ZW3D Primary Tutorial

# Product Design

# ZW3D™ V2023 CAD Product Design



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**ZWSOFT CO., LTD.(GUANGZHOU)**

Room 01-08, 32/F, No.15, Zhujiang West Road,

Tianhe District, Guangzhou 510623, China

(8620)38289780

# *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 CAD  Product Design***, a primary tutorial.

Thanks for being our user!

The ZW3D Team

# Contents

# Introduction

ZW3D is an All-in-one, affordable CAD/CAM solution. It lets engineers design from concept to product in an easy-to-use, single collaborative environment. There are many highlight features in it including Hybrid Modeling, Direct Edit technology, Productive mold design, and High-efficiency manufacturing.

However, all the good application of these highlighted features is based on a good understanding of ZW3D basic and core functions. So, in this tutorial, the most important functions and concepts of ZW3D basic modules will be introduced to you with the briefest way.

The basic modules are **Sketch**, **Part Design**, **Assembly** and **2D Drawing**. Also, to make sure you can start ZW3D smoothly, the installation and activation will be talked first, then some basic settings/operations will be talked in chapter two. Now, let us start.

# 1 Installation and Activation

## 1.1 Installation

Before downloading ZW3D (https://www.zwsoft.com/zw3d/download-center), please go through your computer configuration. The system requirements are listed below.

| Requirements | Recommended |
|---|---|
| **Operating System** | Microsoft® Windows 7_SP1<br>Microsoft® Windows 10 |
| **Processor** | Intel® Core™ 5 or above |
| **RAM** | 8G or above |
| **Graphic** | OpenGL® 3.1 or above<br>NVIDIA® Quadro FX 580 @ 512MB or above |

After downloading ZW3D, please follow the steps below to install it.

STEP 01 Right-click on the ZW3D executable file, select ***Run as administrator*** in the pop-up menu.

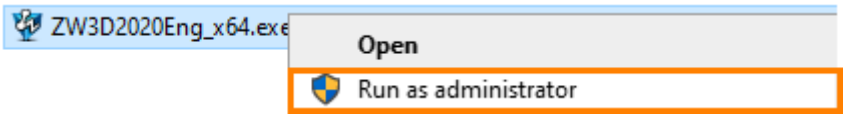



Figure 1 Run as Administrator

STEP 02 Select ***Language*** -> ***Install*** -> Select the version and modules -> Read and accept the terms -> Specify the installation path -> ***Installing***.

## 1.2 Activation

If you install a new ZW3D version for the first time, you can have a free 30-day trial of all ZW3D modules you installed, as shown in the picture below. Please note that add-ons like 5X Milling are not available for the trial version and that some functions, such as Save, Print, Import, and Export will be restricted after the trial expires.

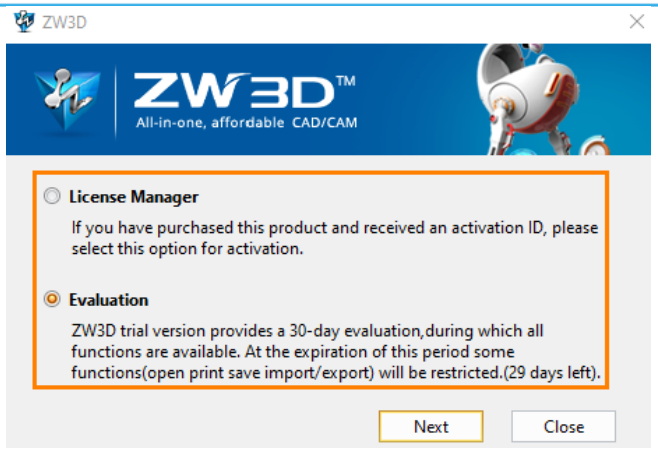



Figure 2 Evaluation

If you already have a standalone license, please activate it following these steps: Go to ***License Manager*** -> Click ***Activate*** -> Tick ***Soft-key Online Activation*** -> Paste your ***Activation ID*** -> ***Verify*** -> Fill the ***User Information***.

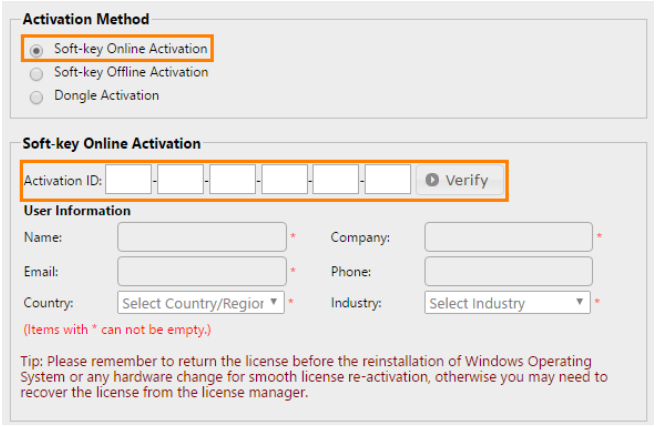



Figure 3 Activate ZW3D

To ensure a smooth activation, please avoid activating the standalone license remotely, turn off the firewall, and remove the port from the white list when the activation fails.

***Notes:*** Floating licenses are more popular in big companies where users need to borrow one from the server when running ZW3D because they are activated on a specific server.

# 2 Basics of ZW3D

## 2.1 User Role Settings

When you start ZW3D for the first time, you will need to select a user role according to your proficiency level. As Figure 4 shows, the default user role is ***Expert***, which means all the ZW3D commands and modules available will appear on the interface. Of course, you can switch roles with the Role manager at any time, as shown in Figure 5.

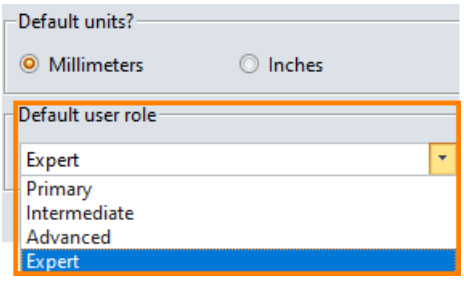



Figure 4 User Role Settings

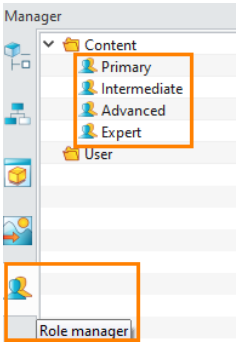



Figure 5 Role Manager

## 2.2 An Introduction to the Interface

The picture below shows the default ZW3D interface after you create a new part file. You can hide or show the menu bar at the top with small blue triangle on the left.

To control the display of the Manager on the left, please click the  icon on the bottom right corner, as speech balloon #1 points.

To adjust the global configurations of ZW3D, such as UI language and background color, please click the  icon on the upper right corner, as speech balloon #2 points.

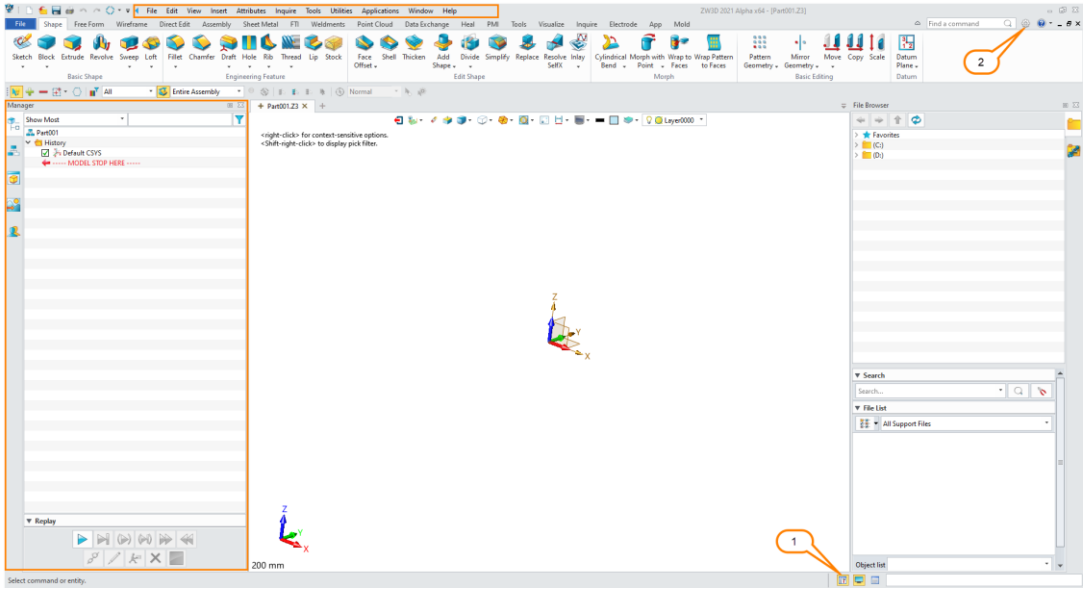



Figure 6 Default ZW3D Interface

If you need to change the UI language or background color, go to ZW3D ***Configuration*** form -> ***General*** or ***Background*** tab to do the set.

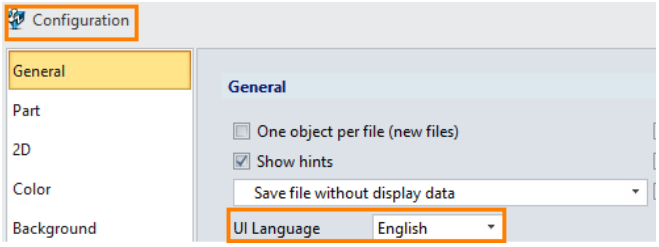



Figure 7 Changing UI Language

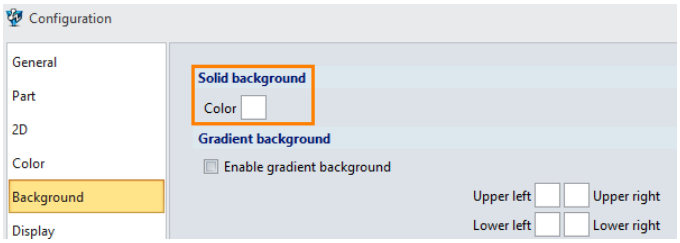



Figure 8 Changing Background Color

## 2.3 Customizing the Interface

To customize the interface, you can first right-click on any blank Ribbon area and select ***Customize...*** in the context menu, as shown in Figure 9. Then, click the ***Transfer*** tab in the Customize dialogue box and rearrange commands as you like.


Figure 9 Invoking the Customize Dialogue Box

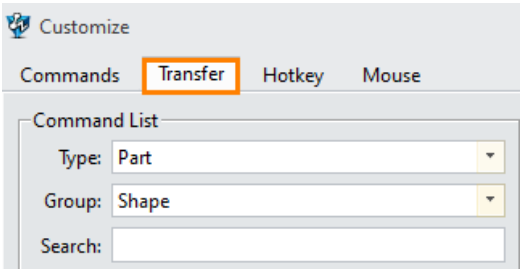



Figure 10 Rearranging Commands in the Transfer Tab

Here's an example of customizing the ***Start*** panel (Figure 11): You can remove the ***Mold Project*** command, add ***Application Plugin Manger*** and ***Library Publisher***.

STEP 01 Uncheck ***Mold Project*** or right-click on it and select Delete in the context menu in the Customize dialogue box.

STEP 02 Select ***Application Plugin Manger*** in the ***Command List***, then drag and drop it to the ***Start*** panel.

STEP 03 Select ***Library Publisher*** in the ***Library*** panel, then drag and drop it to the ***Start*** panel.

STEP 04 Click ***Apply***, then ***OK*** to finish customizing.

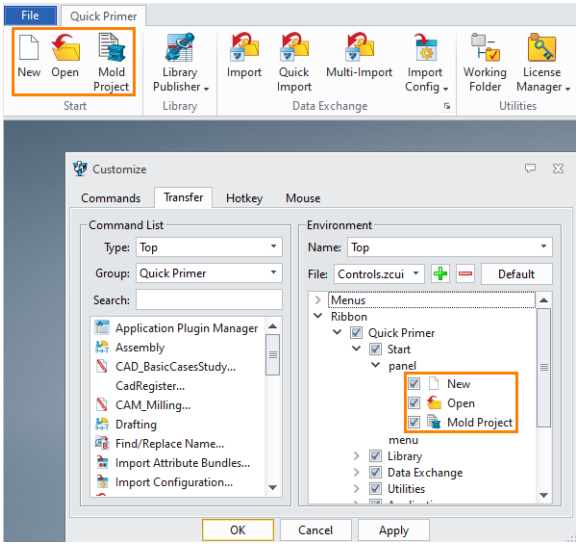



Figure 11 The Default Start Panel

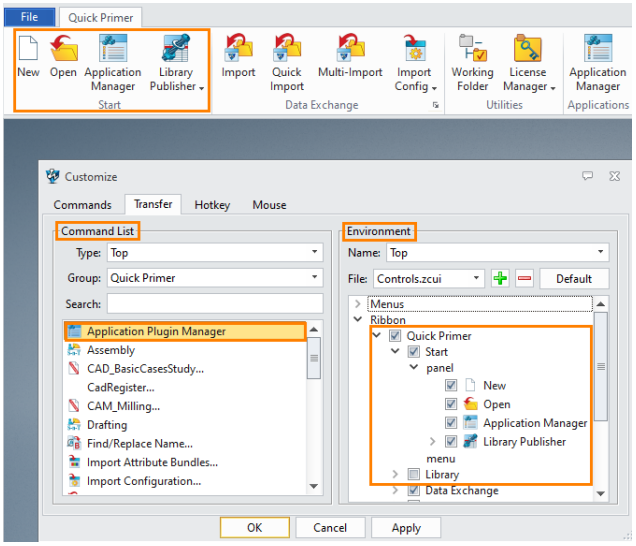



Figure 12 The Customized Start Panel

Besides, you can define your own hotkeys (keyboard shortcuts) and mouse actions of the scroll wheel and the right button.

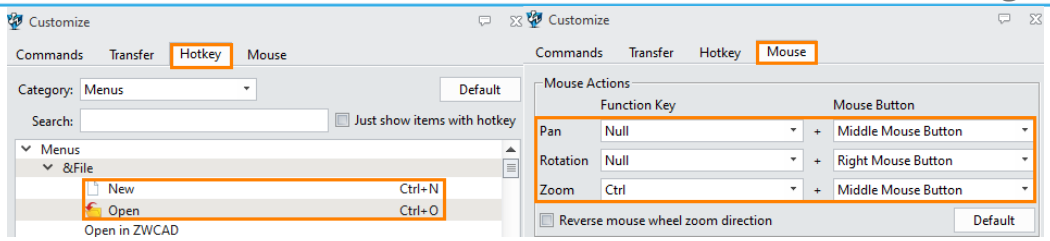



Figure 13 Customizing Hotkeys and Mouse Actions

## 2.4 Working Folder

It is a good habit for you to create a working folder because it makes accessing existing files easier, especially when your project involves many files of different formats.

STEP 01 Click the ***Working folder*** command in the ***Utilities*** panel to invoke the Select a Directory dialogue box.

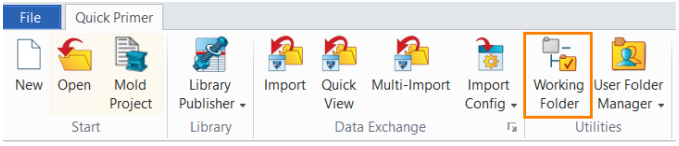



Figure 14 Set Working Folder

STEP 02 Select an existing folder or create a new working folder, then click ***OK***.

And your files will be saved to this folder by default.

## 2.5 File Management

In ZW3D, there can be two types of files: **multi-object** and **single-object** ones.

Multi-object files are in the format of .Z3 and unique to ZW3D. All ZW3D part, assembly, drawing, and CAM plan files can be managed in a single multi-object file, as shown in the figure below.

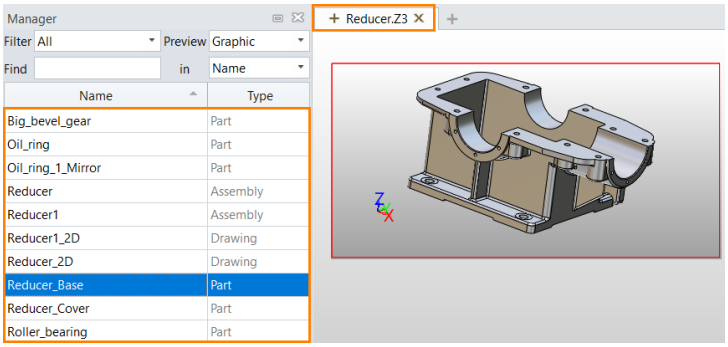



Figure 15 Multi-object File

As for single-object files, they are commonly used in 3D modeling software. It means that a single part/drawing/assembly object is saved as an independent file. Since it is not the default type, you need to tick the corresponding option in the ***General*** tab of the ***Configuration*** dialogue box, as shown in the figure below.

