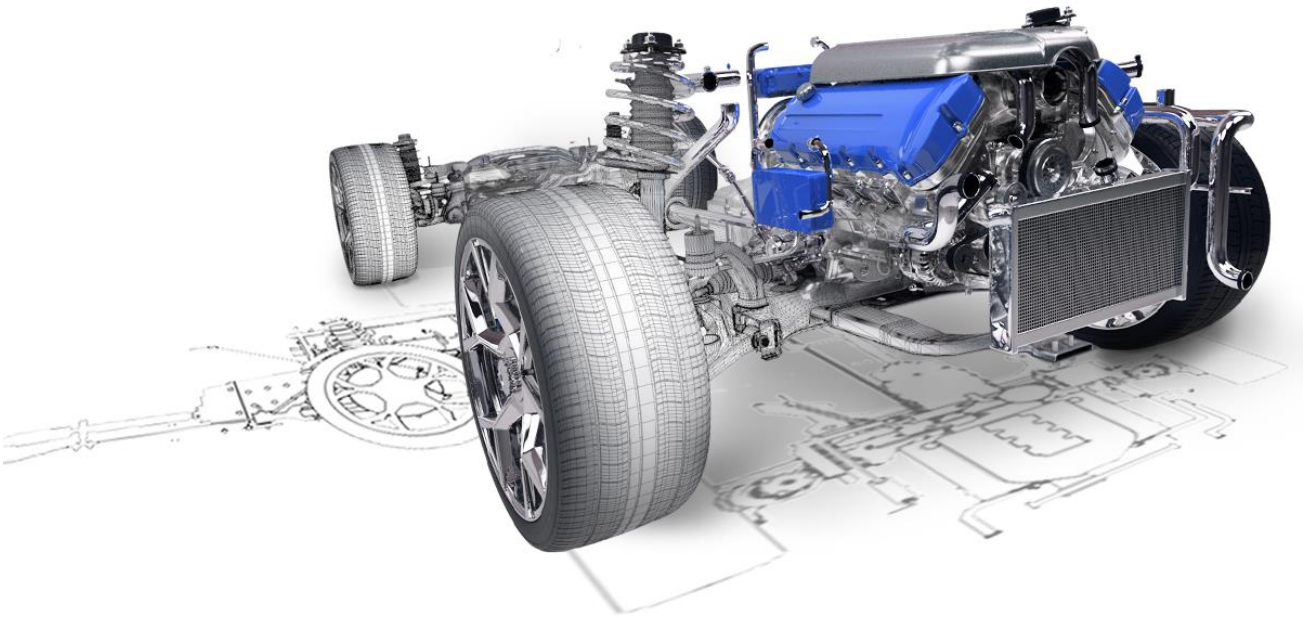


ZW3D From Entry to Master

2D Sheet

Contents

2D Sheet

Key Points:

- ✧ Create the various views for the parts and assemblies
- ✧ Create/edit the dimension
- ✧ Add the annotations and symbols (Datum, Feature Control Symbol, Surface Finish Symbol)
- ✧ Create the text annotation
- ✧ Create a BOM table and the Balloons

1.1 2D Sheet Introduction

In the process of product design and manufacturing, 2D sheet is an important and widely used document even the 3D modeling is very visual.

With ZW3D to design the product 3D mode, then the associative 2D sheet can be generated. When 3D model is changed, the 2D sheet is auto-updated.

1.1.1 Elements of 2D Sheet

Generally, the 2D sheet of a part consists of three parts:

1	Standard view (Top, Front, Right, Left, Bottom, Back and Isometric view), Projection view, Section view, Detail view and so on.
2	Dimension (Shape dimension & Position dimension), Tolerance (Dimension tolerances, Form tolerances & Position tolerances), Datum symbol, Surface finish symbol and Text annotation.
3	Sheet format: Sheet border, Title block and so on.

2D sheet of an assembly includes many different views, assembly dimension, fit dimension, BOM table and so on.

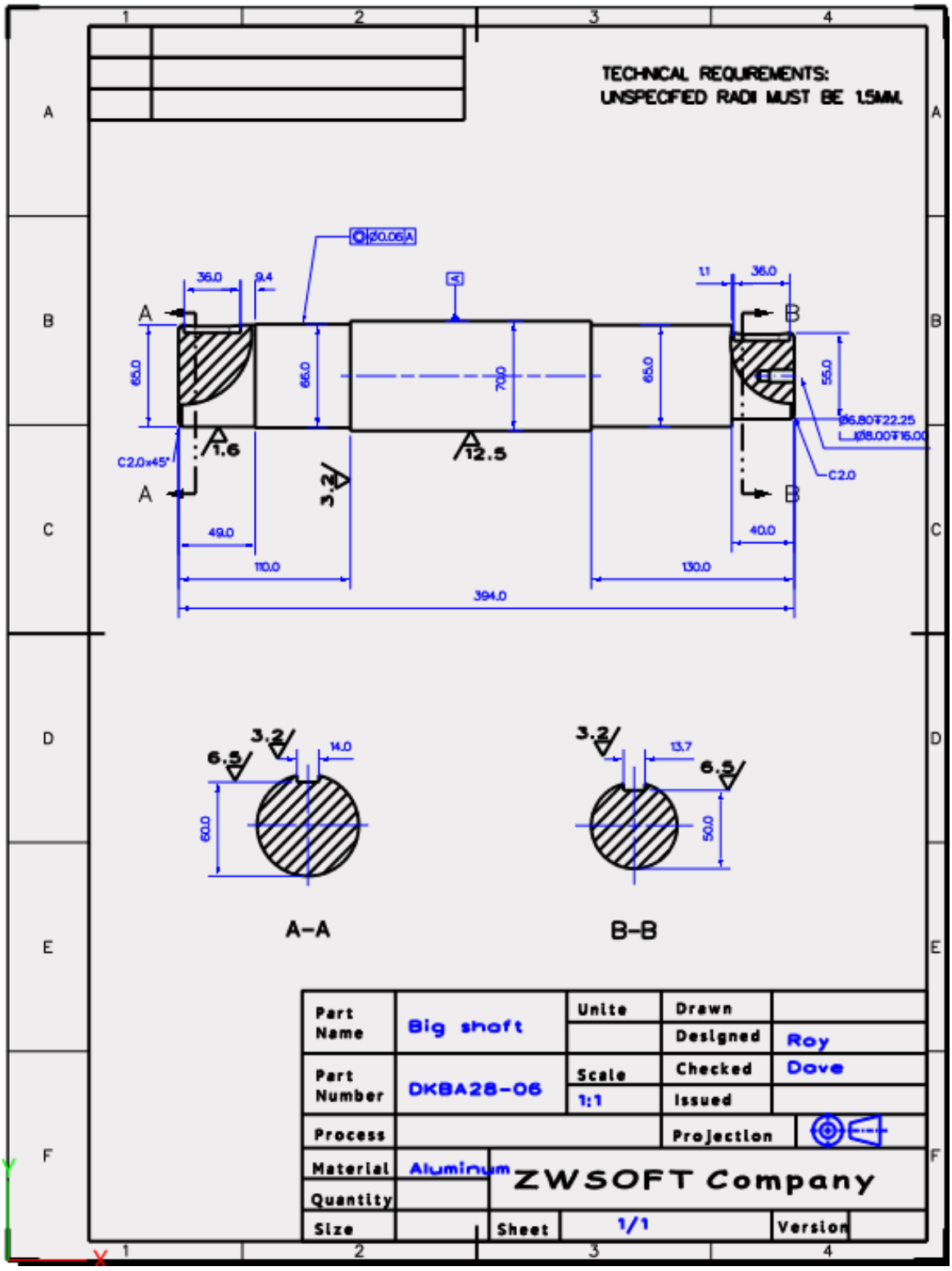


Figure1 2D Sheet

1.1.2 Create a New 2D Sheet

Method1: In modeling environment, insert a new 2D Sheet by picking “2D Sheet” option from Document Aware toolbars or the right menu located working area. Then select a template to

create the new 2D sheet. At the same time, the standard view command is automatically activated.

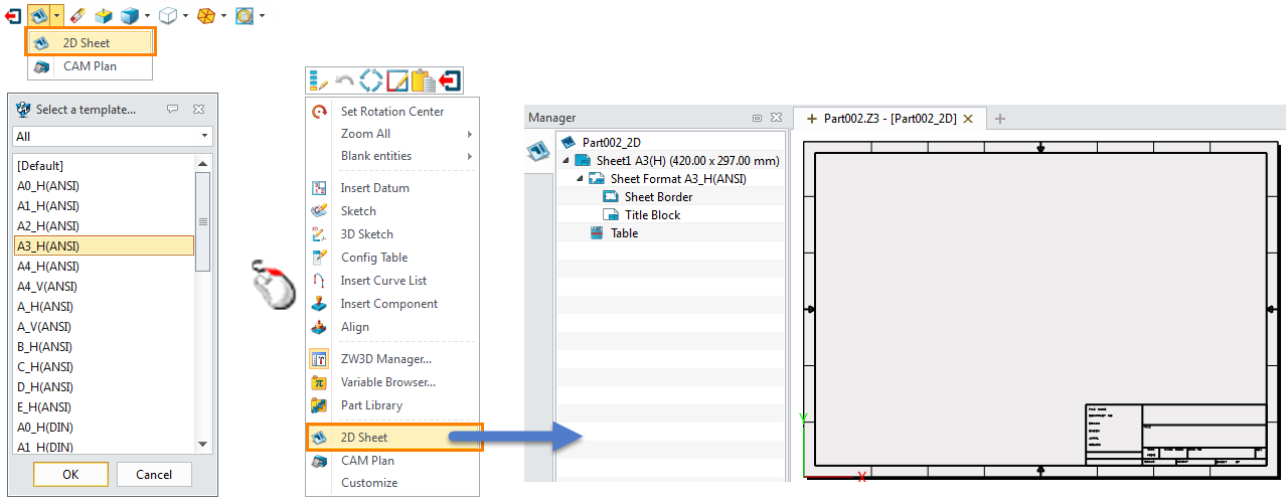


Figure2 Create a New 2D Sheet -Method1

Method2: Click "New" command from the Ribbon toolbar, then select the file type "Drawing Sheet" and drawing template. Give the name and click "OK" button, then the new 2D sheet is created.

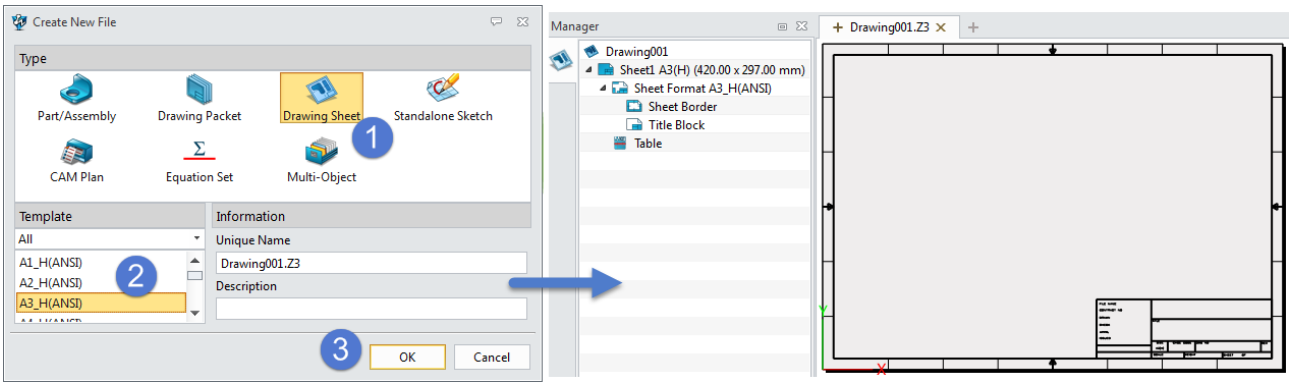


Figure3 Create a New 2D Sheet –Method2

Method3: Click "New" command from the Ribbon toolbar, then select the file type "Drawing Pocket" to create a drawing file. Then click "+" button to create a new 2D sheet in the file level. Other operations are similar with Method2.

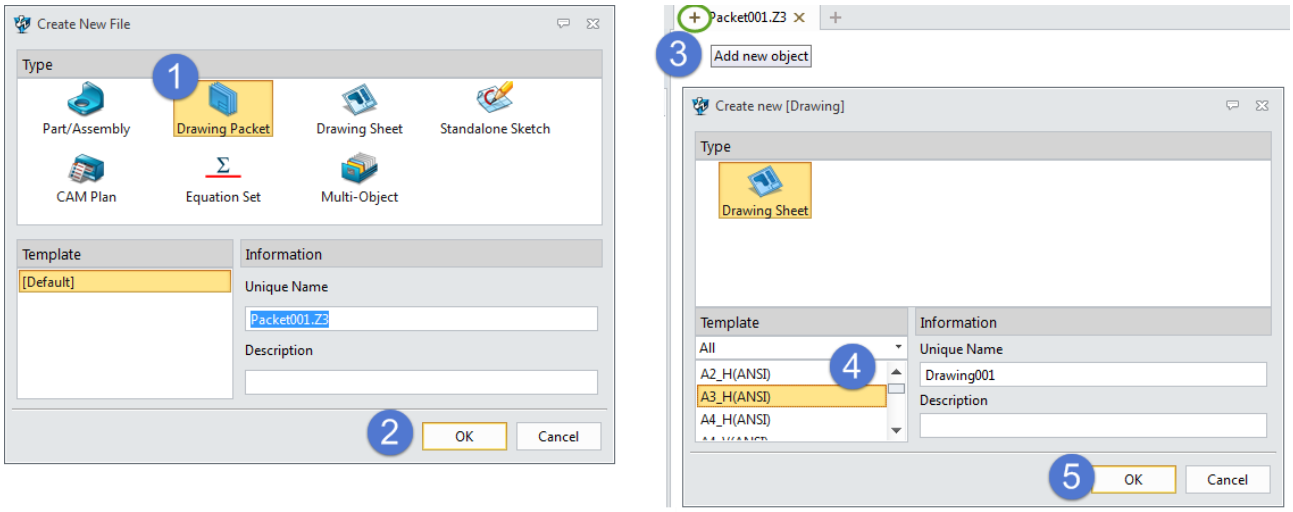


Figure4 Create a New 2D Sheet –Method3

1.1.3 2D Sheet Setting

1. In configuration form, some default parameters of drawing sheet can be modified. See the image below.

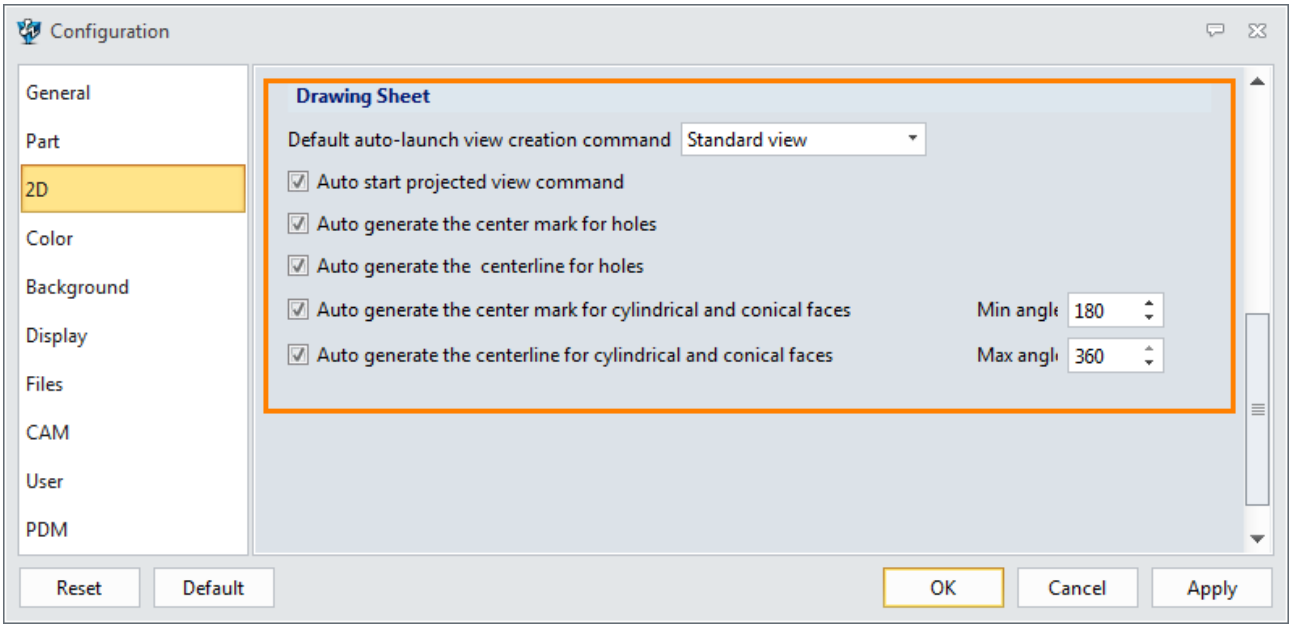
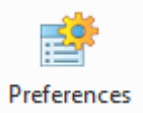


Figure5 Configuration Form

2. **Tools ribbon toolbar->Setting->** 

Use this command to set some global drawing sheet settings, including units, mass units, grid spacing and projection type.

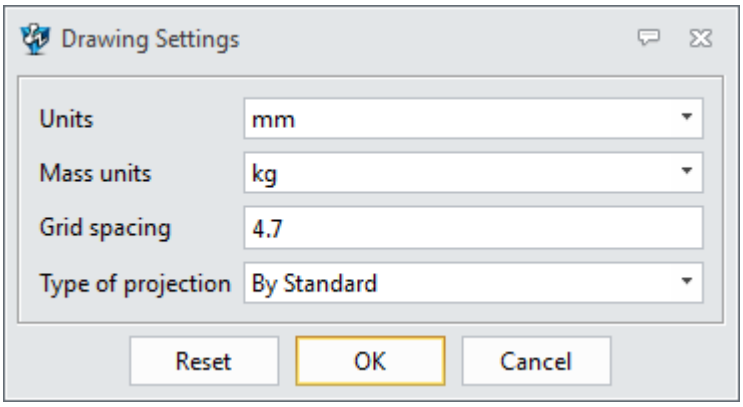


Figure6 Drawing Setting

3. **Sheet Manager->  (on a sheet )-> Attributes**

Set the basic attribute of the selected sheet, such as sheet name, scale, paper color.

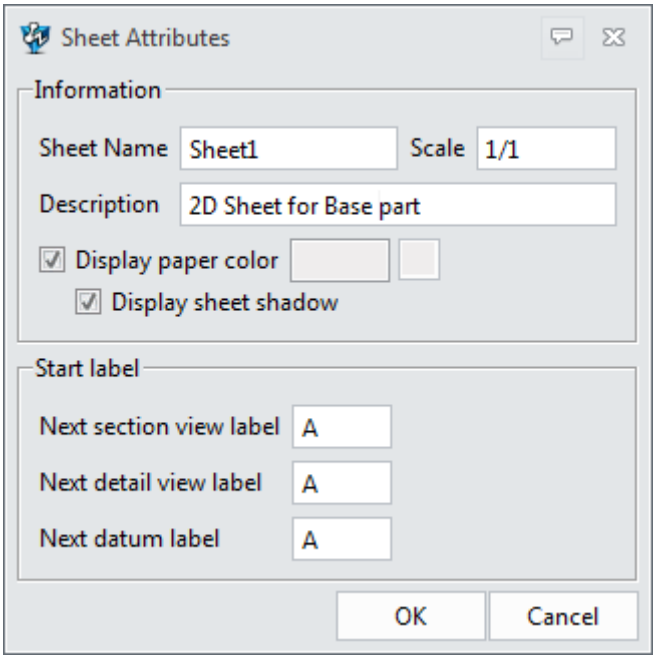


Figure7 Sheet Attributes

4. **Sheet Manager->  (on sheet format )-> Attributes**

You can redefine or customize sheet format according to different requirements.

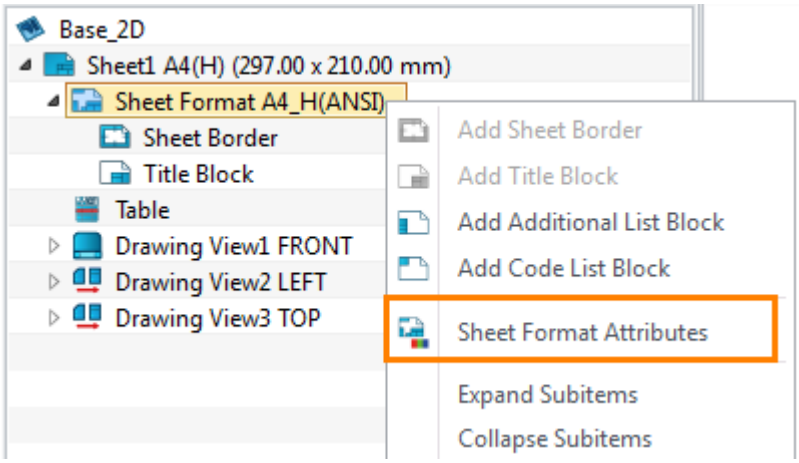


Figure8 Redefine Sheet Format Attribute

1.1.4 Customize 2D Drawing Template

Many different 2D Drawing templates are builtin ZW3D, such as, ANSI, ISO, GB, and DIN. But many companies have their own templates to meet some customized requirements.

Generally, 2D drawing template customization includes three parts. Next, we will introduce how to customize 2D Drawing template from an empty file.

1) Create a new 2D sheet and set 2D sheet size.
2) Define sheet format, including sheet border, title block and so on.
3) Set the layer, or style attribute if necessary.

Task1: Create a new 2D sheet

STEP 01 Confirm the object template file in use.

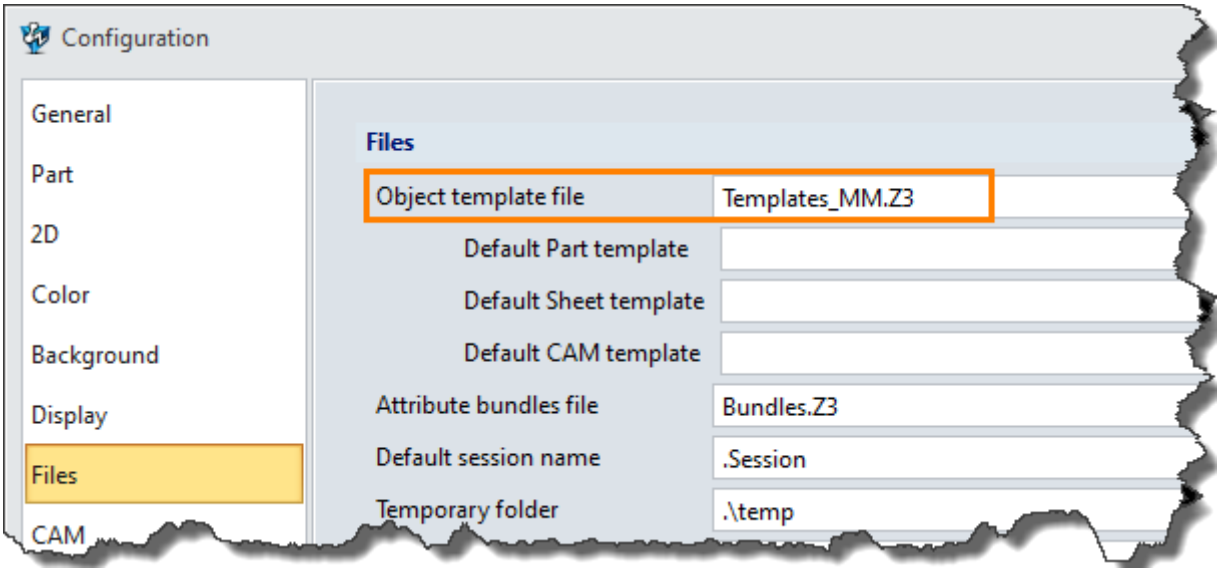


Figure9 Configuration Form

STEP 02 Open the 2D drawing template file (**File menu->Templates...**), then create a new 2D sheet with an empty template.

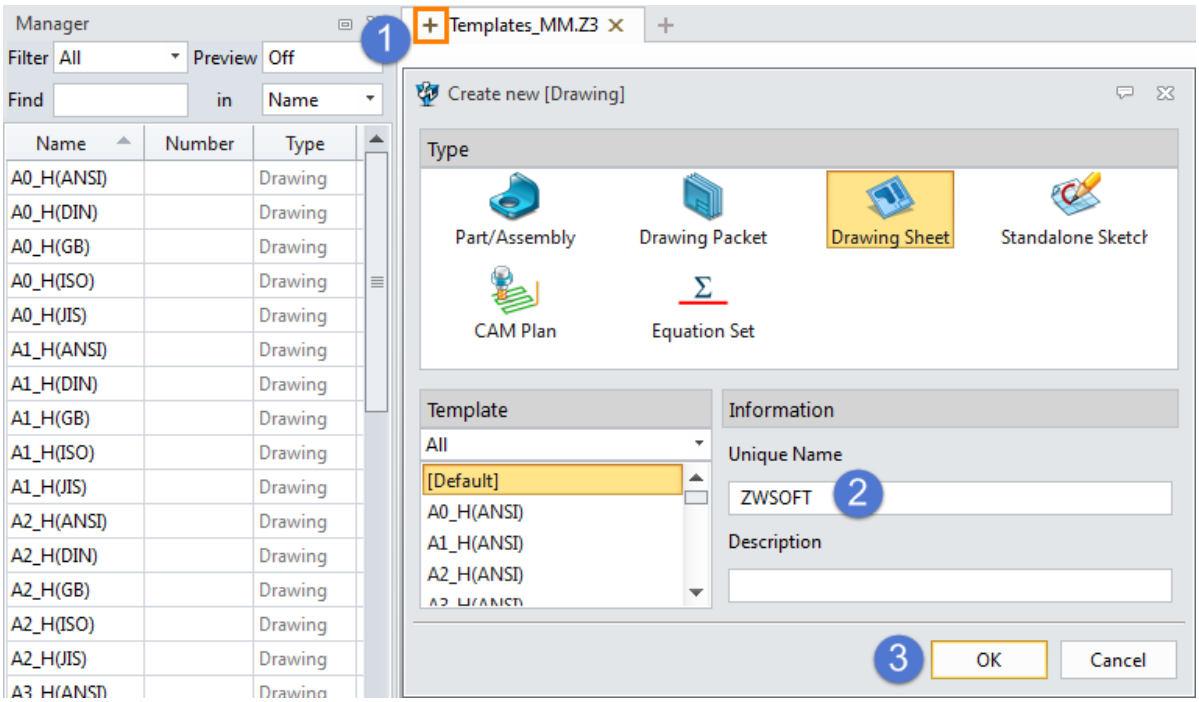


Figure10 Create a new 2D drawing

Task2: Define sheet format

STEP 01 In sheet manager, right click on sheet format, then select "Sheet Format Attributes".

Sheet size: Select the standard size or enter the customized values in **Width** and **Height** fields.

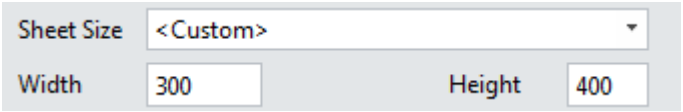


Figure11 Sheet Size

Sheet Border: Define the margin value of sheet border and partition value of trimming mark and centering mark.

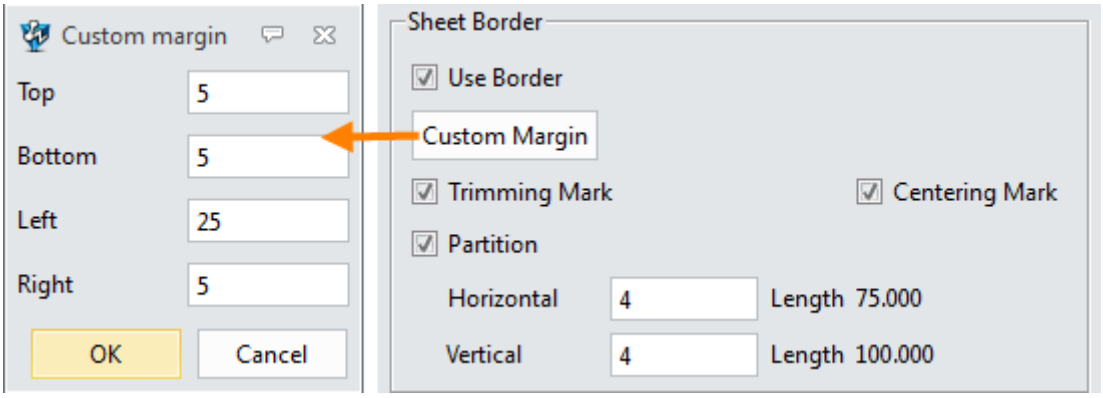


Figure12 Sheet Border

Insert Block: Select a standard block type or predefined block type which is stored in corresponding template file. Or don't define it in here.

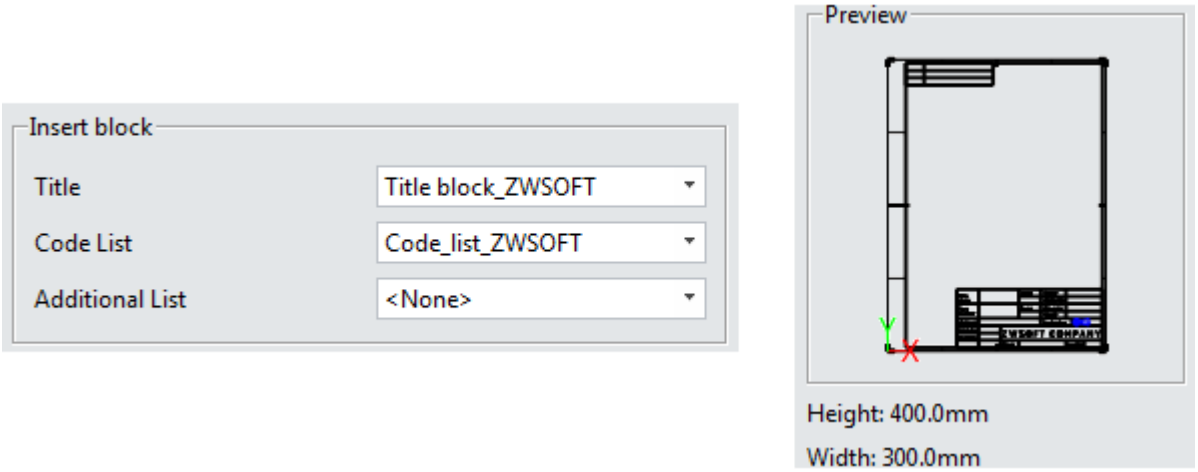


Figure13 Customized Block and Template Preview

Notes: *All 2D drawing template files locate the resource folder of installation directory.*

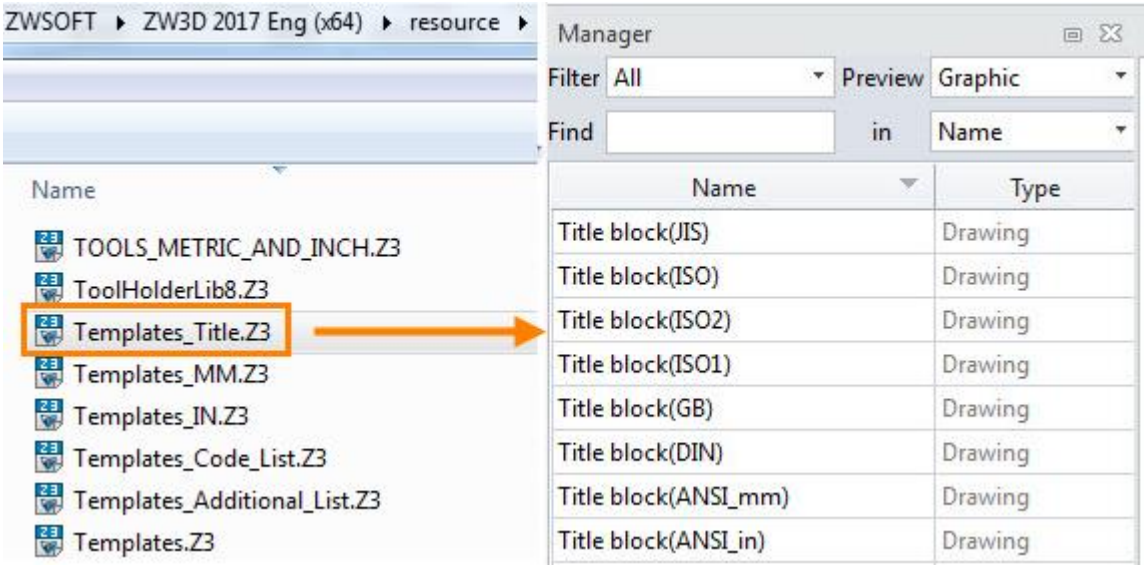


Figure14 2D Drawing Templates

How to add the customized block in the template file, such as Title block?

STEP 01 Open the corresponding template file (Template_Title).

STEP 02 Create a new drawing sheet, and give a name such as "Title block_ZWSOFT".

