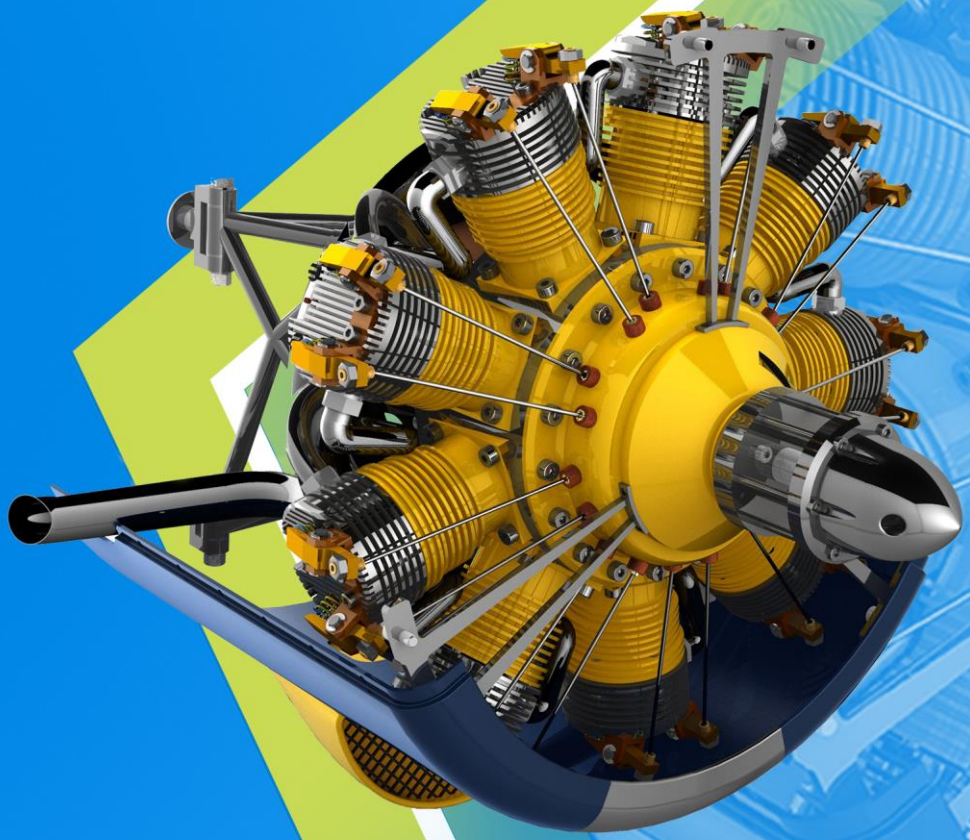
ZW3D from Entry to Master Tutorial

2D Drawing



ZW3D™ V2023 2D Drawing

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Foreword

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

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

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

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

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

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

What you are learning is ***ZW3D 2D Drawing***, a From Entry to Master tutorial.

Thanks for being our user!

The ZW3D Team

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Key Points:

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

1 An Introduction to 2D Sheet

In the process of product design and manufacturing, a 2D sheet is an important and widely used document, although not so visual as a 3D model.

With a 3D model of the product designed in ZW3D, its associative 2D sheet can be generated. When the 3D model is changed, the 2D sheet will be automatically updated.

1.1 Elements of a 2D Sheet

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

1	Standard views (Top, Front, Right, Left, Bottom, Back and Isometric), a Projection view, a Section view, a Detail view, and so on
2	Dimensions (Shape dimension & Position dimension), Tolerances (Dimension tolerance, Form tolerance & Position tolerance), Datum symbols, Surface finish symbols and Text annotations
3	Sheet format: Sheet border, Title block, and so on

Besides, the 2D sheet of an assembly includes different views, assembly dimensions, fit dimensions, a BOM table, and so on.

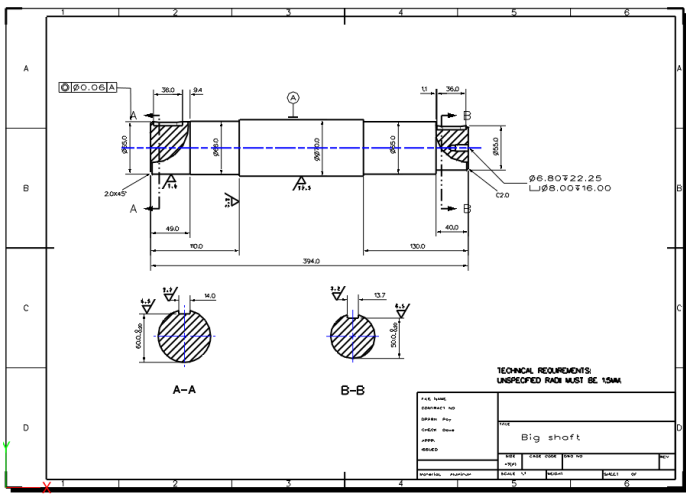


Figure 1 2D Sheet (Part)

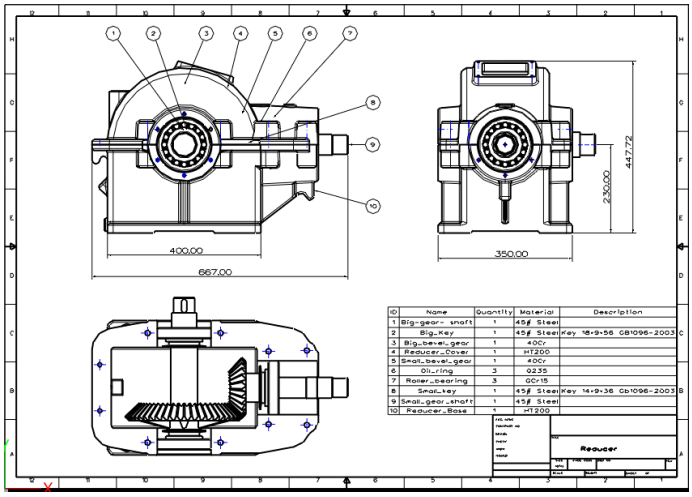


Figure 2 2D Sheet (Assembly)

1.2 Create a New 2D Sheet

Method 1: In the modeling environment, you can insert a new 2D Sheet by clicking ***2D Sheet*** from ***Document Aware Toolbars*** or the context menu in the working area. Then, select a template to create a new 2D sheet. At the same time, the standard view command is automatically activated.

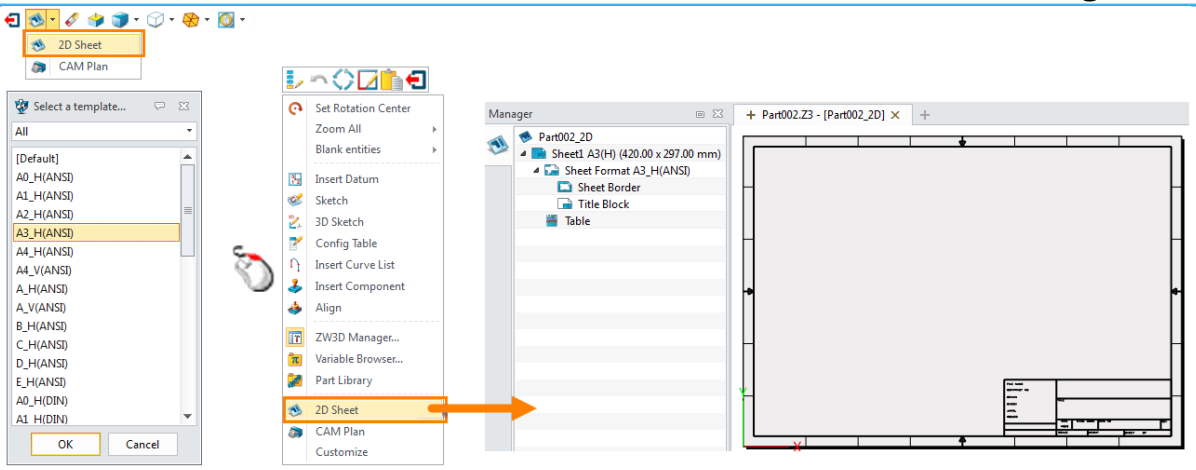


Figure 3 Create a New 2D Sheet—Method 1

Method 2: Click the ***New*** command from the ***Quick Access Toolbar***, then select the ***Drawing Sheet*** file type and a drawing template. Name the sheet and click the ***OK*** button, then a new 2D sheet is created.

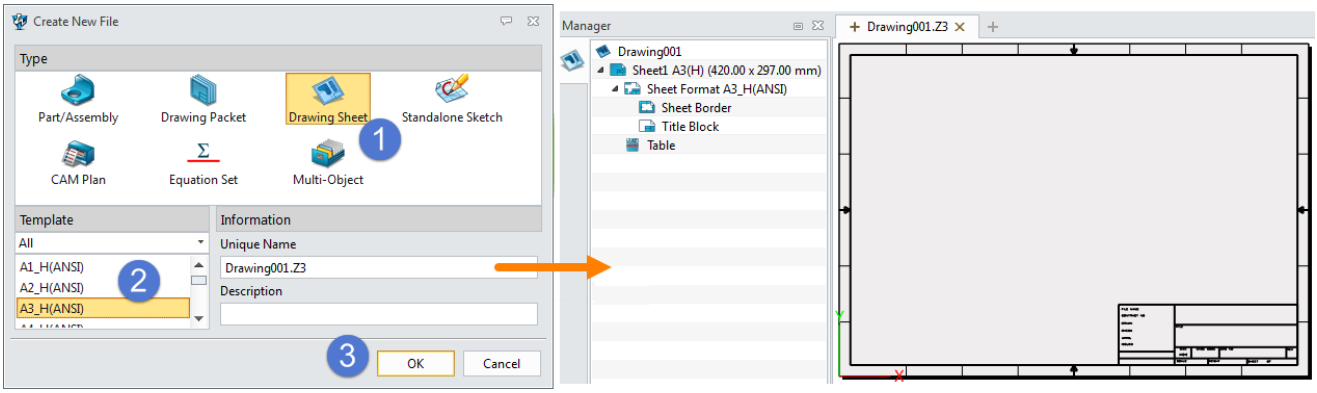


Figure 4 Create a New 2D Sheet—Method 2

Method 3: Click the ***New*** command from the ***Quick Access Toolbar***, then select the ***Drawing Packet*** file type to create a drawing file. After that, click the ***+*** button on the left side to create a new 2D sheet as a file. Other operations are like Method 2.

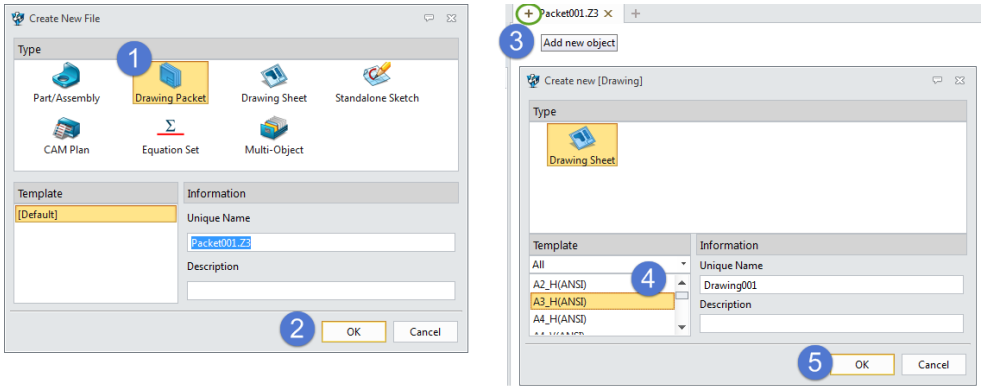


Figure 5 Create a New 2D Sheet—Method 3

1.3 Sheet Setting

- ***Configuration->2D***

  You can modify some default parameters of the drawing sheet (shown in the image below).

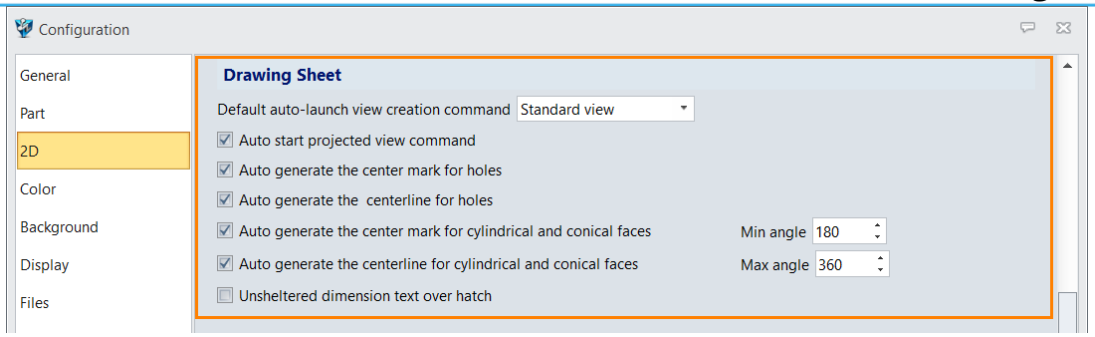


Figure 6 Configuration

- ***Tools Ribbon Tab->Setting-> Preferences***

  Settings for global drawing sheets, including units, mass units, grid spacing, projection type and tolerance. Please refer to Figure 7.

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

  Set the basic attributes of the selected sheet, such as sheet name, scale, paper color, start label and associated model.

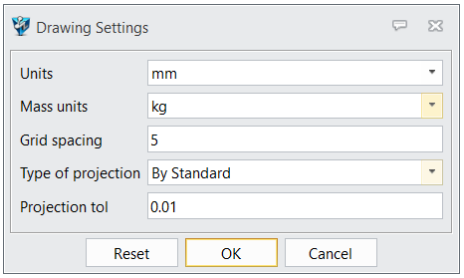


Figure 7 Drawing Settings

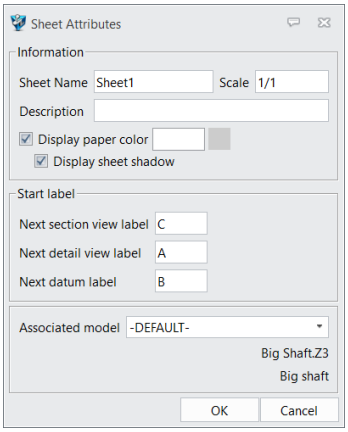


Figure 8 Sheet Attributes

- ***Sheet Manager->*** **(**on ***sheet format*** **)-> *Sheet Format Attributes***

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

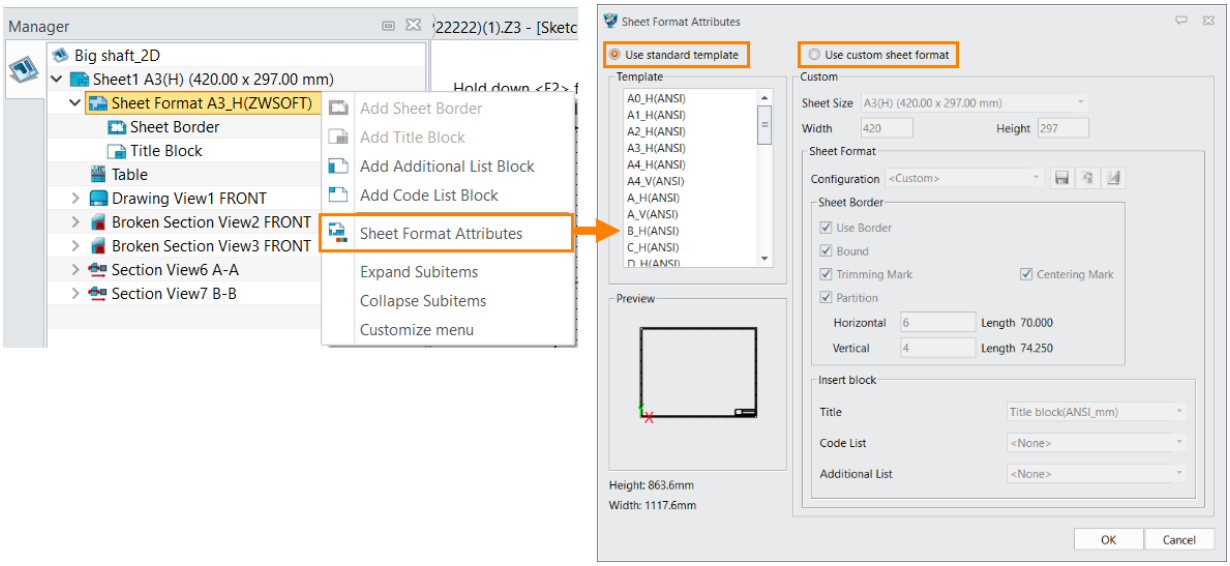


Figure 9 Redefine Sheet Format Attributes

1.4 Customize a 2D Sheet Template

Many different 2D Sheet templates are available in ZW3D, such as ANSI, ISO, GB, and DIN. But companies may have their templates to meet custom requirements.

Generally, there are three tasks to customize a 2D Sheet template.

1) Create a new 2D sheet and set its size.
2) Define the sheet format, including ***Sheet border***, ***Title block,*** and so on.
3) Define the layer attributes (if necessary).
4) Define the style attributes (if necessary).

Task 1: Create a new 2D sheet

STEP 01 Confirm which template you're using from ***Object template file*** in ***Configuration->Files.***

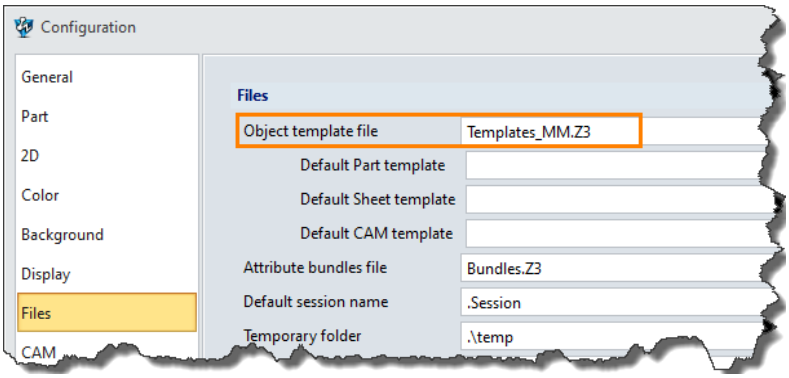


Figure 10 Configuration

STEP 02 Open the 2D drawing template file from ***File->Templates_MM.Z3***, then create a new 2D sheet with the default template.

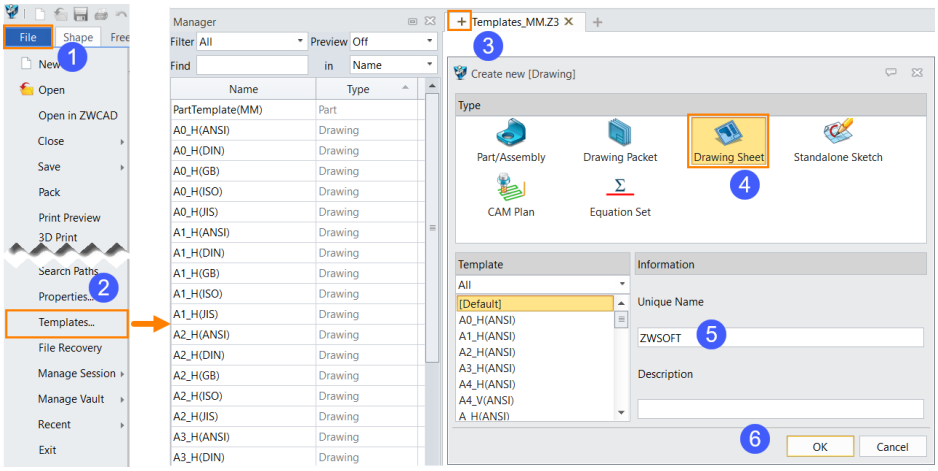


Figure 11 Create a New 2D Drawing

Task 2: Define the Sheet Format

STEP 01 In the ***Sheet Manager***, right-click on ***Sheet Format***, then select ***Sheet Format Attributes*** to invoke the dialogue box. After that, you can do a series of customizations as follows:

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

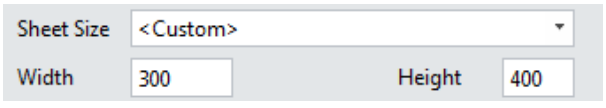


Figure 12 Sheet Size

Sheet Border: Define the values of the ***Top***, ***Bottom***, ***Left***, ***Right*** margins of the ***Sheet Border*** and the ***Partition*** values of ***Trimming Mark*** and ***Centering Mark***.

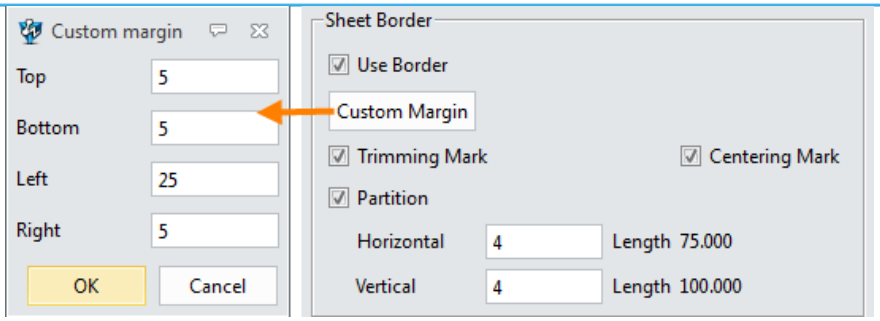


Figure 13 Sheet Border

***Insert Block*:** Select a standard or predefined block type which is stored in the corresponding template file. Or you can just skip this step and do it later.

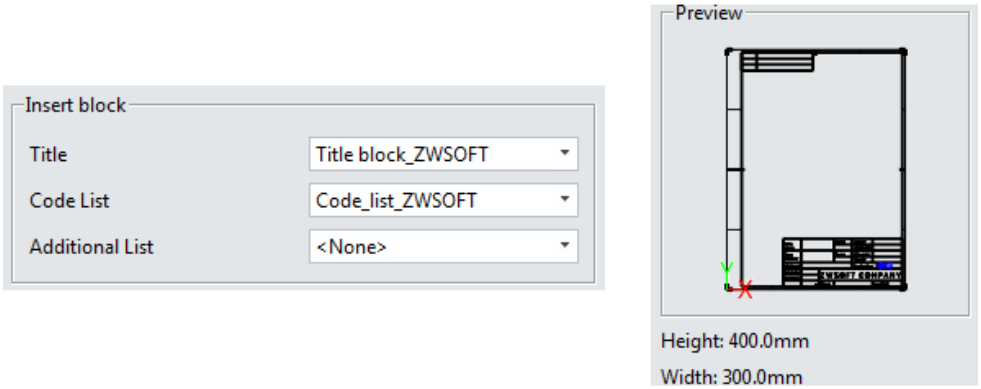


Figure 14 Insert Block

Notes: *All 2D sheet template files are in the resource folder of the installation directory.*

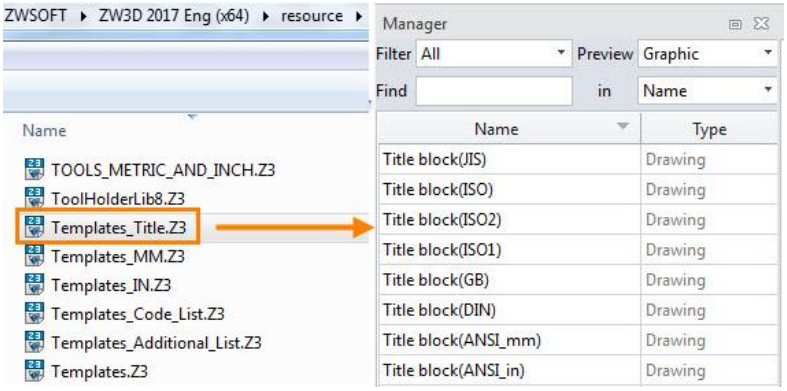


Figure 15 2D Drawing Templates

How to add the customized block (Title Block, Additional List Block, or Code List Block) in the template file?

Here we will take Title Block as an example to show you the steps:

STEP 01 Open the corresponding template file (Template_Title.Z3).

STEP 02 Create a new drawing sheet, and name it (for example, "***Title block_ZWSOFT***").

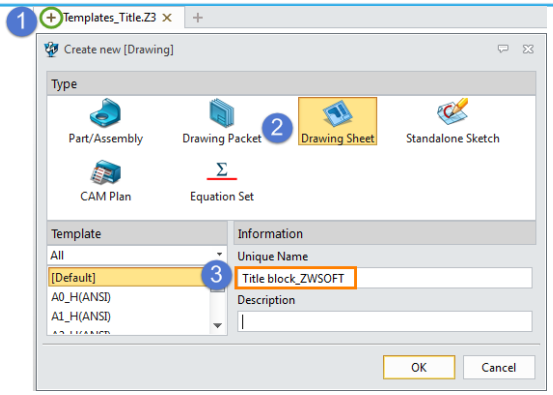
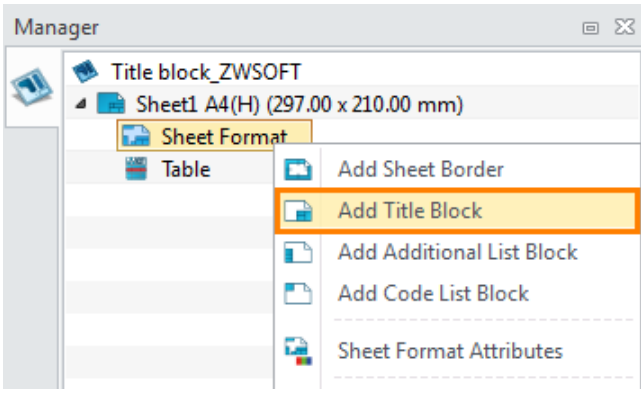


Figure 16 Create a New Drawing in the Template

STEP 03 In the ***Sheet Manager***, right-click on ***Sheet Format***, then click ***Add Title Block*** option. Then draw the title block in the sketch level.



