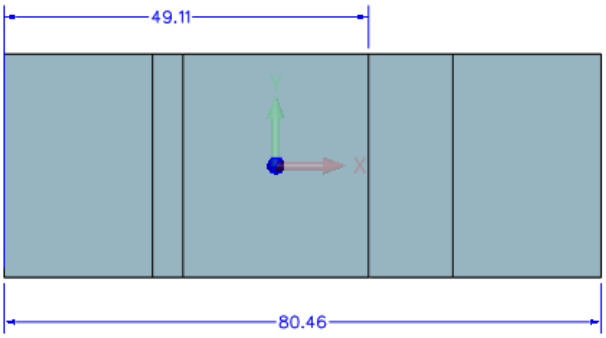


Figure 95 Unfold status after bend changed

If we go to compare the unfold result of unchanged bend and changed bend (Fig97and Fig100), we will find the unfold length has been changed. So we can choose another way to change the bend.

STEP 04 Go to redefine the change bend and change the Type into ***Fixed unfold length***.

Before you go to redefine the feature, turn off all PMI information as follows.

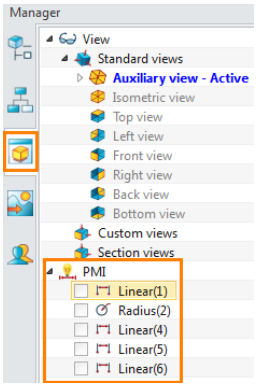


Figure 96 Turn off the PMI information

And then go to redefine the ***Bend Modified*** feature in the history as follows.

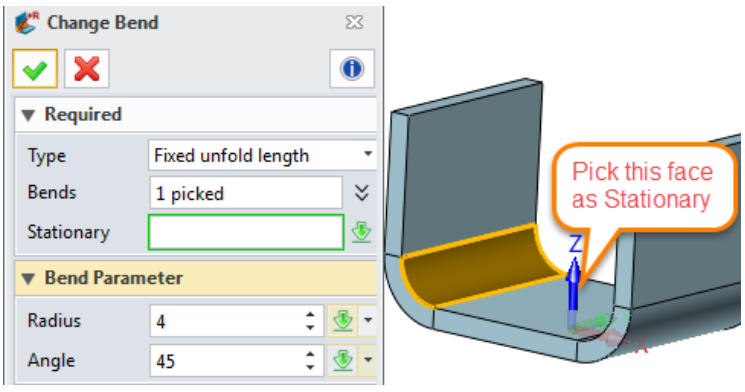


Figure 97 Fixed unfold Length

And then confirm it and unfold it again, at the same time turn on the PMI as follows.

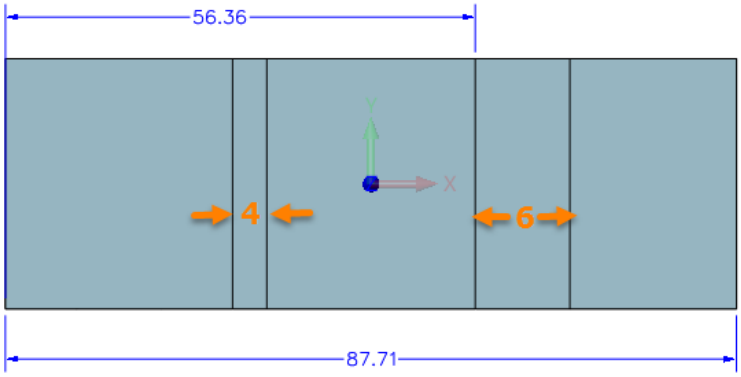


Figure 98 Unfold the changed bend again by fixed unfold length

1.7.4 Show Bend Information

Sheet Metal ribbon tab->Bend->Show Bend Info

It is used to show the bend information such as Bend radius, k-factor etc. and so on.

STEP 01 Choose the command and then pick the desired face as follows.

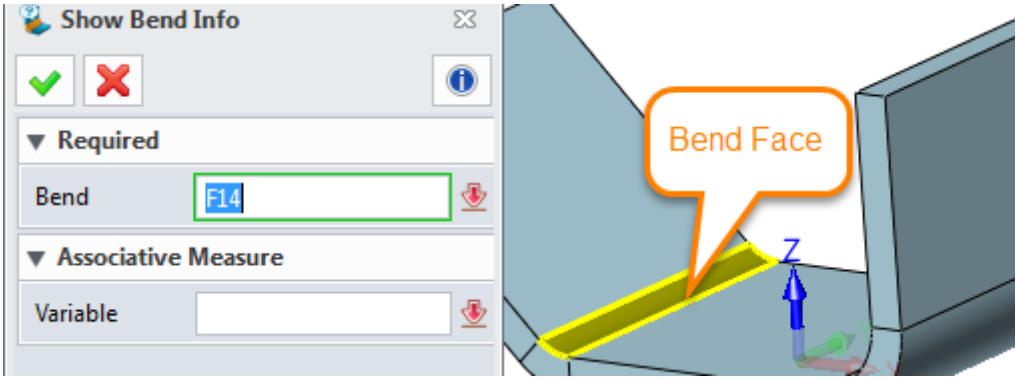


Figure 99 Show Bend Information

STEP 02 After choosing the bend face it will pop up the information form as follows.