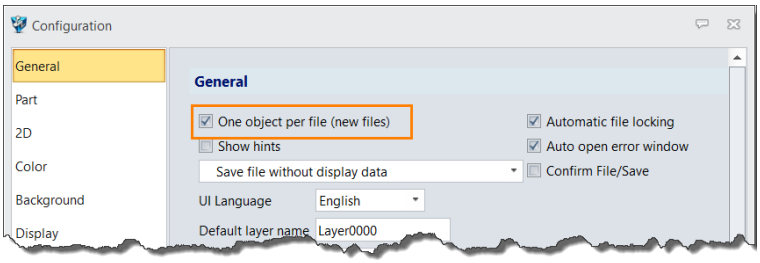



Figure 16 Making Single-Object Files the Default Type of Files

***Note:*** If you need to work with a PDM (Product Data Management) or PLM (Product Lifecycle Management) system, single-object files are strongly recommended for file management.

## 2.6 File Backup

File backup is vital to keep your data safe from sudden incidents like a power cut. In ZW3D, there are two ways to back up your files: auto and manual.

### 2.6.1 Auto Backup

When you create a new .Z3 file, the default backup file (*.Z3.z3bak) will be automatically generated and saved to the folder where your newly created file is.

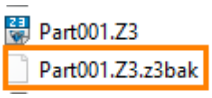



Figure 17 Auto File Backup

***Notes:***

- System backup works only when you do the first-time saving operation within a day, which means the rest of saving will not be backed up.
- The file will still be valid when its extension is changed from Z3.z3bak to Z3.

### 2.6.2 Manual Backup

You can manually back up files in the ***Configuration*** dialogue box, as shown in the figure below.

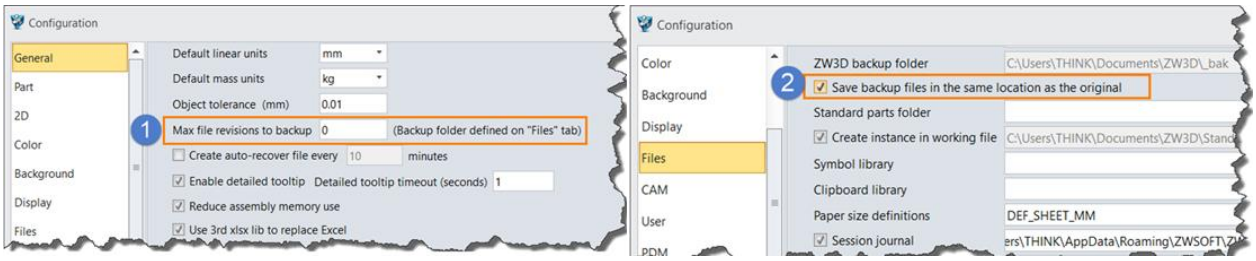



Figure 18 Manual File Backup

STEP 01 Enter a suitable number in the ***Max file revisions to backup*** box in the ***General*** tab, which determines how many backup files could be saved.

STEP 02 Determine the path of the backup folder in the ***Files*** tab. Also, it is suggested that you check the ***Save backup files in the same location as the original*** option for easier management and reuse.

STEP 03 Click ***Apply***, then ***OK*** to finish configuring. A new backup file will be automatically created when you save the file.

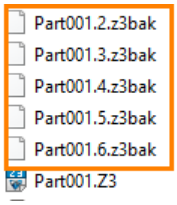



Figure 19 New Backup Files Will Be Automatically Created with Manual Backup

As you can see from Figure 19, Part001.1.z3bak, the first backup file of Part001.Z3 is absent. It's because the value of ***Max file revisions to backup*** was 5, while the original file has been saved for the sixth time. Therefore, please increase the value of ***Max file revisions to backup*** or delete the backup files you don't want to keep.

## 2.7 Object Picking

### 2.7.1 Single and Multi-Object Picking

You can click to pick a single object in the modeling area. If you want to unpick objects, simply keep pressing the ***Ctrl*** key while clicking. To select connected lines, just keep pressing the ***Shift*** key while clicking. Some commands, for example, Chamfer, allow you to pick connected lines before or after invoking them.

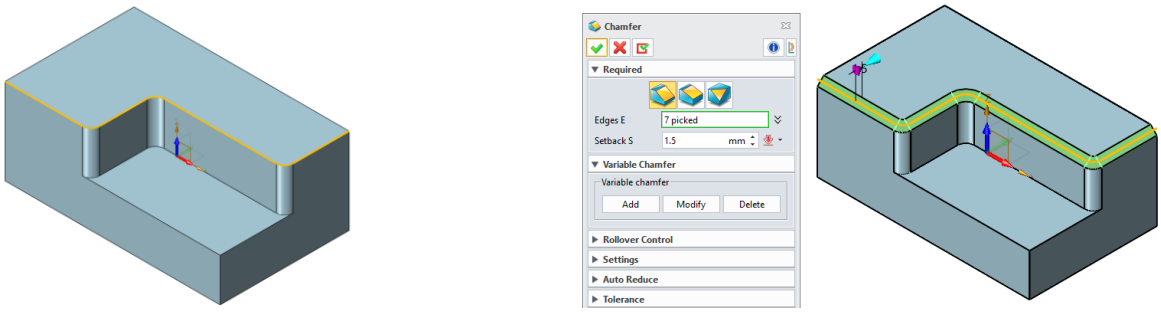



Figure 20 Selecting Connected Lines with Shift

### 2.7.2 Picking with Filter

Filters can help you pick objects faster. For example, as Figure 21 shows, after setting ***Feature*** as the filter, only the features of the part will be pre-highlighted when your mouse hovers over one.

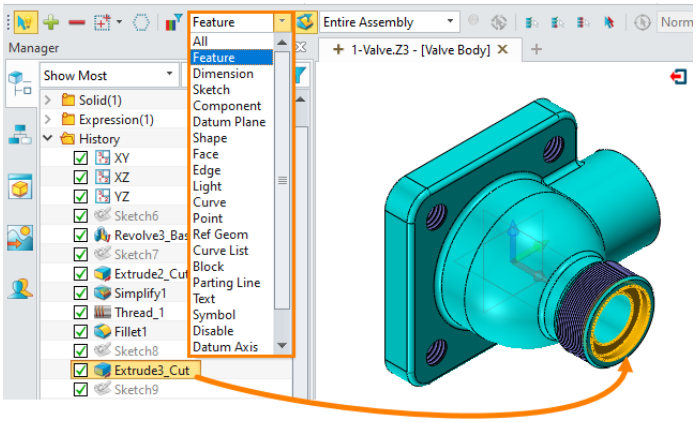



Figure 21 Picking with Filter

### 2.7.3 Picking Covered Objects

There are two methods to pick your target object that is inside or covered by another object.

One is to place your mouse over your target object and keep holding the ***Alt*** key. Then, it will be pre-highlighted in yellow, as shown in Figure 22.

The other is to right-click on the target area, select ***Pick from List*** in the context menu, and pick the target object. As Figure 23 shows, the covered face, F1 is easily picked.

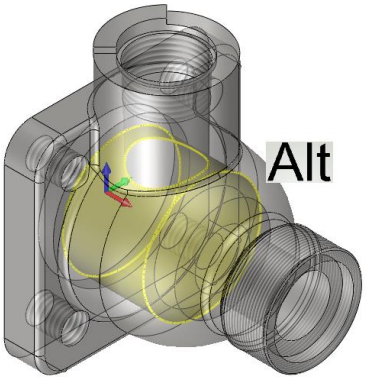



Figure 22 Picking with Alt

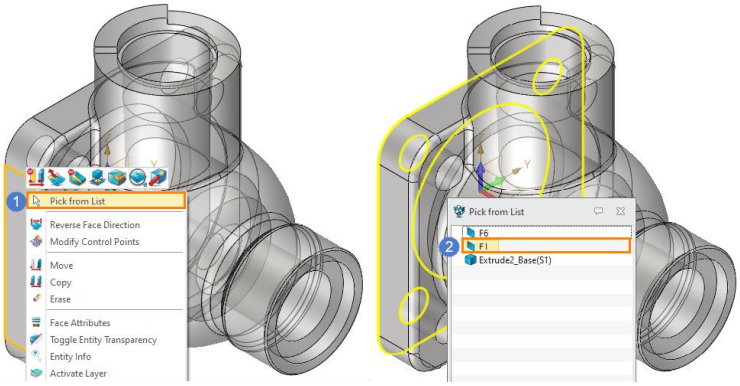



Figure 23 Picking from List

# 3 2D Sketch

2D sketches are fundamental to solid modeling because you can create features and define cross-sectional shapes with them on any planar face. Although most of the time, they will not be outputted as the final designs, they often represent the most important ideas of a feature or even the whole part.

## 3.1 Creating Sketches

In ZW3D, you can create two basic types of sketches. One is sketches that are part of part files, including external and internal ones. The other is standalone sketches, which can be saved as independent files.

### 3.1.1 Sketches as Part of Part Files

In most cases, a sketch is created as a feature in the part modeling environment. After creating a new part, you can invoke ***Sketch*** by right-clicking on the blank area and selecting it in the context menu, or clicking the corresponding icon in the ***Shape*** tab of the Ribbon interface, as shown in Figure 24. Sketches created these ways are called external sketches, which can be reused for many different modeling features.

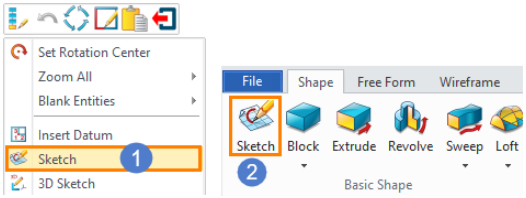



Figure 24 Ways to Create External Sketches

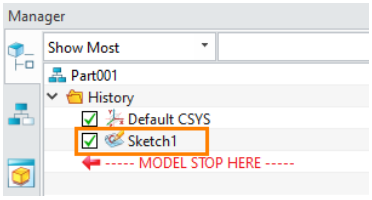



Figure 25 An External Sketch

As for internal sketches, they are created during the execution of a modeling command, such as ***Extrude***, ***Revolve***, ***Sweep***, etc. An internal sketch can only be used for its corresponding modeling feature.

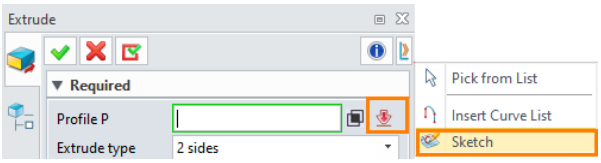



Figure 26 Creating Internal Sketches

You can convert internal sketches into external ones by right-clicking on the internal sketch and selecting ***Make External*** in the context menu, as shown in Figure 28.

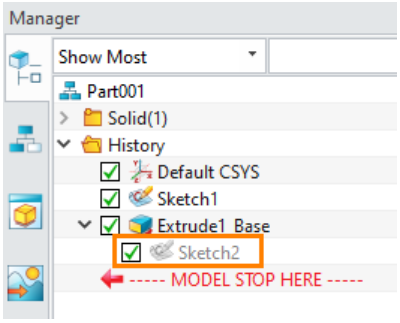



Figure 27 An Internal Sketch Is under Its Corresponding Modeling Feature

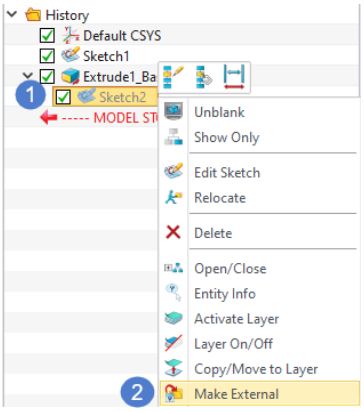



Figure 28 Converting to External Sketches

### 3.1.2 Standalone Sketches

As shown in the figure below, you can choose to create a new ***Standalone Sketch*** file. With this independent file, you can discuss your ideas with team members conveniently.

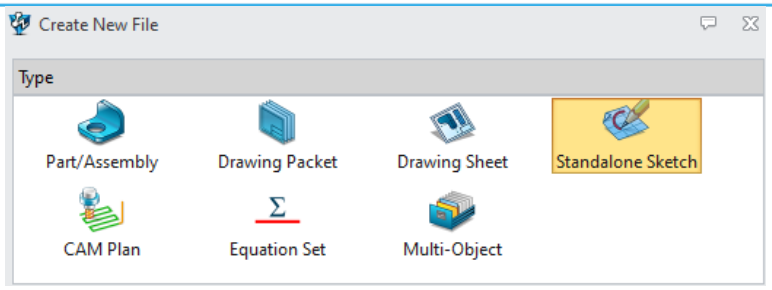



Figure 29 Standalone Sketch

## 3.2 Sketch Settings & Operations

### 3.2.1 Basic Settings and Operations

In the sketch environment of ZW3D, most of the basic sketch settings and operations can be found in the ***Document Aware Toolbars***.


Figure 30 Document Aware Toolbars

The six frequently used functions are explained in the table below.

| | | |
|---|---|---|
| | ***Exit*** | Exit the sketch environment. |
| All | ***Pick Filter*** | Define the filter for object picking |
| | ***Snap Filter*** | |
| | ***Plane View*** | Return to the plane view. |
| | ***Toggle Grid*** | Show or hide the grid. |
| | ***Zoom Control*** | Zoom your sketch view. |

### 3.2.2 Advanced Settings

In the sketch environment, click ***Preferences*** in the ***Settings*** panel to invoke the ***Sketch Settings*** dialogue box. Then, input a number in the ***Grid Spacing*** box according to your need and click ***OK***.

If you want to modify more sketch settings, you can invoke the ***Configuration*** dialogue box, click the ***2D*** tab, and check or uncheck the ***Sketch*** options, as Figure 32 shows.

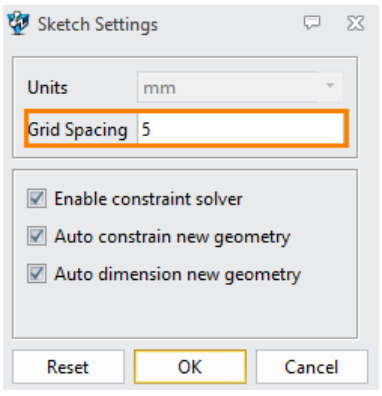



Figure 31 Setting Grid Spacing

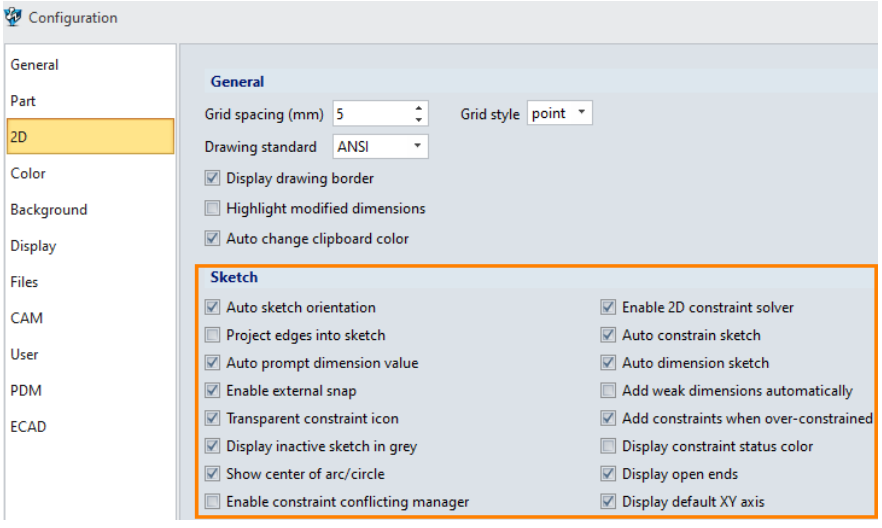



Figure 32 More Sketch Settings

## 3.3 Sketch Flow & Elements

### 3.3.1 Sketch Flow

Below is the sketch flow of ZW3D.

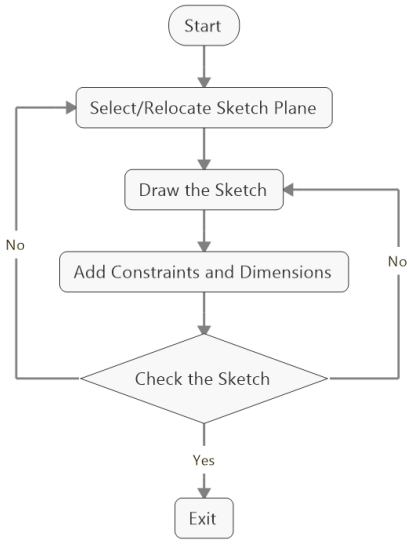



Figure 33 Sketch Flow of ZW3D

To create a new sketch, the first step is to select a ***Sketch Plane***. As Figure 34 shows, the plane you are selecting, in this case, XY will be highlighted. Then, middle-click or click the OK button to enter the sketch environment.

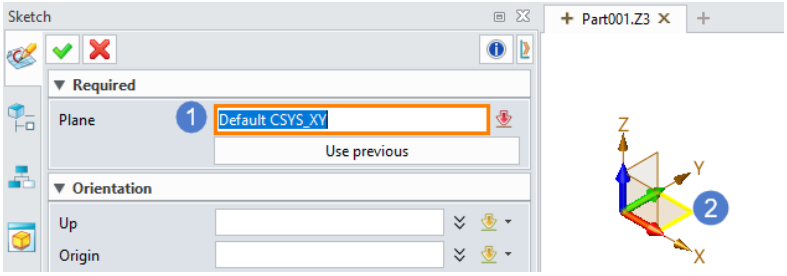



Figure 34 Selecting a Sketch Plane

### 3.3.2 Sketch Elements

After entering the sketch environment, you need to draw some objects and add constraints and dimensions to them. Sketch plane, sketch geometries, sketch constraints are the three basic types of sketch elements. Fully constrained sketches would be regarded well-defined.