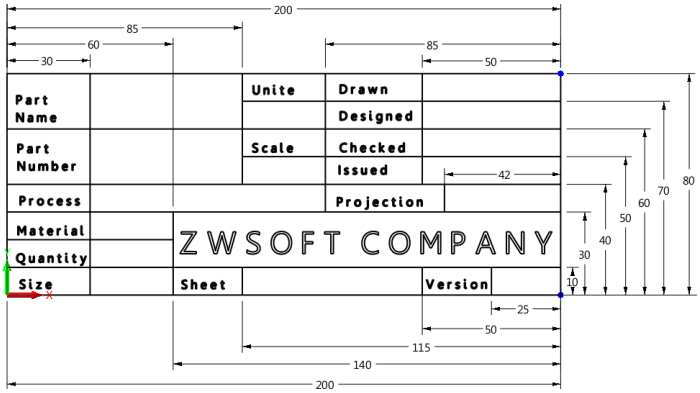


Figure 17 Add Title Block

Figure 18 Draw a Title Block in the Sketch Level

STEP 04 Link part attributes with the title block using the ***Text*** command.

With the ***Variable Browser***, you can browse all available variables, then select the necessary one to embed part attributes in text strings.

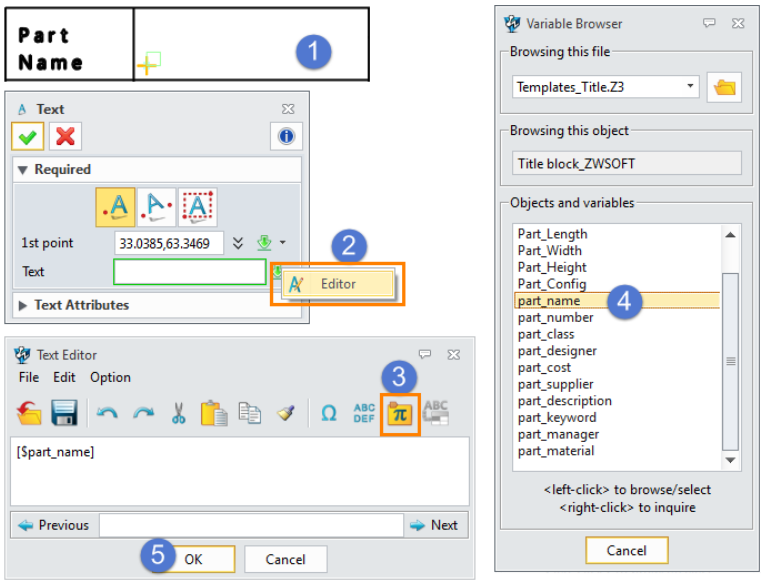


Figure 19 Embed Part Attributes

STEP 05 After defining the link relationship, you will get the following result.

Part Name	[$part_name]	Unite	Drawn	
			Designed	[$part_designer]
Part Number	[$part_number]	Scale	Checked	[$part_manager]
		[$Sheet_scale]	Issued	
Process			Projection	[$Sheet_projection]
Material	[$part_material]	ZWSOFT COMPANY		
Quantity				
Size		Sheet	[$Sheet_sequence] / [$Sheet_amount]	Version

Figure 20 Title Block in the Sketch Level

Notes: *Some texts might seem overlapped when you are defining the part attributes. But it will not affect the results.*

STEP 06 Click the ***Exit*** button to return to the 2D sheet, then you will find the completed customized title block.

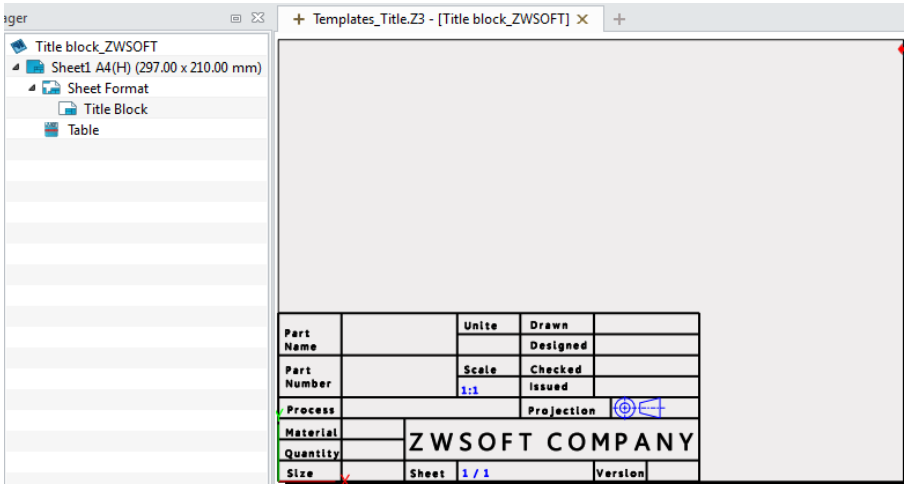


Figure 21 Customized Title Block

Task 3: Define the layer attributes

Click the ***Layer Manager*** from the ***Document Aware Toolbar***, then create new layers and set their parameters, as shown in the image below.

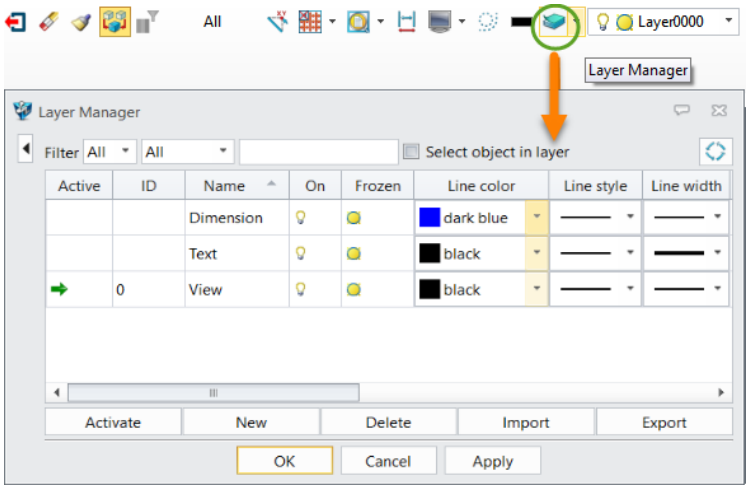


Figure 22 Layer Attribute Definition

Task 4: Define the style attributes

STEP 01 Click the ***Style Manager*** from the ***Tool Ribbon Tab*** or ***Sheet Manager***.

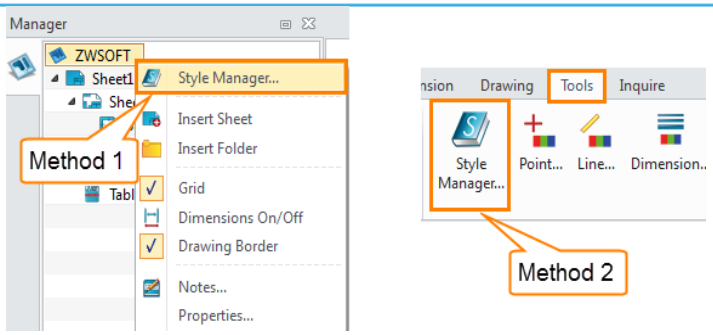


Figure 23 Invoking the Style Manager

STEP 02 Create a new style, for example, linear dimension. Then, set and save the parameters, such as ***Precision*** and ***Text Position***.

STEP 03 Double click the new style to make it the default style.

STEP 04 Save the customized template.

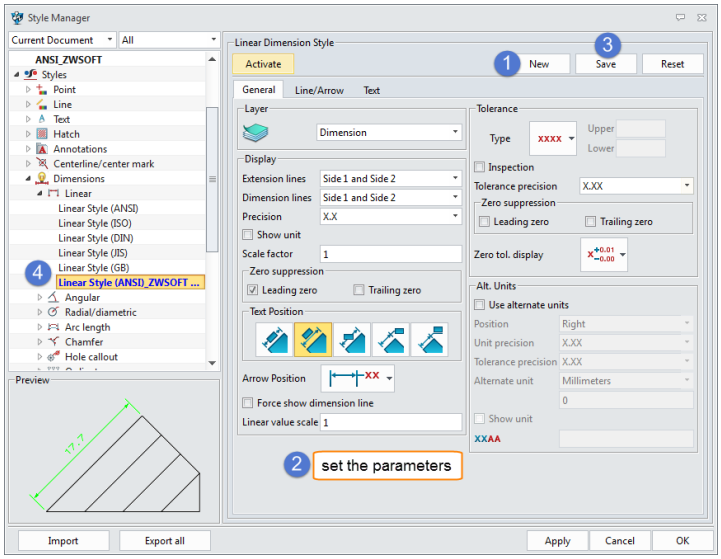


Figure 24 Create a New Style

2 Create Views

2.1 View Layout

Layout Ribbon Tab->View ->Layout

This command can help you generate a view layout of a 3D part, where up to 7 views can be created.

STEP 01 Select the part from a file, then define the views and other parameters, such as ***Label*** and ***Lines*** attributes.

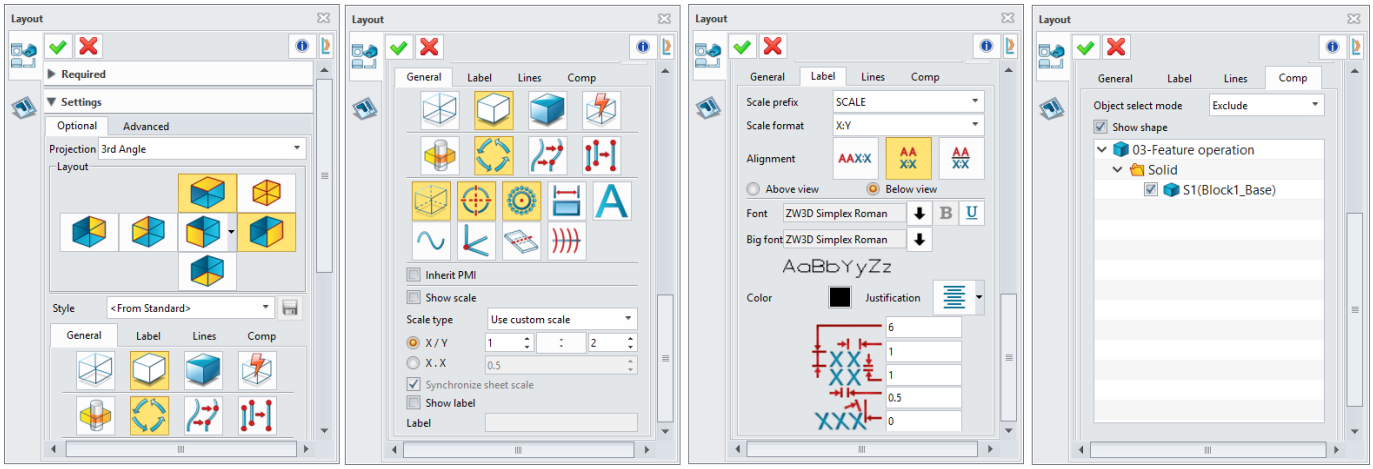


Figure 25 Define the Layout Parameters

STEP 02 The result is shown in the image below.

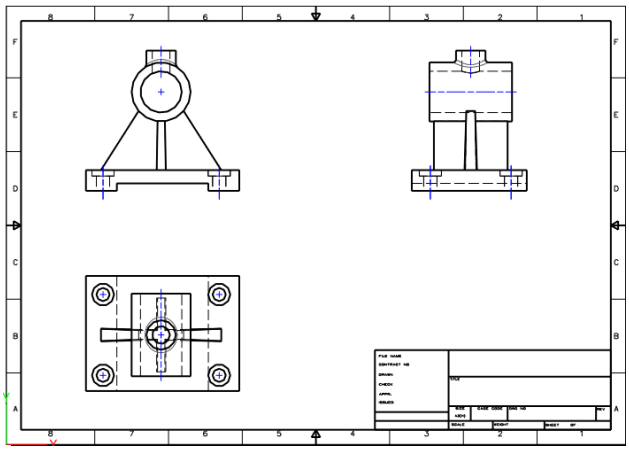


Figure 26 View Layout

2.2 Standard View

Layout Ribbon Tab->View->Standard

This command can help you create the standard view of a 3D part.

STEP 01 Select the part from a file, select a view from the drop-down list, such as ***Top view***, ***Isometric view***, then define its location, as shown in the image below.

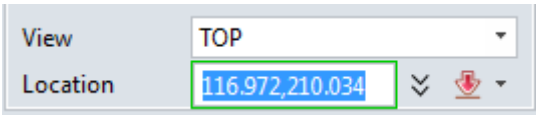


Figure 27 Standard View Definition

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

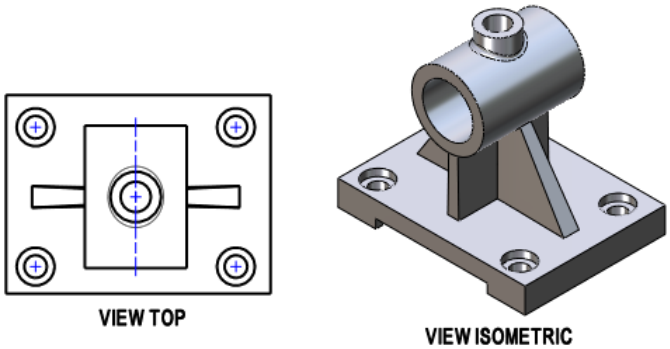


Figure 28 Standard View

General View Attributes

(1)  : ***Show Hidden lines/Center lines/Threads***

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

If ***Show dimension*** *is on*, then you'll get the result as shown in the image below.

Notes: *Sometimes, the default dimension positions are not optimal. So, it is necessary to adjust them manually.*

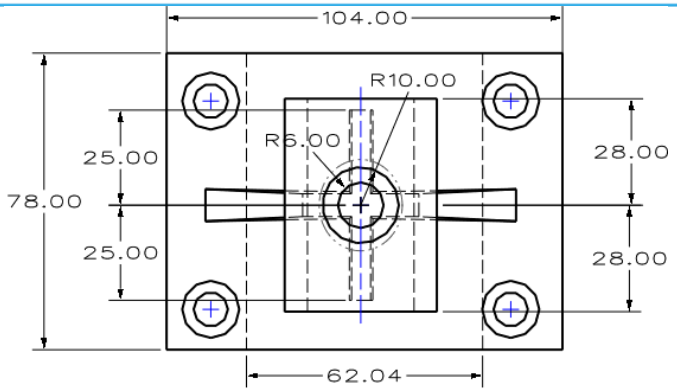


Figure 29 Dimensions from the Part

(3) ***Inherit PMI***: Check this option to show the available PMI in related views.

(4) ***Show scale*** and ***Label***: Refer to the image below to set the parameters and display attributes.

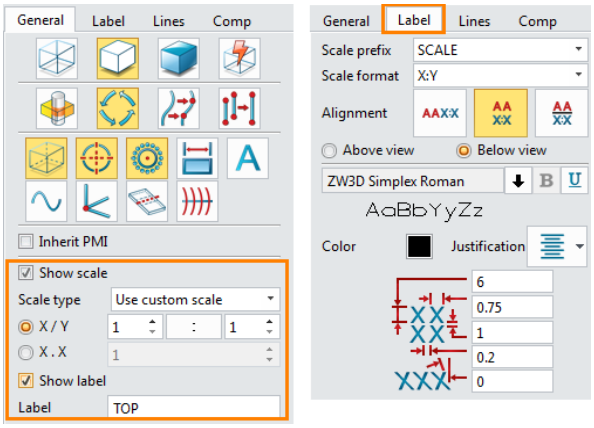


Figure 30 The setting of View Scale and Label

(5) **Set line attributes**: Set the different line attributes (such as ***Color***, ***Width***, and ***Layer***) for each kind of line.

(6) **Set component visibility**: When defining the standard view, you could hide some components or inherit the component visibility from part.

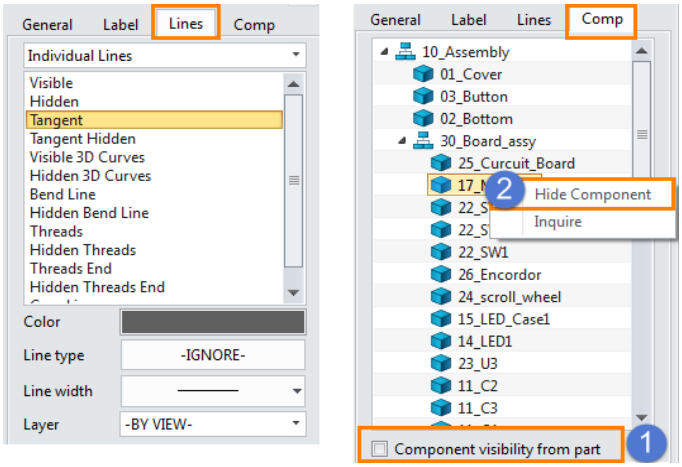


Figure 31 Line Attributes and Component Visibility

2.3 Projection View

Layout Ribbon Tab->View->Projection

Create a view projected from another existing view.

STEP 01 Select the ***Base view*** and define the ***Location*** of the projection view.

STEP 02 Set the ***Projection*** (1st or 3rd angle) and ***Dimension type***.

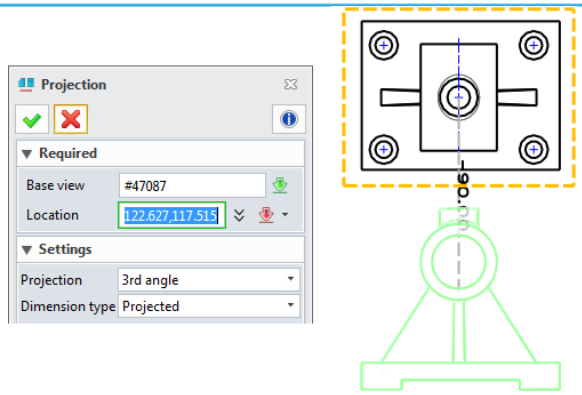


Figure 32 Projection View

2.4 Auxiliary View

Layout Ribbon Tab->View->Auxiliary

Auxiliary view is a projection view that is perpendicular to an edge of another layout view. Normally the edge should be oblique. Hence, the horizontal or vertical edge is not suitable as the reference line.

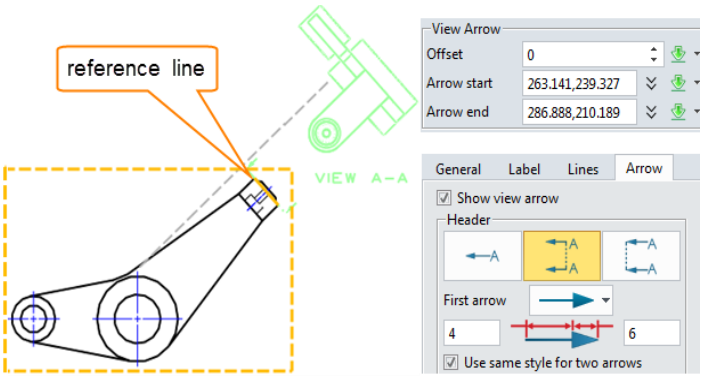


Figure 33 Auxiliary View

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 the ***Arrow*** attribute, as shown in the above image.

2.5 Full Section View

Layout Ribbon Tab->View->Full Section

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

STEP 01 Select the ***Base view***.

STEP 02 Pick points to define the position of section.

Notes: *If two points are picked and full section line is through the model, the full section view will be created. If more points are picked, the stepped section view will be created. A comparison is shown in Figure 34.*

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

STEP 04 Set other parameters if necessary.

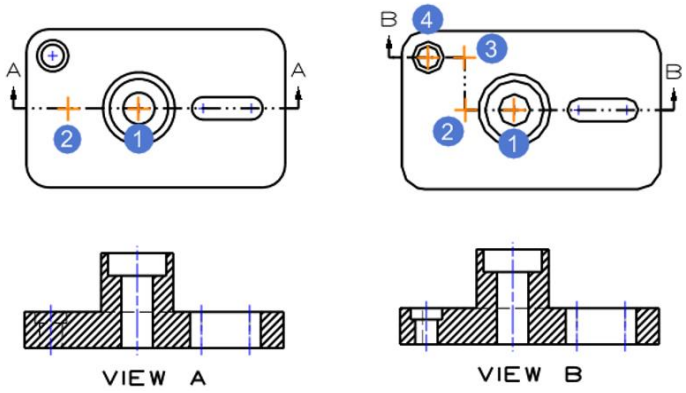


Figure 34 Full Section View and Stepped Section View

- ***Section Methods***

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

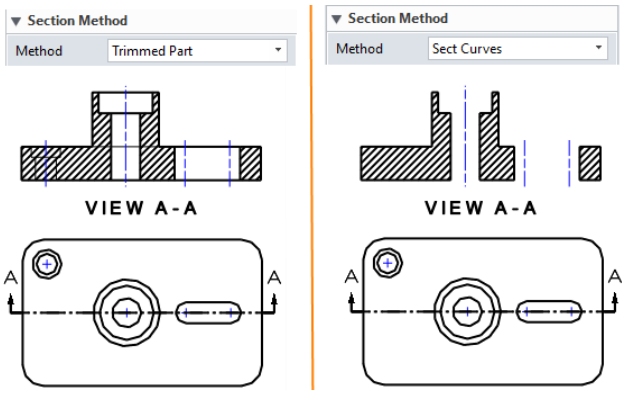


Figure 35 Section View—Section Methods

- ***Section depth***

After defining the section lines, you can use the ***Depth*** option to further clip the model from the final section view, then get the preferred section view. This option is available in ***Full section*** and ***3D named section*** views.

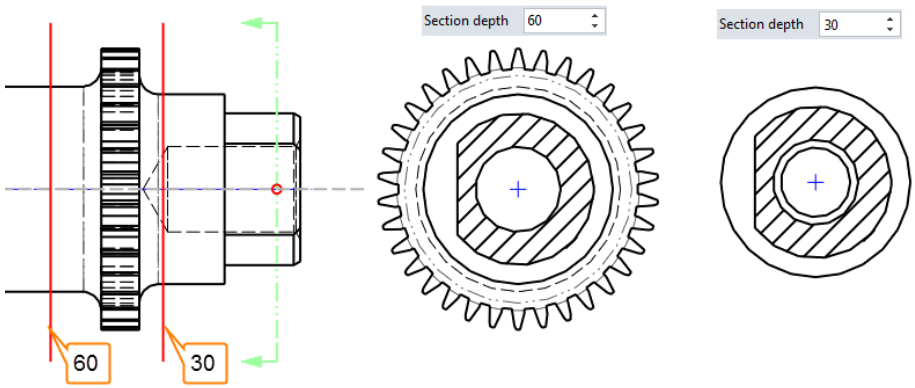


Figure 36 Section Depth

- ***Section Options***

In the ***Part Attributes*** dialogue box, the component section and hatch states can be defined, as shown in the image below.

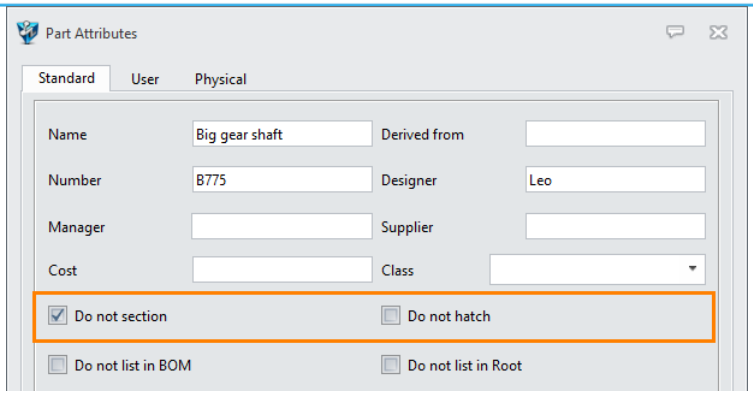


Figure 37 Part Attributes

When creating a section view of the assembly, these attributes can be inherited, as shown in the left image of Figure 38. If you want to redefine the component section and hatch states, uncheck the section options and right-click on the component, as shown the right image of Figure 38.