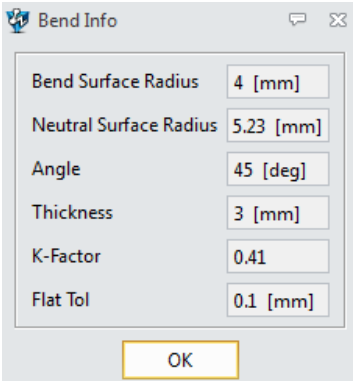


Figure 100 Bend Information

1.7.5 Set Stationary Face

Sheet Metal ribbon tab->Bend->Set Stationary Face

It is used to specify a stationary face which will be used as the default stationary face when unfold a sheet metal part.

Select face as input.

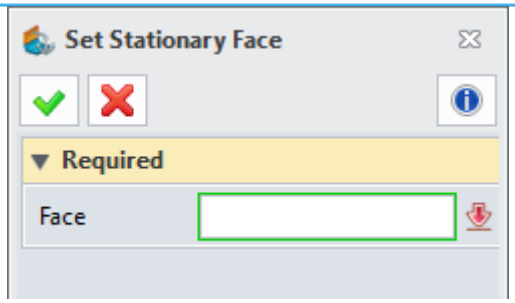


Figure 101 Set Stationary Face

1.8 Convert

1.8.1 Rip

Sheet Metal ribbon tab->Convert->Rip

It is used to open a gap for non-sheet metal part by a line, which can be edge, sketch or wireframe

STEP 01 Open the file named ***Sheetmetal Functions.Z3->20_Convert to sheet metal*** as follows.

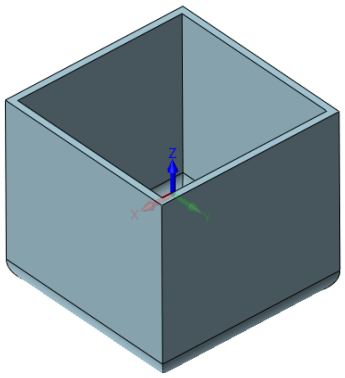


Figure 102 Convert to Sheetmetal part

STEP 02 Choose the ***Rip*** command and then go to choose edge to rip gaps as follows.

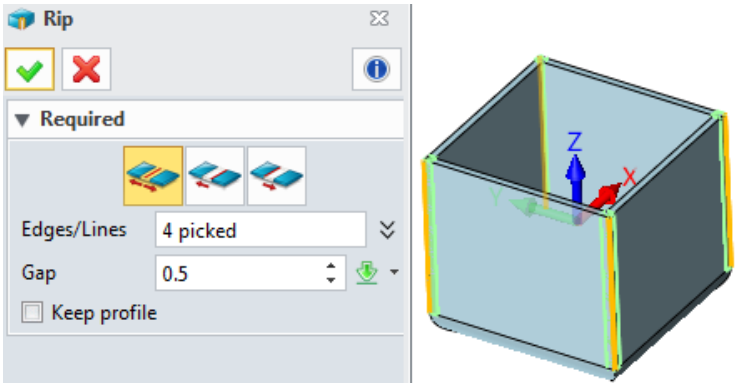


Figure 103 Rip gap

STEP 03 Confirm it to get the following result.

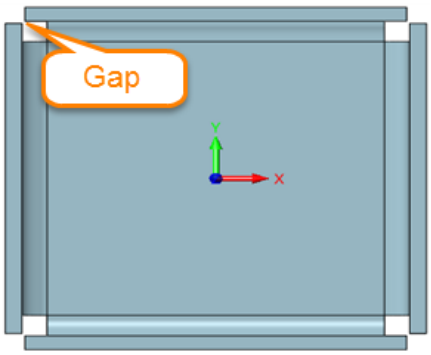


Figure 104 Rip result

1.8.2 Mark Bend

Sheet Metal ribbon tab->Convert->Mark Bend

It is used to mark cylinder face as bend face. Then it can be unfolded by sheet metal unfold command.

STEP 01 Keep on using the Convert to sheet metal part in last step and then mark the bend face.

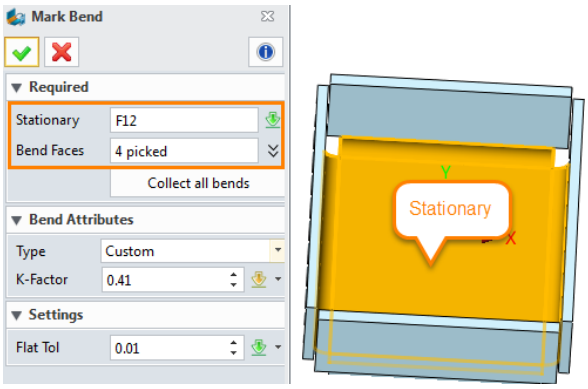


Figure 105 Mark Bend

Collect all bends: This can automatically collect the cylindrical face.