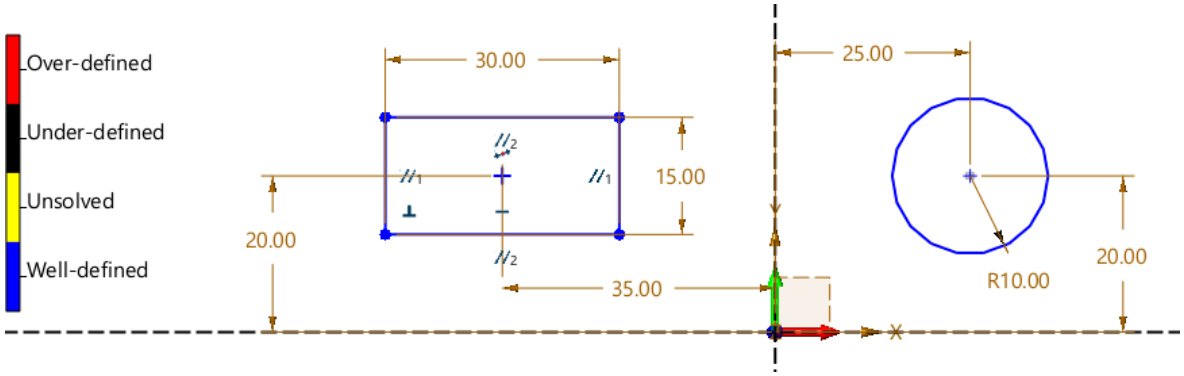



Figure 35 Elements of a Well-Defined Sketch

## 3.4 Sketch Constraints & Dimensions

Theoretically, any sketch can be confirmed with its shape and position properly constrained. In ZW3D, both shape and position can be constrained with dimensions and geometric relationships.

### 3.4.1 Geometric Constraints

In the sketch environment, you can find all the geometric constraints in the ***Constraint*** panel under the ***Constraint*** tab.

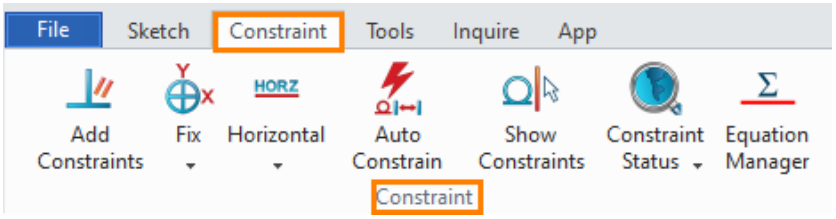



Figure 36 Geometric Constraints

There are two ways to add constraints. One is to pick the objects first, then click the ***Add Constraints*** command and select a proper type of constraint. In this way, a type of constraint is automatically selected by the system. The other is to select a type of constraint first and then pick the target objects.

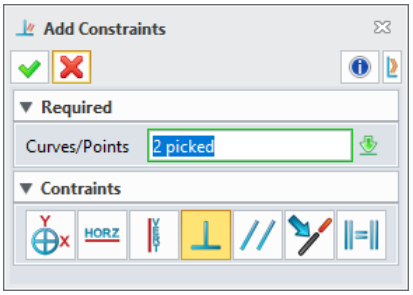



Figure 37 Selecting Objects First

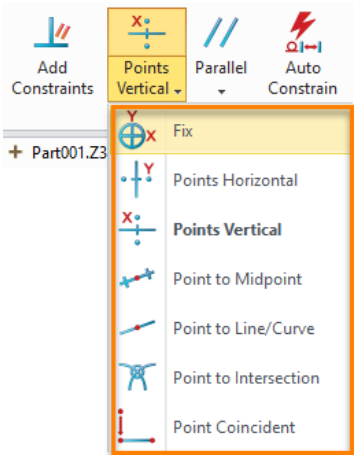



Figure 38 Selecting a Type of Constraint First

If your sketch is complicated, adding constraints could be challenging. To make it easier, please turn on the button so that it can tell you the status of the constraints in real time. Well-defined sketches will be in blue.

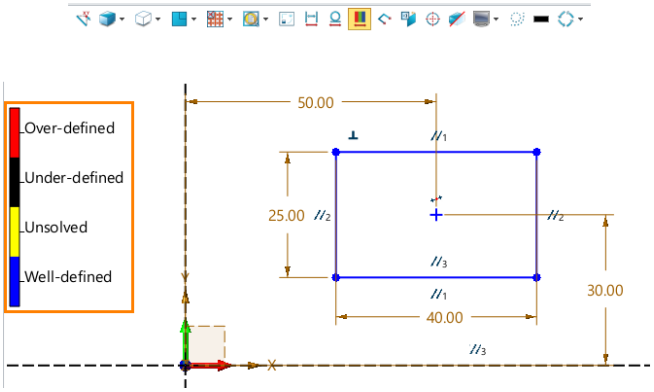



Figure 39 Different Statuses of Constraints Are in Different Colors

When the constraints are conflicted, you can use the ***Conflicting Constraints*** command to invoke the Conflicting Constraints Manager dialogue box and solve the conflicts.

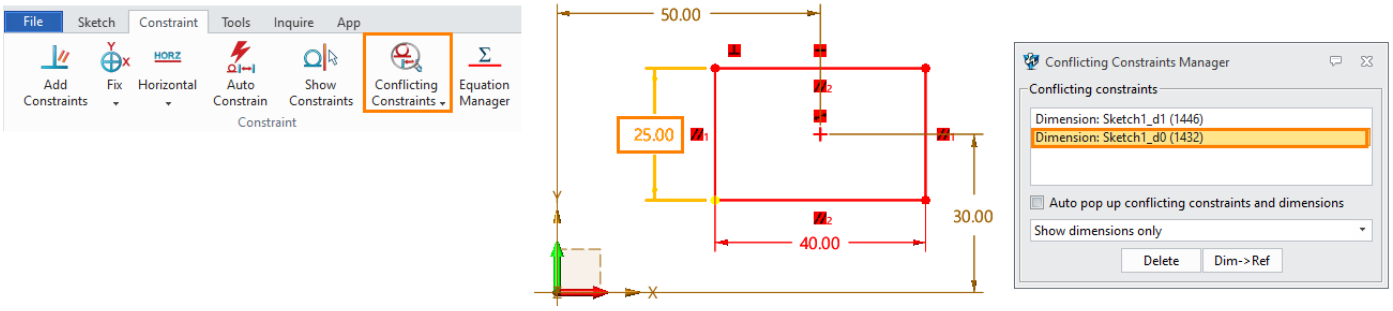



Figure 40 Check and Solve Conflicts of Constraints

### 3.4.2 Dimensional Constraints

With the ***Quick Dimension*** command, you can easily add dimensional constraints.

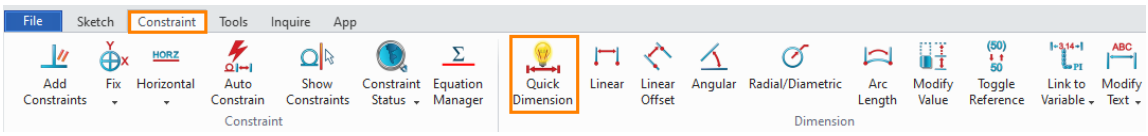



Figure 41 Quick Dimension

As Figure 42 shows, the dimensions you add manually are strong dimensions that determine the change of the whole sketch.

It's recommended that you activate the ***Add Weak Dimensions Automatically*** mode because dimensions created by the system are regarded as weak dimensions so that you can easily constrain the whole sketch. As you can see from Figure 43, weak dimensions are in gray.

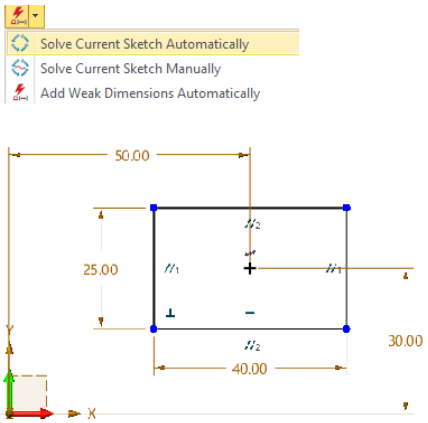



Figure 42 Strong Dimensions

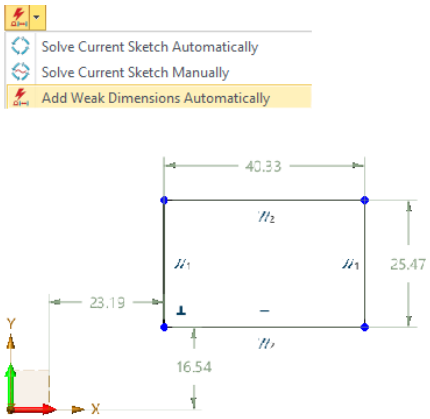



Figure 43 Weak Dimensions

In some cases, **reference dimensions** will be added to the well-defined sketch to make it easier to understand. As Figure 44 shows, they will be in brackets. For example, the length (16.77) is a reference dimension.

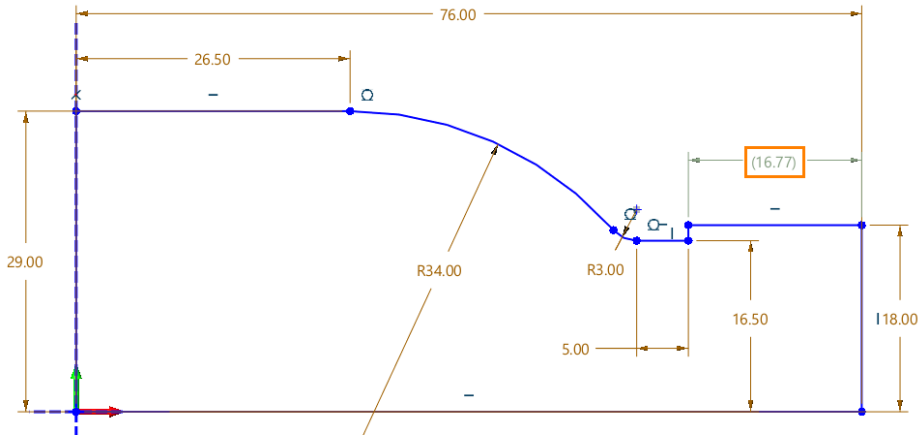



Figure 44 Reference Dimensions

***Note:*** You can hide dimensions and constraints to get a clearer view of the target sketch.


Figure 45 Showing or Hiding Dimensions and Constraints

## 3.5 Points of Attention

### 3.5.1 Sketch Grid Setting

As mentioned in **Chapter 3.2.1**, you can control the display of sketch grid and choose a type of grid in the ***Document Aware Toolbar***.

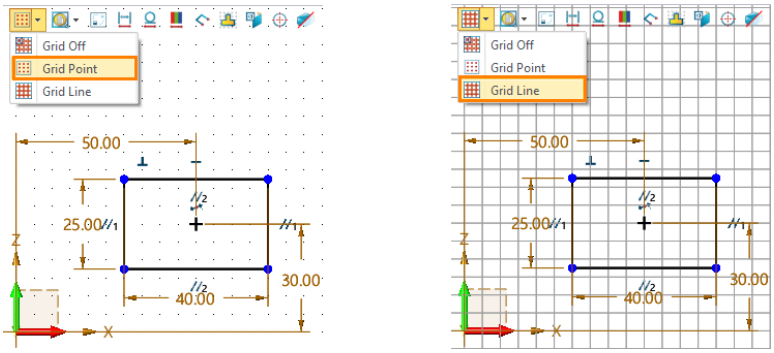



Figure 46 Grid Display

If you need to customize the grid spacing, simply go to ***Preferences*** -> ***Grid Spacing***.

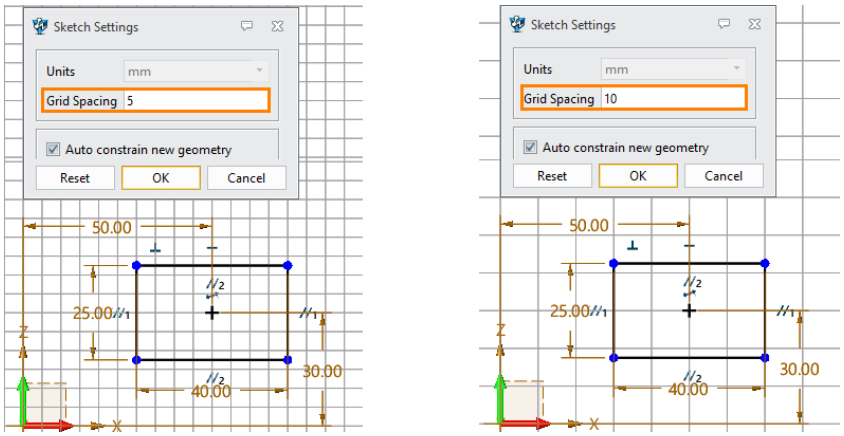



Figure 47 Grid Space Setting

### 3.5.2 Construction Geometries

Construction geometries are references that can help you achieve a final sketch and will not appear in the modeling environment. For example, to quickly draw a hexagon, you can draw a circle first. Then, convert it to a construction geometry by right-clicking on the circle and clicking the ***Toggle Type*** icon in the context menu, as shown in Figure 48.

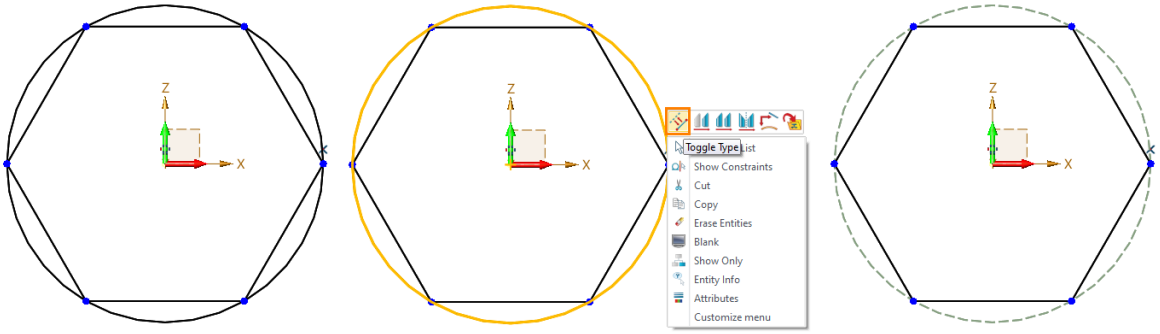



Figure 48 Toggling Types of Geometries

As Figure 49 shows, the construction geometry will not appear in the modeling environment.

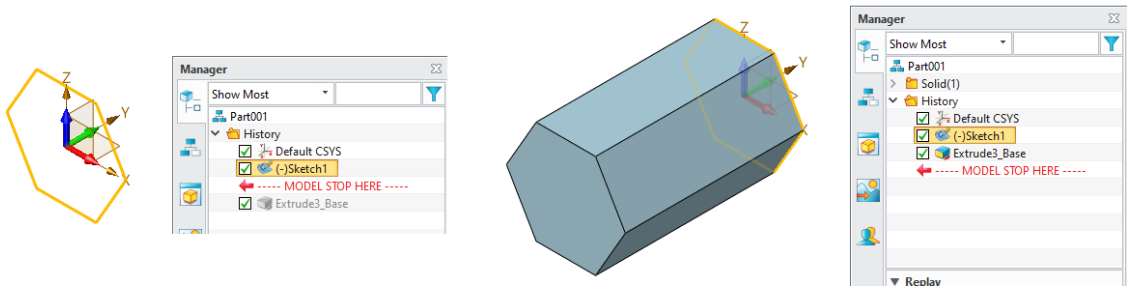



Figure 49 Construction Geometries Will Not Appear in the Modeling Environment

### 3.5.3 Trimming Tools

When drawing a sketch, trimming is an inevitable operation. Among the many trimming tools, ***Power Trim*** and ***One Touch Trim*** allow you to trim the sketch quickly, and ***Trim/Extend to Corner*** to make corners.

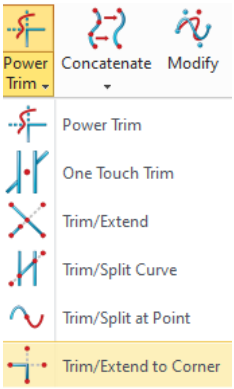



Figure 50 Trimming Tools

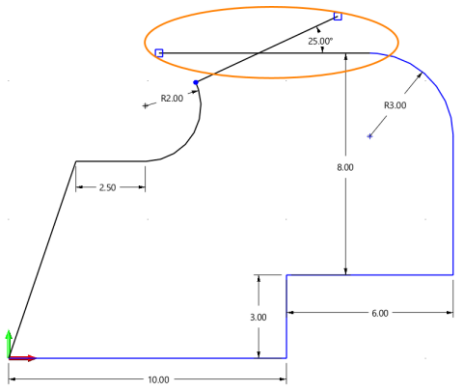



Figure 51 Before Trimming

For example, to trim the sketch in Figure 51, you can use ***Power Trim*** by pressing and holding down the left mouse button while moving the cursor above the parts you want to trim.

