ZW3D From Entry to Master

Sheet Metal
ZW3D Sheet Metal

Key Points:
- Familiar with ZW3D sheet metal functions
- Create sheet metal base, flange, corner, forming feature, etc.
- Convert 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](image)

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

![Sheet Metal Attribute](image)

**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.
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

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:

![Extrude Tab dialog box](image)

**Figure 5** 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.

![Sketch for merged tab](image)

**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”.

2.2 Extrude Flange

Sheet Metal ribbon toolbar->Base->

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.

**STEP 02** Extrude flange as shown in Figure 9.

**STEP 03** Save it as "02_Extrude Flange Open".
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](image)

**STEP 02** Extrude the closed sketch by "Extrude Flange" command as follows.
**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-> 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”.
3. **Flange Creation**

3.1 **Full Flange**

This command can create full flange and S type flange.

**STEP 01** Create a base flange as follows.

![Full Flange](image)
STEP 02 Create a full flange at an edge as follows.

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”.
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
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.
STEP 03 Try other bend types of Hem.

**Open Type**

**S Bend**

**Curl**

**Closed Loop**

**Open Loop**

**Centered Loop**

STEP 04 Create miter hem flange as follows.
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:

**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”.

![Figure 24 Sketch for Loft flange](image-url)
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.*

![Lofted Flange](image)

**STEP 03** Unfold it.

![Unfold loft flange](image)
3.5 Swept Flange

Sheet Metal ribbon toolbar->Flange->

Use this command to create swept flange.

**STEP 01** Create a base flange as follows.

![Figure 28 Base for Swept flange](image)

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

![Figure 29 Profile for Sweeping](image)
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.

![Figure 30 Swept Flange](image)

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

![Figure 31 Filter on Pick Tools Mini Bar](image)

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

![Figure 32 Swept Flange with closed path](image)
STEP 05 Unfold the swept flange.

Figure 33 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 34 Sketch for Swept Flange Base
**STEP 02** Create profile on assigned datum, as shown in Figure 36.

**STEP 03** Create swept flange as follows.
Figure 37 Swept Flange along uncontinuous Path

**STEP 04** Unfold the swept flange.

Figure 38 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.

![Figure 39 Open File “Fold by line”](image)

*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.

![Figure 40 Parameter for Fold by Line](image)
STEP 03 Fold result is shown in Figure 41.

Figure 41 Fold by line Result

STEP 04 Do the same process for other parts.

Figure 42 Finished Fold by line

3.7 Jog

Sheet Metal ribbon toolbar -> 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 43 Jog part
**STEP 02** Click Jog command, then select the curve to create a jog as follows.

![Figure 44: Create Jog](image)

**STEP 03** Confirm it and then get the following result.

![Figure 45: Jog shape](image)
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.

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

a. **Extend by distance**

![](image) Figure 46 Extend flange file

![](image) Figure 47 Extend Flange by Distance
b. Extend flange through until intersection with plane

![Figure 48 Extend Flange through until Intersection with Plane](image)

Figure 48 Extend Flange through until Intersection with Plane

c. Extend flange up to selected plane

![Figure 49 Extend Flange up to Selected Plane](image)

Figure 49 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
Check the option “Extend surface adjacent to the edge” to get the following result.
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.

Figure 54  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 55  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.
Here don’t check the option “Flip side to cut”, and then click ok to get following result.

**STEP 03** Unfold it to check the cut result as follows:
1) **Normal to both sides**: 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 61 Normal to Both Sides](image)

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 62 Normal to Middle](image)
5. Corner

5.1 Close Corner

Sheet Metal ribbon toolbar->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.

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

Figure 63 Close Corner Part

Figure 64 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.

![Figure 65 Underlap Flange Corner](image)

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

![Figure 66 Overlap Flange Corner](image)

**Natural:** Both edges will meet naturally.

![Figure 67 Natural Closed Flange Corner](image)
**STEP 03** Try the send type—Bend. Select two bend faces to close corner as follows.