STEP 02 After confirming the command, the part will have bend information and we can unfold it as follows.

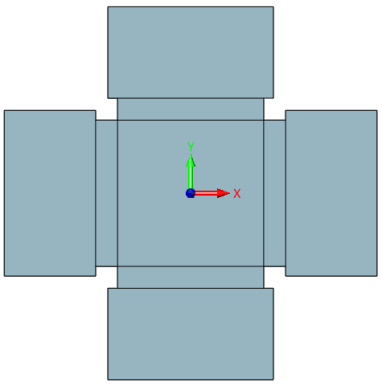


Figure 106 Unfold the Marked bend

1.8.3 Convert to Sheet metal

Sheet Metal ribbon tab->Convert->Convert to sheet metal

This command is a combination of Rip and Mark Bend, which means it includes both.

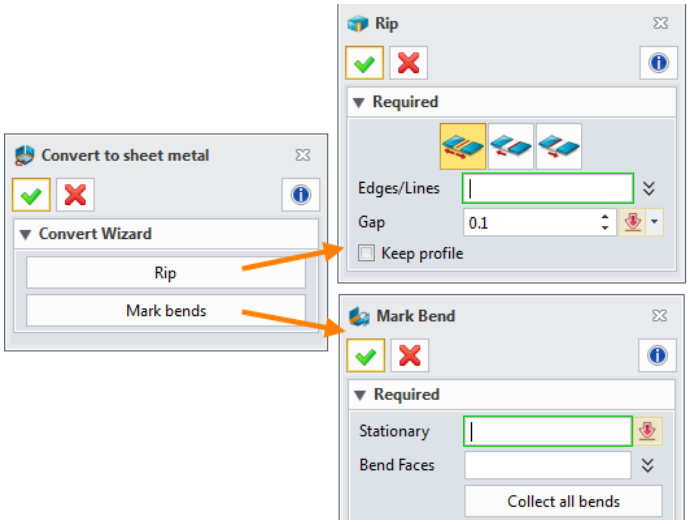


Figure 107 Convert to sheet metal

1.9 Exercises

1.9.1 Exercise1

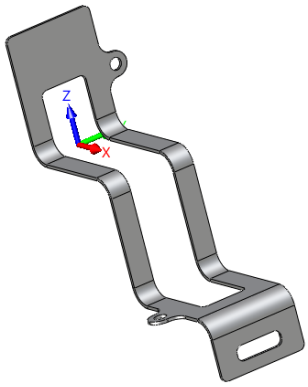


Figure 108 Exercise1

STEP 01 Create a sketch as follows.

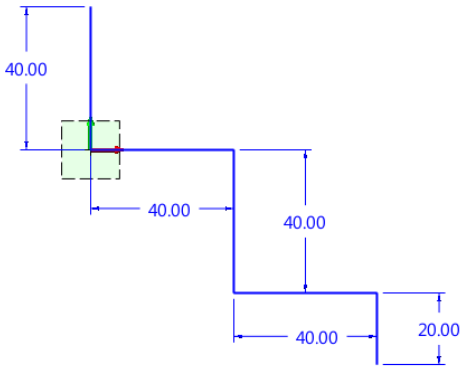


Figure 109 Sketch for exercise1

STEP 02 Exit the sketch and then extrude as follows.

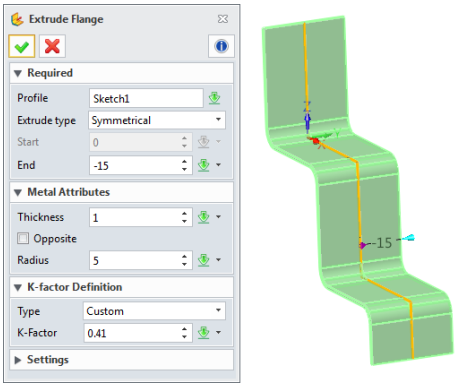


Figure 110 Extrude flange_excecise1

STEP 03 Add a sketch on the right side as follows.

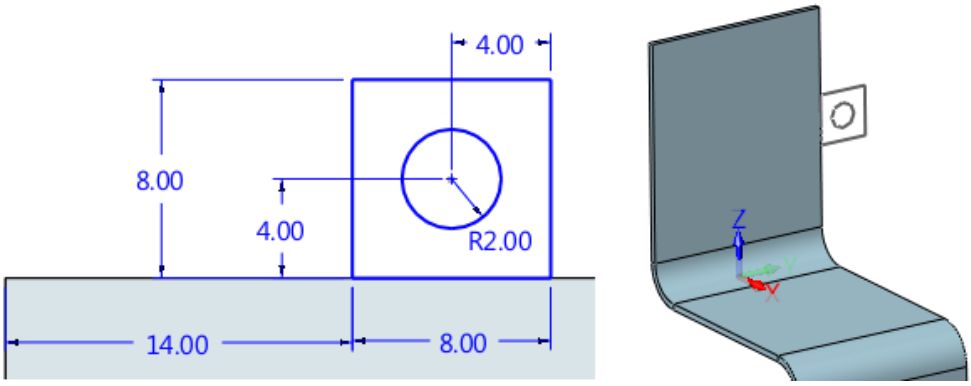


Figure 111 Add sketch_excecise1

STEP 04 Then go to merge a tab to flange as follows.