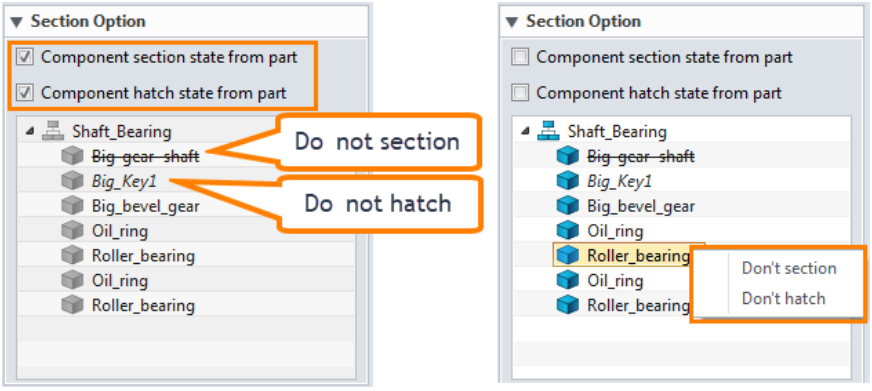


Figure 38 Section Option for an Assembly

2.6 Aligned Section View

Layout Ribbon Tab->View->Aligned Section

This command can help you 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.

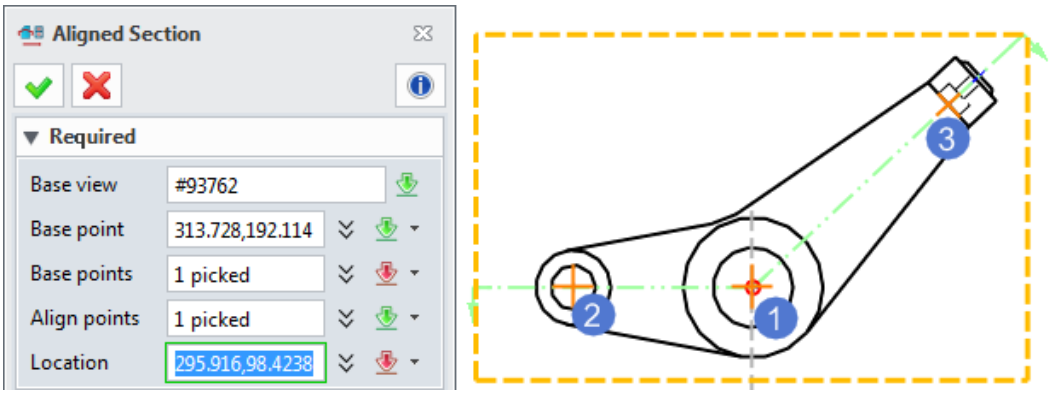


Figure 39 Define the Aligned Section Line

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

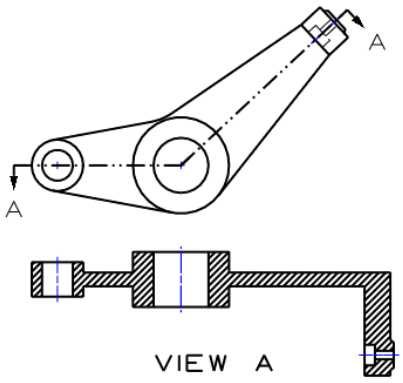


Figure 40 Aligned Section View

2.7 3D Named/Bent Section View

Layout Ribbon Tab->View->3D Named Section/Bent Section

This command can help you insert a named section created in the part. The ***Named Section*** curve 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 an angle of 90 degrees. The function is similar to the stepped section view.

STEP 01 *Draw a sketch.*

STEP 02 Follow ***Wireframe Ribbon Tab->Curve->Named Section*** to create a ***Named Section*** curve.

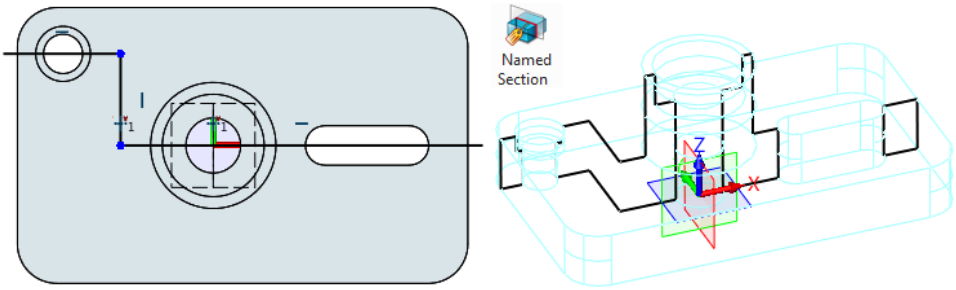


Figure 41 Create a Named Section

STEP 03 In the 2D sheet level, use the ***3D Named Section*** command to create the right section view. If ***Named Section*** curve has been created in the part, then the ***3D name*** option will be activated. The ***Named Section*** curve can be selected from the list.

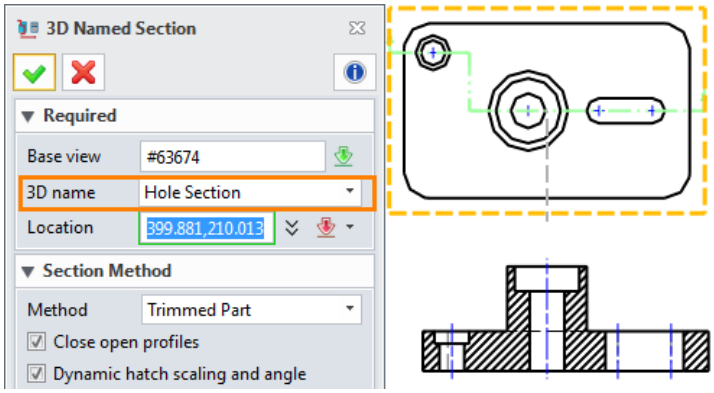


Figure 42 3D Named Section View

- ***Bent Section*** command is used for the sketch that consists of lines with bend points at an angle of degrees other than 90. The operations of creating a ***Bent Section*** view are the same as those of creating a ***3D Named Section*** view.

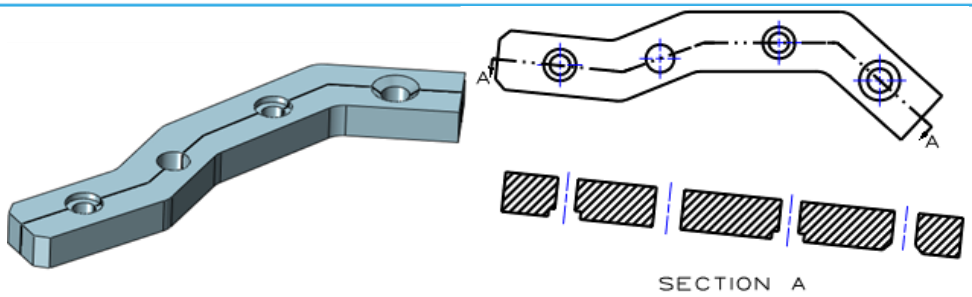


Figure 43 Bent Section View

2.8 Isometric Section View

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

STEP 01 Draw a sketch and create a ***Named Section*** curve.

STEP 02 Use the ***Isometric Section*** command to create the needed views, as shown in Figure 44.

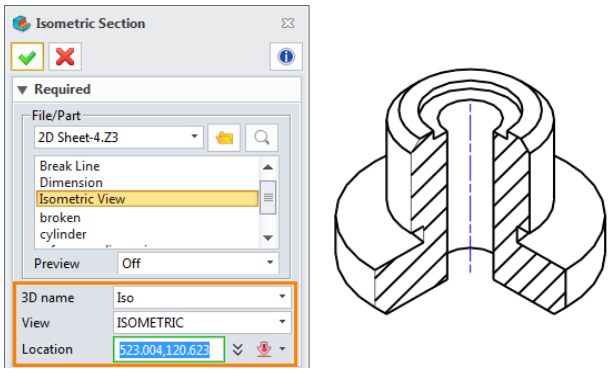


Figure 44 Isometric Section View

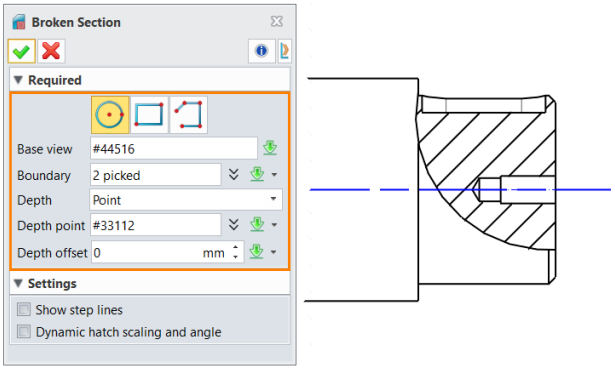


Figure 45 Broken Section View

2.9 Broken Section

Layout Ribbon Tab->View->Broken Section

This command can help you generate the broken section view of an existing view, as Figure 45 above shows.

STEP 01 Select the boundary shape type as circular, rectangle or polyline, and then select the ***Base view***.

STEP 02 Define the boundary of the ***Broken section*** view.

STEP 03 Define the ***Depth type*** and ***Depth point***.

STEP 04 Define the ***Depth offset*** value if necessary.

- ***Section Plane type:*** This type is similar to ***Section View***. Select a projected view to be the ***Depth view***, then pick a location on that view to be the Depth point for the section. You can also define additional ***Offset points*** if necessary (Similar to the stepped section view).

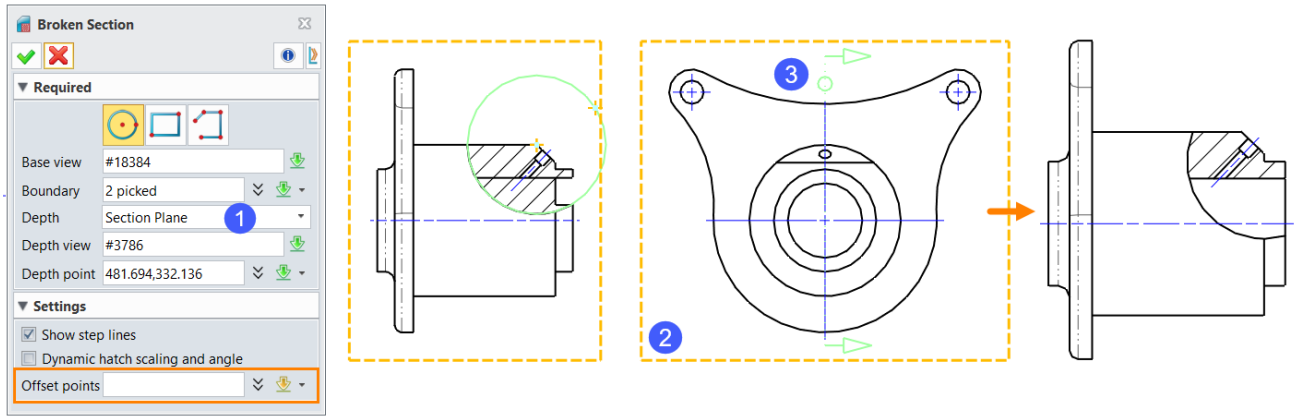


Figure 46 Section Plane Type

- ***3D Named type:*** The operations are like those of creating a ***3D Named Section.***
- ***Show step lines:*** If you are using the ***Section Plane*** type and select additional ***Offset points***, then you can check this option to show the step lines.
- ***Dynamic hatch scaling and angle:*** When this option is checked, the hatching scale and angle calculated based on the section curves will be used to create the hatch. If unchecked, the value entered on the Hatch Attributes Form will be used.

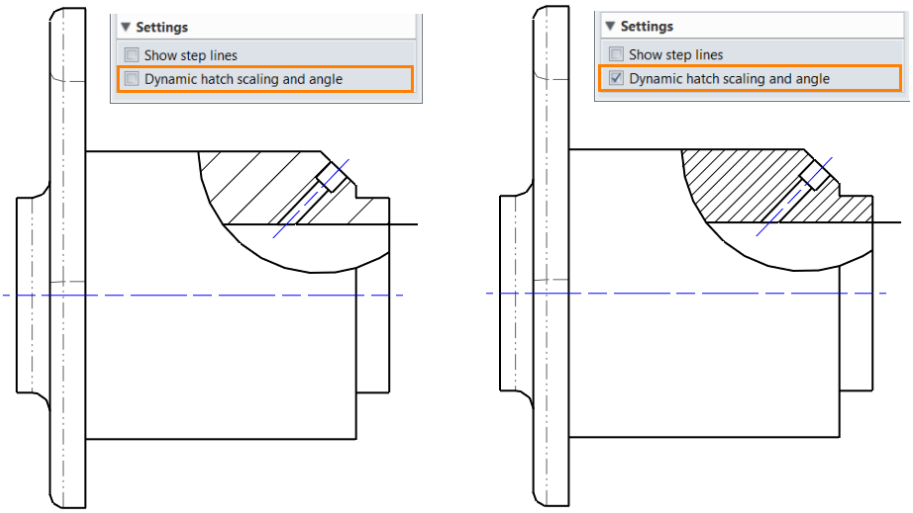


Figure 47 Dynamic Hatch Scaling and Angle

2.10 Detail View

Layout Ribbon Tab->View->Detail

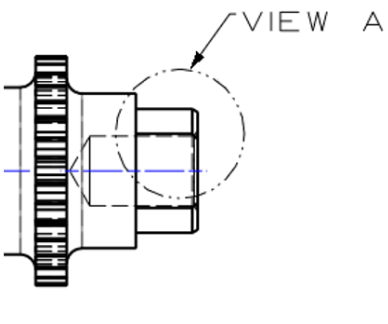
This command can help you generate a detailed view from other views.

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.

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



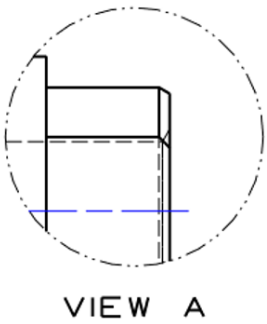


Figure 48 Detail View

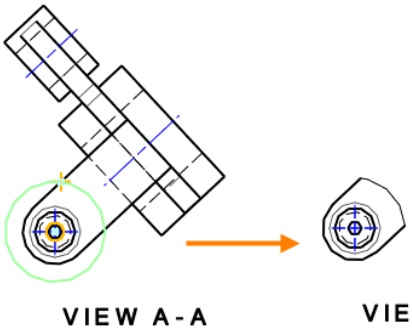


Figure 49 Crop View

2.11 Crop View

Layout Ribbon Tab->View->Crop View

This command can help you generate a partial view by trimming a view with a defined boundary.

STEP 01 Select the boundary type.

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

STEP 03 Define the boundary.

2.12 Break Line View

Layout Ribbon Tab->View->Break Line

A break line view is suitable for a long cylindrical part, for example, a shaft. The dimension value is unchanged.

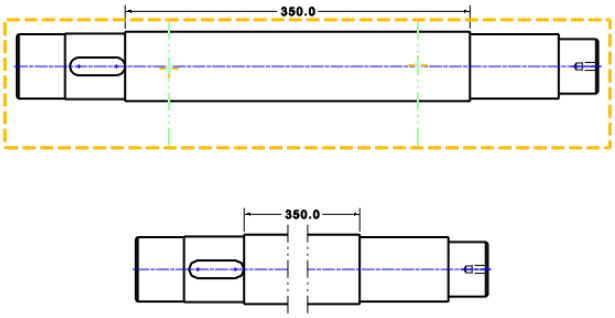


Figure 50 Break Line View

3 Edit Views

3.1 Edit View Attributes

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

Method 1: *Context Menu*

Directly right-click on the view or the view name in the ***Sheet Manager***, then select ***Attribute*** to edit the view attributes.

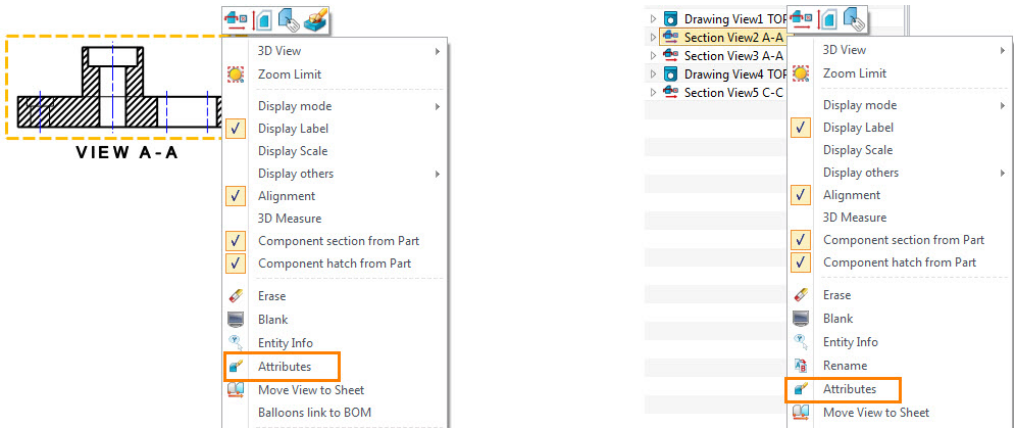


Figure 51 Edit the View Attributes from the Context Menu

Method 2: *Layout Ribbon Tab->Edit View->View Attributes*

STEP 01 Choose ***View Attributes***.

STEP 02 Select the view and middle click to confirm.

STEP 03 Reset the view attributes in the dialogue box.

3.2 Edit the View Label or Scale

Layout Ribbon Tab->Edit View->View Label/View Scale

Although all the view attributes can be modified with the aforementioned ***View Attribute***, you can edit only the view label or scale with some specific edit commands.

If you need to quickly show or hide the view label or scale, you could right-click on the view or the view name in the ***Sheet Manager***, then check or uncheck ***Display Label*** or ***Display Scale***.

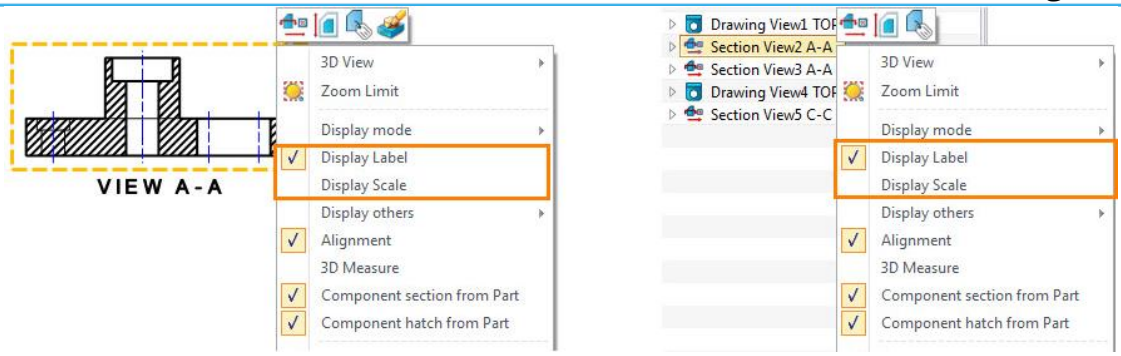


Figure 52 Display the View Label or Scale

3.3 Change Part Configurations

Layout Ribbon Tab->Edit View->Part Config

This command can help you change the view between different ***Part Configurations*** of a 3D part, as shown in the below image.

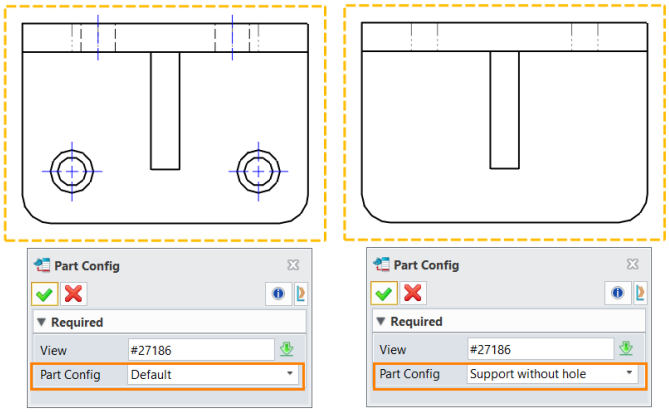


Figure 53 Part Config

3.4 Change Component Configurations

Layout Ribbon Tab->Edit View->Component Config

This command can help you change the view between default views and exploded views of the assembly.

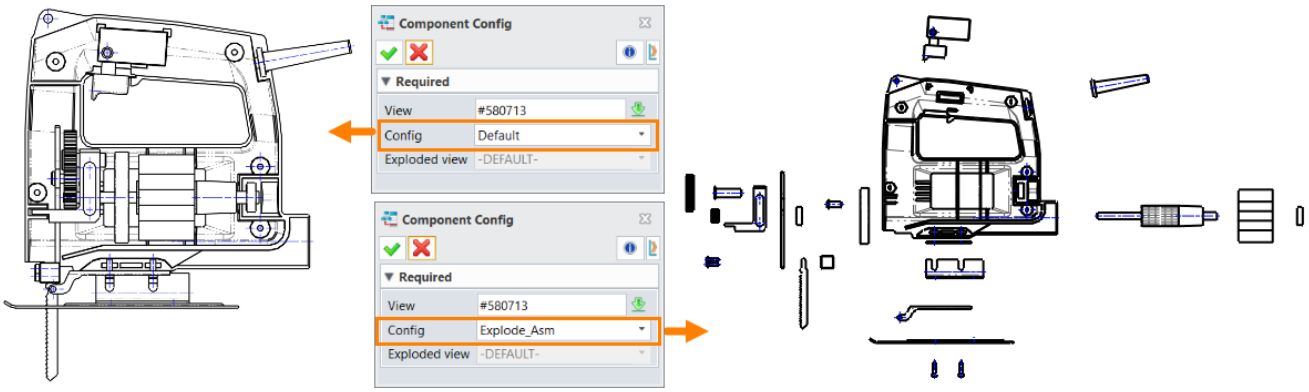


Figure 54 Change Component Config

3.5 Hide Components

Layout Ribbon Tab->Edit View->Hide Components

This command can help you hide the components as you need. Actually, it's a better way to hide the components in the ***View Attribute*** dialogue box, as shown in the image below.

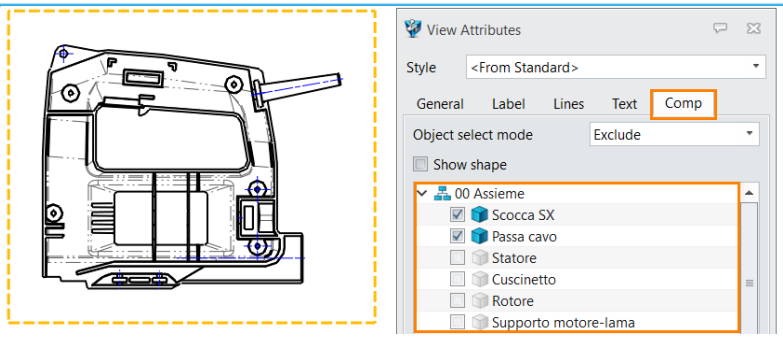


Figure 55 Hide Components in the View Attributes Dialogue

3.6 Redefine the View

After creating the ***Section View***, ***Detail View*** or ***Crop View***, you can follow the methods below to redefine the view.

Method 1: *Context Menu*

STEP 01 Right-click on the view, then select ***Redefine*** to redefine the view.

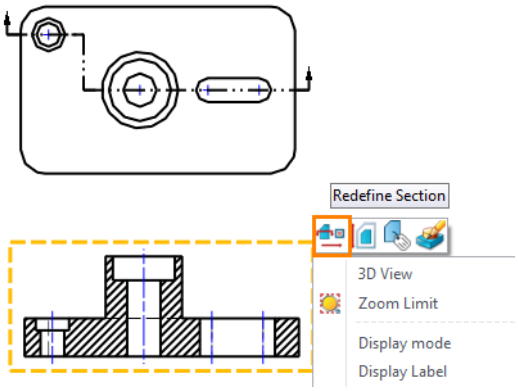


Figure 56 Redefine the View

Method 2: *Layout Ribbon Tab->Edit View->Redefine Detail/Redefine Crop View/Redefine Section*

STEP 01 Choose a suitable command from the ***Edit View*** panel.

STEP 02 Select the view to redefine.

3.7 Edit the Section Line

If the section view is created with the ***Full Section*** command, the section line can be edited after creating a view.

- **Insert the step points**

STEP 01 Right-click on the section line, then select ***Insert step***.

STEP 02 Pick a point to insert the step.

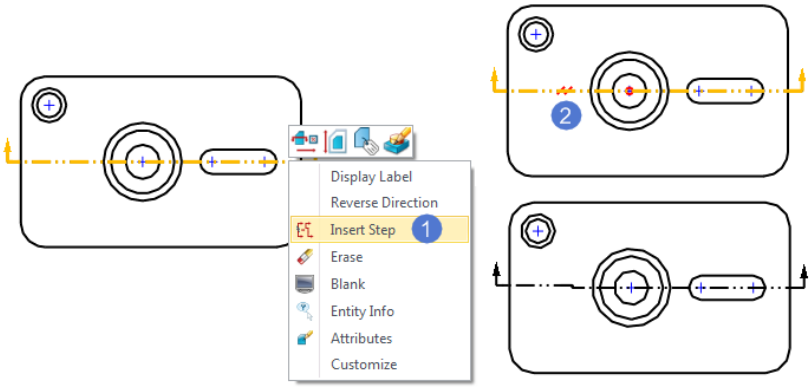


Figure 57 Insert the Step Points

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

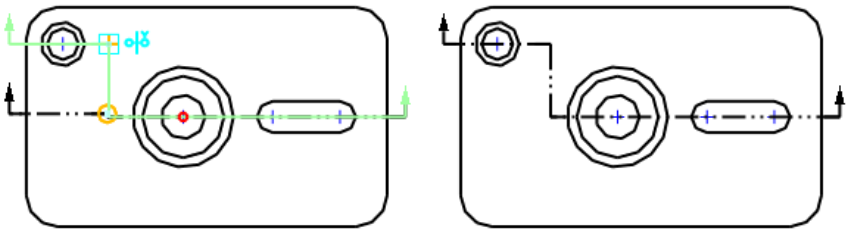


Figure 58 Drag and Drop the Insert Point

- **Reverse the direction of the section line**

Right-click on the section line, then select ***Reverse Direction*** to reverse its direction. The projection view will be automatically updated.