![Figure 69 Close the Corner by Bend Face](image)

The 1\(^{\text{st}}\) and 2\(^{\text{nd}}\) 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](image)

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.

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.

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.

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

![Figure 75 Flare Dimple](image)

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.
Figure 76: Create Louver

Figure 77: Flip Louver 180 Degrees
**STEP 03** Finish the louver in another side as follows.

![Image of louver in two sides]

Figure 78 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.

![Image of punch part]

Figure 79 Punch Part

**STEP 02** Create a sketch which will be used to create a puncher to punch on the datum plane.
**STEP 03** Extrude the sketch as the puncher.

---

**Figure 80** Sketch for Puncher

**Figure 81** Extrude the Sketch as Puncher

**Figure 82** Puncher and Base
**STEP 04** Punch a stiffener as shown in Figure 82.

![Punch a stiffener as shown in Figure 82.](image)

**STEP 05** Confirm it to get the following result.

![Confirm it to get the following result.](image)

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

![Add fillet on the boundary of the stiffener as follows. Then save the file.](image)
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:

**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.

![Linear Unfold](image)

Figure87 Linear unfold

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

![Linear Unfold dialog box](image)

After confirming it we can get the following result:

![Add a new forming status](image)

**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.

![Figure 91 Fold status without change bend](image)

*Note: 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](image)

**STEP 02** After recovering the fold status, we can change its left bend as follows.
Confirm it and the result will be as follows.

STEP 03 Unfold it to get the following result.

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.

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

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

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

![Convert to Sheetmetal part](image)

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

![Rip gap](image)

**STEP 03** Confirm it to get the following 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.

**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](image)

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](image)
9. **Exercises**

9.1 **Exercise1**

**STEP 01** Create a sketch as follows.

![Figure 109 Sketch for exercise 1]
STEP 02 Exit the sketch and then extrude as follows.

![Image: Extrude flange exercise1]

**Figure 110** Extrude flange exercise1

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

![Image: Add sketch exercise1]

**Figure 111** Add sketch exercise1
**STEP 04** Then go to merge a tab to flange as follows.

![Figure 112](image1)

**Figure 112** Merge a tab_exercise1

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

![Figure 113](image2)

**Figure 113** Add one more tab_exercise1

**STEP 06** Select the edges to add the fillet.

![Figure 114](image3)

**Figure 114** Add Fillet_exercise1
**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.

![Sketch plane](image)

Figure 117 Sketch for another slot_exercise1

**STEP 10** Extrude and cut the slot as follows.

![Extrude another slot](image)

Figure 118 Extrude another slot
**STEP 11** Fillet the corners as follows.

![Figure 119](image1)

**STEP 12** Unfold it as follows:

![Figure 120](image2)
**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.
9.2 Exercise2

**STEP 01** Create a sketch as follows.

![Sketch](image1)

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

![Tab](image2)
**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](image)

**STEP 06** Create flare dimple as follows.

![Figure 128 Flare Dimple_exercise2](image)
**STEP 07** Create sketch on XY plane for cutting an open slot as follows.

![Sketch for flare dimpe_exercise2](image)

**Figure 129** Sketch for flare dimpe_exercise2

**STEP 08** Normal cut the flare dimple as follows.

![Normal cut settings](image)

**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.

![Sketch Plane](image)

**Figure 131** Plane for sketch of Louver_exercsie2

**STEP 09** Create the louver as follows.

![Louver Attributes](image)

**Figure 132** Louver_exercsie2
**STEP 10** With “Mirror Feature” command to mirror louver feature as follows.

Figure 133  Mirror louver feature_exercise2

**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.

![Swept Flange dialog box](image1)

**Figure 135 Swept flange_exercise2**

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

![Sketch plane](image2)

**Figure 136 Second new plane for swept flange profile_exercsies2**
**STEP 14** Create the second Swept flange as follows.

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

---

**Figure 137**  Second swept flange_exercesie2

**Figure 138**  Final result for exercise2