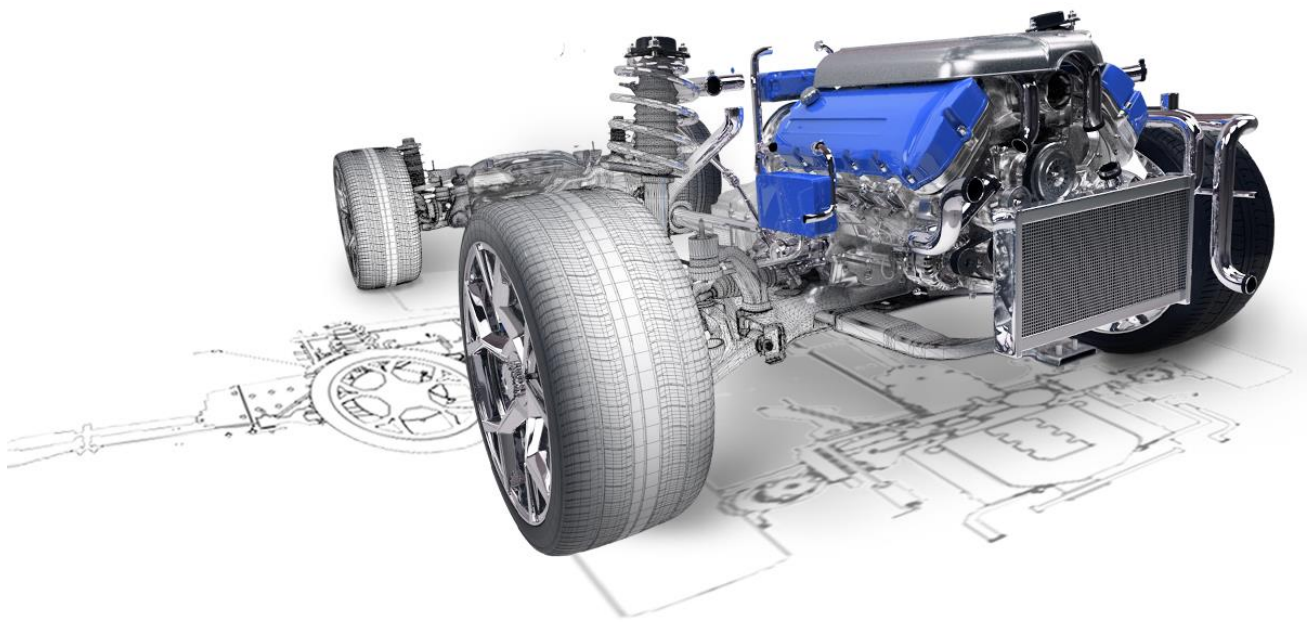


ZW3D



ZW3D From Entry to Master

Sheet Metal

Contents

ZW3D Sheet Metal.....	1
ZW3D Sheet Metal General Introduction.....	1
1. Set up Sheet Metal Attribute.....	2
2. Create Sheet Metal Base.....	3
2.1 Extrude Tab.....	3
2.2 Extrude Flange.....	5
3. Flange Creation.....	8
3.1 Full Flange.....	8
3.2 Partial Flange.....	11
3.3 Hem Flange.....	12
3.4 Loft Flange.....	14
3.5 Swept Flange.....	16
3.6 Fold by Line.....	21
3.7 Jog.....	22
4. Editing Tools.....	24
4.1 Extend Flange.....	24
4.2 Bend Taper.....	27
4.3 Normal Cut.....	29
5. Corner.....	32
5.1 Close Corner.....	32
6. Forming Tools.....	35
6.1 Dimple.....	35
6.2 Louver.....	37
6.3 Punch.....	39
7. Bend Function.....	42
7.1 Fold and Unfold.....	42
7.2 Linear Unfold.....	43
7.3 Change Bend.....	45
7.4 Show Bend Information.....	48
7.5 Set Stationary Face.....	49
8. Convert.....	50
8.1 Rip.....	50
8.2 Mark Bend.....	51
8.3 Convert to Sheet metal.....	52
9. Exercises.....	53
9.1 Exercise1.....	53
9.2 Exercise2.....	60

ZW3D Sheet Metal

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 General Introduction

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:

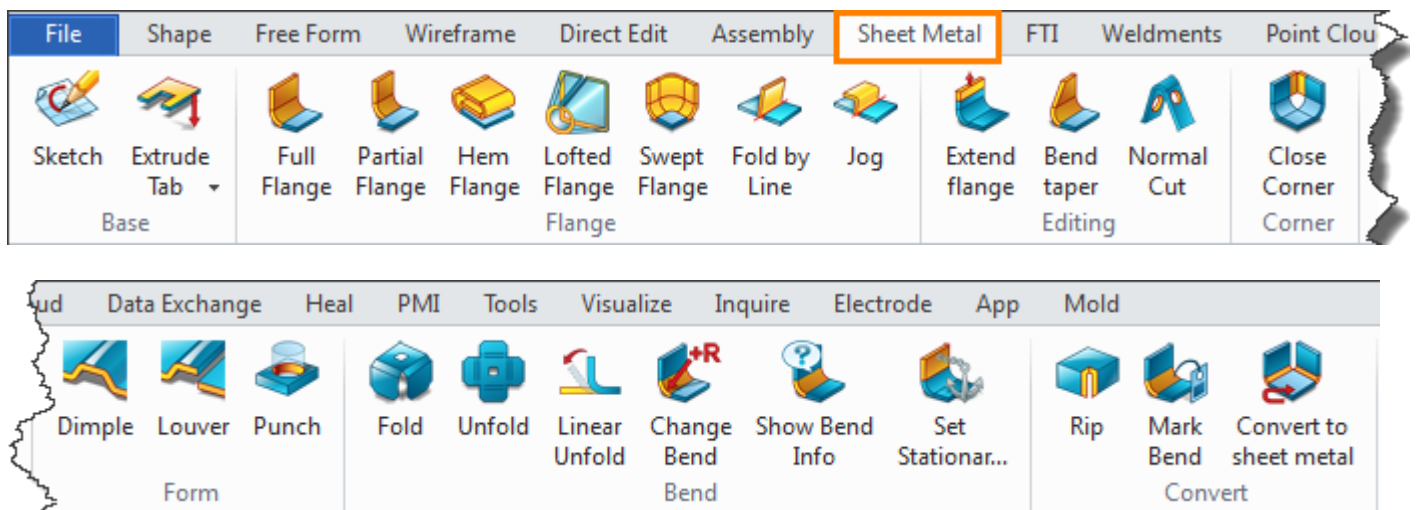
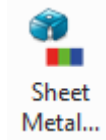


Figure1 ZW3D Sheet Metal Module

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

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 toolbar->Attributes->

STEP 01 Choose the Sheet metal attribute button

STEP 02 Set up the parameters as follows

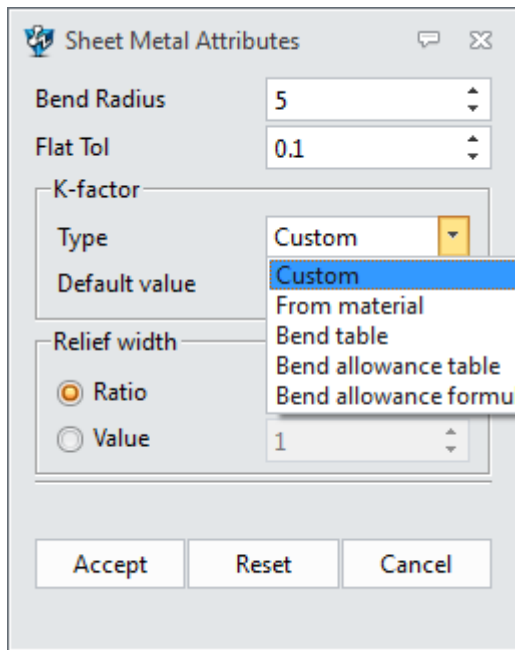


Figure2 Sheet Metal Attribute

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.

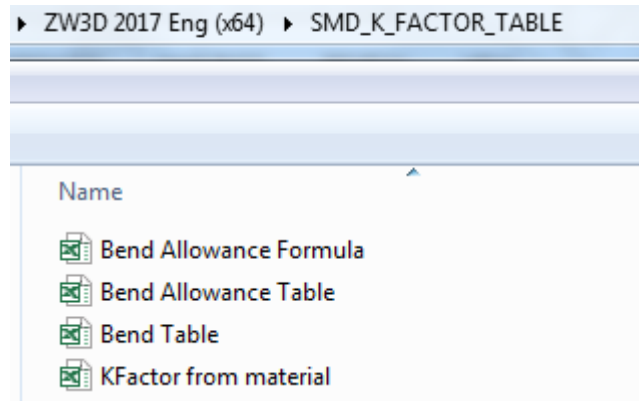


Figure3 K-Factor Table

2. Create Sheet Metal Base

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

2.1 Extrude Tab



Sheet Metal ribbon toolbar->Base->

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

2.1.1 Extrude Base

STEP 01 Create a sketch as follows:

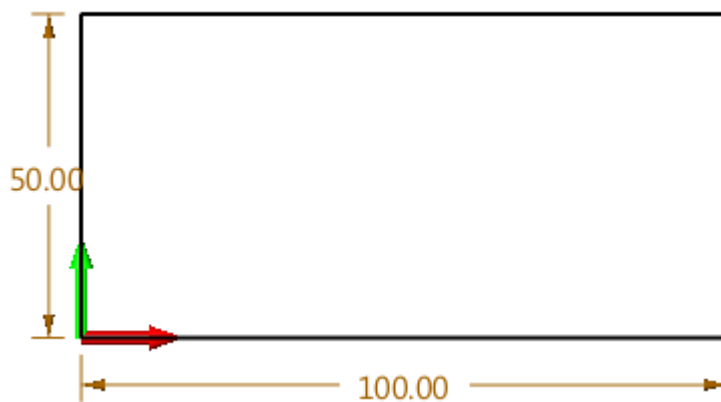


Figure4 Sketch for extrude base

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

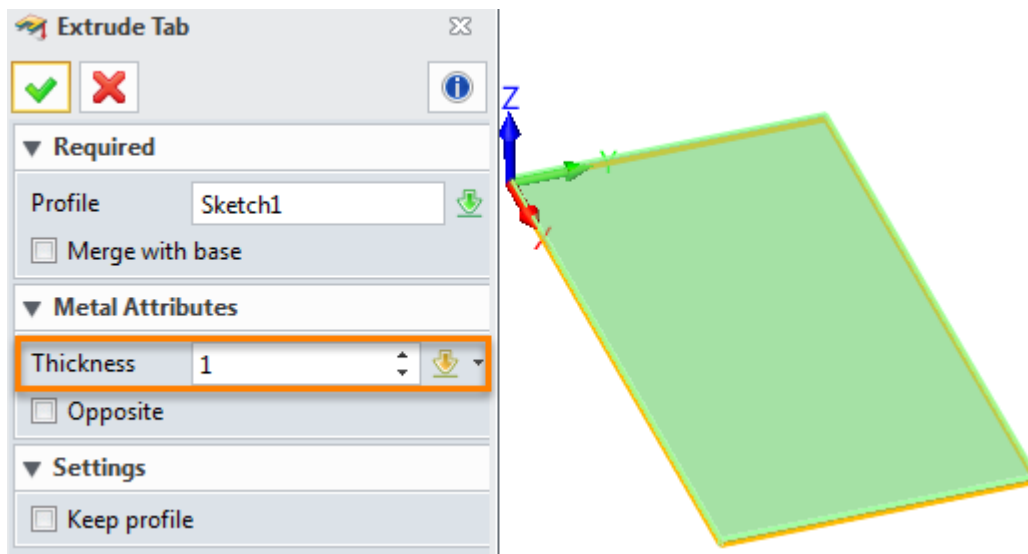


Figure5 Extrude base

Then we can add one more tab based on it.

2.1.2 Add a tab

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



Figure6 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”.

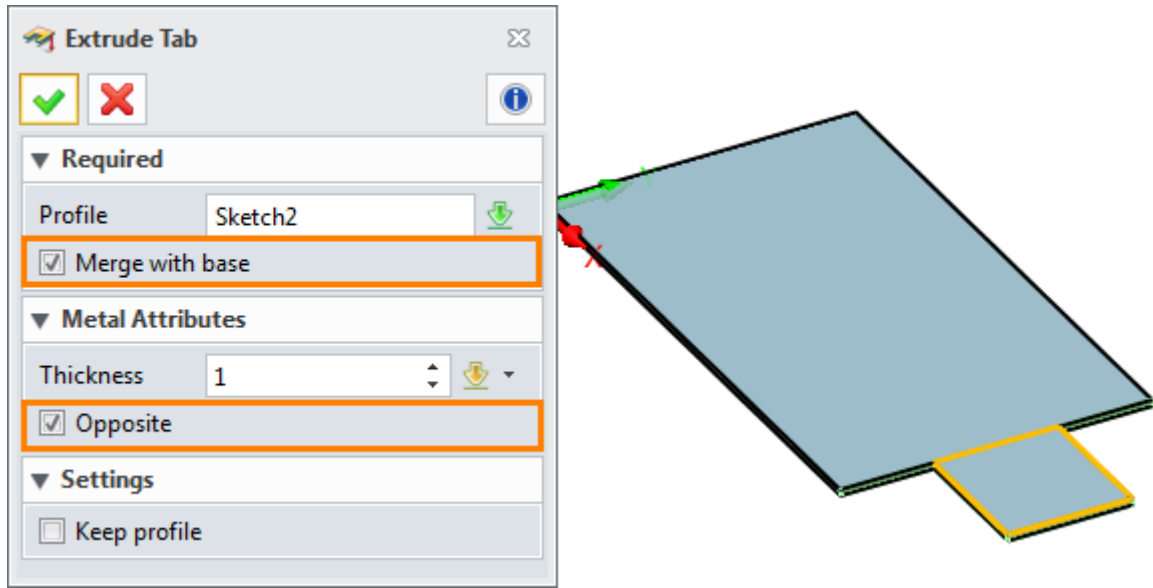


Figure7 Merge Extruded Base

2.2 Extrude Flange



Sheet Metal ribbon toolbar->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.

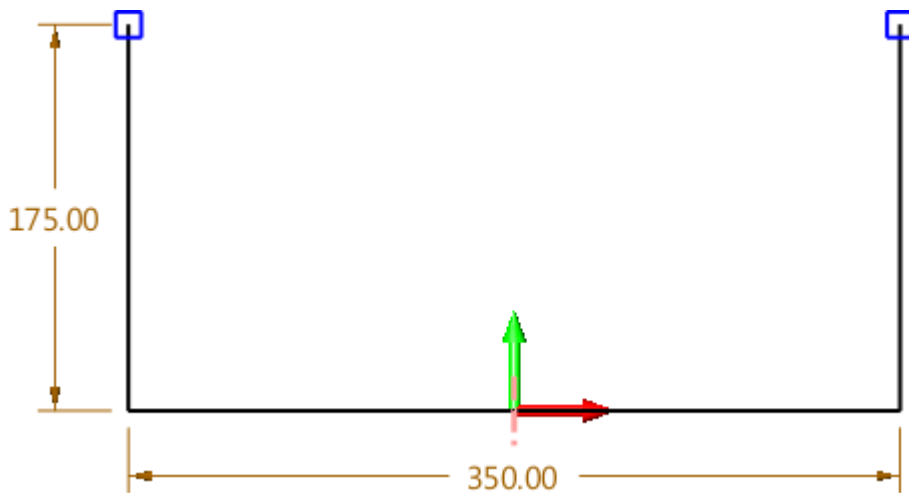


Figure8 Sketch with Open Profile

STEP 02 Extrude flange as shown in Figure 9.

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

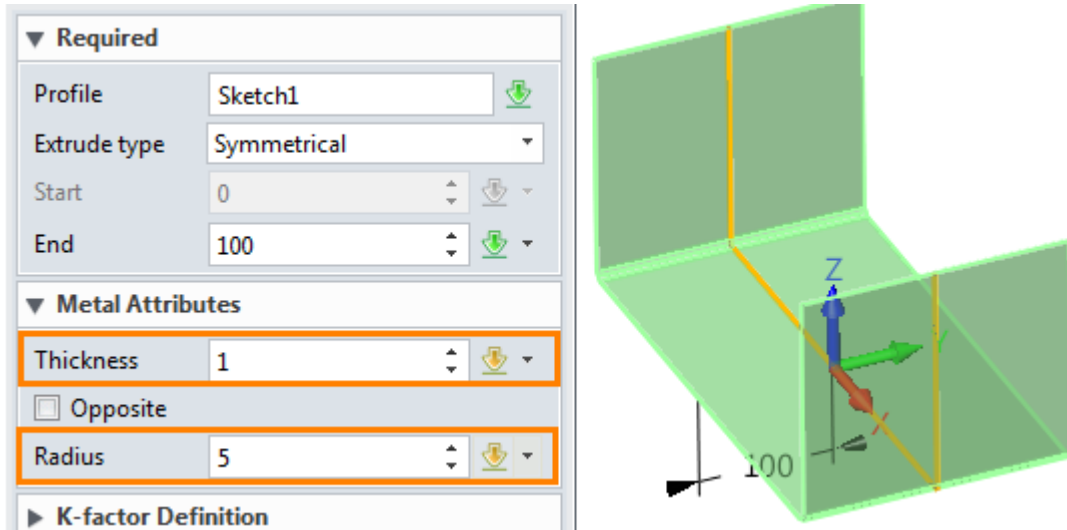


Figure9 Extrude Flange

Note: 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:

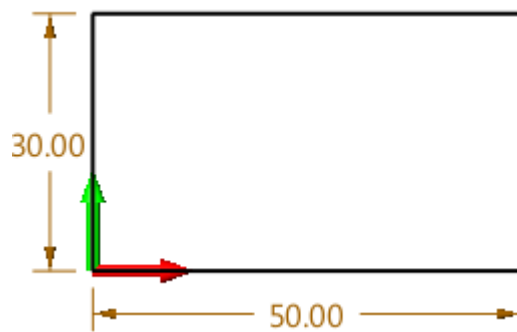


Figure10 Sketch with Closed profile

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

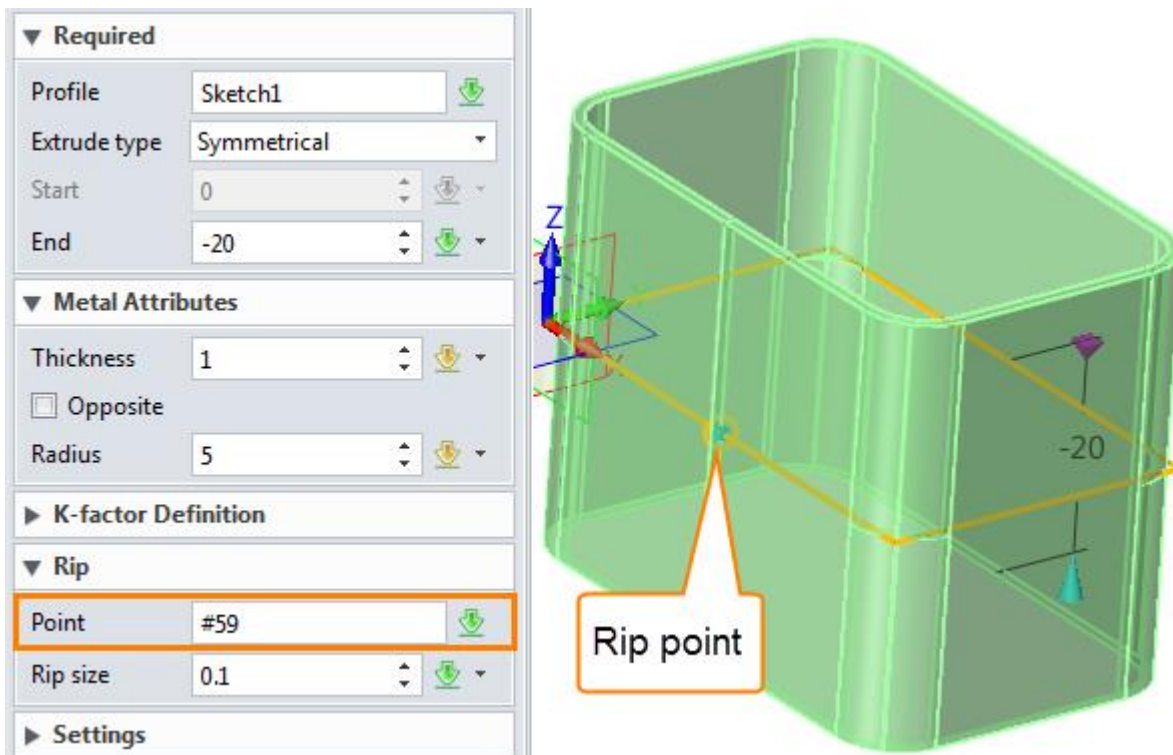


Figure11 Extrude flange by closed sketch

Note: If the Rip option is blank, then it will create a closed shape that can't be unfolded.

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



Sheet Metal ribbon toolbar->Bend->

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".

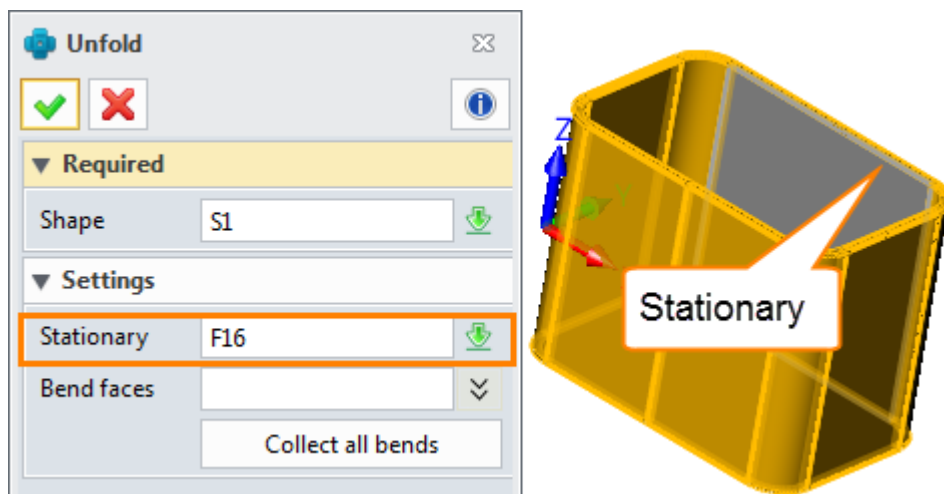


Figure12 Unfold the Part

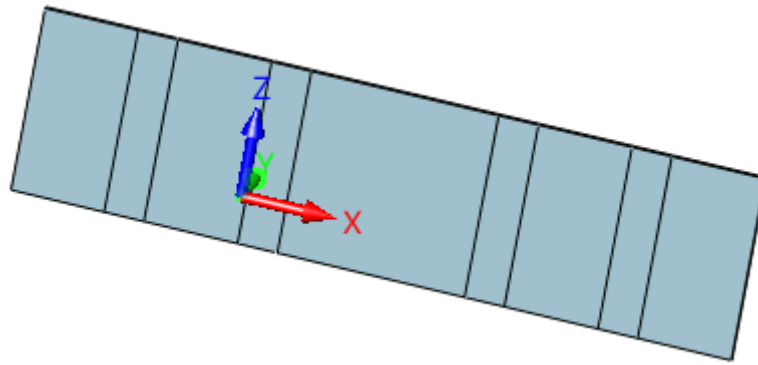


Figure13 Unfold Result

3. Flange Creation

3.1 Full Flange



Sheet Metal ribbon toolbar->Flange->

This command can create full flange and S type flange.

STEP 01 Create a base flange as follows.

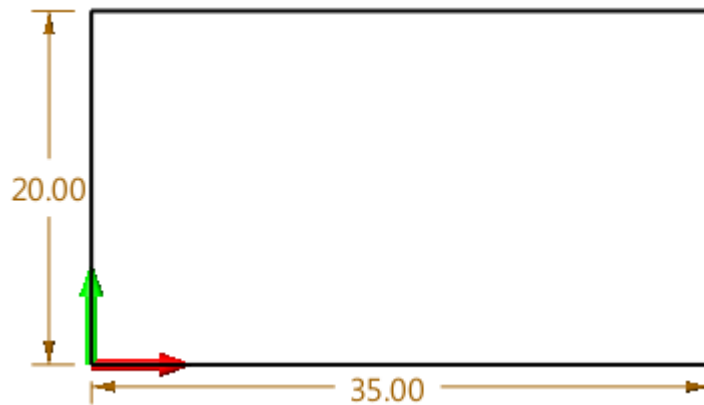


Figure14 Create sketch for base

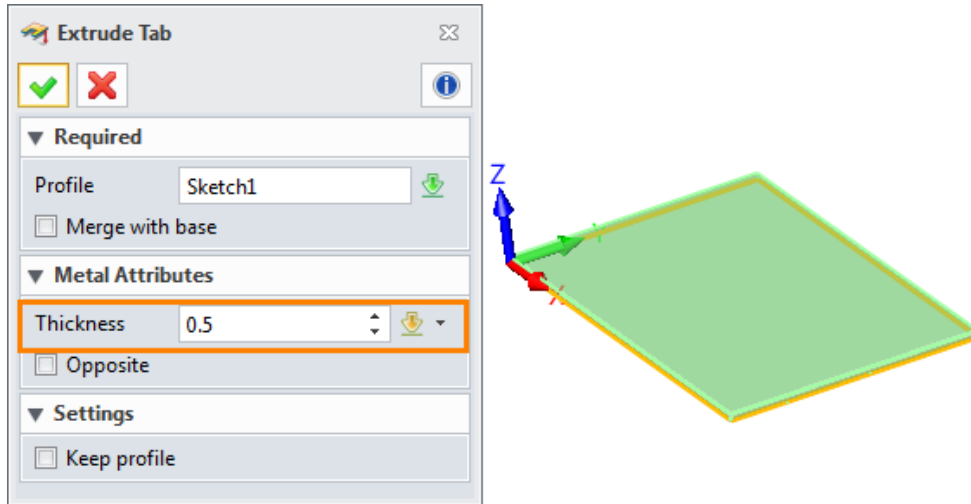


Figure15 Create base

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

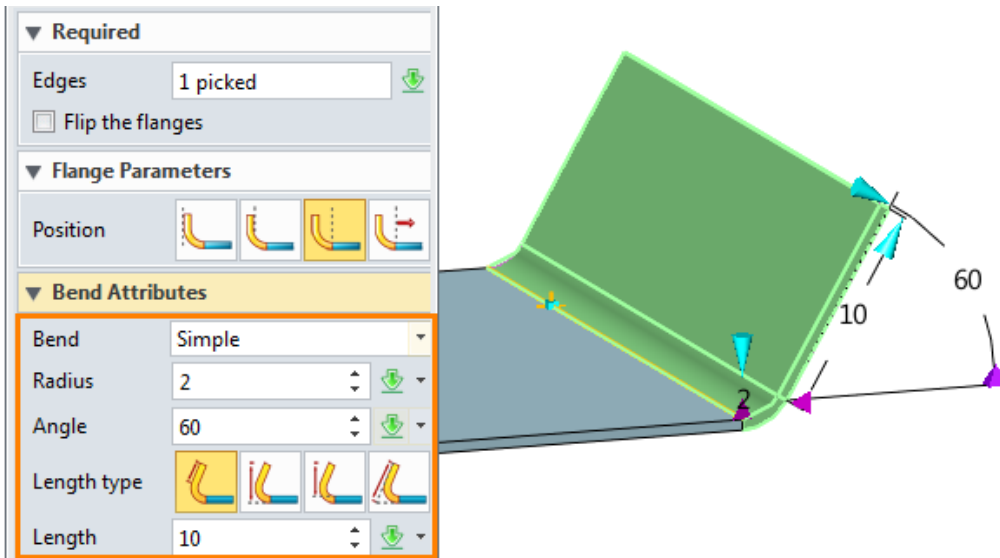


Figure16 Create full flange

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

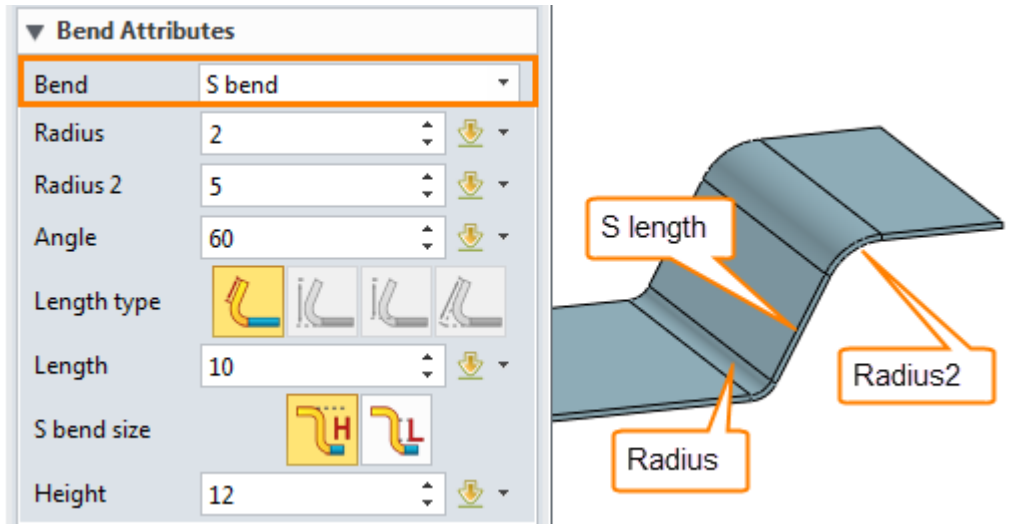


Figure17 Create S bend

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

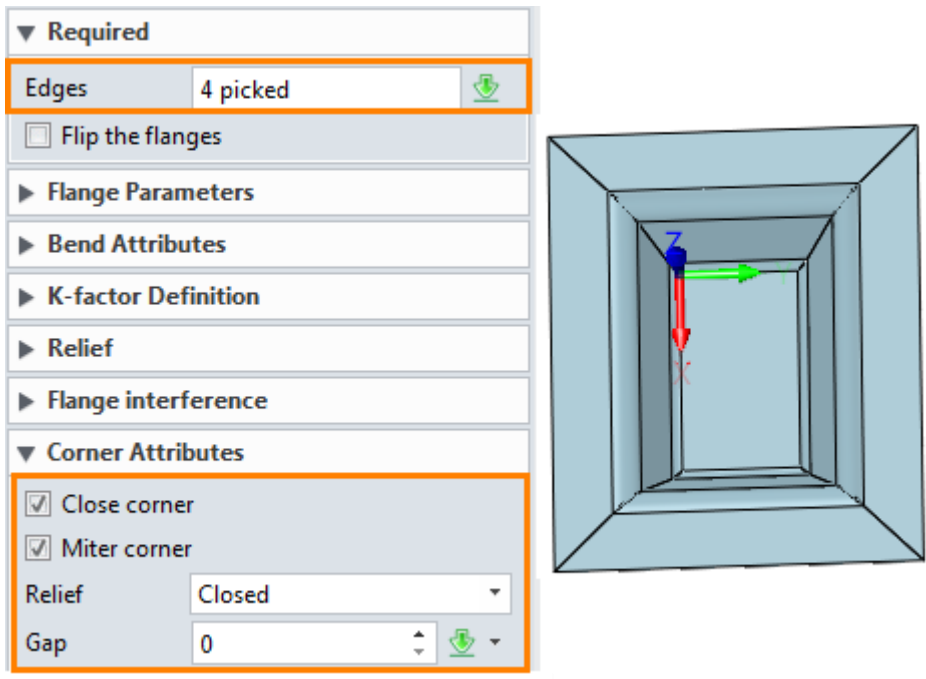


Figure18 Close corner

STEP 05 Add gap in corner.