- **Remove the step point**

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

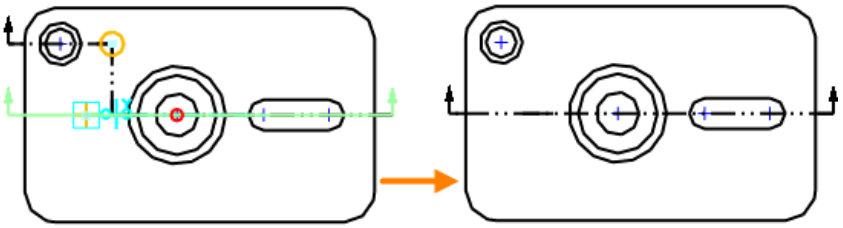


Figure 59 Remove the Step Point

3.8 Move the View

By default, not every view can be dragged to any position you want. For example, the ***Projection View*** and ***Section View*** can only be moved along the projection direction. Therefore, if you want to move them to any position, please follow the steps below.

STEP 01 Right-click on the view, then uncheck ***Alignment***.

Note*: *Alignment *controls the association between the view projection and the* ***Base view****.*

STEP 02 Drag and drop the view to any position.

STEP 03 If you want to regain the original association, check ***Alignment*** and move the view again.

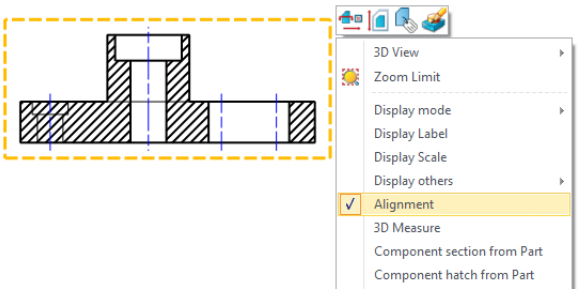


Figure 60 Uncheck Alingment to Move the View

3.9 Rotate the Base View

Layout Ribbon Tab->Edit View->Rotate View

This command can help you rotate the Base view. Select the Base view, then adjust the parameters in the pop-up edit toolbar.

: The view can be rotated by right-clicking when this icon is activated.

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

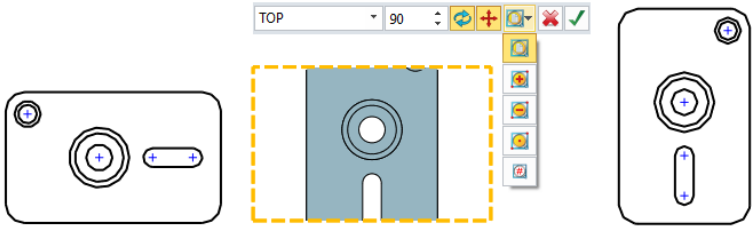


Figure 61 Rotate the Base View

4 Create Dimensions

4.1 Set the Dimension Attribute

It's a good design habit to set the dimension attribute before creating the dimensions.

STEP 01 Invoke the ***Style Manager*** and edit the dimension style.

Right-click on the 2D sheet object in the ***Sheet Manager*** or directly select ***Dimension Attribute*** from the ***Tools Ribbon Tab***.

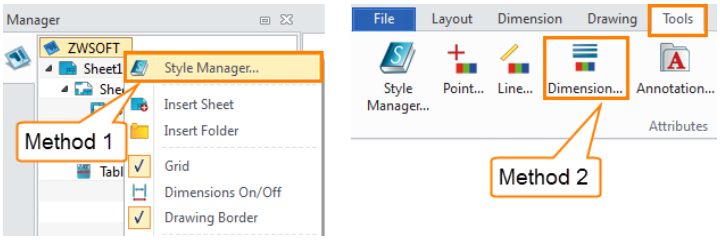


Figure 62 Dimension Style Manager

STEP 02 Double-click on a dimension style, for example, a linear one, to set it as currently activate style (or create a new dimension style by "***New***" and set it as the active style).

STEP 03 Specify the necessary parameters and click the ***Save*** button to save the modifications.

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

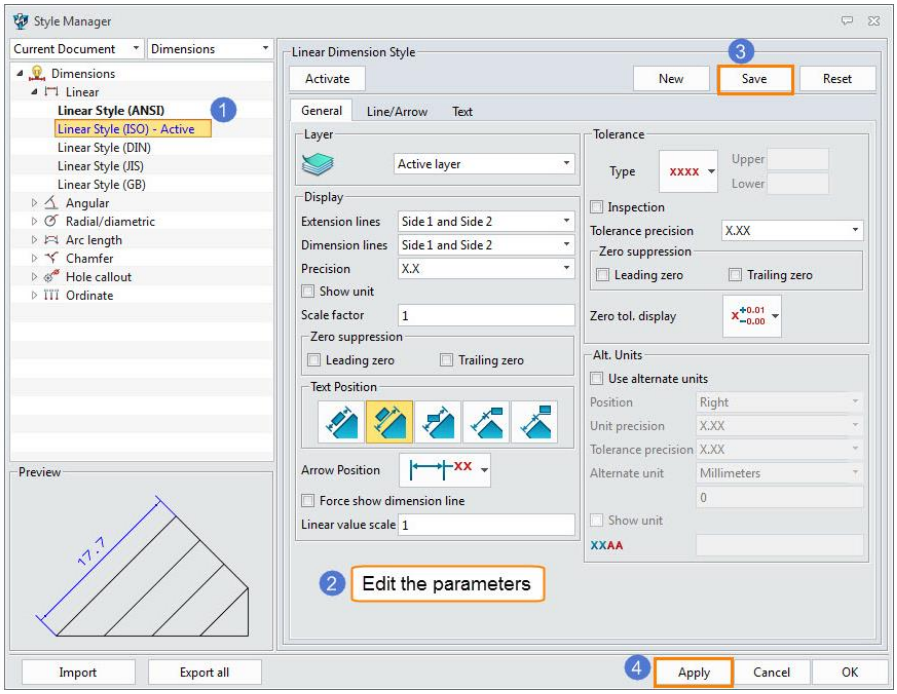
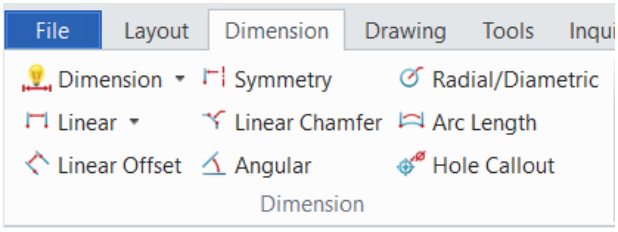


Figure 63 Create a New Style and Define the Dimension Style

4.2 Add Basic Dimensions

You could create the required dimensions with the quick ***Dimension*** tool or other dimension tools.



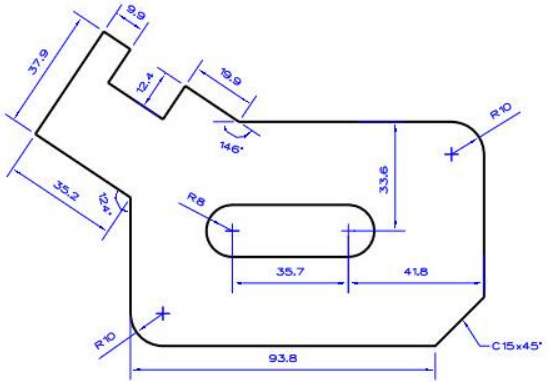


Figure 64 Dimension Tools and View with Dimensions

4.3 Auto Dimension

Dimension Ribbon Tab->Dimension->Auto Dimension

This command can help you create dimensions in batches.

STEP 01 Select the target view and define a global origin.

STEP 02 Select the types of entities that you are adding dimensions to.

STEP 03 Define the placement of the horizontal and vertical dimensions, then click ***OK*** to finish.

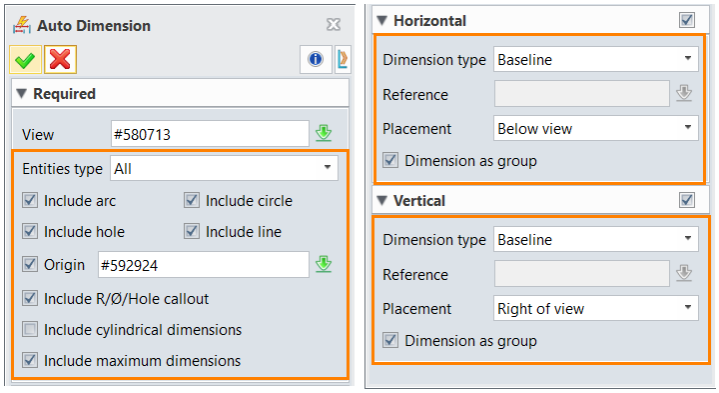


Figure 65 Settings of Auto Dimension

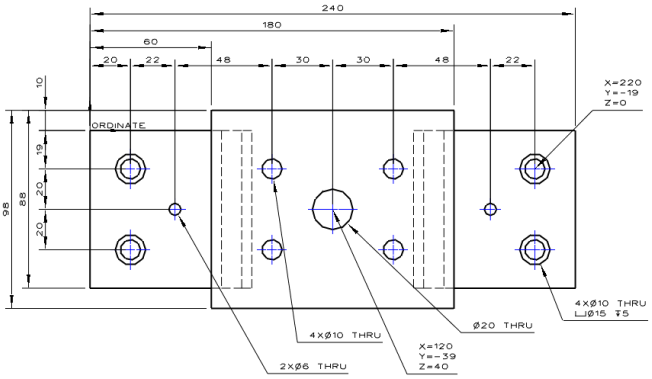


Figure 66 Results of Auto Dimension

4.4 Linear Dimension Mode

Dimension Ribbon Tab->Dimension->Linear

This command can help you create a linear dimension in various ways.

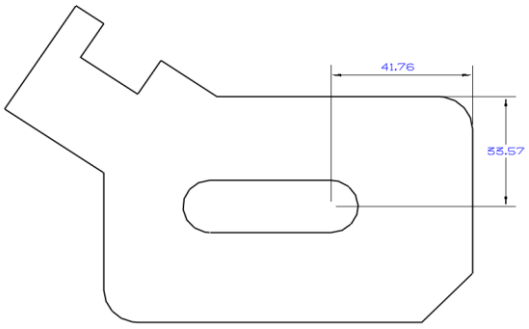


Figure 67 Linear—Horizontal & Vertical

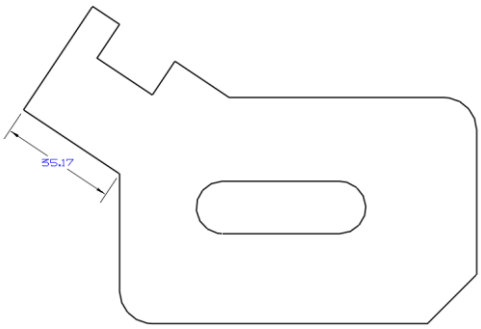


Figure 68 Linear—Aligned

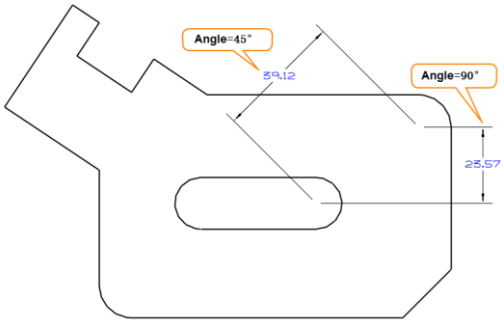


Figure 69 Linear—Rotated

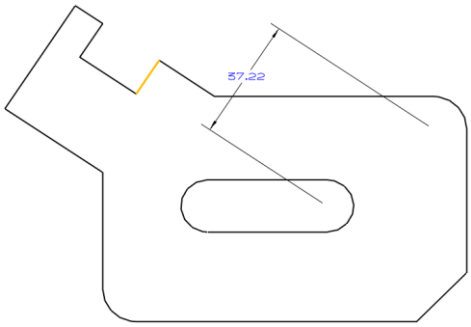


Figure 70 Linear—Projected

Dimension Ribbon Tab->Dimension->Baseline/Continuous/Ordinate

This command can help you create a 2D linear baseline/continuous/ordinate dimension group.

STEP 01 Select the first point and the second point if needed.

STEP 02 Pick a point to place the dimension text.

STEP 03 Select other points to define the dimension group.

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

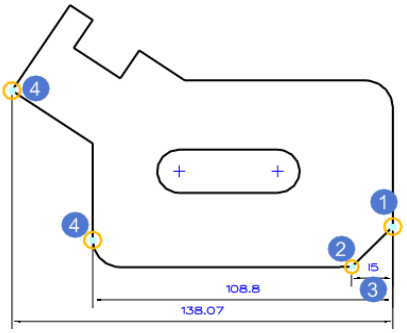


Figure 71 Baseline

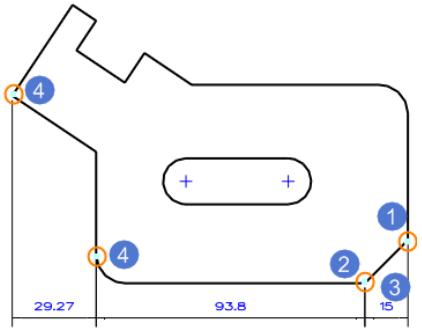


Figure 72 Continuous

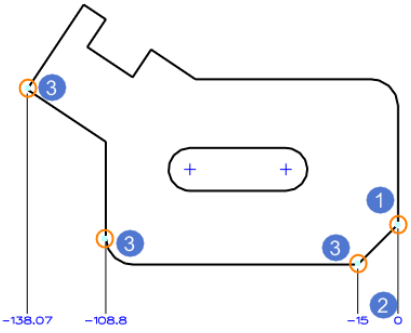


Figure 73 Ordinate

4.5 Hole Callout

Dimension Ribbon Tab->Dimension->Hole Callout

This command can help you create one or more hole callout dimensions.

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

STEP 02 Check the needed hole parameters.

STEP 03 Click ***OK*** to finish.

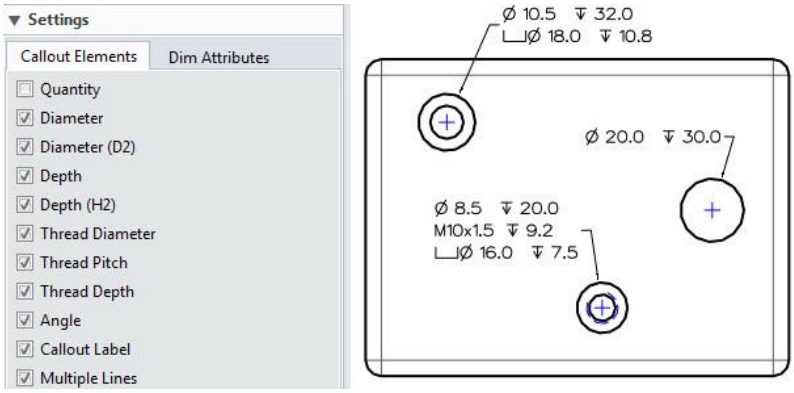


Figure 74 Hole Callout Dimensions

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

4.6 Other Dimensions

Dimension Ribbon Tab->Dimension->Linear Offset/Arc Length/Radial/Angular/Linear Chamfer/Symmetry

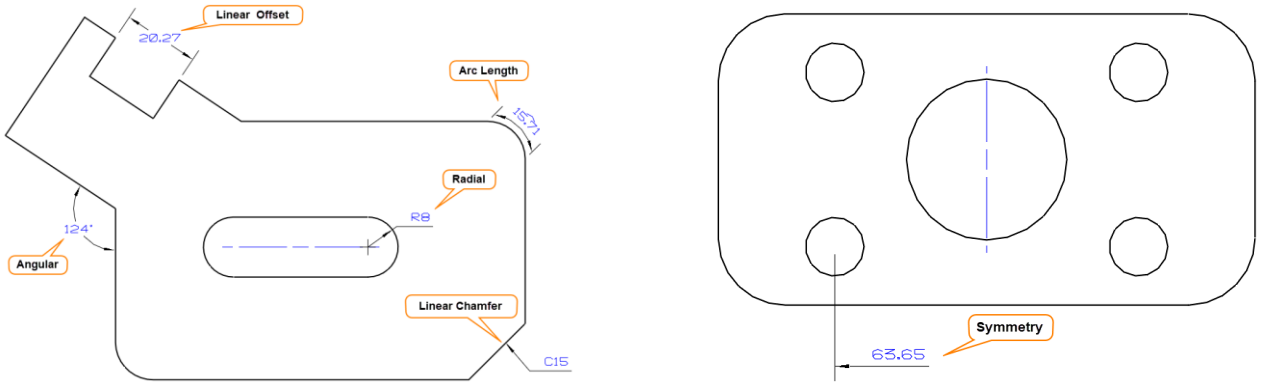


Figure 75 Other Dimensions

5 Edit Dimensions

5.1 Edit Dimension Attributes

Method 1: Right-click on the dimension, then select ***Attributes*** to edit all dimension attributes, including tolerance, dimension line/arrow, text, font, etc.

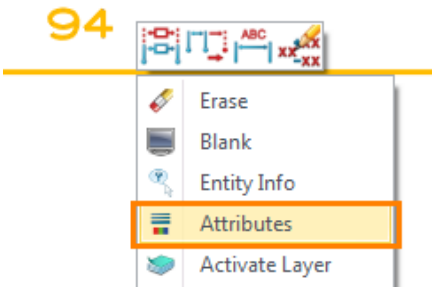


Figure 76 Edit Dimension Attributes

Method 2: ***Dimension Ribbon Tab->Edit Dimension->Dimension Quick Edit***

This command can help you quickly edit partial dimension attributes, such as dimension tolerance, precision and appended text.

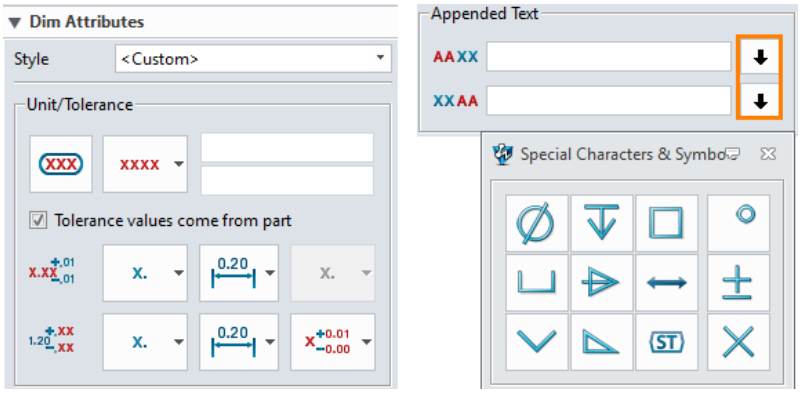


Figure 77 Dimension Quick Edit

5.2 Quick Dimension Tools

In ZW3D, quick tools for editing dimensions, including dimension symbol, tolerance and precision are available.

STEP 01 Right-click on the blank area in the ***Ribbon Tab***, then ***ToolBars*** and ***DimTool***.

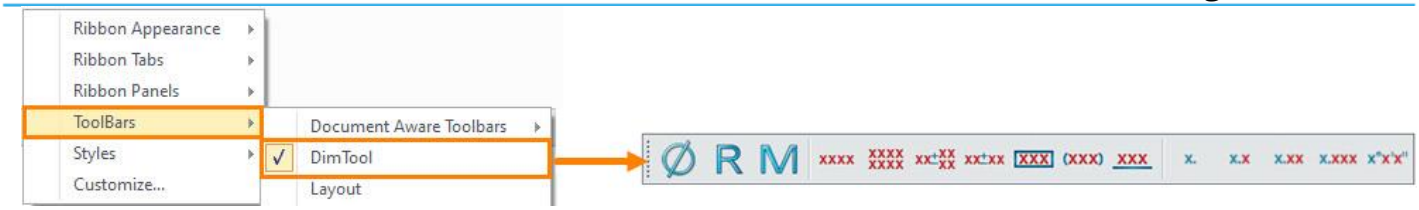


Figure 78 Quick DimTool

STEP 02 Pick any dimension, then select the command to quickly add a symbol, edit tolerance or change precision, as the images below show.

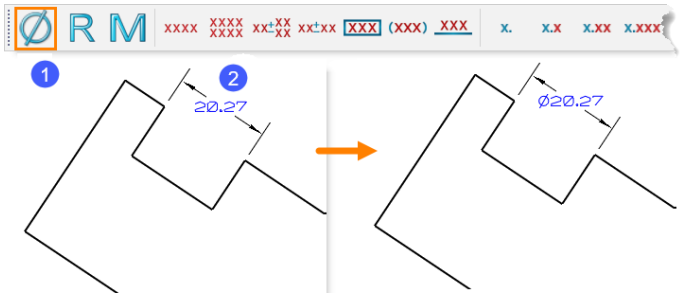


Figure 79 Add Symbol

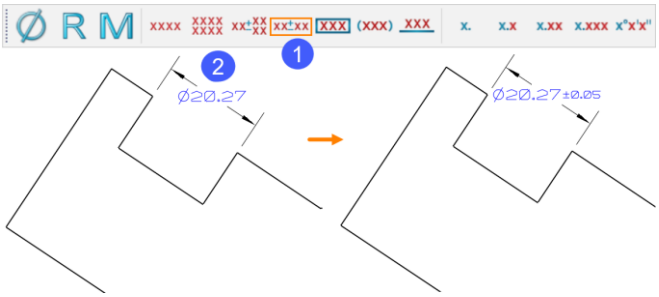


Figure 80 Add Equal Tolerance

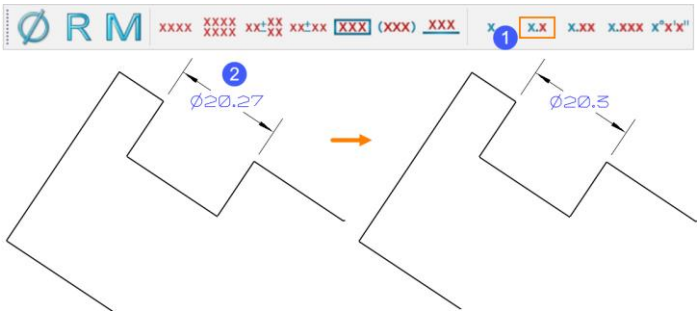


Figure 81 Change precision

5.3 Add Tolerance

Method 1: ***Dimension Ribbon Tab->Edit Dimension->Modify Tolerance***

Method 2: Right-click on the dimension, then select ***Modify Tolerance*** to edit.

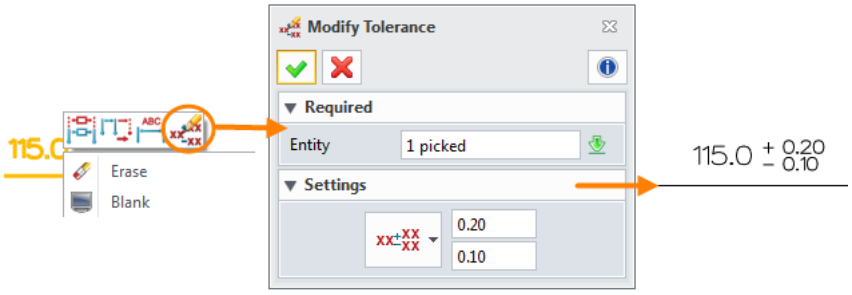


Figure 82 Method 2—Unequal Tolerance

Method 3: Right-click on the blank area in the ***Ribbon Tab***, then ***ToolBars*** and ***DimTool***.

Notes: *If you want to add the tolerance zone, just select the* ***Tolerance zone*** *type in the* ***Modify Tolerance*** *dialog, then click* **Inquire tolerance** *to inquire the tolerance zone. Select the suitable tolerance zone, then click* **OK**, *as shown in the image below.*

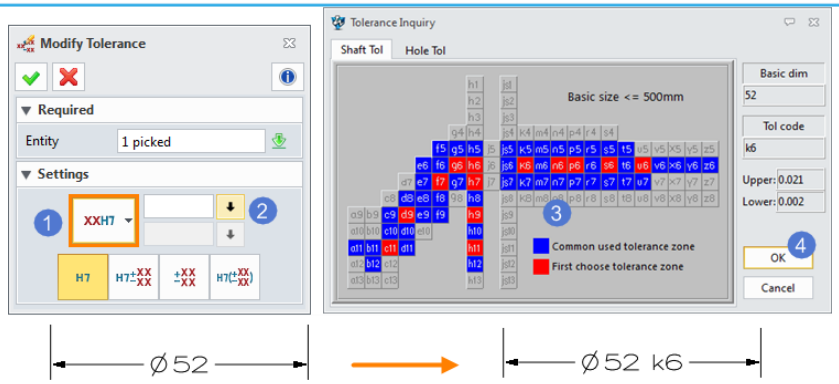


Figure 83 Define the Tolerance Zone

5.4 Toggle Reference Dimension

Dimension Ribbon Tab->Edit Dimension->Toggle Reference

This command can help you toggle a dimension to or from the reference dimension.