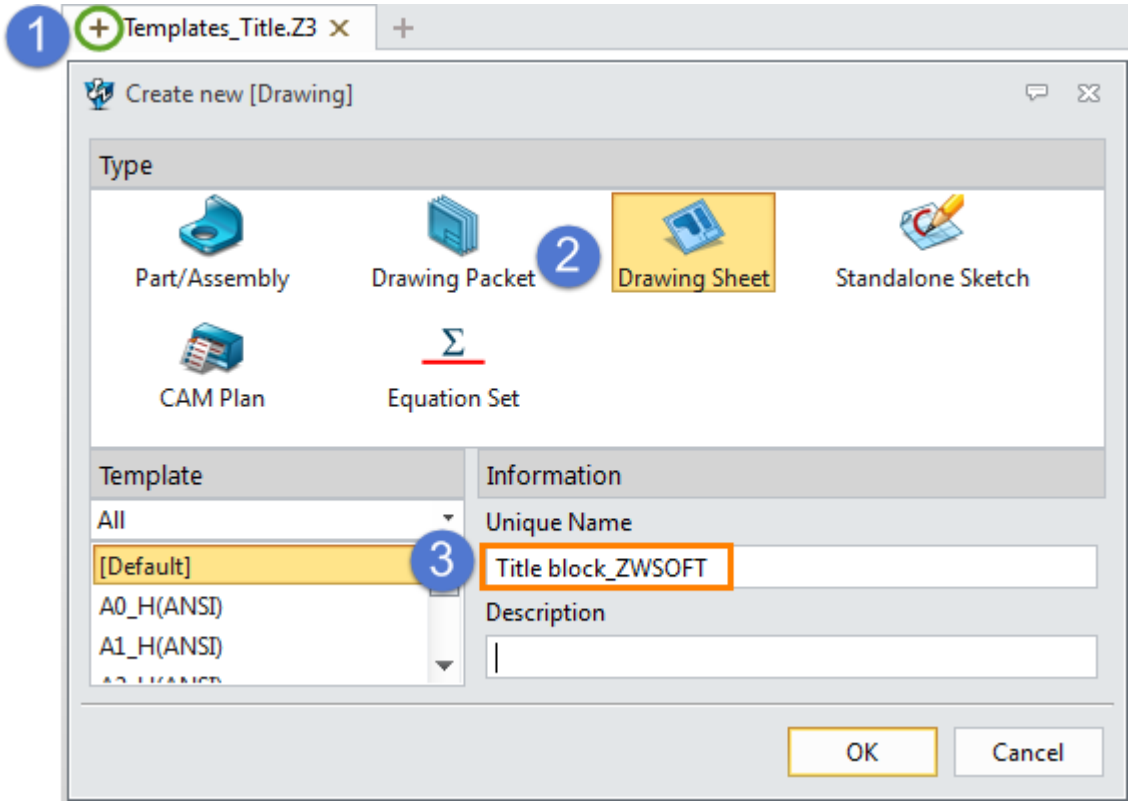


Figure15 Create a New Drawing in the Template

STEP 03 In sheet manager, right click on sheet format, then click "Add Title Block" option. Then draw the code list block in the sketch level.

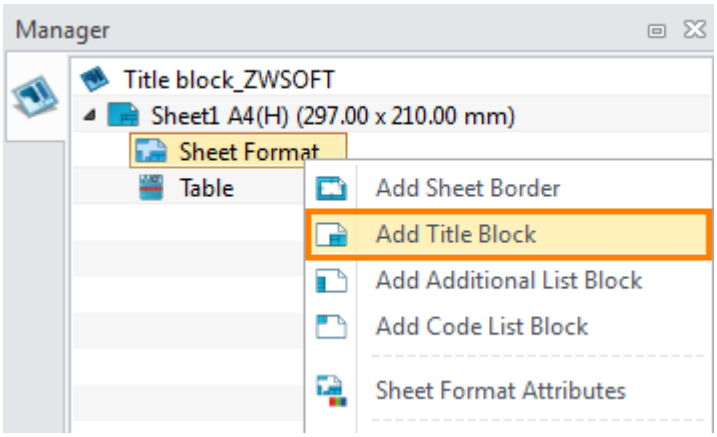


Figure16 Add Title Block Manually

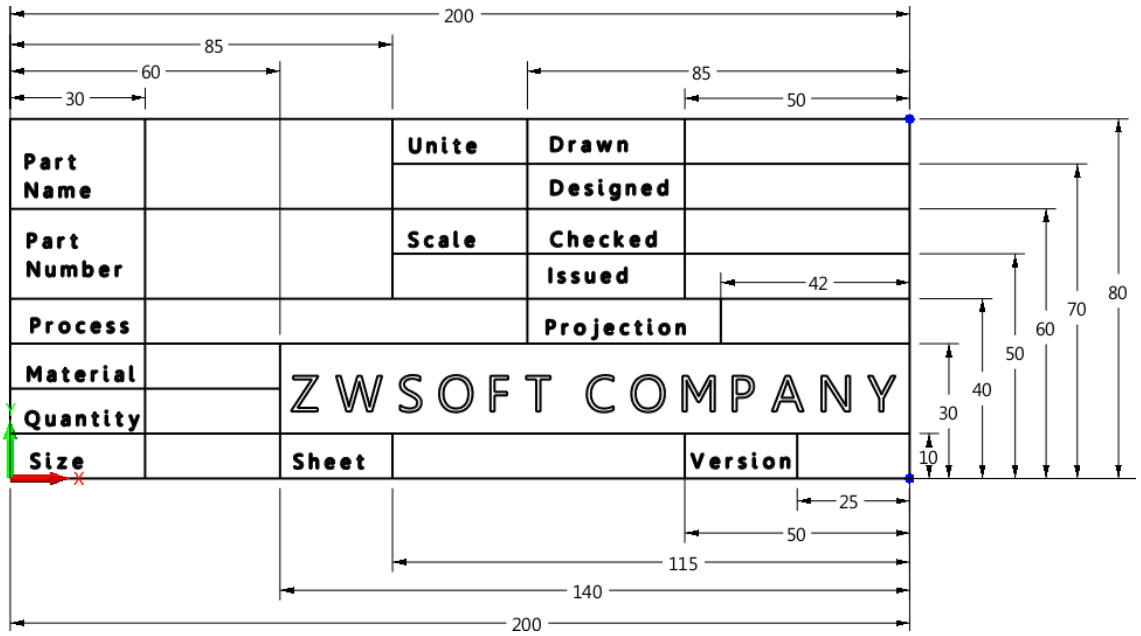


Figure17 Draw Title Block in Sketching

STEP 04 In title block, create needed links with part attribute by text command.

With the variable browser you can browse all available variables, then select the needed one to embed part attribute data in text strings. See the image below.

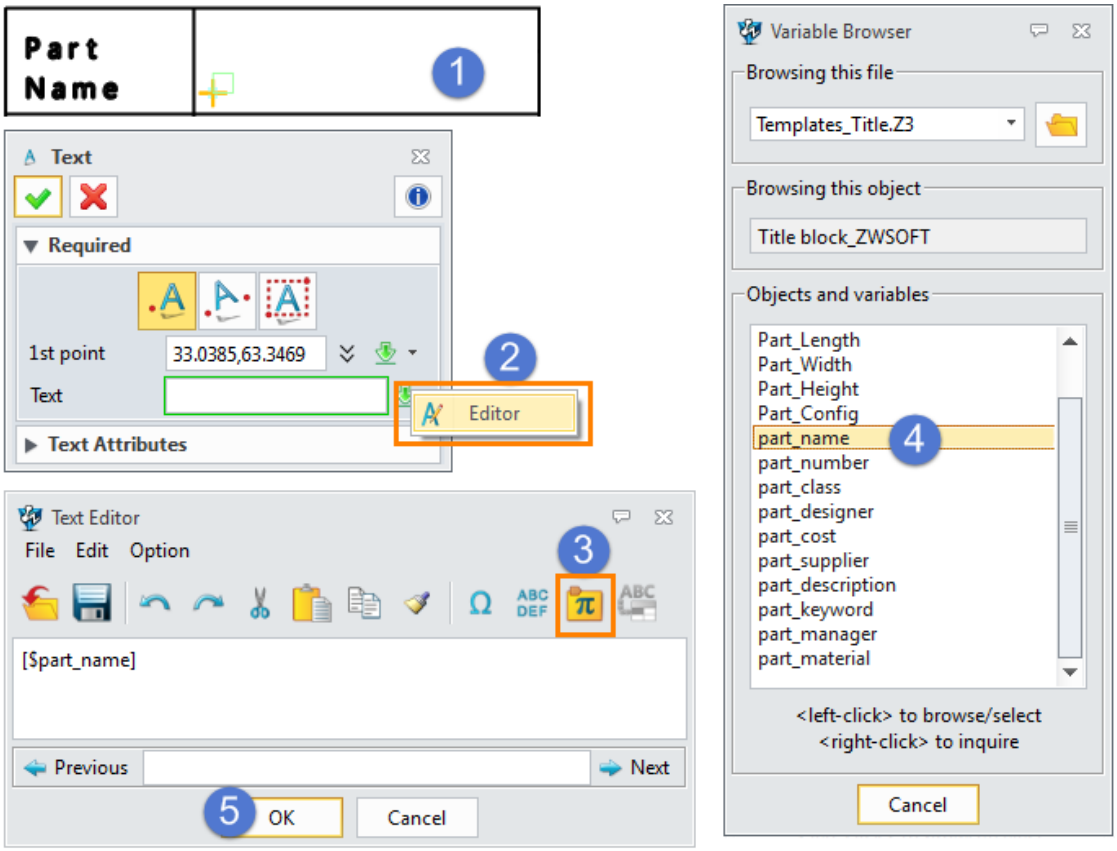


Figure18 Embed Part Attribute

Note: *User attribute links also could be added in template. Just keep the same attribute name.*

STEP 05 When finishing the definition of link relationship, you will get the following result.

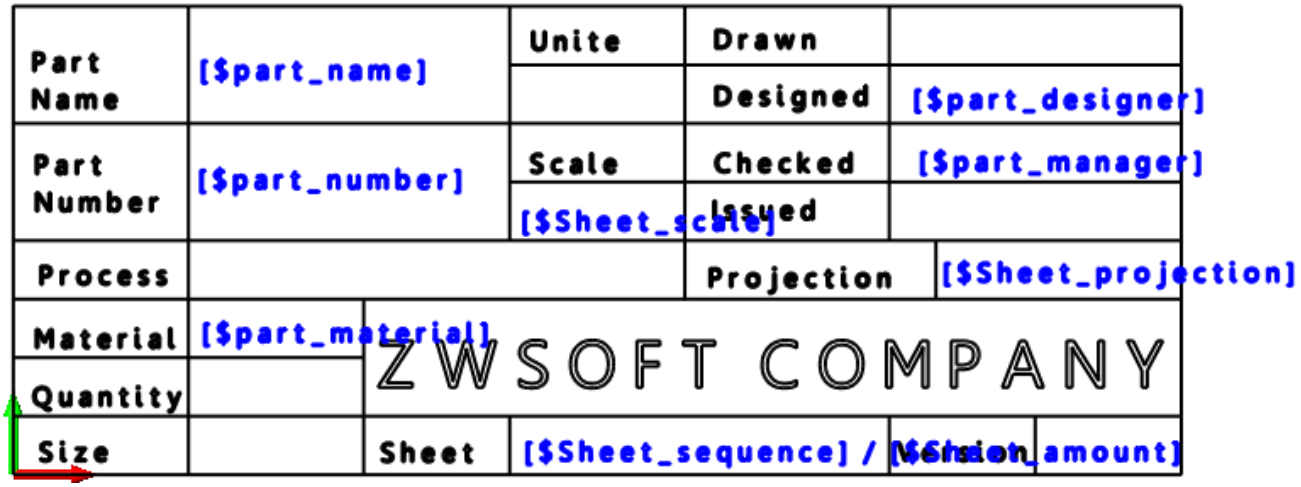


Figure19 Title Block in Sketch Level

STEP 06 Back to 2D drawing level, then the customized title block is done.

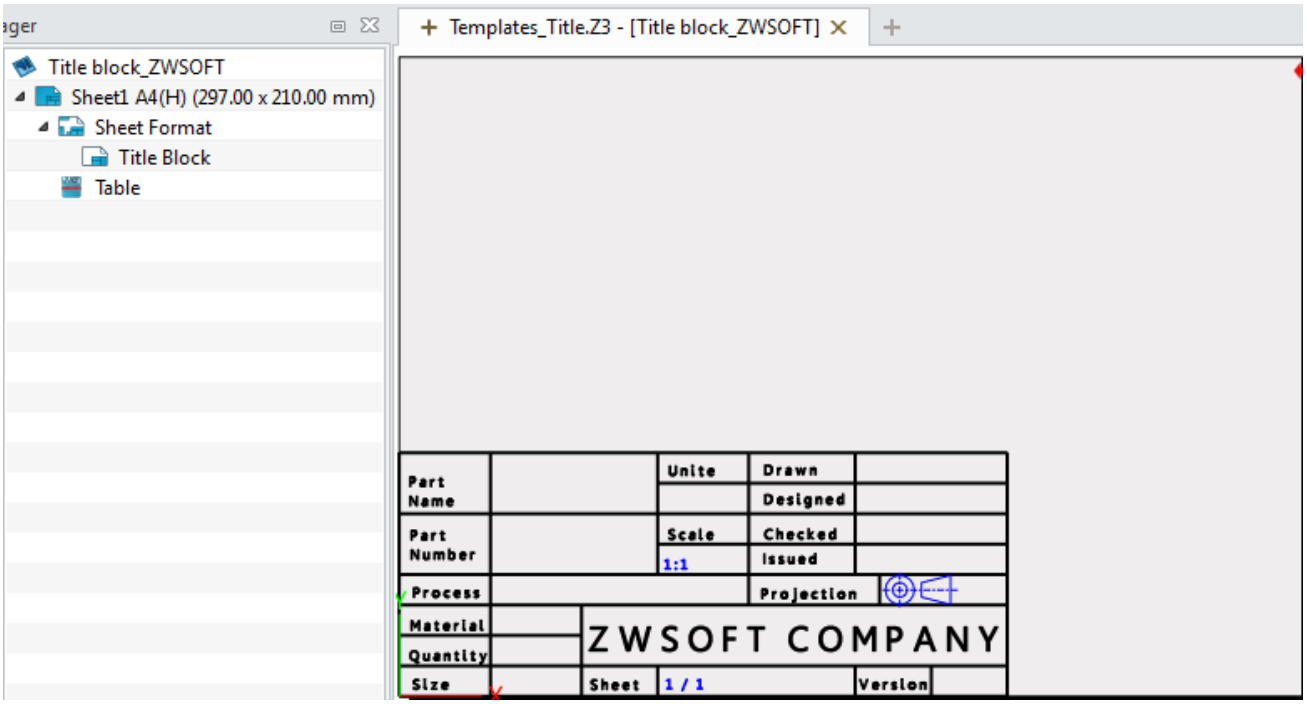


Figure20 Customized Title Block

Task3: Define the layer attribute

Select layer manager from the document aware toolbars, then create the new layers and set the parameters, as shown in the image below.

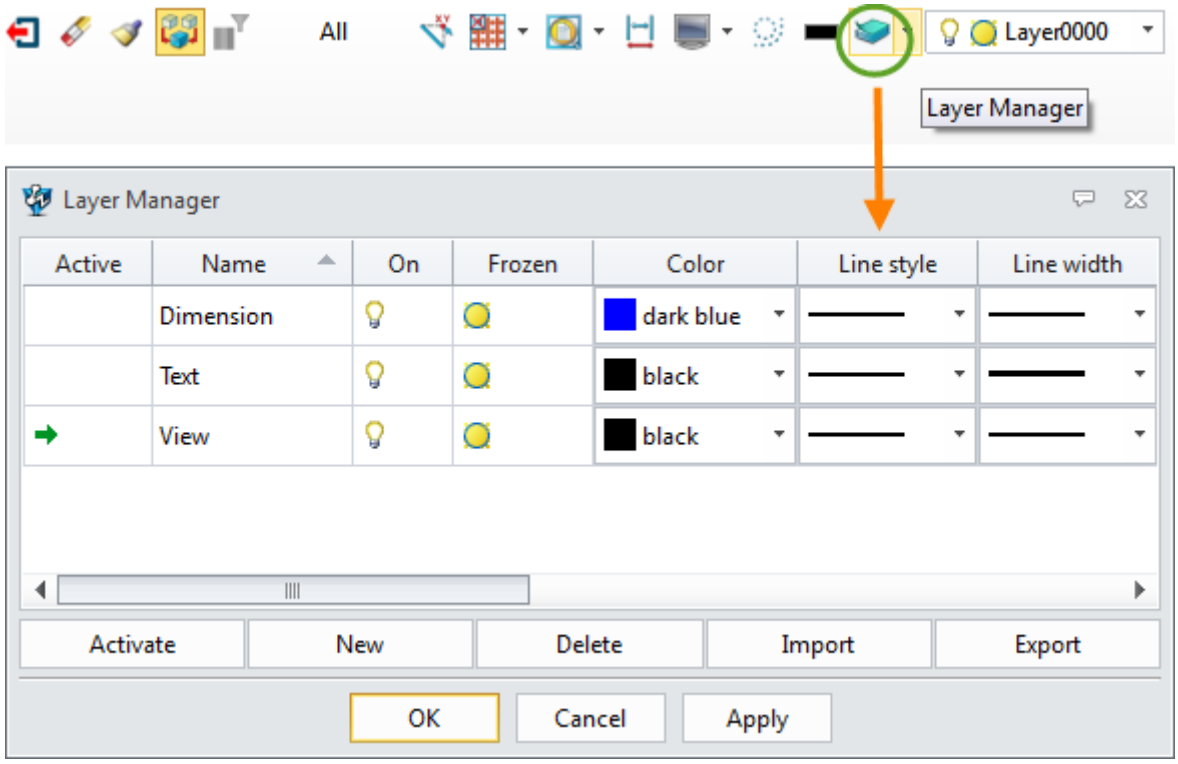


Figure21 Layer Manager

Task4: Define the style attribute

STEP 01 Select style manager command from the ribbon toolbar or sheet manager.

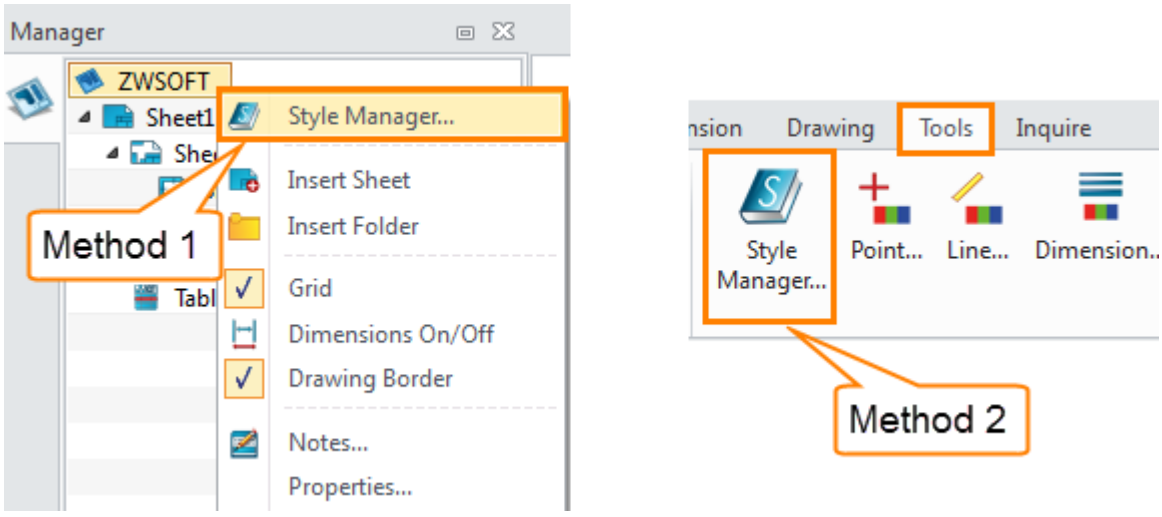


Figure22 Launch Style Manager

STEP 02 Create a new style, such as linear dimension style. Then set the parameters and save them, such as precision, text position.

STEP 03 Double click the new style to activate it as the default style.

STEP 04 Save the customized template at last.

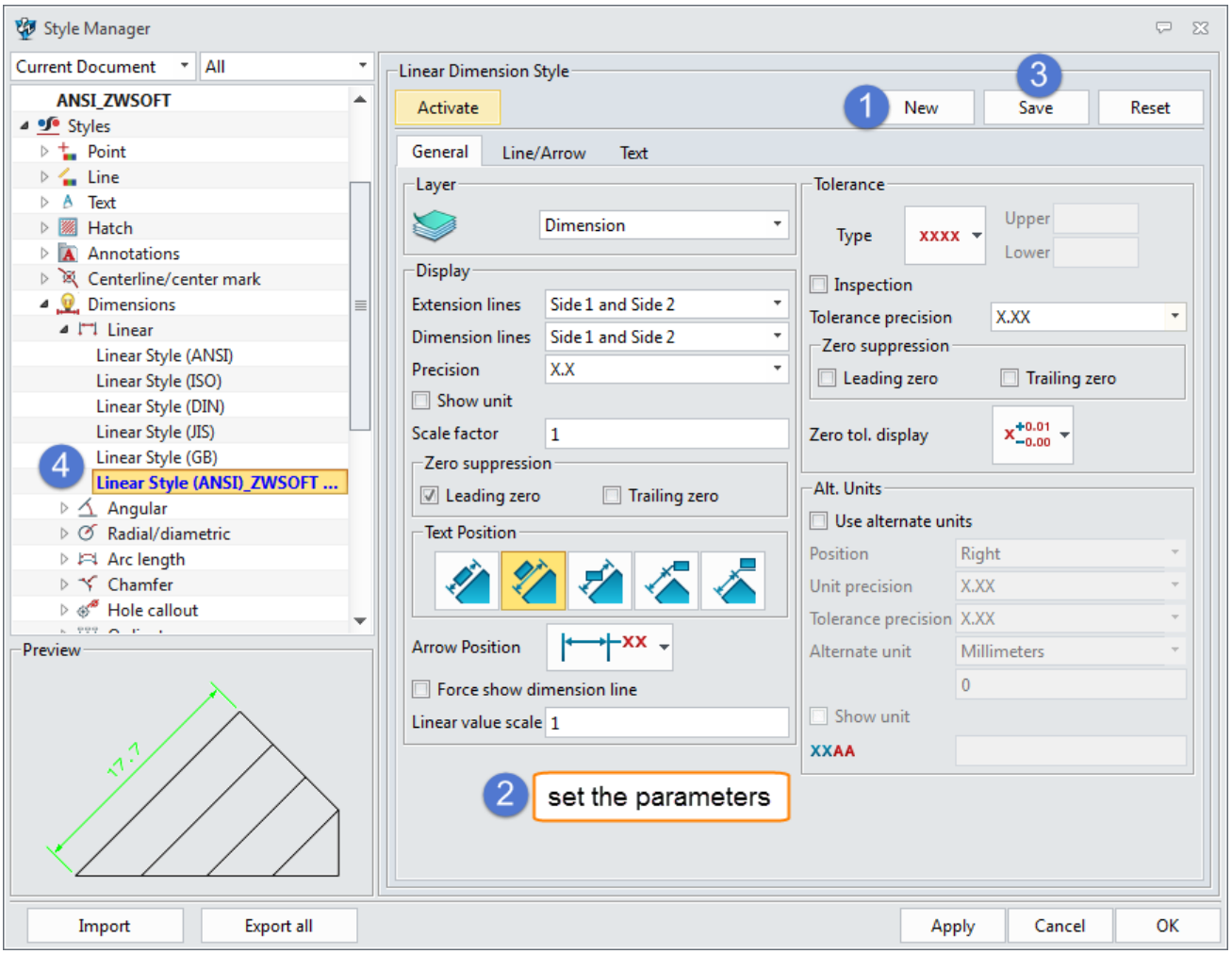


Figure23 Create the new Style

1.2 Create the Views

1.2.1 View Layout

Layout ribbon toolbar->View -> 

Use this command to create a 1 to 7 view layout of a 3D part.

STEP 01 Select one part, then define the layout view and other parameters, such as view label, line attribute.

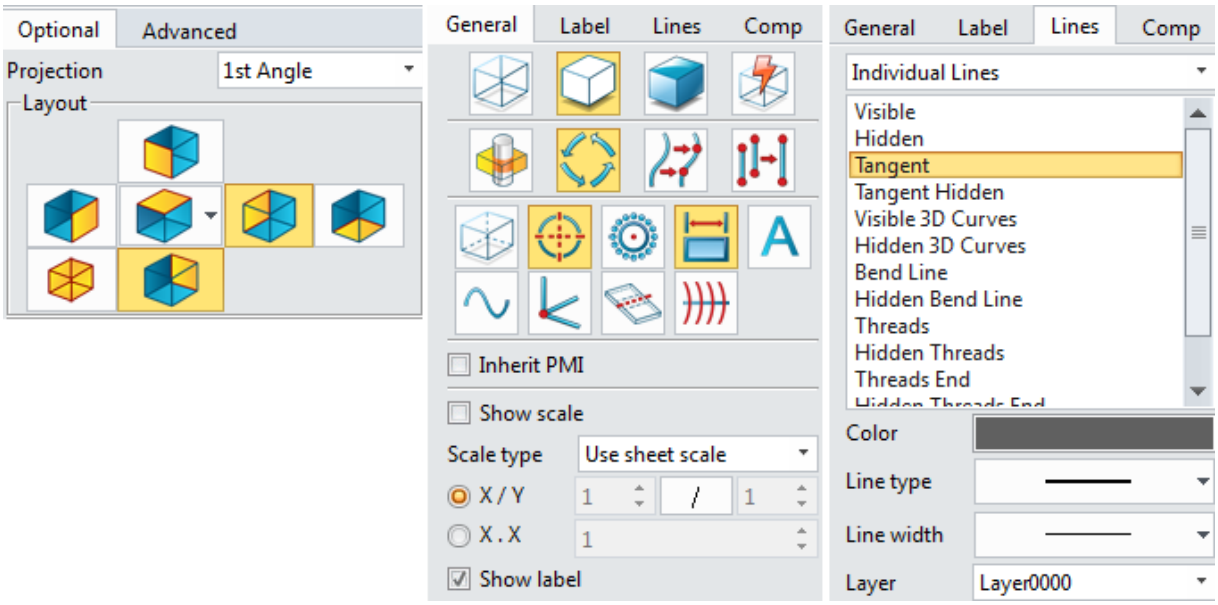


Figure24 Define the Layout Parameters

STEP 02 The result is shown in the image below.

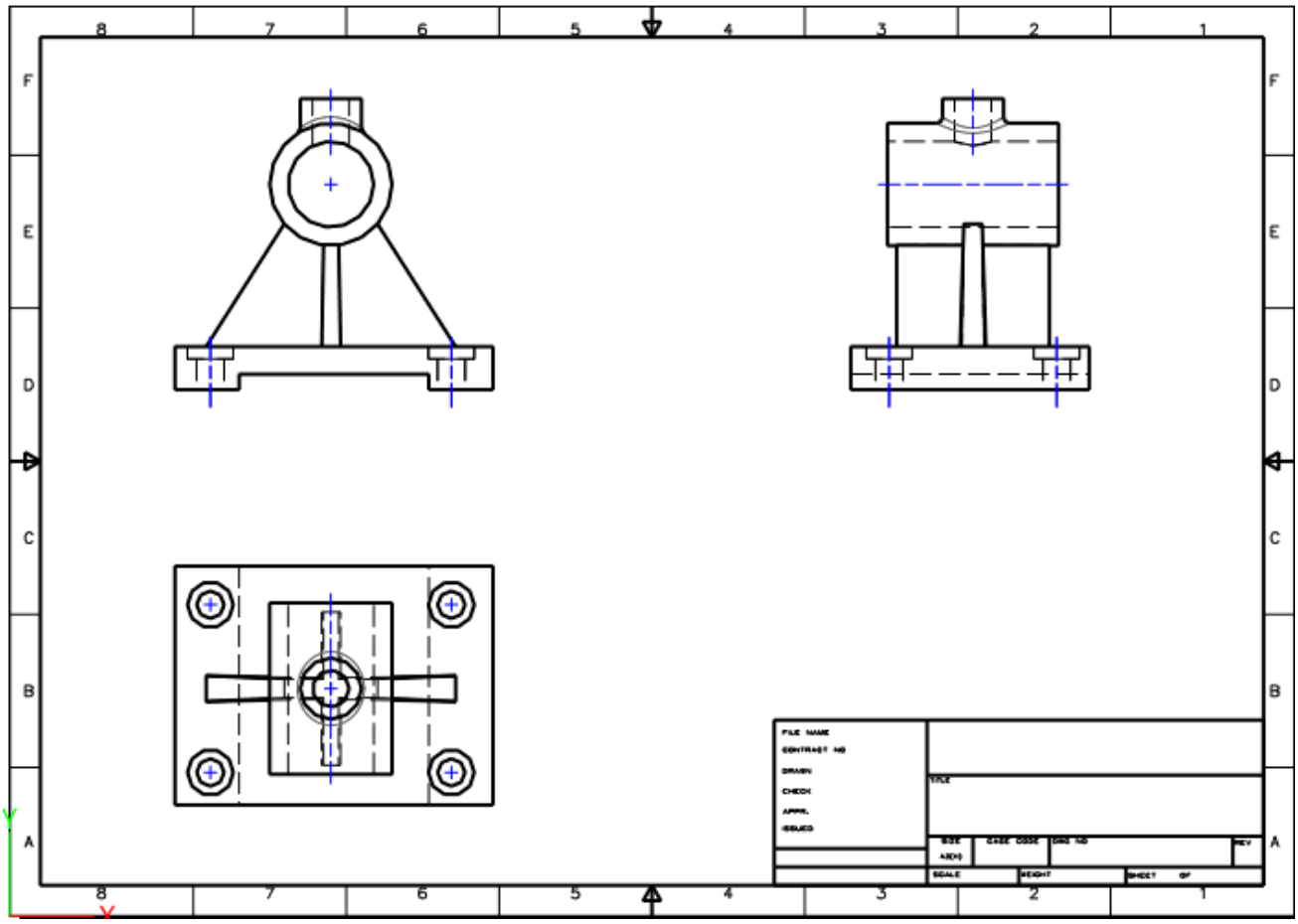


Figure25 View Layout

1.2.2 Standard View

Layout ribbon toolbar->View -> 

Use these commands to create a standard view from a 3D part.

STEP 01 Select the part from a file, then select the view from the drop-down list, such as top view, isometric view, and define the view location, as shown in the image below.

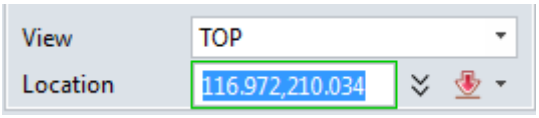


Figure26 Definition of Standard View

STEP 02 Set the other parameters, then click OK to get the view.

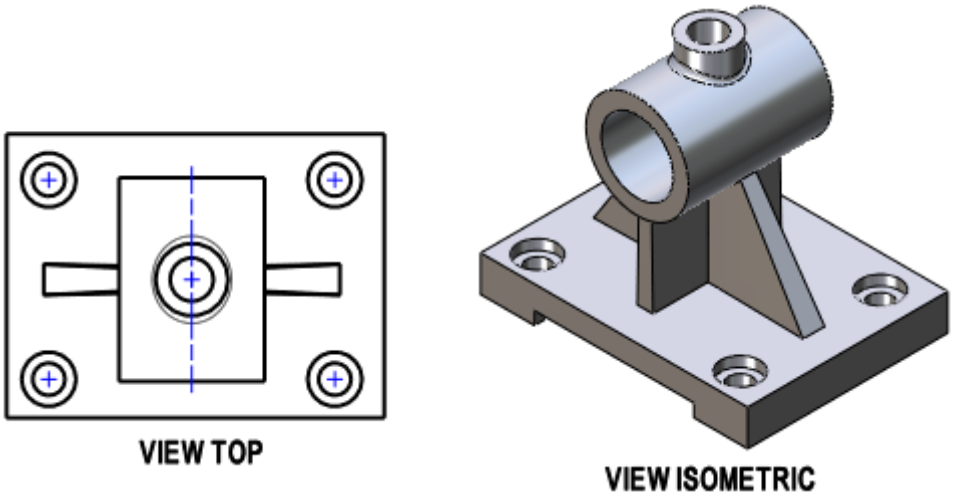


Figure27 Standard View

General View Attributes

(1)  : Show hidden lines / center lines/ threads

(2)  : Show dimensions / text / 3D curves /3D zero point from part

Eg. Turn on “Show dimension”, then get the result as show in the image below.

*(**Note:** Sometimes, the default dimension positions are not optimal. So it is necessary to adjust the dimension position manually.)*

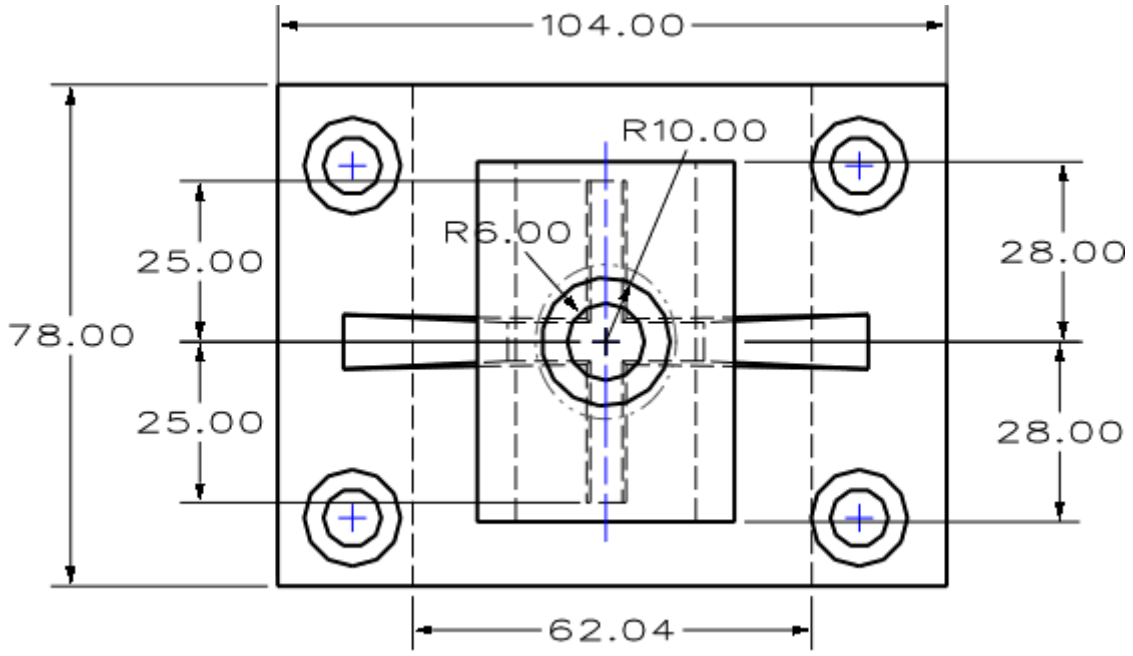
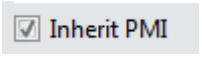


Figure28 View with Dimensions from Part

(3)  : Check this option to show avaliable PMI on related views.