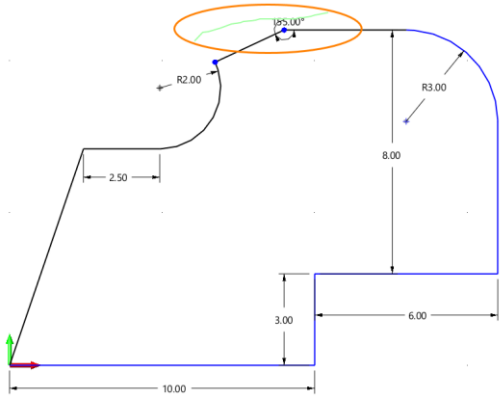



Figure 52 Using Power Trim

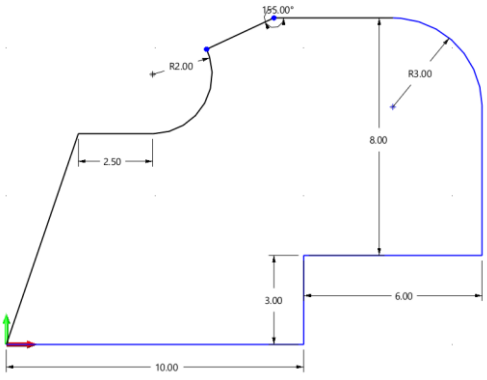



Figure 53 After Trimming

### 3.5.4 Checking the Sketch

After finishing the whole sketch, you might get confused when some 3D modeling commands can't be used on it. Oftentimes, it is because there are overlapping or unconnected lines in the sketch.

To check whether the sketch is closed or not, you can use two tools in the ***Document Aware Toolbars***. One is ***Display open ends on/off***, which will emphasize the unconnected endpoints of lines. The other is ***Closed Rings On/Off***, which will color the closed area.

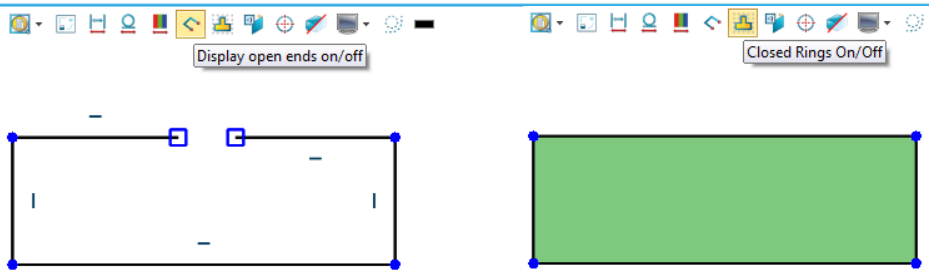



Figure 54 Checking Unconnected Lines

To check overlapping lines, you can either use the ***Overlap*** command *(**Inquire*** tab -> ***Sketch Doctor*** panel -> ***Overlap**)* or the ***Display open ends on/off*** command.

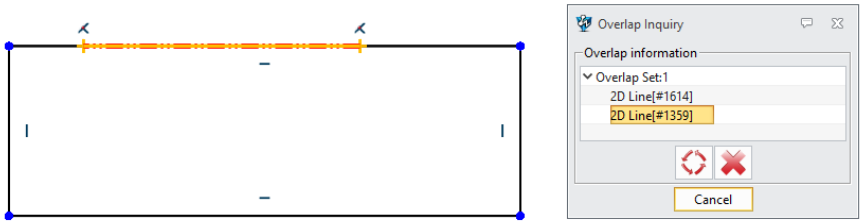



Figure 55 Checking Overlapping Lines

### 3.5.5 Modifying Dimensions and Delay Update

Modifying dimensions sometimes results in strange shapes because the whole sketch will change after one of the dimensions is modified. As shown in Figure 56, the whole sketch became deformed when the radius was changed from 2.00 to 8.00.

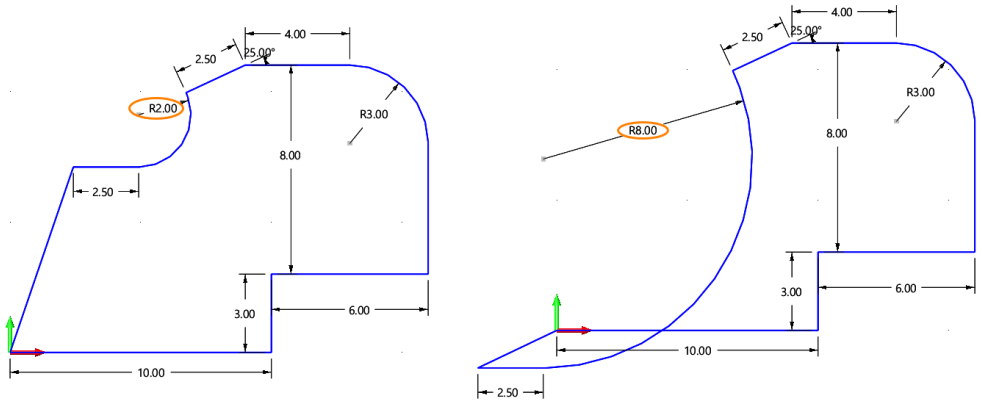



Figure 56 Deformed Sketch Due to Modification of Dimensions

Besides, you can activate ***Delay Update*** when modifying a set of dimensions.

STEP 01 Double-click on an existing dimension or create a new one to invoke the ***Input Dimension Value*** dialog box, tick the ***Solve manually*** option, input a value, and click ***OK***.

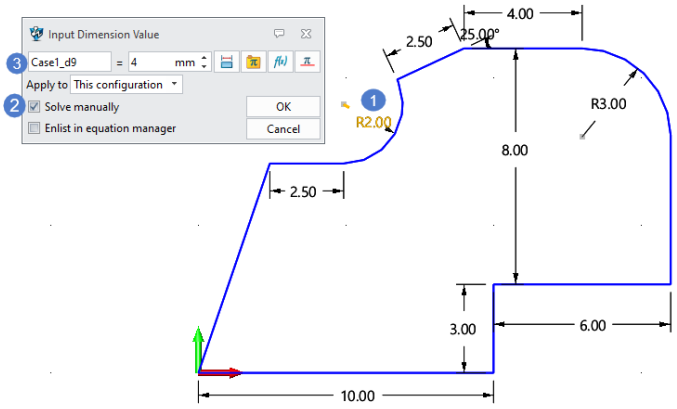



Figure 57 Input Dimension Value

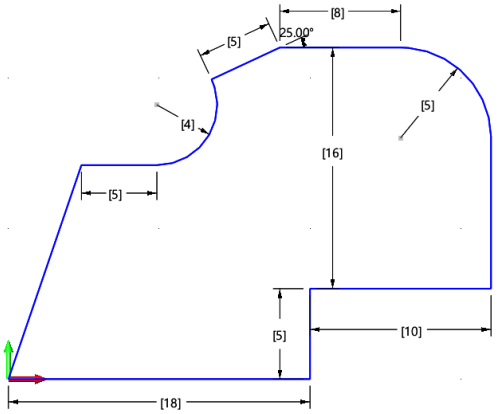



Figure 58 Modifying Other Dimensions

STEP 02 Modify other dimensions, and their values will be in brackets, meaning that they are not updated yet.

STEP 03 Click ***Solve Current Sketch Manually*** to update the whole sketch.

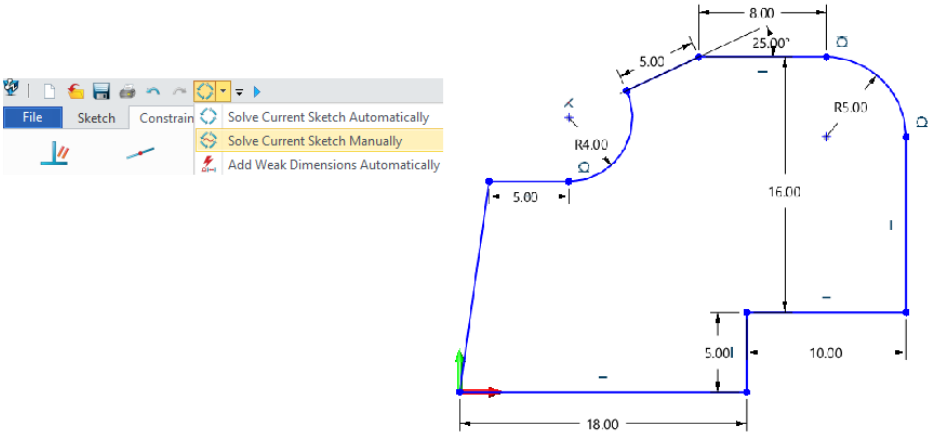



Figure 59 Update All the Dimensions

### 3.5.6 Dimension Display

Some dimensions of a parametric sketch are linked with variables or expressions. In this case, you can change the type of dimension display (***Tools*** tab -> ***View*** panel -> the accordion -> ***Dimension Display*** -> ***Expressions***), as shown in Figure 60. After that, both dimension values and expressions will appear, as shown in Figure 61.

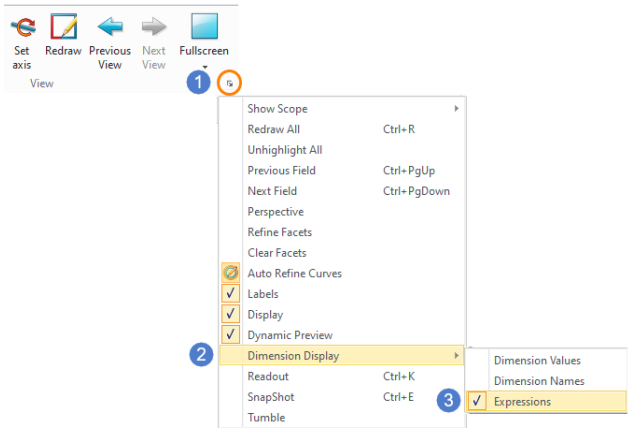



Figure 60 Switch the Type of Dimension Display

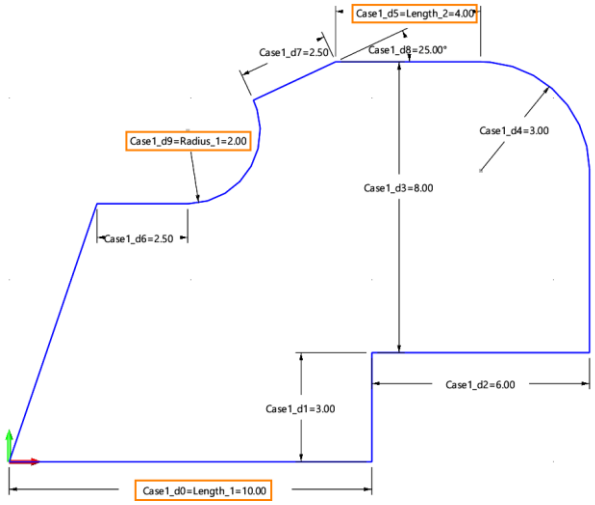



Figure 61 Displaying Dimensions and Expressions

### 3.5.7 Relocating the Sketch Plane

The ***Relocate*** command in the ***Settings*** panel under the ***Sketch*** tab can help you specify a sketch plane or switch to another.

When modeling, you can directly right-click on the target sketch in the history manager and select ***Relocate***.

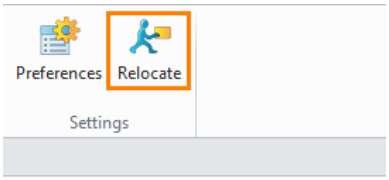



Figure 62 Relocating in the Sketch Environment

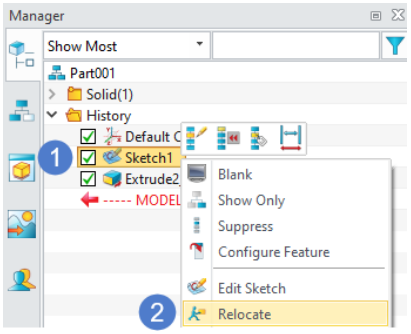


Figure 63 Relocating in the Modeling Environment

For example, if you want to change the sketch plane from XY to XZ, you can pick the target XZ plane in the Relocate dialogue box.

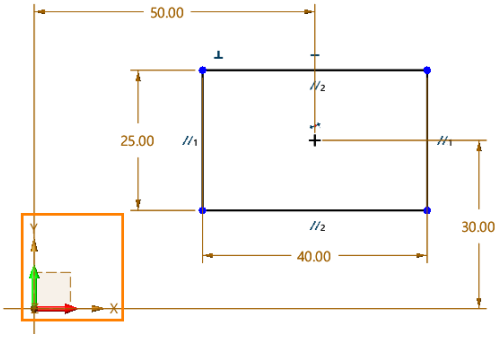
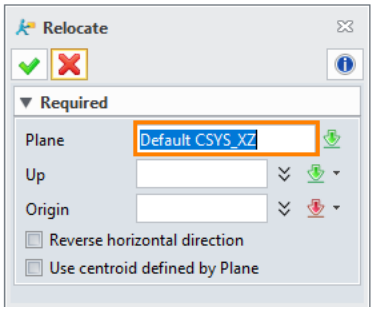



Figure 64 The Sketch in the XY Plane



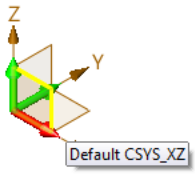




Figure 65 Picking the Target Sketch Plane

Click ***OK***, then the sketch plane will be changed to XZ, as shown in the figure below.

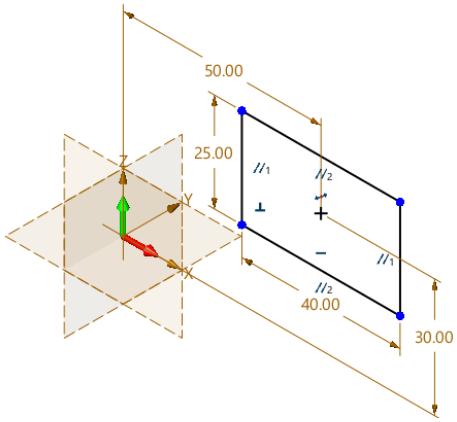



Figure 66 Relocating the Sketch Plane to XZ

## 3.6 Sketch Cases

Please follow the steps of the cases and practice in ZW3D so that you will have a better understanding of the sketch module and making a well-defined sketch.

### 3.6.1 Case 1 – Creating the Profile of the Feature of a Valve Body

The main feature of the valve body below is ***Revolve***. The revolved part is highlighted in yellow in Figure 67. The profile of this feature is highlighted in yellow in Figure 68. Follow the steps and you shall recreate one.

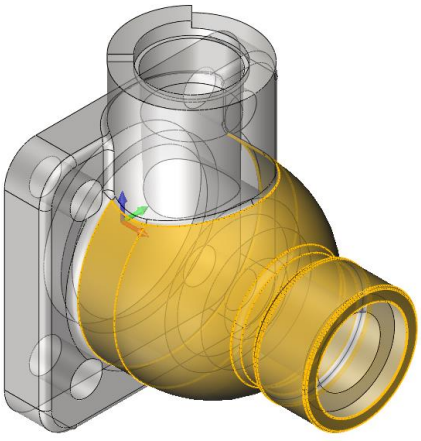


Figure 67 The Revolved Feature of the Valve Body

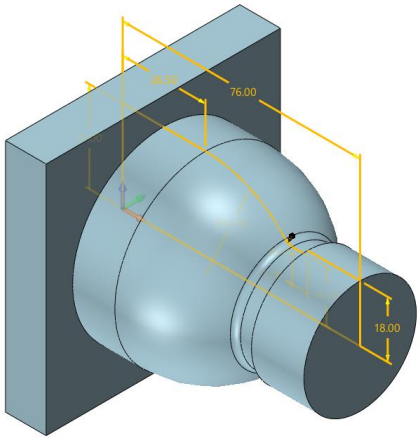



Figure 68 The Profile of the Revolved Feature

STEP 01 Go to the ***Shape*** tab and click ***Sketch***, then select the XZ plane as the sketch plane, as shown in Figure 69.

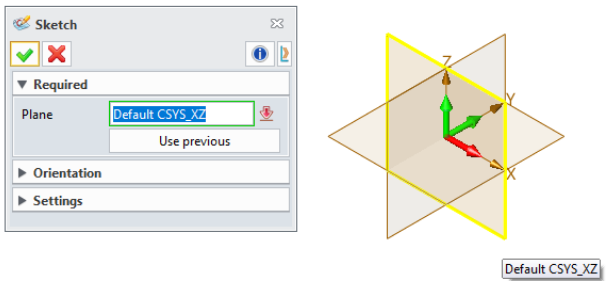



Figure 69 Select the XZ Plane as the Sketch Plane

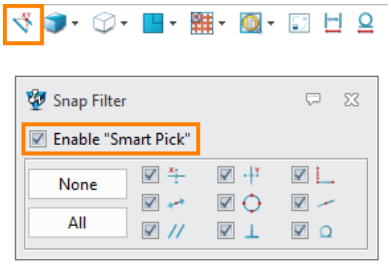



Figure 70 Enabling Smart Pick

STEP 02 Use the ***Draw*** command to draw a continuous line vertically from the origin and then horizontally. Make sure ***Smart Pick*** is enabled in the ***Snap Filter*** dialogue box for a quicker sketch process.