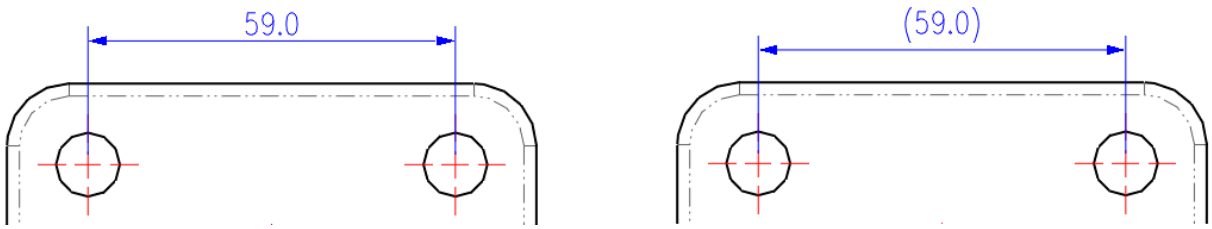


Figure 84 Toggle Reference Dimension

5.5 Edit Dimension Texts

Method 1: ***Dimension Ribbon Tab->Edit Dimension->Modify Text***

Method 2: Right-click on the dimension, then select ***Modify Text*** to invoke the Dimension Editor, as shown in the image below. You can directly input the customized text to replace the system value and add some symbols or link the variables to the dimensions.

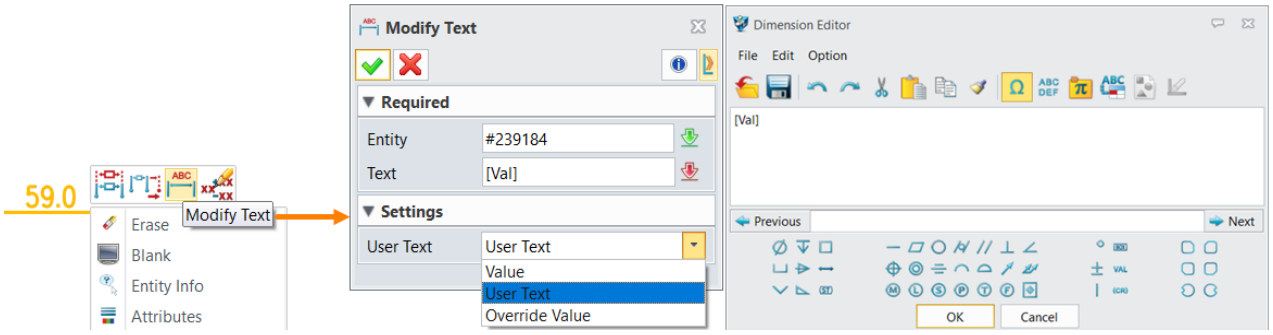


Figure 85 Modify Dimension Text

5.6 Modify Text Point

Right-click on the dimension or follow ***Dimension Ribbon Tab->Edit Dimension,*** you can select ***Modify Text Point*** to modify the location of the text point of a dimension or change the mode of text placement (automatic/manual). The dimension text and extension lines will adapt accordingly.

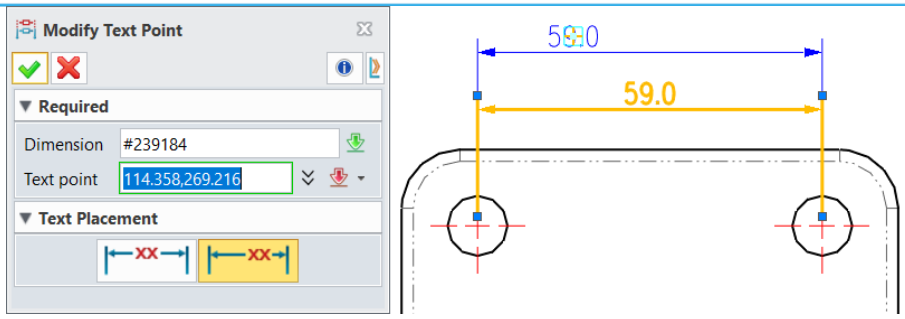


Figure 86 Modify Text Point

***Notes**: A more convenient way to change the location of the text is to drag and drop the dimension text based on the defined placement mode.*

5.7 Modify Dimension Points

This command can help you redefine a dimension. For example, if a dimension becomes invalid, you could make it valid again by redefining the dimension point with this command.

Method 1: *Dimension Ribbon Tab->Edit Dimension->Modify Dimension Points*

Method 2: Right-click on the dimension, then select ***Modify Dimension Points*** to edit.

***Notes**: This quick method to modify the dimension points or the position of texts is highly recommended: click the dimension, then drag and drop the control point or text to a new position.*

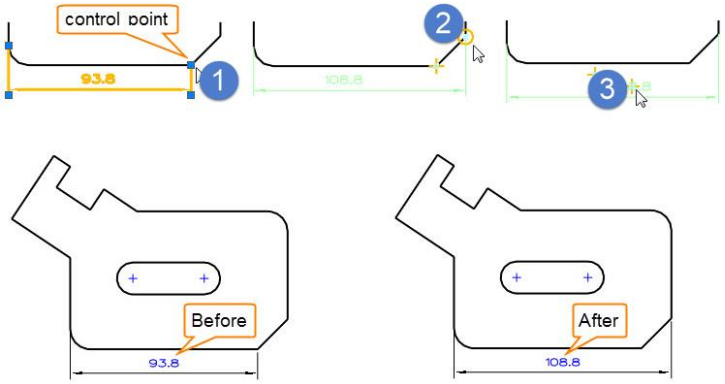


Figure 87 Modification of Dimension Points

5.8 Modify Part Dimensions

When creating a 2D view, invoke the ***Show dimensions from part*** option, and a view with part dimensions will be created.

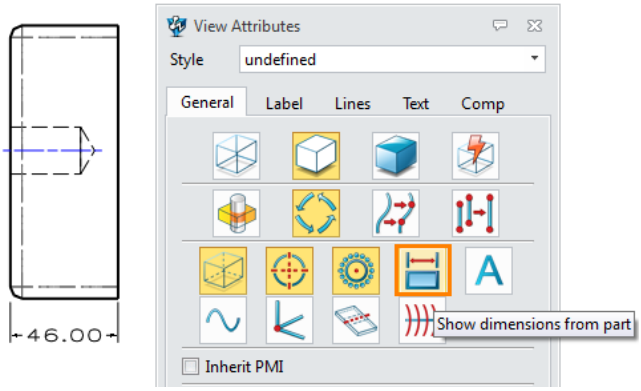


Figure 88 The View with a Part Dimension

Modifying the value of this kind of part dimensions in the 2D view will drive the part geometry. Meanwhile, other views will be updated automatically.

Method 1: *Dimension Ribbon Tab->Edit Dimension->Modify Part Dim*

Method 2: Right-click on the dimension, then select ***Modify Part Dimension***.

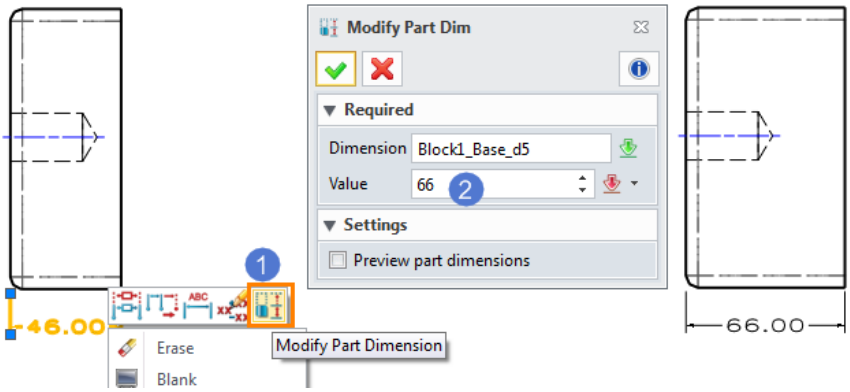


Figure 89 Modifying a Part Dimension

5.9 Align Dimensions

Dimension Ribbon Tab->Edit Dimension->Modify Text Alignment

This command can help you align one dimension with another.

Select two dimensions and the first dimension will be aligned with the second one.

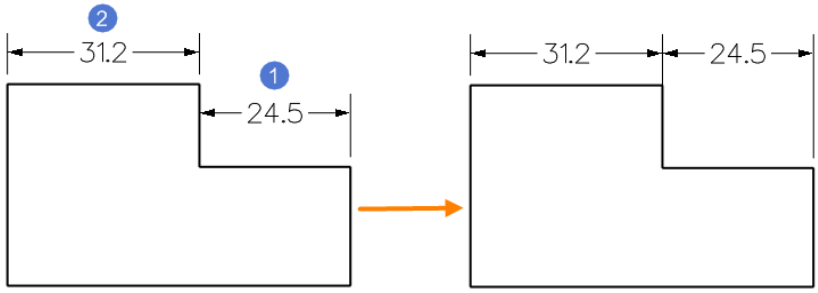


Figure 90 Align the Dimensions

5.10 Edit Dimension Groups

Dimension Ribbon Tab->Edit Dimension->Add to Group

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

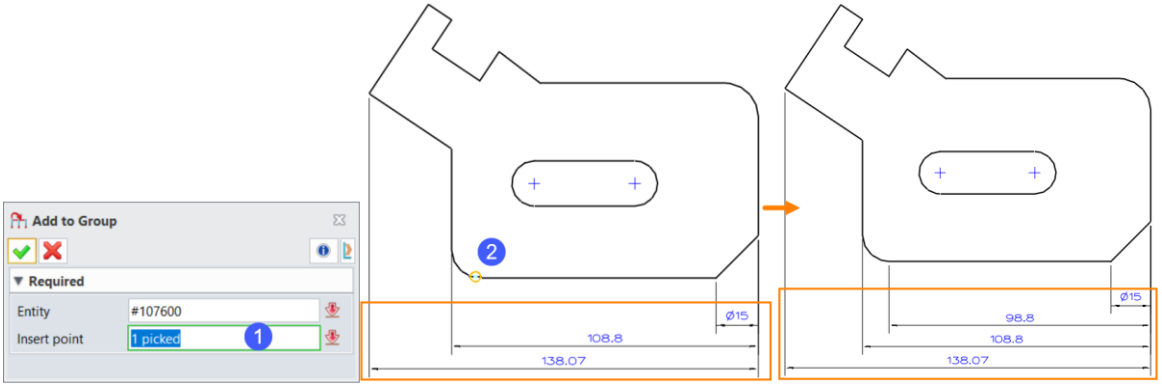


Figure 91 Add to Group

Dimension Ribbon Tab->Edit Dimension->Remove from Group

Select a dimension from a dimension group to remove.

Dimension Ribbon Tab->Edit Dimension->Explode Group

This command will turn each dimension of a dimension group into an individual object.

6 Annotations and Symbols

6.1 Label Coordinate

Dimension Ribbon Tab->Annotation->Label Coordinate

This command can help you create labels for the coordinate of the points.

STEP 01 Pick a point as the ***Origin***, then the coordinate of the point will be calculated accordingly.

STEP 02 Select one or multiple points to create the label.

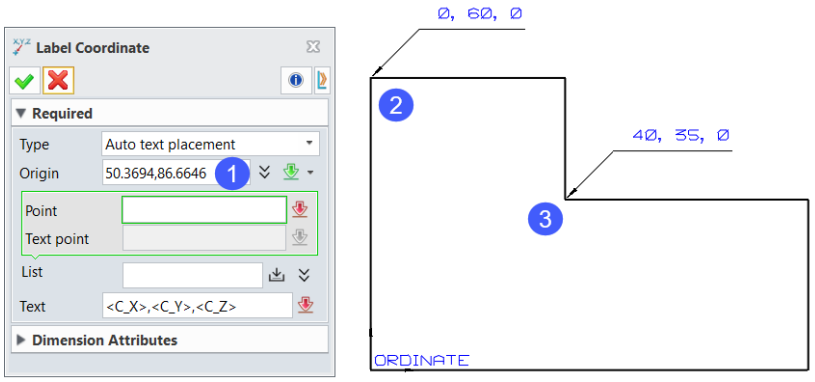


Figure 92 Label Coordinate

- ***Text:*** You can determine which coordinates will be displayed in the view, as shown in the below image.

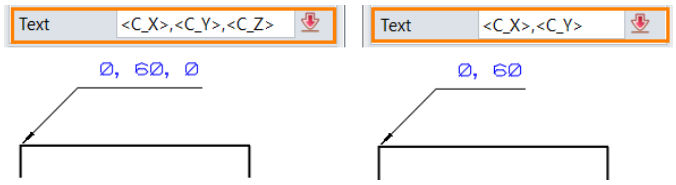


Figure 93 Text Settings

6.2 Label

Dimension Ribbon Tab->Annotation->Label

You can manually create a label with this command. The procedures will be illustrated with the example of a counter-bore hole.

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

STEP 02 Input the dimension text in the Dimension Editor dialogue box, as shown in Figure 94 below.

Click ∅, add hole dimension D1 value(32); then click ↧, add hole depth H1 value (95); line feed; click ⌴ and ∅, add hole dimension D2 value (52), then click ↧, add hole depth H2 value (5).

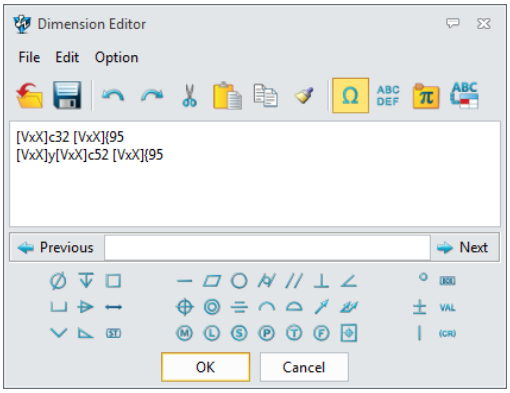
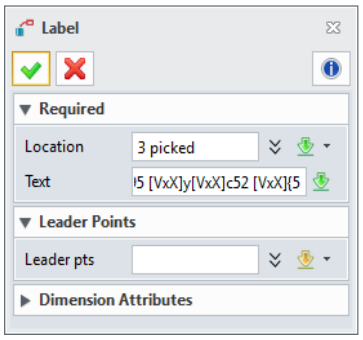


Figure 94 Define Dimension Text with Editor



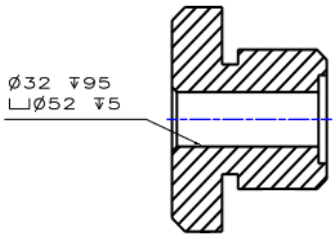


Figure 95 Hole Dimension Label

STEP 03 Click ***OK*** and the label will appear, as shown in the image above.

STEP 04 Use ***Leader pts*** to locate additional leader arrows if necessary.

6.3 Datum Feature/Target

Dimension Ribbon Tab->Annotation->Datum Feature

This command can help you create a datum feature, which can be used in Feature Control Symbol.

STEP 01 Input 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 in accordance with the requirements.

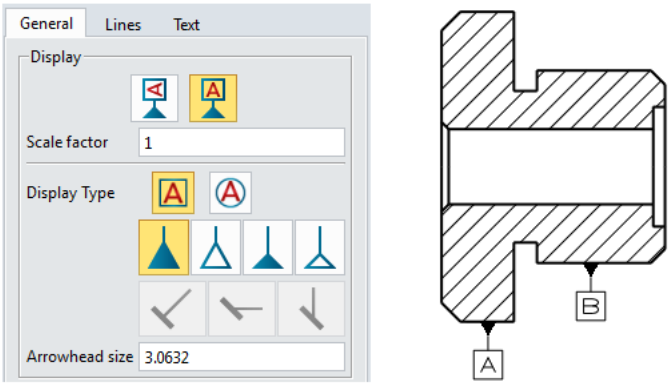


Figure 96 Datum Feature

Dimension Ribbon Tab->Annotation->Datum Target

This command can help you establish a datum by creating a datum target (a specific point, line or area on a part).

STEP 01 Determine the type of the ***Datum Target***, which can be a ***Target point, Target area (circle or rectangle)*** or ***Target line***.

STEP 02 Determine the position of the ***Datum Target***.

STEP 03 Define the size of the target area if necessary.

STEP 04 Define the ***Text point***.

STEP 05 Define the Datum Text and the ***Datum Text 2*** or ***Datum Text 3*** if necessary.

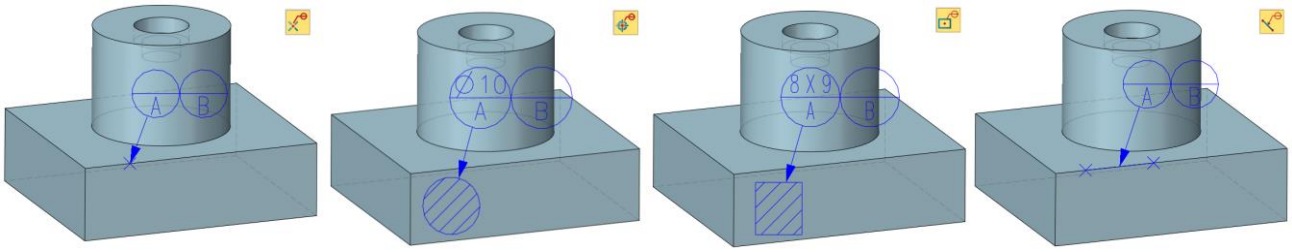


Figure 97 Datum Target

6.4 Feature Control Symbol

Dimension Ribbon Tab->Annotation->Feature Control

This command can help you create a feature control symbol.

STEP 01 Define the FCS text in the Feature Control dialogue box.

STEP 02 Pick the location points of the FCS text.

STEP 03 Define other parameters, such as additional Leader Points and the Display type.

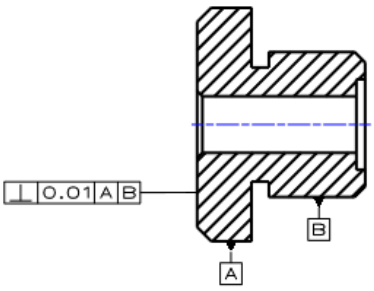


Figure 98 Feature Control Symbol

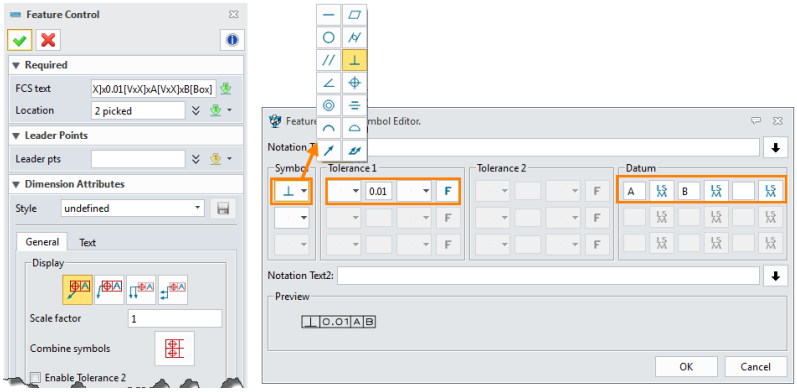


Figure 99 Defining the Feature Control Symbol

6.5 Center Mark/Center Line/Center Mark Circle

Dimension Ribbon Tab->Symbol->Center Mark

This command can help you create a center mark on an arc or circle, as shown in Figure 100.

Dimension Ribbon Tab->Symbol->Center Line

This command can help you create a centerline on a line, arc or circle, as shown in Figure 101. Also, you can manually draw a centerline by picking two points.

- **Use individual extension**

Check the ***Use individual extension*** option.

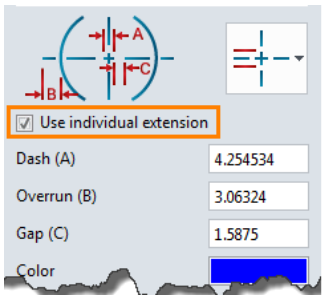


Figure 100 Center Mark

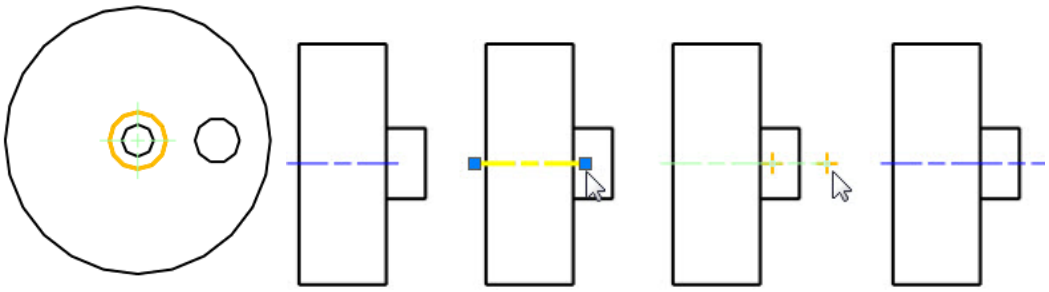


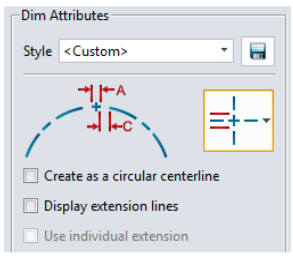
Figure 101 Drag and Drop the Center Line

Dimension Ribbon Tab->Symbol->Center Mark Circle

This command can help you place centerlines on entities following a circular pattern, for example, a bolt hole.

STEP 01 Select the center points of all holes.

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



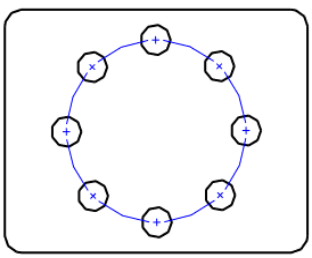
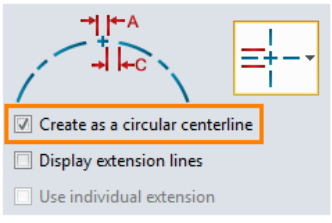


Figure 102 Center Mark Circle



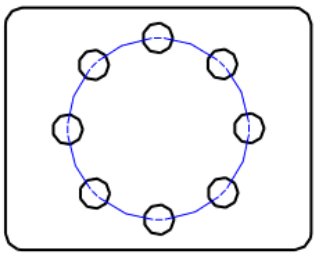


Figure 103 Center Mark Circle—Circular Centerline

- **Create as a circular centerline**

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

6.6 Surface Finish Symbol

Dimension Ribbon Tab->Symbol->Surface Finish

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

STEP 01 Pick a point to locate the symbol.

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

STEP 03 Select the machining type and set the value.

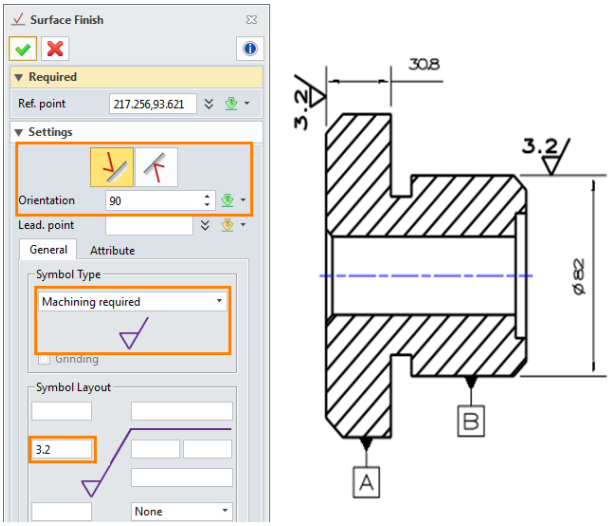


Figure 104 Surface Finish Symbol

6.7 Intersection Symbol

Dimension Ribbon Tab->Symbol->Intersection Symbol

This command can help you easily add an intersection symbol between 2 entities.

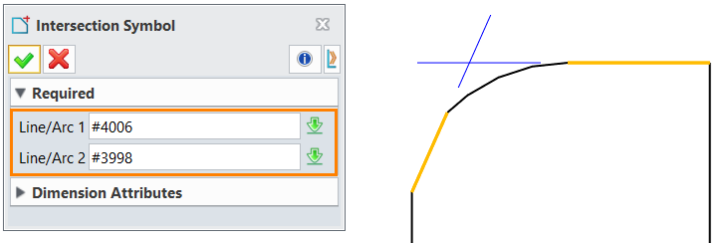


Figure 105 Intersection Symbol

6.8 User Symbol Management

Dimension Ribbon Tab->Symbol->Insert/Add/Manage

These commands can help you insert a customized symbol from the symbol library.

- ***Add*** is used for saving entities to the active symbol library. You can select a part of or the entire drawing sheet, define the ***Base Point*** and ***Name***, then click ***OK*** to finish.