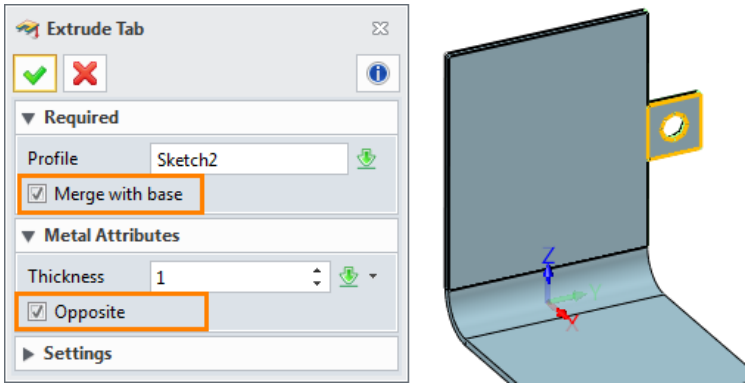


Figure 112 Merge a tab_excecise1

STEP 05 Add the same tab in another place as follows.

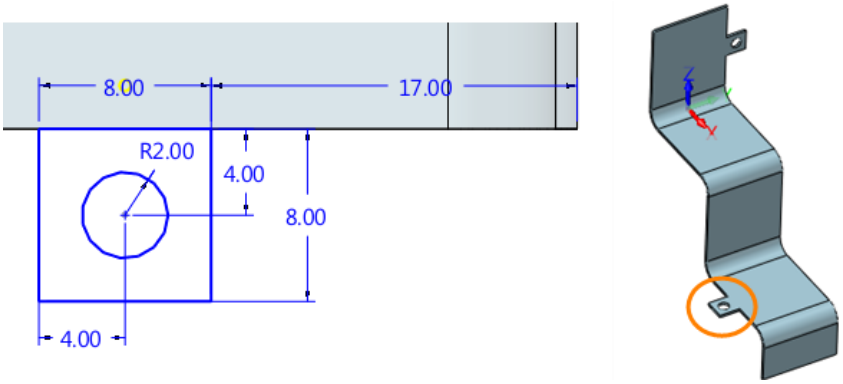


Figure 113 Add one mere tab_excesice1

STEP 06 Select the edges to add the fillet.

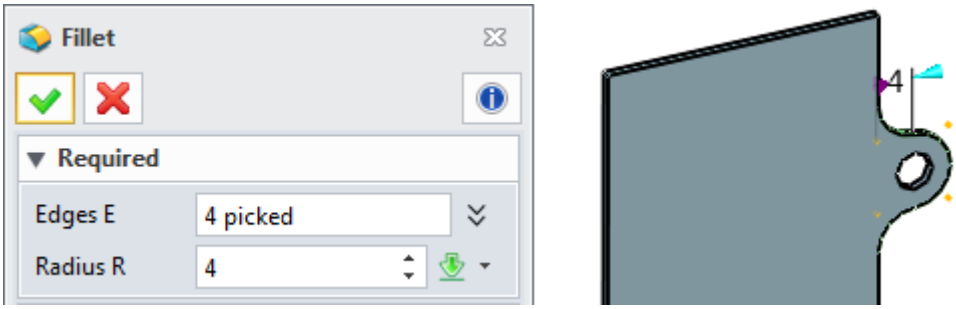


Figure 114 Add Fillet_excesice1

STEP 07 Create a sketch for middle slot as follows.

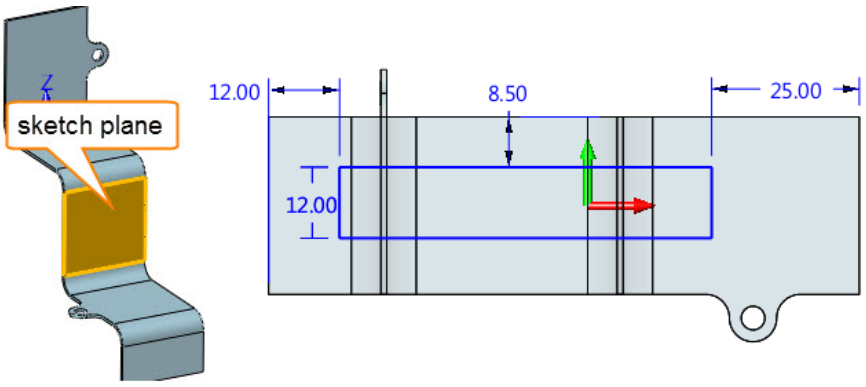


Figure 115 Create cut out sketch_excesice1

STEP 08 Extrude and cut the middle slot as follows.