(4) Show scale and label

Refer to the image below to set the parameters and display attributes.

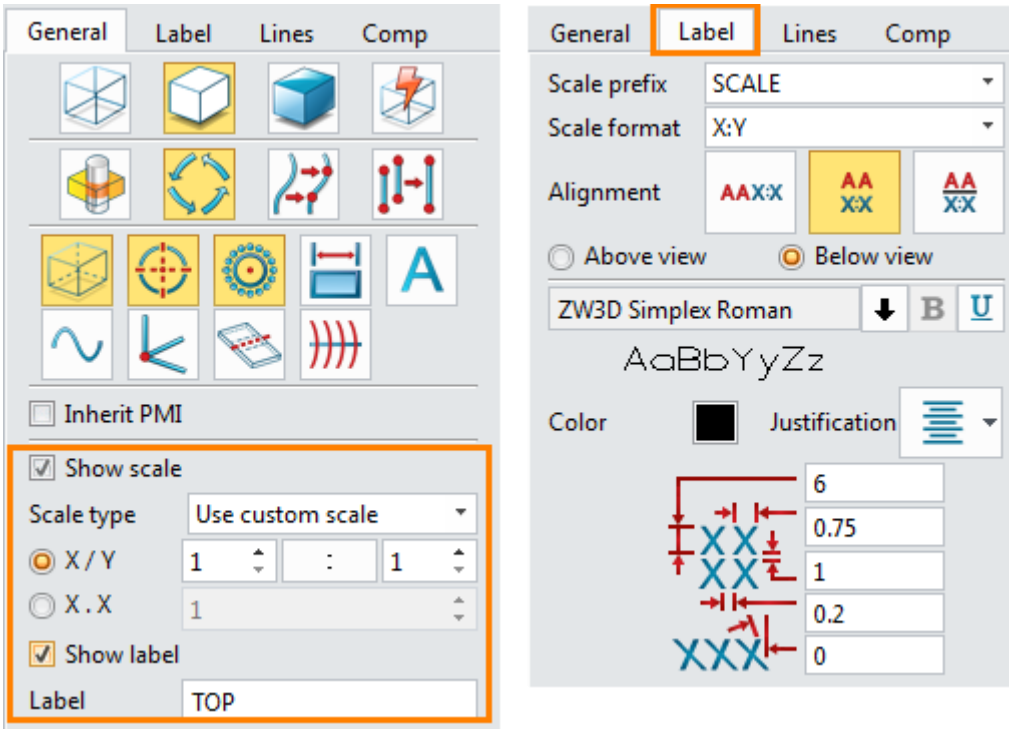


Figure29 Setting of View Scale and Label

(5) Set line attributes

Set the different line attribute (such as color, width and layer) for each line.

(6) Set component visibility

When defining the standard view, you could hide some components or inherit the component visibility from part.

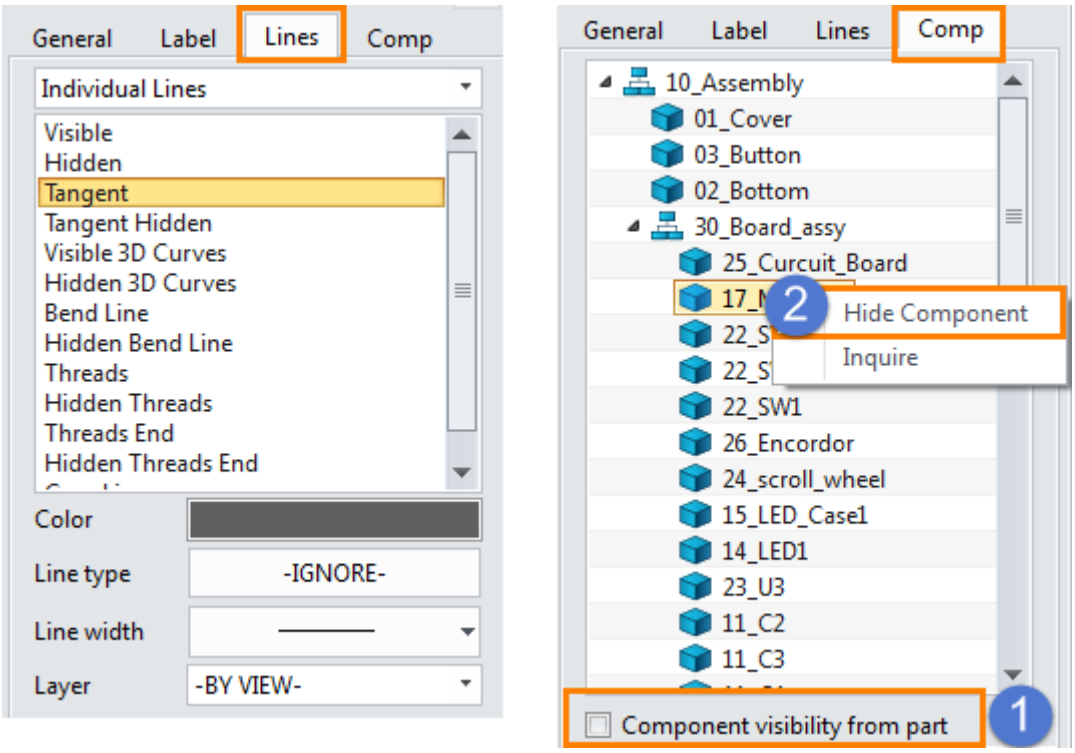


Figure30 Line Attribute and Component Visibility

1.2.3 Projection View

Layout ribbon toolbar->View -> 

Create a view projected from another existing 3D layout view.

STEP 01 Select the base view and define the location of the projection view.

STEP 02 Set the projection type—1st / 3rd angle projection and set the view attribute.

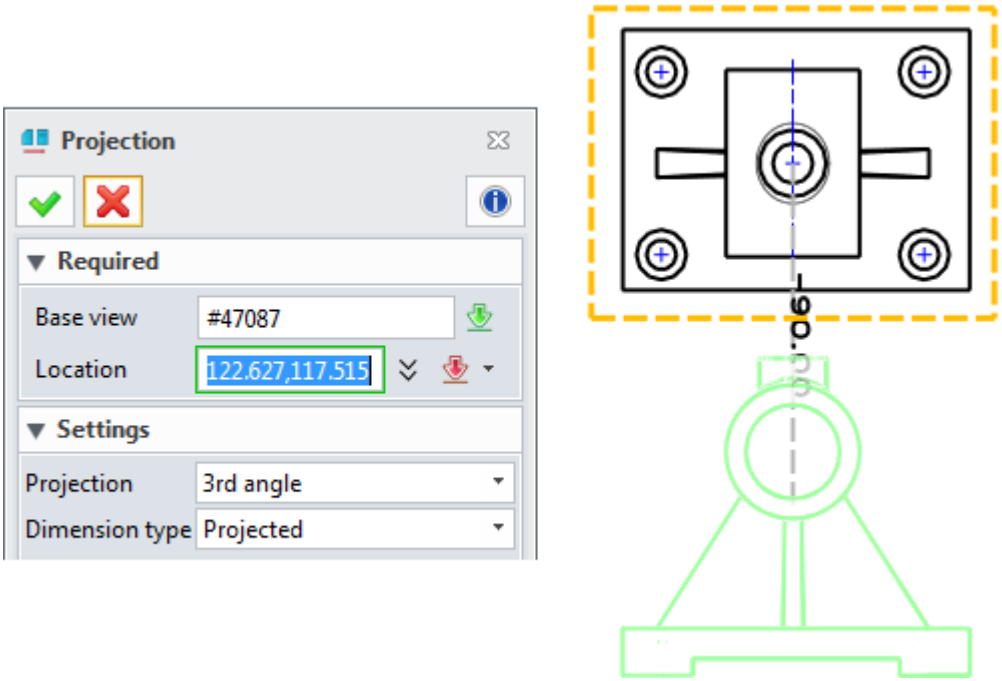


Figure31 Projection View

1.2.4 Auxiliary

Layout ribbon toolbar->View -> 

Auxiliary view is a projection view that is perpendicular from an edge of another layout view. The horizontal or vertical edge is not suitable as the reference line.

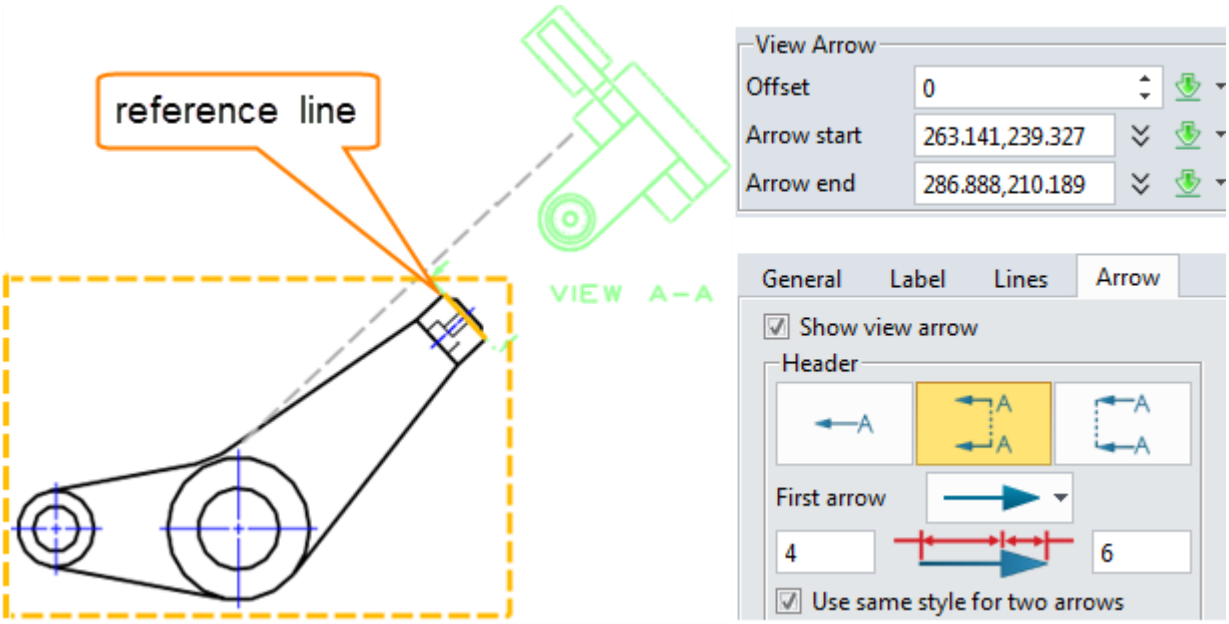


Figure32 Auxiliary View

STEP 01 Select the base view and select the edge as the reference line.

STEP 02 Pick a point as the location of the view.

STEP 03 Set other parameters if necessary, such as arrow attribute, as shown in the above image.

1.2.5 Full Section View

Layout ribbon toolbar->View -> 

Use this command to create a variety of section views of a 3D layout view.

STEP 01 Select the base view.

STEP 02 Pick points to define the section position.

Note: *If two points are picked and full section line through the model, the full section view is created. If more points are picked, the stepped section view could be created, as shown in the right image of Figure 33.*

STEP 03 Select a point as the location of section view.

STEP 04 Set other parameters if necessary.

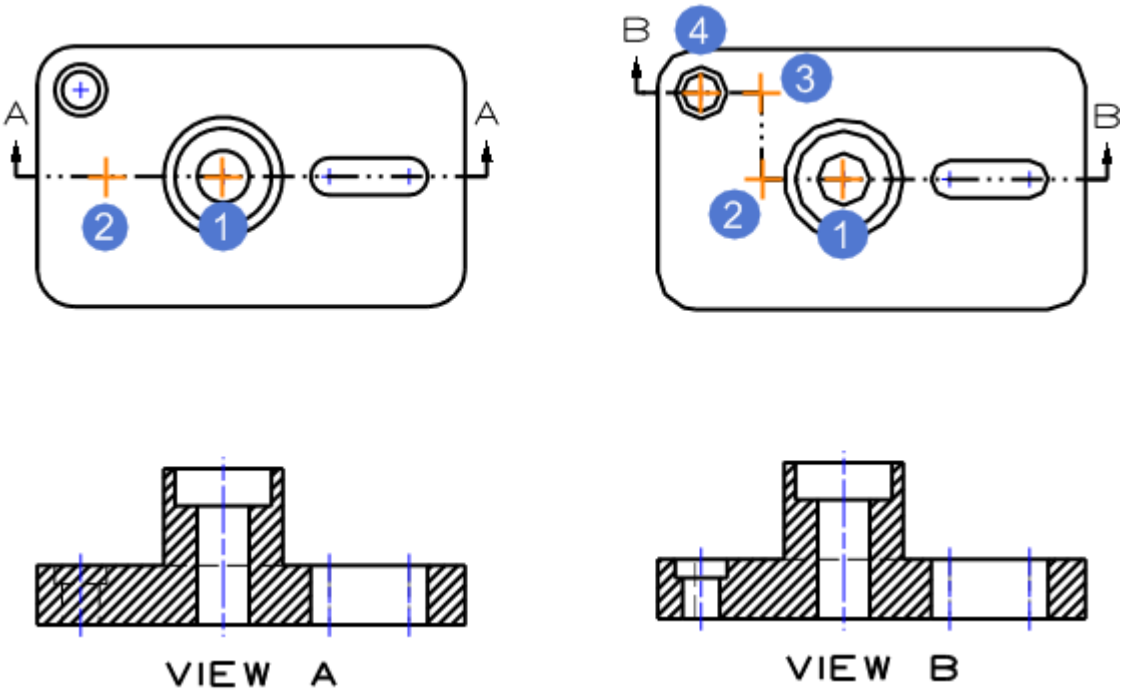


Figure33 Full Section View and Stepped Section View

Section Method

Trimmed Part section method shows a hidden line view of the entire part cut by the section plane. Section Curves section method shows the cross section profile only.

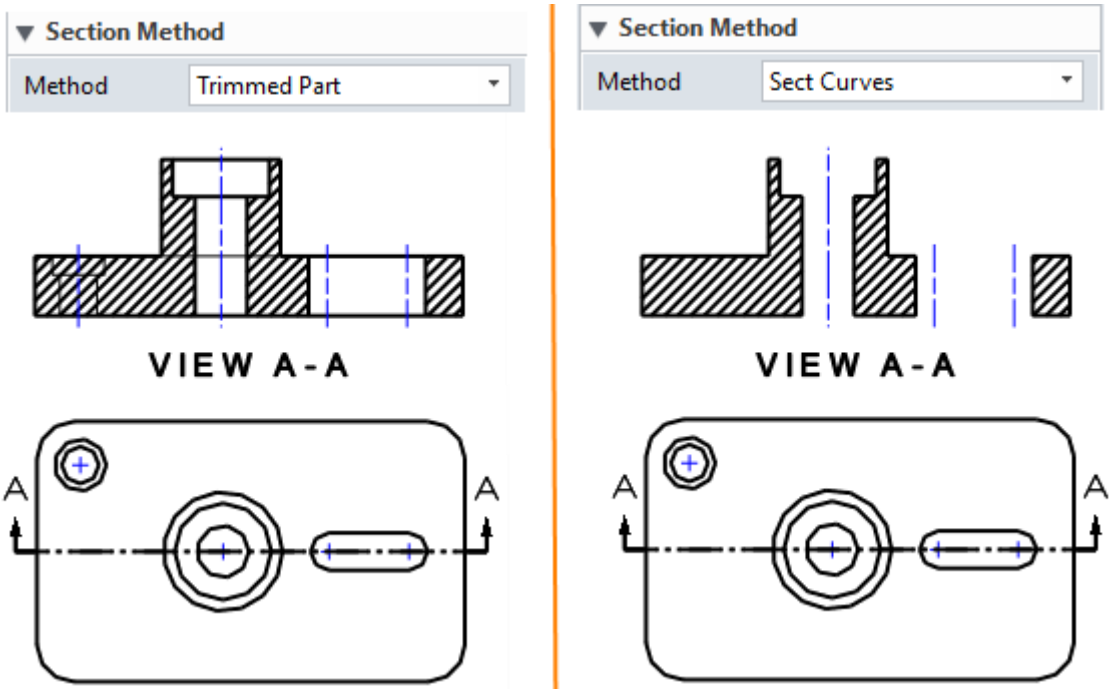


Figure34 Section View-Section Method

Section depth

After defining the section lines use this “Depth” option to further clip the model from the final section view, then get a leaner section view. Full section and 3D named section view provide this option.

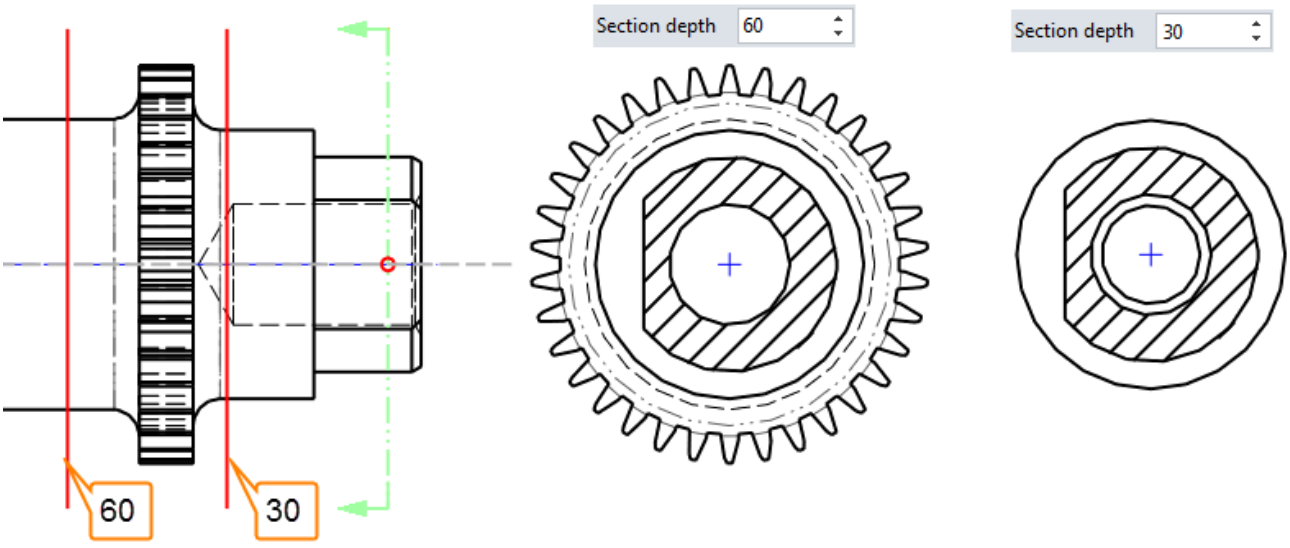


Figure35 Section Depth

Section Option

In part attribute form, the component section/hatch state can be predefined, as shown in the image below.

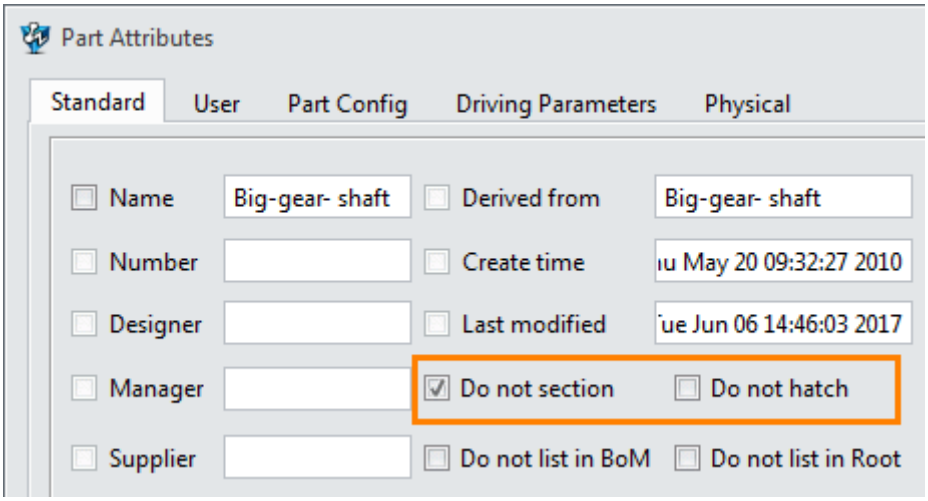


Figure36 Part Attribute

When creating a section view of the assembly, these attributes can be inherited, as shown the left image of Figure 37. If you want to redefine the component section state and hatch state, unpick the section options, then right click the component to define, as shown the right image of Figure 37.

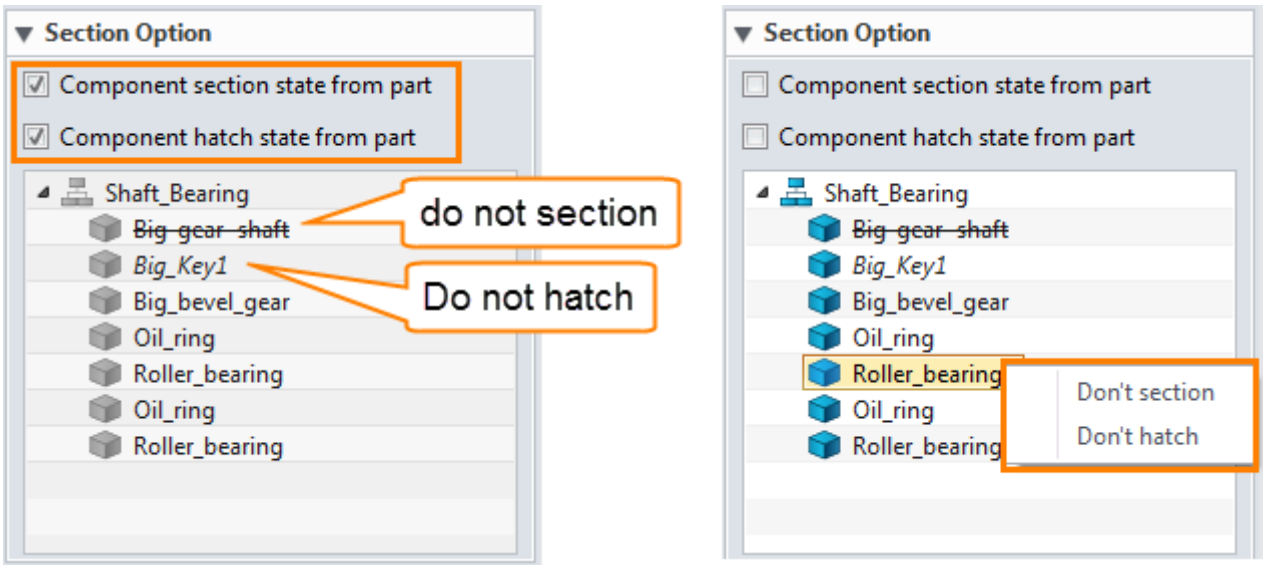


Figure37 Section Option for an Assembly

1.2.6 Aligned Section View

Layout ribbon toolbar->View-> 

Use this command to create a section view in two directions.

STEP 01 Select the base view.

STEP 02 Pick the points to define the section line, including base points and align points, as shown in the image below.

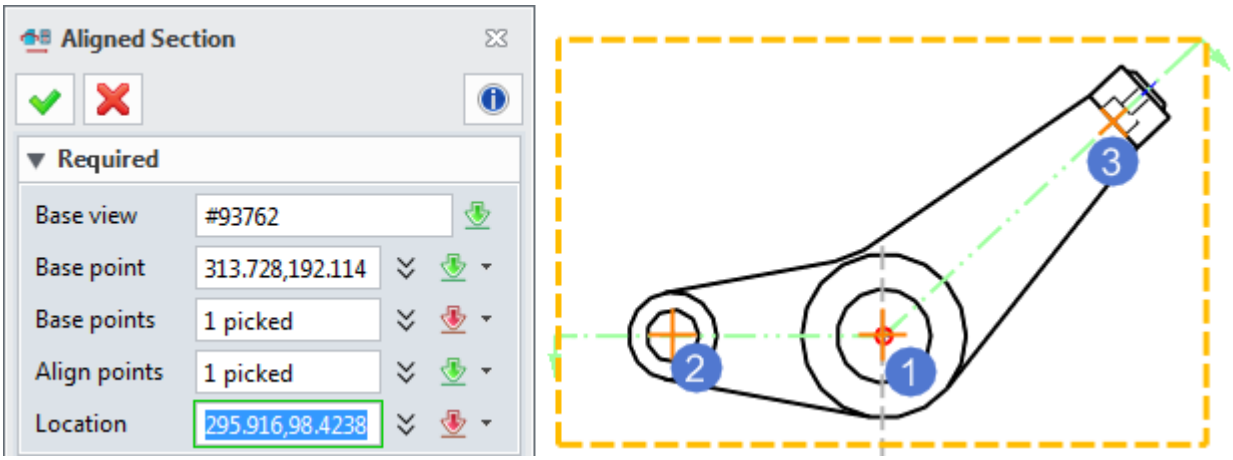


Figure38 Define the Aligned Section Line

STEP 03 Define the view label location and set other parameters if necessary.

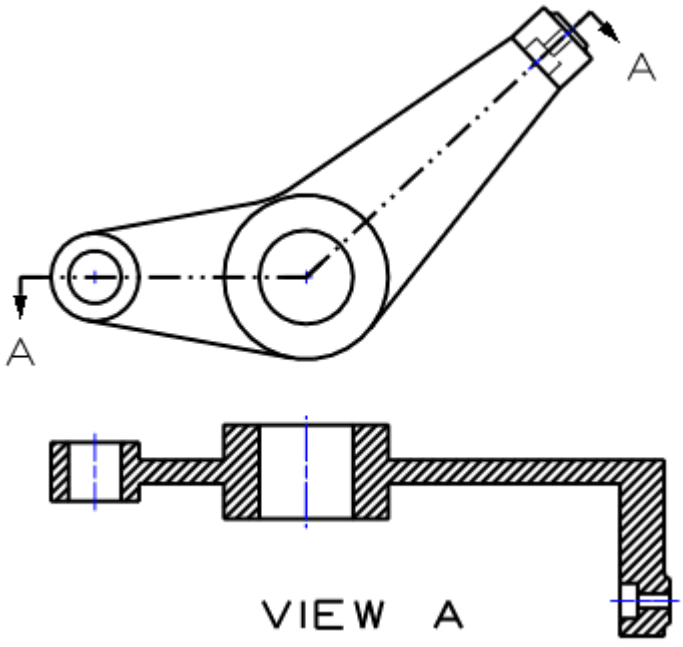


Figure39 Aligned Section View

1.2.7 3D Named/Bent Section View

Layout ribbon toolbar->View->  / 

Use this command to insert a named section created in the part using the Named Section Feature command. The Named Section Feature must be created with a sketch.

"3D Named Section" command is used for the sketch that consists of lines with zero or more bend points at 90 degrees. The function is similar with the stepped section view.

STEP 01 Draw a sketch.

STEP 02 Use **Wireframe->Named Section** command to create a named section feature.

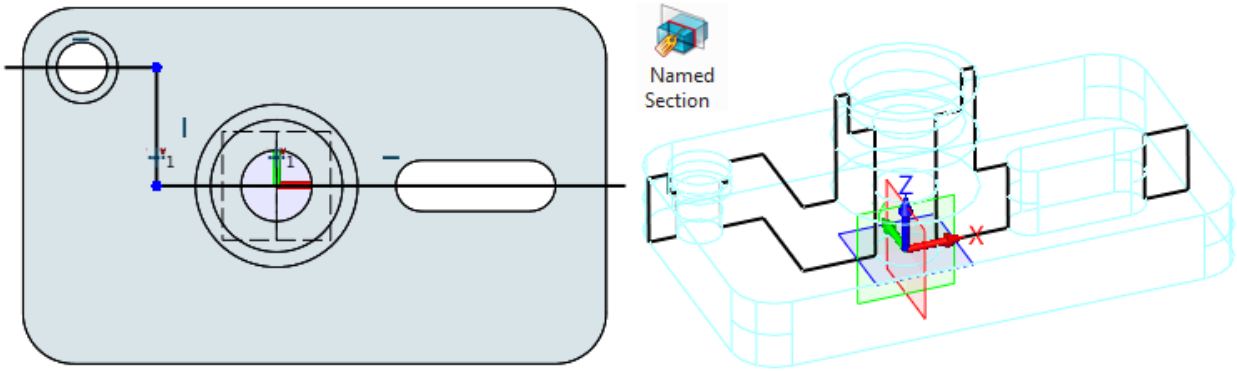


Figure40 Create a Named Section Feature

STEP 03 In 2D sheet level, use 3D Named section view command to create the right section view. When sections have been created in the part, 3D name option is activated. The named section can be selected from the list.

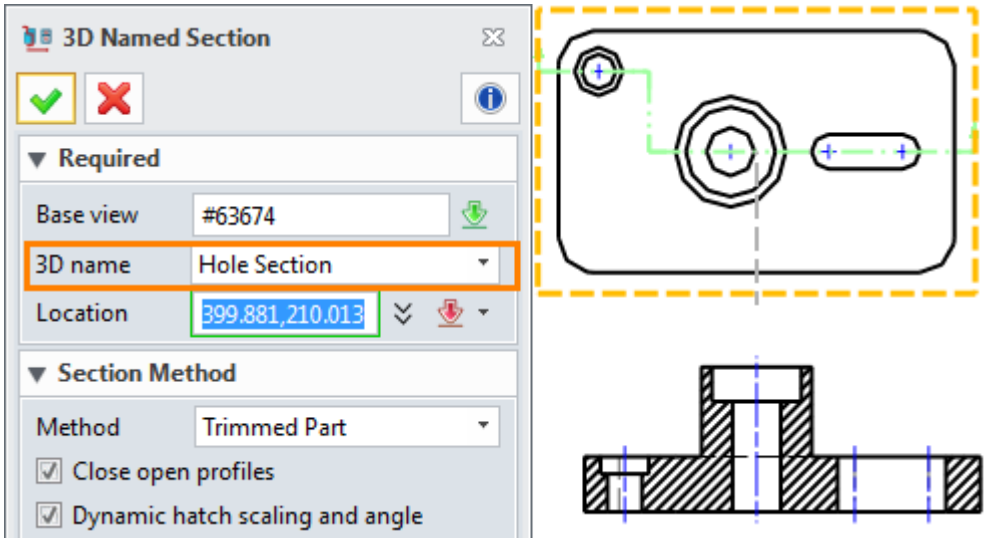


Figure41 3D Named Section View

"Bent Section" command is used for the sketch that consists of lines with bend points at other than 90 degrees. The operations of creating Bent Section view are the same as before.

Notes: *On current version, Bent section view only provides the section curves section method.*

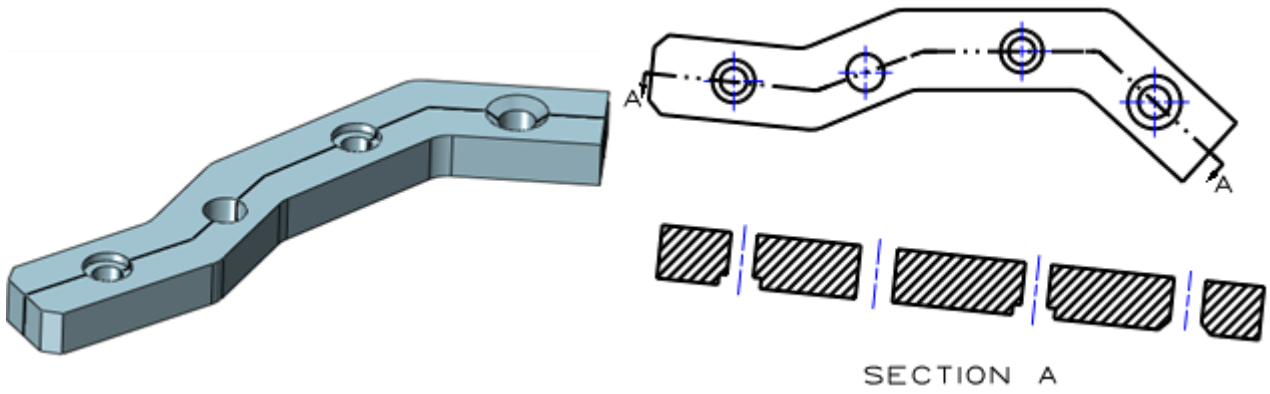


Figure42 Bent Section View

1.2.8 Isometric Section View

Just like the 3D Named Section command, the section line needs to be defined in the part by **Wireframe->Named Section** command. And the sketch line should be open.

STEP 01 Finish the preparation work: draw a sketch and create a named section feature.

STEP 02 Use Isometric Section View command to create the needed view, as shown in the image below.