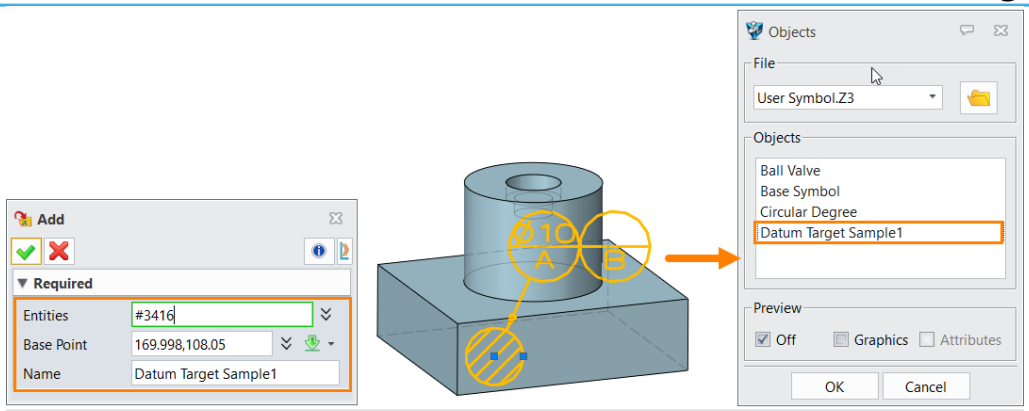


Figure 106 Add Symbol to Library

Note: *By default, the symbol will be restored in the current file. But if you have added a file to the* ***Configuration->Files tab->Symbol library****, then the symbol will be restored to this file instead, as shown in the below image.*

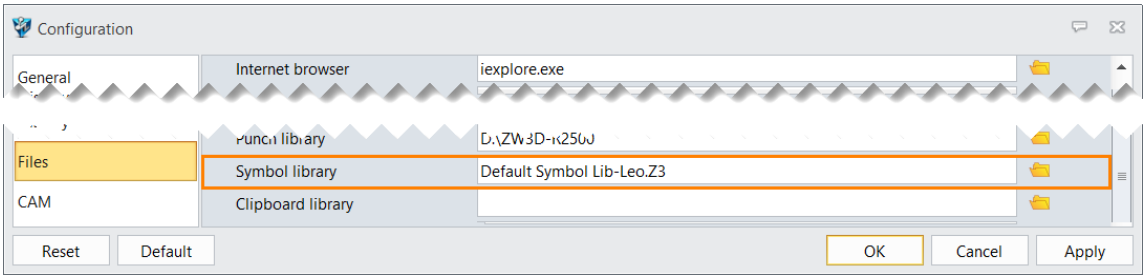


Figure 107 Symbol Library Definition

- ***Insert*** is used for inserting symbols from the library to the current sheet. Simply select the file and needed symbol, then drag it to the position you want.
- ***Manage*** is used for managing the symbols in the library, such as deleting or renaming them.

7 Table and Balloon

7.1 Create a BOM Table

Dimension Ribbon Tab->Table->BOM

This commands can help you create a BOM table for the selected view (detail or section).

The image below is an assembly consisting of some components and sub-assemblies.

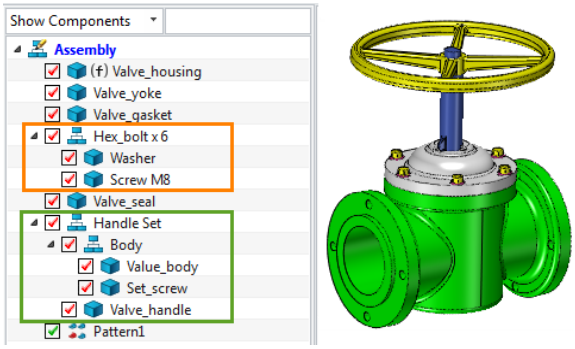


Figure 108 Assembly Model

- **Level Setting**

Top-level only means that parts and sub-assemblies will be listed, but not the components of the sub-assemblies.

Parts only means that all parts (including the ones of all sub-assemblies) will be listed, but not sub-assemblies. Each subassembly component is an individual item.

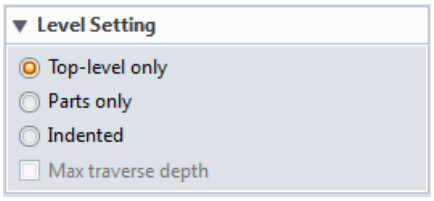


Figure 109 Level Setting of BOM

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

Figure 110 Top-level only VS. Parts only

Indented means that all the parts , sub-assemblies and their components will be listed. Also, you can define the ID with one of the three types of numbering, 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

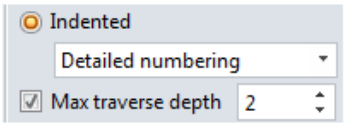
ID	Name	Quantity
1	Handle Set	1
2	Body	1
3	Set_screw	1
4	Value_body	1
5	Valve_handle	1
6	Hex_bolt	6
7	Screw M8	1
8	Washer	1
9	Valve_gasket	1
10	Valve_housing	1
11	Valve_seal	1
12	Valve_yoke	1

Figure 111 Indented Level

- ***Max traverse depth***

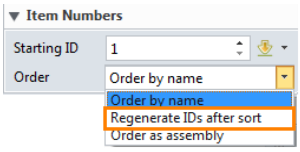
This option is to define the level of assembly from which the BOM table should read the data.

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



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

Figure 112 Different Traverse Depth



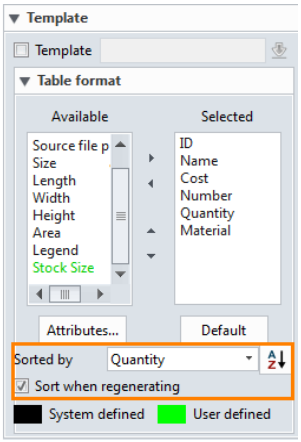


Figure 113 Item Number and Template

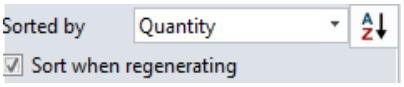
- ***Table ID Order***

There are 3 different methods to sort the table IDs.

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

Order as assembly: Based on the order of component insertion to sort the table IDs.

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

Figure 114 Table Order

Note: *If the* ***Sort when regenerating*** *option is unchecked, the order will not change, so IDs also will not change.*

All available part attributes are listed in here, including black-colored system attributes (such as Legend) and green-colored user attributes (such as ***Stock Size***), as shown in Figure 113.

The icons allow you to add or delete attributes.

The icons allow you to rearrange the sequence of the attributes.

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	

Figure 115 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.

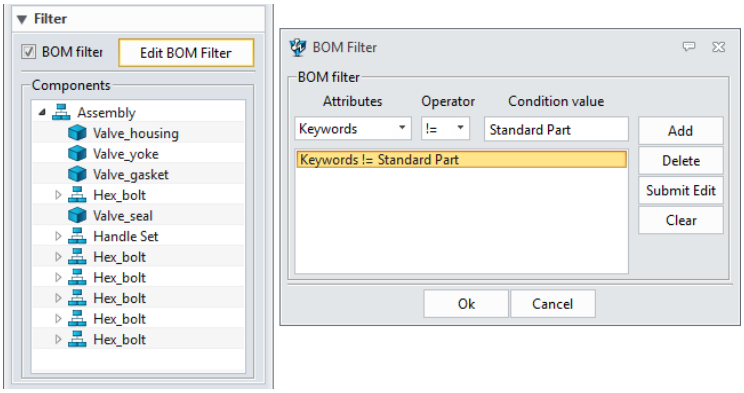


Figure 116 BOM Filter

7.2 Edit the Table

As the image below shows, you can click to invoke the Table dialogue box and click any column or row to invoke the Table Column/Row dialogue box. Then you can add or delete columns or rows, set the text alignment and text attributes, etc.

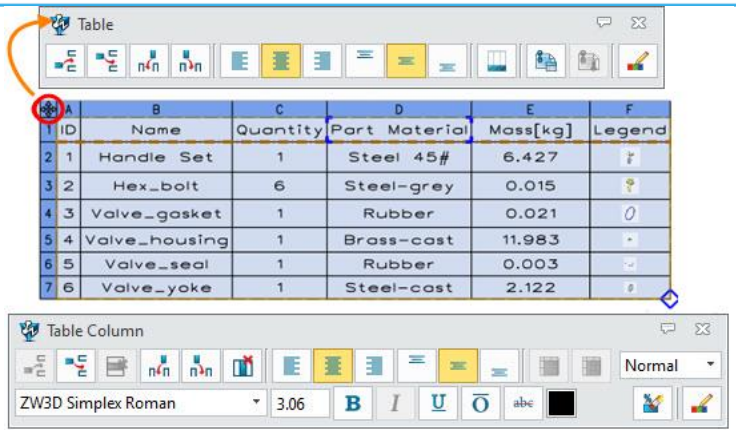


Figure 117 Table Editor

➢ **Edit the Column Name**

Double click the column header to edit, such as ***Material*** ->***Part Material***.

➢ **Edit the Column Attributes**

Right-click on the column header, then click ***Attributes*** to edit attributes in the Table Attributes dialogue box, for example, the value of precision.

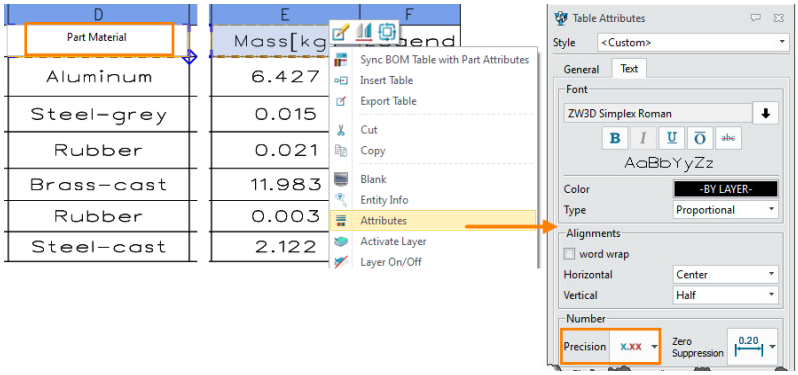


Figure 118 Edit Column Name and Attributes

➢ **Edit Cell Value**

Double-click on the cell, then click ***Yes*** to edit.

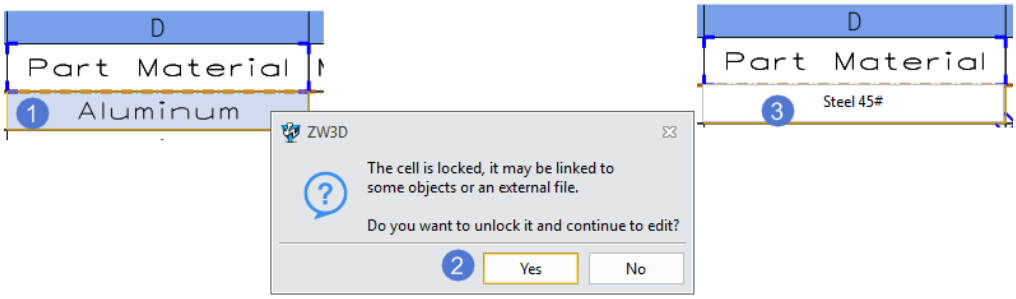


Figure 119 Edit Cell Value

➢ **Sync BOM Table with Part Attributes**

Method 1: Right-click on  and then click ***Sync BOM Table with Part Attributes***.

Method 2: Right-click on the name of the BOM table in the ***Manager***, then click ***Sync BOM Table with Part Attributes***.

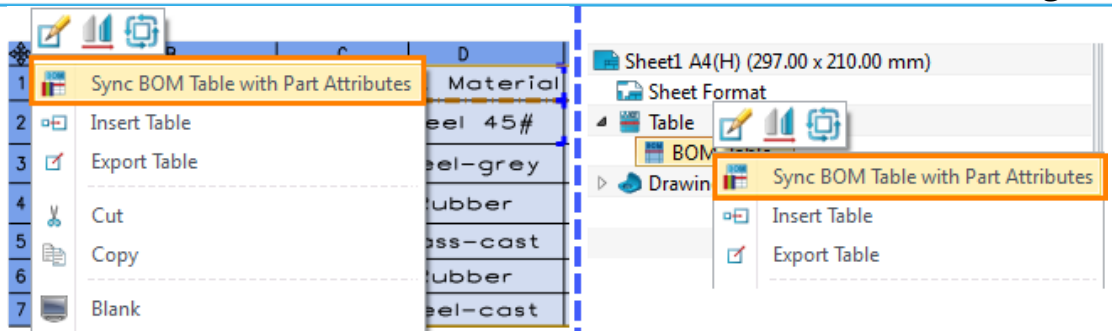


Figure 120 Sync BOM Table with Part Attributes

Next, you can check the ***Part Attributes*** in the modeling level following ***Tool Ribbon Tab->Attributes->Part Attributes***.

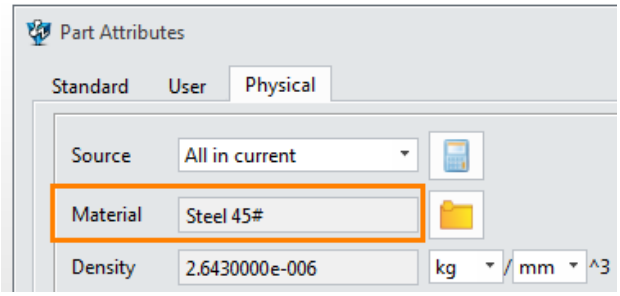


Figure 121 Updated Part Attributes

➢ **Edit Table Header**

Right-click on  ***->Insert->Head Row***. Then you can merge the cells or edit the header cells.

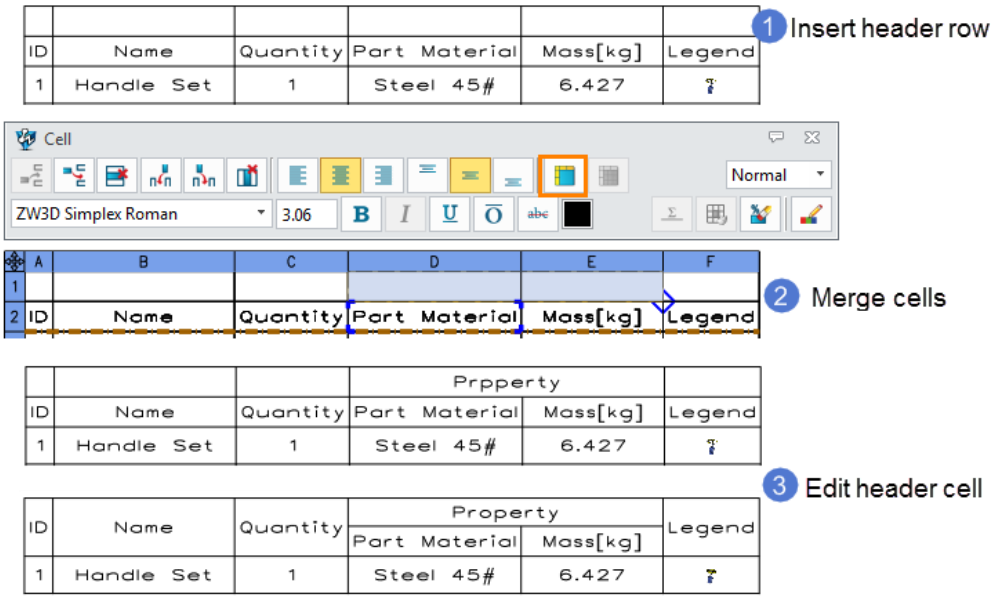


Figure 122 Edit Table Header

➢ **Table Template**

Right-click on  ***->Save as Template…*** to save this table header as a BOM template for later use.

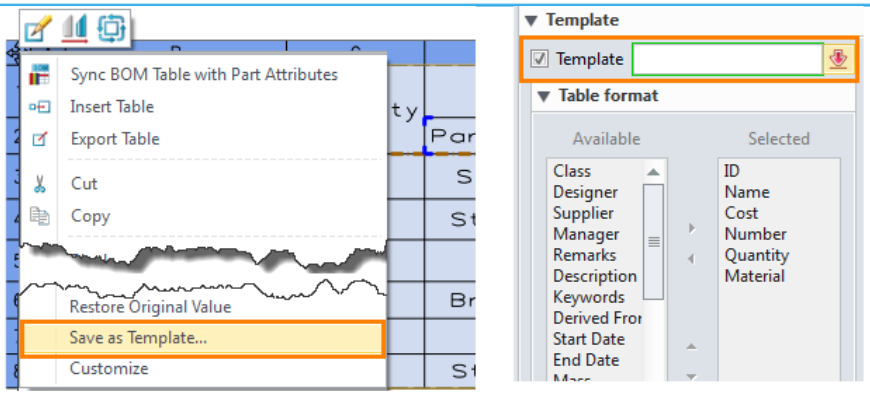


Figure 123 Table Template

➢ **Import/Export Table**

The table can be exported with ***Export Table*** and saved as an Excel® file for later modification. Then the edited Excel® file can be inserted back into the 2D sheet, as shown in Figure 124,

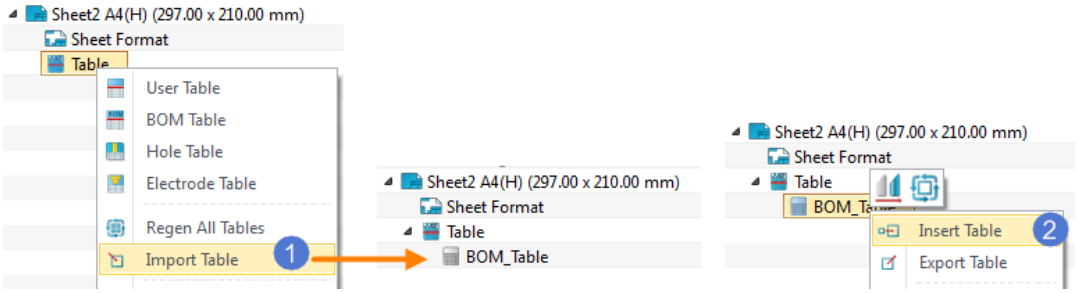


Figure 124 Import & Insert Table

7.3 Auto Balloons

Dimension Ribbon Tab->Annotation->Auto Balloon

Balloons can be automatically generated in a view based on the visibility of components and inserted into the appropriate views without duplicates. You can also specify whether the balloons follow the order of assembly 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****.*

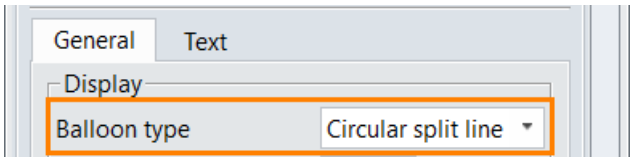


Figure 125 Balloon type-Circular split line

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.

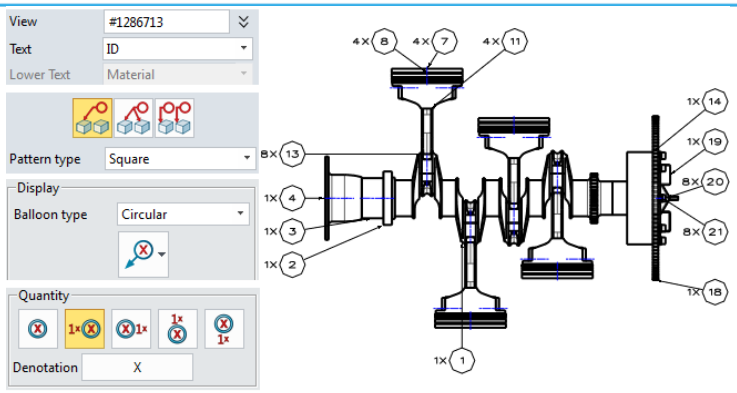


Figure 126 Auto Balloon Result 1 (Pattern type: Square, Balloon type: Circular, Quantity: the 2nd type)

You could try other combinations of parameters to get different results, as shown in the image below.

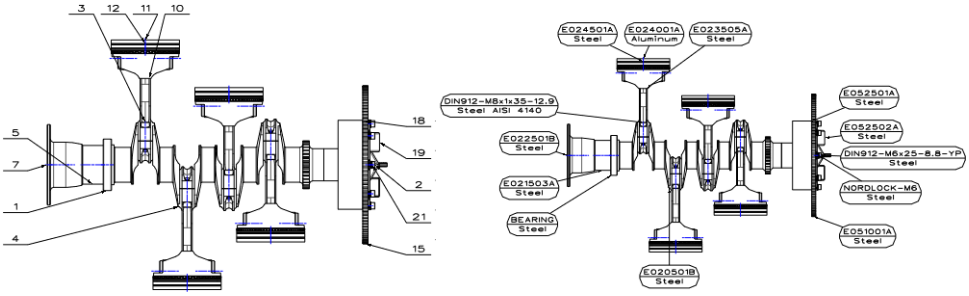
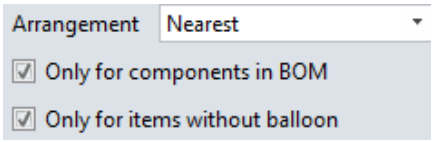


Figure 127 Various Auto Balloon Results

More Layout Parameters

If there is a BOM inserted in the drawing sheet, these 2 options are activated.



Only for components in BOM is to control whether the components excluded from the BOM should be labeled. If the excluded components are included, a ***** character will be used as their ID and you can modify it.

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

Below is an example.

STEP 01 Create the top view and the projection view on the left.

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

STEP 03 Select the top view to create the balloon.

STEP 04 Select the projection view to create the balloon and set the parameters.

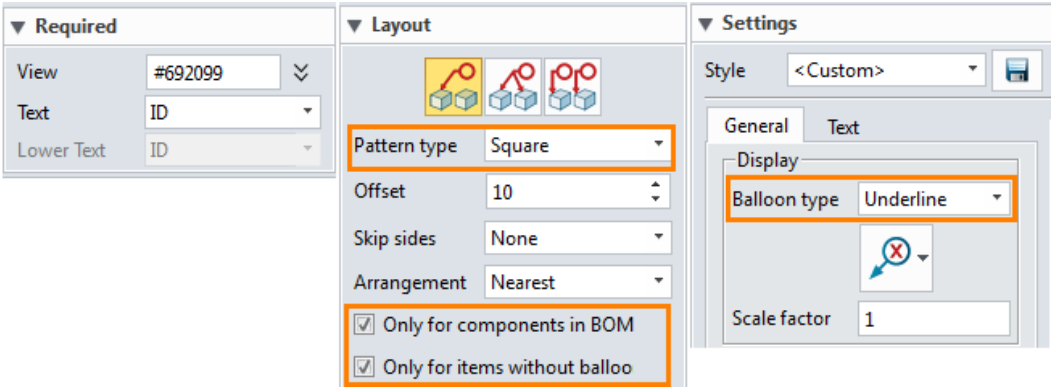


Figure 128 Definition of Parameters

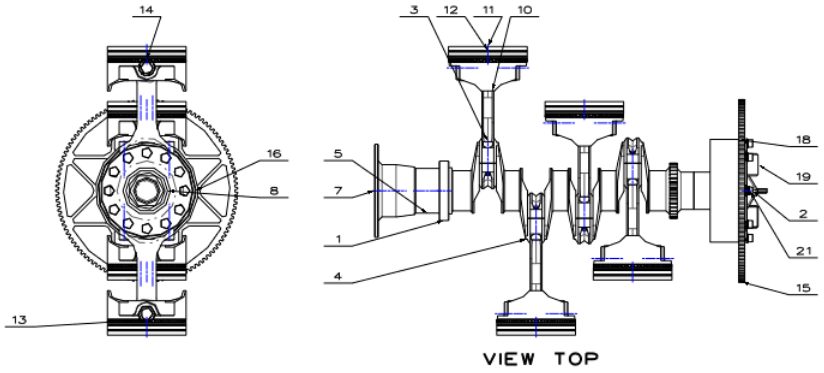


Figure 129 Balloons

7.4 Balloons

Dimension Ribbon Tab->Annotation->Balloon

This command allows you to manually create a balloon for the picked entity, even with multiple base points and leader arrows, as shown in the image below.

The parameters of dimension attributes are the same as those of ***Auto Balloon***.

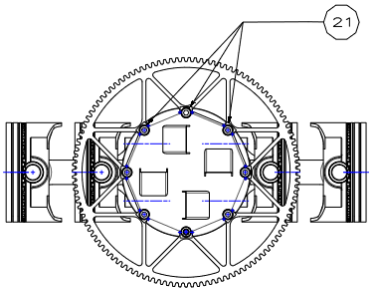


Figure 130 Manual Balloon

7.5 Stack Balloons

Dimension Ribbon Tab->Annotation->Stack Balloons

This command can help you stack multiple ballons together, as shown in the image below.

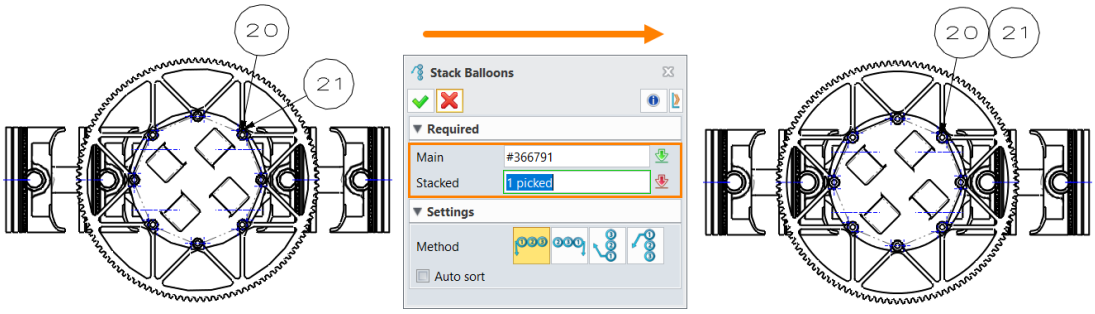


Figure 131 Stack Balloons

7.6 Hole Table

Dimension Ribbon Tab->Table->Hole

This command can help you create a hole table based on a layout view that contains real holes and user-defined holes. The model below is an example.