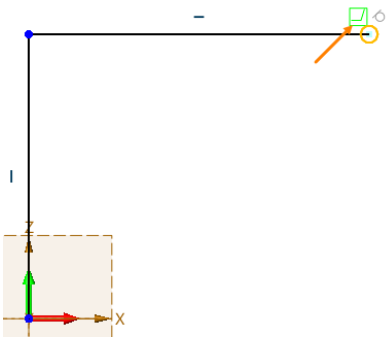


Figure 71 Drawing a Continuous Line

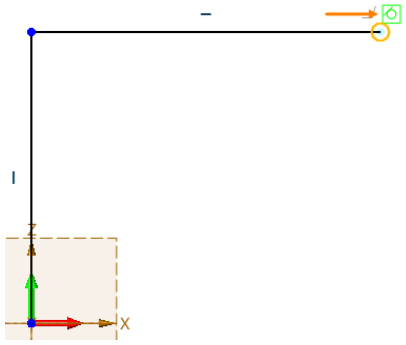


Figure 72 Switching to the Arc Mode

STEP 03 Click inside the yellow circle at the endpoint of the line and switch from the line mode to the arc mode of the ***Draw*** command.

STEP 04 Draw two arcs, as shown in Figure 73.

STEP 05 Switch back to the line mode and draw the rest of the lines.

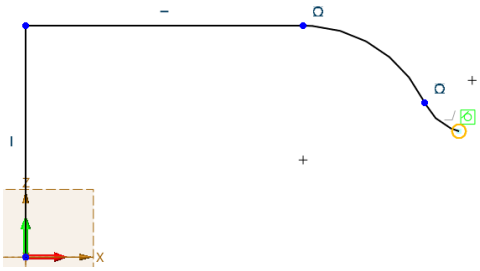


Figure 73 Drawing Two Arcs

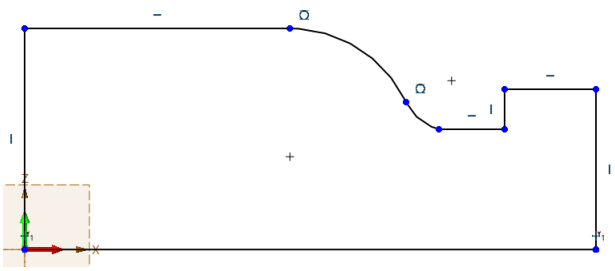


Figure 74 Drawing the Rest of the Lines

***Notes:*** Buttons related to the status of the sketch in the ***Document Aware Toolbar***, especially the color codes of definition, should be turned on. Such a real-time view of the sketch will help you in adding constraints and dimensions.

STEP 06 Use the ***Add Constraints*** command to add proper constraints one by one. As shown in Figure 75, you can easily add tangent constraints to the arcs and lines.

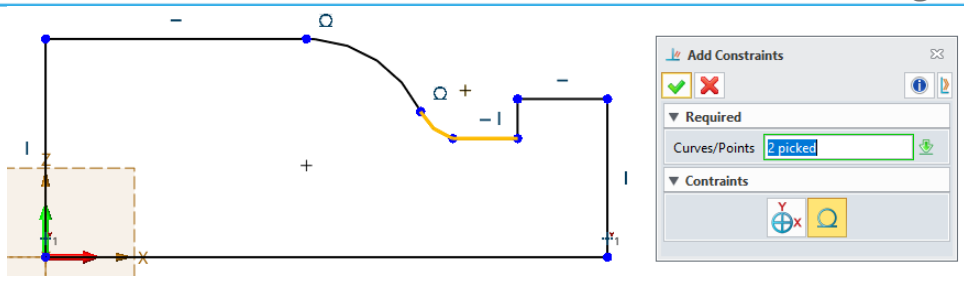



Figure 75 Adding Proper Constraints

STEP 07 Add dimensions with the ***Quick Dimension*** command until the whole sketch is well-defined.

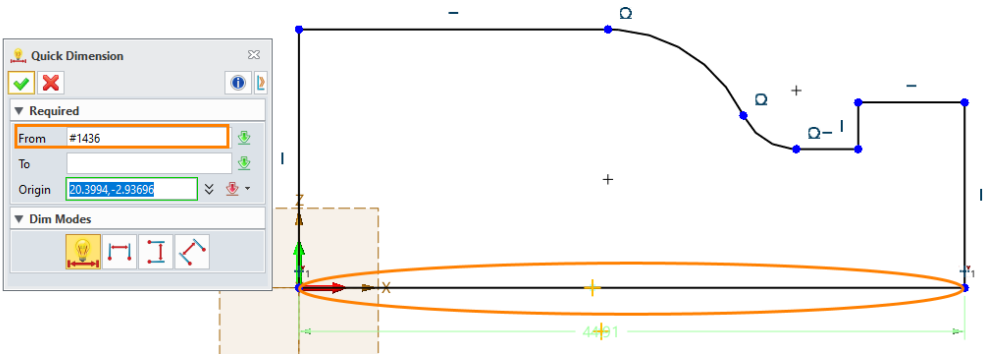



Figure 76 Adding Dimensions

STEP 07 Click the graphic area, input the value of the length of the line, and click ***OK***. The result is shown in Figure 77. Or you can specify the dimension with two points, as Figure 78 shows.

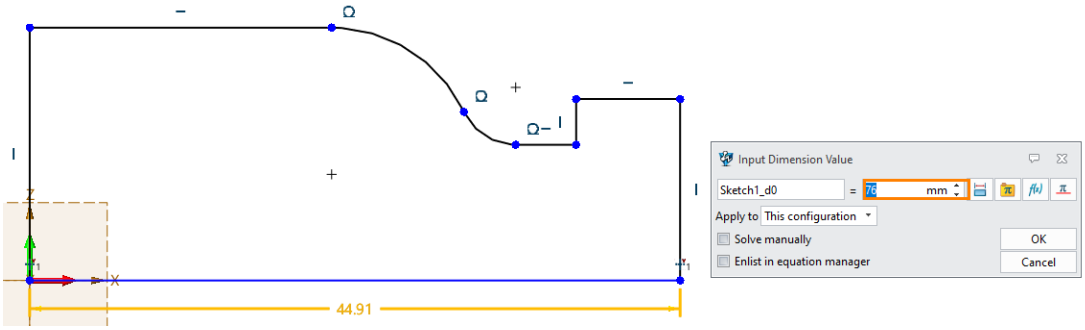



Figure 77 Defining the Value of the Dimension

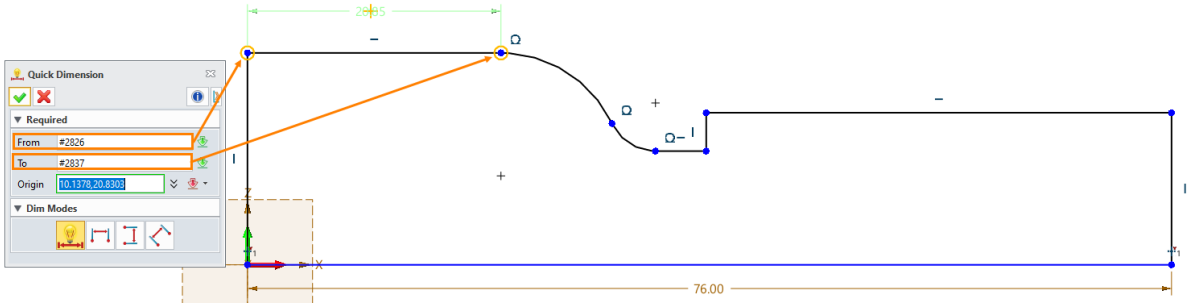



Figure 78 Specifying the Dimension with Points

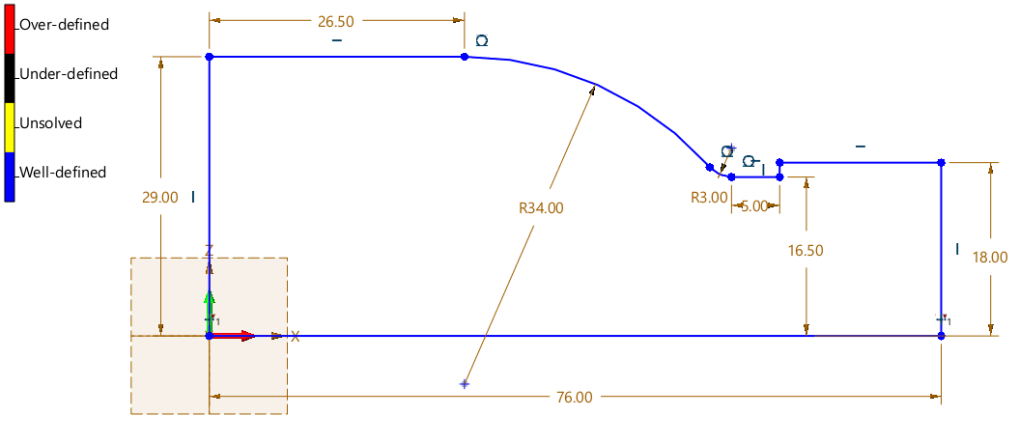



Figure 79 All the Dimensions Are Added

STEP 08 Check the profile with the ***Overlap*** or ***Display open ends on/off*** command and exit the ***Sketch*** module.

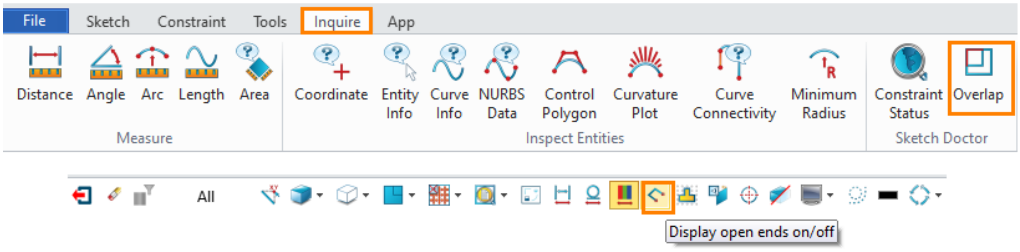



Figure 80 Checking the Profile

### 3.6.2 Case 2 – Creating the Profile of the Feature of a Wrench

The main feature of the wrench below is ***Extrude***, which means you'll need to draw the profile first.

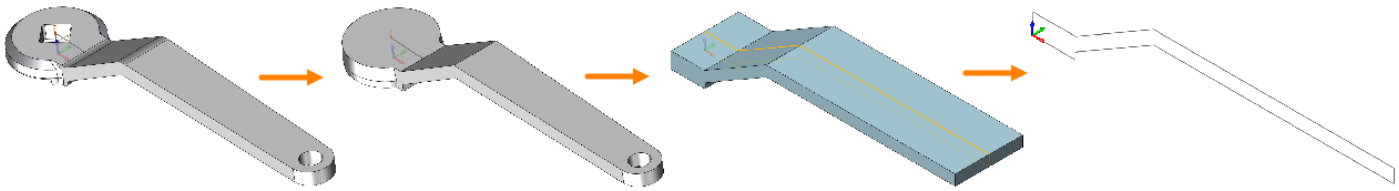


Figure 81 Profile for Extrude Feature of Wrench

STEP 01 Go to the ***Shape*** tab and click ***Sketch***, then select the XZ plane as the sketch plane.

STEP 02 Use the ***Draw*** command to draw a continuous line as shown in Figure 82. Make sure ***Smart Pick*** is enabled.

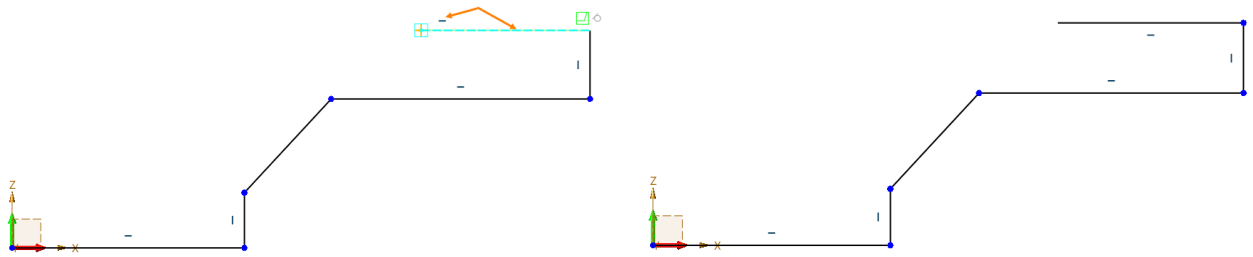


Figure 82 Drawing a Continuous Line

STEP 03 Add the ***Equal*** constraint to the highlighted lines, as shown in the left side of Figure 83 to get the result on the right side.

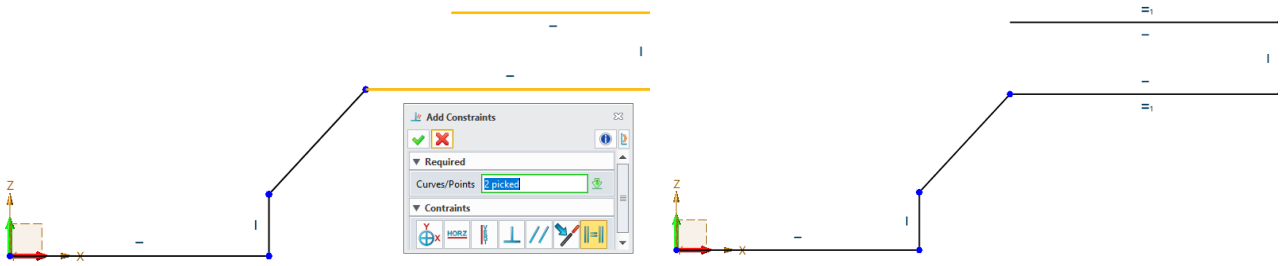



Figure 83 Adding the Equal Constraint to the Lines

STEP 04 Draw a slanting line parallel to the slanting line below and keep their endpoints vertically aligned, as shown in the left side of Figure 84 to get the result on the right side.

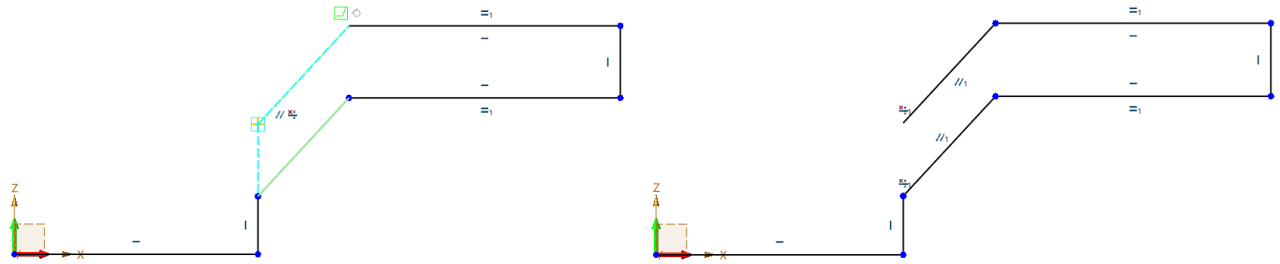


Figure 84 Drawing the Parallel Slanting Line

STEP 05 Draw the rest of the lines to get a rough profile is finished, as shown in the right side of Figure 85.

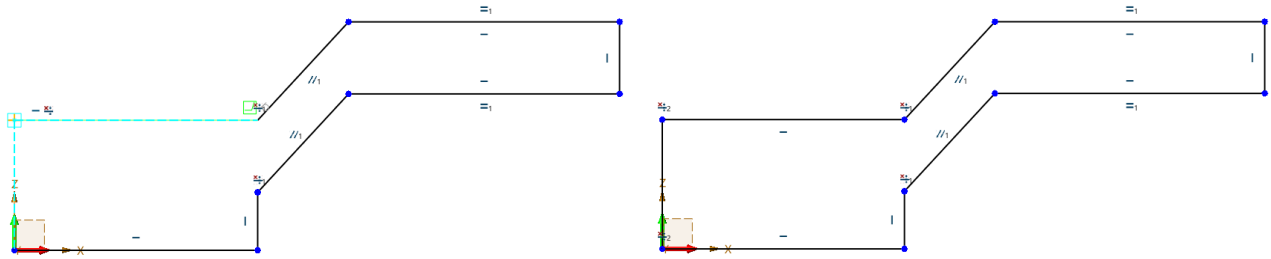


Figure 85 Finishing the Rough Profile

STEP 06 Add dimensions until the profile is blue (well-defined). For example, use the ***Angular*** command to specify the angle between two lines.