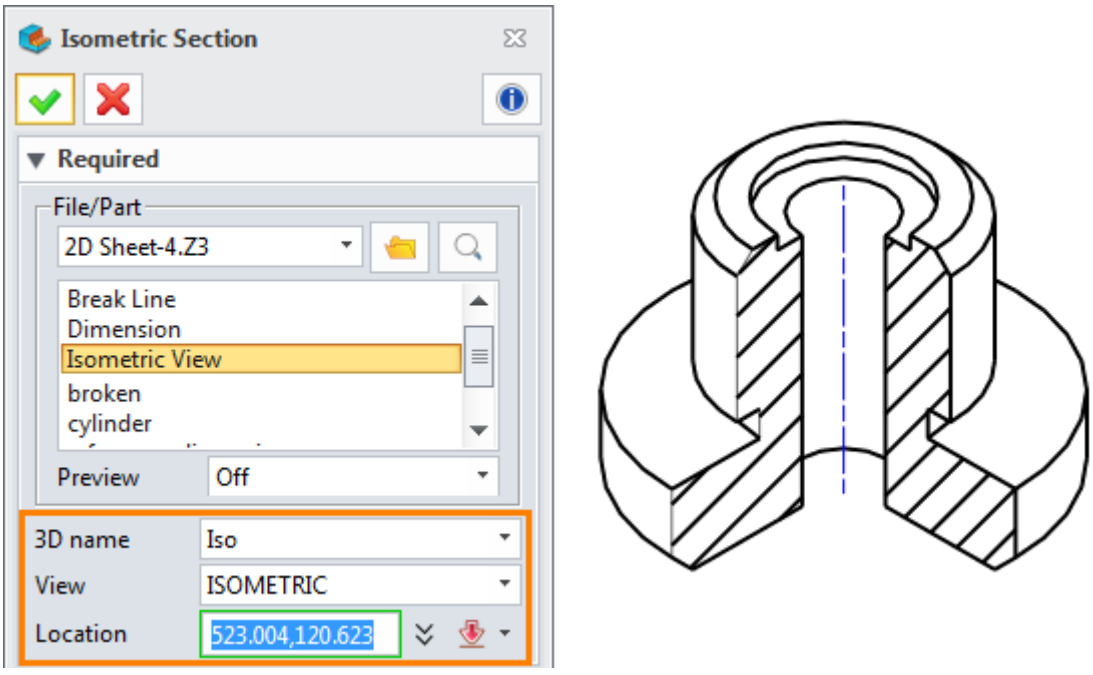


Figure43 Isometric Section View

1.2.9 Detail View

Layout ribbon toolbar->View -> 

Use this command to generate a detail view from other view.

STEP 01 Select the creation method and base view.

STEP 02 Pick the points to define the detail view border.

STEP 03 Define the view label location.

STEP 04 Set the scale factor for detail view and define the view location.

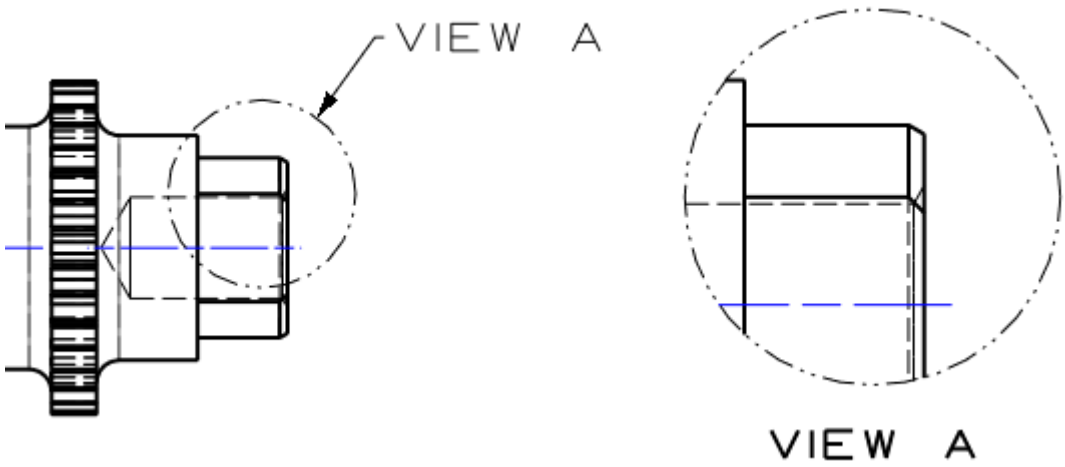
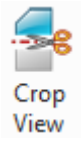


Figure44 Detail View

1.2.10 Crop View

Layout ribbon toolbar->View -> 

Use this command to generate a partial view by trimming a drawing view with a defined boundary.

STEP 01 Select the boundary type.

STEP 02 Select the view to crop (except detail view, define view, break line view).

STEP 03 Define the boundary to finish.

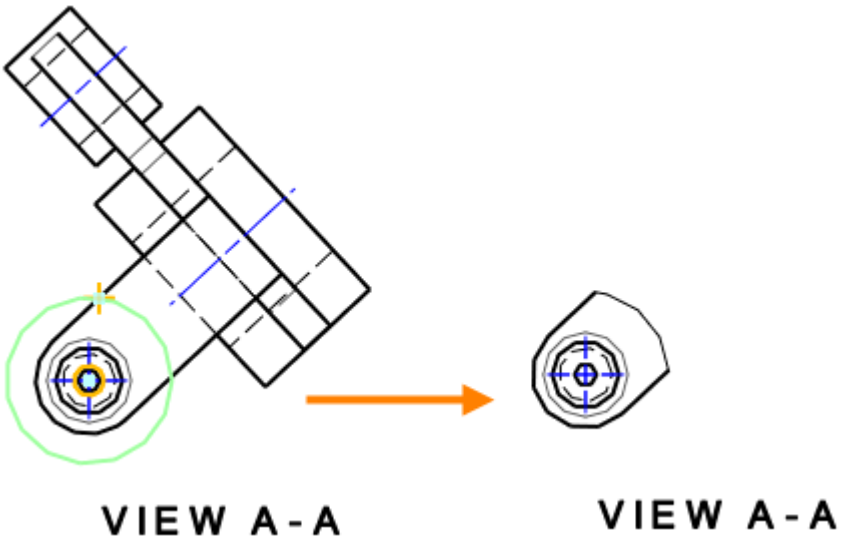


Figure45 Crop View

1.2.11 Break Line View

Layout ribbon toolbar->View -> 

Break line view is suitable for the long-cylinder part. The diemnsion value is unchanged.

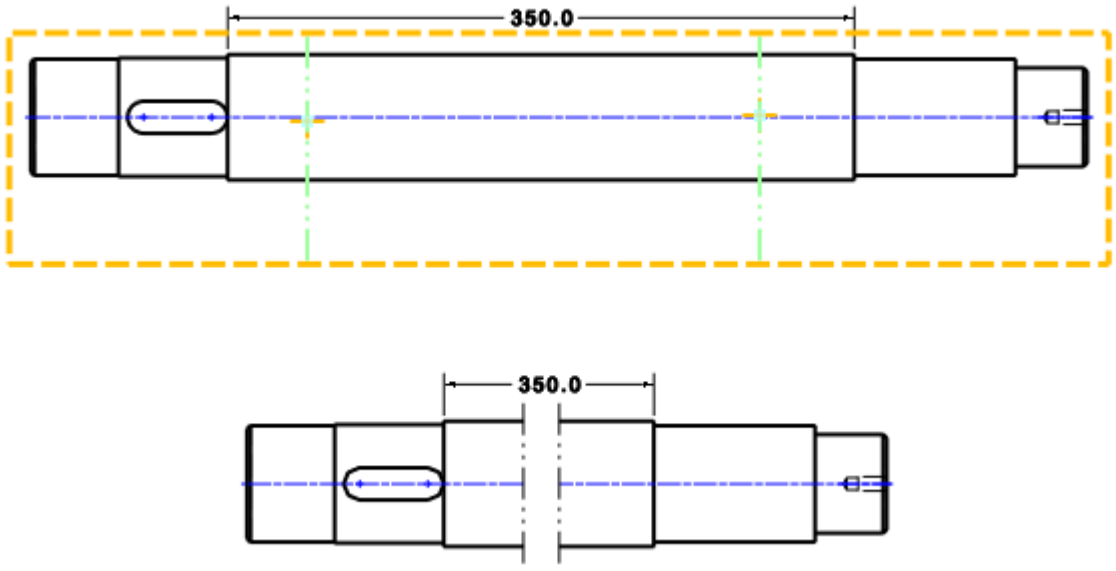
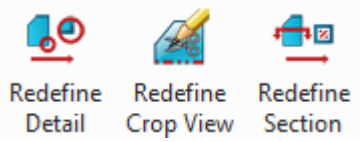


Figure46 Break Line View

1.3 Edit the Views

1.3.1 Redefine the View

After creating the view, refer to the following methods to redefine the view, including the various section view, detail view and crop view.

Method1：Layout ribbon toolbar->Edit View -> 

STEP 01 Choose the suitable redefine command from the ribbon tab.

STEP 02 Select the view to redefine.

Method2: Right Menu

STEP 01 Right click the view, then select Redefine command to redefine the view .

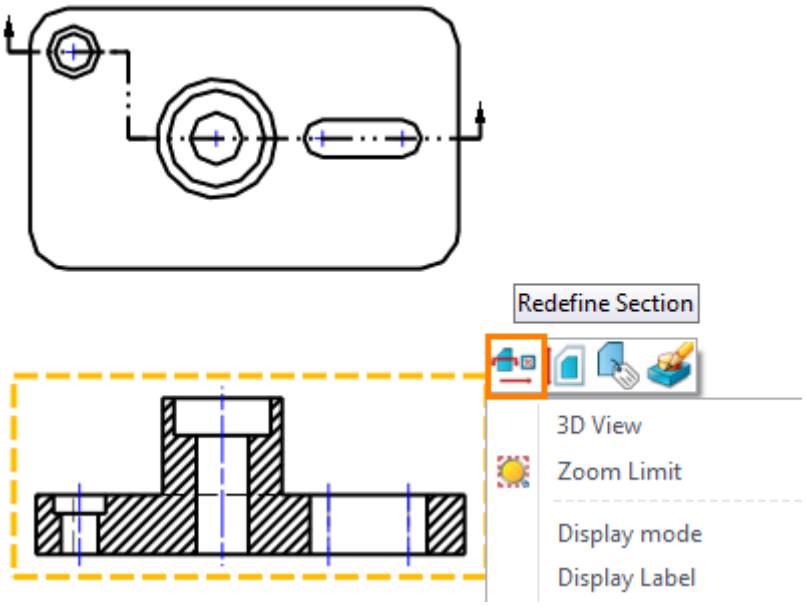


Figure47 Redefine the View—Method2

1.3.2 Edit the Section Line

If the section view is created by “Full Section” command, the section line can be edited after the view creation.

Insert the step points

STEP 01 Right click the section line, then select “Insert step” option.

STEP 02 Point a point to insert the step.

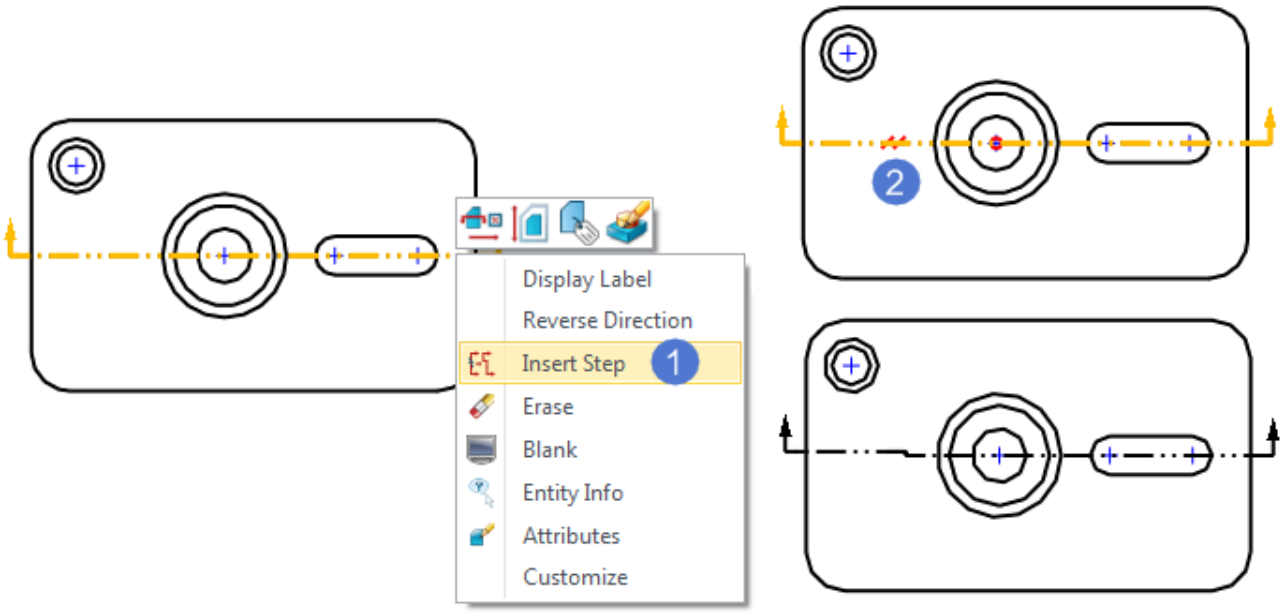


Figure48 Insert the Step Point

STEP 03 Drag and drop the insert point to the suitable position to get a new section line. The projection view is automatically updated.

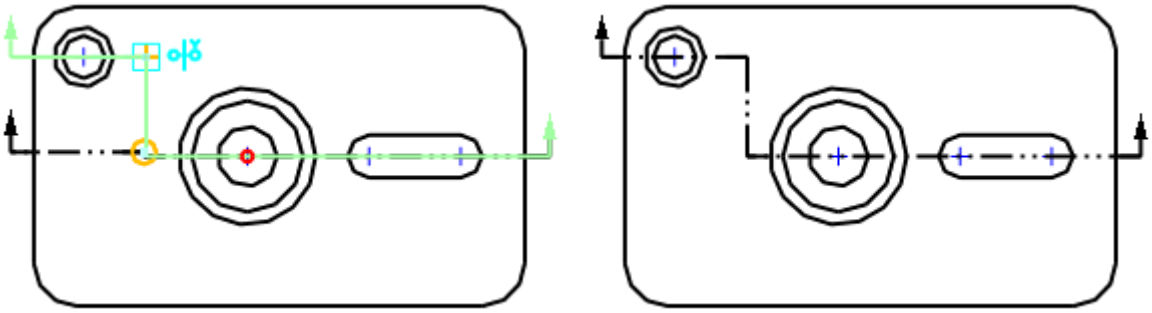


Figure49 Drag and drop the Insert Point

Reverse the direction of section line

Right click the section line, then select “Reverse Direction” option to reverse the direction of section line. The projection view is auto-updated.

Remove the step point

Directly drag and drop the step line to align another line, then two step lines are merged into one.

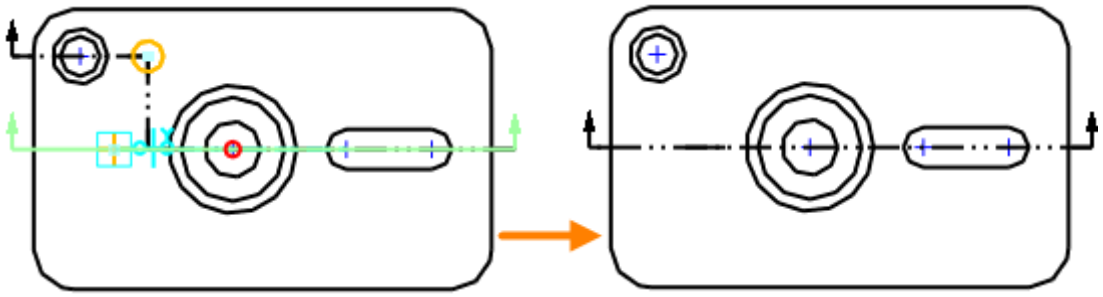


Figure50 Remove the Step Point

1.3.3 Edit the View Attribute

You could quickly modify the view attribute with different methods after creating the view.

Method1: Right Menu

Directly right-click the view or right-click the view name in sheet manager, then select Attribute command to edit the view attribute.

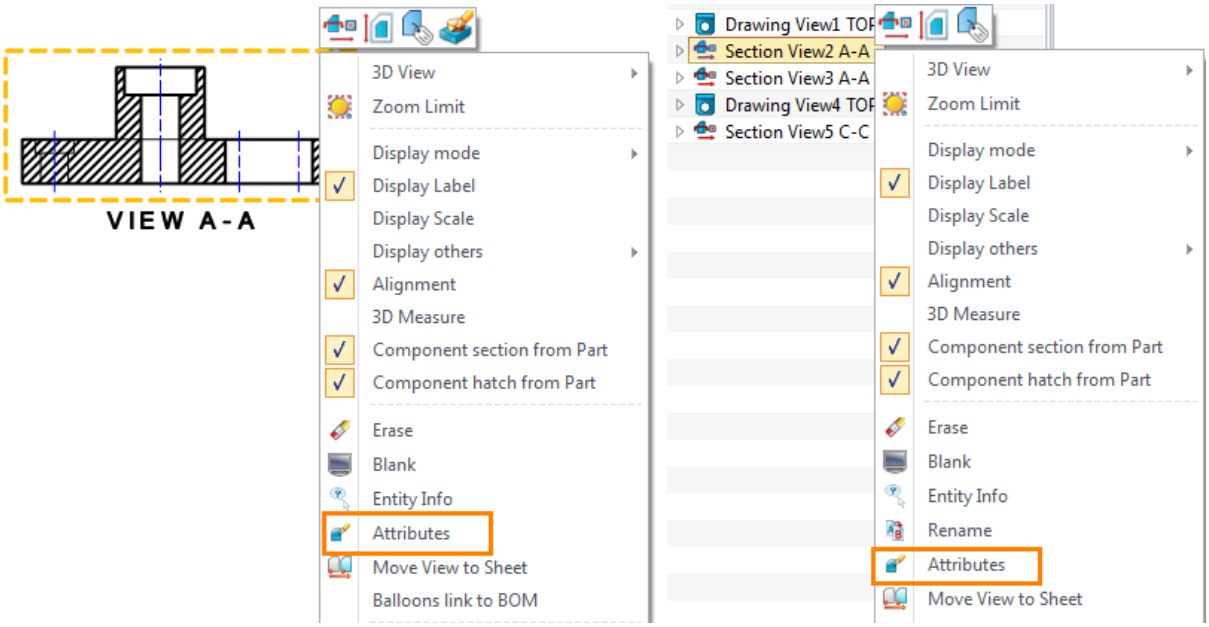
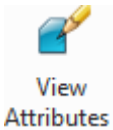


Figure51 Edit the View Attribute-Method 1

Method2：Layout ribbon toolbar->Edit View -> 

STEP 01 Choose the View Attribute command.

STEP 02 Select the view and middle-click to confirm.

STEP 03 Reset the view attribute in the form.

1.3.4 Edit the View Label and Scale

Use the above methods, you could edit the view label and view scales in view attribute form, as shown in Figure 52.

How to quickly show/hide the view label and scale?

As shown in Figure 48, check/uncheck “Display Label” or “Display Scale” to quickly show/hide the view label and view scale.

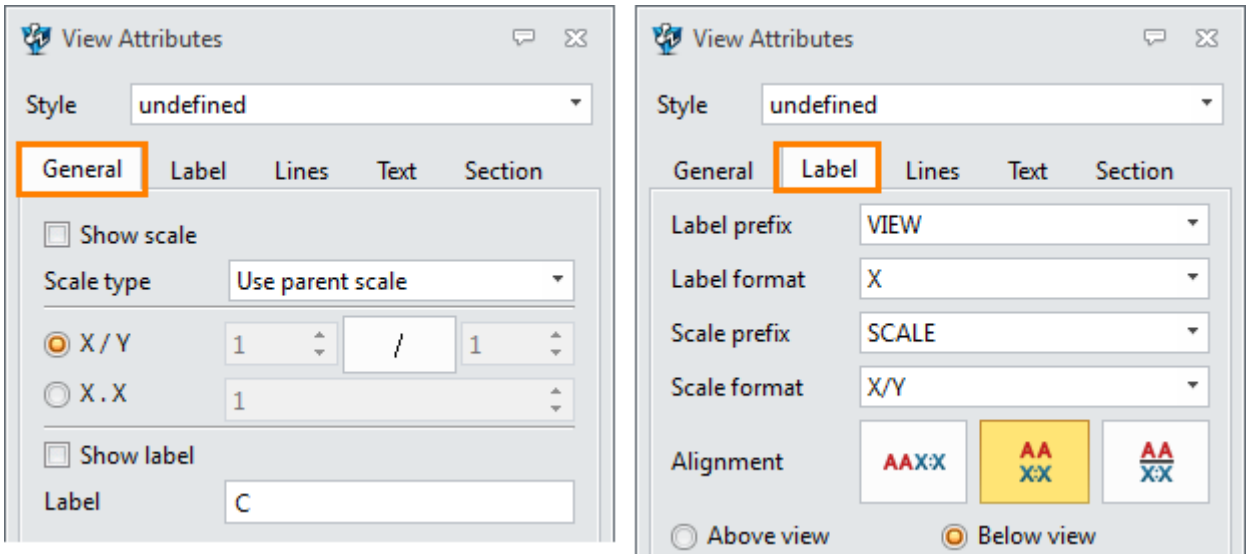


Figure52 Edit the View Label and Scale

1.3.5 Move the View

Not any view can be moved to any position. Such as, the default projection view only can be moved along the projection direction. For a projection view/ section view, follow these steps to move it to any position.

STEP 01 Right click the view, then uncheck “Alignment” option.

Notes: *Alignment option controls the view projection association.*

STEP 02 Drag and drop the view to any position.

STEP 03 If you want to regain the original position relationship, check “Alignment” option and move the view again.

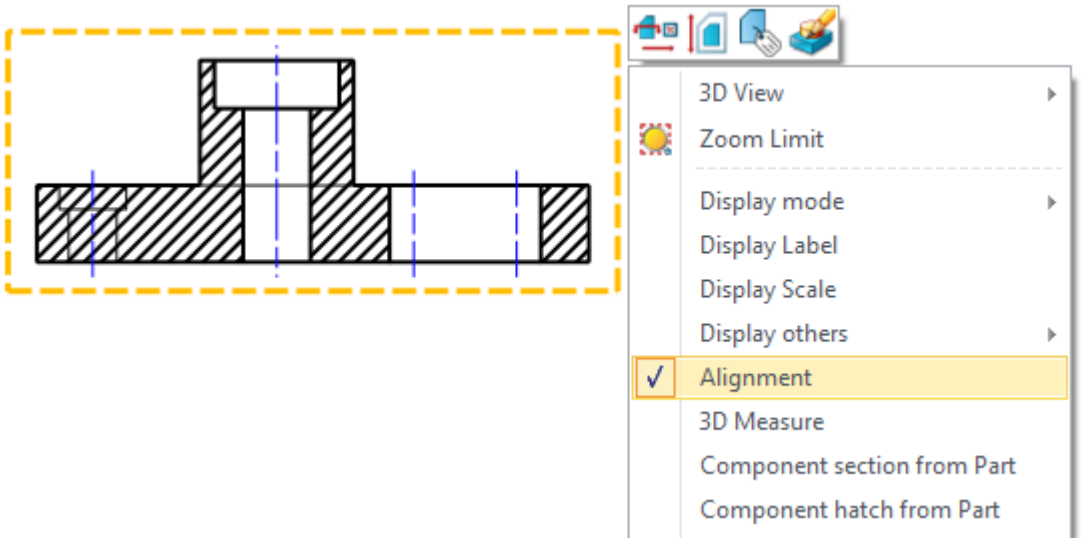
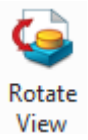


Figure53 Uncheck the Alingment to Move

1.3.6 Rotate the Base View

Layout ribbon toolbar->Edit View -> 

Use this command to rotate view in place（only for base view）.

Select the base view, then adjust the parameters in pop-up edit toolbar.

 : The view can be rotated by right mouse button when this icon is activated.

 : The view can be moved by draging the middle mouse button when this icon is activated.

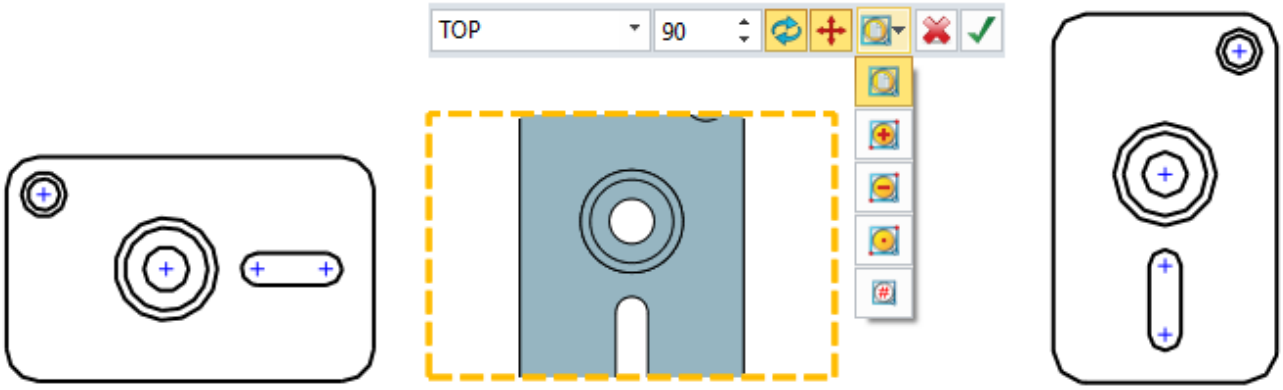


Figure54 Rotate the Base View

1.4 Dimension

1.4.1 Set the Dimension Attribute

It's a good design habit that setting the dimension attribute before creating the dimensions.

STEP 01 Open the style manager to edit the dimension style.

Right click the 2D sheet object in sheet manager or directly select the Dimension Attribute command from Tools ribbon toolbar.

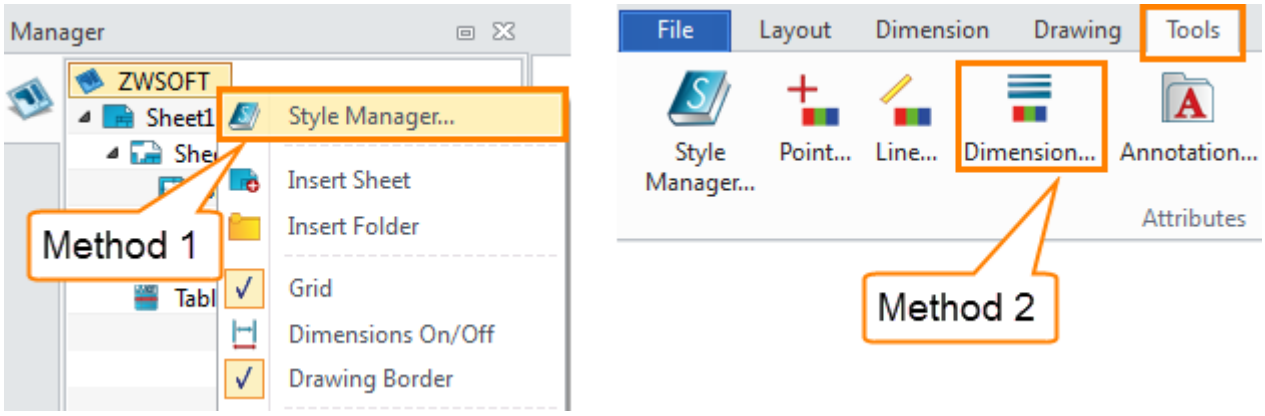


Figure55 Dimension Style Manager

STEP 02 Double click one dimension style to activate, such as linear dimension style. Or create a new dimension style and set as the activated style.

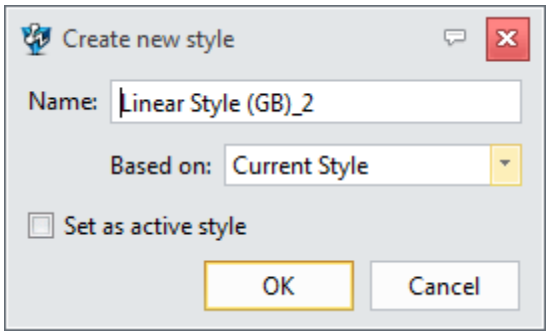


Figure56 Create a New Style

STEP 03 Set the needed parameters and click "Save" button to save the modification.

STEP 04 Click "Apply" button to apply the activated style.

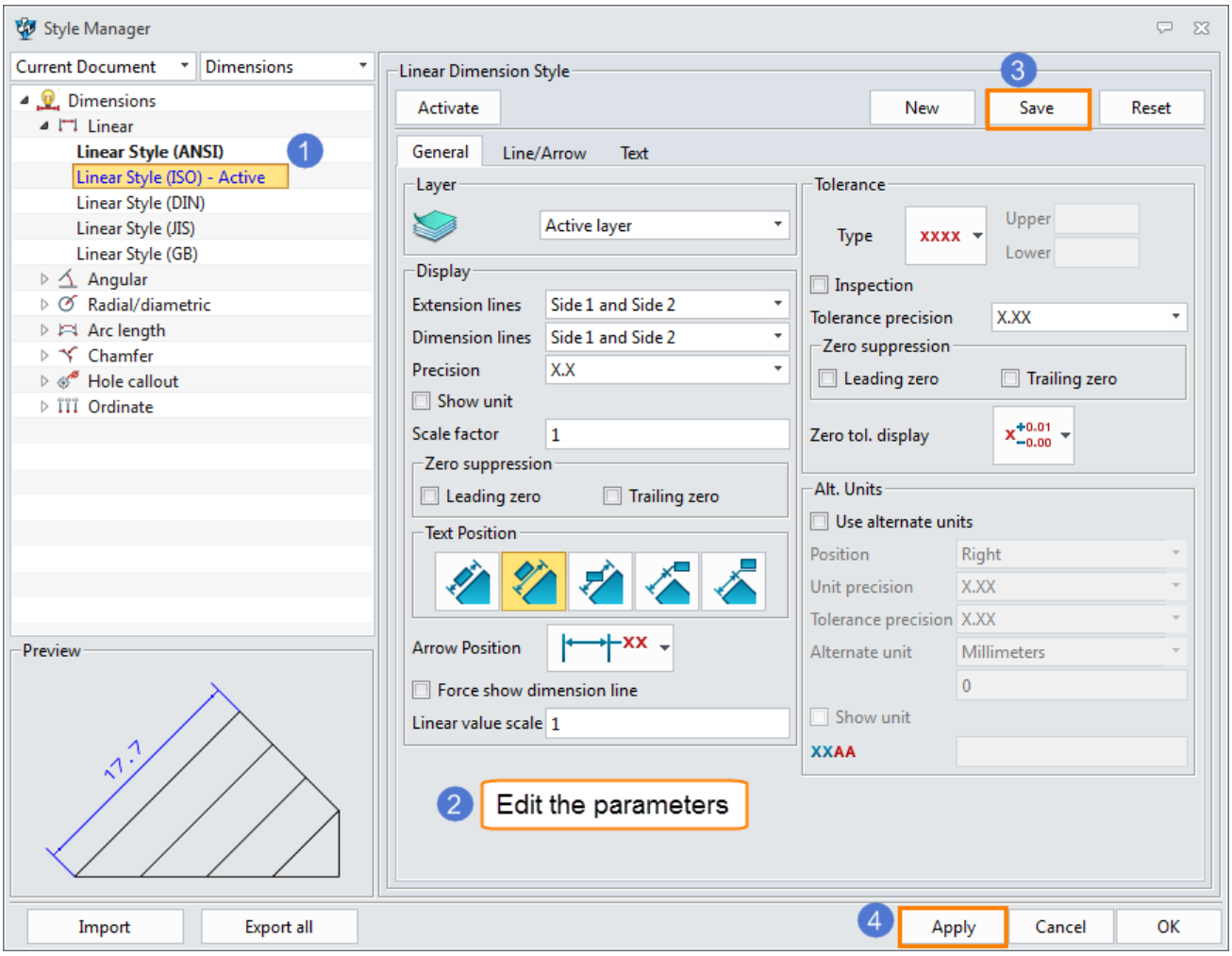


Figure57 Define the Diemnsion Style

1.4.2 Add Basic Dimensions

Use Quick Dimension Tool or other dimension tools to create the required dimensions.

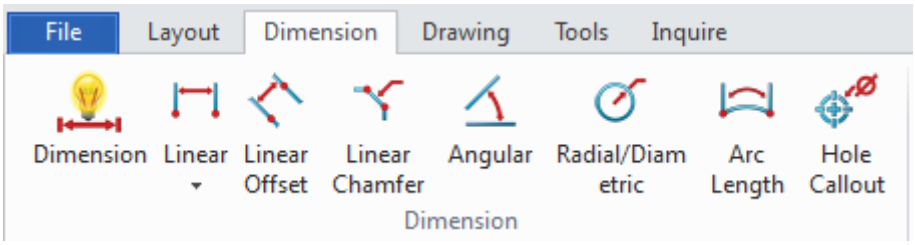


Figure58 Dimension Tools

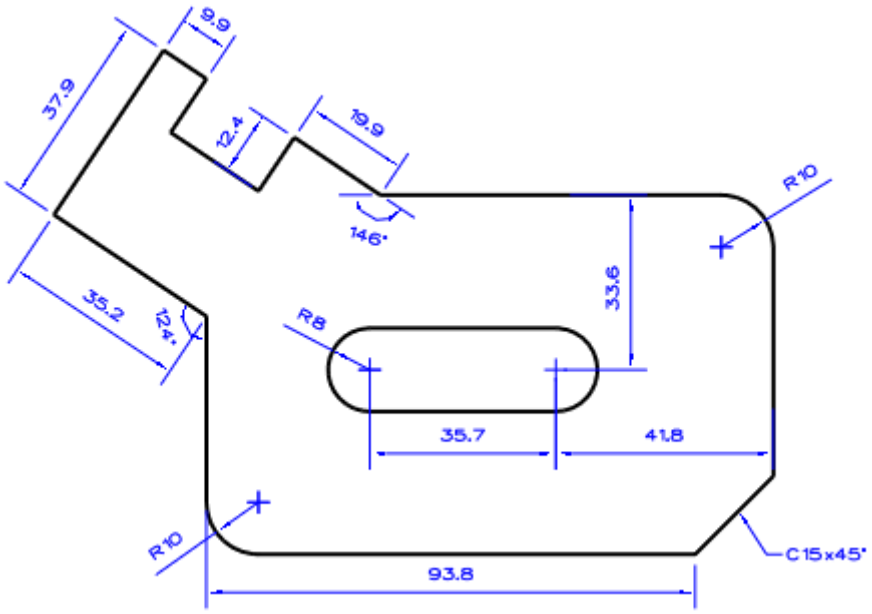


Figure59 View with Dimensions

1.4.3 Linear Dimension Mode

Baseline Use this command to create a 2D linear baseline dimension groups.