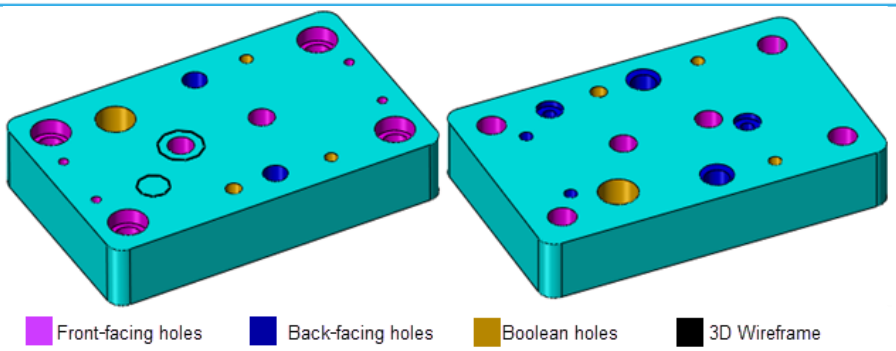


Figure 132 Hole Model

STEP 01 Create a top view of the model.

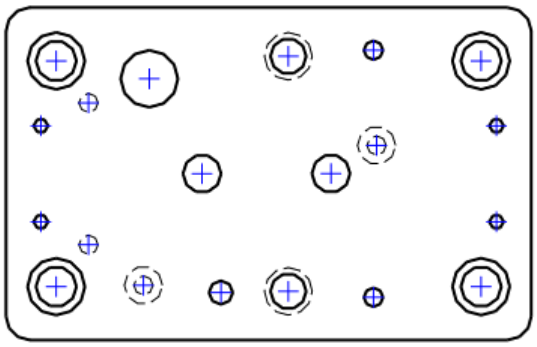


Figure 133 Top view-Hole

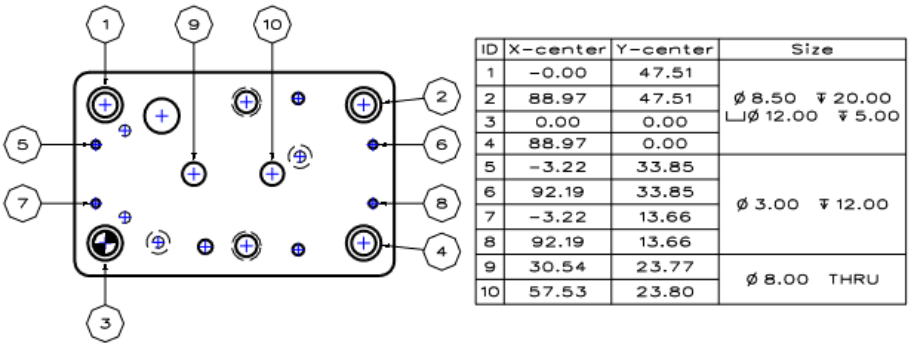
STEP 02 Select the 2D view and name the hole table.

STEP 03 Pick a point as the ***Base point***, according to which the x & y coordinates of the hole will be defined in the hole table.

STEP 04 Set the ***Hole Filter***. By default, only front-facing holes are selected.

STEP 05 Set other parameters if required. For example, check the ***Combine same size*** option.

STEP 06 Click ***OK***. Then pick a point to insert the hole table.



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	

Figure 134 Hole Table

- **Hole Filter**

Backfacing Holes/Hidden Holes

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

When both the ***Backfacing holes*** and ***Hidden holes*** options are checked, all back-facing holes will be selected.

Boolean Holes

If this option is checked, 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. Also, this option is ineffective when you're selecting user-defined holes.*

Hence, please redefine the ***View*** *option after resetting the hole filter. Then the selected hole features will be updated, as shown in the image below.*

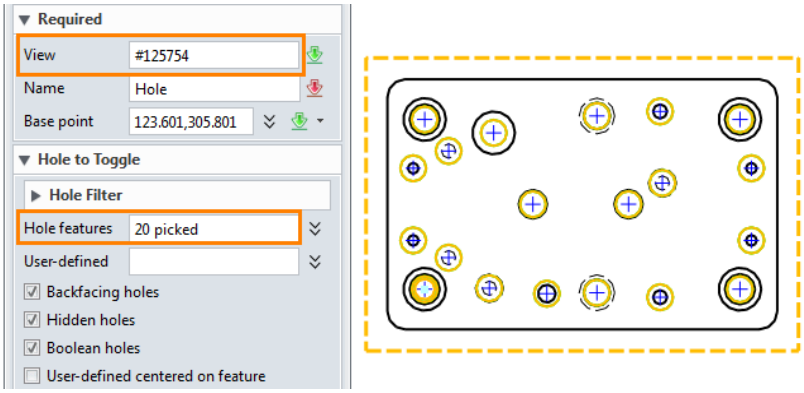


Figure 135 Reset Hole Filter

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

STEP 01 If some circles are drawn in the part level, you can edit the View Attributes to show them first. Right-click on the view-***>Display others->Show 3D Curves from Part.***

In the image below, there are two circles. Circle 2 is concentric with one of the hole features.

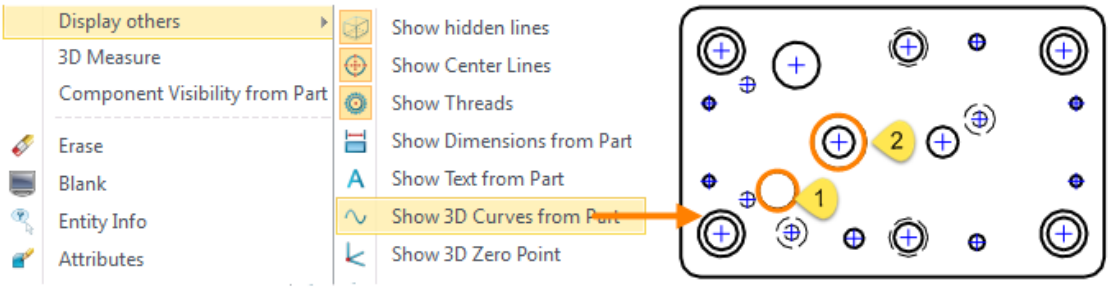


Figure 136 Show 3D Curves from Part

STEP 02 Activate the ***User-defined*** option, and Circle 1 can be picked. If you want to pick Circle 2, the ***User-defined centered on feature*** option should be checked.

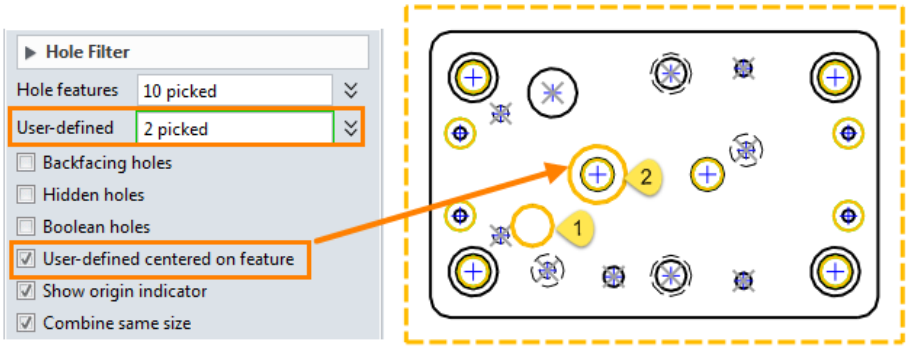
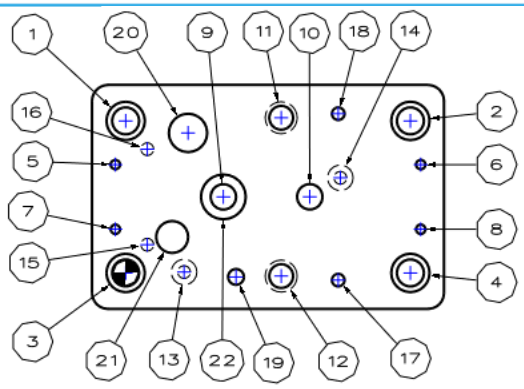


Figure 137 User-defined Hole Feature

STEP 03 Pick all hole features and add user-defined holes for the hole table.



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

Figure 138 The Whole Hole Table

7.7 Annotation Table

Layout Ribbon Tab->Table->Annotation

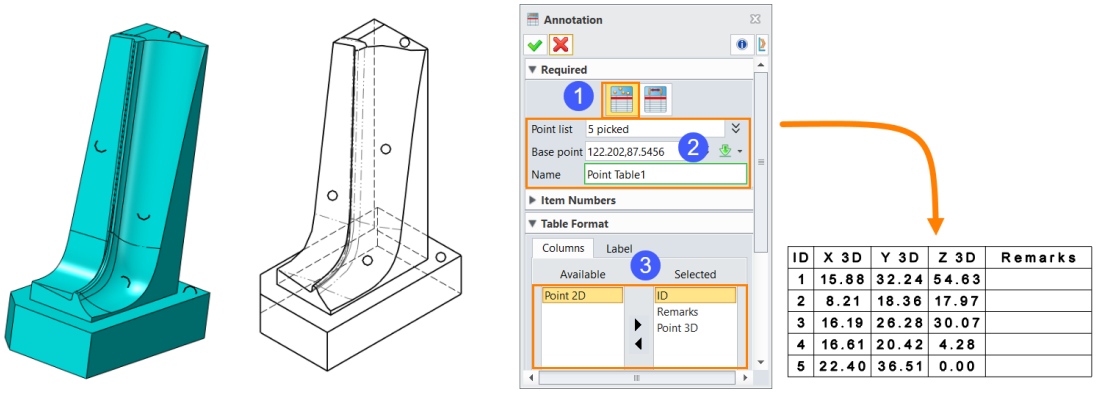
This command allows you to generate ***Point Table*** or ***Dimension Table***.

- ***Point Table***

STEP 01 Select the needed points for the ***Point list*** and define one as the ***Base point.***

STEP 02 Name the Point Table.

STEP 03 Select the available attributes for the columns of the table.

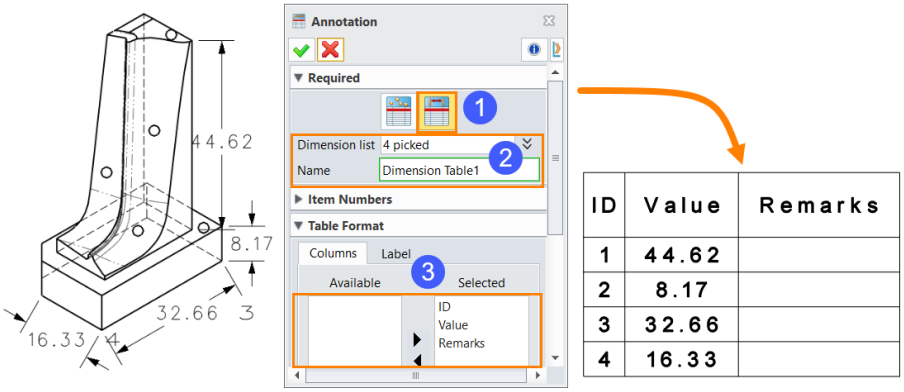


ID	X 3D	Y 3D	Z 3D	Remarks
1	15.88	32.24	54.63	
2	8.21	18.36	17.97	
3	16.19	26.28	30.07	
4	16.61	20.42	4.28	
5	22.40	36.51	0.00	

Figure 139 Point Table

- ***Dimension Table***

The operation is similar to that of ***Point Table***. You can get the result by following the 3 steps in the image below.



ID	Value	Remarks
1	44.62	
2	8.17	
3	32.66	
4	16.33	

Figure 140 Dimension Table

8 Auto Drafting

Part/Assembly context->Tools Ribbon Tab->Insert->Auto Drafting

Auto Drafting can generate a drawing sheet automatically for the selected assemblies, components or shapes at once to improve your work efficiency.

STEP 01 Select the models to draft.

STEP 02 Define the size and saving path of the sheet, and its name and scale if necessary.

STEP 03 Define the layout of the sheet, as how you'd use the ***Layout*** command in the ***2D sheet*** level.

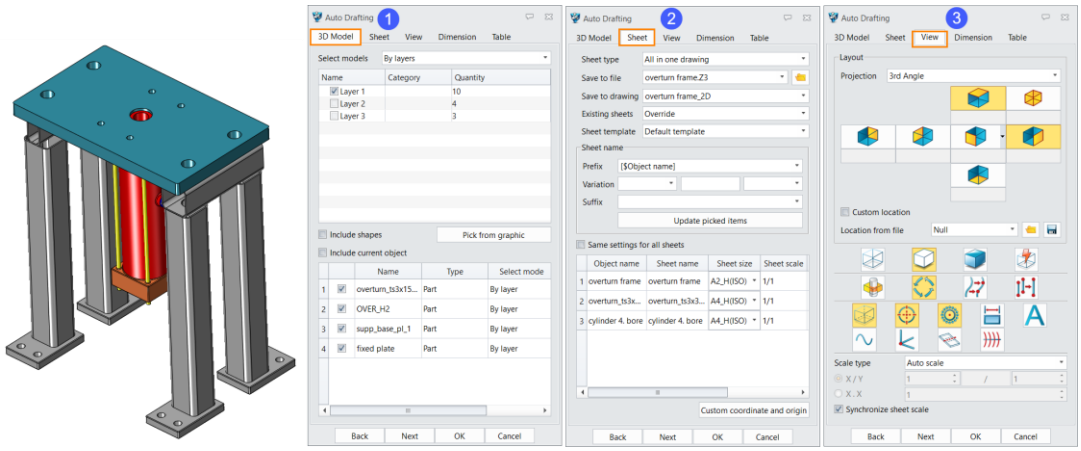


Figure 141 Steps of Auto Drafting

STEP 04 Define the settings of ***Auto Dimension*** and ***Table*** if necessary, then you'll get multiple sheets all at once.

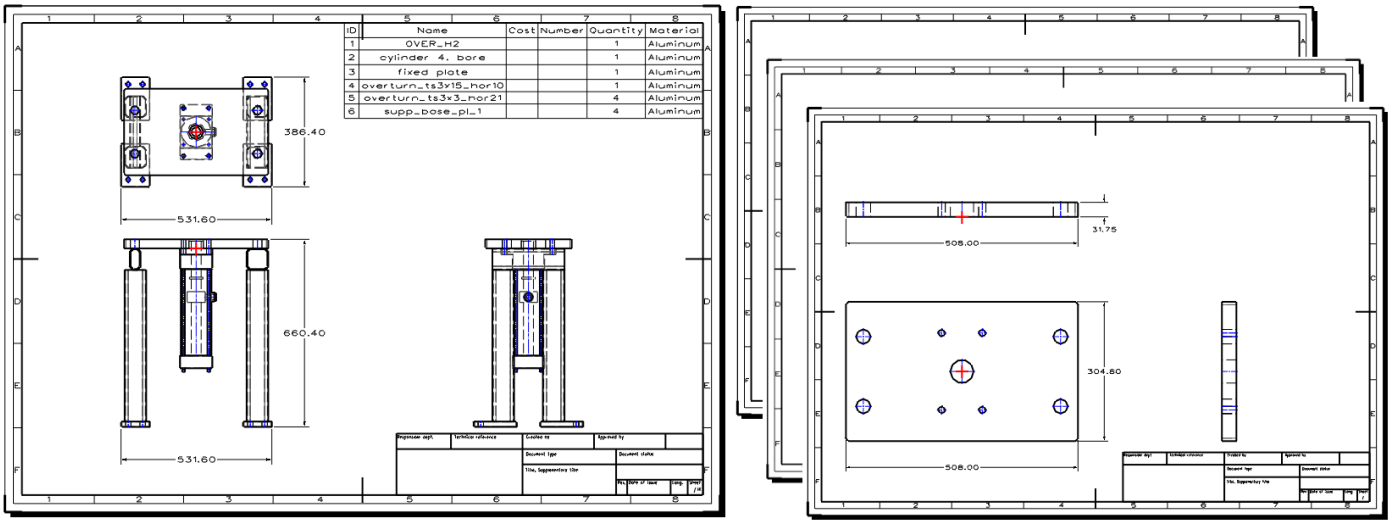


Figure 142 Auto Drafting Results

The aforementioned steps are just the basic way to use Auto Drafting in ZW3D. Let's see how to use other options to meet your different needs.

3D Model Selection Settings

There are 3 methods to select models to create a drawing sheet in ZW3D: ***By Layers***, ***By Filters*** and ***Pick from graphic***.

- **Auto Selection-By Layers**

You can pick layers to determine the objects that will be generated in the drawing sheet. After you picked the layers from the layer list, you can keep picking the objects from the object list for the drawing sheet.

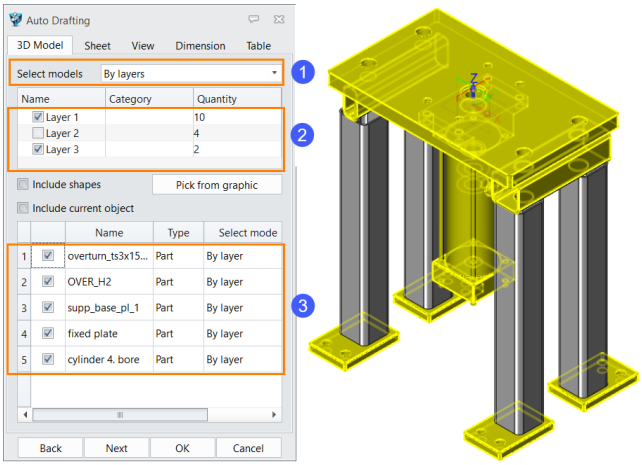


Figure 143 By Layers

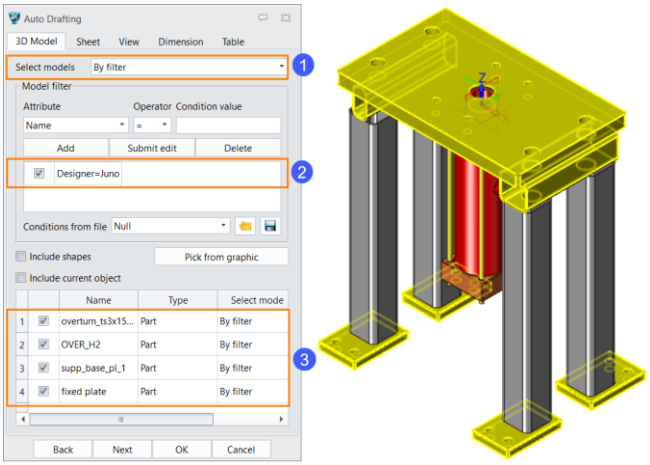


Figure 144 By Filters

➢ **Auto Selection-By Filter**

You can filter out objects to create a drawing sheet. The standard attributes can be used as a filter, such as a name, designer, cost, etc. And you can keep picking the objects from the object list for the drawing sheet.

Besides, the defined condition can be saved as a template for future use. Click the  button, input a name and click ***OK***, then the new filter will be saved as a template file.

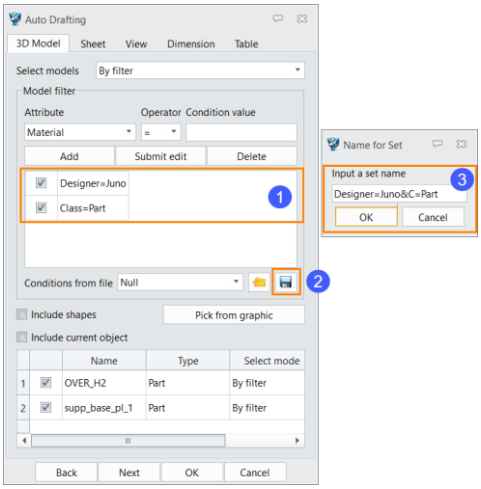


Figure 145 Filter Template

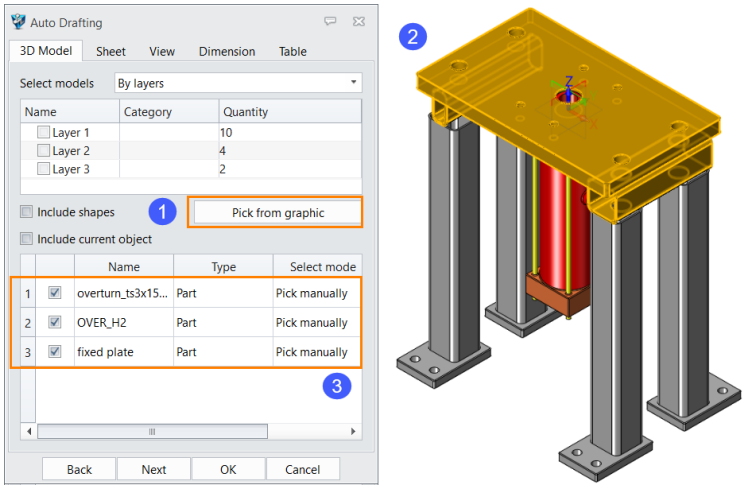


Figure 146 Pick from graphic

➢ **Manual Selection- Pick from graphic area**

You can pick objects in the graphic area manually to create a drawing sheet, as shown in Figure 146.

Besides, 2 more options are provided to control whether to create the drawing sheets for shapes or the top-assembly.

1) ***Include Shapes***:

When there are shapes in the current assembly level, the shapes are available to create the drawing sheets.

2) ***Include current object***:

The top-assembly of the current level is available to create the drawing sheets.

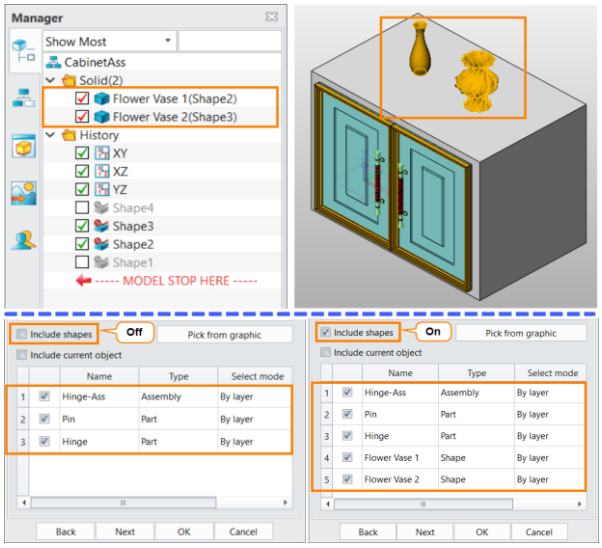


Figure 147 Check “Include shapes”

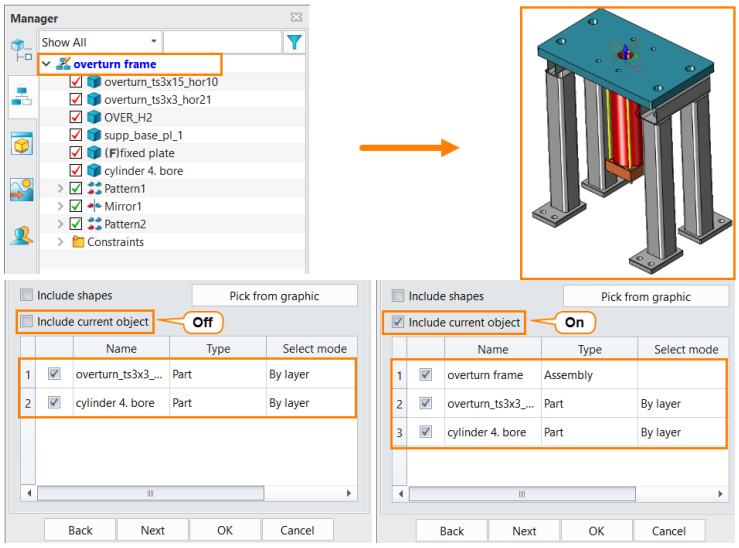


Figure 148 Check “Include current object”

Set Sheet Attributes

In the ***Sheet*** Tab, you can specify the name, template, size and save type of the sheet.

- **Types to Save the Drawing Sheet**

  1) ***All in one drawing*****:** All sheets will be saved into one drafting object.

  2) ***One object for each*****:** All sheets will be saved in different drafting objects but one Z3 file.

  3) ***One file for each*****:** All sheets will be saved as different Z3 files.

- **Sheet Template Settings**

The default template of ***Auto drafting*** will be the template set in ***Configuration***. But, you can also specify an external template to be the new drawing sheet. When the template file is specified, all the ***Sheet Size*** in sheet templates and sheet sizes will be read for you to specify the sheet size of different objects.

- **Sheet Name Setup**

The name of each drawing sheet can be generated according to the rule defined by ***Prefix***, ***Variation*** or ***Suffix***.

The ***Prefix*** and ***Suffix*** fields can define by 4 types of attributes:

1) Name of the object to draft: The reference format is ***[$Object name]***.

2) Number of the object to draft: The reference format is ***[$Number]***.

3) Sheet Code of the object to draft: The reference format is ***[$Sheet code]***.

4) Root assembly name: The reference format is ***[$Assembly name]***.

The “***Variation***” field is used to define the increment method of sheet name.