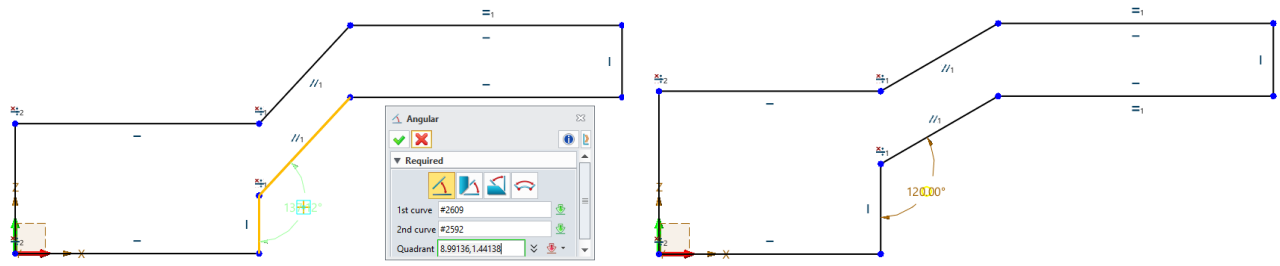



Figure 86 Specifying the Angular Dimension

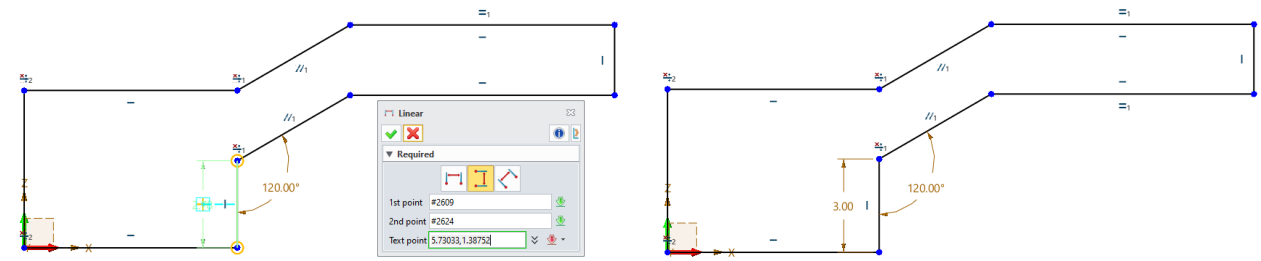



Figure 87 Defining Linear Dimensions with the Quick Dimension or Linear command

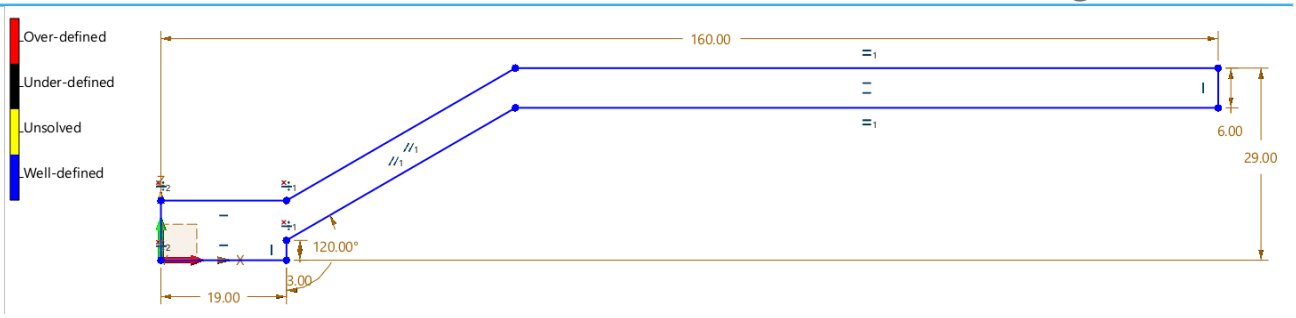



Figure 88 The Final Well-Defined Sketch

STEP 07 Check the profile and exit the ***Sketch*** module.

**Summary**

The points below can help you draw a well-defined sketch efficiently.

- Choose the right sketch plane.
- Better start from the default origin.
- Pay attention to the ***Smart Snap*** prompt. Accept the confirmed constraints and avoid adding constraints you are uncertain of to avoid making an over-defined sketch.

# 4 Modeling

Creating a 3D geometric model is fundamental to the whole product design and future applications of its data like engineering drawing, CAM(Computer-Aided Manufacturing), and CAE(Computer-Aided Engineering ), etc.

## 4.1 Basic Modeling Concepts

### 4.1.1 Feature-Based Modeling

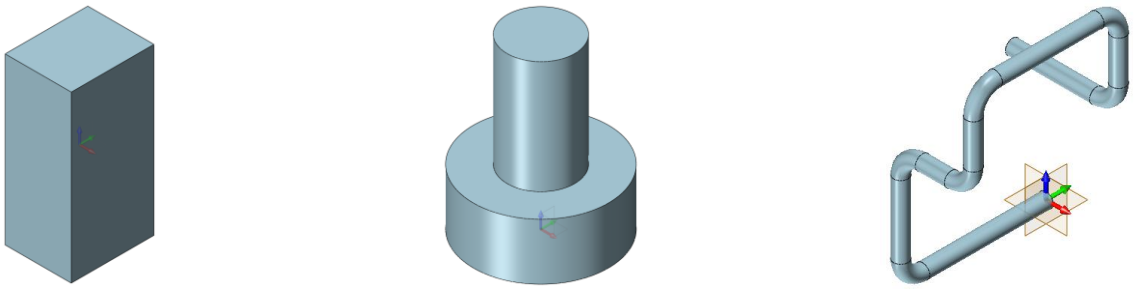
Feature-based modeling means that a feature is regarded as the basic modeling unit and that a 3D model consists of various features. Generally, modeling features can be divided into three types.

- **Datum feature**

Datum features include datum planes, axes, and point.

- **Basic feature**

Basic features include extrude, revolve, and sweep features.



Extrude Feature　　Revolve Feature　　Sweep Feature

Figure 89　Basic Features

- **Engineering feature**

Engineering features are mostly applied to the actual field of engineering, including chamfer, fillet, draft, etc.

Fillet Feature　　Chamfer Feature

Figure 90　Engineering Features

### 4.1.2 Solid Shape and Surface Shape

There are two types of shapes: solids and surfaces. In ZW3D, a closed geometry is considered a solid, otherwise, it is a surface, as shown in Figure 91. The unique modeling method of ZW3D, hybrid modeling, allows you to switch between solids and surfaces freely. It means that if a shape is geometrically closed, it will be automatically filled and regarded a solid.

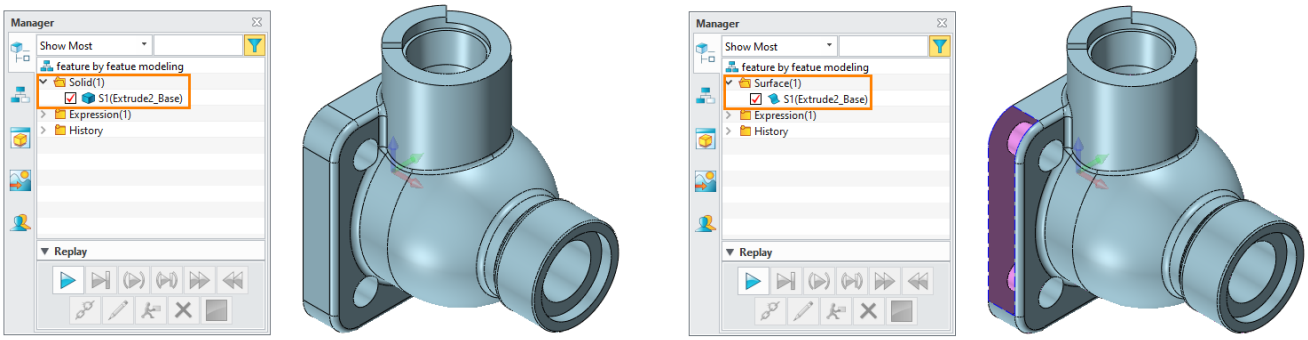
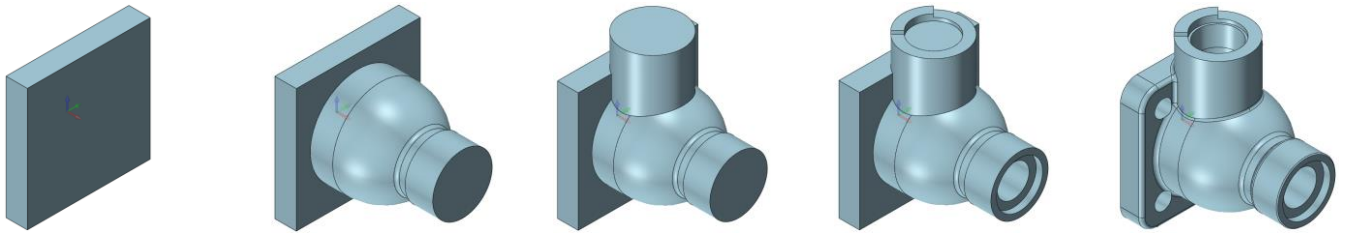



Figure 91 Solid Shape and Surface Shape

## 4.2 Parametric Modeling

### 4.2.1 Feature-Based Parametric Modeling

Feature-based parametric modeling means that the 3D model is created with various features and that the features are controlled by parameters. When you modify the feature parameters, the model will be modified or updated. As shown in Figure 92, the valve body model can be created with different features.



Extrude Base | Add a Revolve | Add Another Revolve | Cut Features | Engineering Features

Figure 92 Modeling with Features

### 4.2.2 Parametric Modeling Process

Generally, parametric modeling includes the following three steps.

- **Define the parameters (variables and expressions)**

In ZW3D, all the parameters can be defined in the ***Equation Manager***, which can be accessed in both the modeling (***Tools*** tab -> ***Insert*** Panel -> ***Equation Manager***) and sketch (***Tools*** tab -> ***Utilities*** Panel -> ***Equation Manager***) environments.

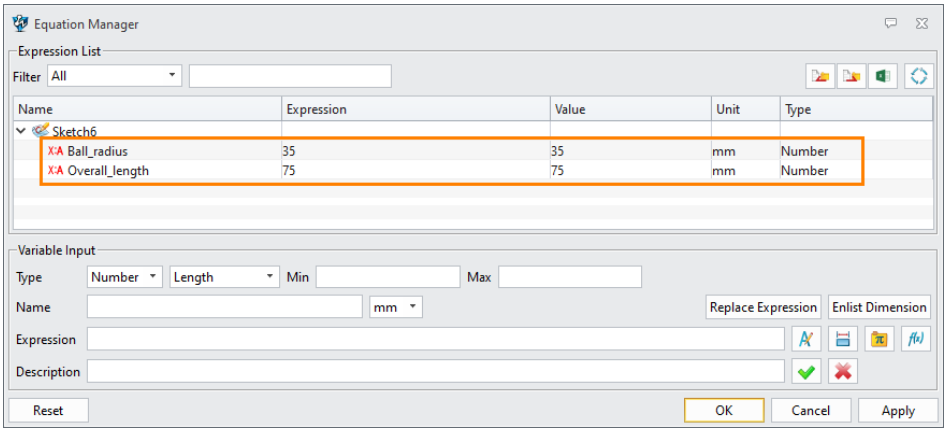



Figure 93 Equation Manager

- **Feature modeling and apply the parameters**

In the course of the feature modeling, you can apply parameters while adding dimensions to your sketch or creating features.

As Figure 94 shows, you can select the target parameters directly in the ***Variable Browser*** while adding dimensions to the sketch.

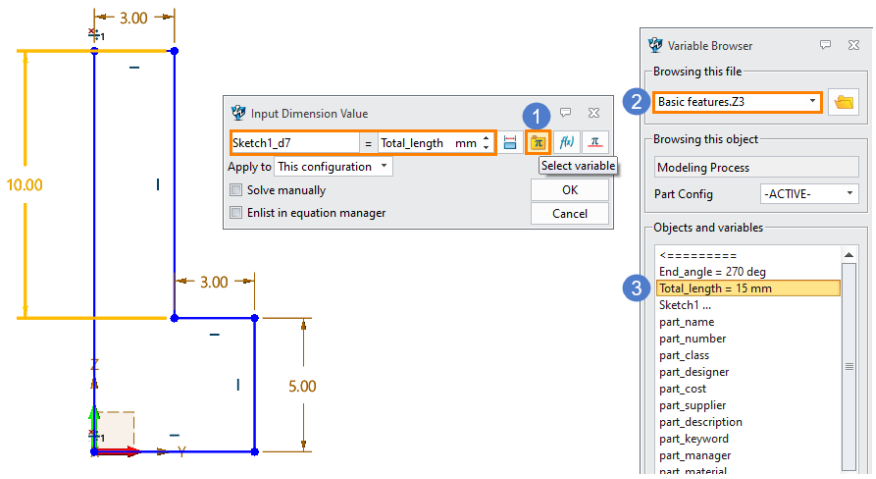



Figure 94 Applying Parameters While Adding Dimensions to the Sketch

When creating features, for example, the revolve one below, you can set an existing parameter as its end angle by following the 4 steps in the figure below.

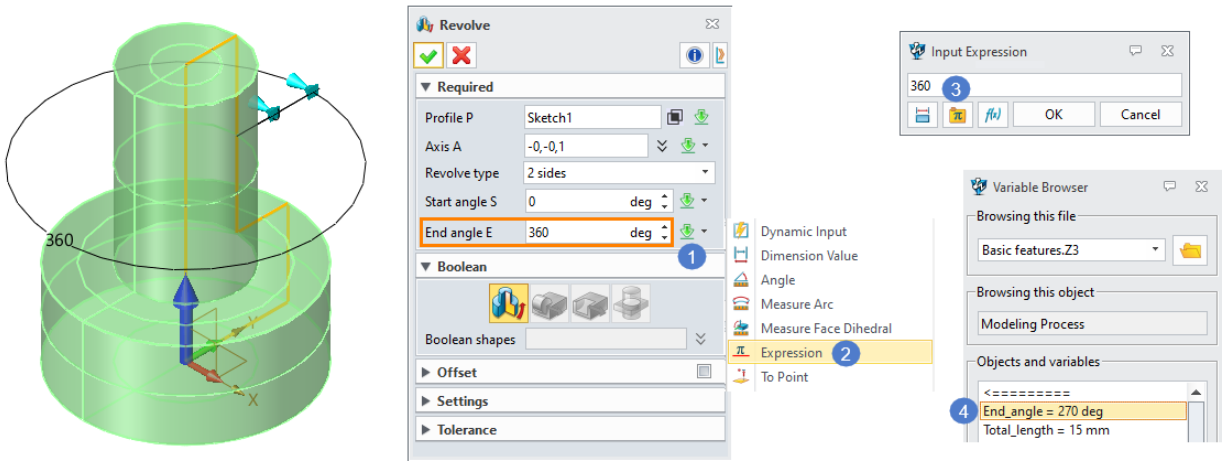



Figure 95 Setting the Parameter While Creating the Revolve Feature

➢ **Modify the parameters to update the model**

To apply a new value to some parameters and update the model, you can double-click on the target parameter in the ***History Manager*** and change the value. After that, the file name in the ***History Manager*** will be in red and marked as outdated. To update it, simply click the ***Regen*** command.

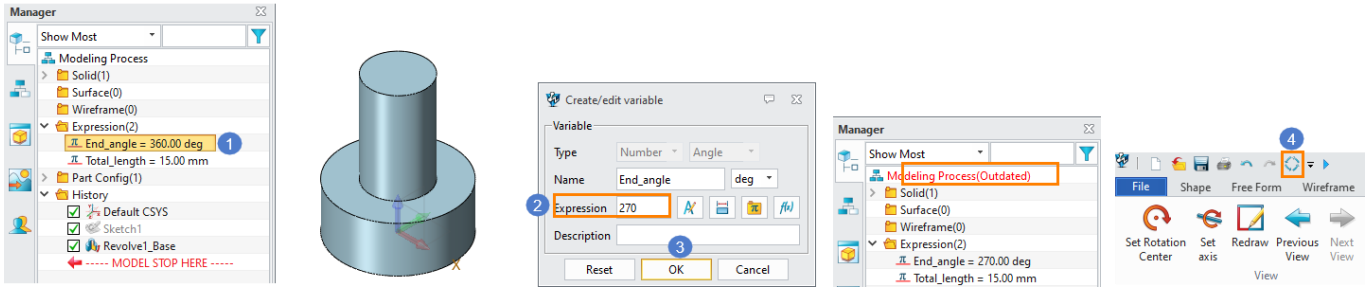



Figure 96 Modifying the Parameters and Updating the Model

Now, you shall find it easy to change and manage the key dimensions by means of parametric modeling.

## 4.3 Modeling Settings

It is suggested that you modify some basic settings (working folder, file type, linear units, mass units, and object tolerance) before modeling. You can refer to **Chapter 2.4** for the setting of working folders, and **Chapter 2.5** for the management of different types of files.