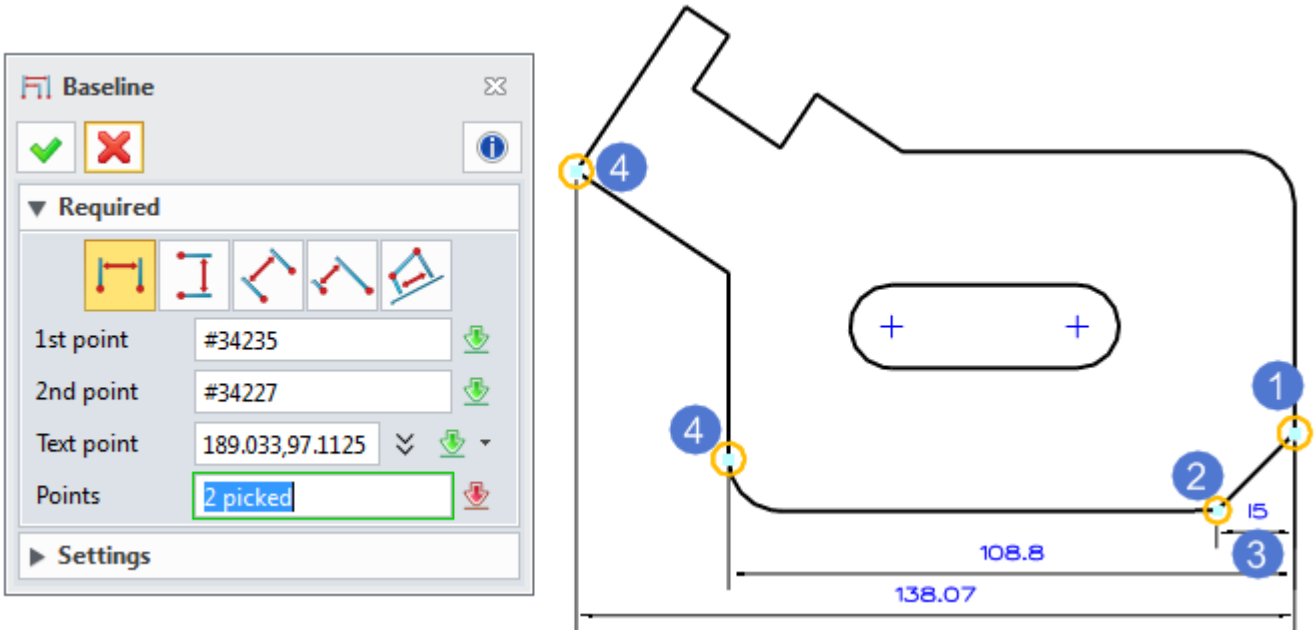


Figure60 Linear Baseline Dimension

Notes: *In these images, the multiple 3or 4 markers indicate the additional dimension points.*

Continuous Use this command to create a 2D linear continuous dimension groups.

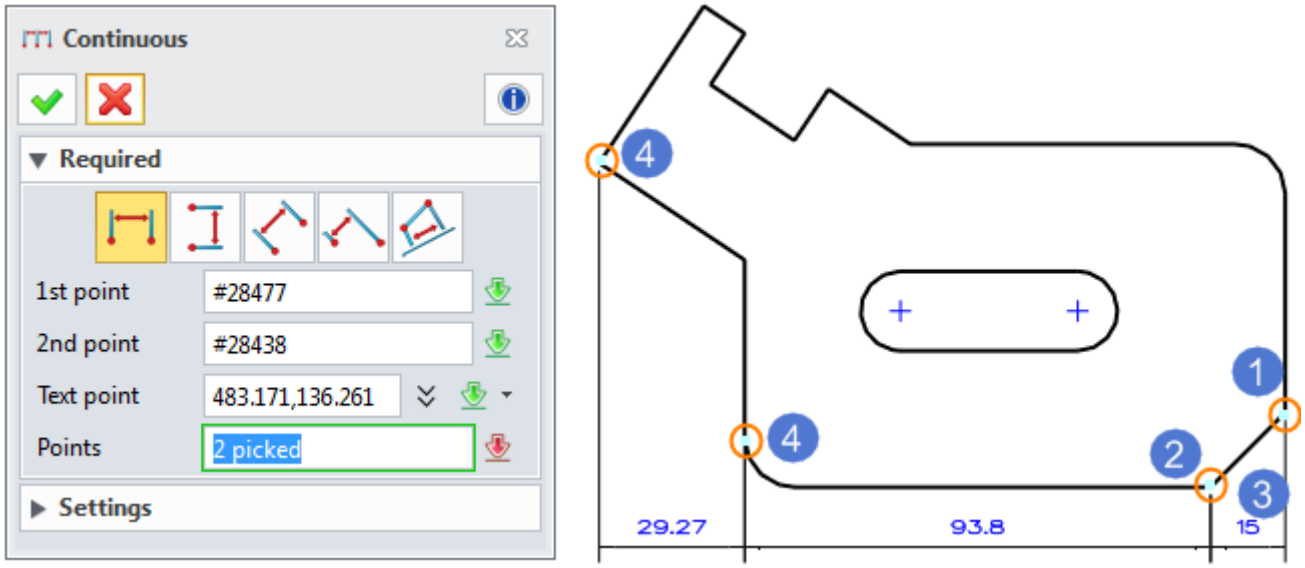


Figure61 Linear Continuous Dimensions

Ordinate Use this command to create a 2D linear ordinate dimension groups.

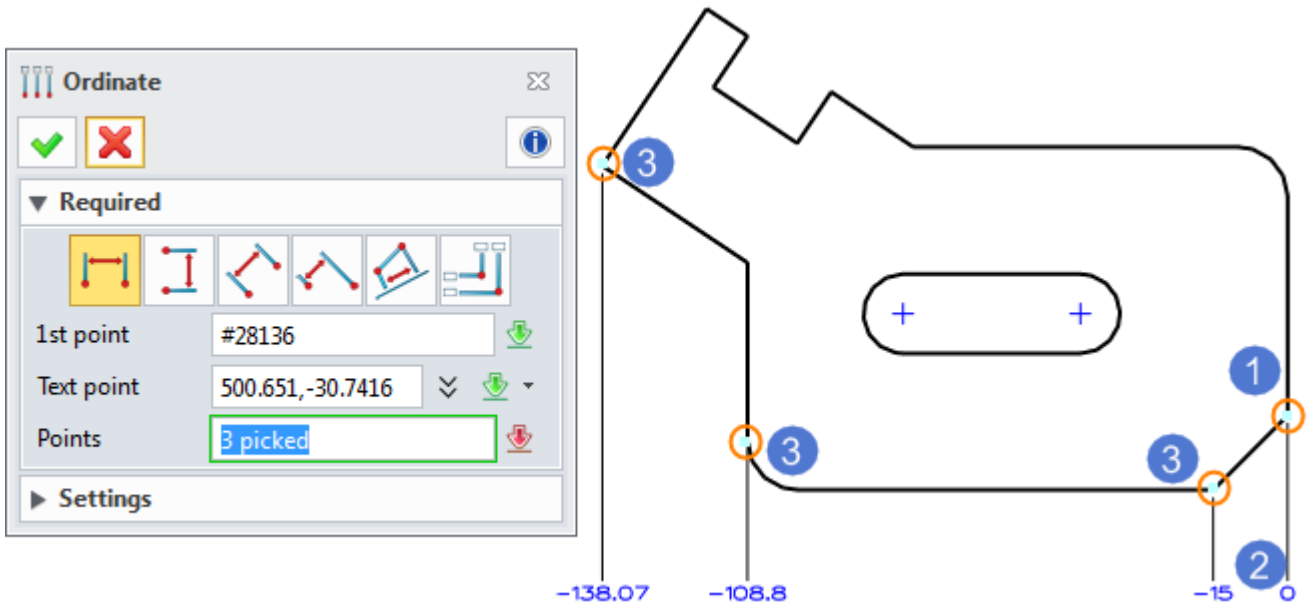


Figure62 Linear Ordinate Dimensions

How to edit the linear dimension?

Dimension ribbon toolbar->Edit Dimension -> Add to Group

Select a dimension in the group and specify a point to insert a new dimension into this group.

Dimension ribbon toolbar->Edit Dimension -> Remove from Group

Select a dimension from a dimension group to remove.

Dimension ribbon toolbar->Edit Dimension -> Explode Group

Use this command to explode a dimension group. Then each dimension becomes an individual object.

1.4.4 Add the Tolerance

Method1: Dimension ribbon toolbar->Edit Dimension -> Modify Tolerance

Method2: Right click the dimension, then select "Modify Tolerance" command to edit.

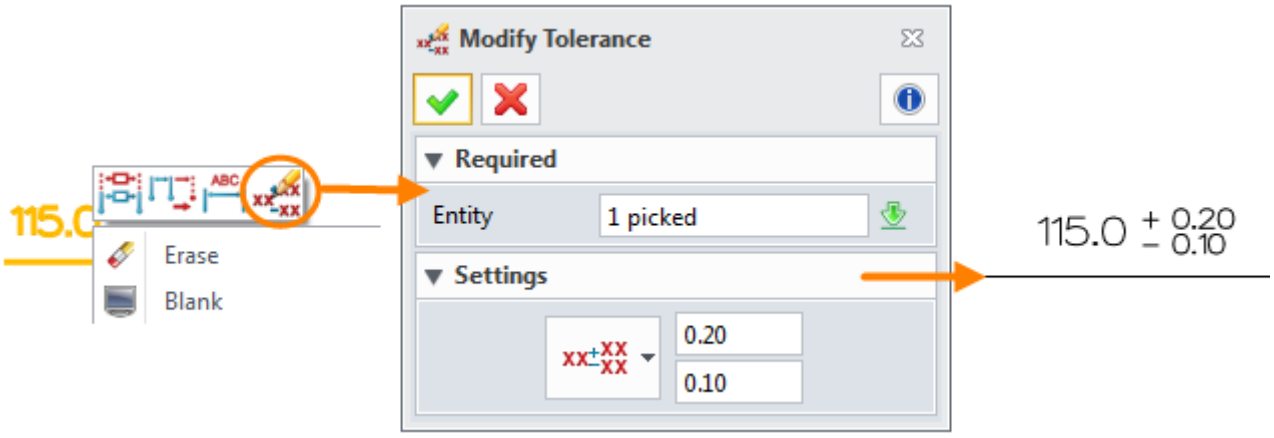


Figure63 Unequal Tolerance-Method 1

Method3: Modify the tolerance with Quick DimTool.

STEP 01 Right click the blank area in the ribbon toolbar, ToolBars->DimTool.

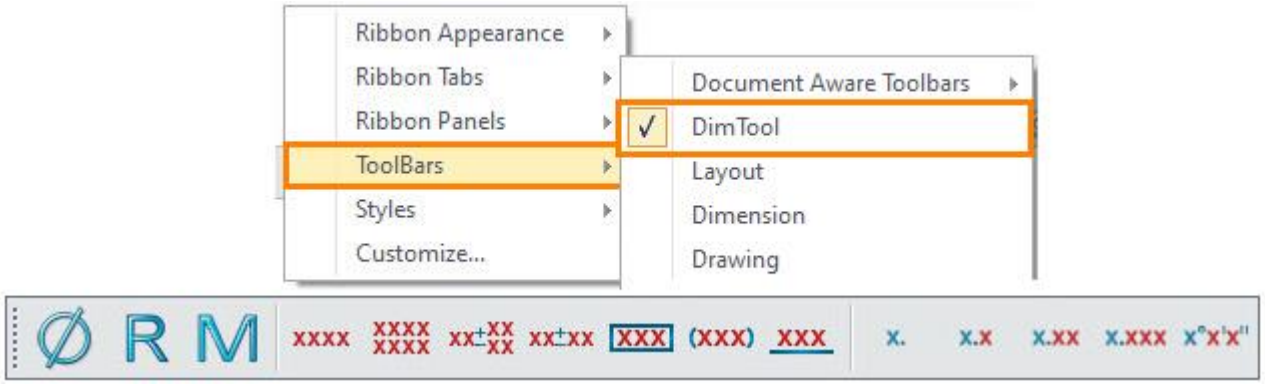


Figure64 Quick Dimension Tool

STEP 02 Select the dimension and then choose the tolerance type from the toolbar.

Figure65 Tolerance Command

STEP 03 Modify the tolerance according to the requirements.

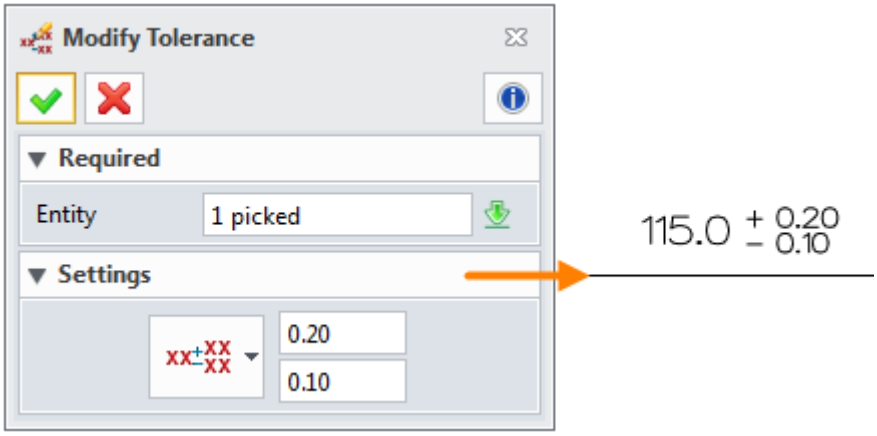


Figure66 Modify Tolerance-Method 2

With Quick Dimension Tool to add dimension symbol

With quick dimension tools, we could quickly add some dimension symbols, such as: Ø / R, directly set dimension tolerance or precision.

How to add tolerance zone?

Refer to the image below, select the "Tolerance zone" type, then click "Inquire tolerance" button to inquire the tolerance zone. Select the suitable tolerance zone, then click OK.

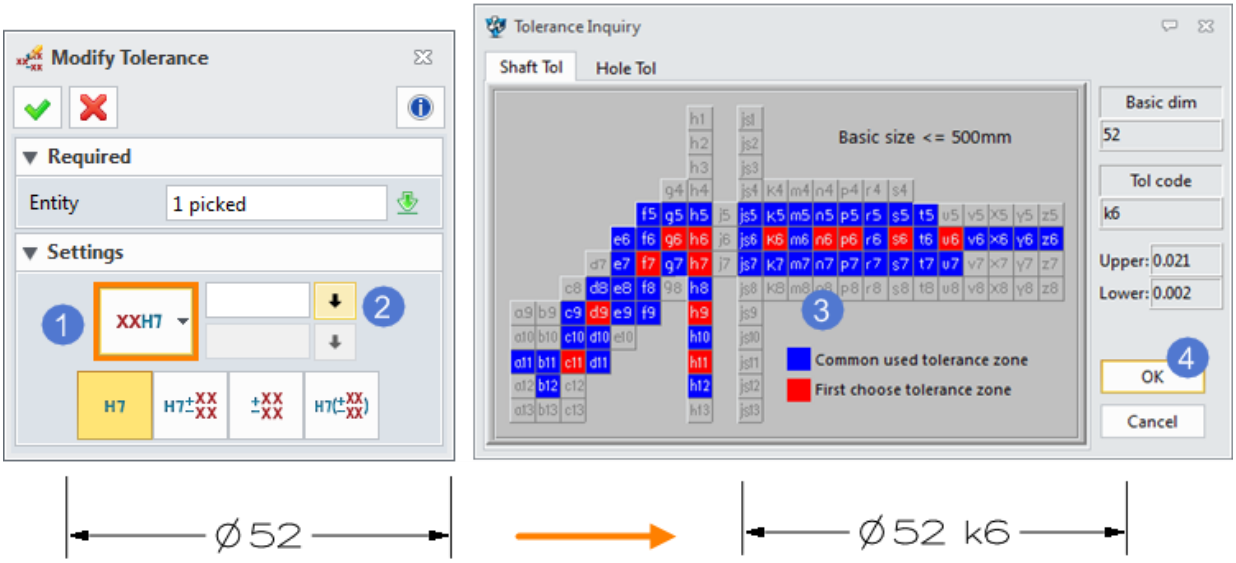
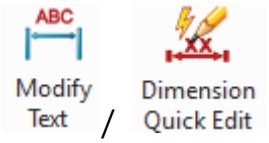


Figure67 Define Tolerance Zone

1.4.5 Edit Dimension Text

Method1: Dimension ribbon toolbar->Edit Dimension -> 

Method2: Right-click the dimension, then select "Modify Text" command. At the same time, the dimension editor form will automatically pop up, as shown in Figure 69.

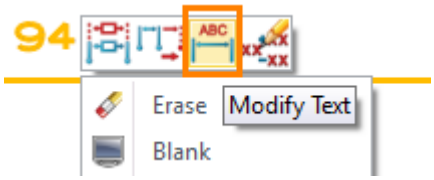


Figure68 Modify Dimension Text

This form allows the insertion of special characters, symbols and variables into the text string.

Notes: *When using "Dimension Quick Edit" command, you need to manually open this editor form.*

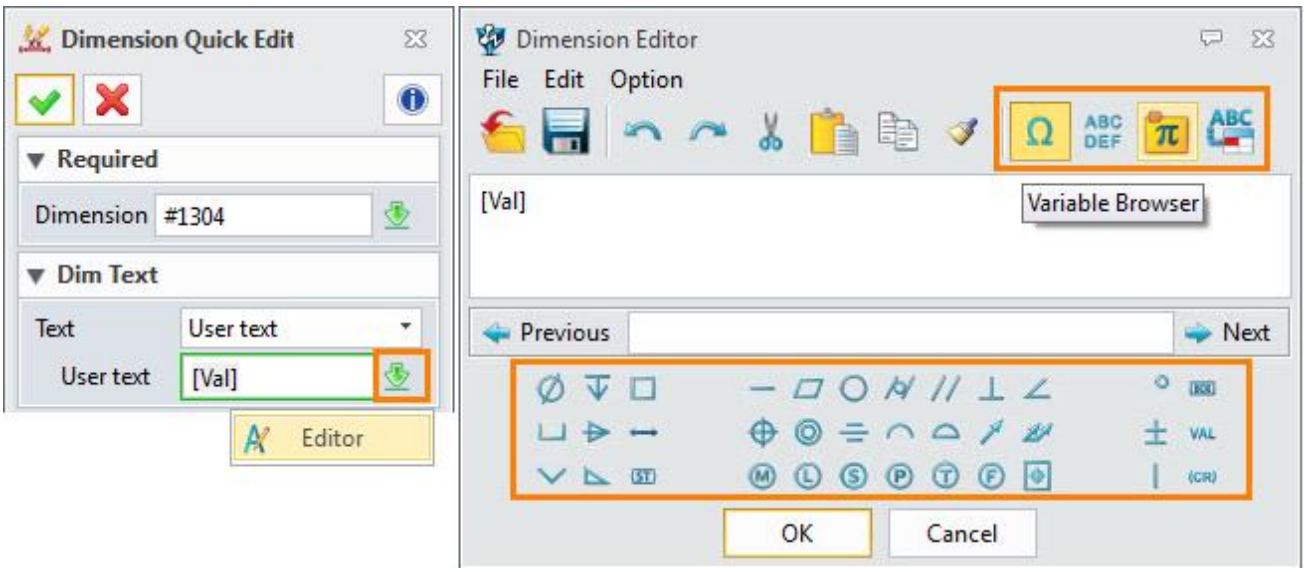


Figure69 Dimension Editor

1.4.6 Edit Dimension Attribute

Method1: Dimension ribbon toolbar->Edit Dimension -> 

Use this command to quickly edit partial dimension attributes, including dimension tolerance, presion and appended text.

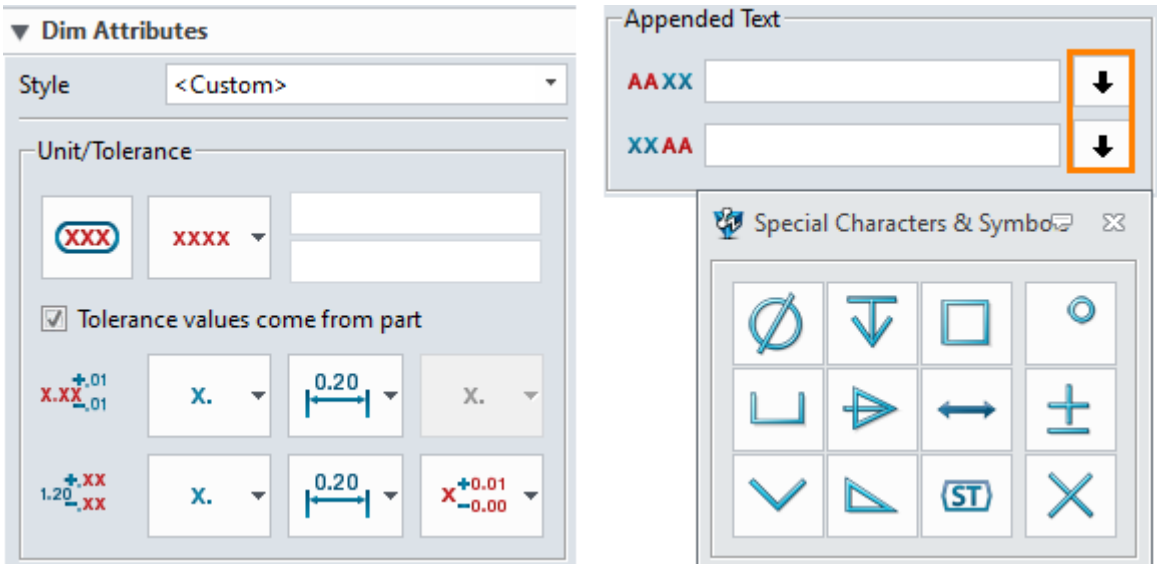


Figure70 Dimension Quick Edit

Method2: Right-click the dimension, then select “Attribute” command to edit all dimensions attributes, including tolerance, dimension line/arrow,text ,font and so on.

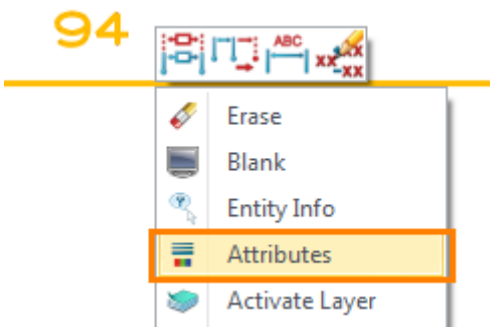


Figure71 Edit Dimension Attribute

1.4.7 Modify Text Point

Use this command to modify the text point location of a dimension or change the mode of text placement (automatic / manual placement). The dimension text and extension lines will adapt accordingly.

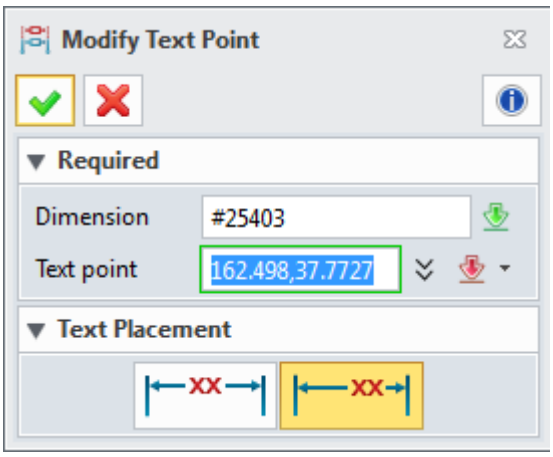


Figure72 Modify Text Point

Notes: *Drag and drop the dimension to directly change the text point based on defined placement mode.*

1.4.8 Modify Dimension Points

Use this command to redefine a dimension. E.g. a dimension becomes invalid, you could use it to redefine the dimension objects to make it valid.

Method1: Dimension ribbon toolbar->Edit Dimension -> 

Method2: Right-click the dimension, then select “Modify Dimension Points” command to edit.

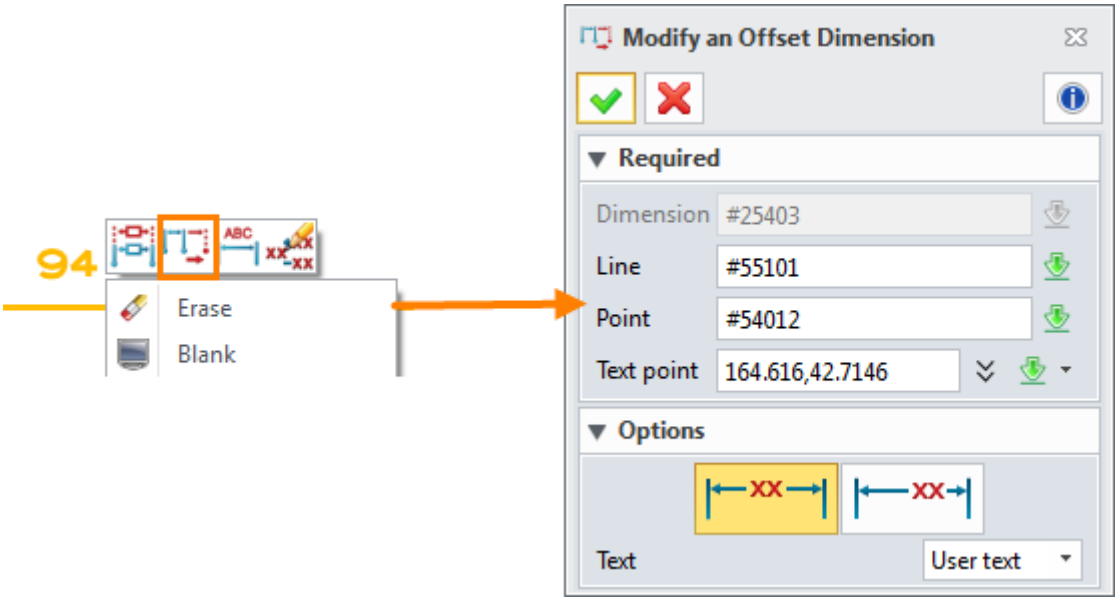


Figure73 Modify Dimension Points

With Quick Dimension Tool to add dimension symbol

It is highly recommended to use the quick method to modify dimension points or text position.

Move the cursor to the dimension/click the dimension, then drag and drop the control point or text to a new position.

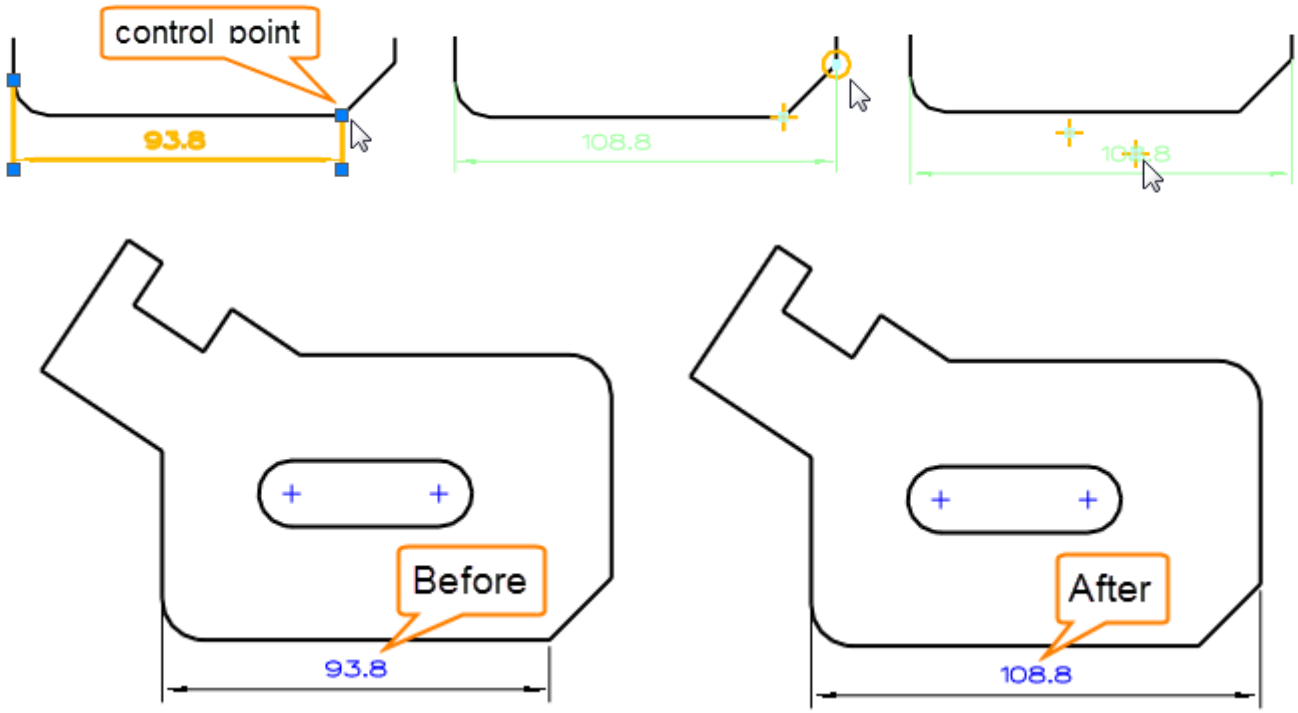


Figure74 Modification of Dimension Points

1.4.9 Create / Modify Part Dimension

When creating 2D view, turn on the option of "Show dimension from part", the view with part dimensions is created.

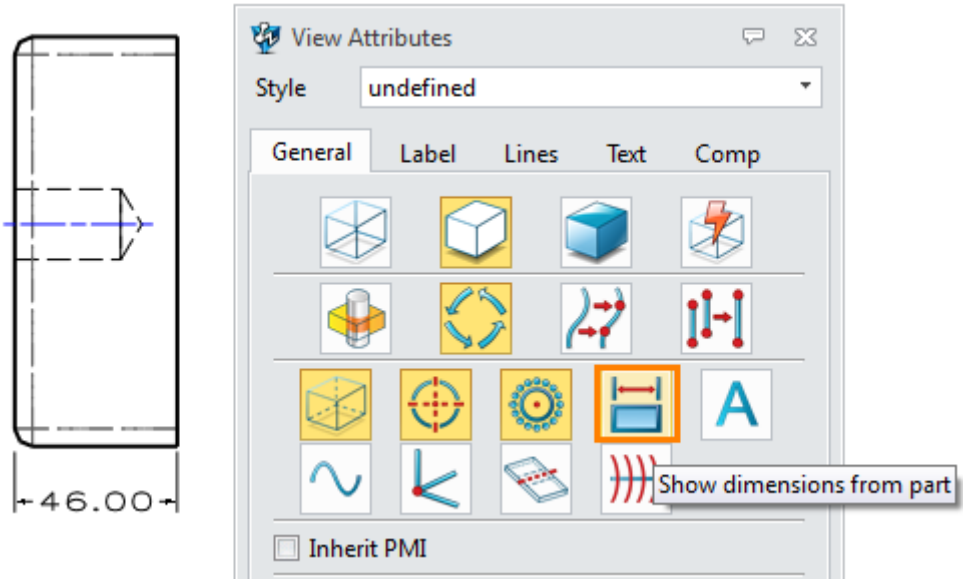


Figure75 The View with Part Dimension

Modify a part level dimension in a 2D view, then this new dimension value will automatically drive the part geometry. At the same time, the other views will automatically update.

Method1: Dimension ribbon toolbar->Edit Dimension -> 

Method2: Right-click the dimension, then select "Modify Part Dimension" command.

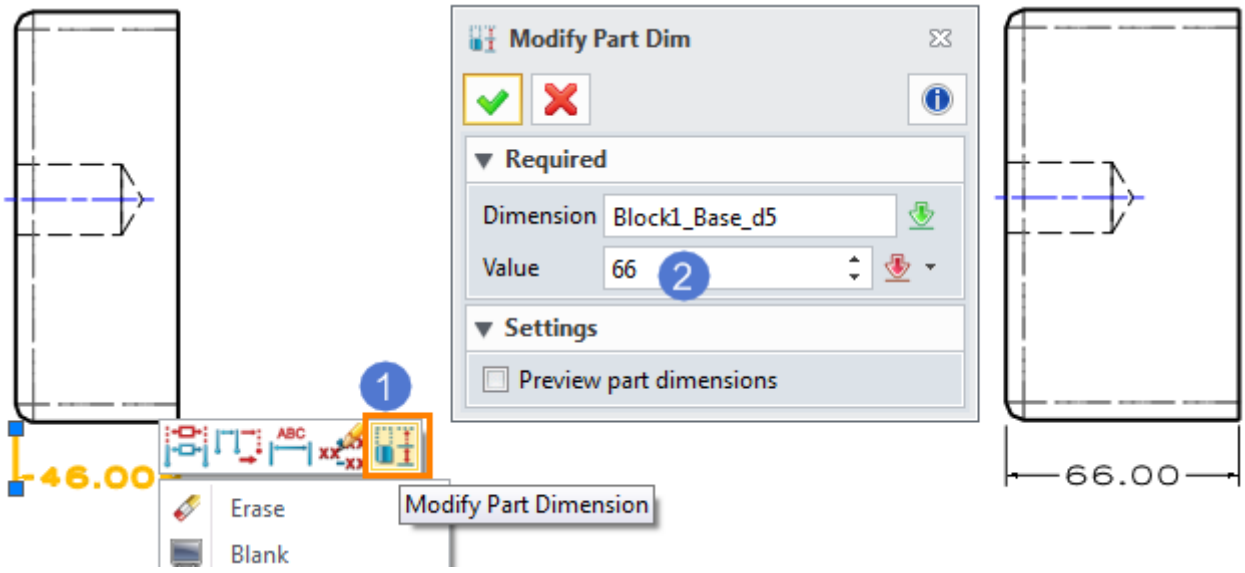
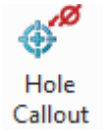


Figure76 Modify Part Dimension

1.4.10 Hole Dimension

Dimension ribbon toolbar-> Dimension -> 

Use this command to create one or more hole callout dimensions. First select the layout view and then select the holes to add callouts.