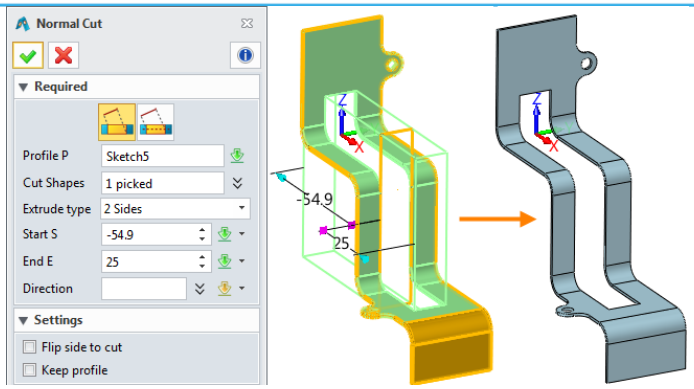


Figure 116 Extrude cut the middle slot_excesice1

STEP 09 keep on adding a sketch on the following position.

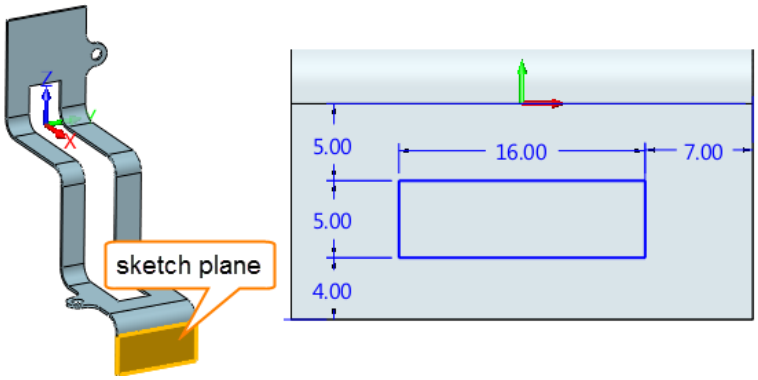


Figure 117 Sketch for another slot_excesice1

STEP 10 Extrude and cut the slot as follows.

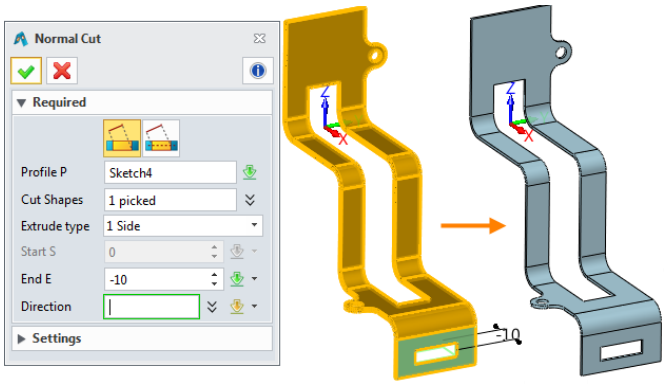


Figure 118 Extrude another slot

STEP 11 Fillet the corners as follows.

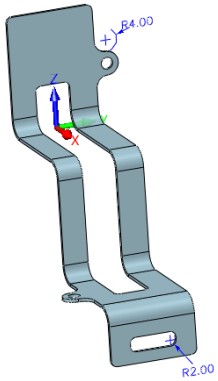


Figure 119 Add fillet at corners

STEP 12 Unfold it as follows:

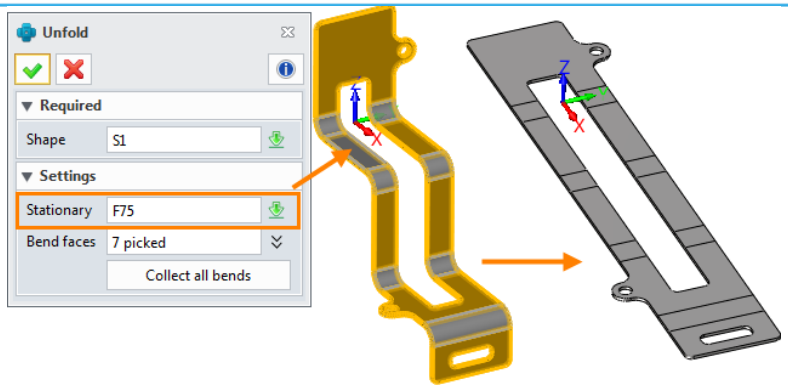


Figure 120 Unfold

STEP 13 Create 2D sheet for it and show the bend information as follows:

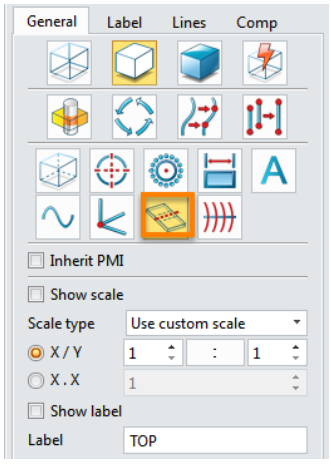


Figure 121 Show Bend information in 2D sheet

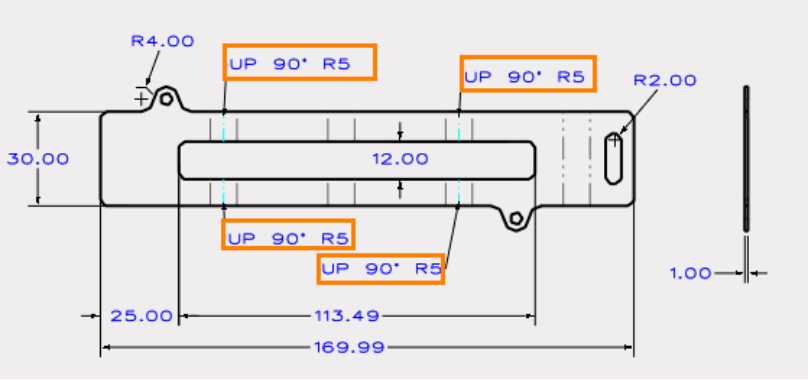


Figure 122 2D sheet with bend information

STEP 14 Save it.

1.9.2 Exercise2

STEP 01 Create a sketch as follows.

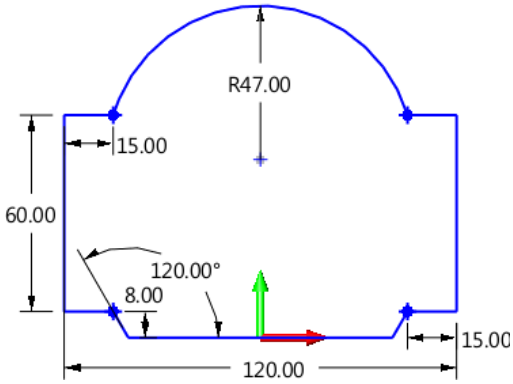


Figure 123 Sketch_ exercise2

STEP 02 Create a tab with thickness ***0.5*** and fillet the corners with ***R3*** as follows.

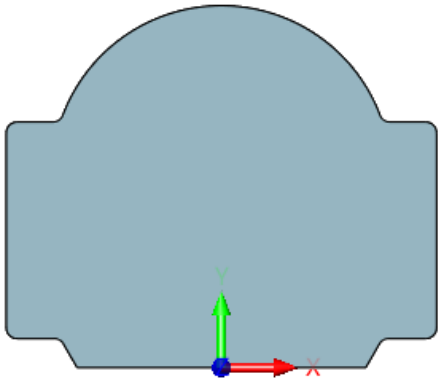


Figure 124 Tab_exercise2

STEP 03 Create sketch for dimple as follows.

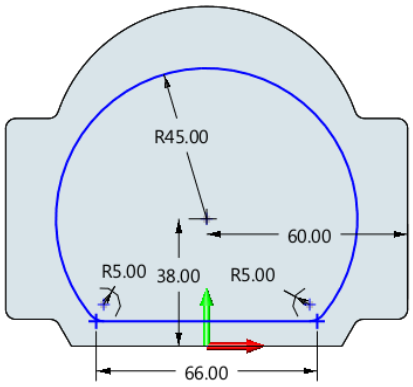


Figure 125 Sketch for dimple_exercise2

STEP 04 Create dimple as follows.

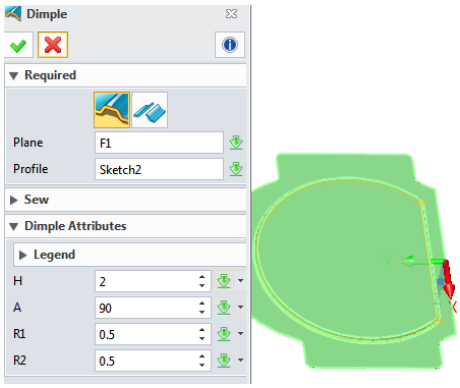


Figure 126 Dimple_exercise2

STEP 05 Create sketch for flare dimple as follows.

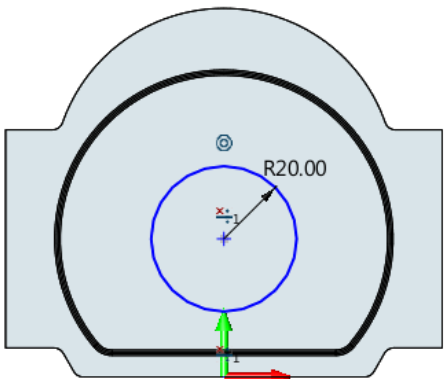


Figure 127 Sketch for flare dimple_exercise2

STEP 06 Create flare dimple as follows.