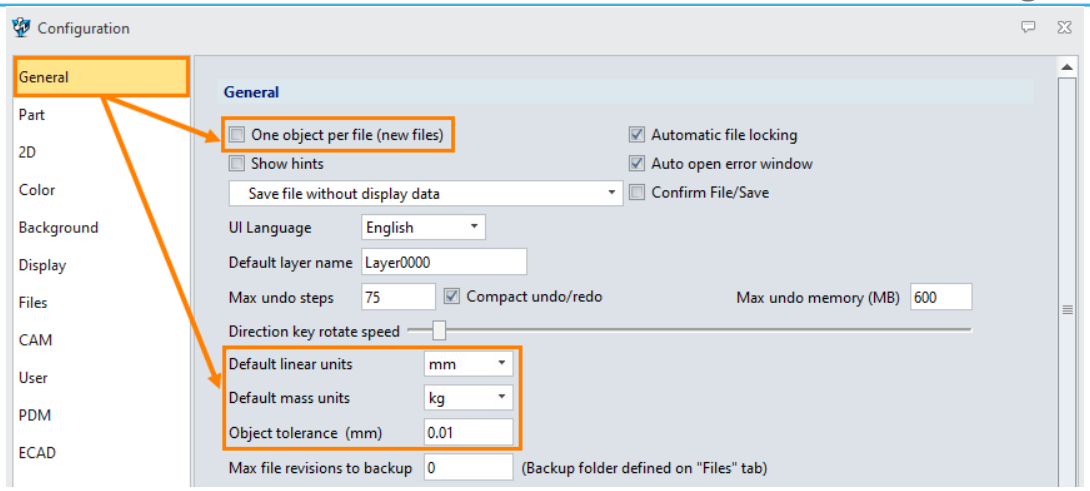



Figure 97 Basic Settings before Modeling

## 4.4 Modeling Guidelines

### 4.4.1 History Manager

The default History Manager helps you manage the modeling history. Besides History, folders reflecting the status of the model (Solid/Surface/Wireframe/Expression/Explicit Objects/Part Config) will also appear in it. You can change how their display (except the History folder) in the ***Configuration*** dialogue box.

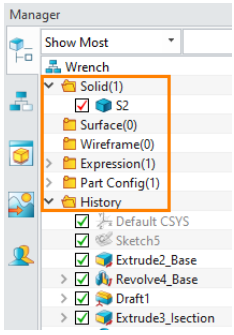



Figure 98 Folders in the History Manager

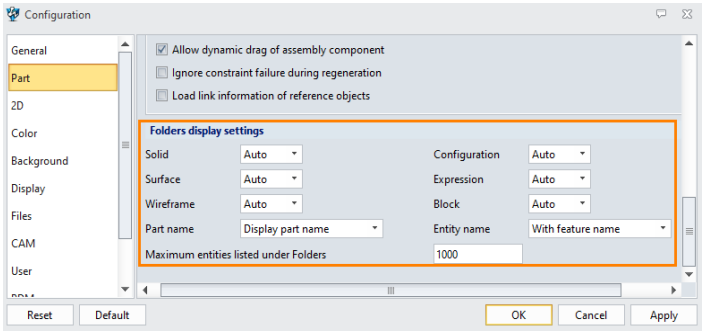



Figure 99 Folder Display Settings

To replay the modeling history, you can either drag the ***History Indicator*** or click the ***Replay*** buttons.

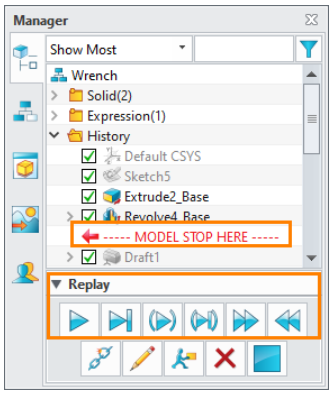



Figure 100 Replaying the Modeling History

### 4.4.2 Datum

In ZW3D, the default datum is a ***Default CSYS*** (Coordinate System), which is coincident to the WCS (World Coordinate System). As shown in Figure 101, the CSYS in brown is the ***Default CSYS***, which includes the XY, XZ, and YZ datum planes and the X, Y, and Z datum axes.

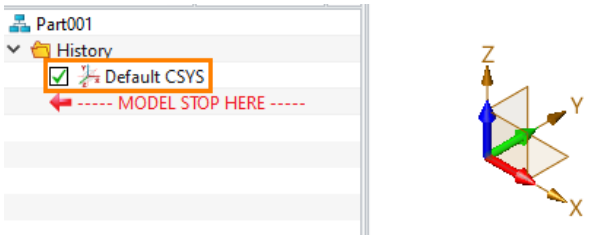



Figure 101 Default CSYS

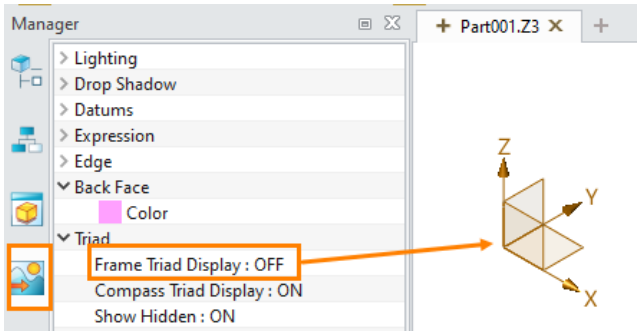



Figure 102 Hiding the Frame Triad

The red, green, and blue axes form the ***Frame Triad***. You can turn off/on the display of it by double-clicking on the ***Frame Triad Display*** option in the ***Visual Manager***, as shown in Figure 102.

➢ **Datum Creation**

Apart from the datum CSYS, you also can create datum planes whenever and wherever you need during the modeling. As shown in the figure below, there are multiple ways for you to create a datum plane.

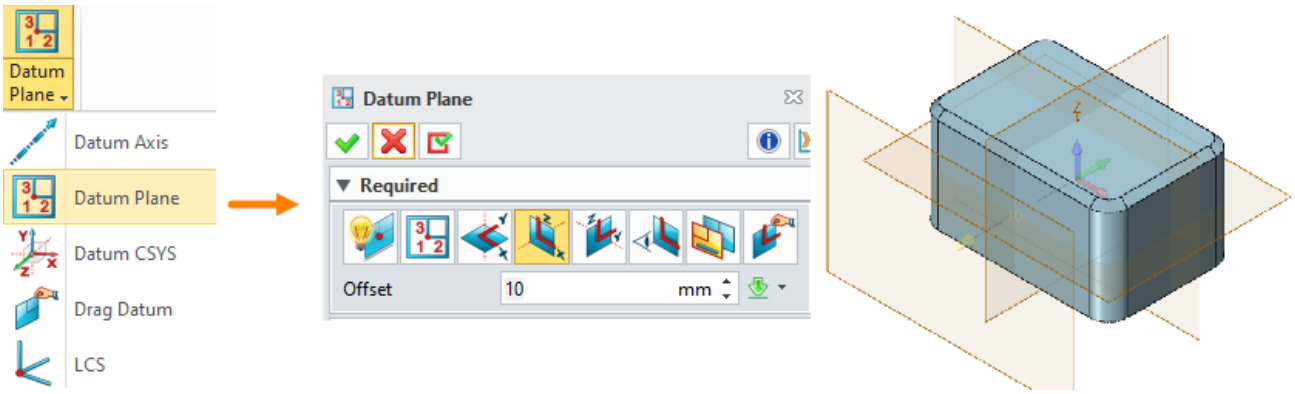



Figure 103 Creating Datum Planes

➢ **Auto Size of Datum**

If you prefer the datum plane to be in a bigger size, you can turn on ***Auto Size*** in the ***Visual Manager***.

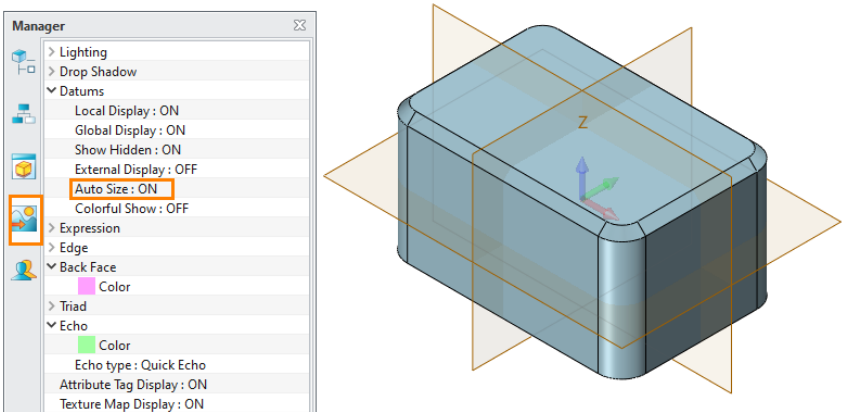



Figure 104 Making the Datum Planes Appear Bigger

➢ **Show or Hide**

You can show or hide datum planes.

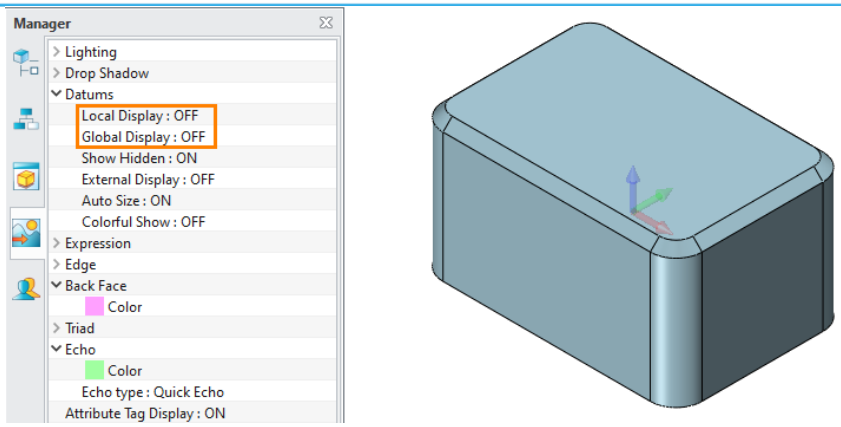



Figure 105 Turning off the Display of Datum Planes

### 4.4.3 Layer Manager

A 3D model is usually composed of different types of geometric entities, including points, lines, curves, sketches, datums, surfaces, solid shapes, etc. A complicated model may contain thousands of entities. Therefore, managing these entities properly is vital to a clear view of the model.

In ZW3D, you can easily access layer-related tools in the ***Document Aware Toolbar***.

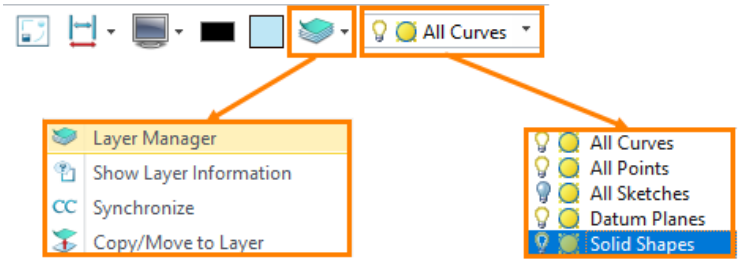



Figure 106 Layer-Related Tools in ZW3D

A default layer ***Layer0000*** would be created in the ***Layer Manager***. It can be renamed, but not deleted. If you did not create new layers, all the entities will be automatically assigned to this default layer. Also, when you turn off the bulb icon, all the entities of this layer will be blanked.

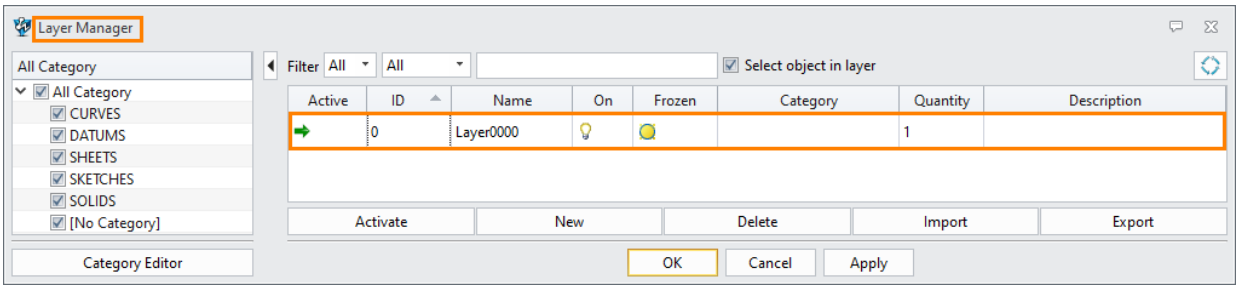



Figure 107 Layer Manager

The good habit of creating proper layers at the beginning of the modeling stage and assigning each entity to the corresponding layer will pay off when you are trying to view certain types of entities. As shown in the figure below, you can view all the sketches by checking the corresponding layer in the ***Layer Manager***.

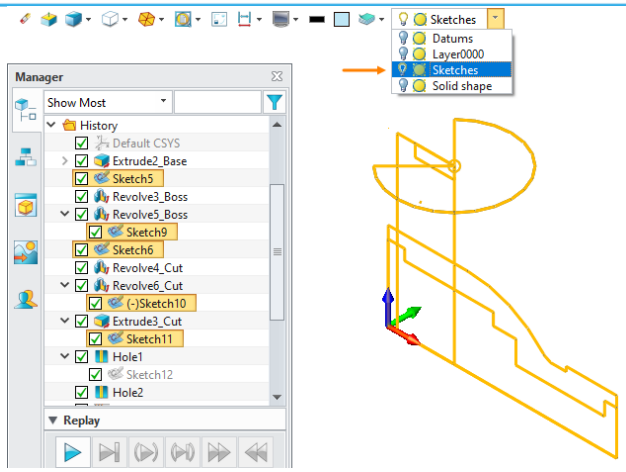



Figure 108 Display All Sketches

### 4.4.4 Feature Operations

As mentioned in **Chapter 4.1.1**, the modeling method of ZW3D is feature-based modeling, which allows you do a lot of operations based on the features.

- **Edit/Redefine Features**

As shown in Figure 109, after right-clicking on any feature, you can perform operations, such as Redefine, Suppress, and Delete on it.

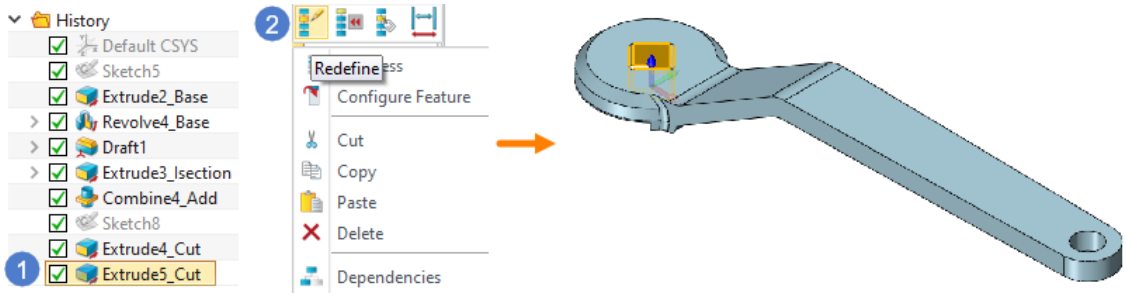



Figure 109 Feature Operations

- **Rearrange Features**

To adjust the sequence of features, you can drag them around in the ***History Manager***.

For example, in the model below, **Fillet 1** is done prior to **Shell 1**.

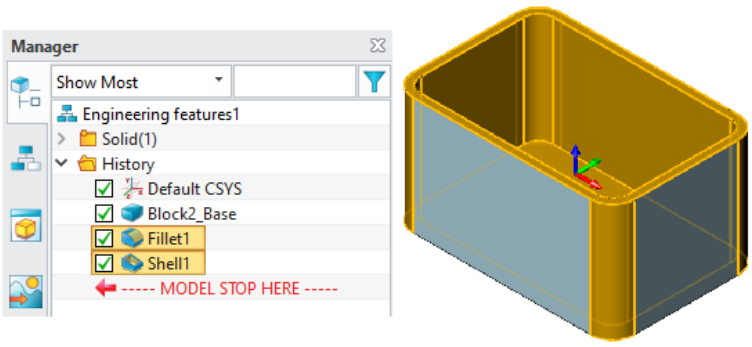



Figure 110 Make the Shell After Fillet

If you drag **Shell 1** above **Fillet 1**, the result will be different.

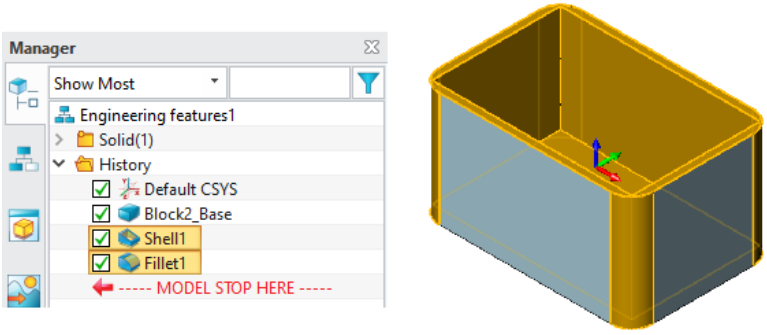



Figure 111 Fillet After Making the Shell

➢ **Insert Features**

If you need to add features before the last operation, you can drag the ***History Indicator*** to the right position and add them.