STEP 01 Select the layout view and pick the holes (circle or arc).

STEP 02 Check the needed hole parameters.

STEP 03 Click "OK" button to finish. The result is shown in the image below.

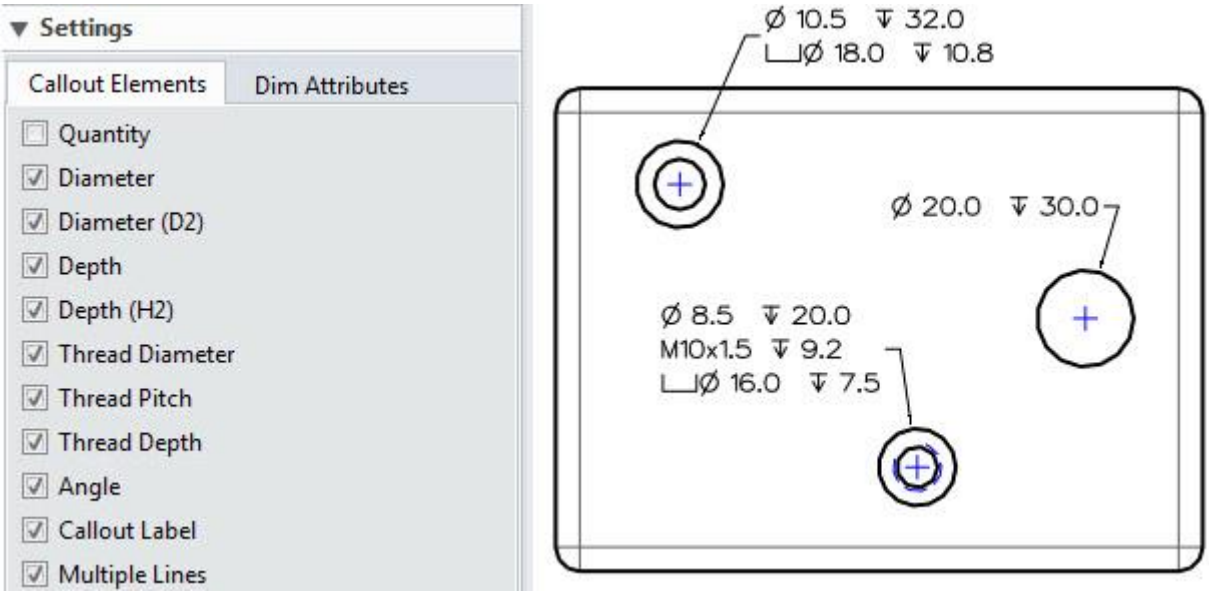


Figure77 Hole Callout Dimension

Notes: *If you want to add the hole dimension on the projection view of hole, please use Label command to manually create the hole dimension.*

1.4.11 Align the Dimension

Dimension ribbon toolbar-> Edit Dimension -> 

Use this command to align the position of a dimension with another dimension.

Select two dimensions. Then the first dimension is aligned to the second one.

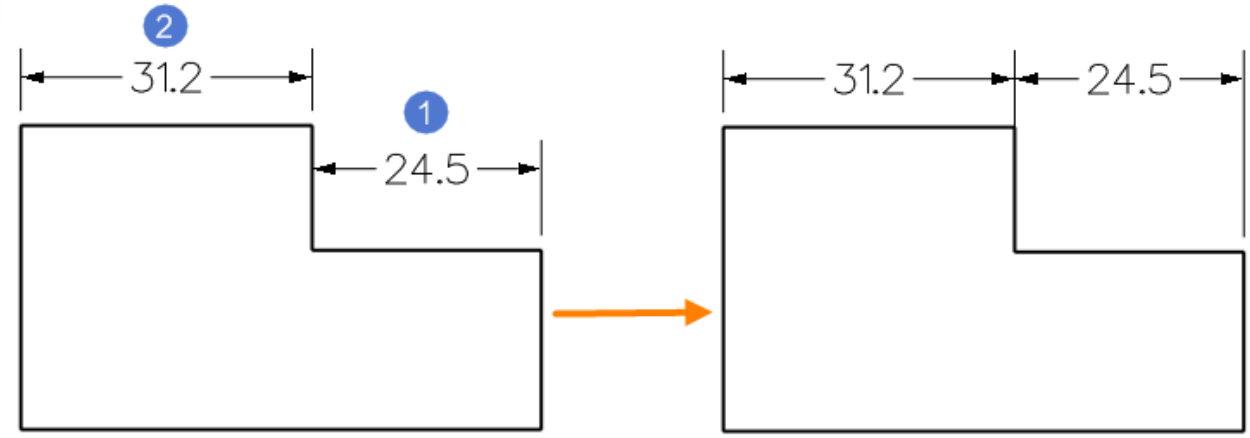


Figure78 Align the Dimension

1.5 Annotation and Symbol

1.5.1 Center Mark/Line

Dimension ribbon toolbar-> Edit Dimension -> 

Use this command to create a center mark at an arc or circle, as shown in Figure 79.

Dimension ribbon toolbar-> Dimension -> 

Use this command to create a centerline mark between lines, arcs or circles, as shown in Figure 80. Also you can manually draw a centerline by picking two points.

Use individual extension

Check the option of "Use individual extension" to create the center mark /line with individual extension. Then you could drag and drop the control point of the extension line to adjust it.

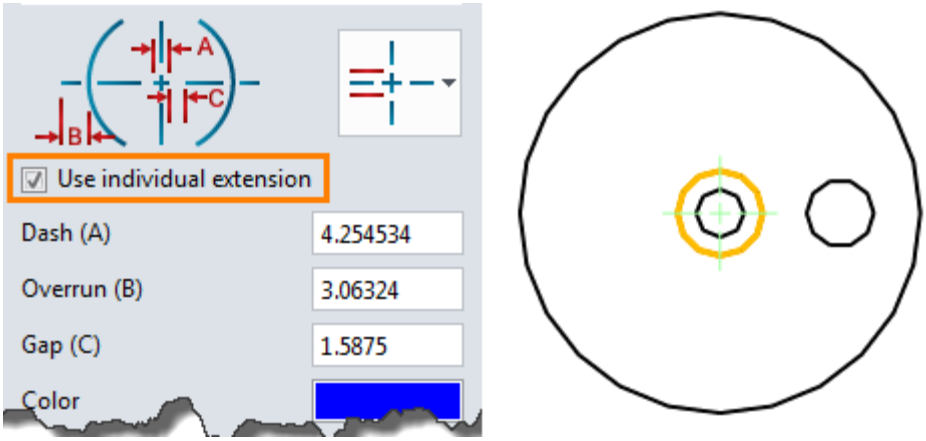


Figure79 Center Mark

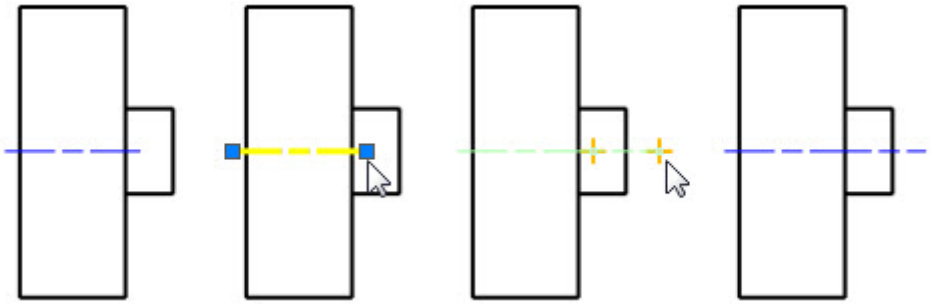


Figure80 Drag and Drop the Extension Line

Dimension ribbon toolbar-> Dimension -> 

Use this command to place a centerline through a circular pattern such as bolt holes.

STEP 01 Select the center points of all holes.

STEP 02 Set the center mark type and define other parameters if required.

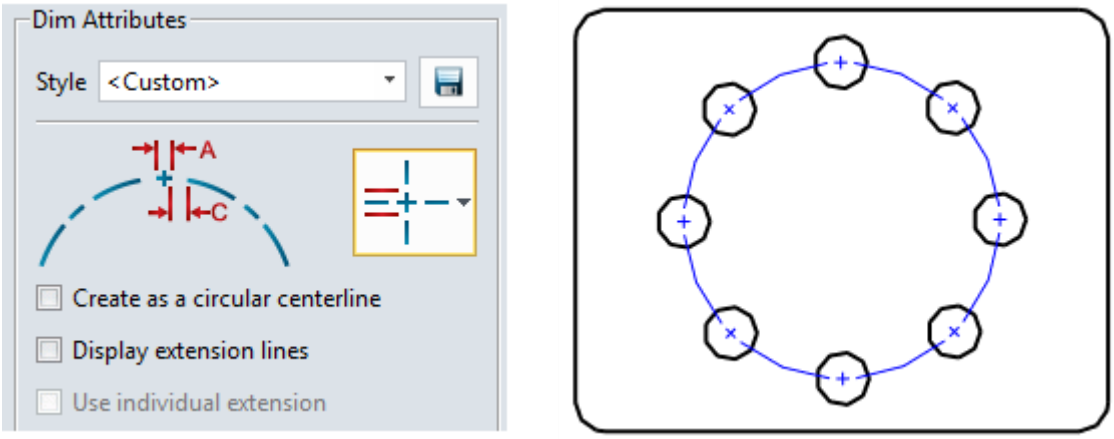


Figure81 Center Mark Circle

Create as a circular centerline

Check this option to show the circle instead of the center mark.

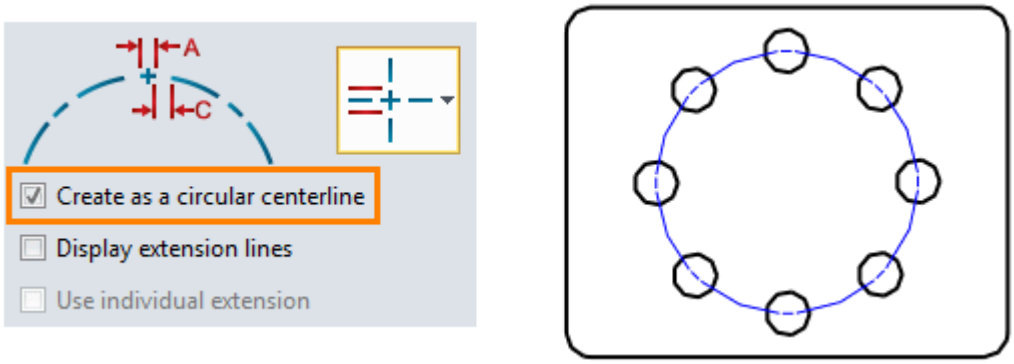


Figure82 Center Mark Circle-Circular Centerline

1.5.2 Datum

Dimension ribbon toolbar-> Annotation -> 

Use this command to create a datum feature, which can be used in Feature Control Symbol.

STEP 01 Set the label text or use the default value.

STEP 02 Select the target entity.

STEP 03 Pick a point to locate the text.

STEP 04 Set the display parameters according to requirements.

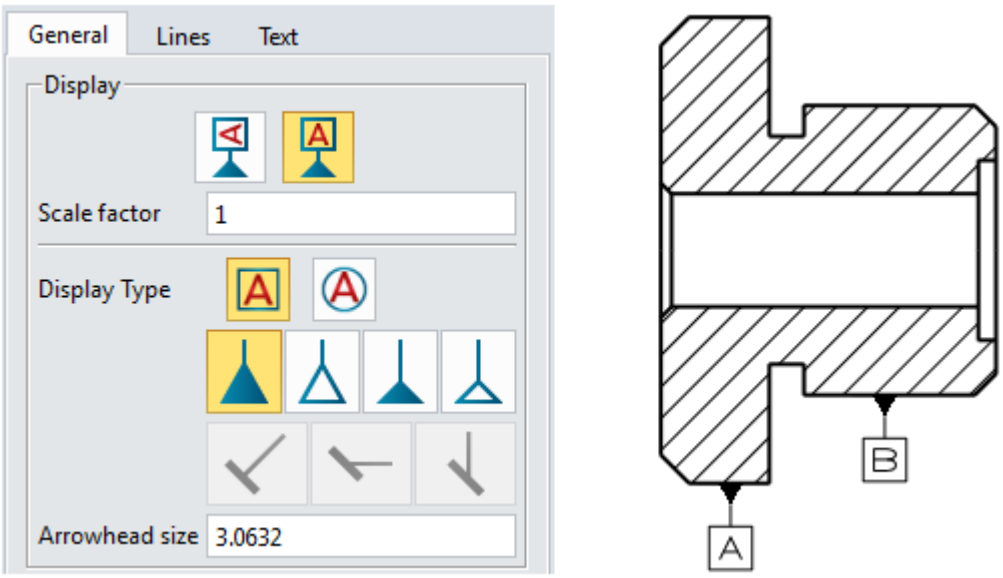
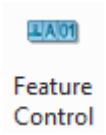


Figure83 Datum Feature

1.5.3 Feature Control Symbol

Dimension ribbon toolbar-> Annotation -> 

Use this command to create a feature control symbol.

STEP 01 Define the FCS text by the editor.

STEP 02 Pick points to locate the FCS text..

STEP 03 Define other parameters, such as additional leader points, display type.

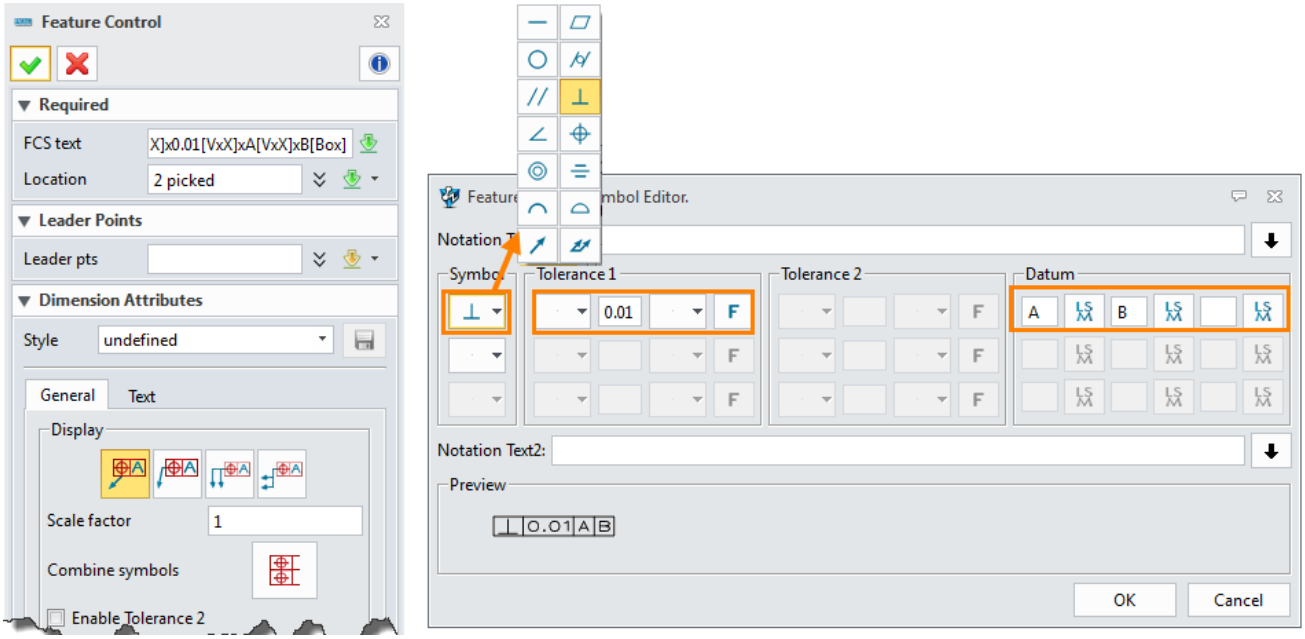


Figure84 Feature Control Symbol Defination

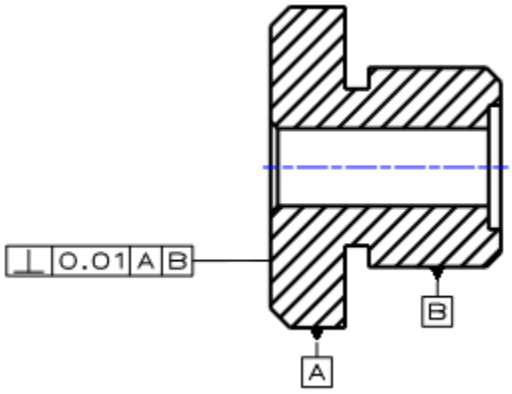


Figure85 Feature Control Symbol

1.5.4 Surface Finish Symbol

Surface finish represents the machining quality of part surface. So in 2D view, it is required to select the edge to define the surface finish symbol.

STEP 01 Pick a point to locate the symbol.

STEP 02 Define the orientation angle or lead point if needed.

STEP 03 Select the machining type and set the value.

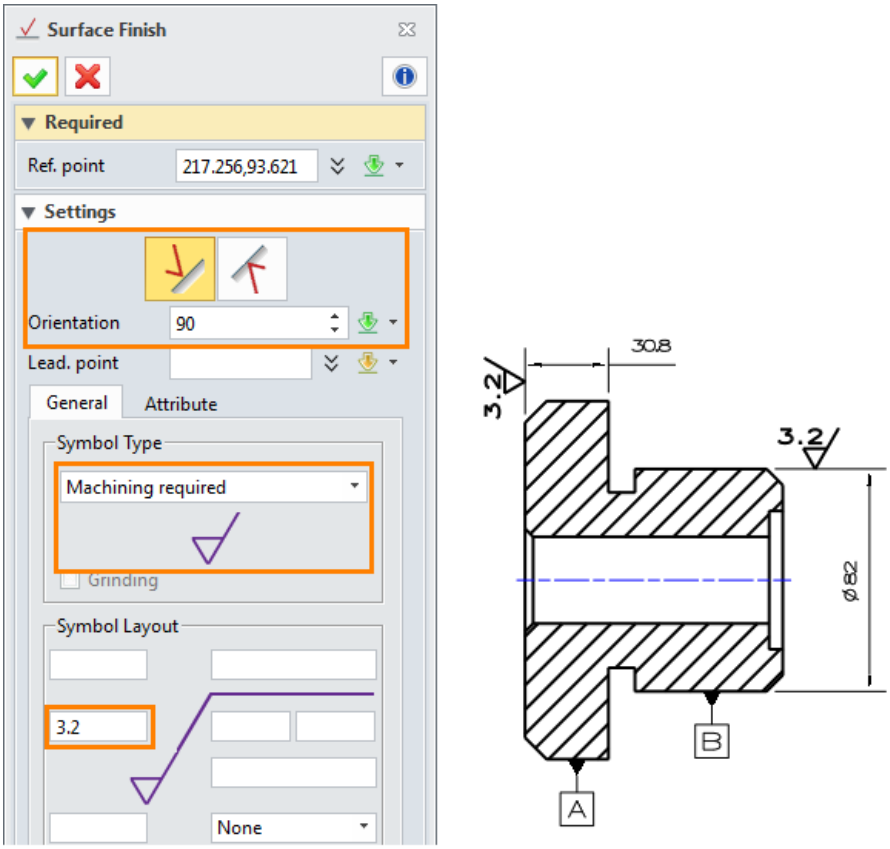


Figure86 Surface Finish Symbol

1.5.5 Label/Text

Dimension ribbon toolbar-> Annotation -> 

Use this command to manually create a label. Take a counter-bore hole as example.

STEP 01 Pick one or multi points to locate the label.

STEP 02 With editor to input the dimension text, as shown in the image below.

Click , add hole dimension D1 value(32); then click , add hole depth H1value (95); line feed; click and , add hole dimension D2 value (52), then click , add hole depth H2 value (5).

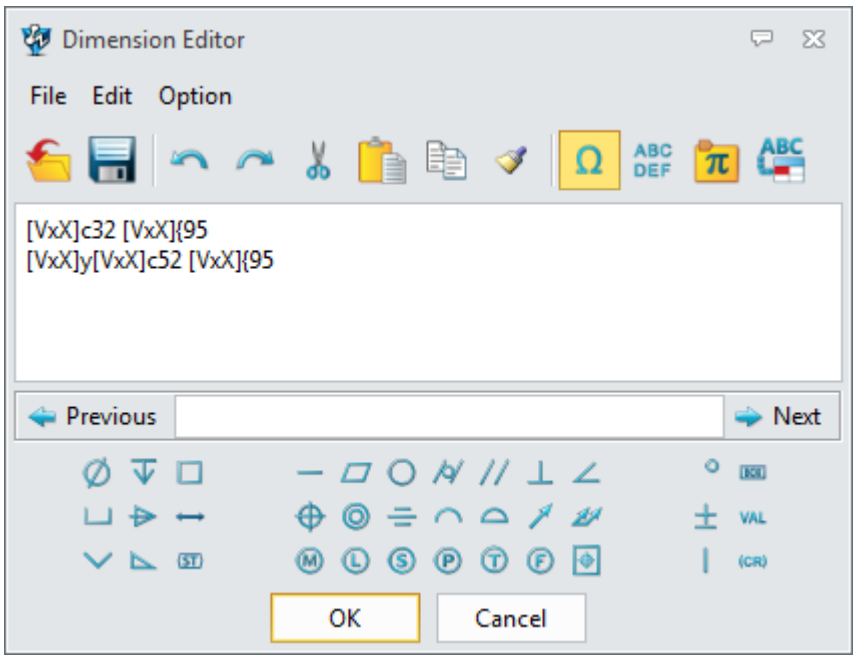
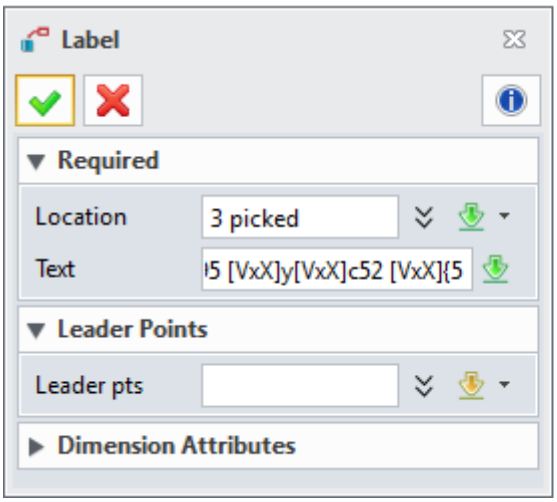


Figure87 Define Dimension Text with Editor

STEP 03 Click OK to get the label, as shown in the image below.



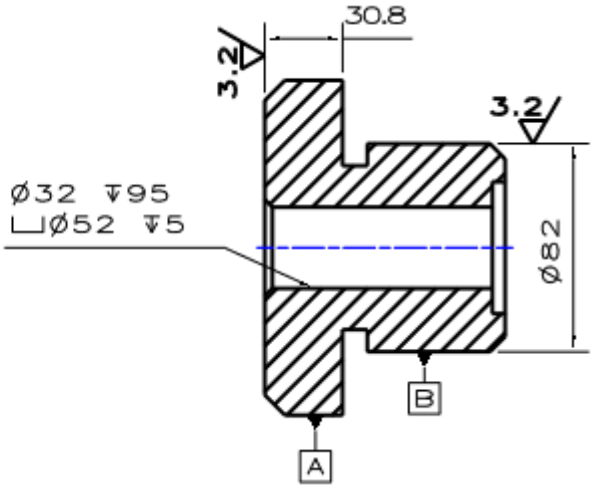


Figure88 Hole Dimension Label

Leader pts option： Select points to locate additional leader arrows.

1.6 Table

1.6.1 Create a BOM Table

Dimension ribbon toolbar-> Table -> 

BOM

Select the view to create a BOM table, including detail and section view.

Let's see the image below, this is an assembly including some components and subassemblies.

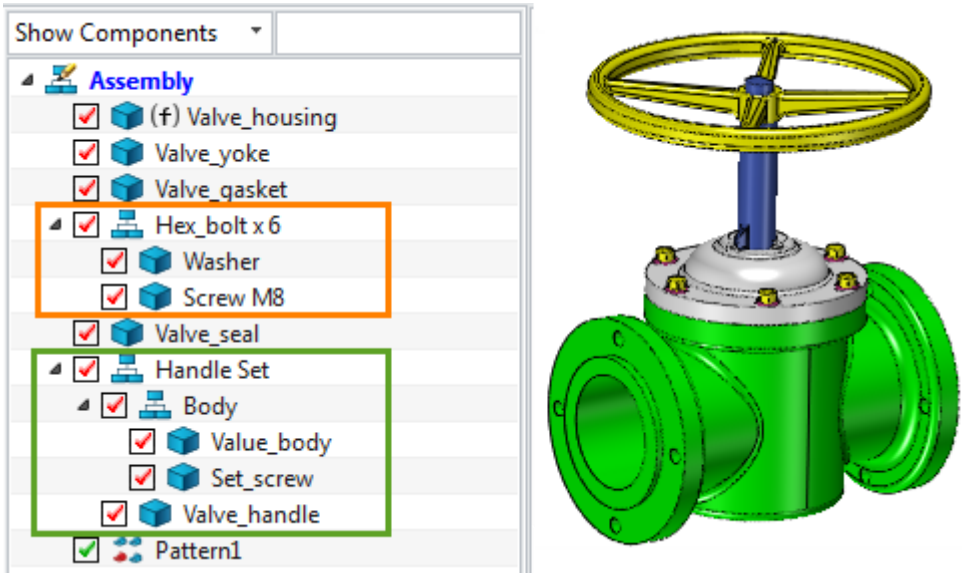
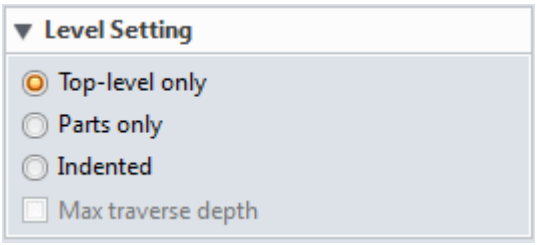
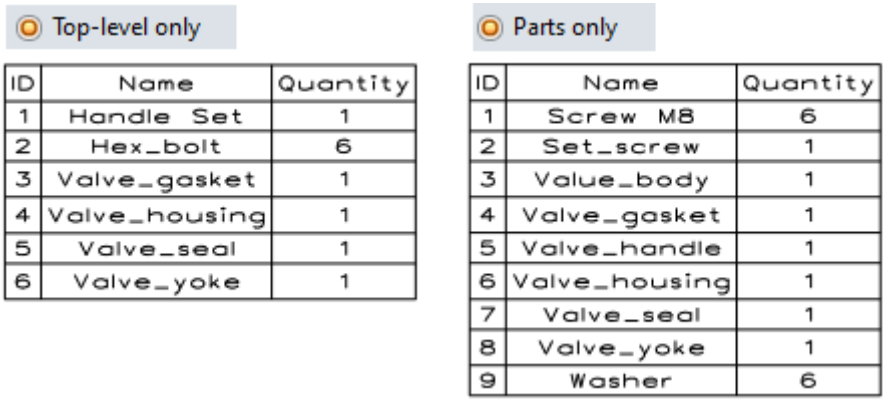


Figure89 Assembly Model

Level Setting 

Top-level only: only list out parts and sub-assemblies excluding sub-assembly components

Parts only: only list out all parts including the one from all sub-assemblies, but not list sub-assembly. Each subassembly component is an individual item.



Top-level only

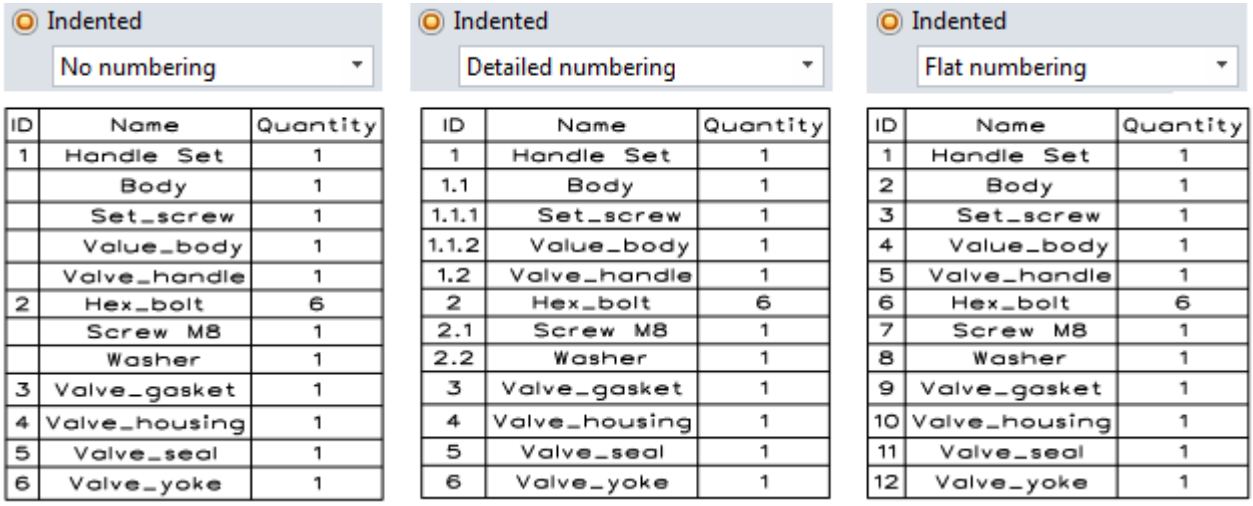
ID	Name	Quantity
1	Handle Set	1
2	Hex_bolt	6
3	Valve_gasket	1
4	Valve_housing	1
5	Valve_seal	1
6	Valve_yoke	1

Parts only

ID	Name	Quantity
1	Screw M8	6
2	Set_screw	1
3	Value_body	1
4	Valve_gasket	1
5	Valve_handle	1
6	Valve_housing	1
7	Valve_seal	1
8	Valve_yoke	1
9	Washer	6

Figure90 Top-level only VS Part only

Indented: list out all the parts and sub-assemblies and their components, and more further controls are provided. Three methods are provided to define ID, as shown in the image below.



Indented — No numbering

ID	Name	Quantity
1	Handle Set	1
	Body	1
	Set_screw	1
	Value_body	1
	Valve_handle	1
2	Hex_bolt	6
	Screw M8	1
	Washer	1
3	Valve_gasket	1
4	Valve_housing	1
5	Valve_seal	1
6	Valve_yoke	1

Indented — Detailed numbering

ID	Name	Quantity
1	Handle Set	1
1.1	Body	1
1.1.1	Set_screw	1
1.1.2	Value_body	1
1.2	Valve_handle	1
2	Hex_bolt	6
2.1	Screw M8	1
2.2	Washer	1
3	Valve_gasket	1
4	Valve_housing	1
5	Valve_seal	1
6	Valve_yoke	1

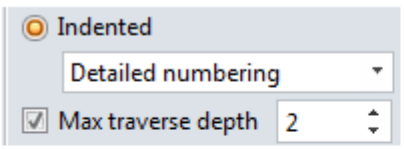
Indented — Flat numbering

ID	Name	Quantity
1	Handle Set	1
2	Body	1
3	Set_screw	1
4	Value_body	1
5	Valve_handle	1
6	Hex_bolt	6
7	Screw M8	1
8	Washer	1
9	Valve_gasket	1
10	Valve_housing	1
11	Valve_seal	1
12	Valve_yoke	1

Figure91 Indented Level

Max traverse depth：This option is to define which assembly level BOM should read out up to.

When this option is checked and the value is set as 2, you will get the following BOM table.