STEP 06 Save this file as “04_Full Flange_S bend”.

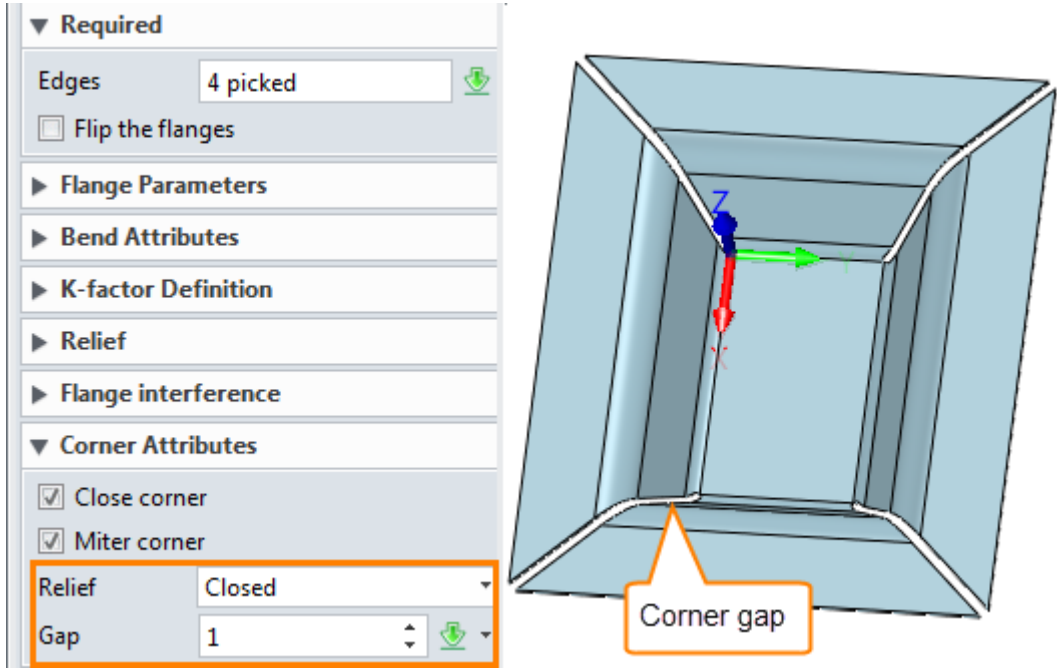
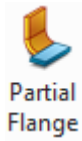


Figure19 Add gap for corner

3.2 Partial Flange



Sheet Metal ribbon toolbar->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 20.

STEP 03 Save this file as "05_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

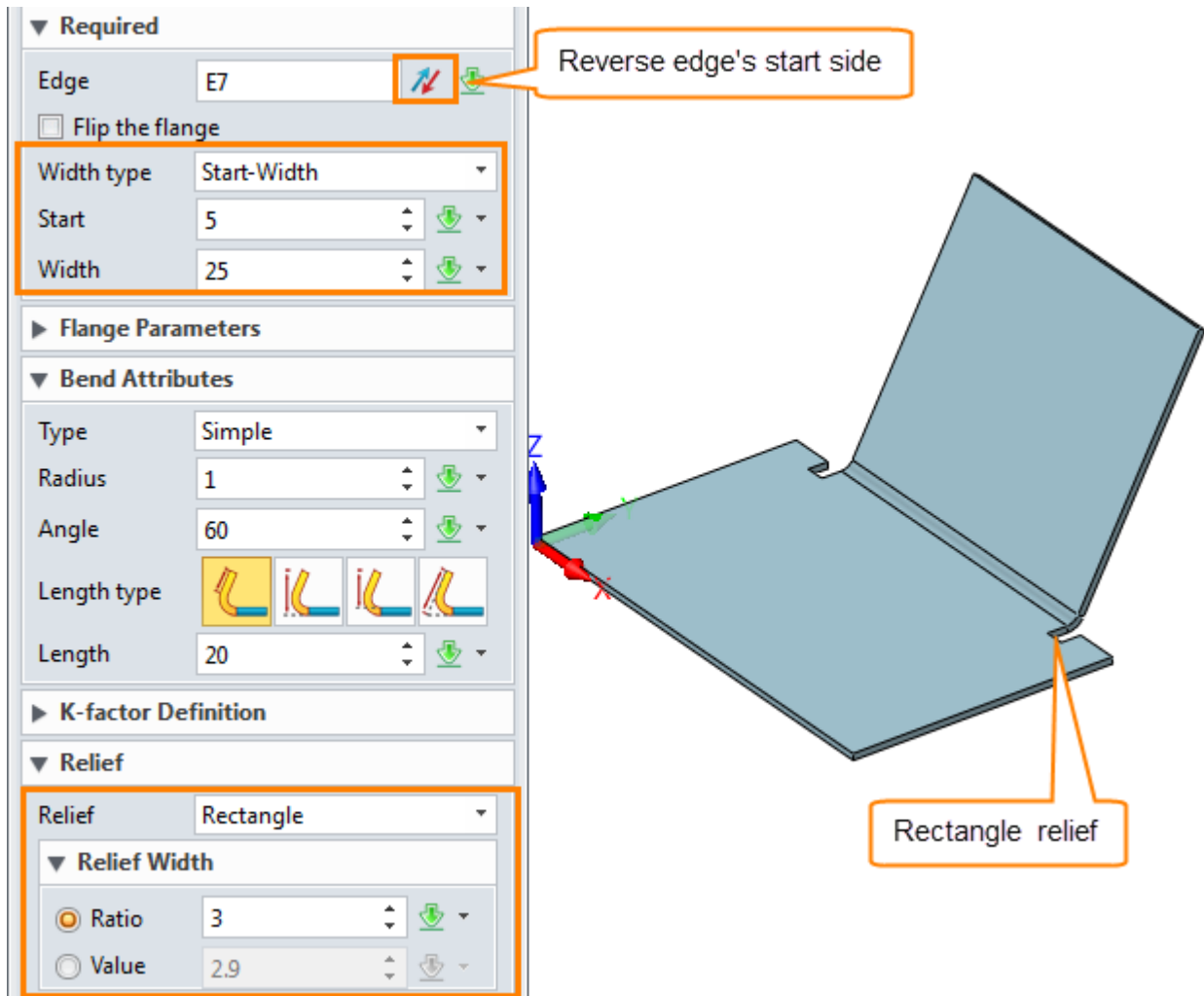


Figure20 Create Partial Flange

3.3 Hem Flange



Sheet Metal ribbon toolbar->Flange->

STEP 01 Open the file "Sheetmetal Function.Z3->06_Hem"

STEP 02 Choose Hem Flange to create hem flange as shown in Figure 21.

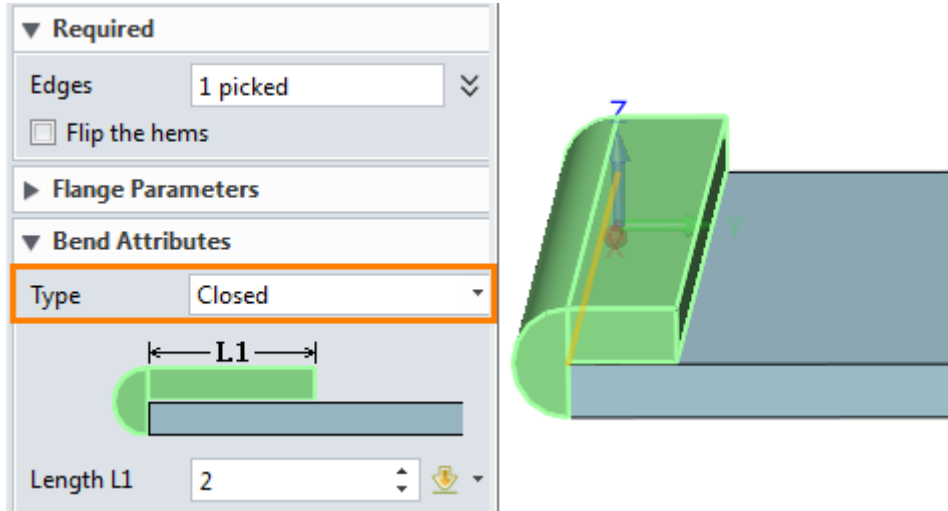


Figure21 Hem Flange

STEP 03 Try other bend types of Hem.

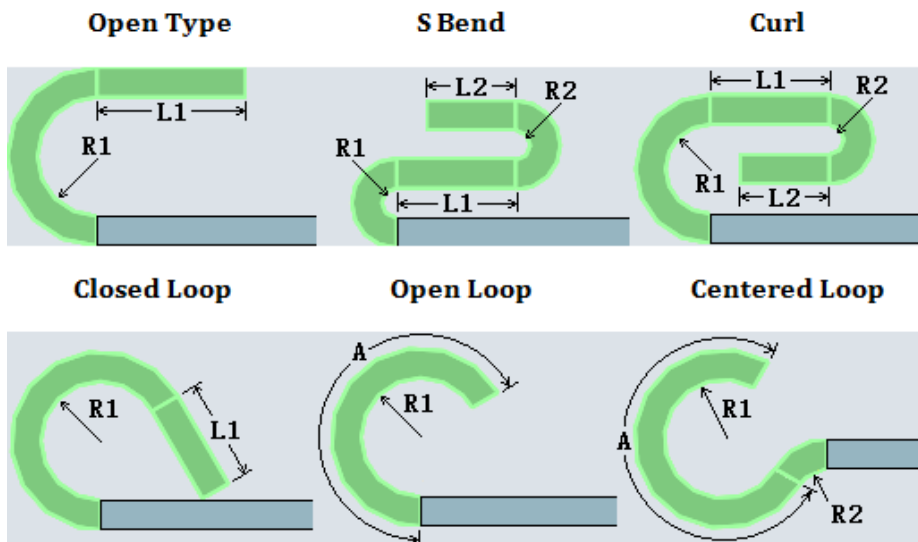


Figure22 Hem Flange Type

STEP 04 Create miter hem flange as follows.

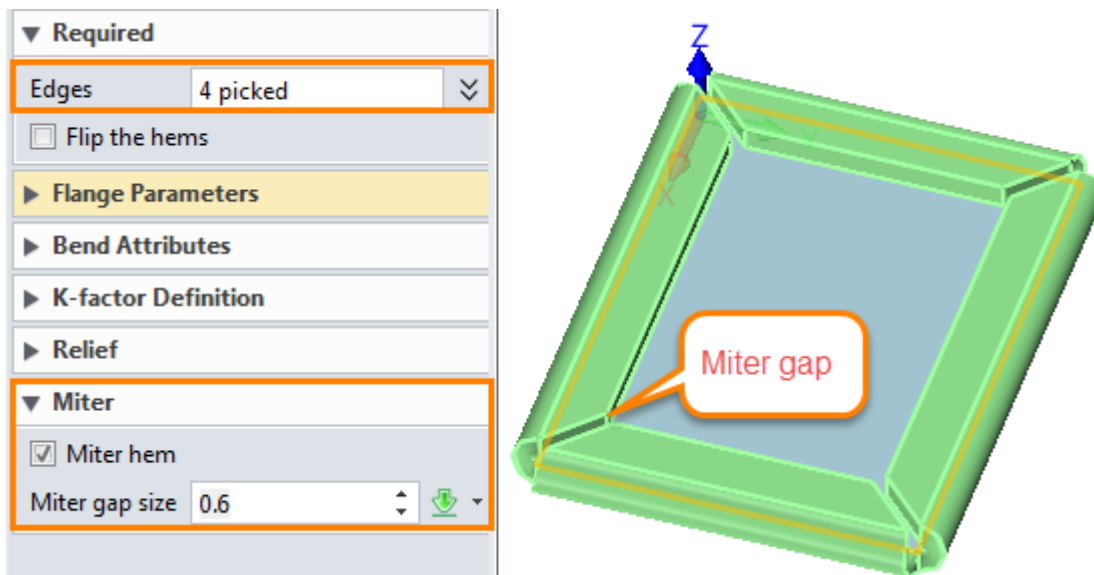
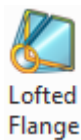


Figure23 Miter Hem Flange

3.4 Loft Flange



Sheet Metal ribbon toolbar->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:

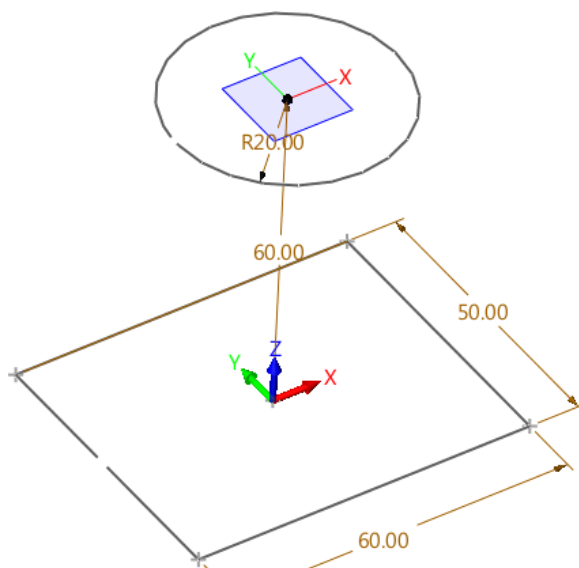


Figure24 Sketch for Loft flange

Note: 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".

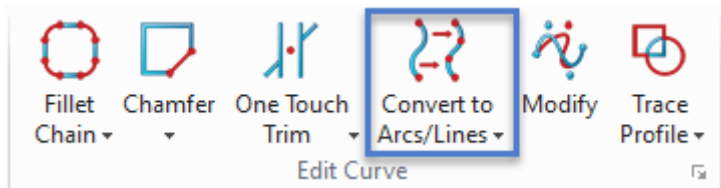


Figure25 Convert to Arc/Lines

2) Make sure the profile is open.

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

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

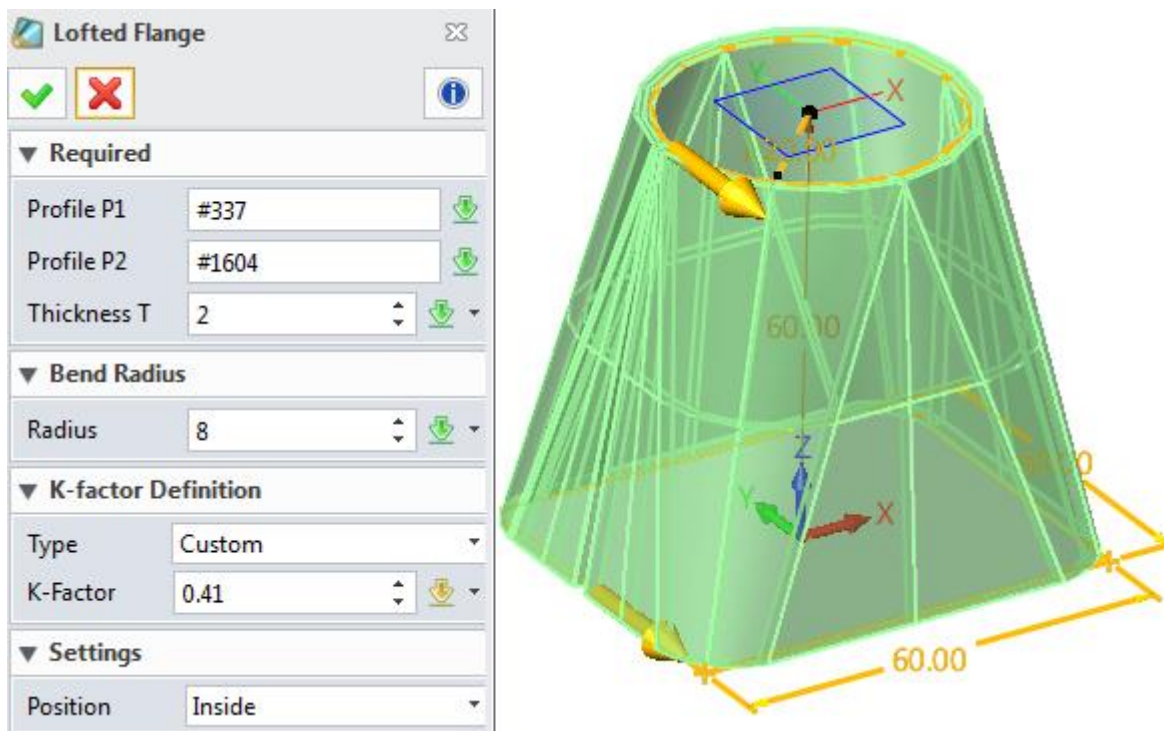


Figure26 Loft Flange

STEP 03 Unfold it.

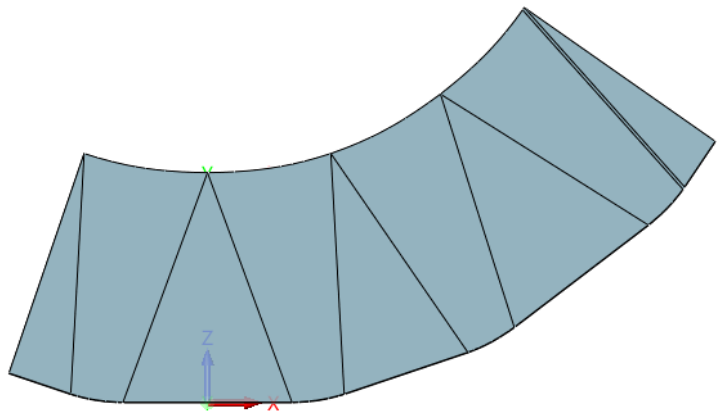
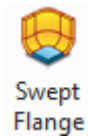


Figure27 Unfold loft flange

3.5 Swept Flange



Sheet Metal ribbon toolbar->Flange->

Use this command to create swept flange.

STEP 01 Create a base flange as follows.

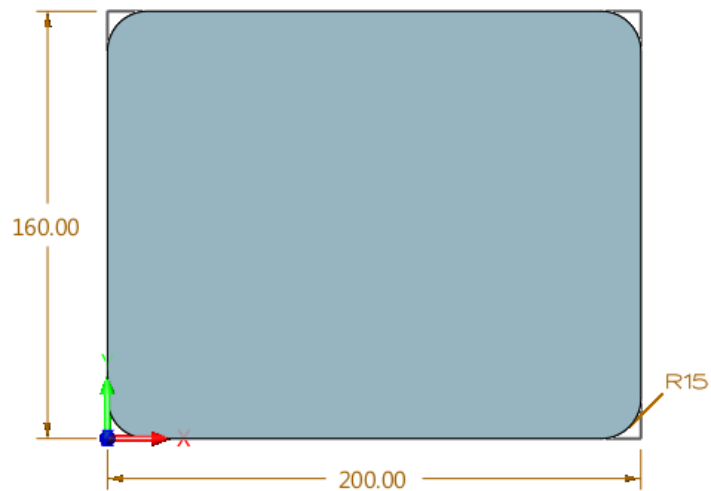


Figure28 Base for Swept flange

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

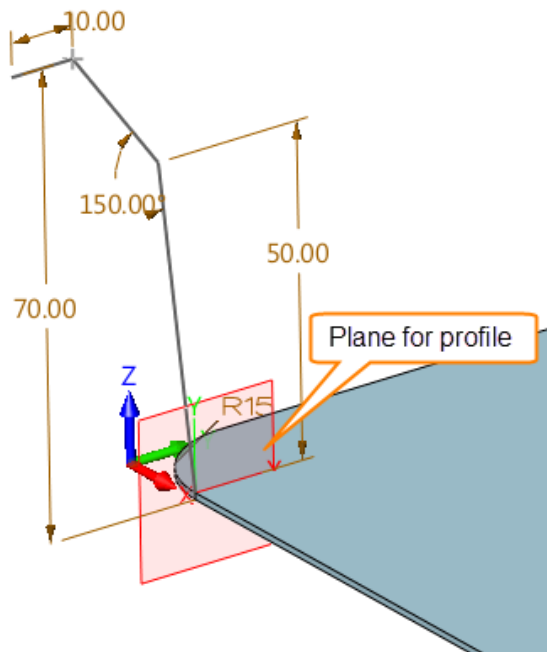


Figure29 Profile for Sweeping

Note: 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.

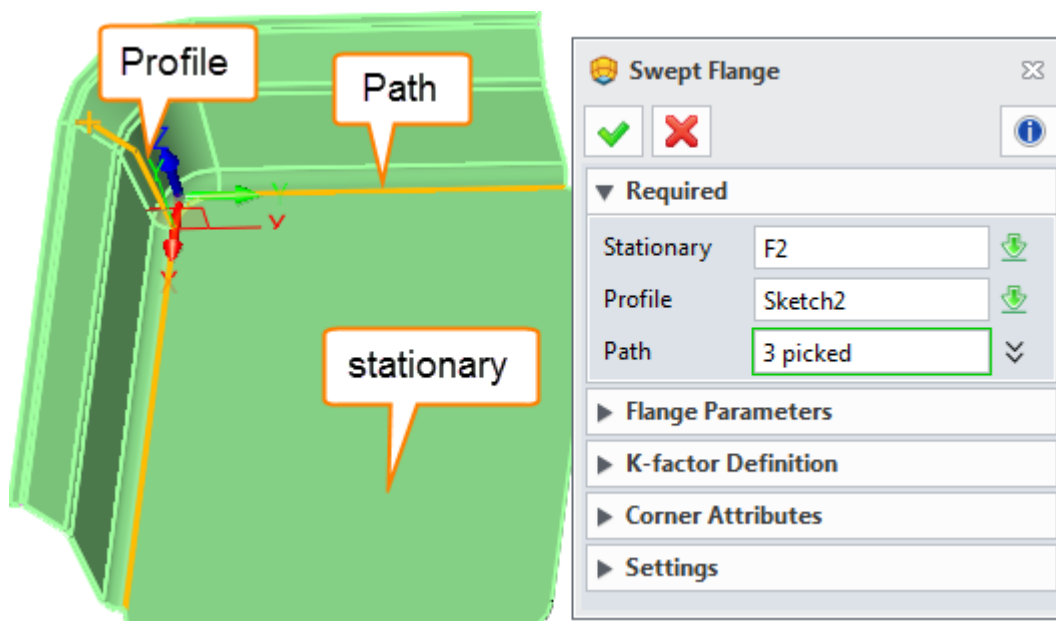


Figure30 Swept Flange

Tips: It is able to pick all of the continuous edges in one click by setting the filter as follows.

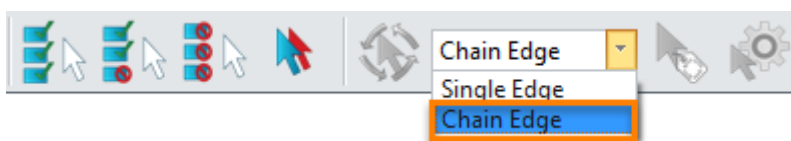


Figure31 Filter on Pick Tools Mini Bar

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

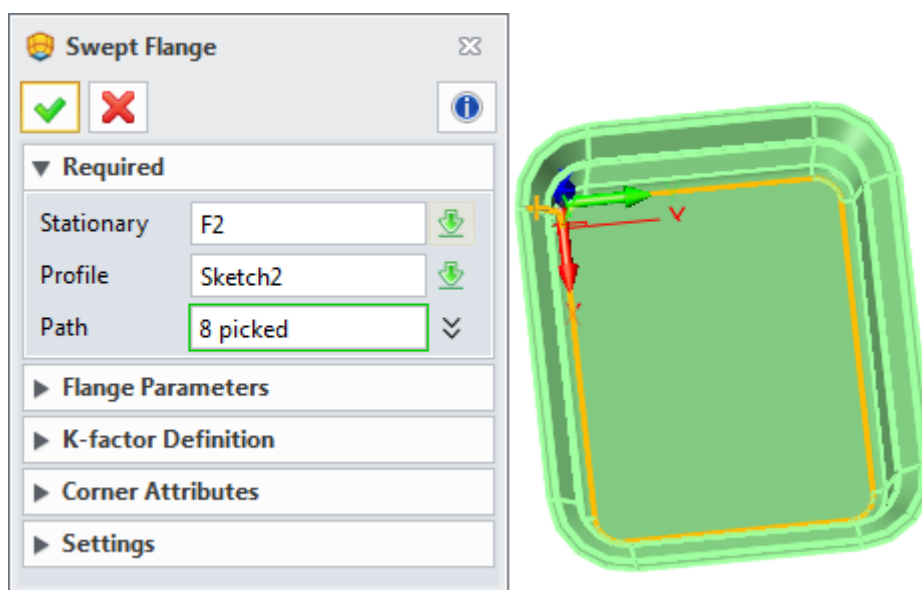


Figure32 Swept Flange with closed path

STEP 05 Unfold the swept flange.