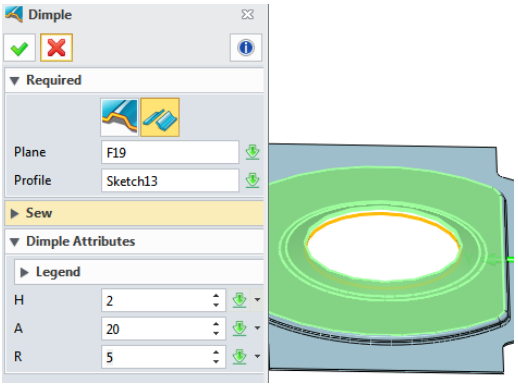


Figure 128 Flare Dimple_ exercise2

STEP 07 Create sketch on XY plane for cutting an open slot as follows.

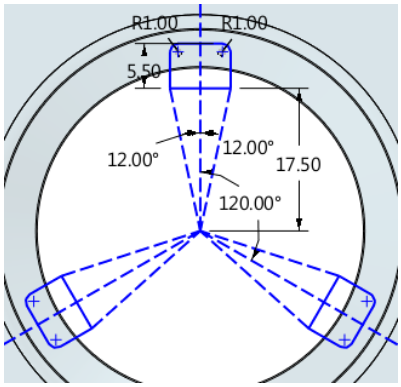


Figure 129 Sketch for flare dimpe_exercise2

STEP 08 Normal cut the flare dimple as follows.

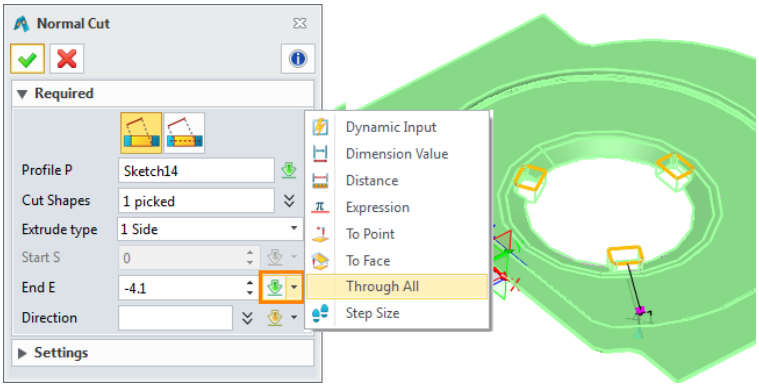


Figure 130 Cut the open slots on flare dimpe_exercise2

Tips: *choose* ***1 side*** *and* ***Through All*** *option from the contextual menu to quickly finish it.*

STEP 08 Create a sketch on the following plane for creating Louver sketch as follows.

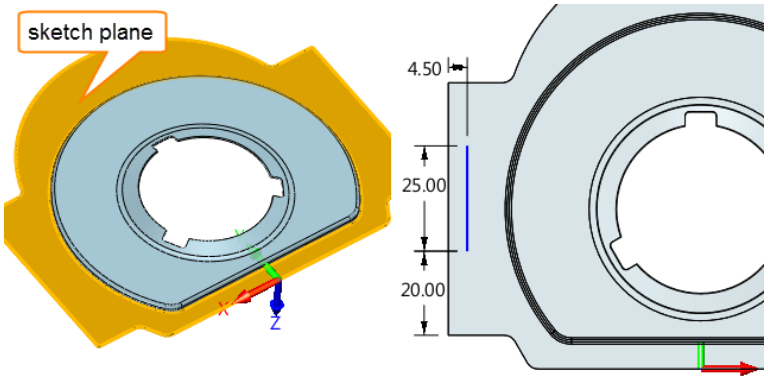


Figure 131 Plane for sketch of Louver_exercesie2

STEP 09 Create the louver as follows.

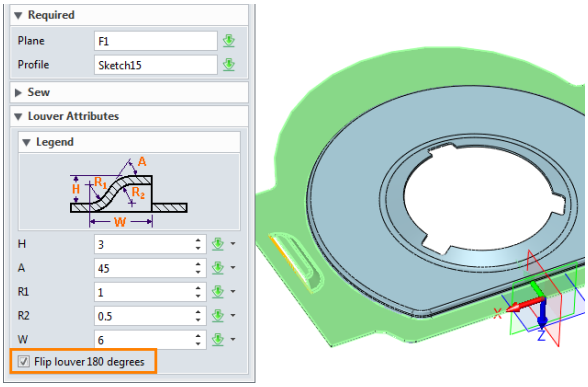


Figure 132 Louver _exercesie2

STEP 10 With ***Mirror Feature*** command to mirror louver feature as follows.

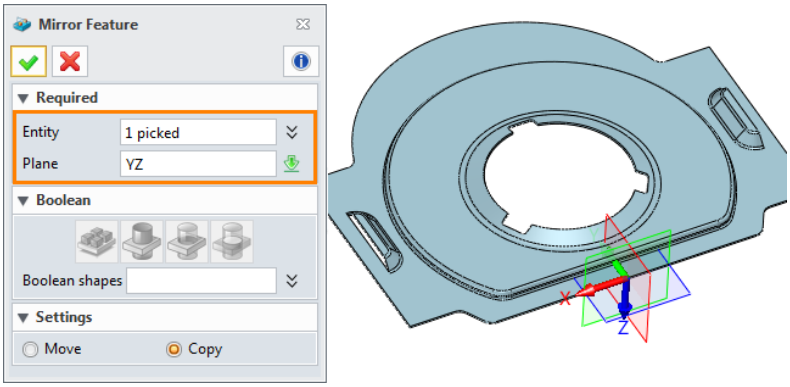


Figure 133 Mirror louver feature _exercesie2

STEP 11 Create a new plane for swept flange's sketch, and draw the sketch.