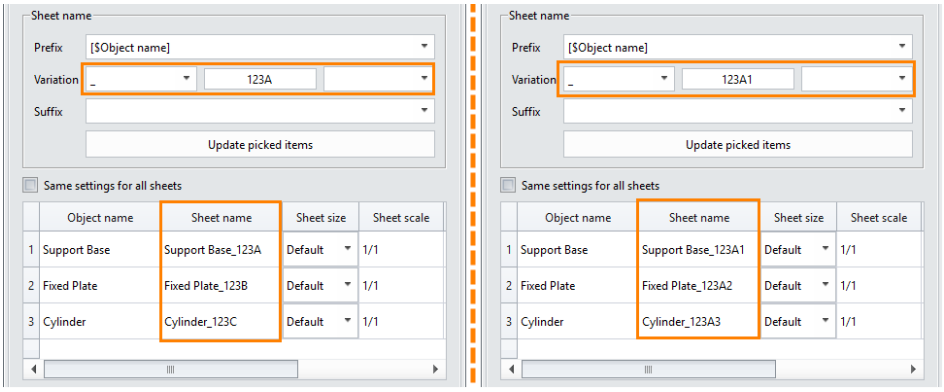


Figure 149 Sheet Name Setup

Notes: *After finishing setting up* ***Prefix***, ***Variation*** *and* ***Suffix****, you can click* ***Update picked item*** *to update the sheet name preview in the dialog.*

- ***Custom Coordinate and Origin***

View projection can have various results based on different frames. By default, ***Auto Drafting*** use the WCS (World Coordinate System) as the projection coordinate system. If you want to redefine a coordinate system to create a drawing sheet, pick the redefined items first, then click ***Custom coordinate and origin*** to invoke a dialogue box to define.

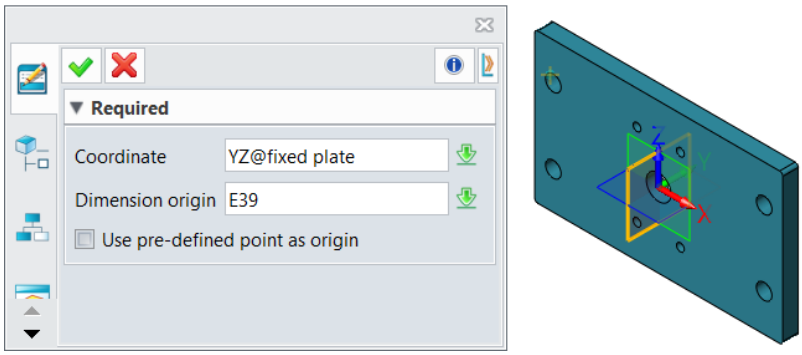


Figure 150 Frame and Origin Setting for View Projection

Notes: *The* ***Auto dimension*** *is supported by* ***Auto Drafting***. Therefore*, the default origin will be used to create dimensions. You can use the* ***Custom coordinate and origin*** *to redefine the origin.*

Set View Attributes

Please refer to ***Chapter 2.1*** for procedures of setting view attributes.

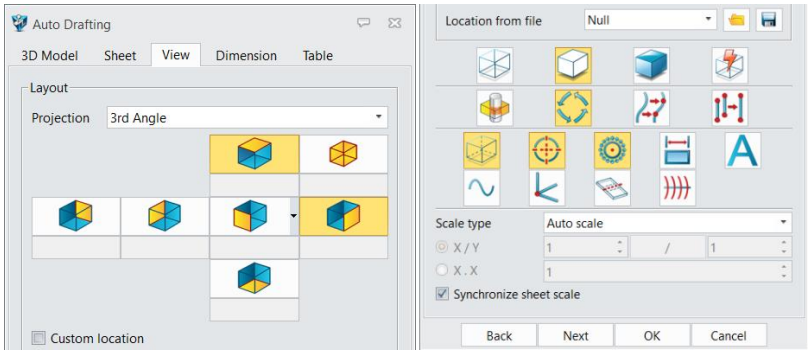


Figure 151 View Projection Settings

Auto Dimension Settings

Please specify the type of views and objects to generate the dimensions. In the ***Dimension*** tab, you can specify the view where you're creating dimensions with auto dimension, and the entities to create the dimensions and type of dimensions as well.

- **Enable Auto Dimension**

Check this option and other settings in this tab will be available. Otherwise, you will not be able to generate any dimension in the drawing sheet.

- **Layout**

Specify the views you're adding the dimensions to. Only the views enabled in the ***View*** Tab can be selected to create dimensions***.***

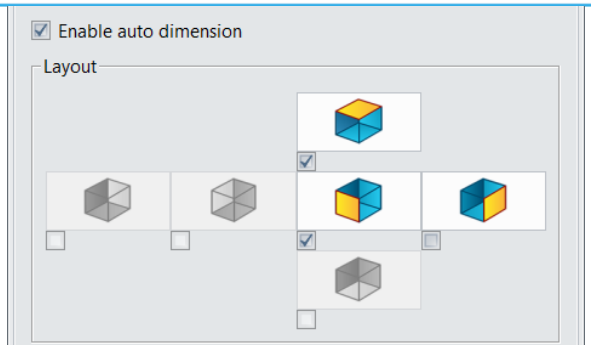


Figure 152 Views for Auto Dimension

➢ **Entities to dimension**

You can specify the detailed settings of ***Auto Dimension***. For more details, please refer to Chapter 1.4.3.

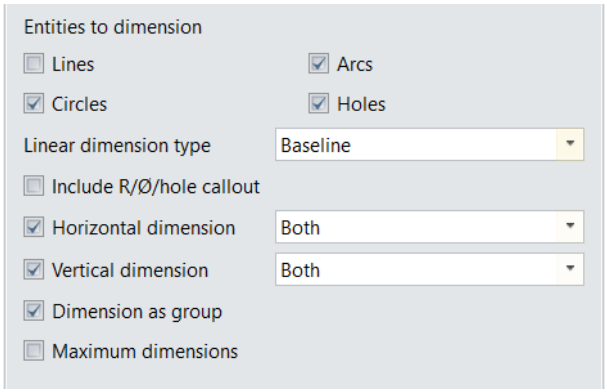


Figure 153 Auto Dimension Settings

Table Settings

BOM and hole tables can be generated during the ***Auto Drafting*** process. Also, the BOM table will be generated in each assembly drawing sheet. The location, template and other settings can be saved to the template.

❖ ***BOM Table Settings***

➢ **Enable BOM table**

Check this option to enable the creation of BOM tables and other settings for the BOM table.

➢ **Location**

The location of the BOM is defined by its origin and the position of the origin in the drawing sheet (represented by the percentage of the sheet size).

➢ **Table origin**

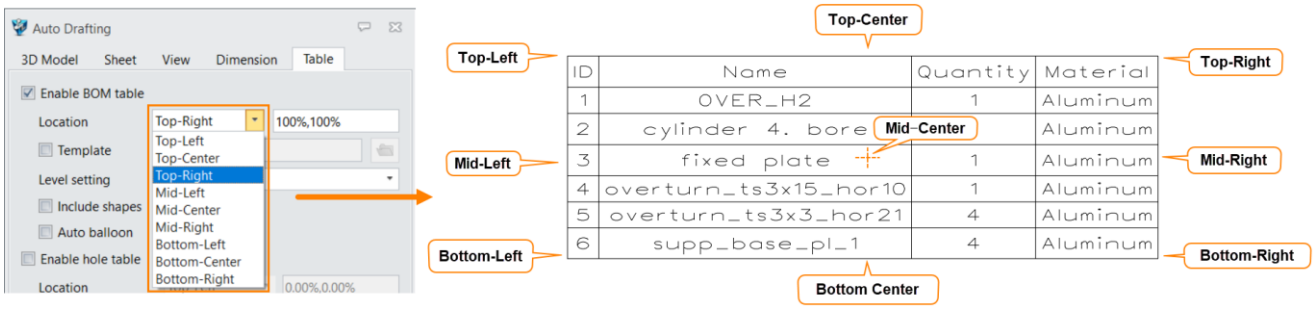


Figure 154 Table Origin

➢ **Sheet origin and percentage division**

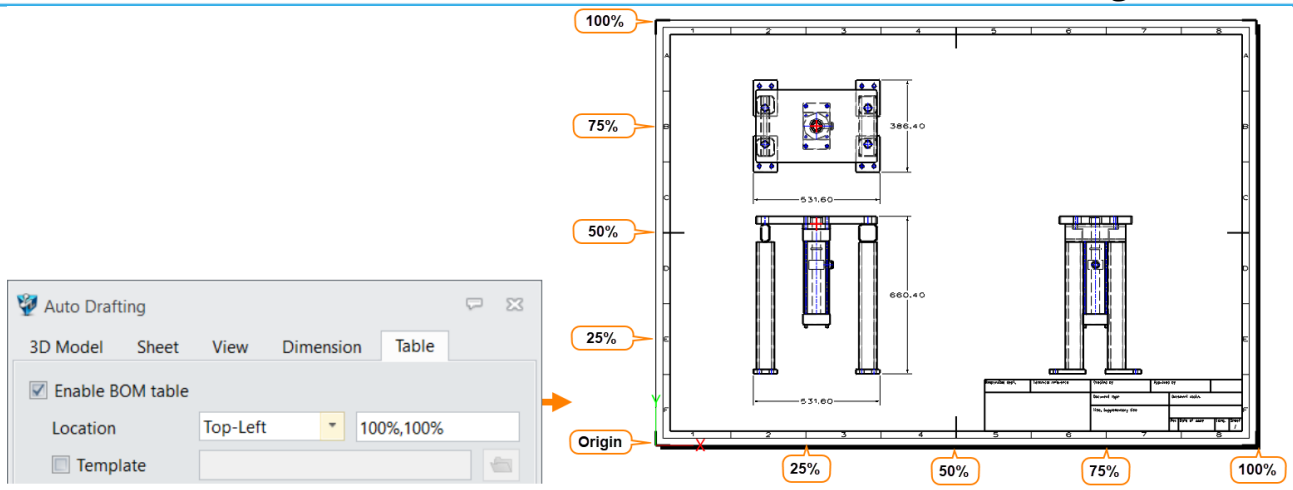


Figure 155 Sheet Origin and Percentage Division

For example, if the location has been set to ***Top-Right*** and ***100%,100%*** at the same time, the table will be located as the image below shows:

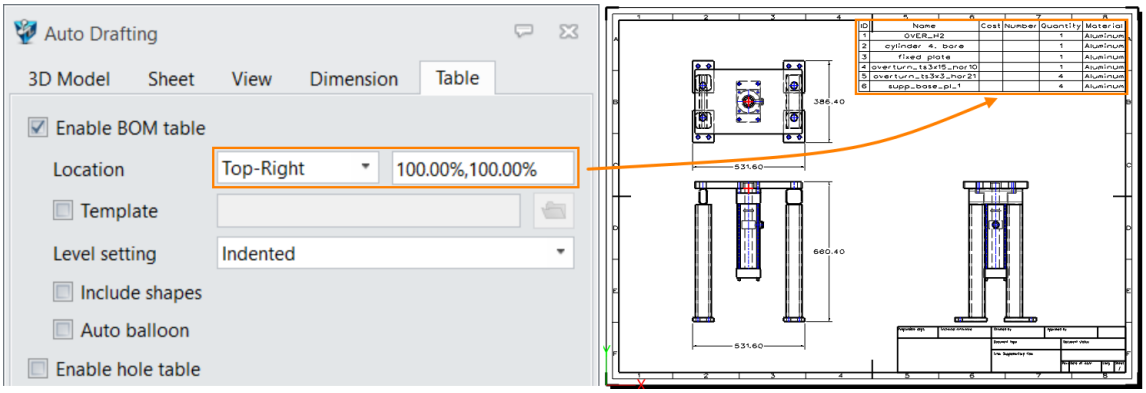


Figure 156 Table Location Definition

➢ **Template**

You can specify an external BOM table template.

➢ **Level setting**

You can choose from the 3 output types of BOM: ***Top-level only***, ***Parts only*** or ***Indented***. Then, check the ***Include shapes*** option to integrate shape items into the BOM table. You can refer to Chapter 7.1 for more information.

➢ **Auto balloon**

Check this option so that balloons will be added to the main view automatically.

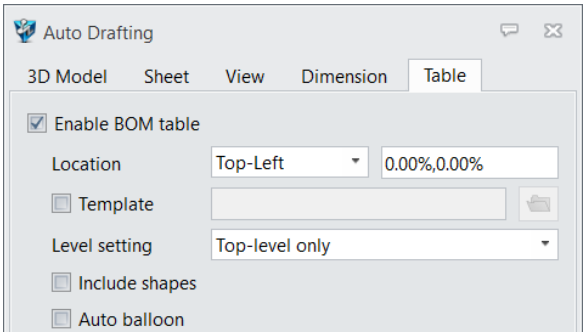


Figure 157 BOM Table Settings

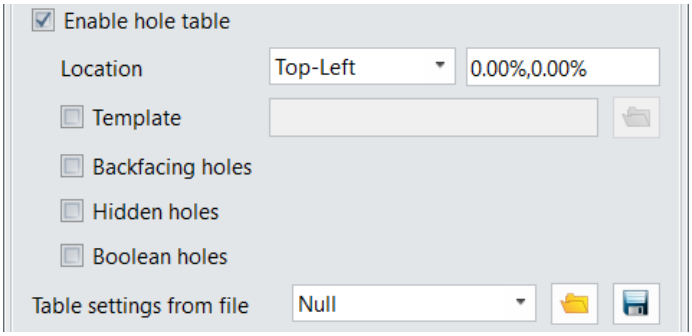


Figure 158 Hole Table Settings

❖ ***Hole Table Settings***

Setting a hole table is mostly similar to setting of BOM table, except that you can determine whether the

back-facing holes, hidden holes and Boolean holes will be displayed in the hole table, as Figure 159 shows.

The current settings can be saved as a template for future use. Click the button, then name the template.

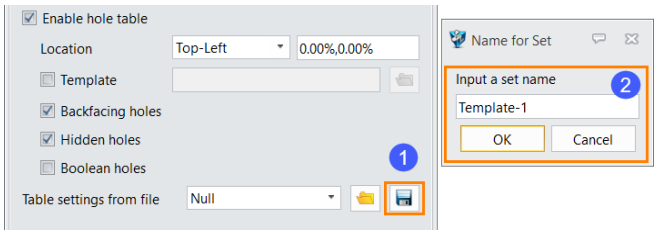


Figure 159 Table setting template

9 Case—2D Drawing

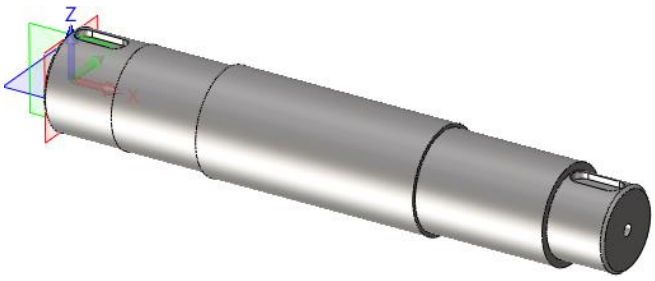


Figure 160 The 3D Model

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

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

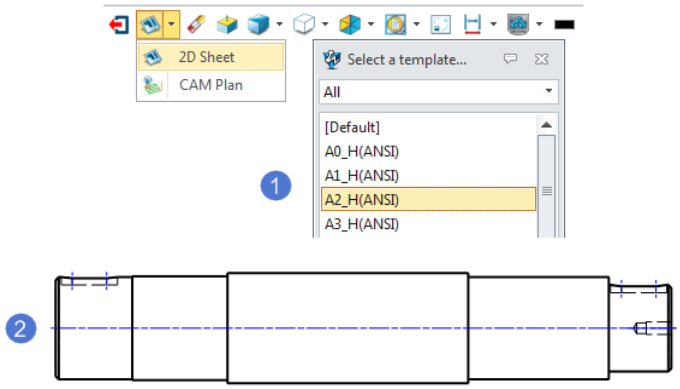


Figure 161 Top View

STEP 03 Use the ***Full Section*** command to create two section views. The parameters are set as the image below shows.

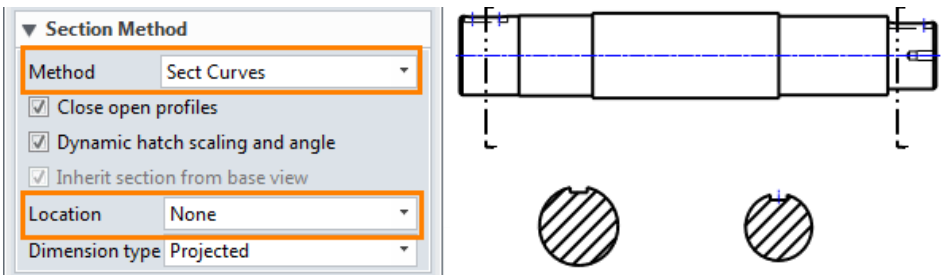


Figure 162 Section Views

STEP 04 Drag and drop the endpoints of the section line to get a better position. And right-click on the section line to select the ***Display label*** command.

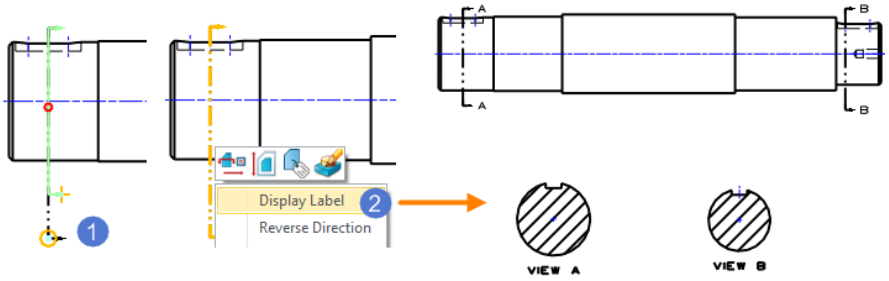


Figure 163 Section Views with Labels

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

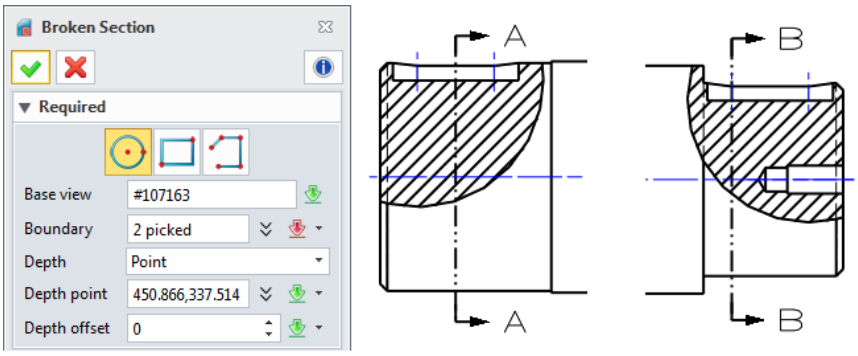


Figure 164 Broken Section Views

STEP 06 Invoke the ***Style Manager*** to modify the value of precision of the linear dimension, the text format of the chamfer dimension, the text shape, etc.

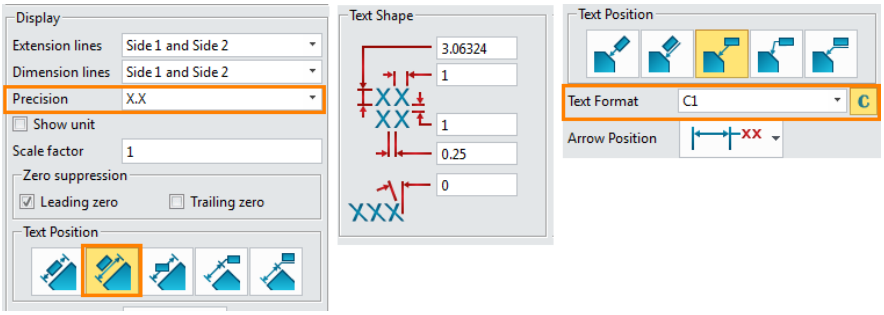


Figure 165 Dimension Style

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

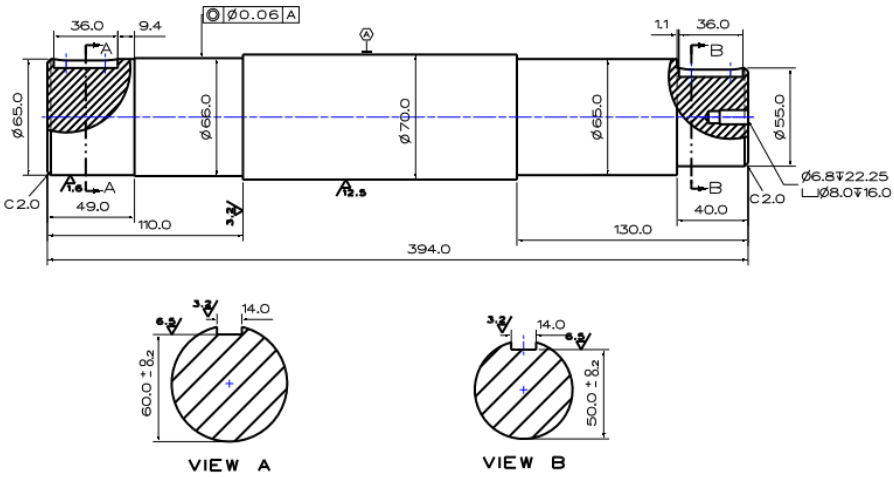


Figure 166 View with Dimensions

STEP 08 Set the default drawing units. ***Tool Ribbon Tab->Setting->Preference***

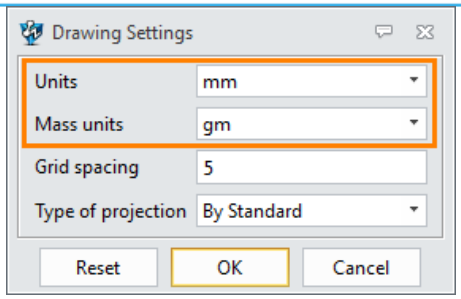


Figure 167 Drawing Settings

STEP 09 Edit the title block. You could add some part attributes with the ***Text*** command or in the editor.

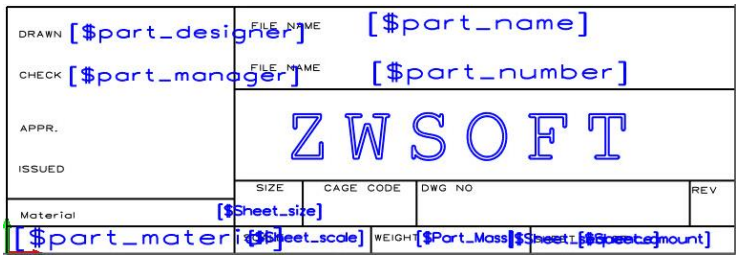


Figure 168 Edit the Title Block

Notes: *Some texts might seem overlapped when you are defining the part attributes. But it will not affect the results.*

STEP 10 After editing, go back to the 2D drawing level, and you will find all the information linked with the part attributes.

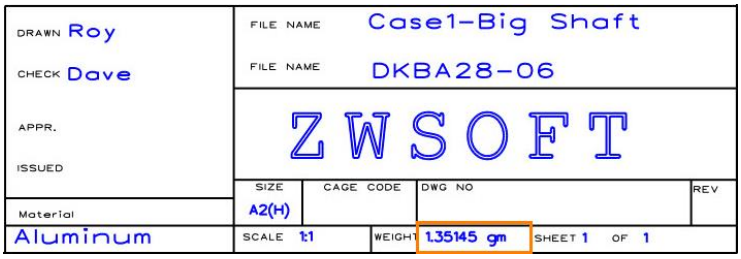


Figure 169 Title Block

STEP 11 Add some technical requirements with the ***Text*** command.

STEP 12 The final 2D sheet should be how the image below shows.

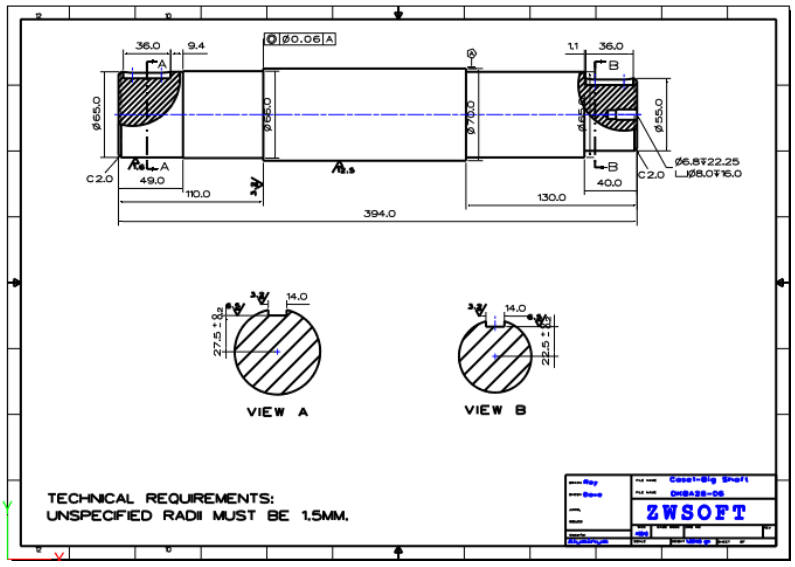


Figure 170 2D Sheet of Part