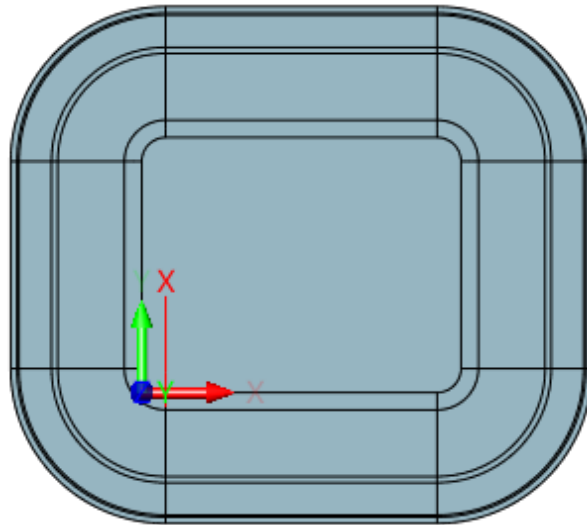


Figure33 Unfold the Swept Flange

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:

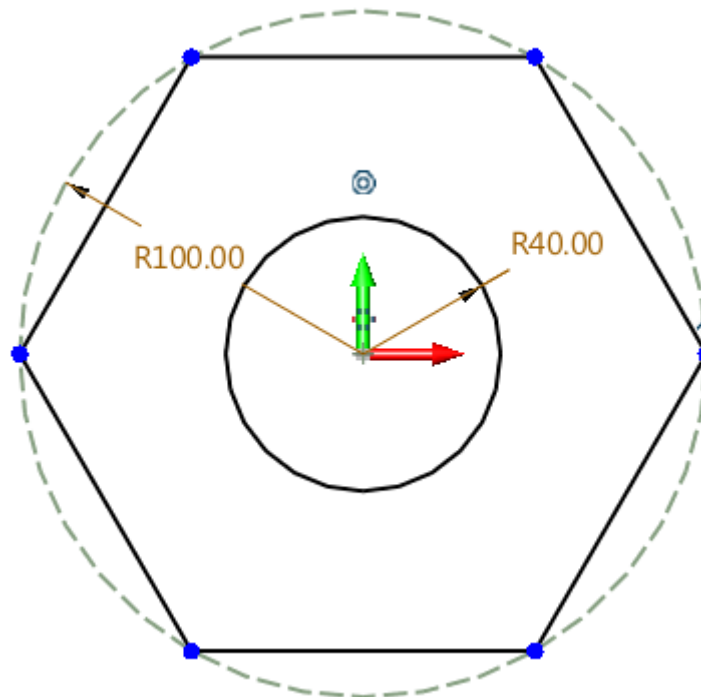


Figure34 Sketch for Swept Flange Base

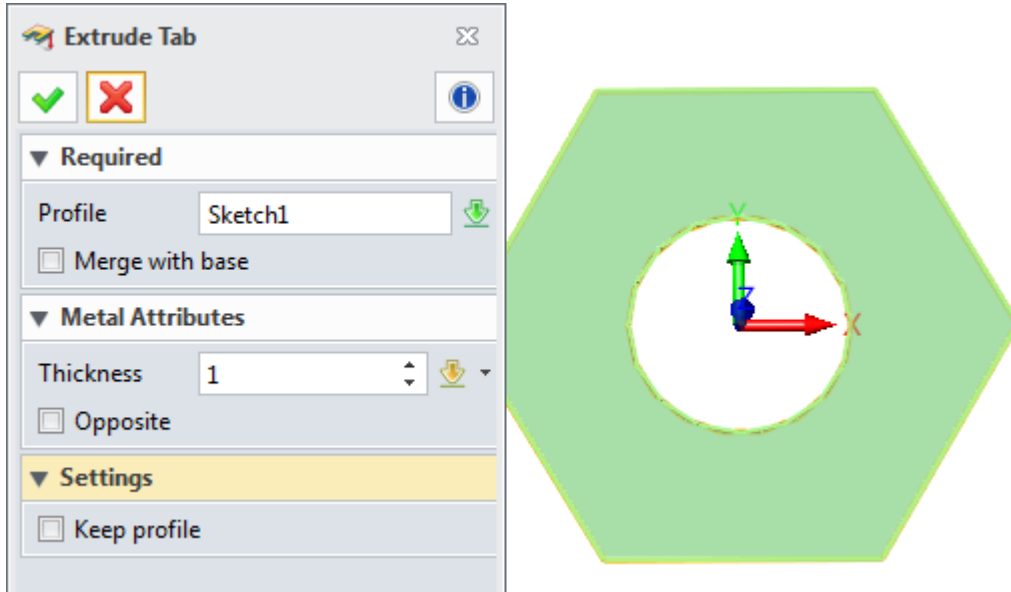


Figure35 Base for Swept flange

STEP 02 Create profile on assigned datum, as shown in Figure 36.

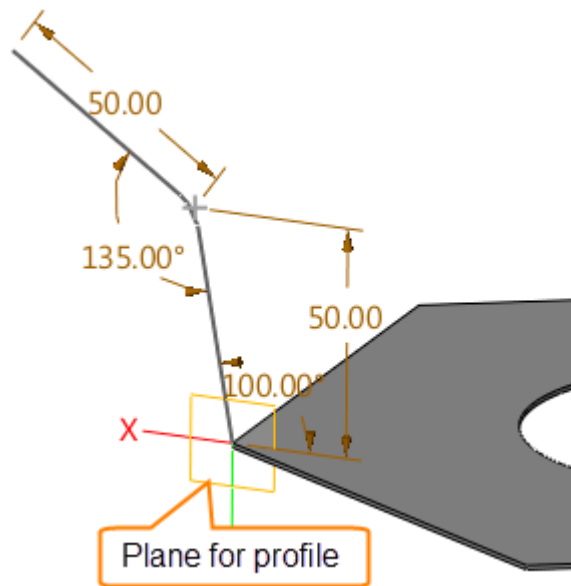


Figure36 Profile for Swept Flange along uncontinuous Path

STEP 03 Create swept flange as follows.

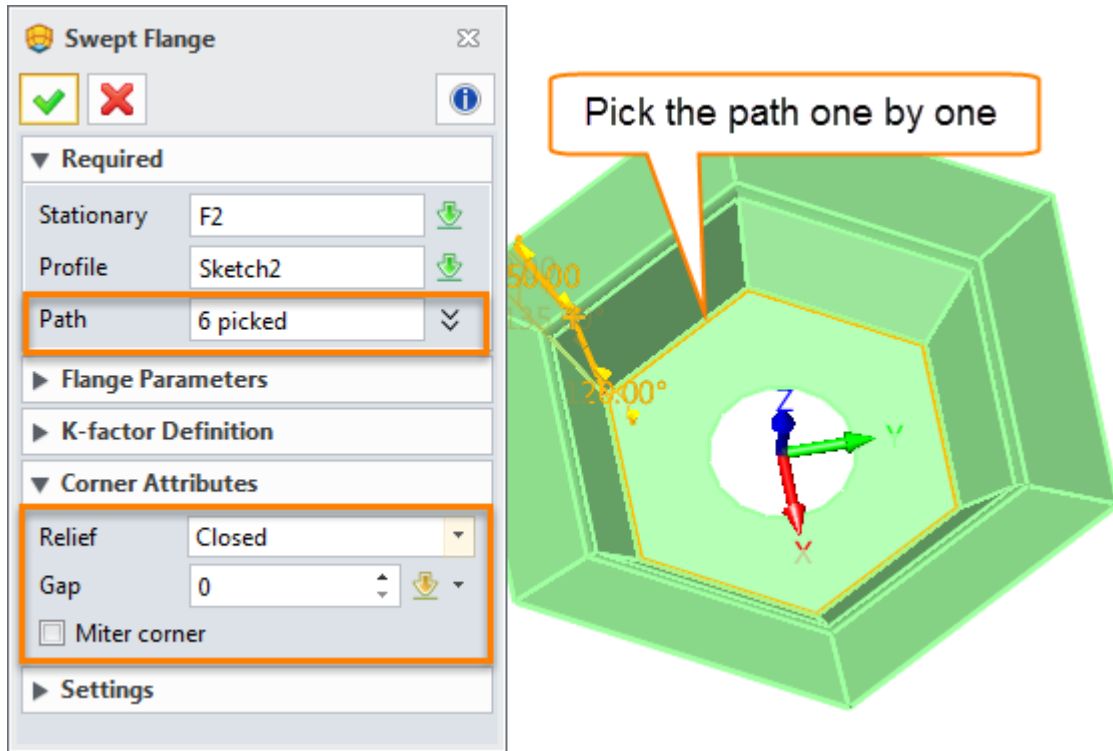


Figure37 Swept Flange along uncontinuous Path

STEP 04 Unfold the swept flange.

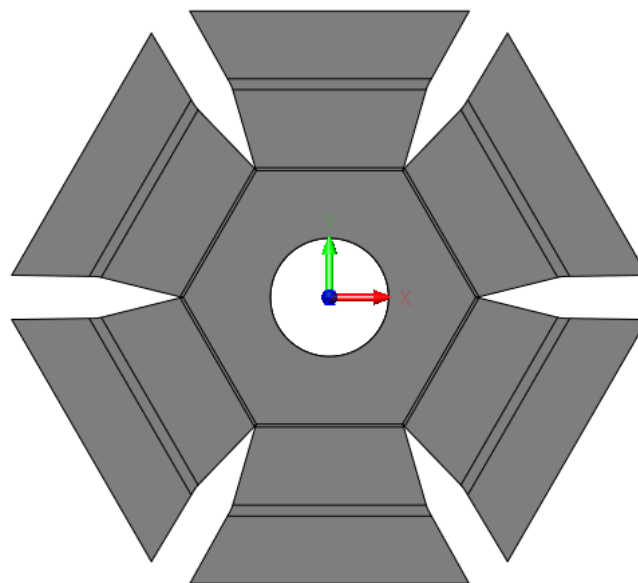
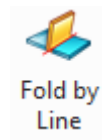


Figure38 Unfold Swept Flange on uncontinuous path

3.6 Fold by Line



Sheet Metal ribbon toolbar->Flange->

Use this command to create bends along a line.

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

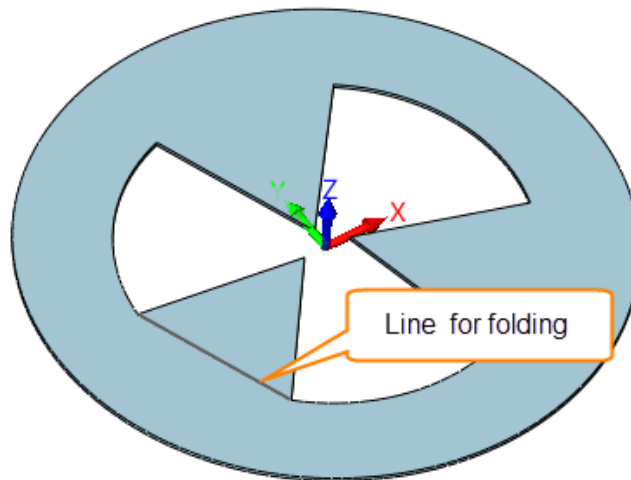


Figure39 Open File “Fold by line”

Note: 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.

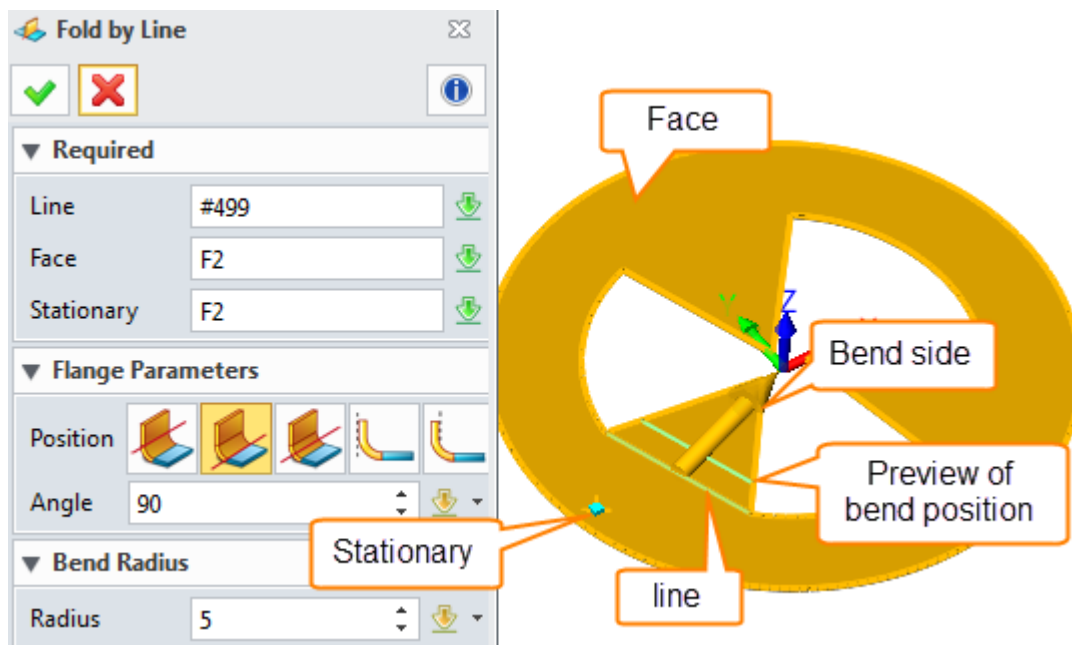


Figure40 Parameter for Fold by Line

STEP 03 Fold result is shown in Figure 41.

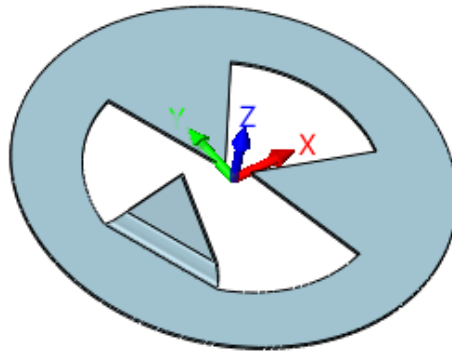


Figure41 Fold by line Result

STEP 04 Do the same process for other parts.

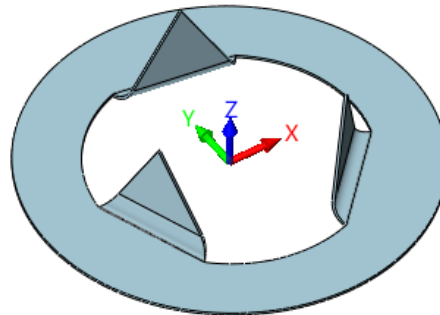
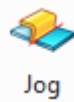


Figure42 Finished Fold by line

3.7 Jog



Sheet Metal ribbon toolbar->Flange->

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

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

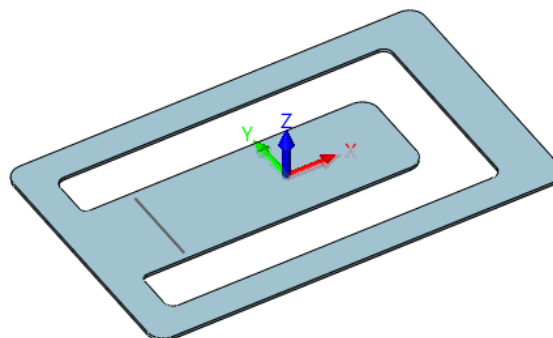


Figure43 Jog part

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

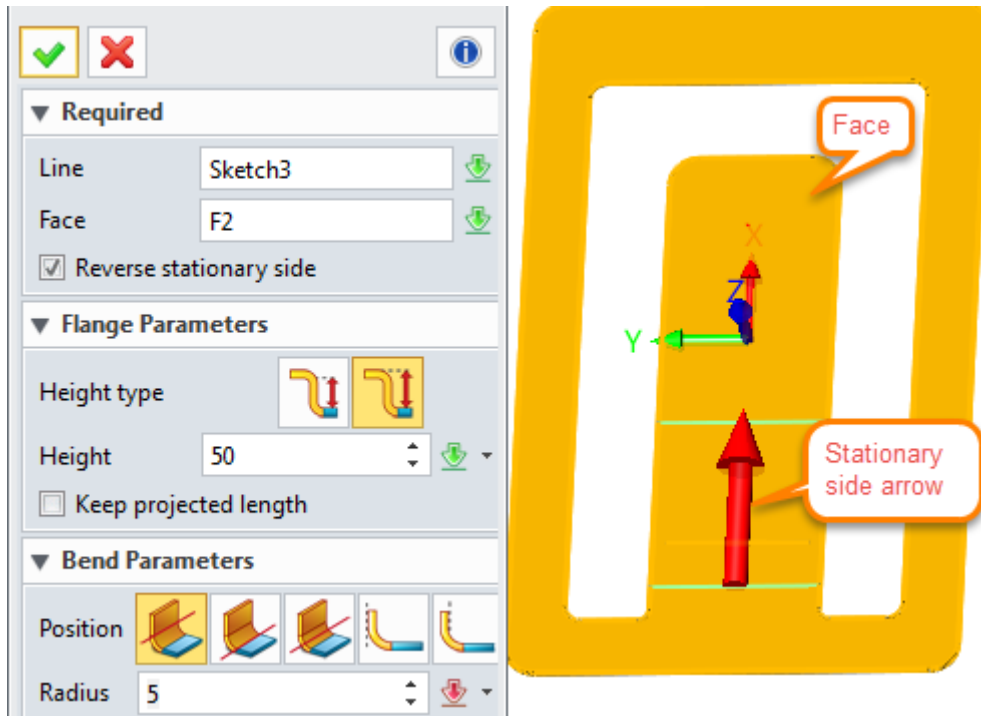


Figure44 Create Jog

STEP 03 Confirm it and then get the following result.

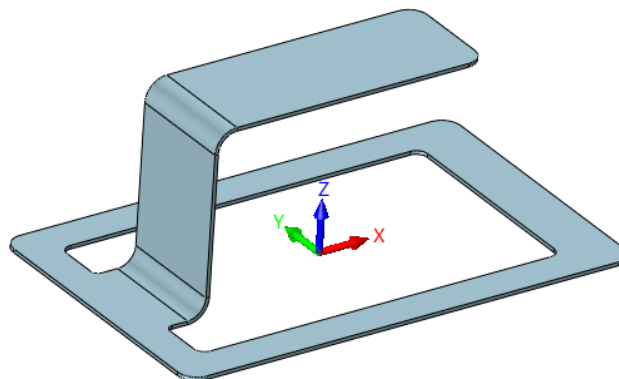


Figure45 Jog shape

4. Editing Tools

4.1 Extend Flange



Sheet Metal ribbon toolbar->Editing->

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.

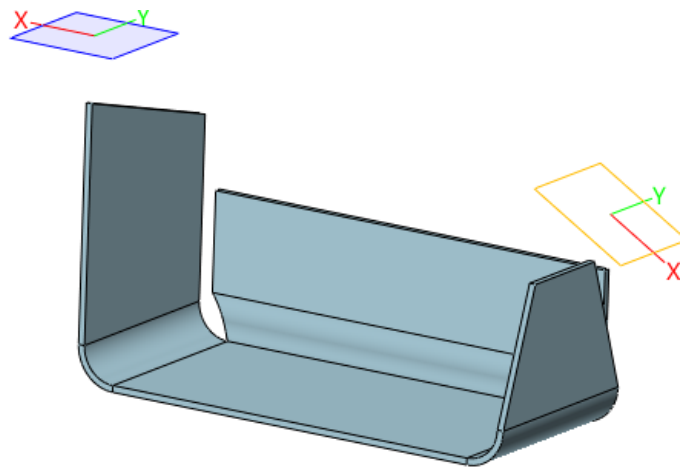


Figure46 Extend flange file

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

a. Extend by distance

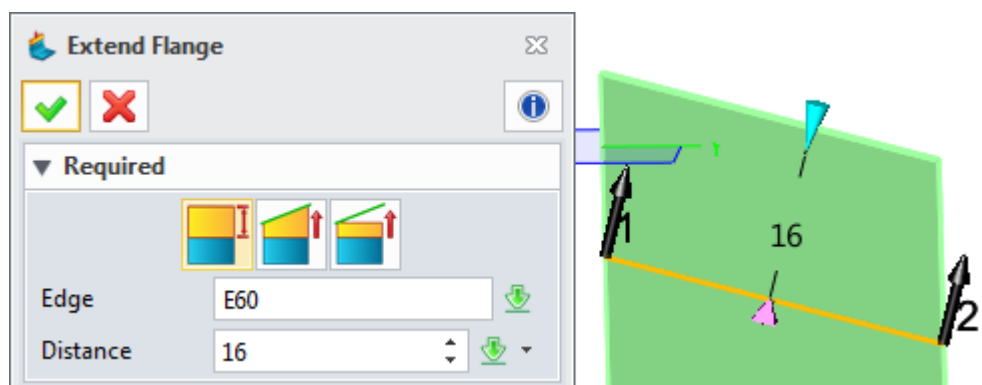


Figure47 Extend Flange by Distance

b. Extend flange through until intersection with plane

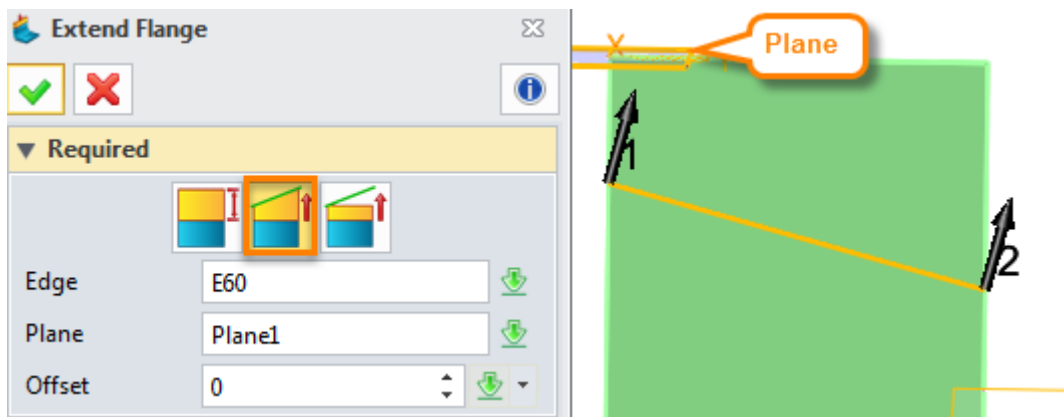


Figure48 Extend Flange through until Intersection with Plane

c. Extend flange up to selected plane

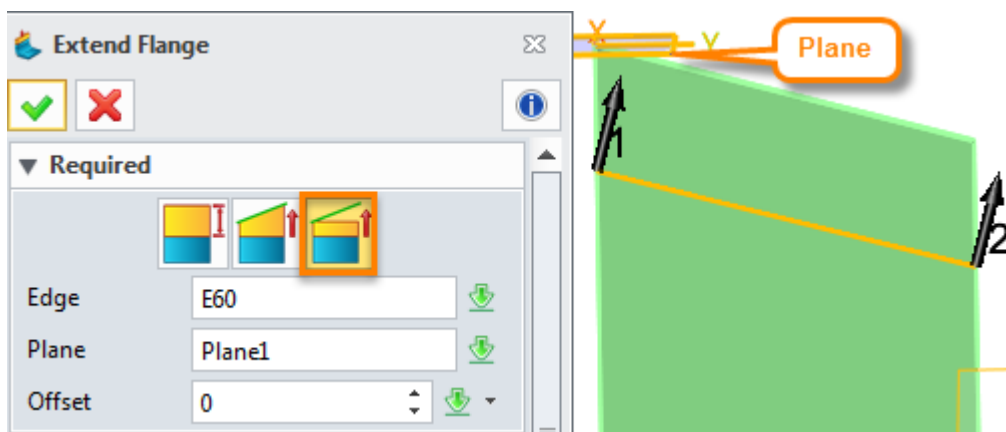


Figure49 Extend Flange up to Selected Plane

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

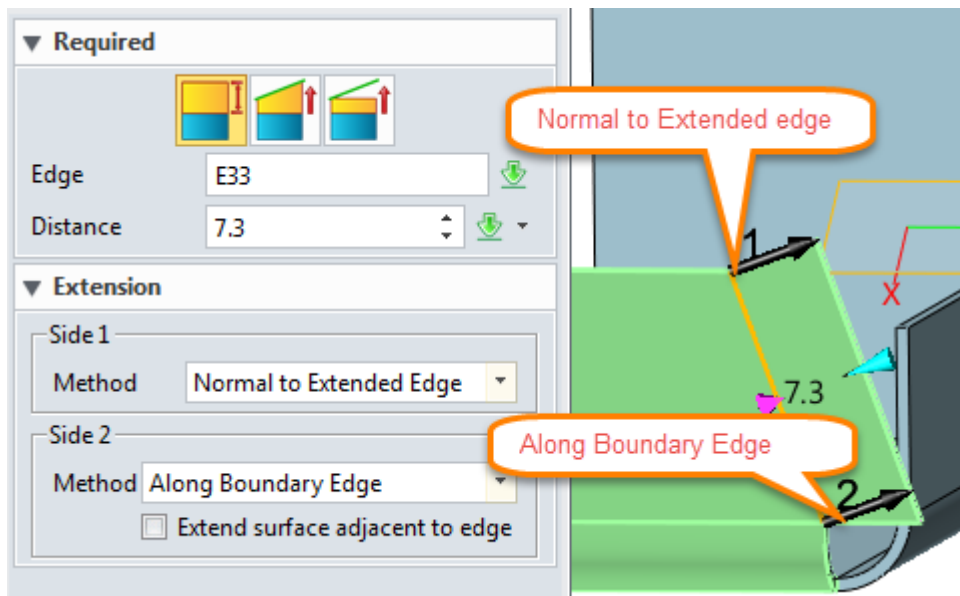


Figure50 Different Extension Method

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

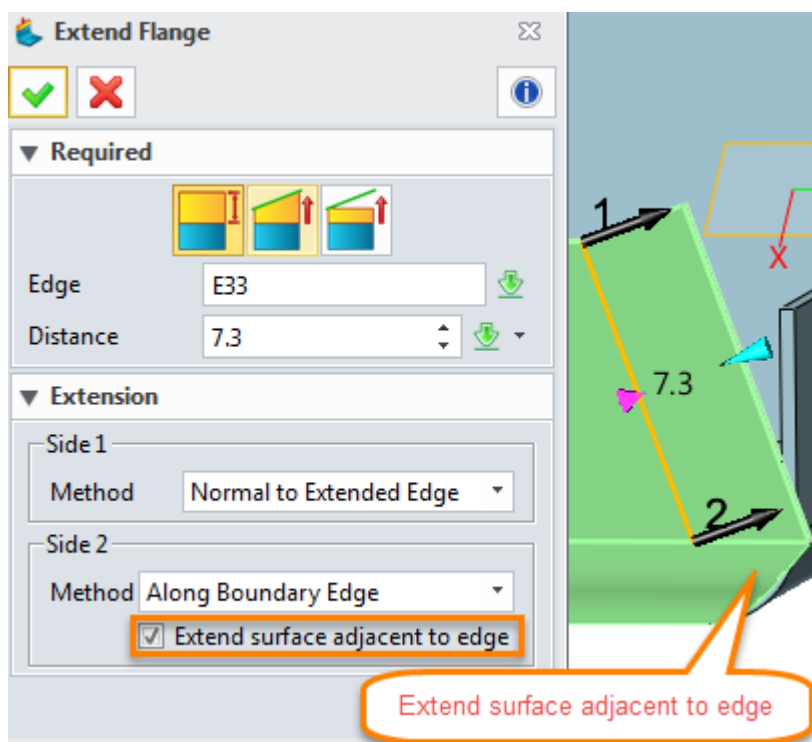


Figure51 Extend surface adjacent to the edge

4.2 Bend Taper



Sheet Metal ribbon toolbar->Editing->

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.

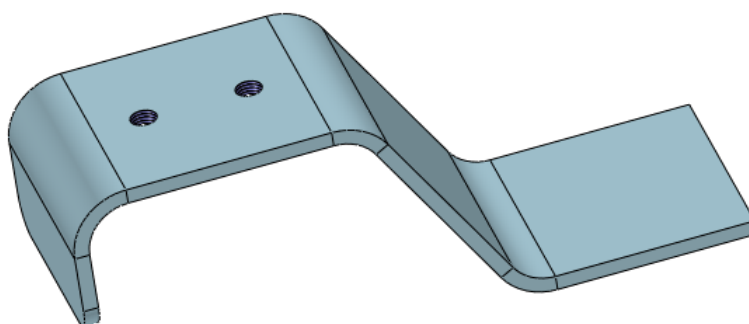


Figure52 Bend Taper part

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

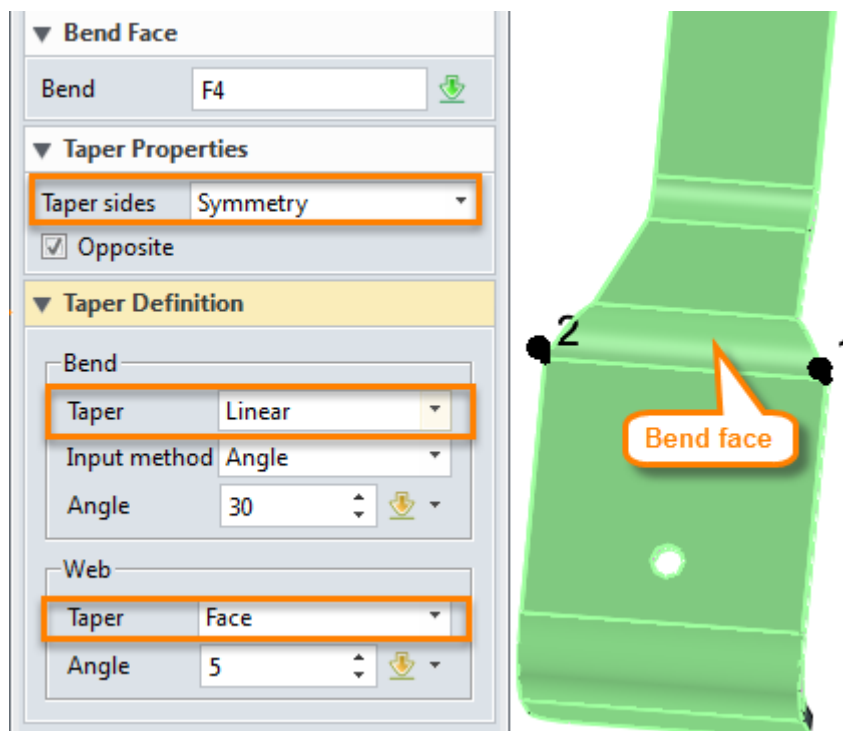


Figure53 Bend Taper

Bend Taper definition

There are two methods (Linear/Tangent) to define bend taper, as shown in Figure 54. The edge of the bend is tangent to the web when the tangent method is used.