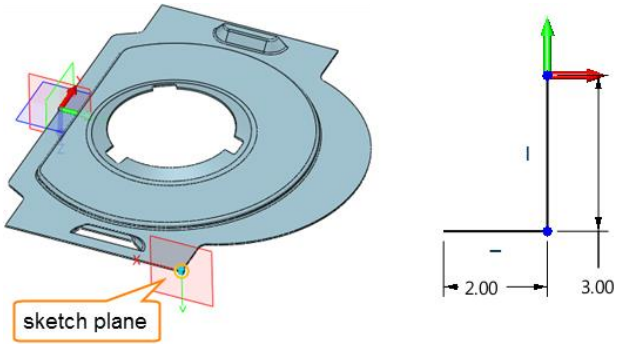


Figure 134 First sketch for swept flange_exercise2

STEP 12 Create Swept Flange, details are as follows.

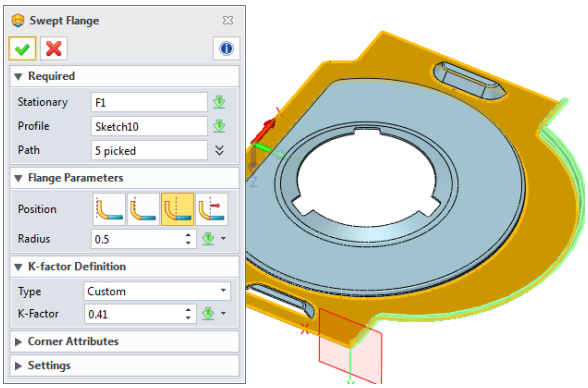


Figure 135 Swept flange_exercise2

STEP 13 Create another plane for a second Swept flange's sketch and draw the sketch as follows.

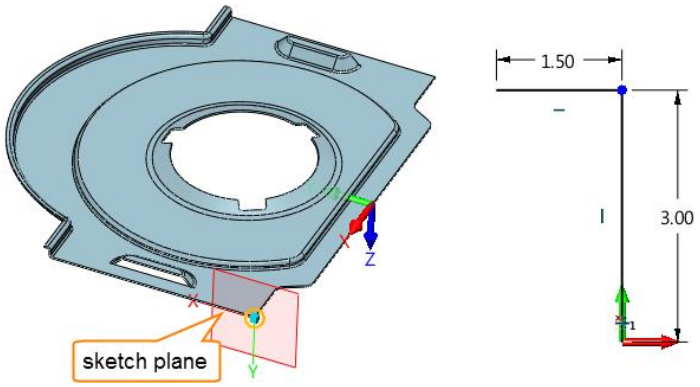


Figure 136 Second new plane for swept flange profile_exercesie2

STEP 14 Create the second Swept flange as follows.

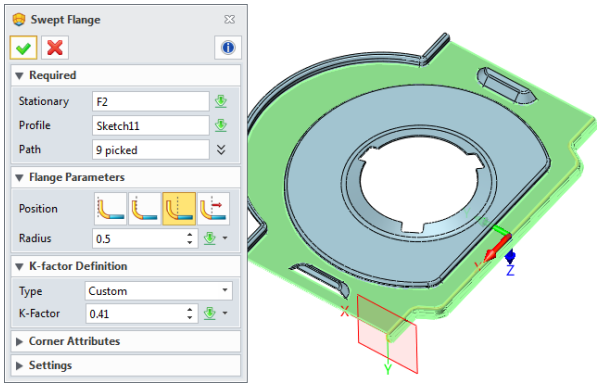


Figure 137 Second swept flange_exercesie2

After that we can get the following final result. Then save the file.

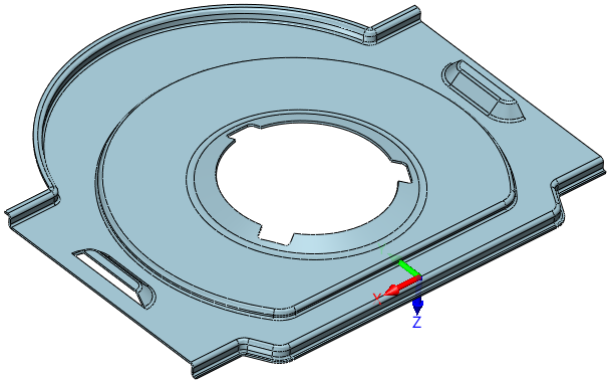


Figure 138 Result for exercise2