Indented — Detailed numbering — Max traverse depth 2

ID	Name	Quantity
1	Handle Set	1
1.1	Body	1
1.2	Valve_handle	1
2	Hex_bolt	6
2.1	Screw M8	1
2.2	Washer	1
3	Valve_gasket	1
4	Valve_housing	1
5	Valve_seal	1
6	Valve_yoke	1

Figure92 Different Traverse Depth

Item Numbers and Template

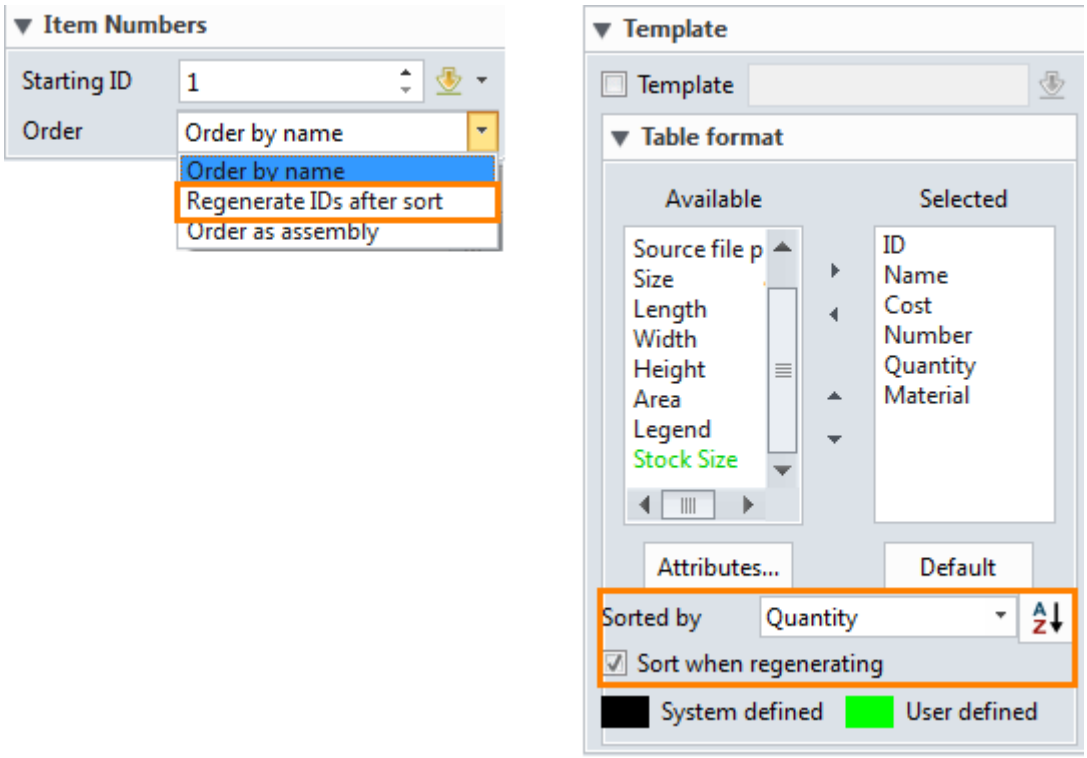


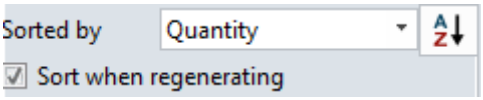
Figure93 Item Numbers and Template

There are three different methods to sort the table ID.

Order by name: Based on the part name to sort the table ID.

Order as assembly: Based on the components insertion order to sort the table ID.

Regenerate IDs after sort: When table order is changed, IDs will be regenerated. The table order is controlled by another parameter, as shown in the image below.



ID	Name	Quantity
1	Handle Set	1
2	Valve_gasket	1
3	Valve_housing	1
4	Valve_yoke	2
5	Valve_seal	2
6	Hex_bolt	6

ID	Name	Quantity
1	Valve_yoke	1
2	Valve_housing	1
3	Valve_seal	1
4	Handle Set	2
5	Valve_gasket	3
6	Hex_bolt	6

Figure94 Table order

Notes: *If the option of" Sort when regenerating" is unchecked, the table order will not change, so IDs also will not change.*

All available part attributes are listed in here, including black system attributes (such as Legend) and green user attributes (such as Stock Size), as shown in Figure 95.

 : Add or delete the attribute  : Rearrange the sequense of the attribute

ID	Name	Quantity	Material	Mass[kg]	Legend
1	Handle Set	1	Aluminum	6.427	
2	Hex_bolt	6	Steel-grey	0.015	
3	Valve_gasket	1	Rubber	0.021	
4	Valve_housing	1	Brass-cast	11.983	
5	Valve_seal	1	Rubber	0.003	
6	Valve_yoke	1	Steel-cast	2.122	

Figure95 BOM Table

BOM Filter

Use BOM filter to set up some conditions to generate a specified table.

Part attributes and customized attributes are available for this filter condition.

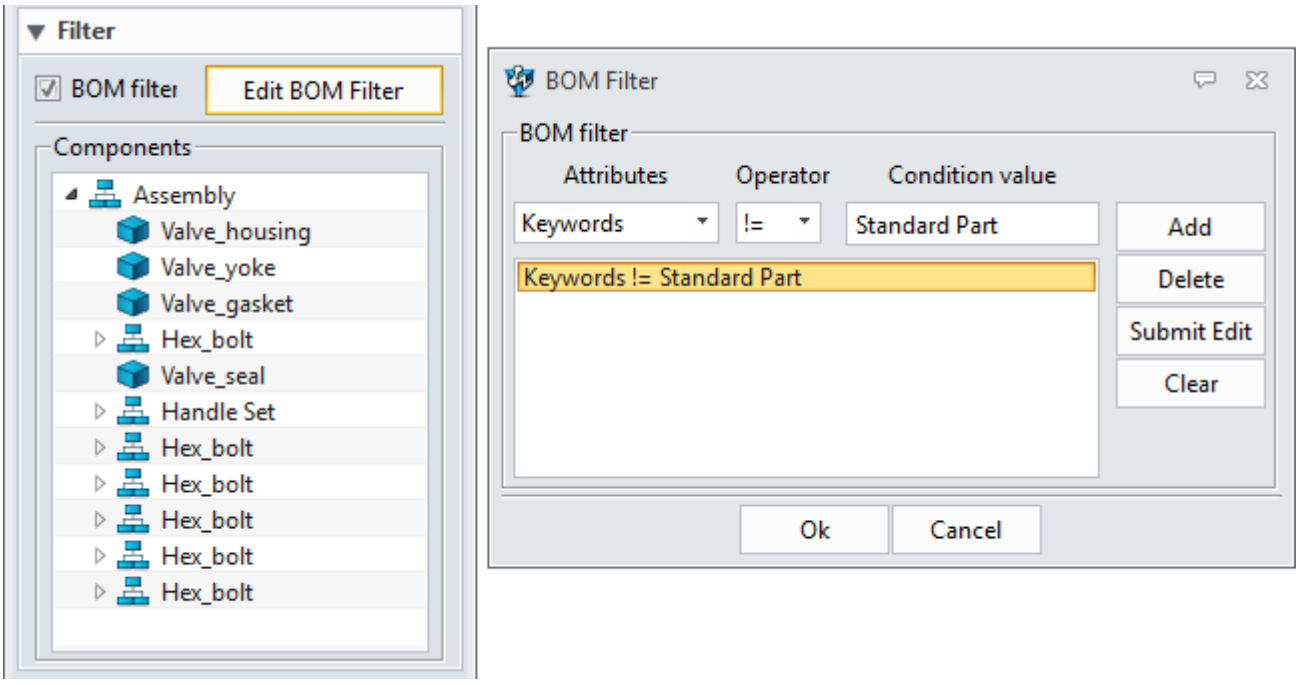


Figure96 BOM Filter

1.6.2 Edit the Table

Move the cursor on the table, click  to activate the table editor. Click any cloumn or any row to get the table cloumn/row editor, as shown in the image below. Then you can add/delete the column/row, set the text alignment,text attribute and so on.

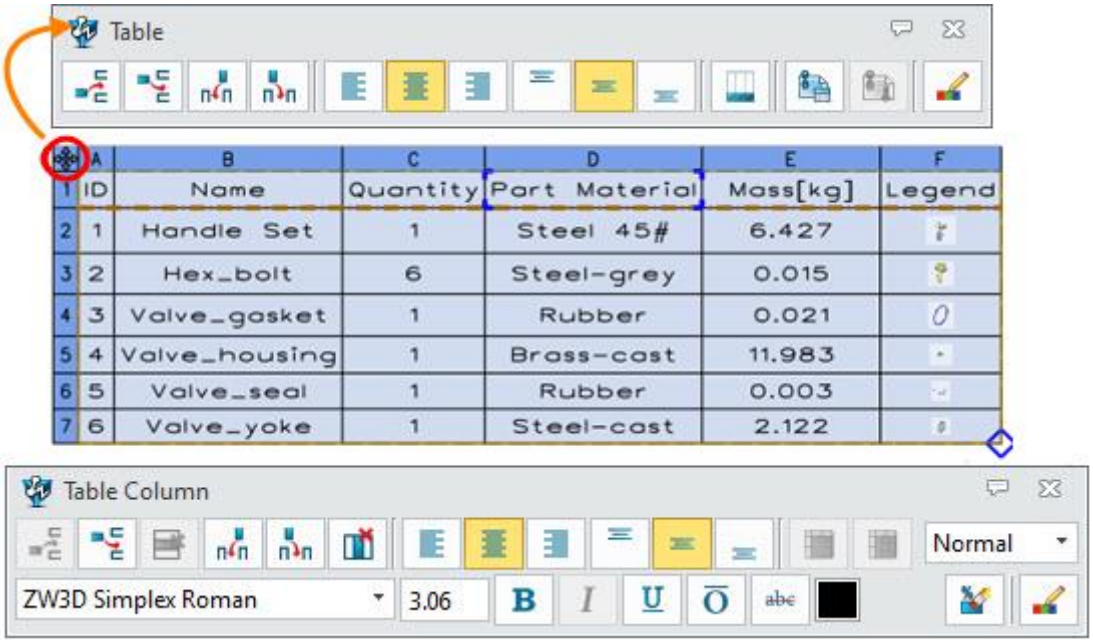


Figure97 Table Editor

Q1. Edit the Column Name

A1: Double click the column head to edit, such as Material ->Part Material.

Q2. Edit the Column Attribute

A2: Right click the column head, then select the attribute to edit, such as value precision.

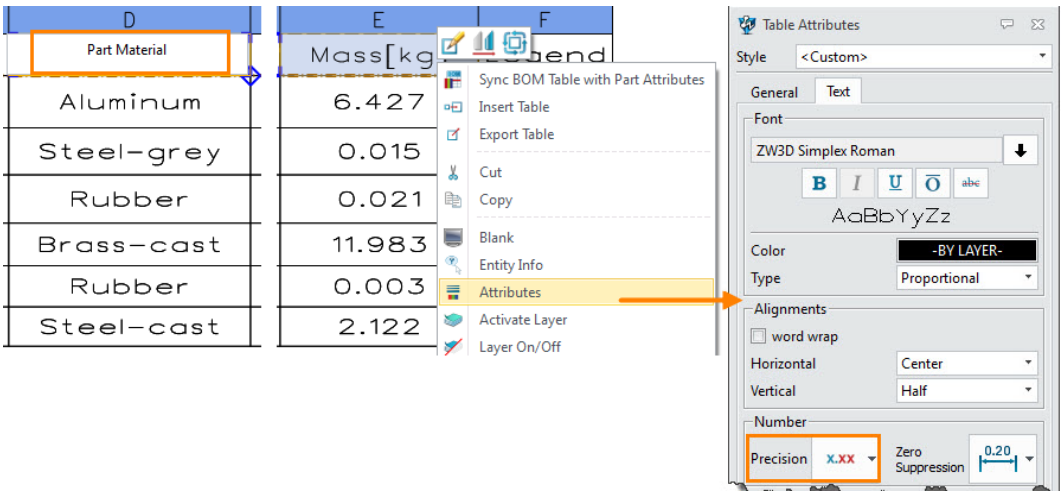


Figure98 Edit Column Name and Attribute

Q3. Edit Cell Value

A3: Double click the cell, then "Yes" button to unlock the value to edit. Such as changing the material of one component.

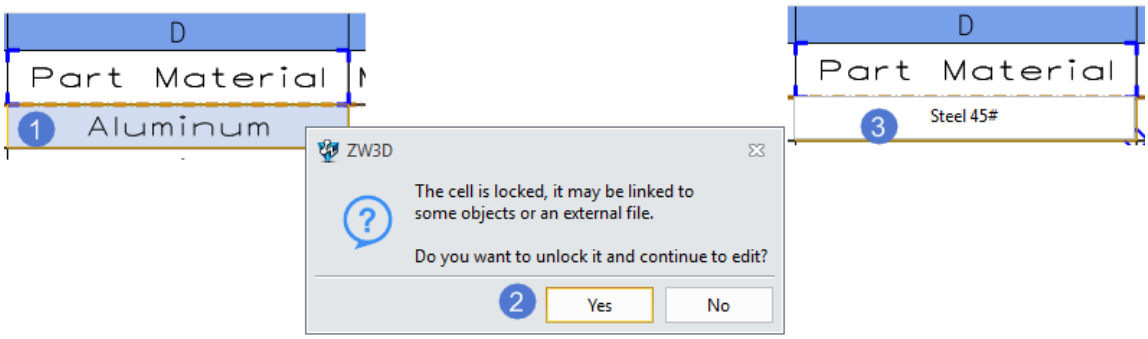


Figure99 Edit Cell Value

Q4. Sync BOM table with Part Attributes

A4: Move the cursor on the table, then right click  ->click this command "Sync BOM table with part attributes". Then you could check the part attribute in modeling level by Tool ribbon tab-> Attribute->Part attribute.

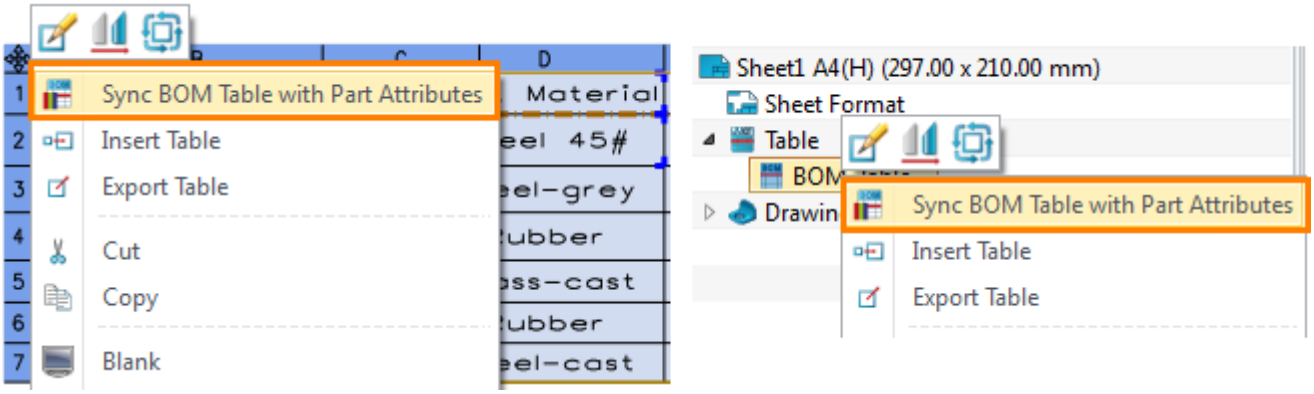


Figure100 Sync BOM table with part attributes

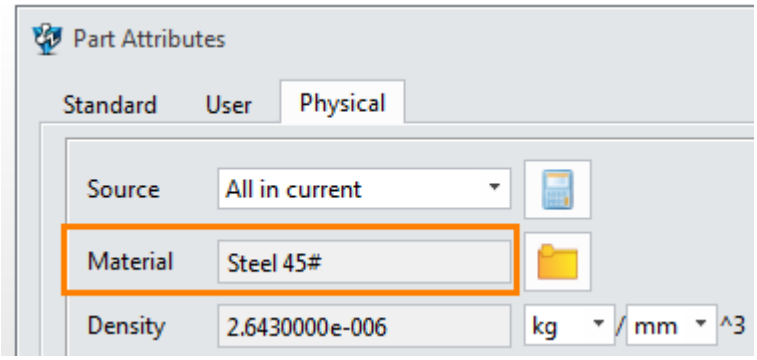


Figure101 Updated Part Attribute

Q5. Edit Table Header

A5: Move the cursor on the table, then right click  ->Select Insert –> Head Row. Then you could merge the cells or edit the header cells, as shown in the image below.

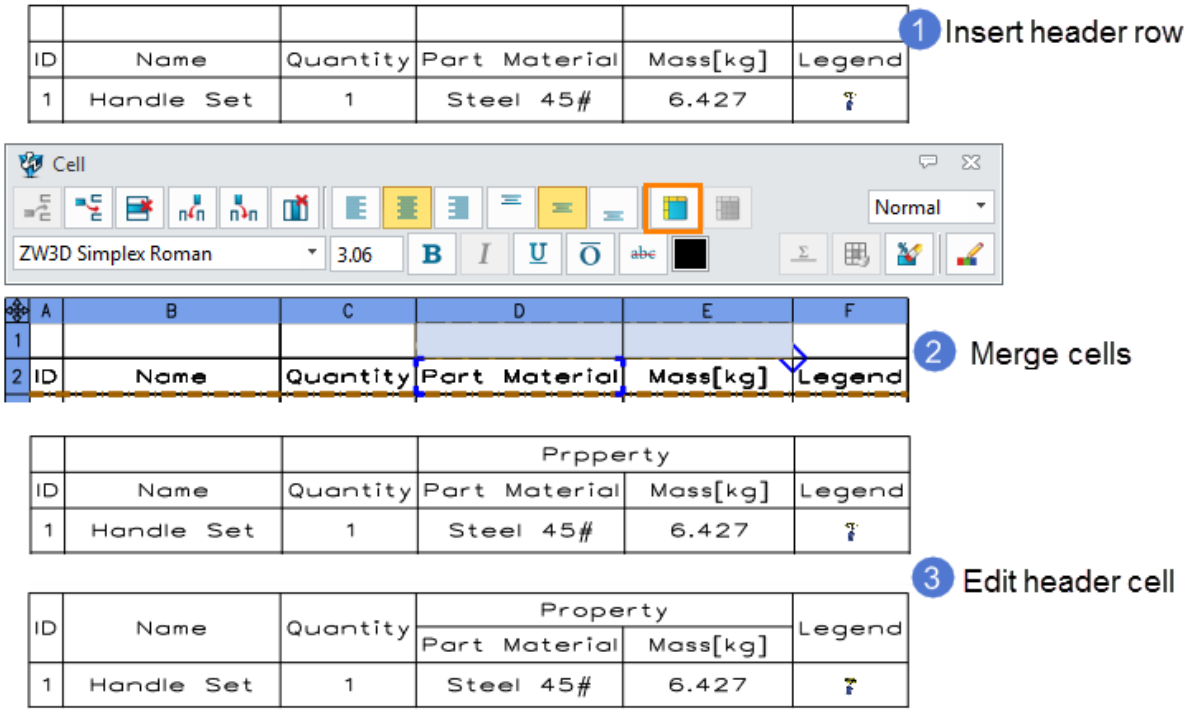


Figure102 Edit Table Header

Q6. Table Template

A6: Select the whole table, then you could use "Save as Template…" to save this table header as a BOM template. Then when you create a new BOM table, you could import this tmeplate.

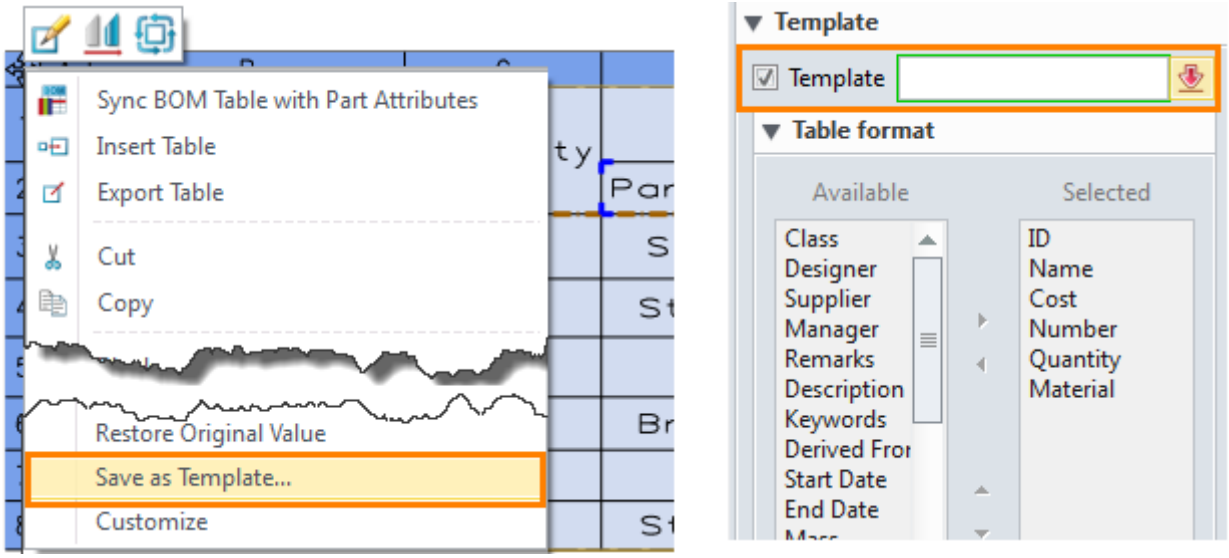


Figure103 Table Template

Q7. Import/ Export Table

A7: As shown in Figure 100, the table can be exported by "Export Table" command and saved as an excel file. Then you could edit this excel file. After editing, the table file can be imported into 2D sheet, as shown in the image below.

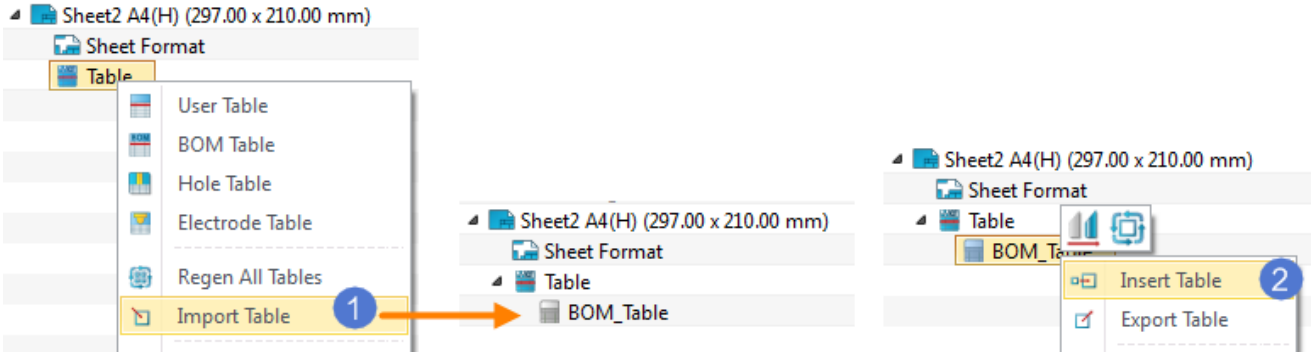
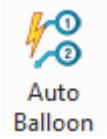


Figure104 Import & Insert Table

1.6.3 Auto Balloon

Dimension ribbon toolbar-> Annotation -> 

Balloons are automatically generated in a view based on component visibility. Balloons are inserted into the appropriate views without duplicates. You can specify that the balloons follow the assembly order or are numbered sequentially.

STEP 01 Select the 2D view. The default balloon text is ID.

Notes: *The Lower Text option is activated when the balloon type is circular split line.*

STEP 02 Set the layout parameters. Pattern type is defined as square type.

STEP 03 Set balloon type as circular and use the second quantity type.

STEP 04 Click OK to get the result, as shown in the image below.

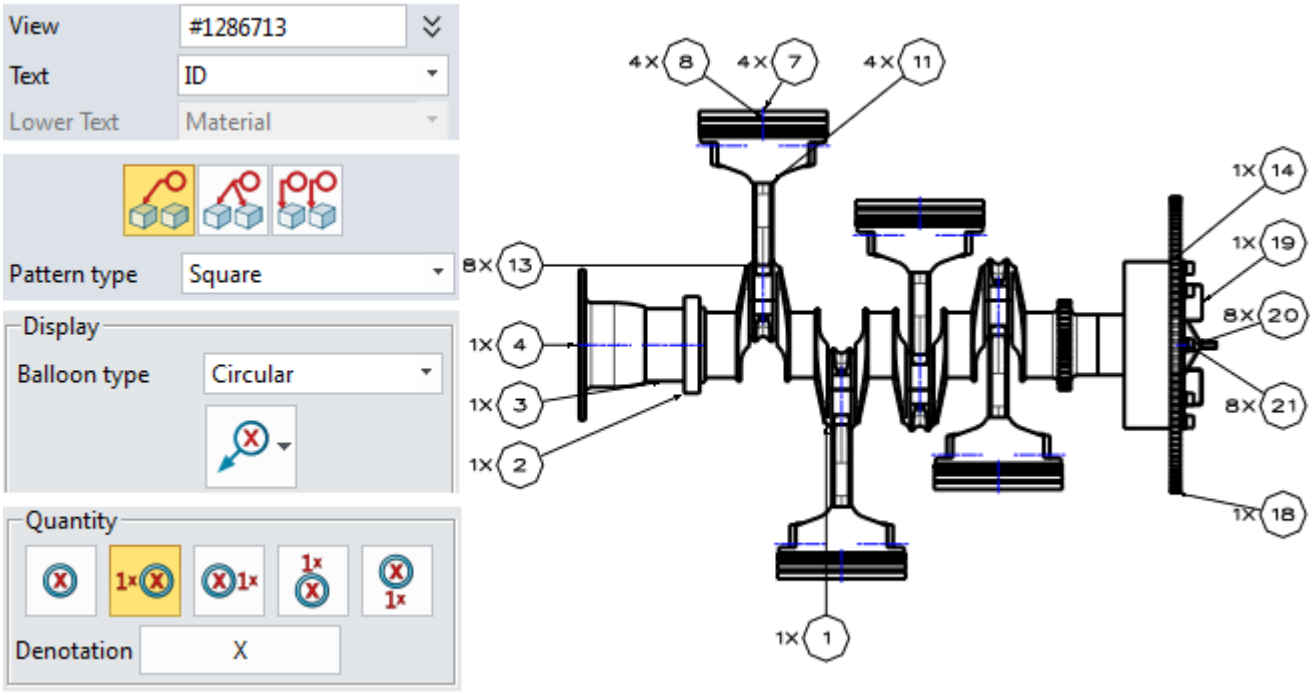


Figure105 Ballloon-Result1

You could try other parameters to get the different balloon display effect, as shown in the image below.

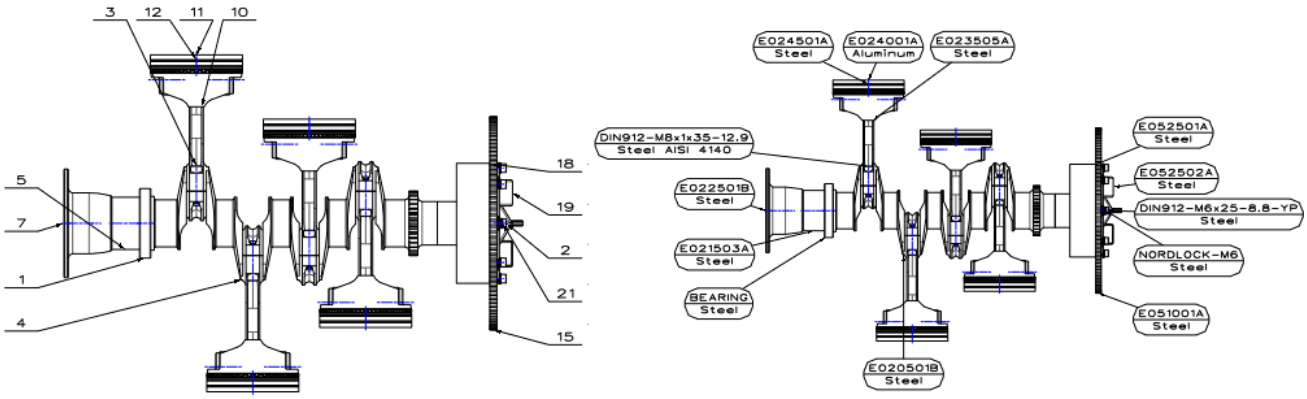
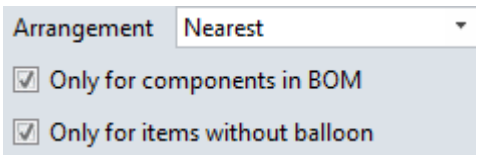


Figure106 Different Balloon Display

More Layout Parameters

If there is a BOM inserted in drawing sheet, these options are activated.



“Only for components in BOM” option is to control whether the excluded components from BOM should be labeled. If the excluded components are included, a * character will be used as their ID which you can modify to the one you like.

"Only for items without balloon" is to label the components which are not labeled in any other views.

STEP 01 Create the top view and the projection view on the left, as shown in Figure 108.

STEP 02 Create the BOM table with "Part only" method.

STEP 03 Select the top view to create the balloon.

STEP 04 Select the projection view to create the balloon. The parameters are set as below.

STEP 05 Get the balloon result as shown in Figure 108.

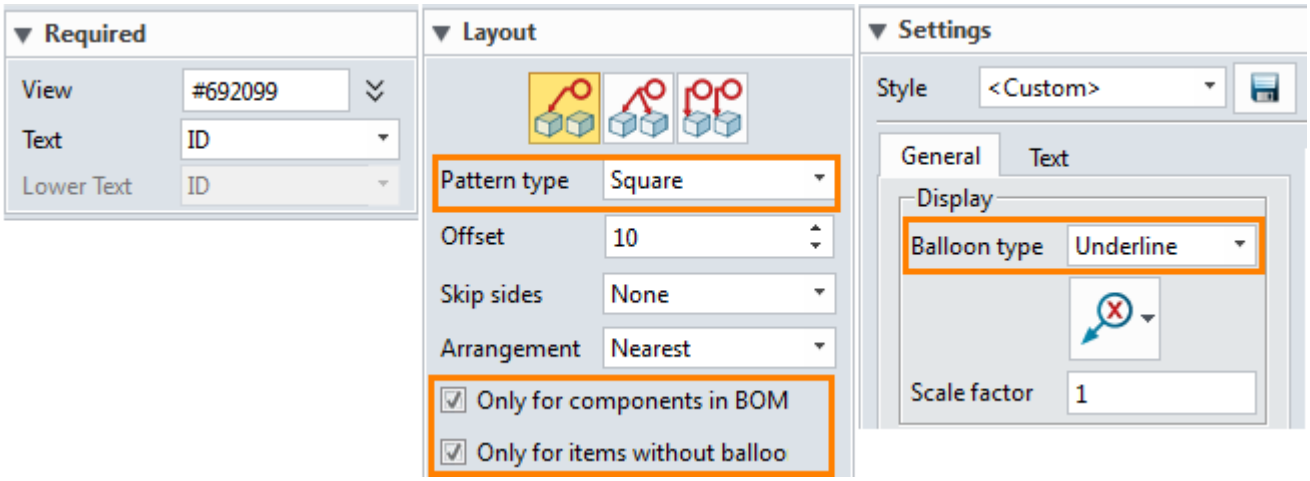


Figure107 Definition of Parameters

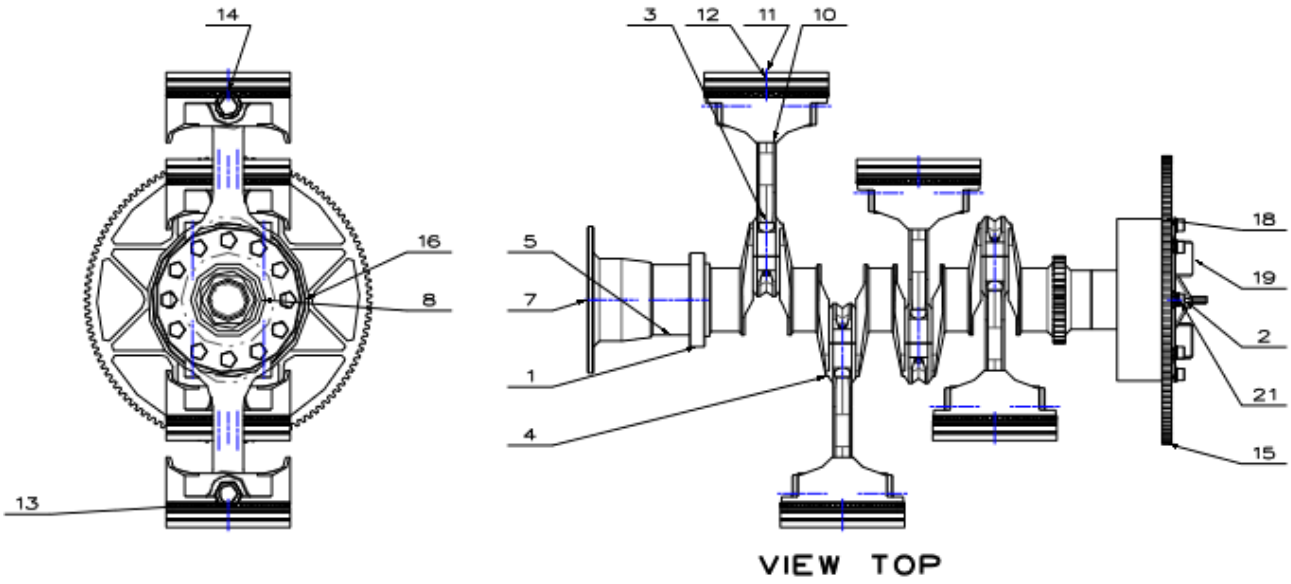


Figure108 Balloon of Cross-views

Relations between Balloon ID and BOM ID

How the balloon ID is generated from BOM ID can follow the rules listed below:

1) If no BOM has been generated from any view of an assembly, balloon will label every component in part level with IDs that follow their assembly sequences.

2) If a BOM is generated from a view of an assembly, when “Arrangement” is using “Nearest”, balloon ID will use BOM ID to label components.

3) If a BOM is generated from a view of an assembly, when “Arrangement” is using “CW” or “CCW”, balloon ID will be generated and BOM ID will change to the new ID.

4) If a BOM is generated from a view of an assembly, when “Arrangement” is using “CW” or “CCW” and “Keep the item numbers” option is checked, balloon ID will follow BOM ID but re-place them in defined style.

5) If several BOMs have generated within one sheet, “Balloons link to BOM” from the right-click menu of a view can be used to link the view to a specific BOM, then balloon ID can inherit the BOM ID.

1.6.4 Balloon

Dimension ribbon toolbar-> Annotation -> 

Pick drawing entities to manually create the balloons by with command.

You can define multiple base points and multiple leader arrows to generate the same balloon text, as shown in the image below.

The parameters of dimension attributes are the same with Auto Balloon command.