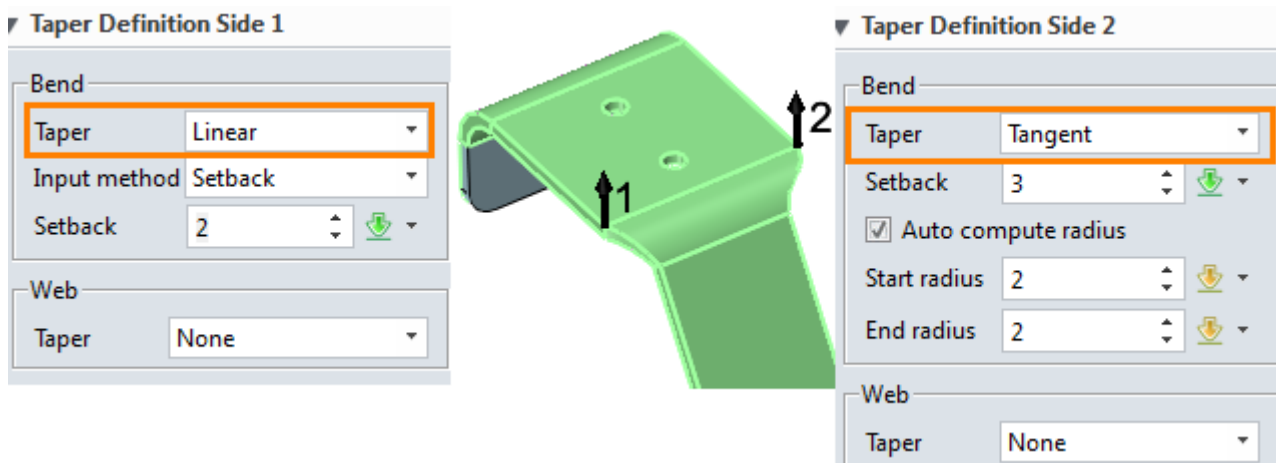


Figure54 Bend Taper

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.

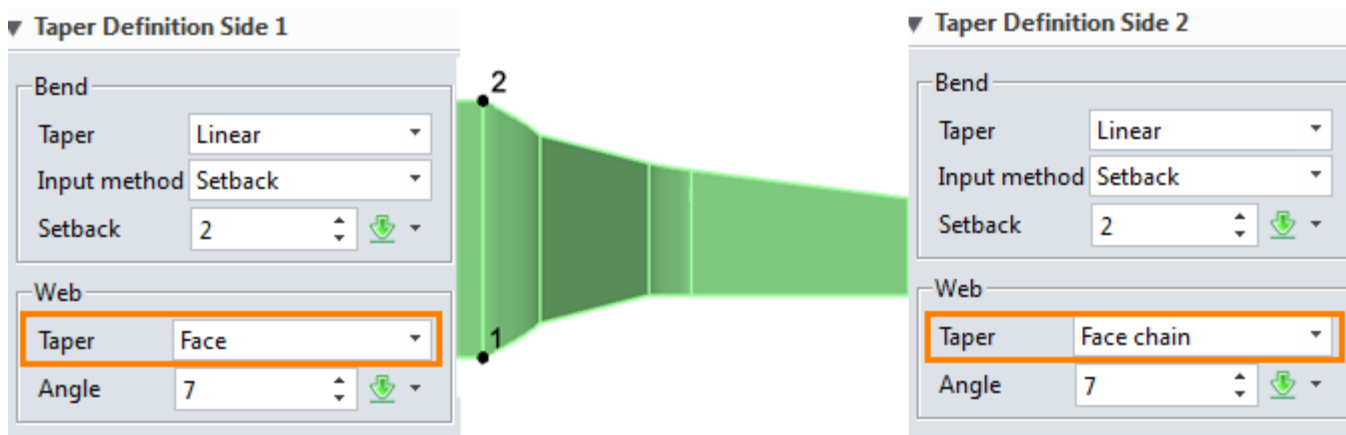
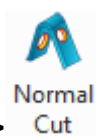


Figure55 Web Taper

4.3 Normal Cut



Sheet Metal ribbon toolbar->Editing->

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.

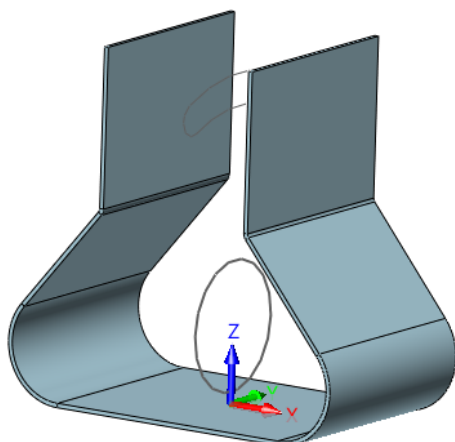


Figure56 NormalCut File

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

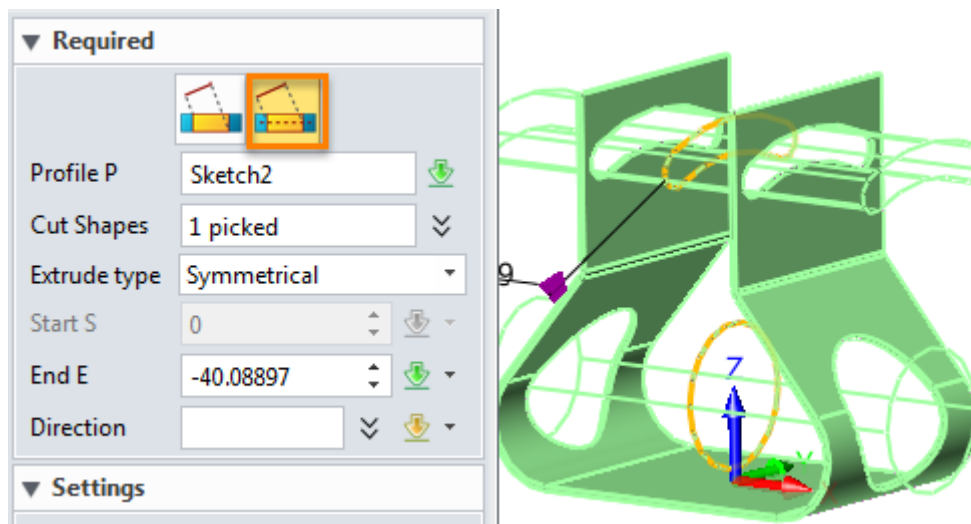


Figure57 Normal Cut to Remove Material

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

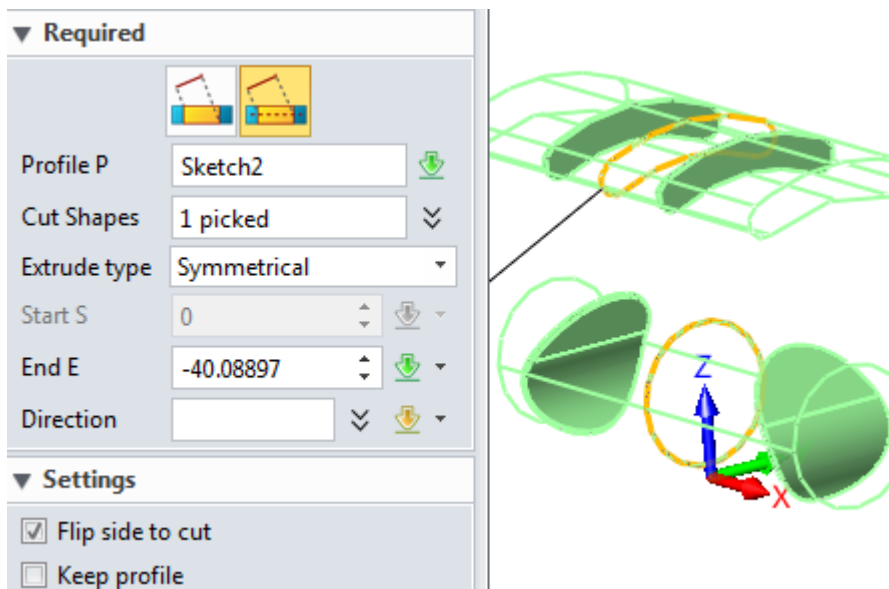


Figure58 Normal cut to fall the material

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

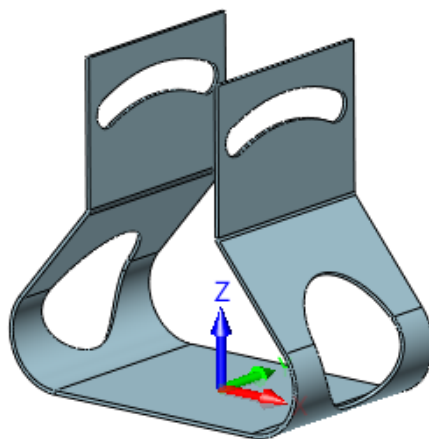


Figure59 Normal Cut Result

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

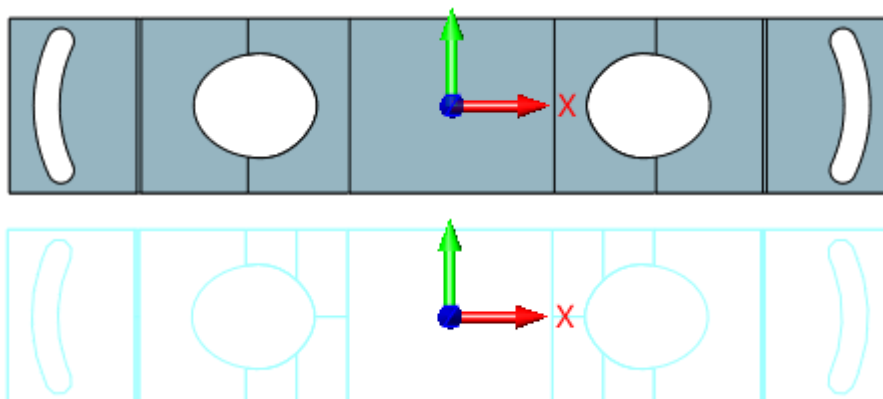



Figure60 Unfold the Normal Cut Result

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.

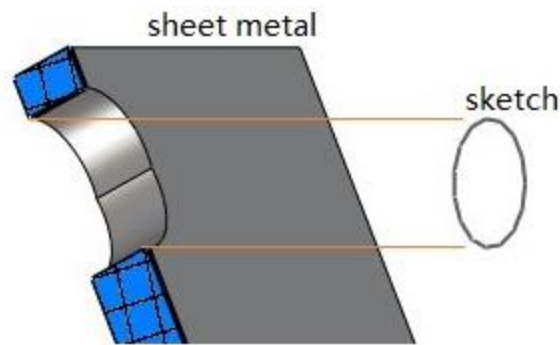



Figure61 Normal to Both Sides

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.

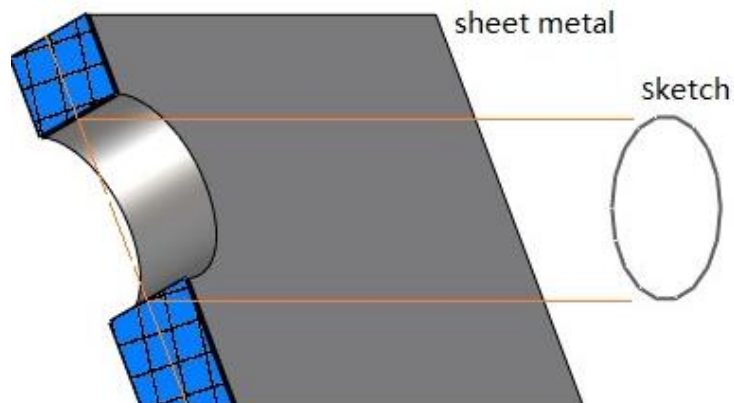


Figure62 Normal to Middle

5. Corner

5.1 Close Corner



Close Corner

Sheet Metal ribbon toolbar->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.

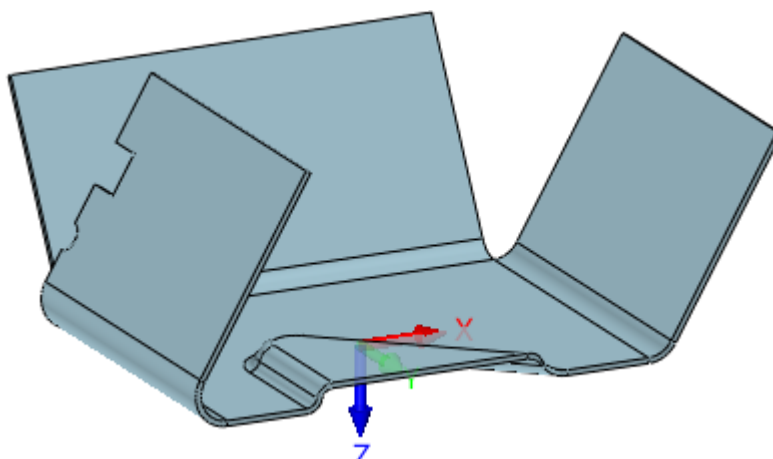


Figure63 Close Corner Part

STEP 02 Choose Close Corner command and apply the first type-Edge, as shown in Figure 64.

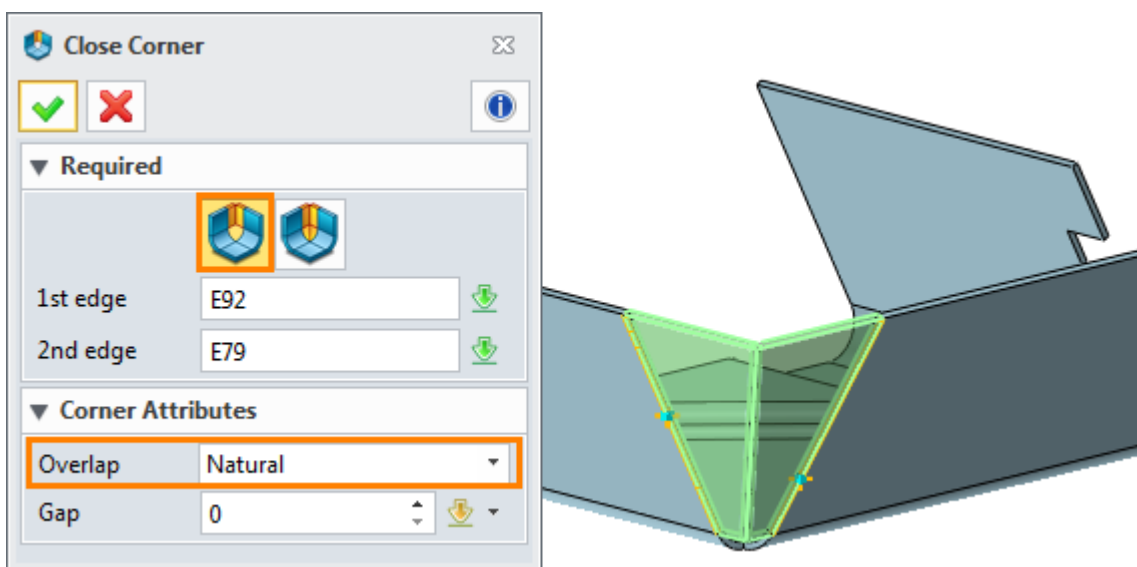


Figure64 Close Corner by Edge

Note: 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.

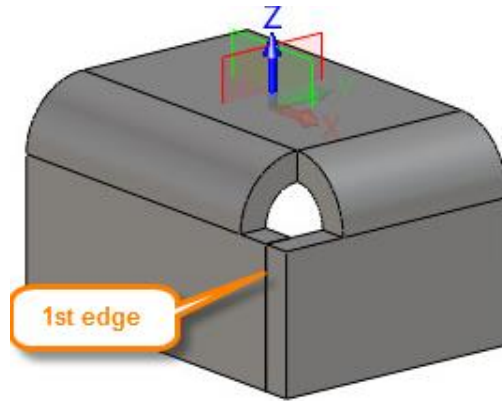


Figure65 Underlap Flange Corner

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

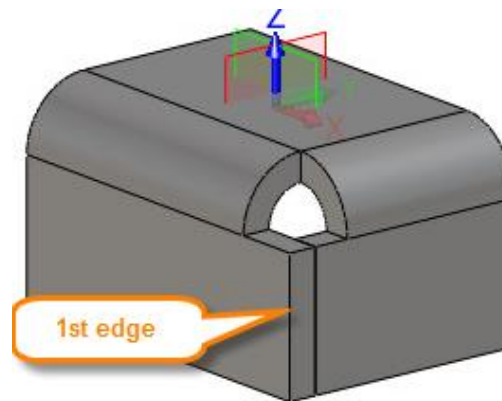


Figure66 Overlap Flange Corner

Natural: Both edges will meet naturally.

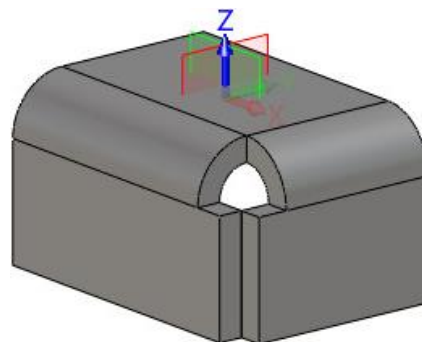


Figure67 Natural Closed Flange Corner

Figure68

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

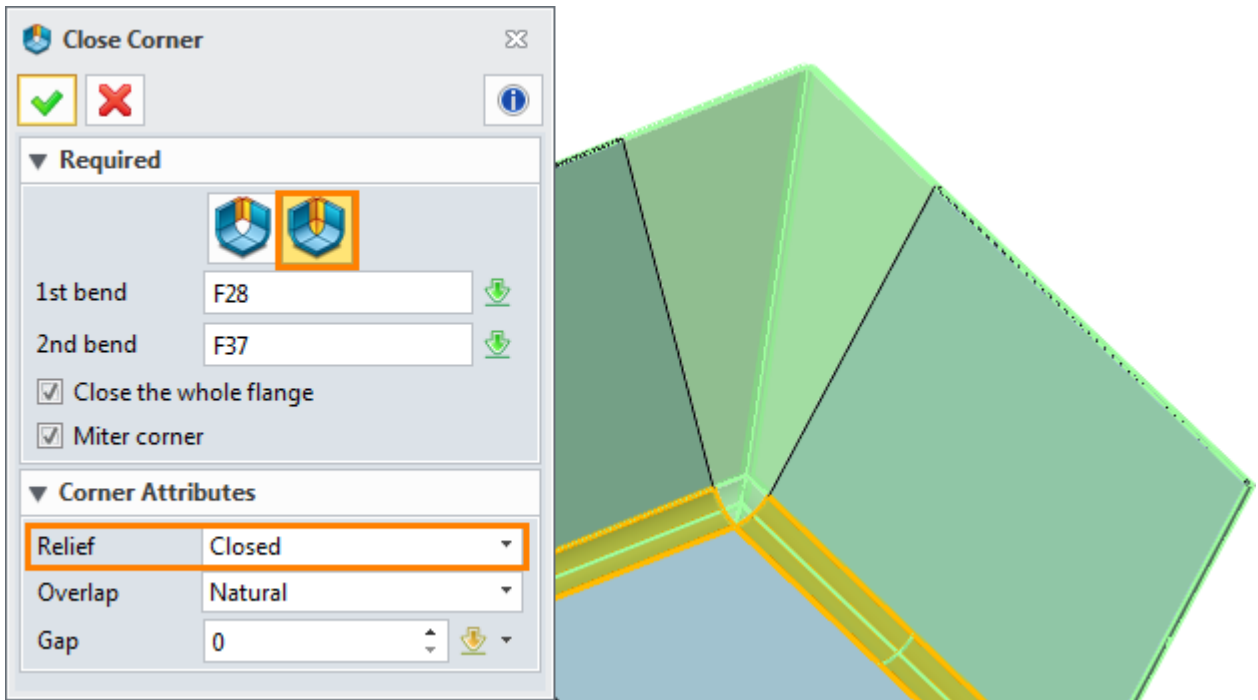


Figure69 Close the Corner by Bend Face

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.

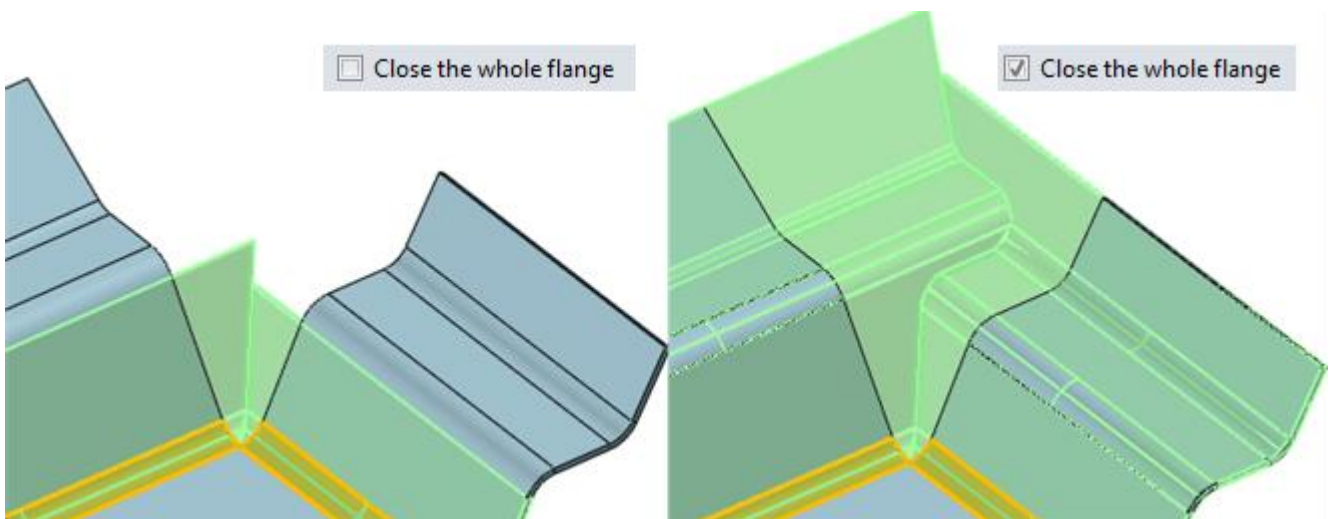


Figure70 Close the Whole Flange

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 70.

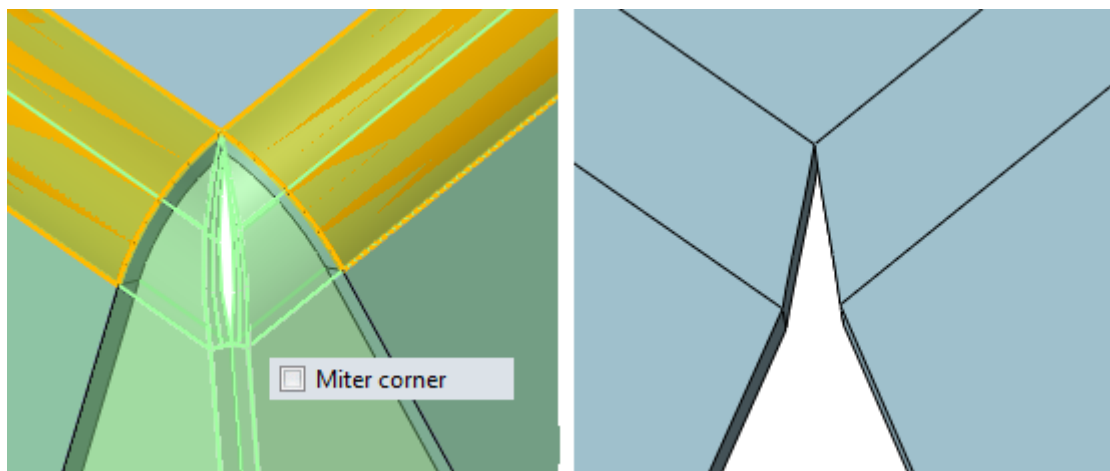


Figure71 Unfolded Un-mitered Corner

Check this option, the corner will be close to meet the gap shape. Then the unfold shape is not linear as shown in Figure 71.

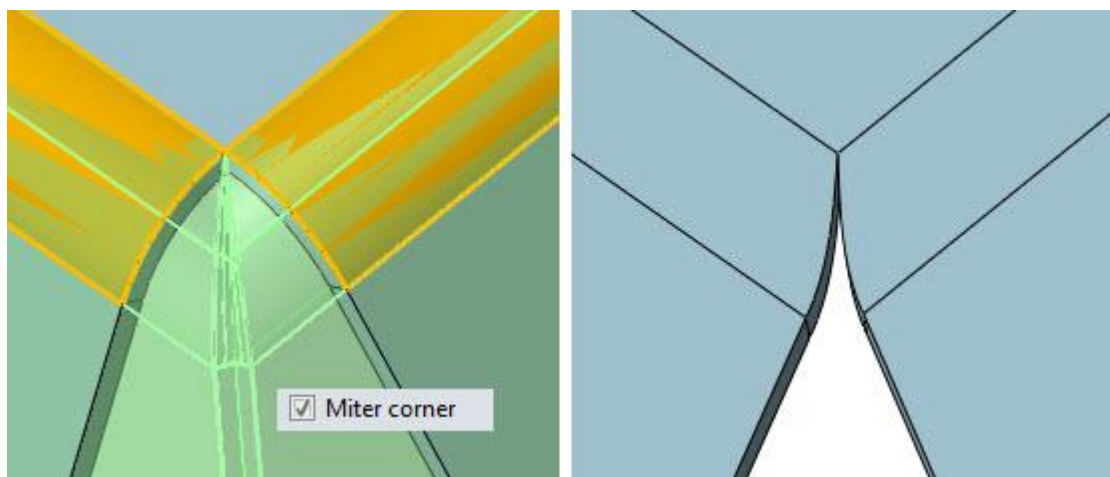


Figure72 Unfolded Mitered Corner

6. Forming Tools

6.1 Dimple



Sheet Metal ribbon toolbar->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.

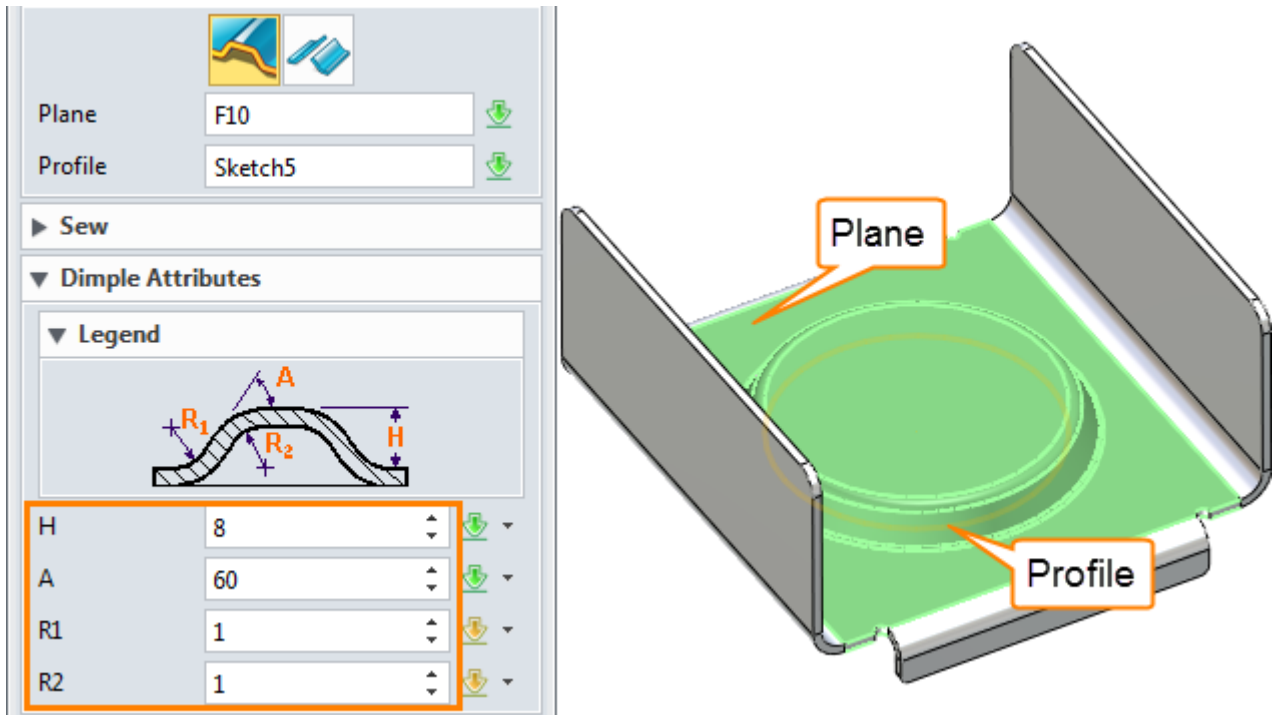


Figure73 Create Plain Dimple

Note: 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 73.

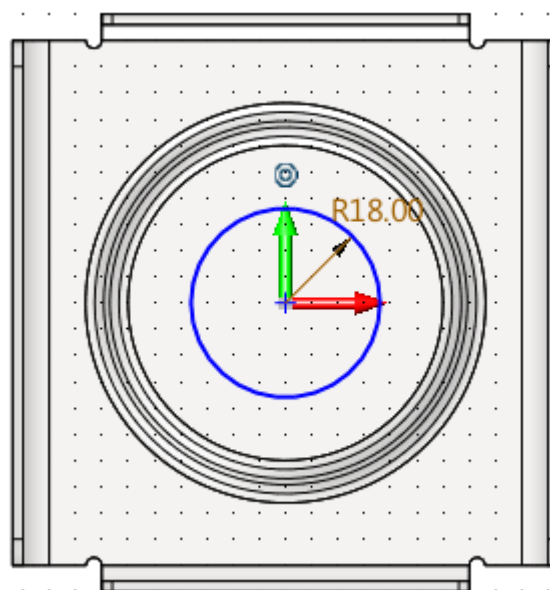


Figure74 Sketch on Top Face of Plain Dimple

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

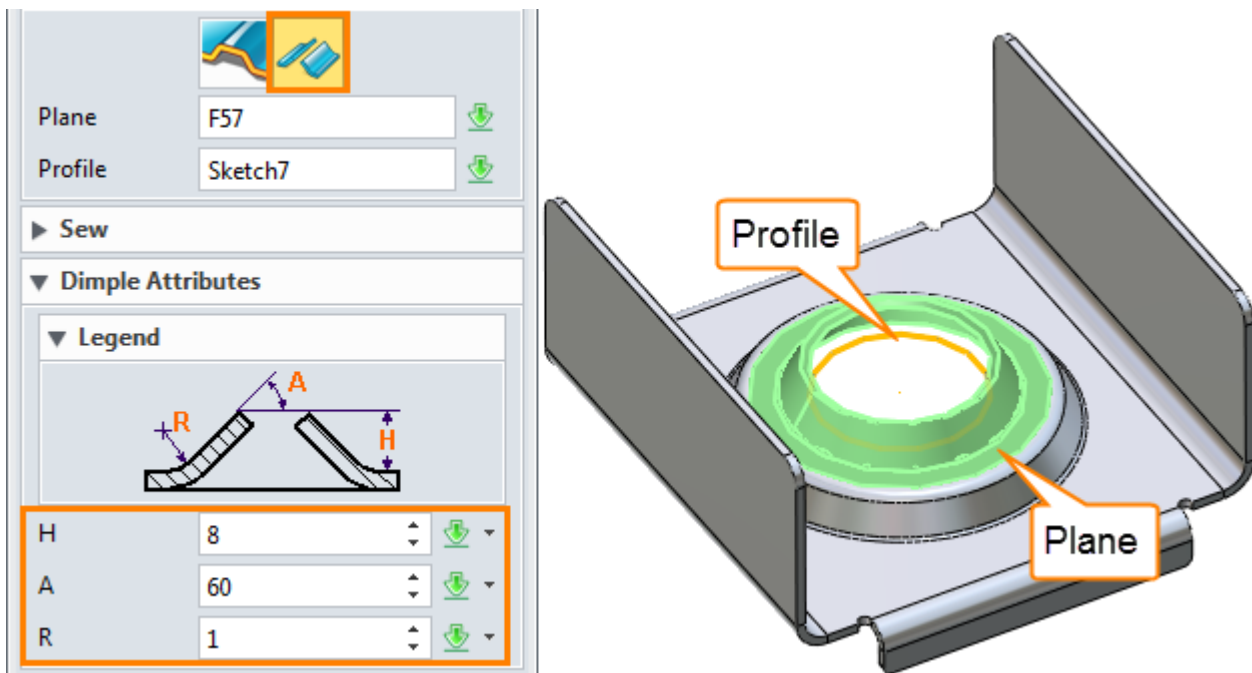


Figure75 Flare Dimple

6.2 Louver



Sheet Metal ribbon toolbar->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 75.

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

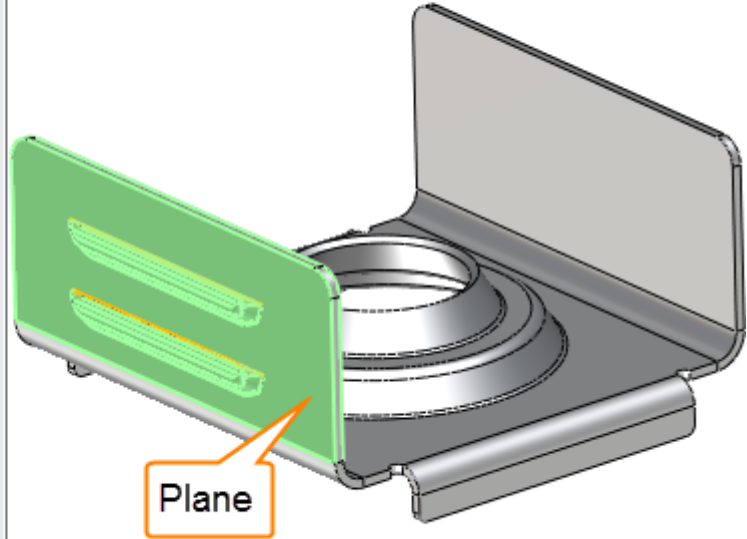
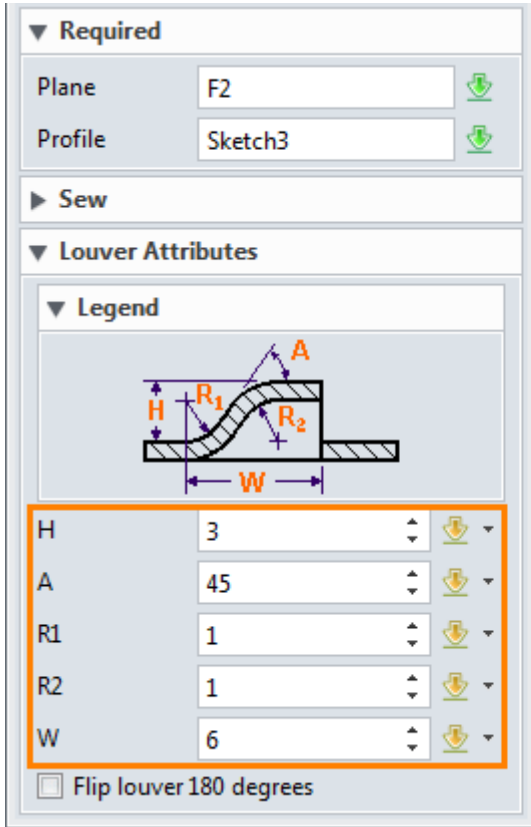


Figure76 Create Louver

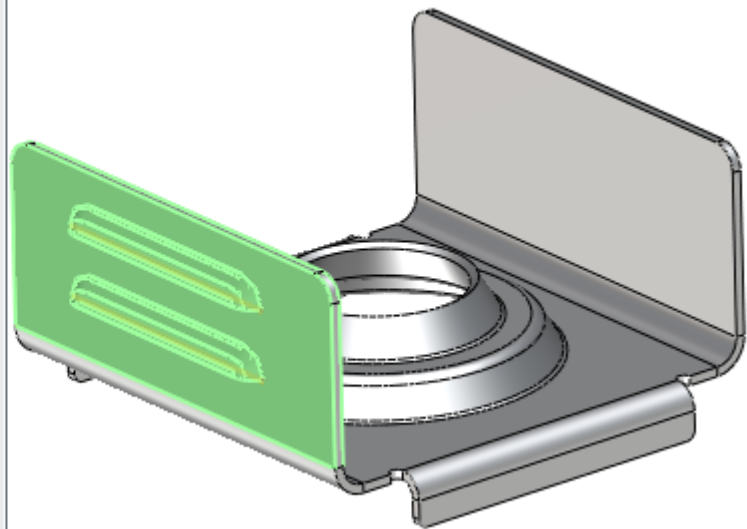
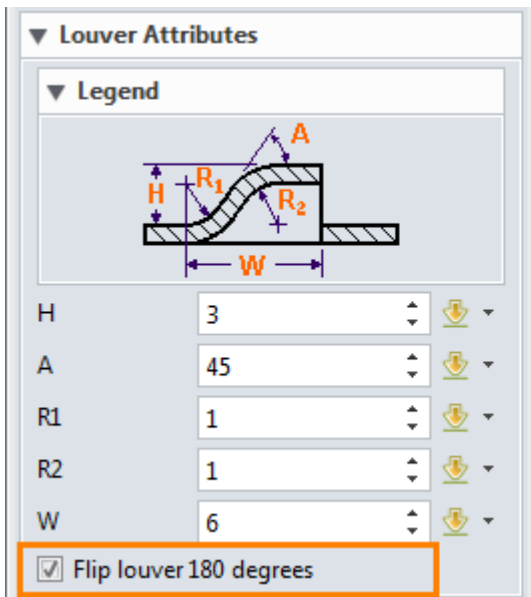


Figure77 Flip Louver 180 Degrees

STEP 03 Finish the louver in another side as follows.

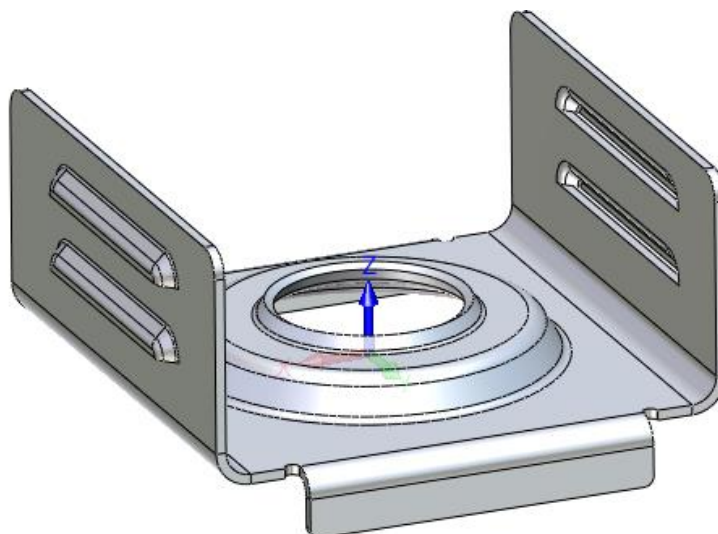


Figure78 Louver in two Sides

6.3 Punch



Sheet Metal ribbon toolbar->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.

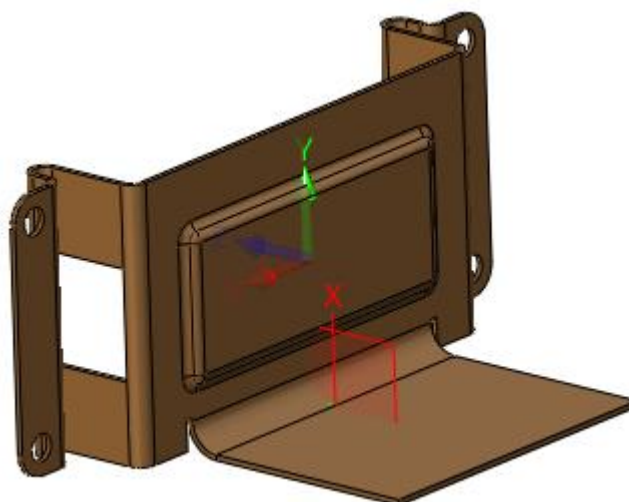


Figure79 Punch Part

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

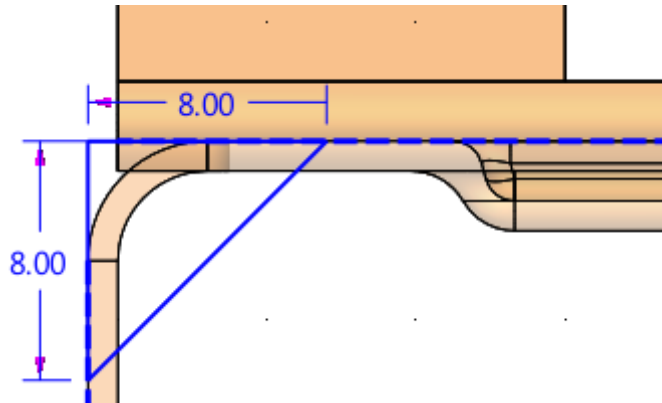


Figure80 Sketch for Puncher

STEP 03 Extrude the sketch as the puncher.

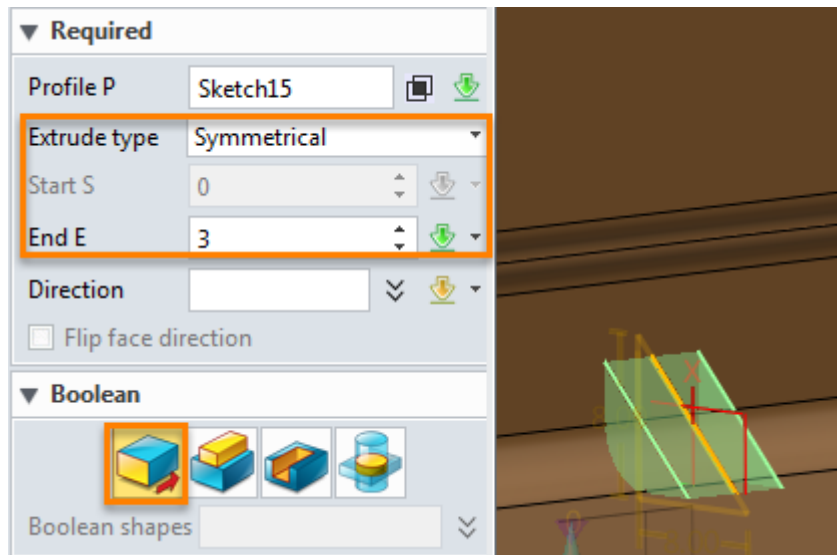


Figure81 Extrude the Sketch as Puncher

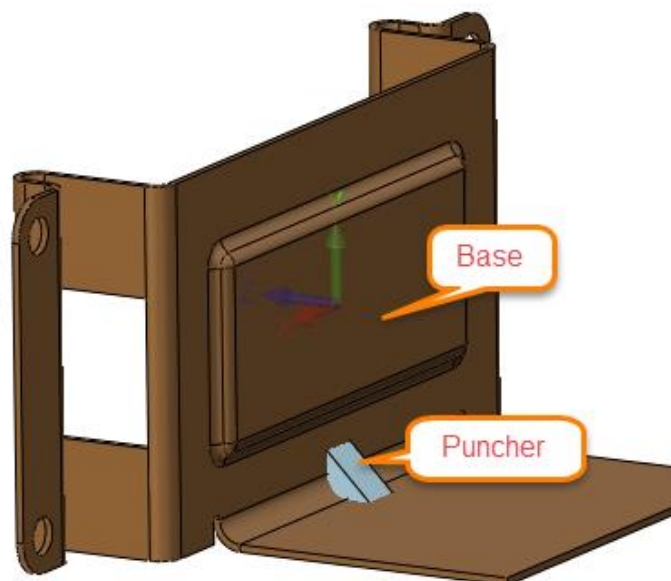


Figure82 Puncher and Base

STEP 04 Punch a stiffener as shown in Figure 82.

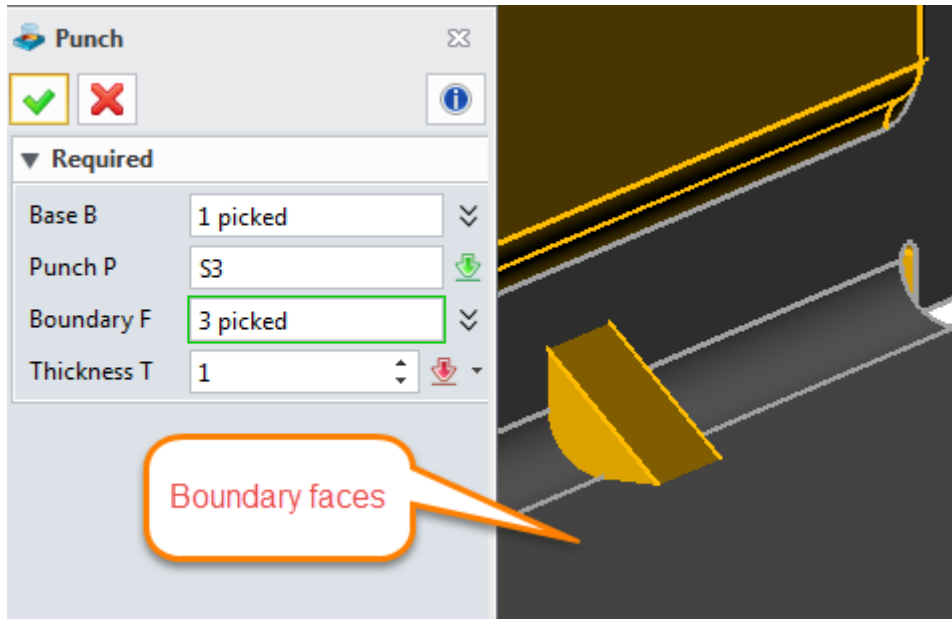


Figure83 Punch a Stiffener

STEP 05 Confirm it to get the following result.

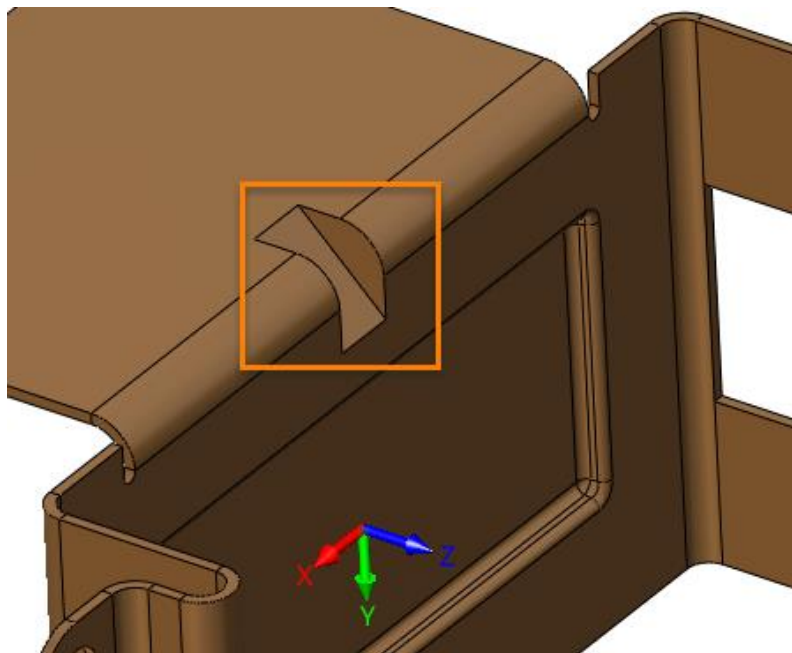


Figure84 Punch Result

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

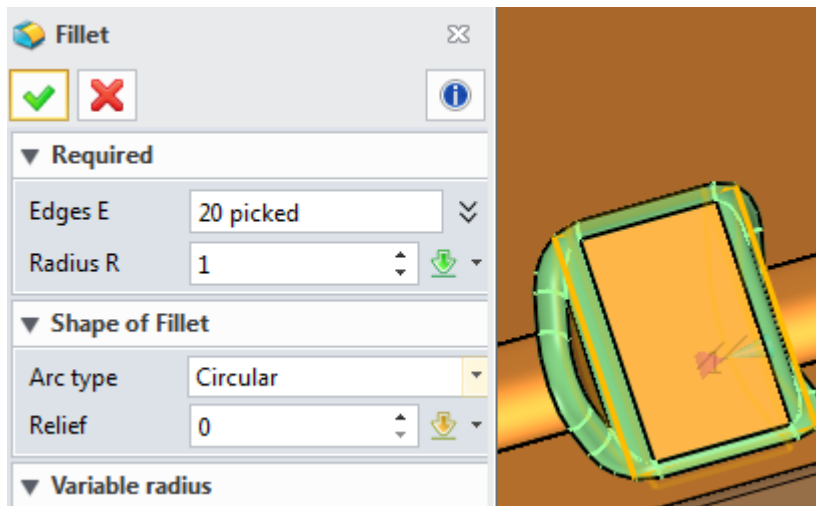


Figure85 Fillet for the Stiffener

7. Bend Function

7.1 Fold and Unfold



Sheet Metal ribbon toolbar->Bend->

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:

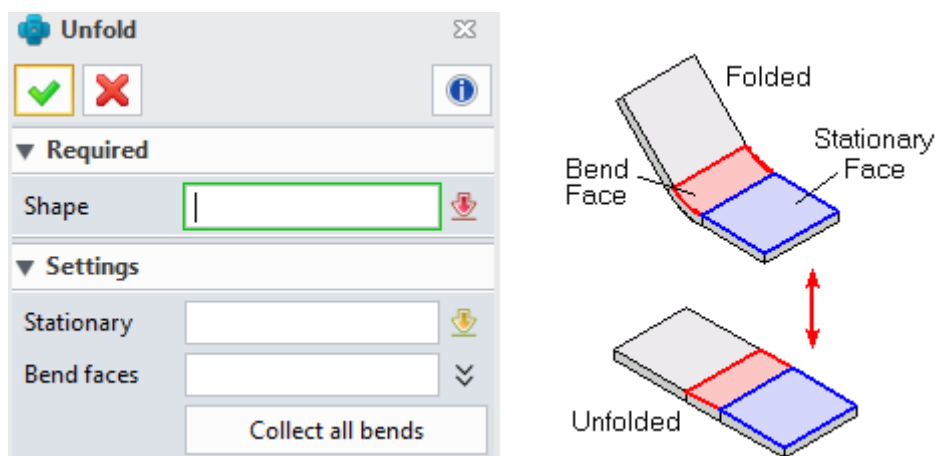


Figure86 Unfold

Shape: Choose sheet metal shape to unfold or fold.

Stationary face: choose a face as the stationary face, which will determine the unfolded

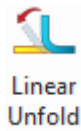
shape's location plane. This option is not mandatory, if this option is blank, it will use the default base flange's top face as the stationary face.

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.

7.2 Linear Unfold



Sheet Metal ribbon toolbar->Bend->

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.

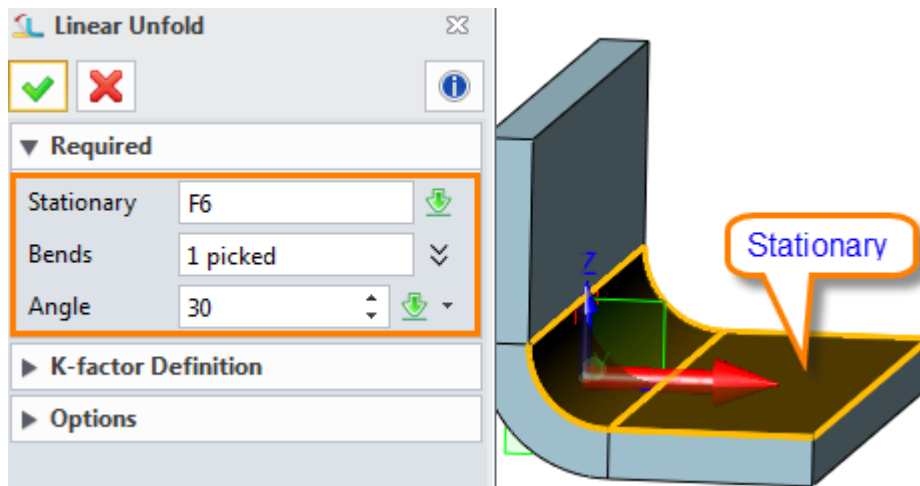


Figure87 Linear unfold

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

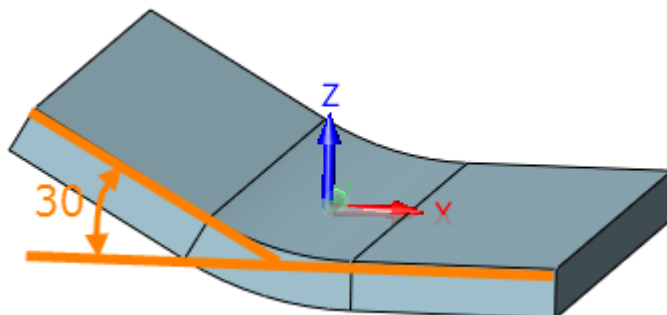


Figure88 Linear unfolded shape

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:

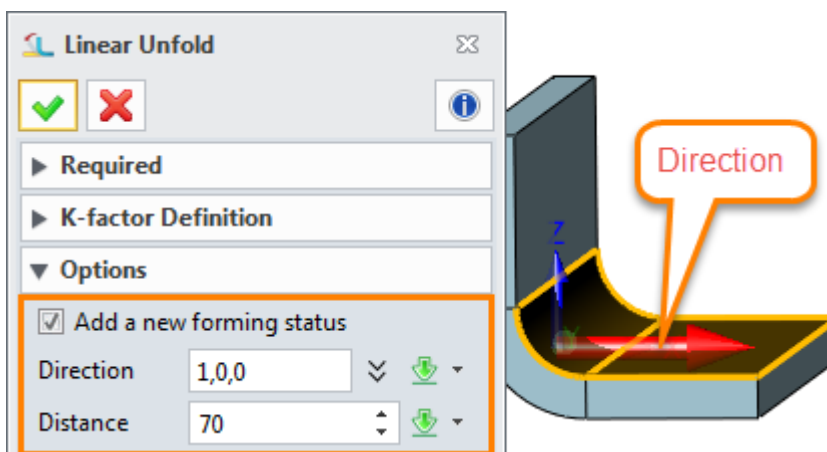


Figure89 Add a new forming status

After confirming it we can get the following result:

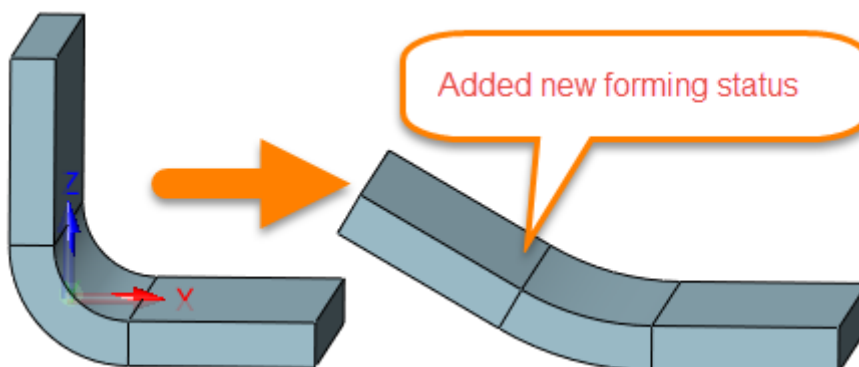


Figure90 Added new forming status

Note: The added new forming status is associated with the original part.

7.3 Change Bend



Sheet Metal ribbon toolbar->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.

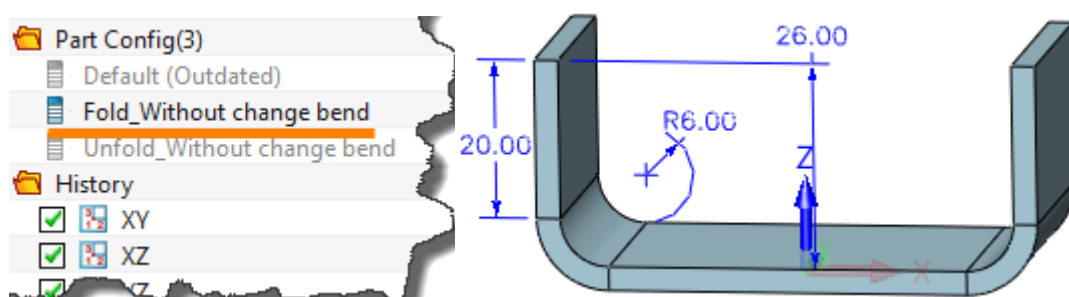


Figure91 Fold status without change bend

Note: These 3D dimensions are created by PMI function.