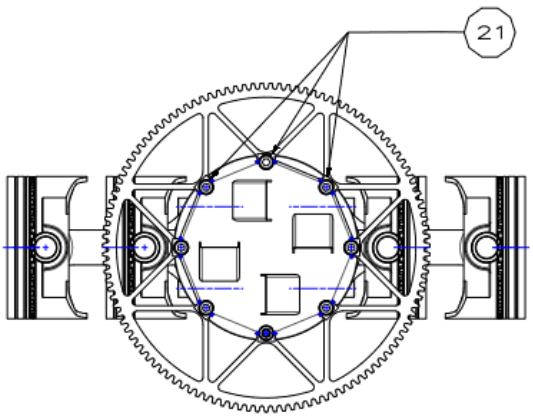


Figure109 Manual Balloon

1.6.5 Hole Table

Dimension ribbon toolbar-> Table ->

Use this command to create a hole table based on a layout view that contains real holes and user-defined holes. Take this model as an example

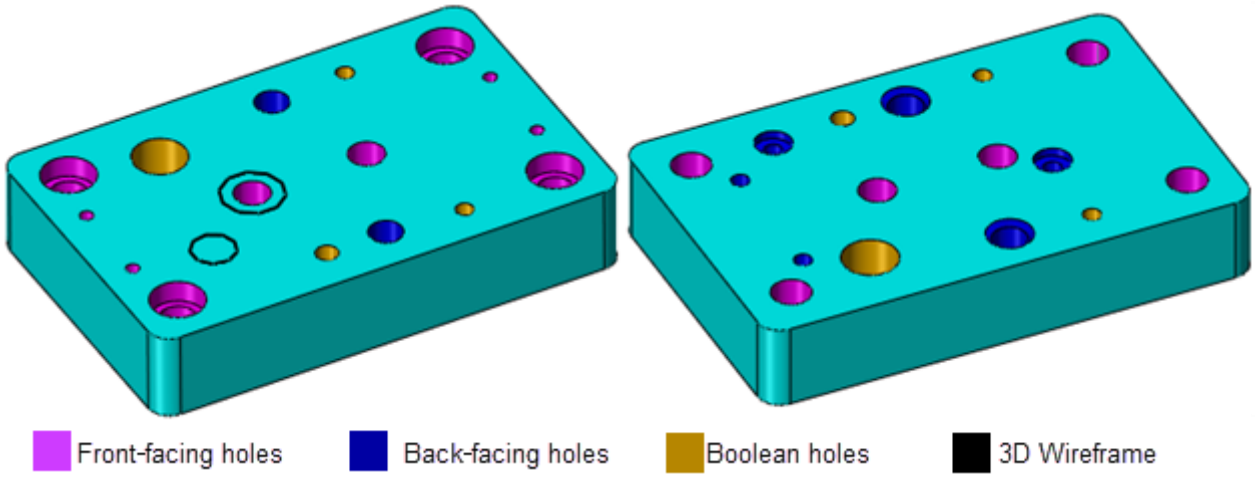


Figure110 Hole Model

STEP 01 Create a top view based on this model.

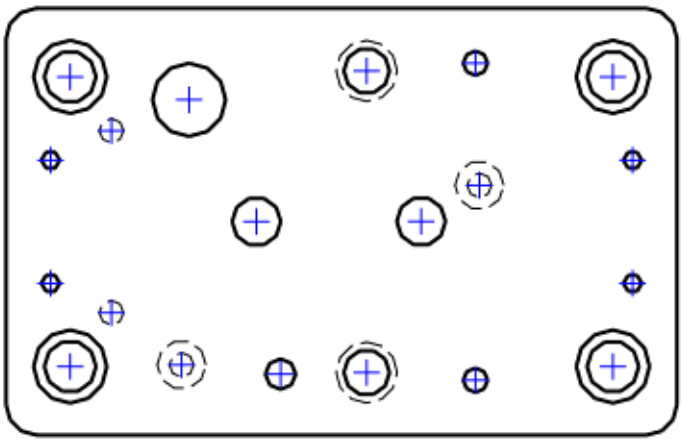


Figure111 Top view-Hole

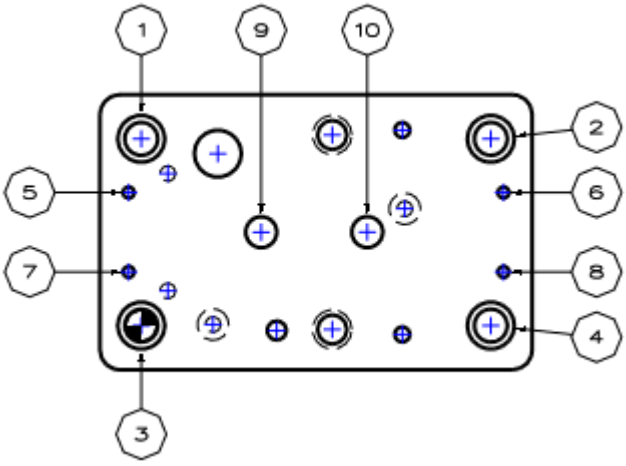
STEP 02 Select the 2D view and give an name for the hole table.

STEP 03 Pick a point as the base point. Then, in hole table, the x & y coordinates of the hole will be defined relative to this point

STEP 04 Set the hole filter. As default, only front-facing holes are selected.

STEP 05 Set other parameters if required. Such as check the option “Combine same size”.

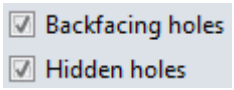
STEP 06 Click OK. Then pick a point to insert the hole table. The result is shown in the image below.



ID	X-center	Y-center	Size
1	-0.00	47.51	Ø 8.50 ↧ 20.00 ⌴Ø 12.00 ↧ 5.00
2	88.97	47.51	
3	0.00	0.00	
4	88.97	0.00	
5	-3.22	33.85	Ø 3.00 ↧ 12.00
6	92.19	33.85	
7	-3.22	13.66	
8	92.19	13.66	
9	30.54	23.77	Ø 8.00 THRU
10	57.53	23.80	

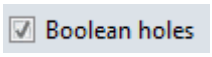
Figure112 Hole Table

Hole Filter



When only “Backfacing holes” option is checked, back-facing hole features based on this view plane are selected. If the back-facing hole is not a thru-hole, it will not be selected.

When “Backfacing holes” option and “Hidden holes” option both are checked, all back-facing holes are selected.



Check this option, the imported holes and Boolean holes will be included.

Notes： *Checking or unchecking these three options does not affect holes that you have already selected for the hole table. This option has no effect when selecting user-defined holes.*

So on current version, after resetting the hole filter, please redefine the view option. Then the selected hole features are updated, as shown in the image below.

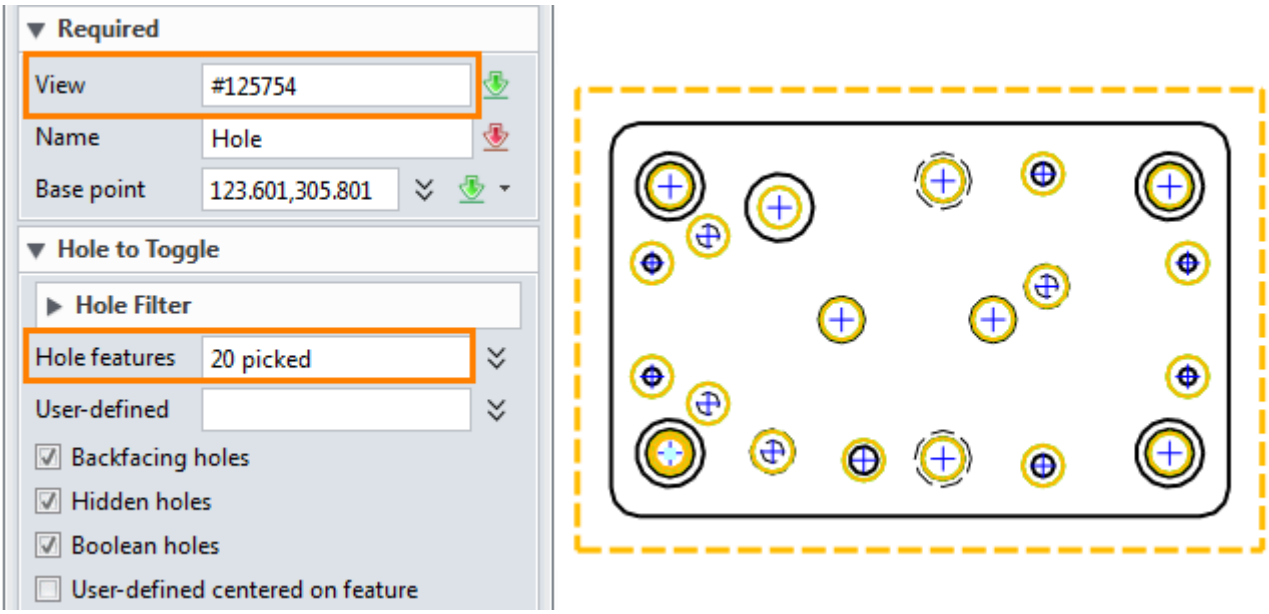


Figure113 Reset Hole Filter

How to add the user-defined hole into hole table?

STEP 01 If some circles are drawn in part level, you could edit the view attribute to show them firstly. Right click the view-> Display others-> Show 3D Curves from Part.

See the image below, there are two circles. Circle2 is concentric with one hole feature.

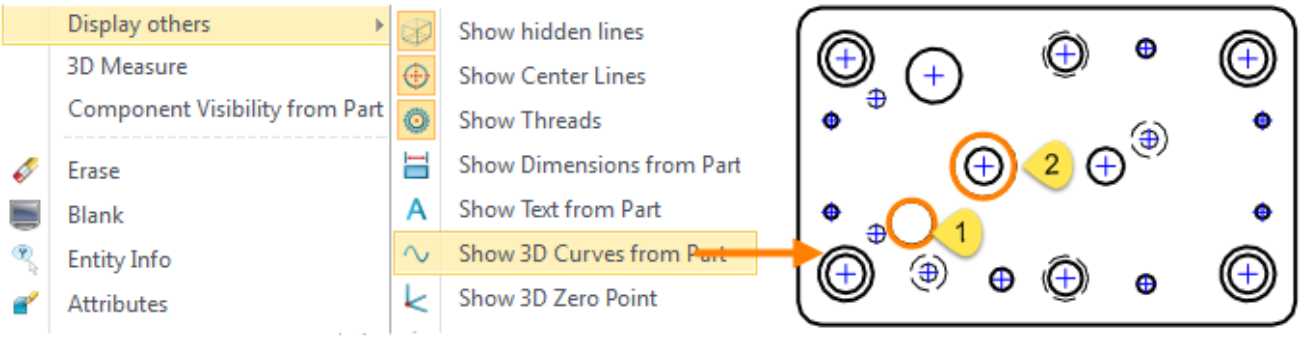


Figure114 Show 3D Curves from Part

STEP 02 Activate “User-defined” option, then the circle1 can be picked. If you want to pick circle2, the option “User-defined centered on feature” should be checked.

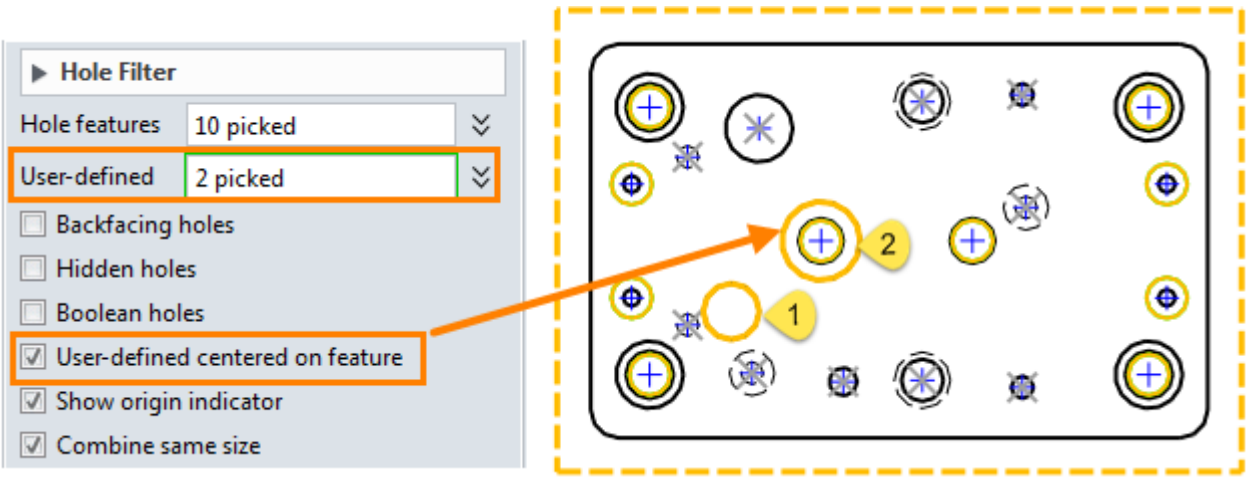
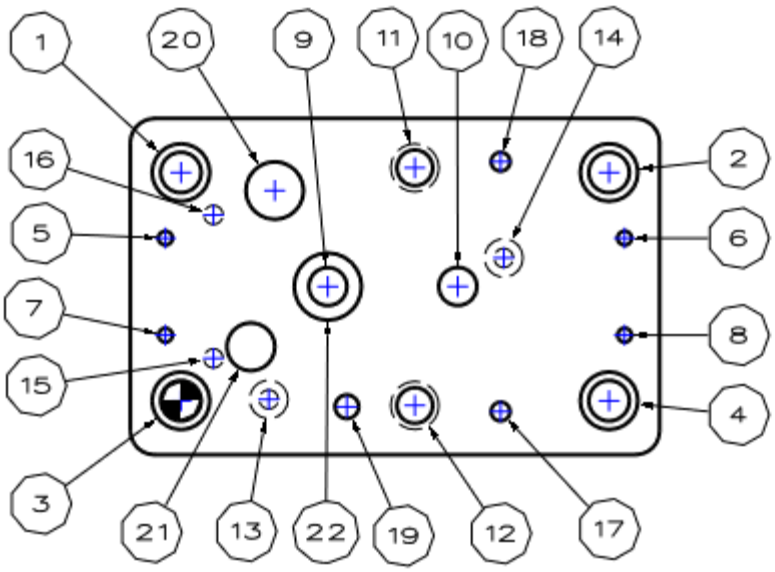


Figure115 User-defined Hole Feature

STEP 02 Pick all hole features and add user-defined holes for hole table. The result is shown in the image below.



ID	X-center	Y-center	Size
1	-0.00	47.51	Ø 8.50 ↧ 20.00 ⌴Ø 12.00 ↧ 5.00
2	88.97	47.51	
3	0.00	0.00	
4	88.97	0.00	
5	-3.22	33.85	Ø 3.00 ↧ 12.00
6	92.19	33.85	
7	-3.22	13.66	
8	92.19	13.66	
9	30.54	23.77	Ø 8.00 THRU
10	57.53	23.80	
11	48.60	48.49	Ø 7.50 THRU ⌵Ø 10.00 X 90°
12	48.60	-0.98	
13	18.26	0.27	Ø 4.00 ↧ 10.00 ⌴Ø 8.00 ↧ 2.00
14	67.08	29.72	
15	6.76	8.83	Ø 4.00 ↧ 12.00
16	6.76	38.68	
17	66.43	-2.24	Ø 4.00 ↧ 20.00
18	66.43	49.76	
19	34.49	-1.24	Ø 5.00 ↧ 20.00
20	19.49	43.76	Ø 12.00 ↧ 20.00
21	14.54	11.23	Ø 10.00
22	30.54	23.77	

Figure116 Whole Hole Table

1.7 Case---2D Drawing

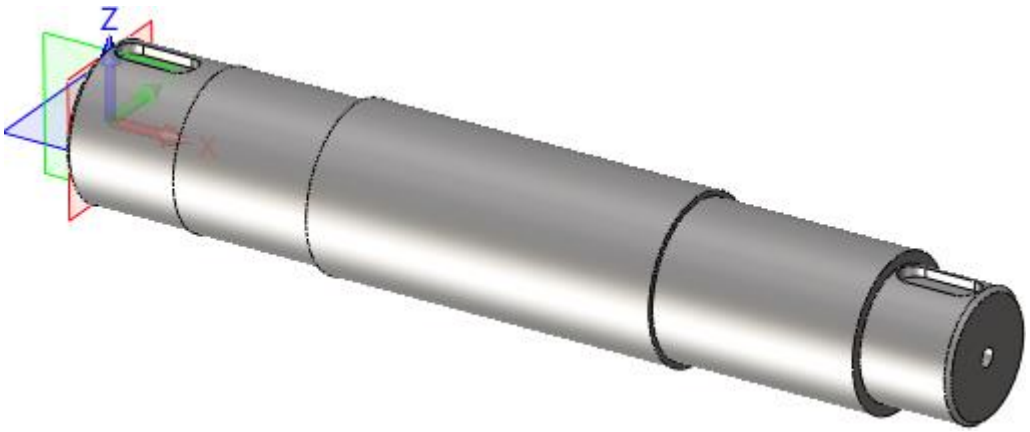


Figure117 Case1--- 3D Model

STEP 01 In modeling level, select “2D Sheet” command and “A2_H(ANSI)”template to create a 2D sheet.

STEP 02 Create a standard Top view, as shown in the image below.

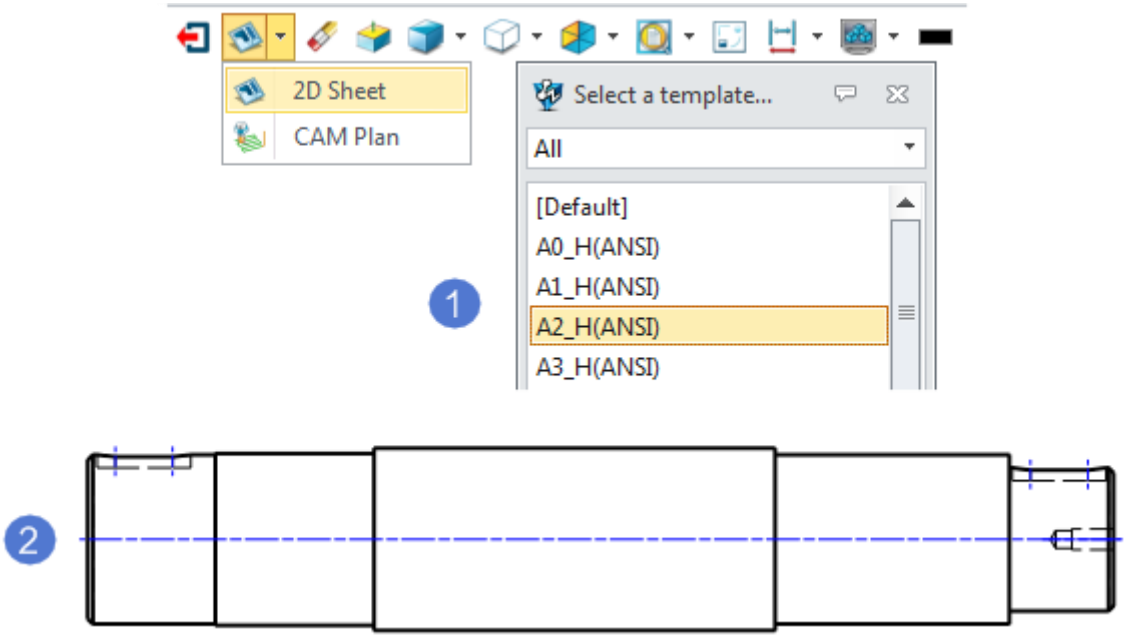


Figure118 Top View

STEP 03 Use “Full Section” command to create two section views. The parameters are set as shown in the image below.

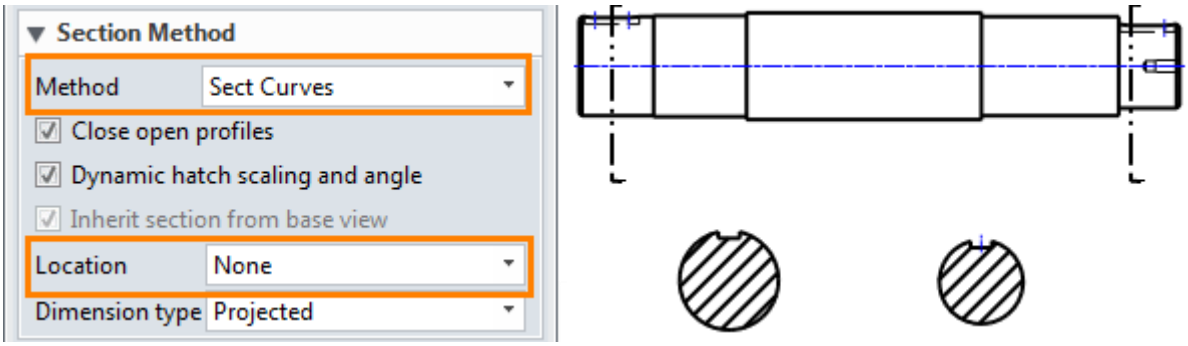


Figure119 Section Views

STEP 04 Drag and drop the end point of the section line to get a better position. And right click the section line to select "Display label" command.

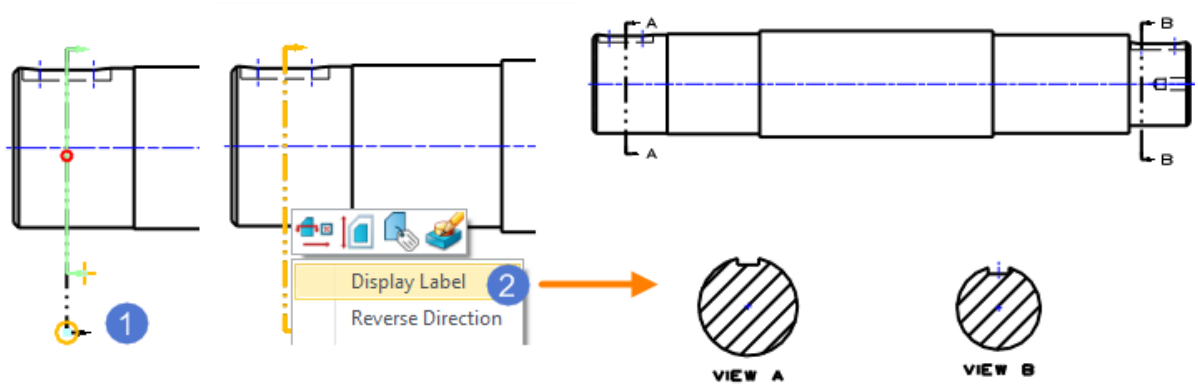


Figure120 Section Views with Label

STEP 05 Use "Broken Section" command to create two broken section views on top view.

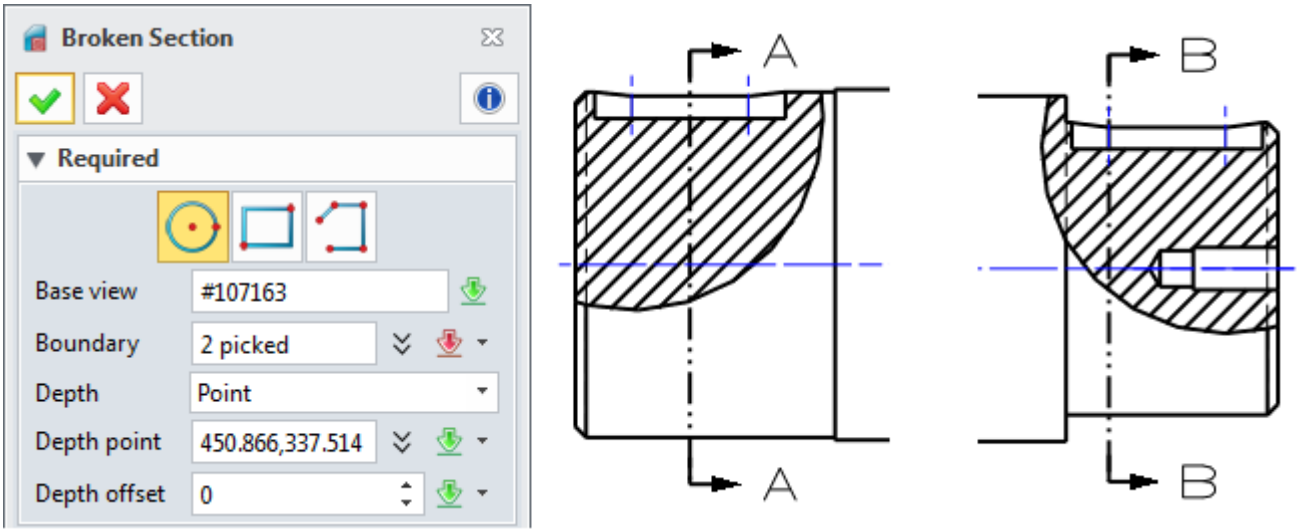


Figure121 Broken Section Views

STEP 06 Open the Style Manager to make some modifications, such as value precision of linear dimension, text format of chamfer dimension, text shape.

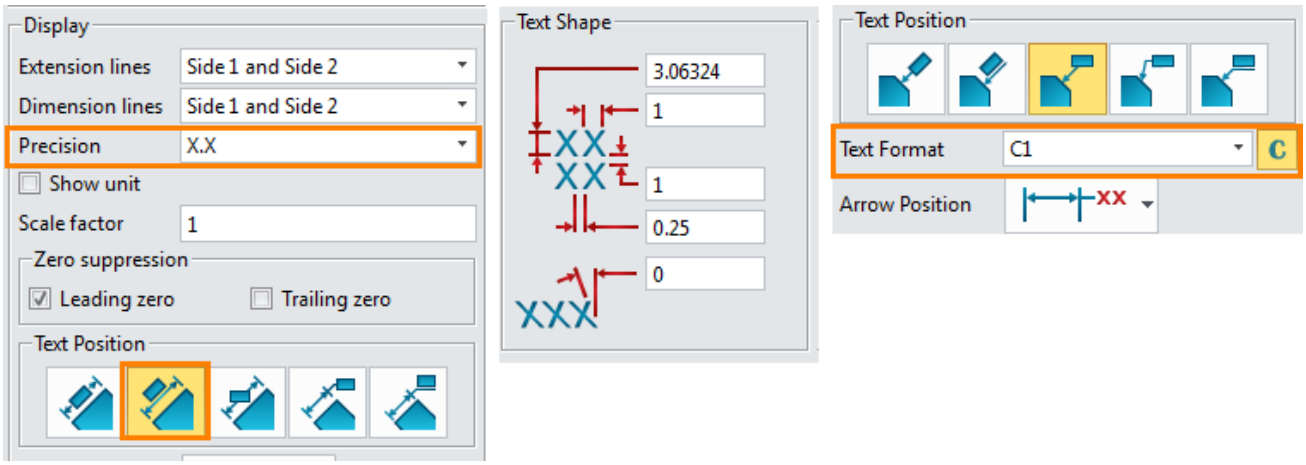


Figure122 Dimension Style

STEP 07 Create the dimensions, annotations, hole labels, surface finish symbols and so on.

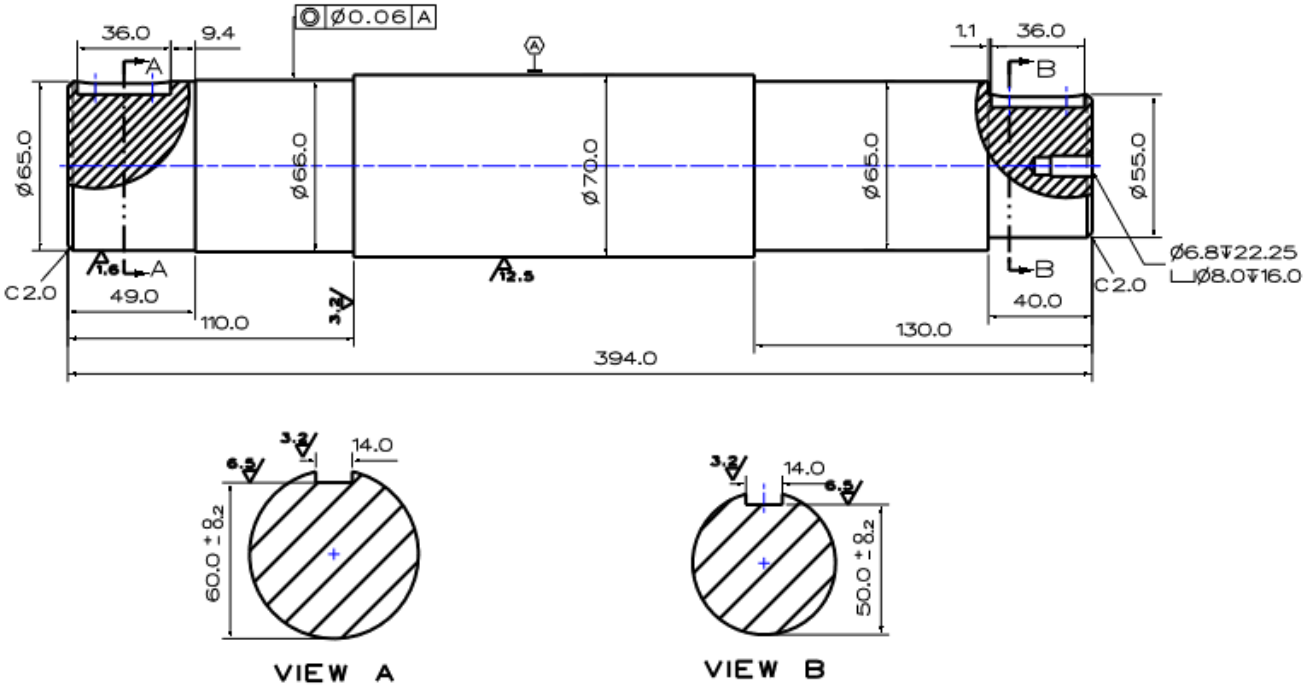


Figure123 View with Dimensions

STEP 08 Set default drawing units. Tool ribbon toolbar->Setting->Preference

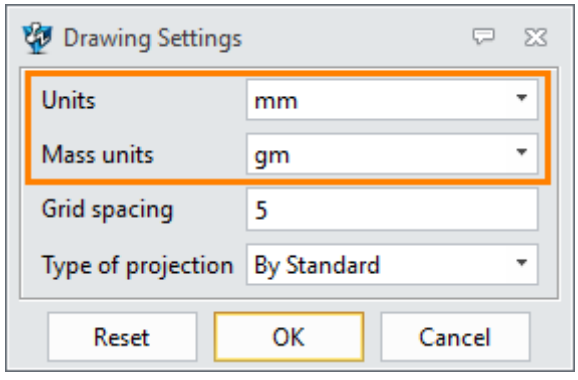


Figure124 Drawing Setting

STEP 09 Edit the title block. You could add some part attributes with text command and editor.

Figure125 Edit the Title Block

Notes: *Some texts are overlap when defining part attribute. But it doesn't matter.*

STEP 10 After editing, when back to 2D drawing level, all information has been linked with part attribute.

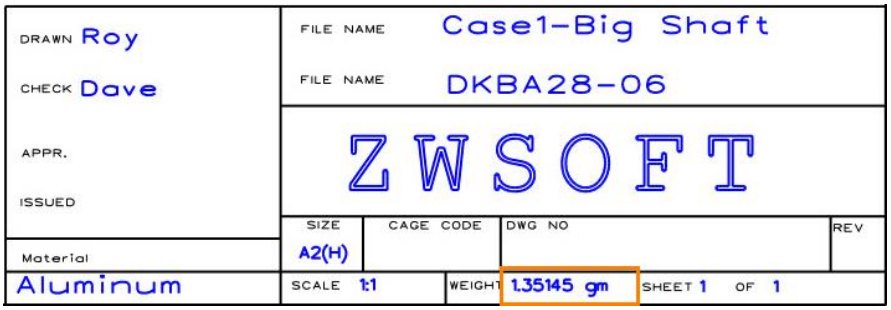


Figure126 Title Block

STEP 11 Add some technical requirements by text command.

STEP 12 The final 2D sheet is shown in the image below.

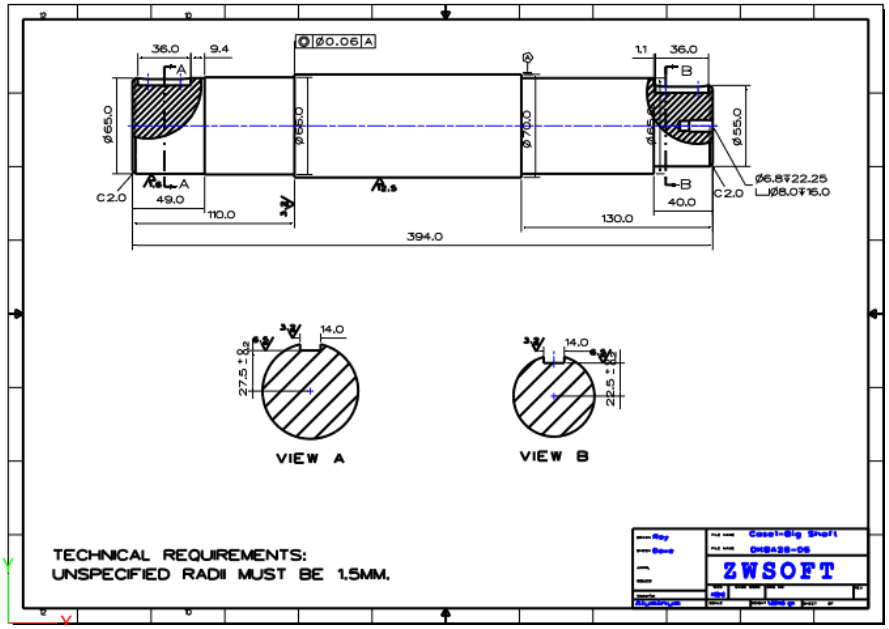


Figure127 2D Sheet of Part