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

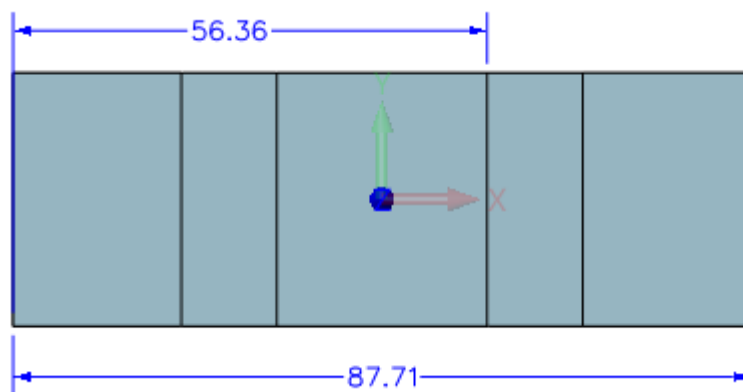


Figure92 Unfold status without change bend

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

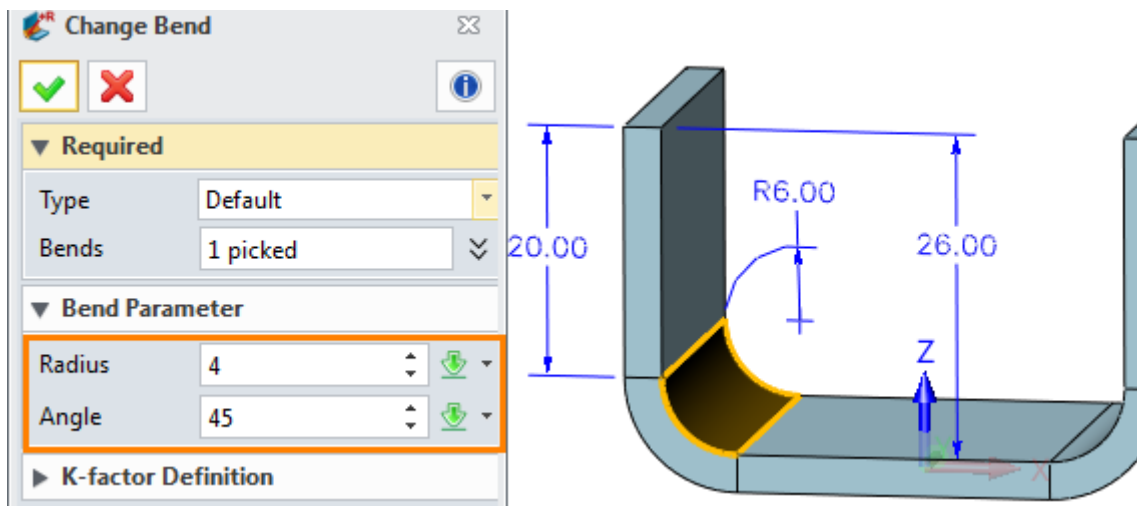


Figure93 Default way to change bend

Confirm it and the result will be as follows.

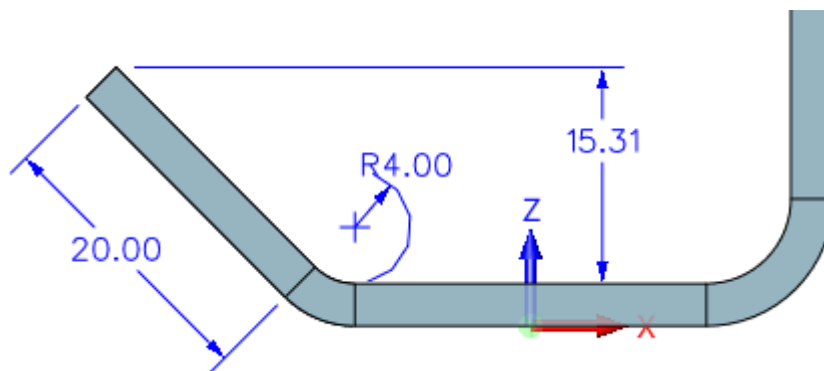


Figure94 Default way changed bend result

STEP 03 Unfold it to get the following result.

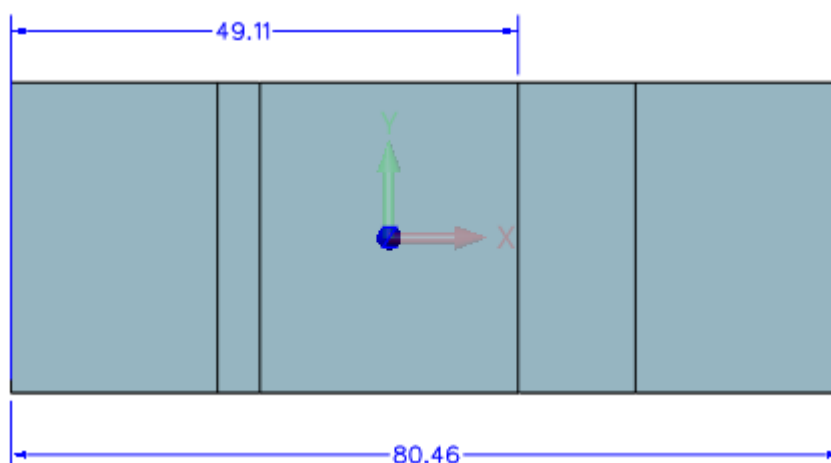


Figure95 Unfold status after bend changed

If we go to compare the unfold result of unchanged bend and changed bend (Fig91 and Fig94), 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.

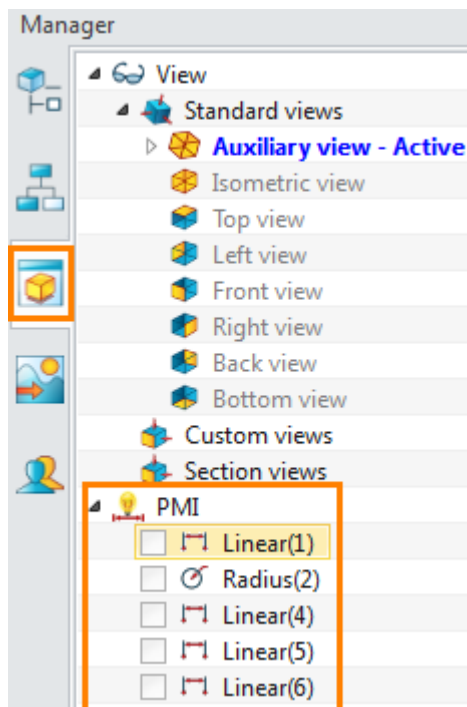


Figure96 Turn off the PMI information

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

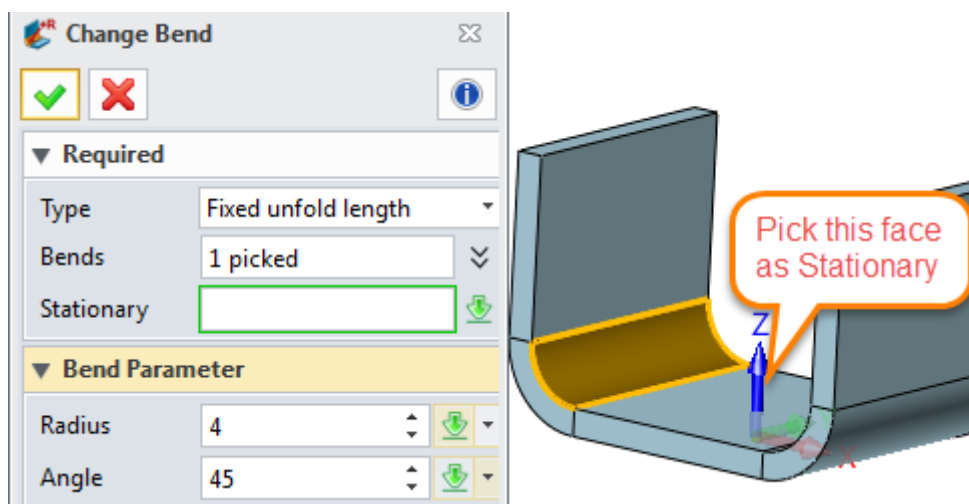


Figure97 Fixed unfold Length

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

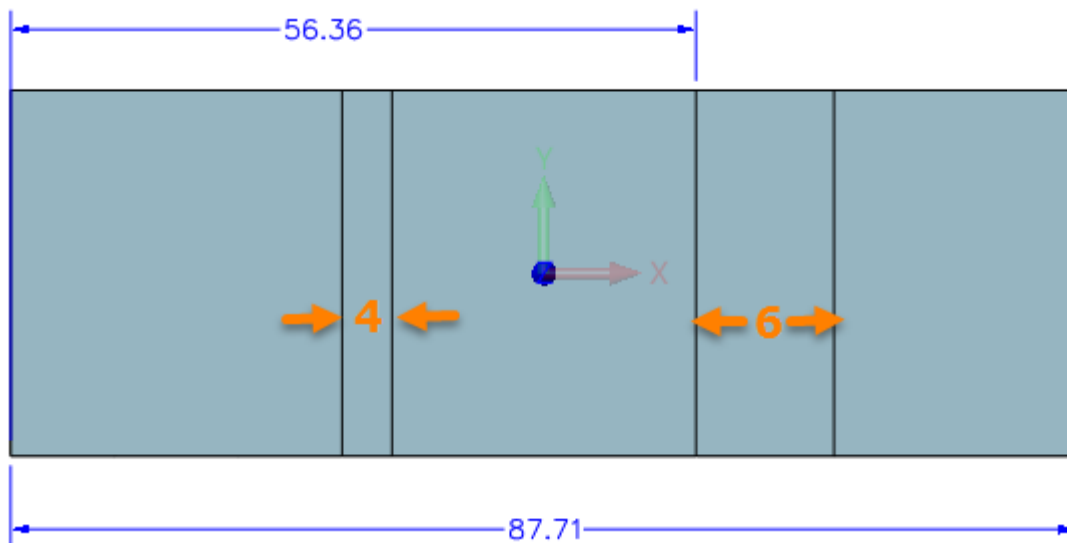
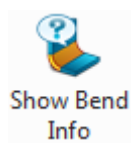


Figure98 Unfold the changed bend again by fixed unfold length

7.4 Show Bend Information



Sheet Metal ribbon toolbar->Bend->

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.

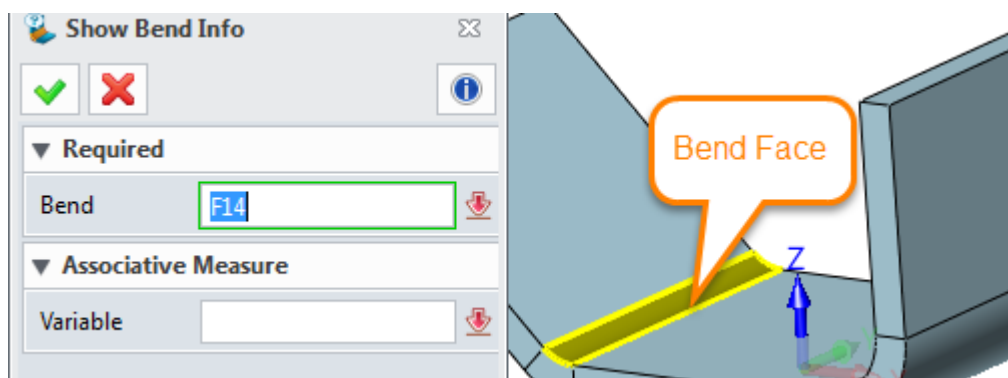


Figure99 Show Bend Information

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

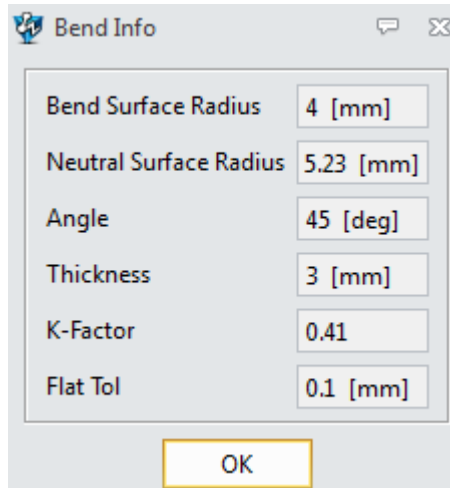


Figure100 Bend Information

7.5 Set Stationary Face



Sheet Metal ribbon toolbar->Bend->

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.

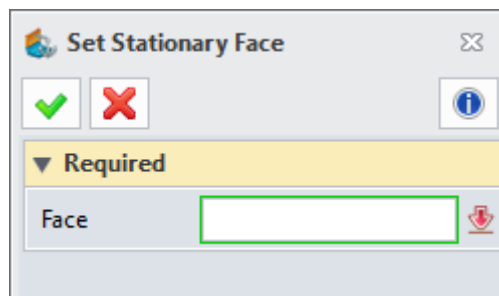


Figure101 Set Stationary Face

8. Convert

8.1 Rip



Sheet Metal ribbon toolbar->Convert->

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.

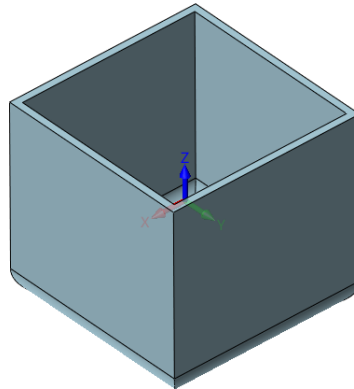


Figure102 Convert to Sheetmetal part

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

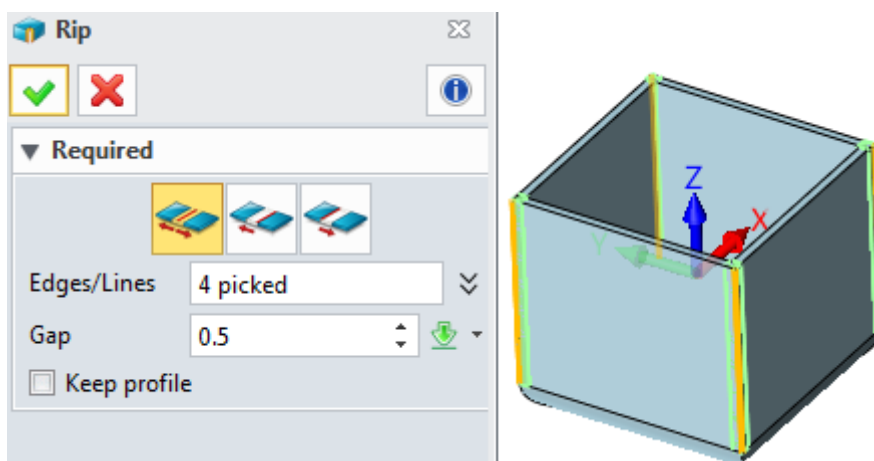


Figure103 Rip gap

STEP 03 Confirm it to get the following result.

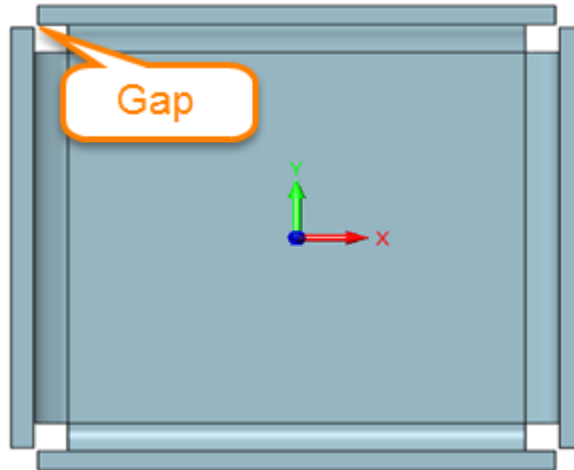


Figure104 Rip result

8.2 Mark Bend



Sheet Metal ribbon toolbar->Convert->

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 sheetmetal part in last step and then mark the bend face.

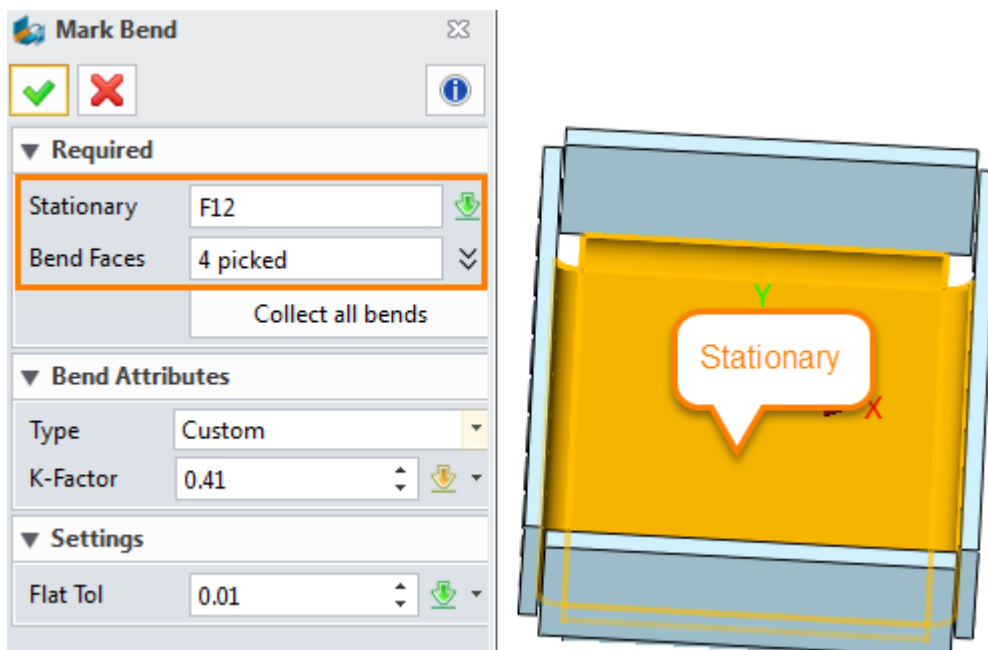


Figure105 Mark Bend

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.

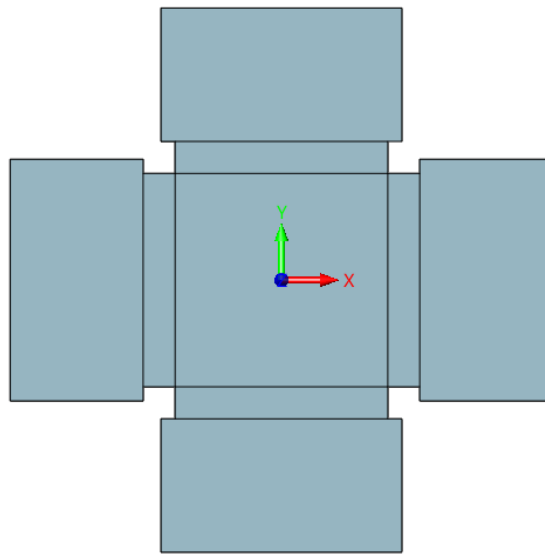
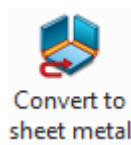


Figure106 Unfold the Marked bend

8.3 Convert to Sheet metal



Sheet Metal ribbon toolbar->Convert->

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

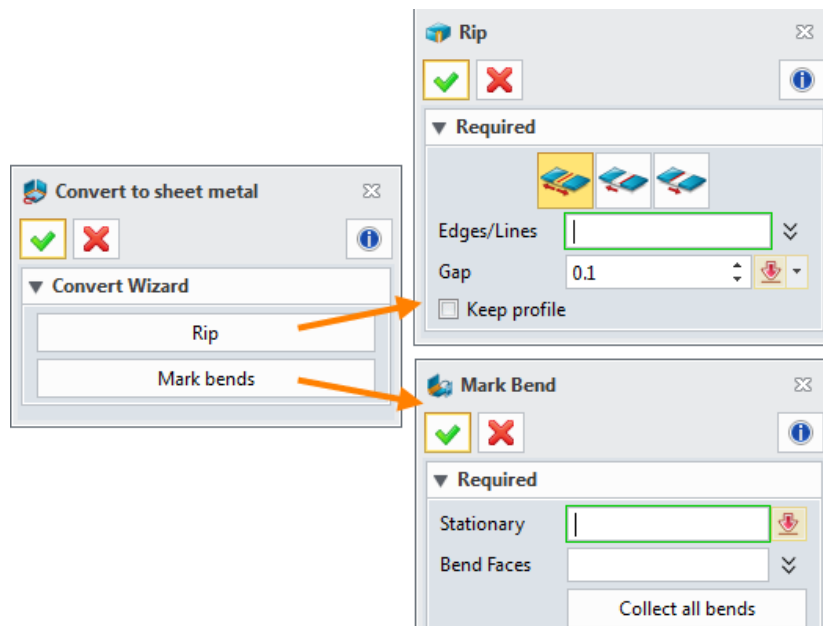


Figure107 Convert to sheet metal

9. Exercises

9.1 Exercise1

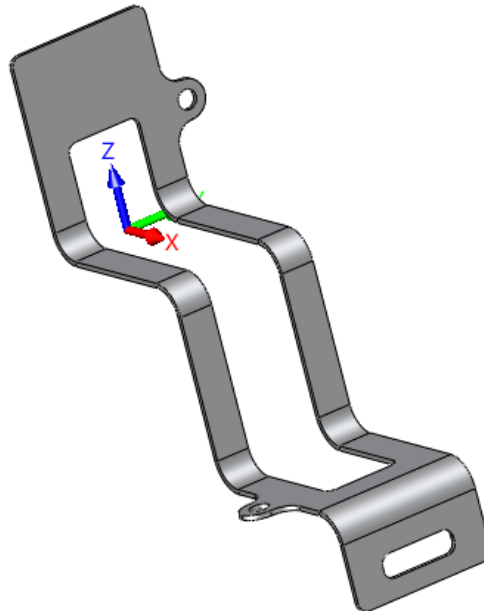


Figure108 Exercise1

STEP 01 Create a sketch as follows.

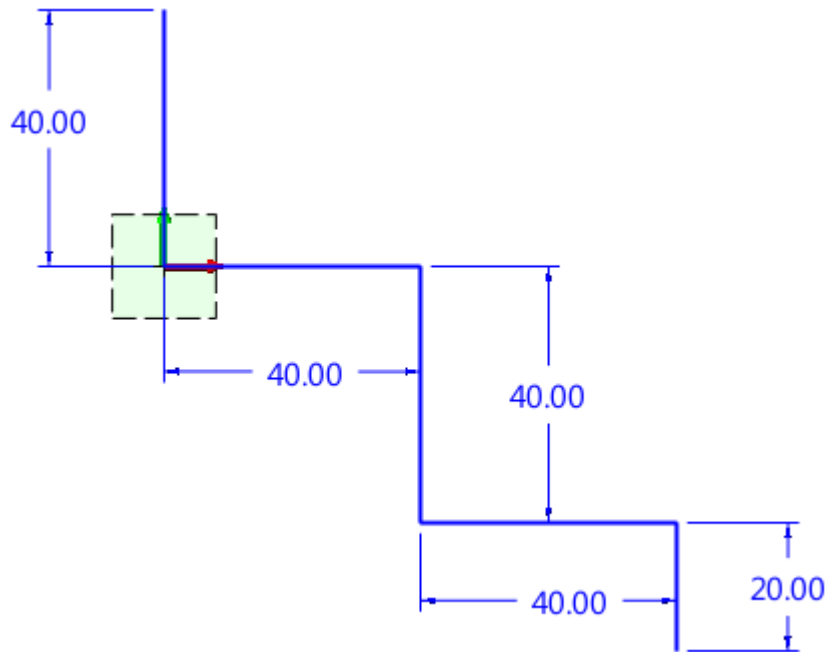


Figure109 Sketch for exercise1

STEP 02 Exit the sketch and then extrude as follows.

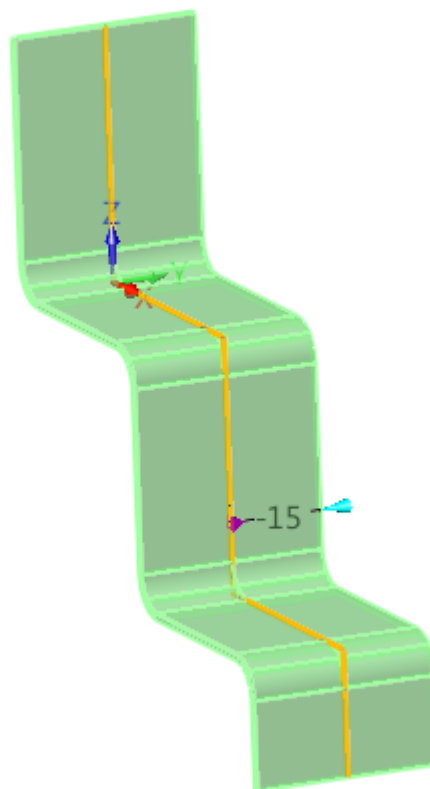
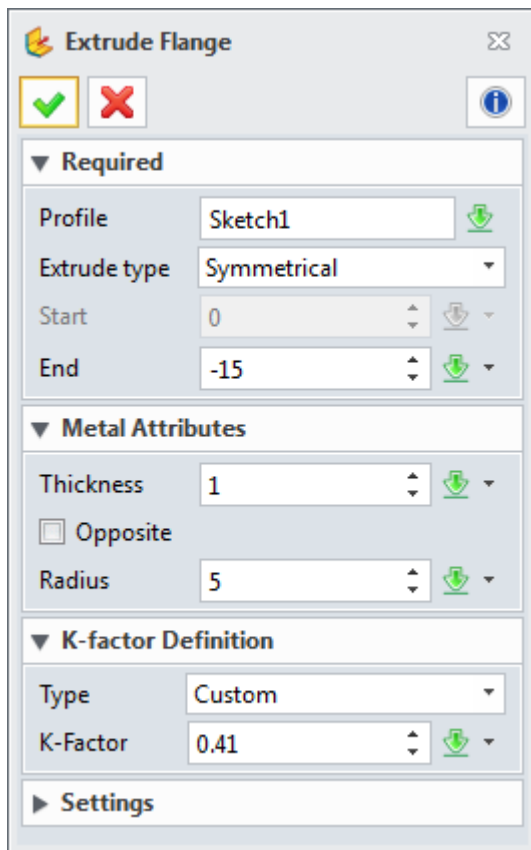


Figure110 Extrude flange_excercise1

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

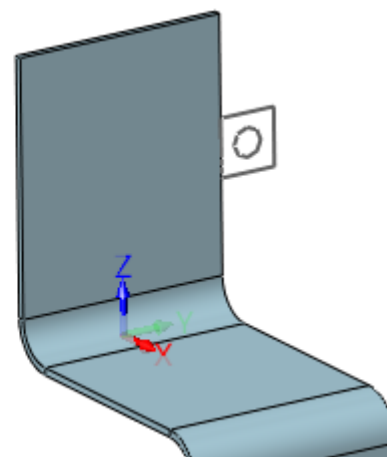
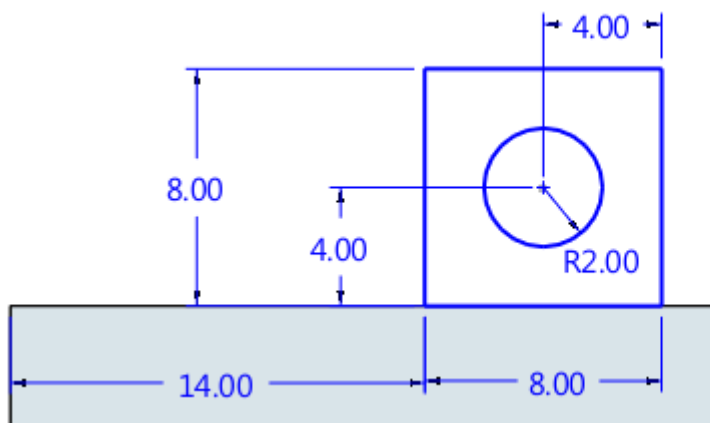


Figure111 Add sketch_excercise1

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

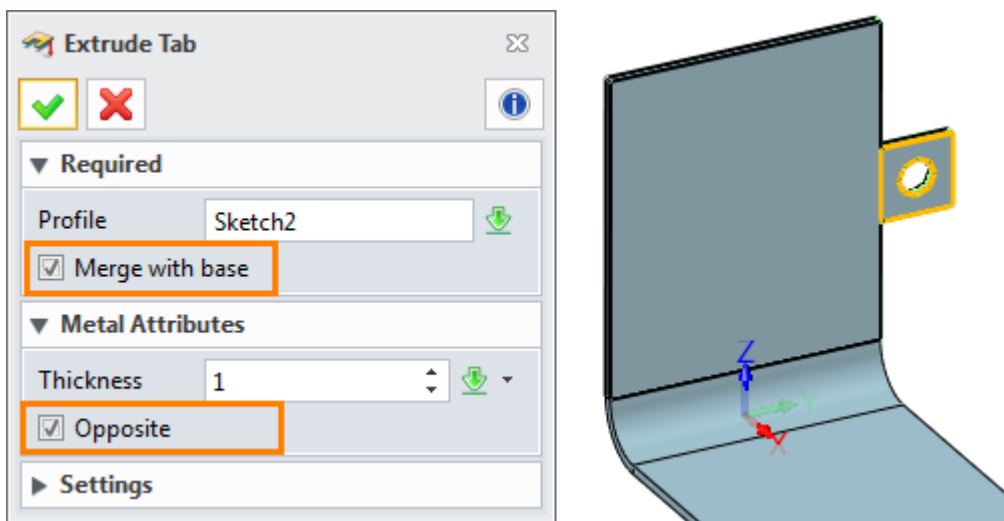


Figure112 Merge a tab_excecise1

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

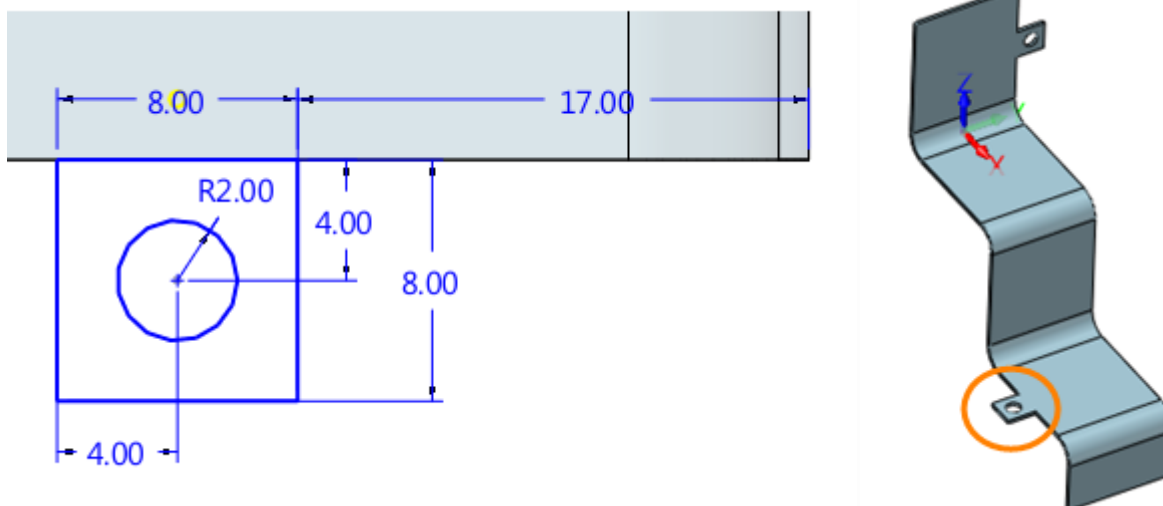


Figure113 Add one mere tab_excesice1

STEP 06 Select the edges to add the fillet.

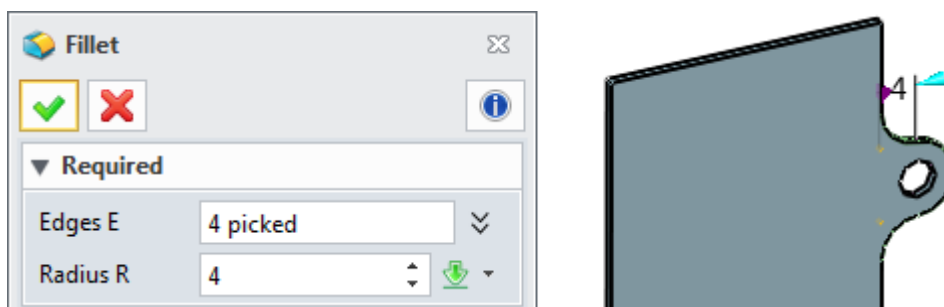


Figure114 Add Fillet_excesice1

STEP 07 Create a sketch for middle slot as follows.

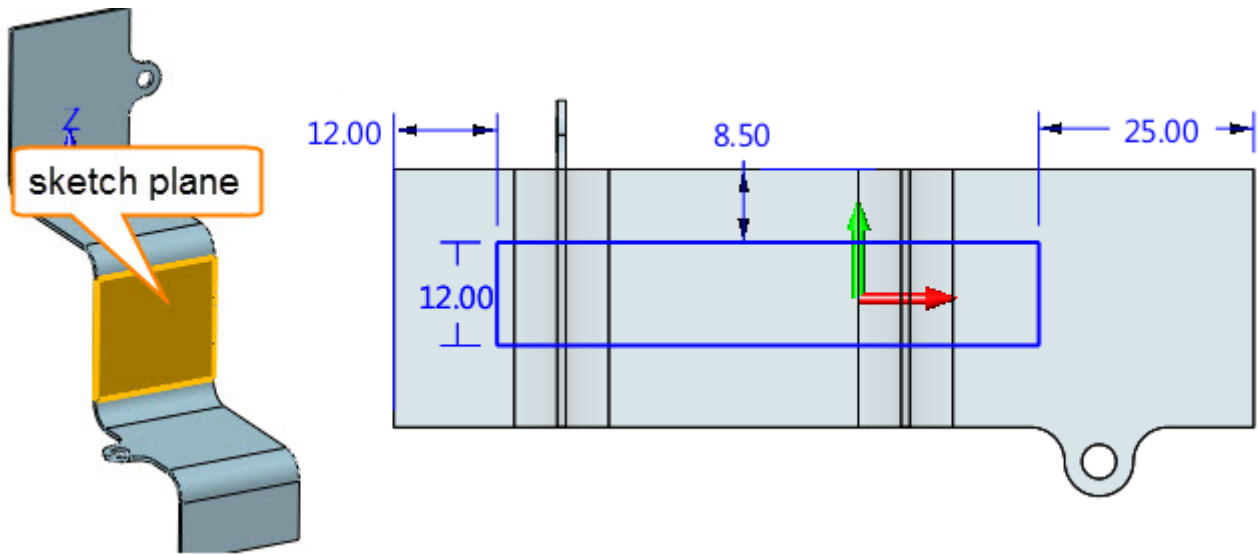


Figure115 Create cut out sketch_excesice1

STEP 08 Extrude and cut the middle slot as follows.

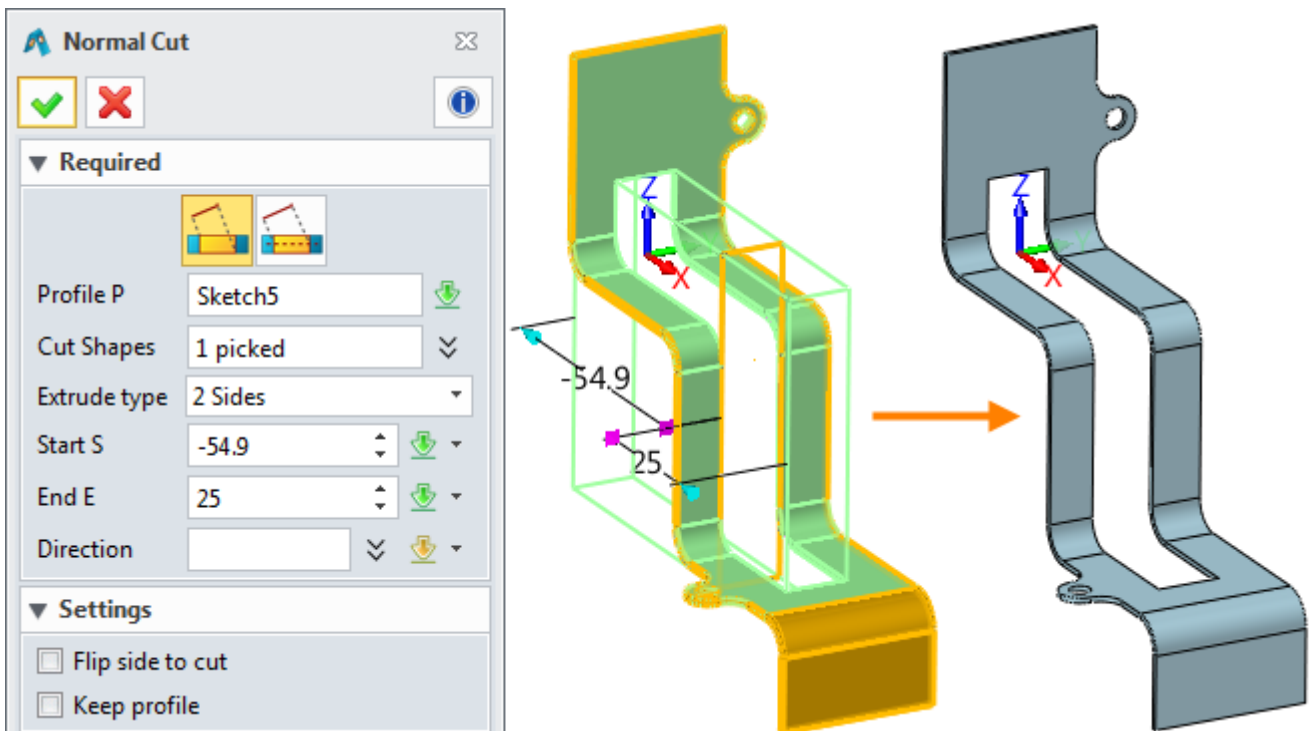


Figure116 Extrude cut the middle slot_excesice1

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

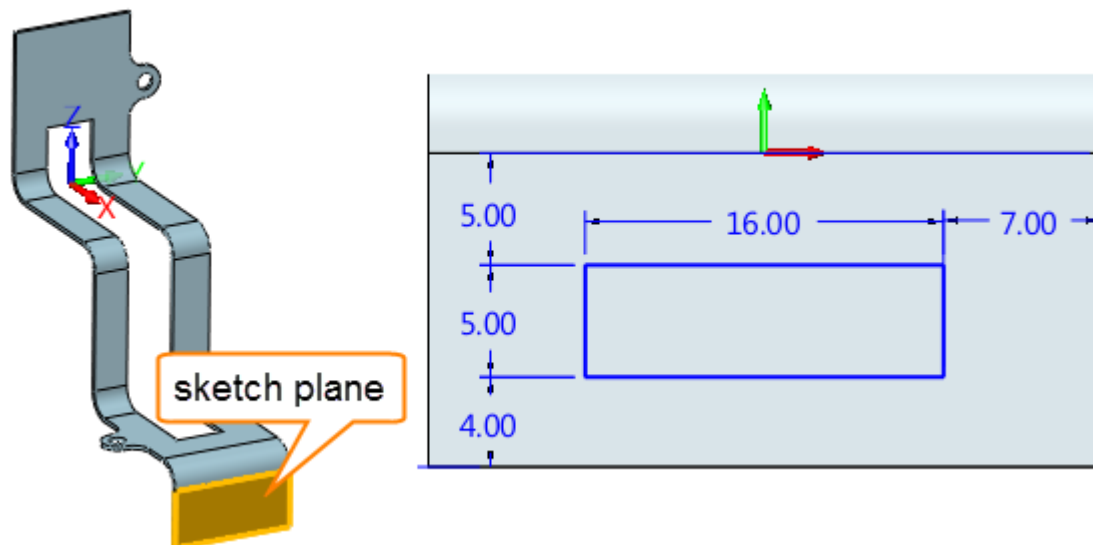


Figure117 Sketch for another slot_excesice1

STEP 10 Extrude and cut the slot as follows.

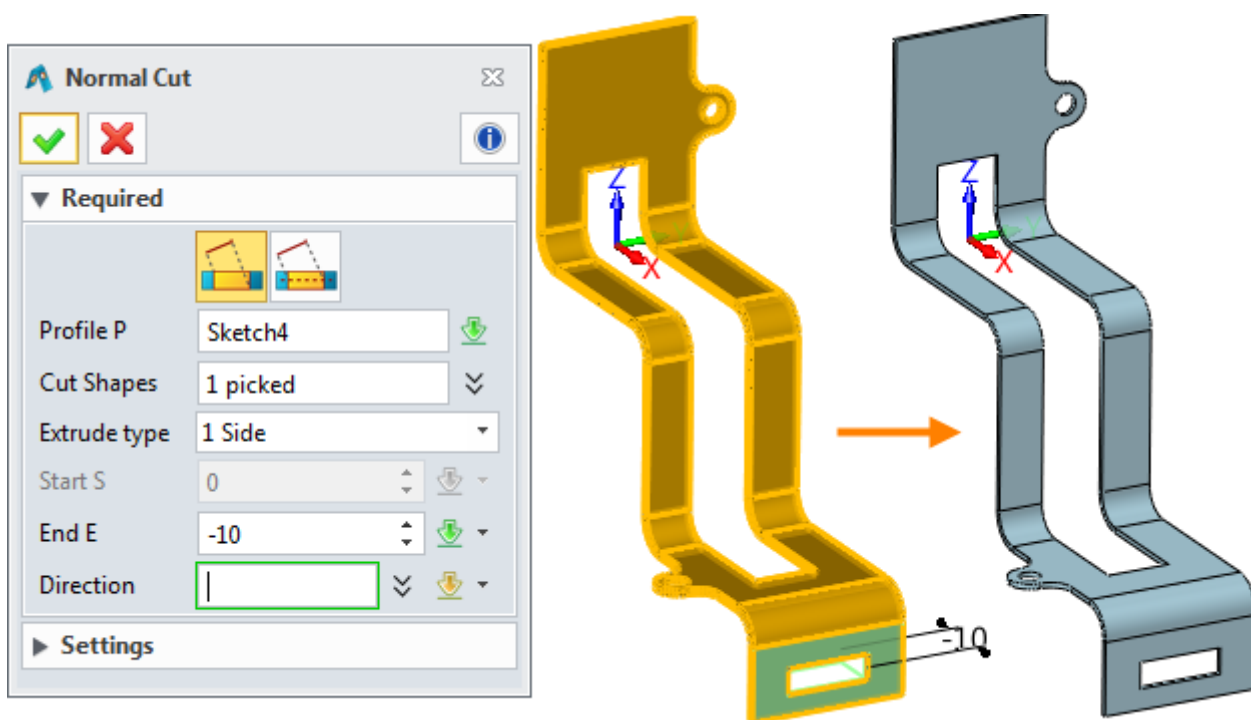


Figure118 Extrude another slot

STEP 11 Fillet the corners as follows.

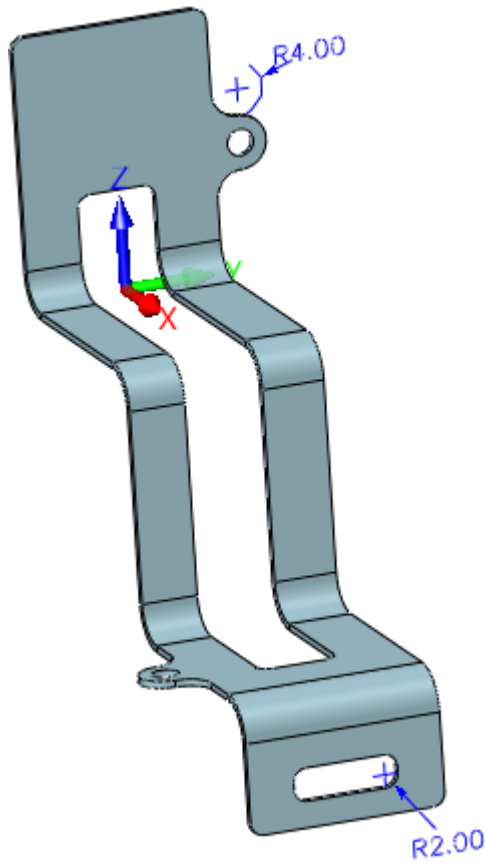


Figure119 Add fillet at corners

STEP 12 Unfold it as follows:

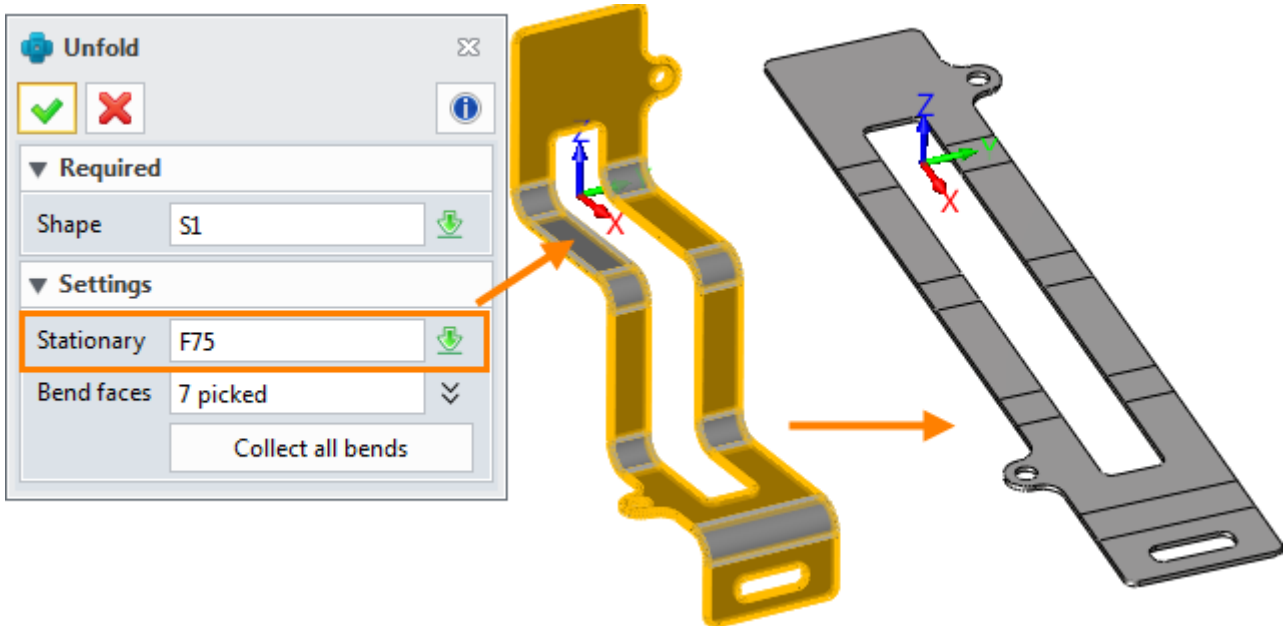


Figure120 Unfold

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

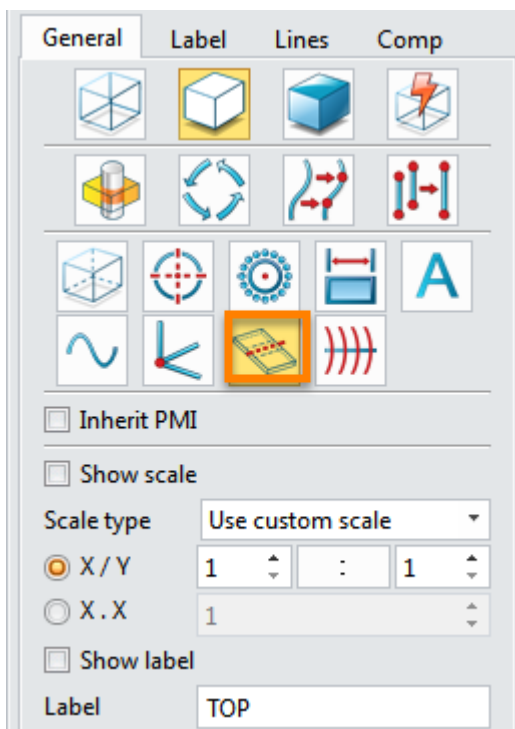


Figure121 Show Bend information in 2D sheet

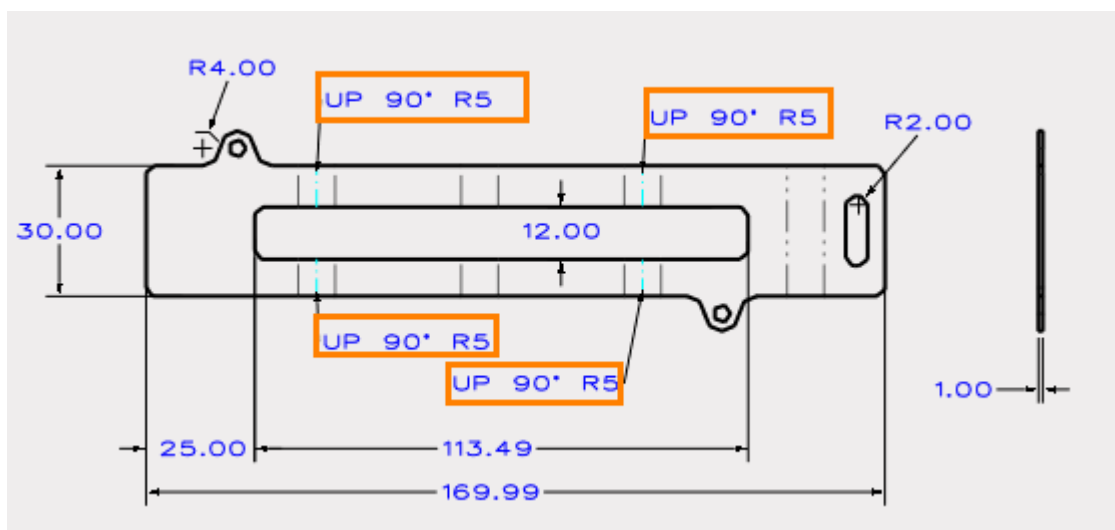


Figure122 2D sheet with bend information

STEP 14 Save it.

9.2 Exercise2

STEP 01 Create a sketch as follows.

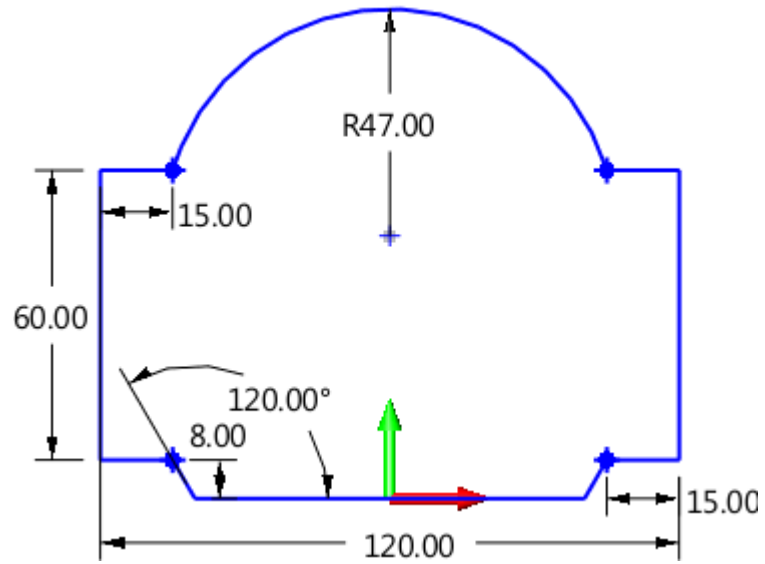


Figure123 Sketch_exercise2

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

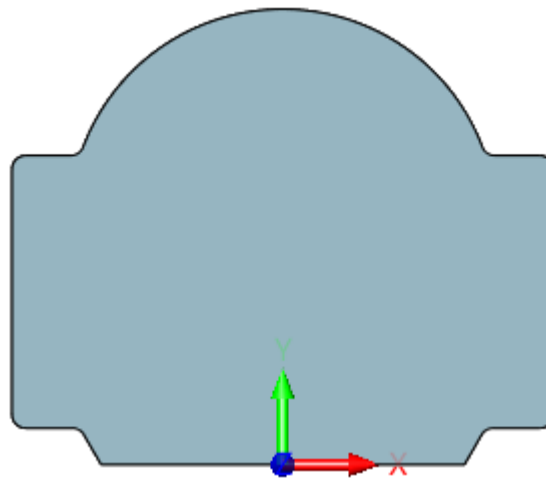


Figure124 Tab_exercise2

STEP 03 Create sketch for dimple as follows.

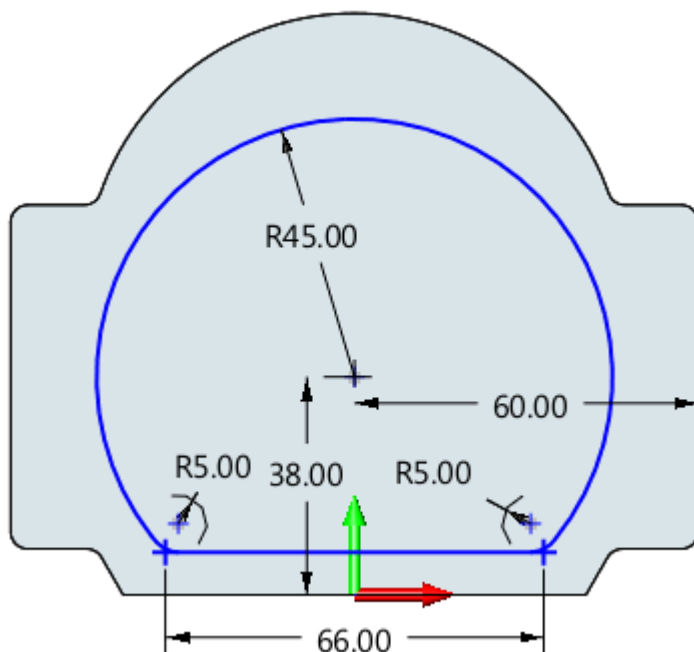


Figure125 Sketch for dimple_exercise2

STEP 04 Create dimple as follows.

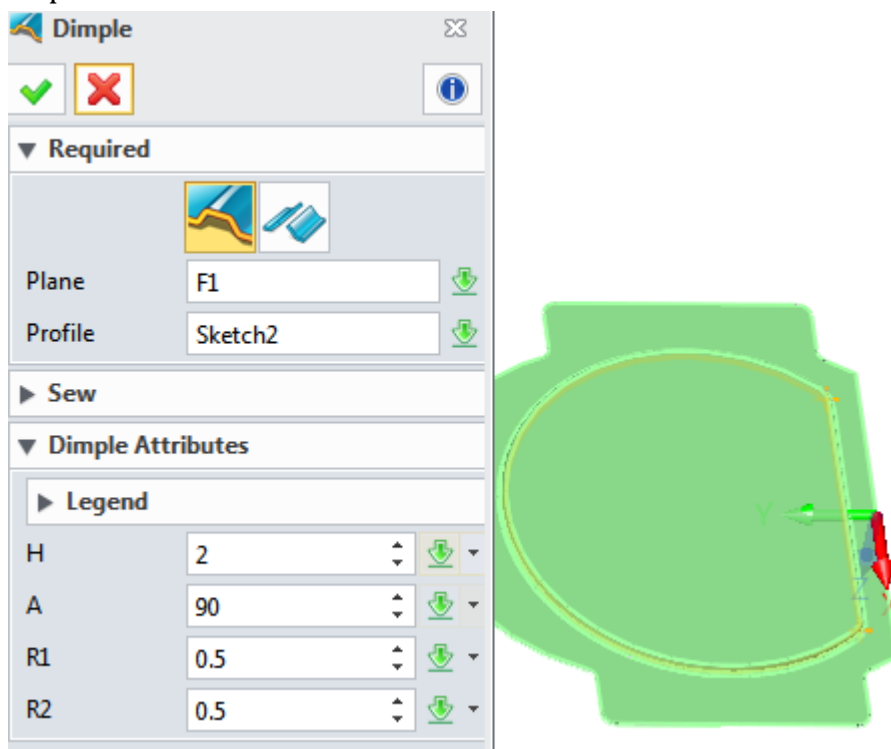


Figure126 Dimple_exercise2

STEP 05 Create sketch for flare dimple as follows.

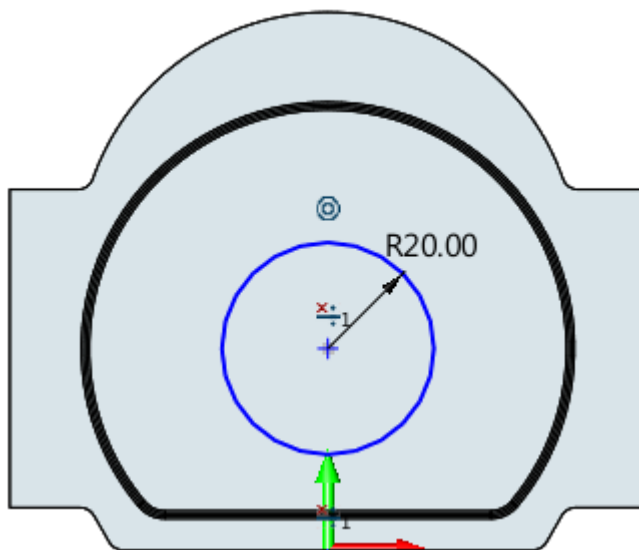


Figure127 Sketch for flare dimple_exercise2

STEP 06 Create flare dimple as follows.

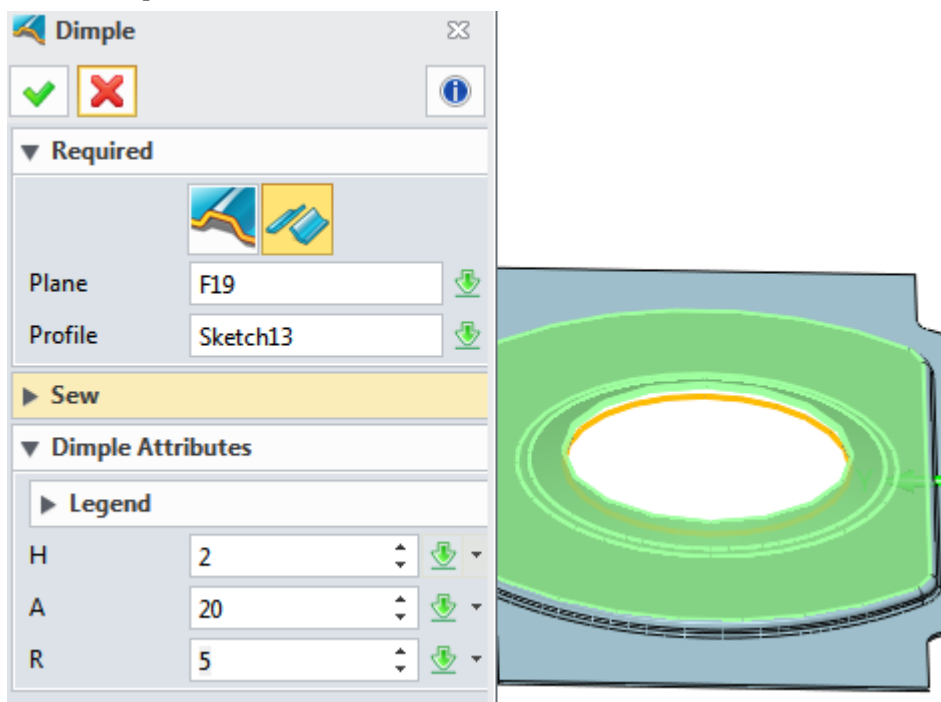


Figure128 Flare Dimple_exercise2

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

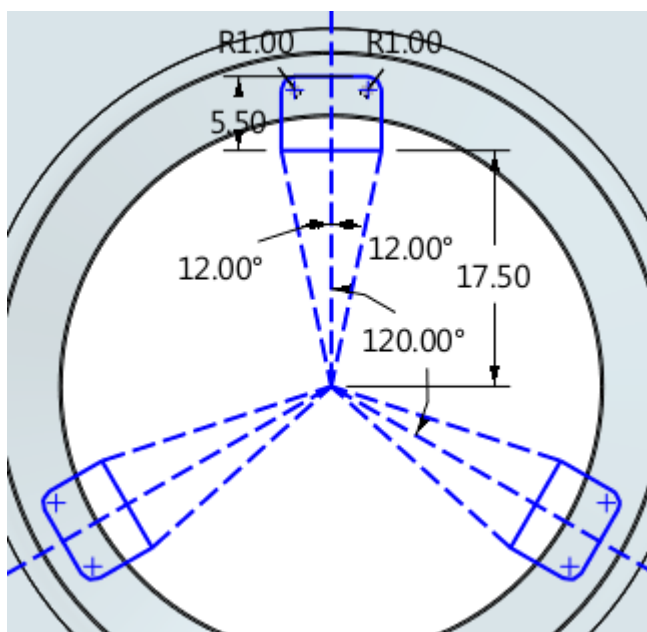


Figure129 Sketch for flare dimpe_exercise2

STEP 08 Normal cut the flare dimple as follows.

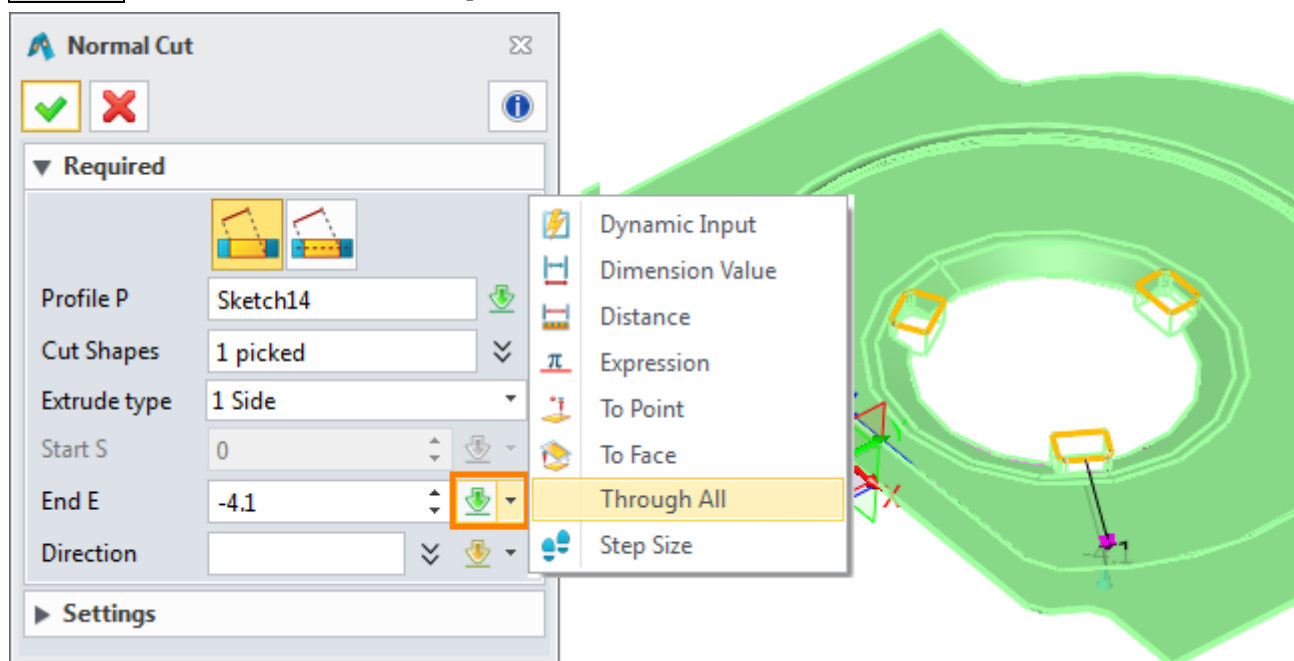


Figure130 Cut the open slots on flare dimpe_exercise2

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.

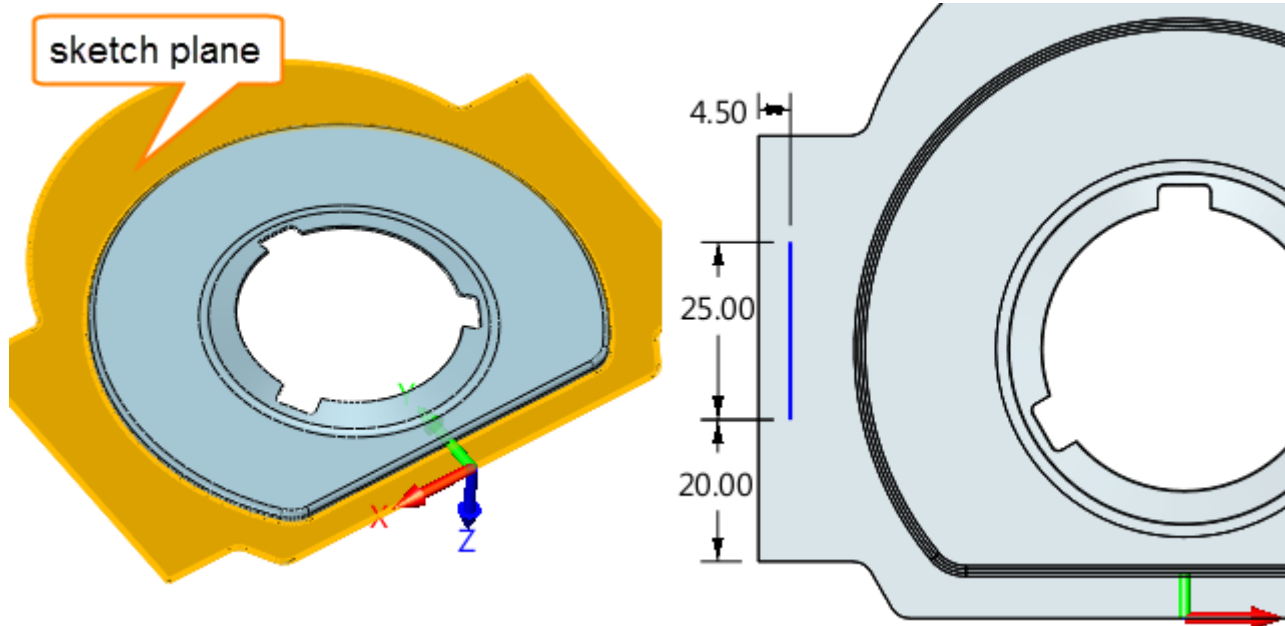


Figure131 Plane for sketch of Louver_exercsie2

STEP 09 Create the louver as follows.

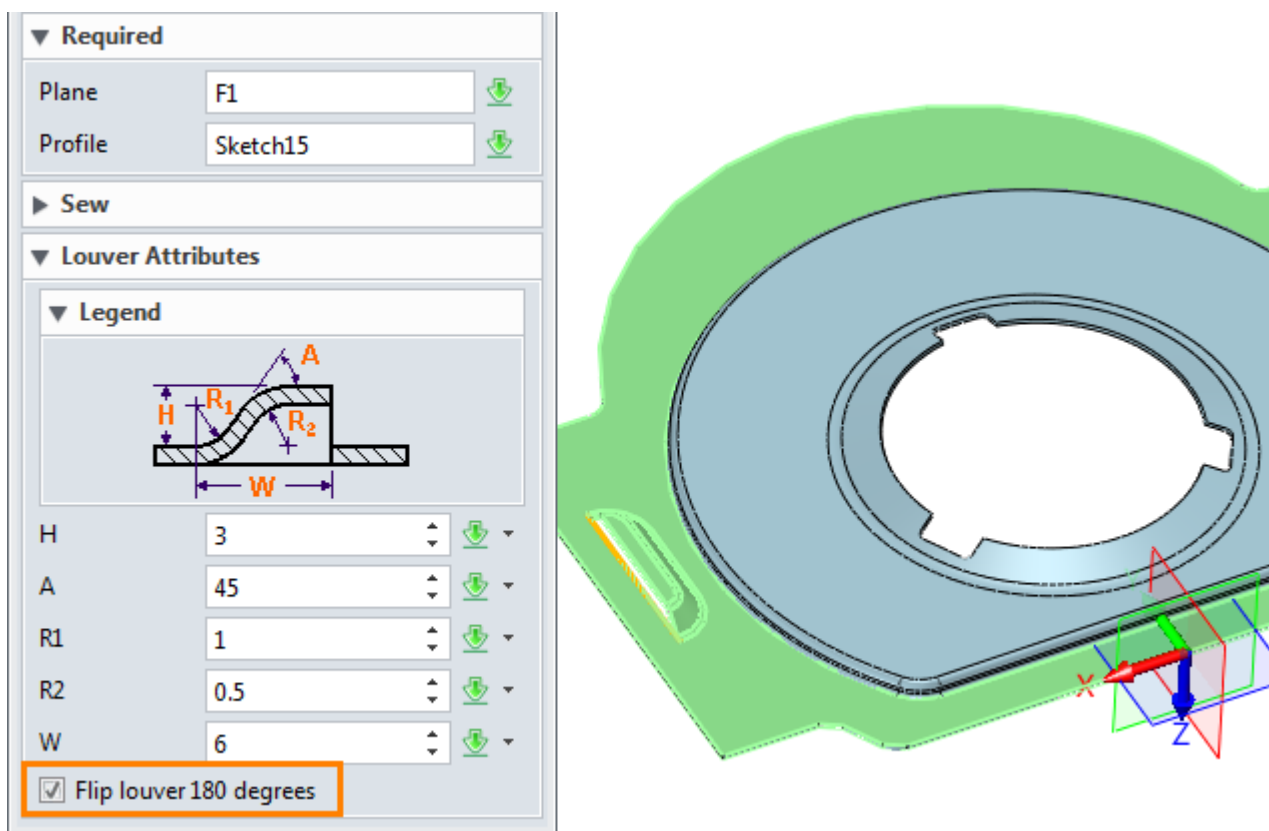


Figure132 Louver_exercsie2

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

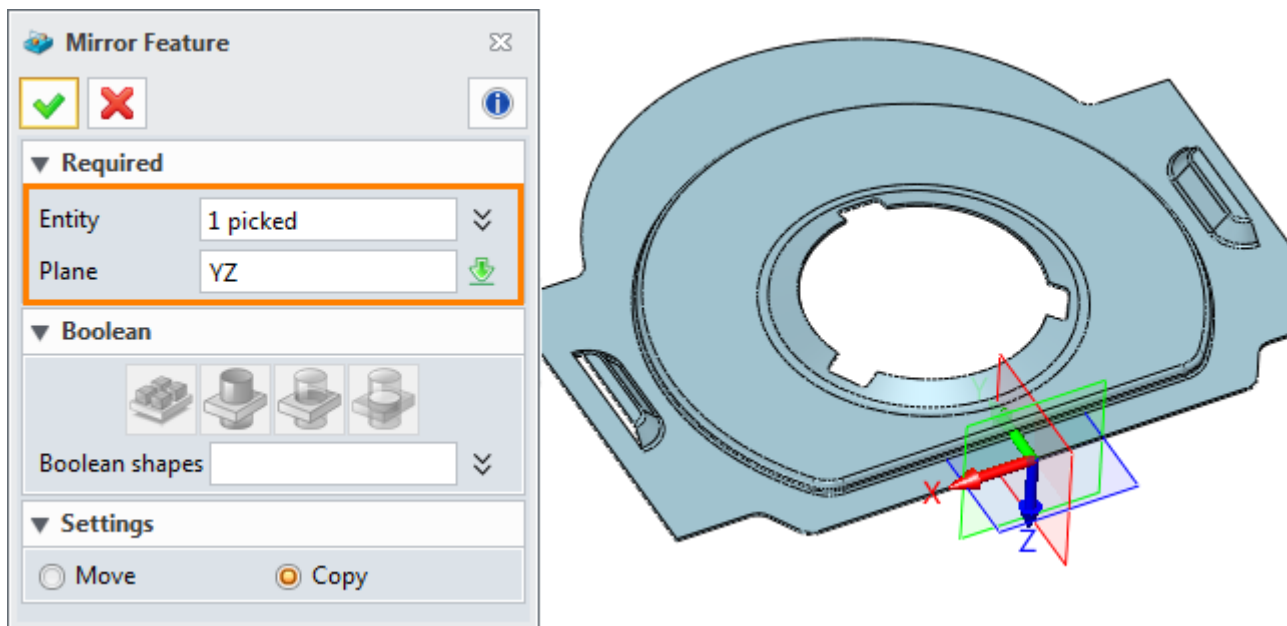


Figure133 Mirror louver feature_exercsie2

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

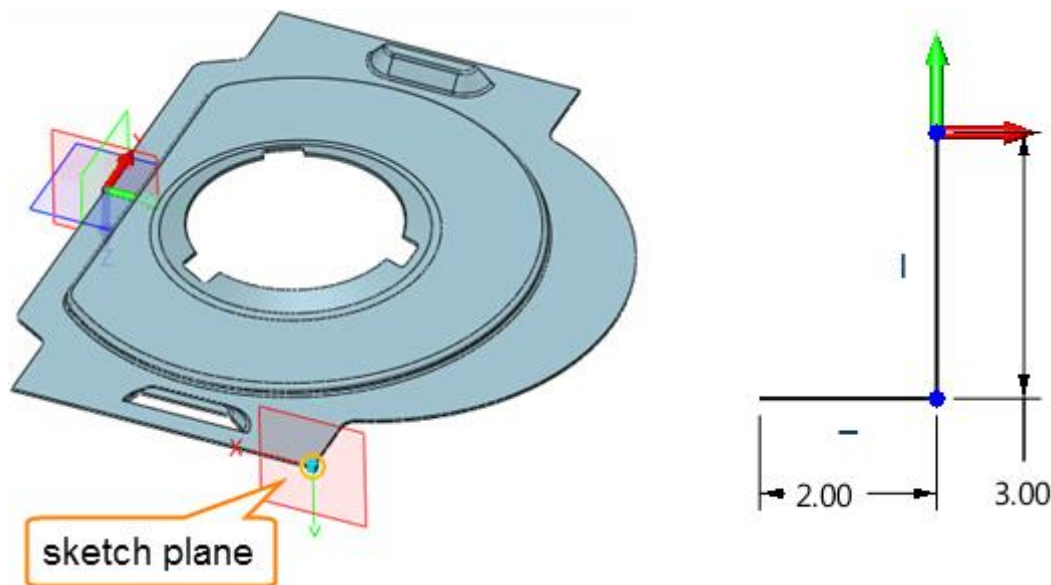


Figure134 First sketch for swept flange_exercise2

STEP 12 Create Swept Flange, details are as follows.

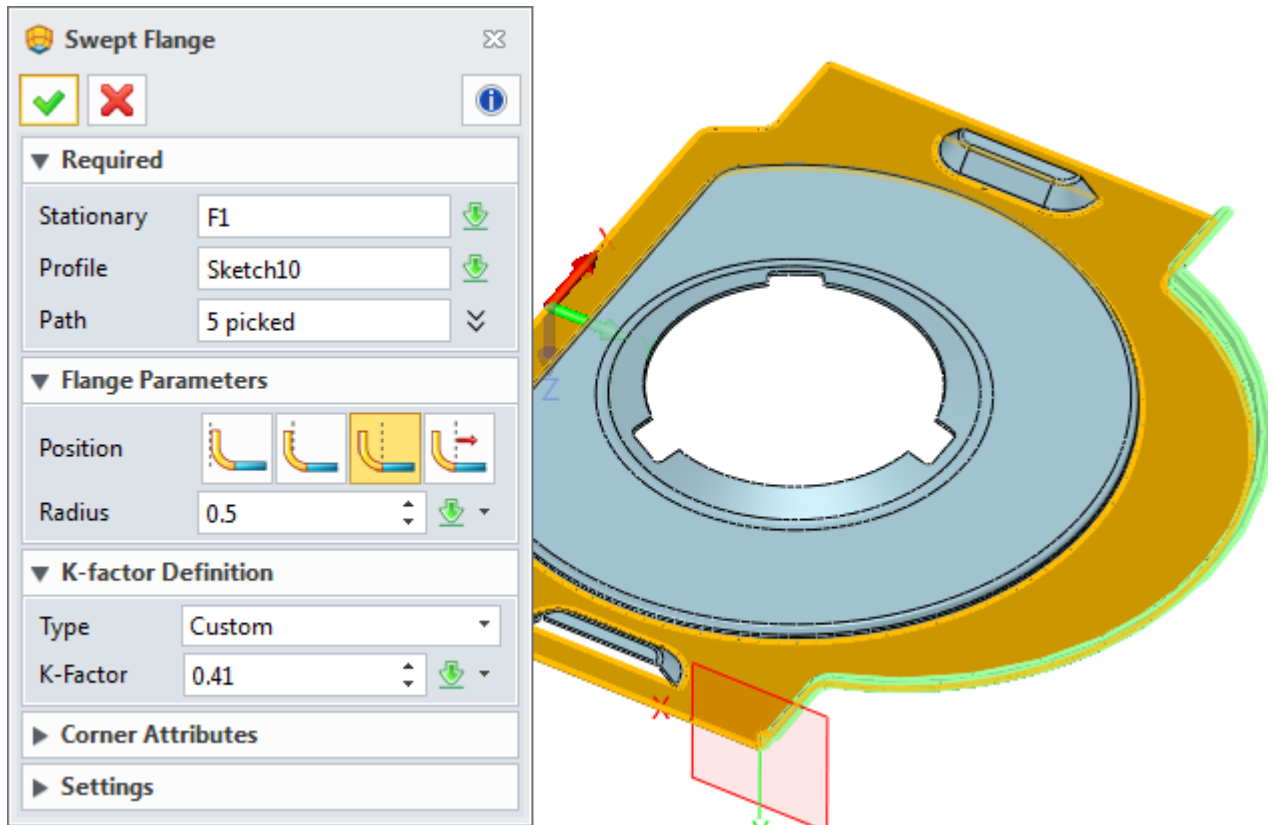


Figure135 Swept flange_exercise2

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

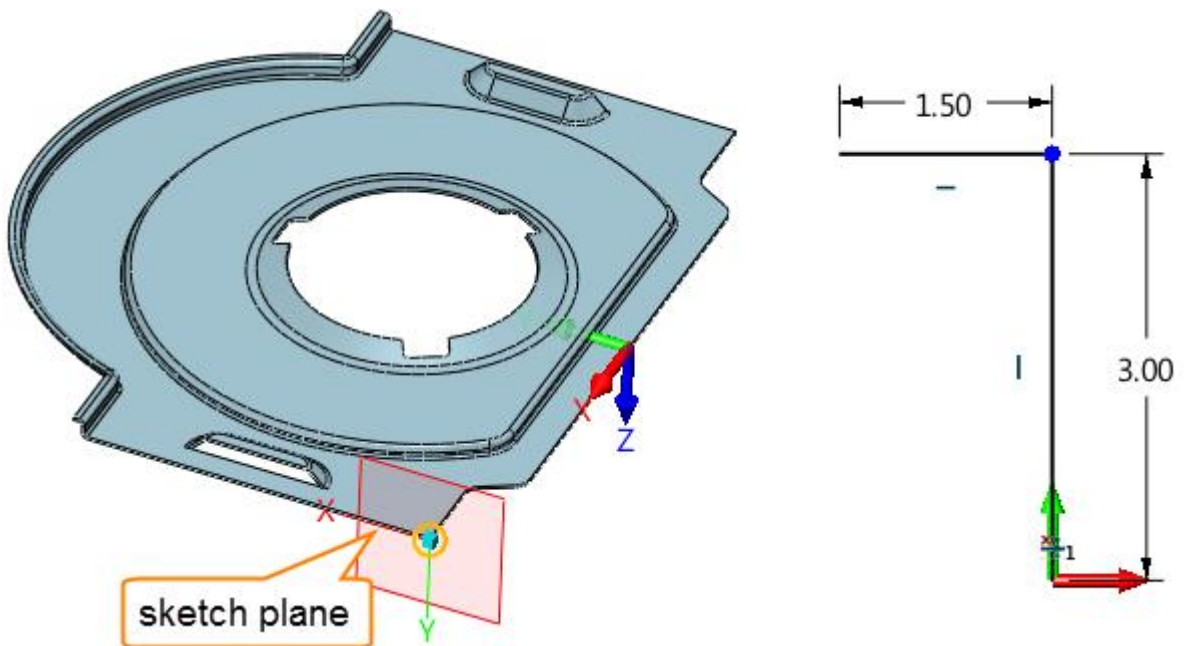


Figure136 Second new plane for swept flange profile_exercises2

STEP 14 Create the second Swept flange as follows.

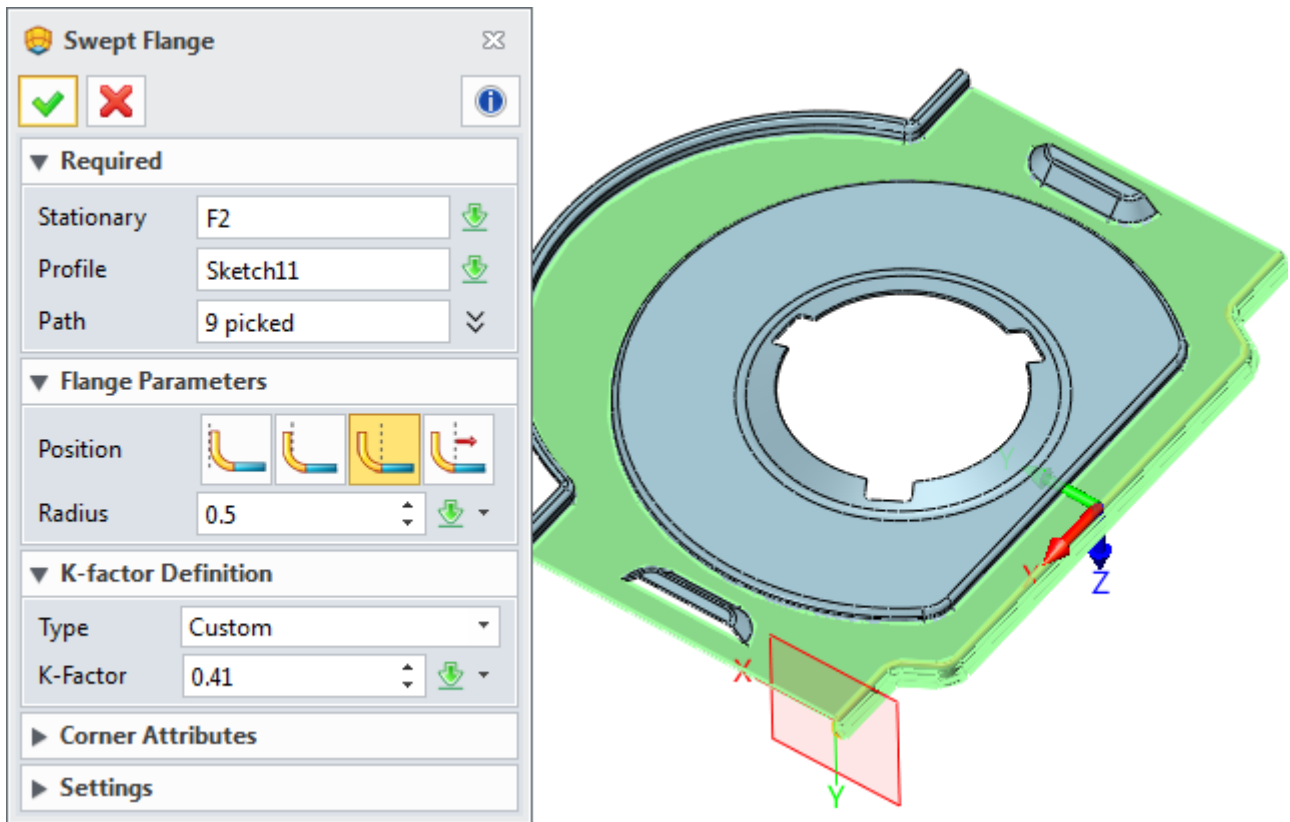


Figure137 Second swept flange_exercsie2

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

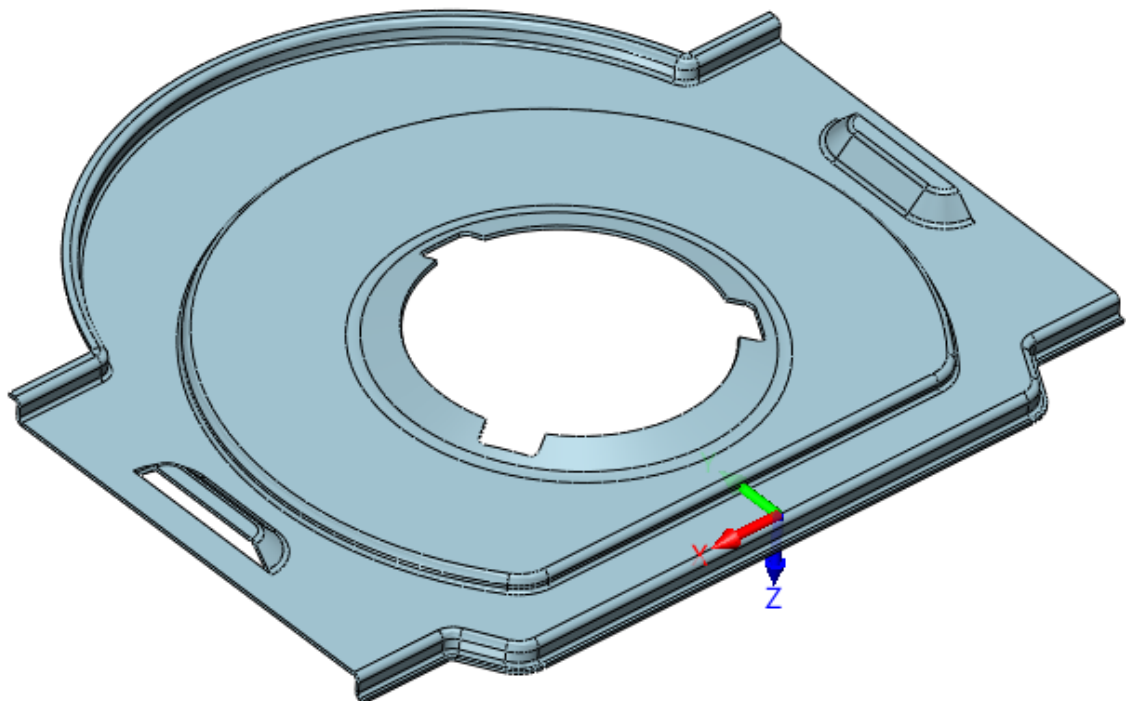


Figure138 Final result for exercise2