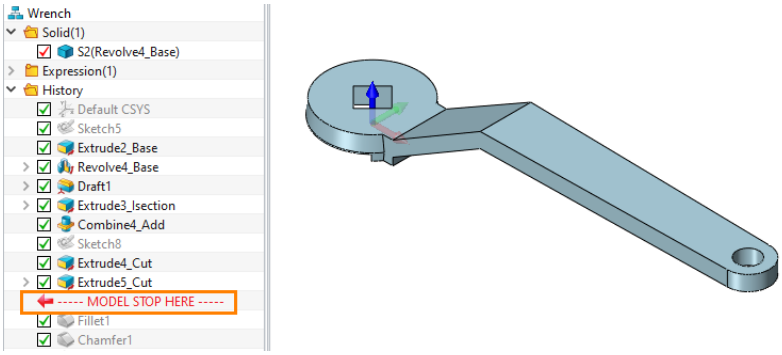



Figure 112 Insert Features

### 4.4.5 Display Modes and View Types

You can switch both the display mode and view type in the ***Document Aware Toolbar***.

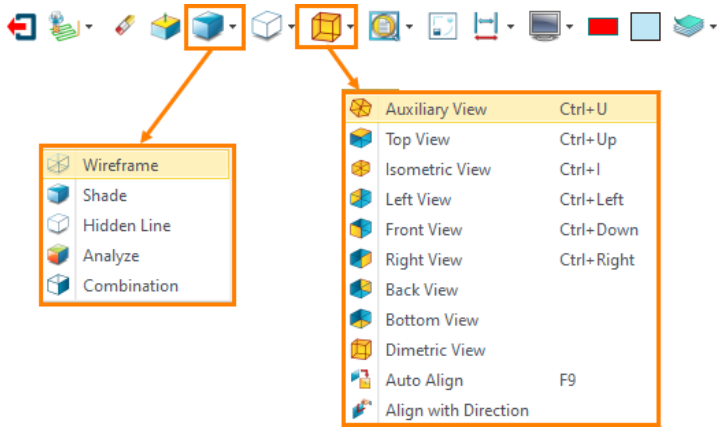



Figure 113 Display Modes and View Types

Also, you can switch between two frequently used display modes, ***Shade*** and ***Wireframe***, using the keyboard shortcut, ***CTRL+F***.

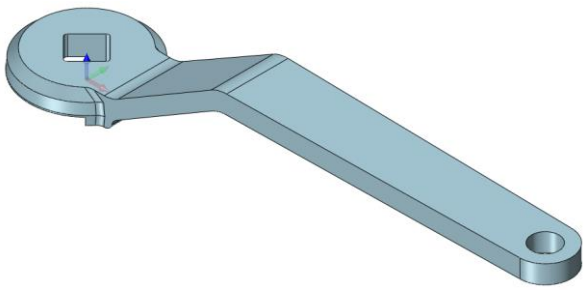


Figure 114 Shade

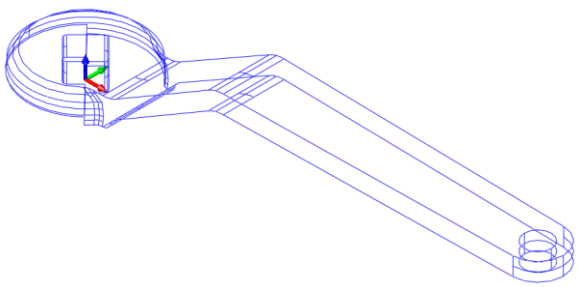


Figure 115 Wireframe

### ➢ Customize Views

To save a specific view, you can go to the ***View Manager,*** right-click on ***Custom Views***, and select ***New*** to create a new view. After that, you can switch to this custom view whenever you need.

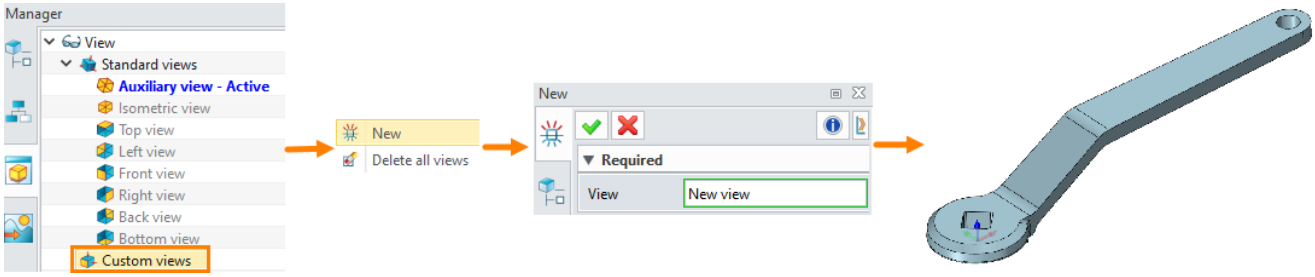



Figure 116 Creating a Custom View

A default isometric view is shown on the left side of the figure below. Double-click on the newly created view, and the model will be displayed from your customized view, as shown on the right side of the figure below.

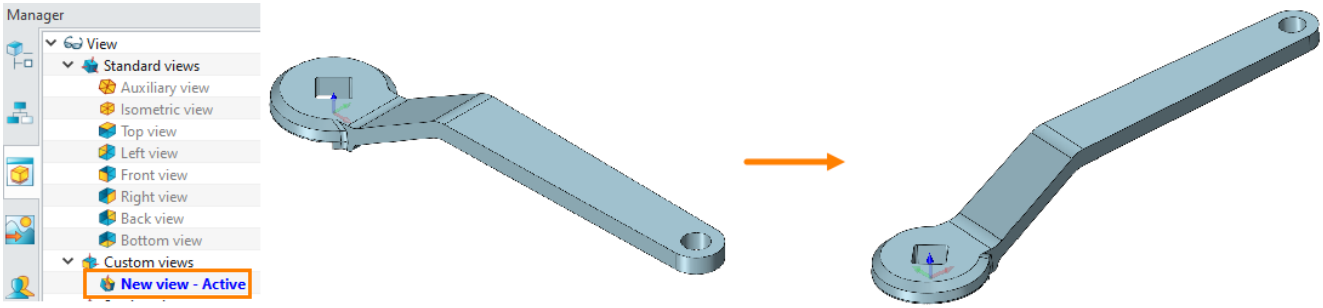



Figure 117 Change to the Custom View

## 4.4.6 Part Appearance

You can make the part look more vivid using the ***Face Attributes*** command in the ***Texture*** panel of the ***Visualize*** tab.

In the ***Face Attributes*** dialogue box, use ***Face*** as the filter. Then, pick the target faces and the defined attributes will be applied to them. If you want to apply the attributes to the whole shape, you can use ***Shape*** as the filter.

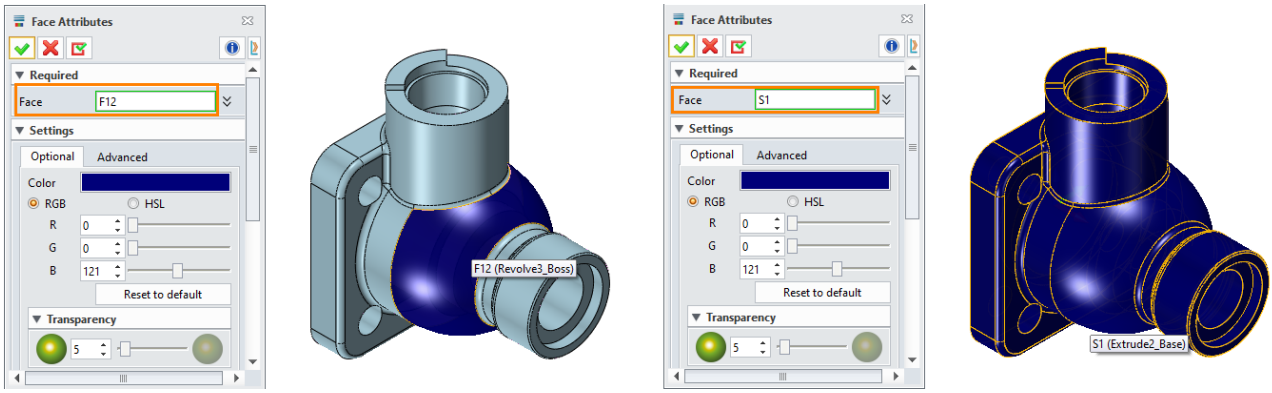



Figure 118 Setting Face Appearance

If you already know what the part will be made of, you can texturize it. For example, if the material used to manufacture the Valve Body is metal, you can apply ***Metal(cast)*** to it, as shown in Figure 120.

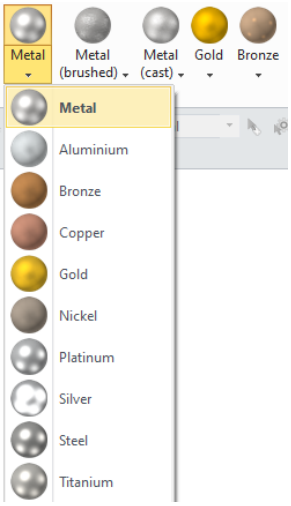



Figure 119 Texture List

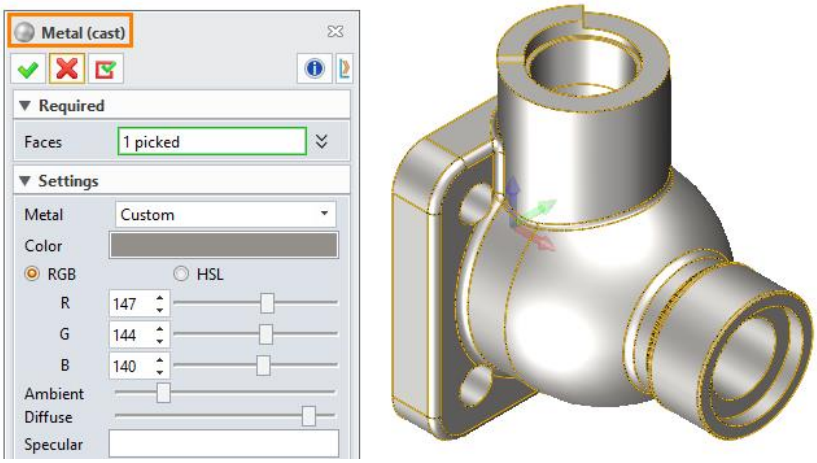



Figure 120 Texturizing the Part

***Notes:*** Since the texture is of higher priority than the face appearance, you need to remove the texture with the ***Erase Texture*** command if you only want to display the face attributes.

### 4.4.7 Material Attributes

To set the material attributes of the part, you can go to ***Tools*** tab -> ***Attributes*** panel -> ***Material Attributes***. First, select the target shape or part. Then, you can either define your own material or choose from the existing material list.

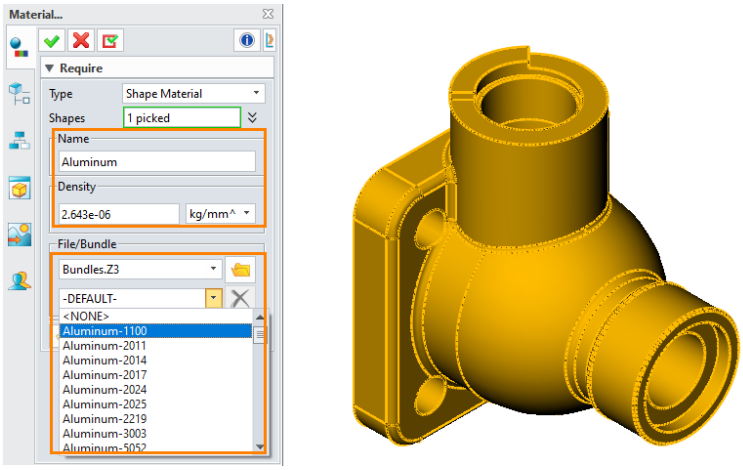



Figure 121 Setting the Material

### 4.4.8 Part Attributes

If you want to set or check part attributes, you can go to ***Tools*** tab -> ***Attributes*** panel -> ***Part Attributes***. In ZW3D, part attributes include ***Standard***, ***User*** and ***Physical***.

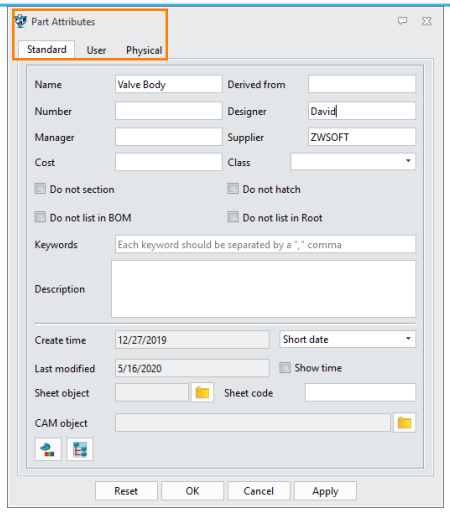



Figure 122 Part Attributes

For example, you can calculate the physical attributes of the current part, including mass, volume, etc. in two steps.

STEP 01 In the ***Physical*** tab of the ***Part Attributes*** dialogue box, click ***Material*** folder to invoke the ***Material Attributes*** dialogue box. You can either define a new material or select one from the existing ***File/Bundle*** box, then click ***OK***.

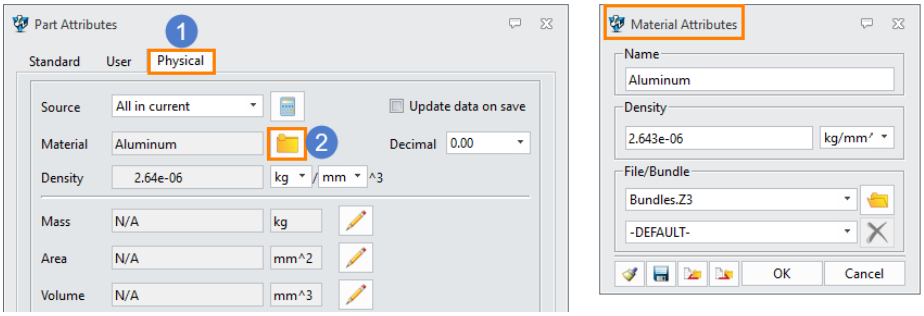



Figure 123 Checking Material Attributes

STEP 02 Click the ***Update*** button to calculate the part attributes. As shown in the figure below, physical attributes like ***Mass*** are updated.

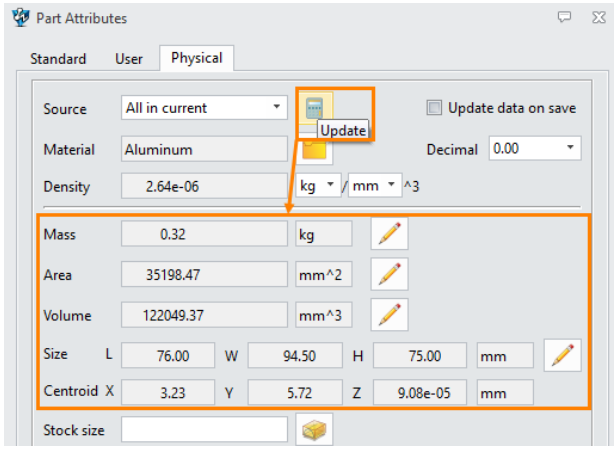



Figure 124 Updating Part Attributes

## 4.5 Modeling Cases

### 4.5.1 Case 1 – Valve Body

The general process of modeling the valve body below has been introduced in **Chapter 4.2.1**. Now, the process will be broken down into small steps for you to practice.

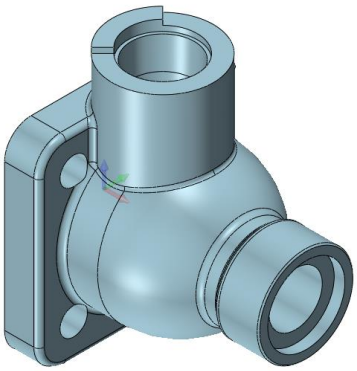


Figure 125 Valve Body

STEP 01 Create a new .Z3 file named **Valve.Z3** and click **OK** to enter the part modeling environment.

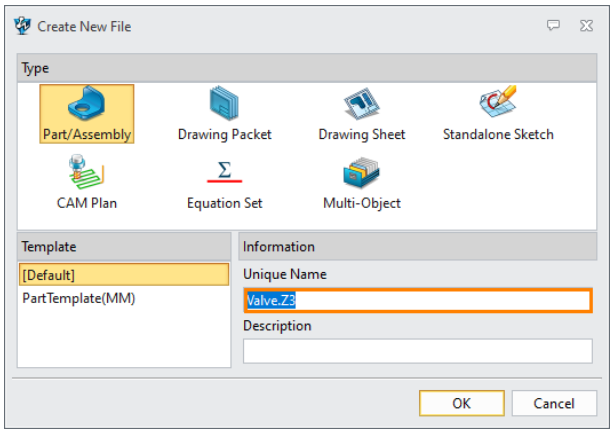



Figure 126 Creating a New .Z3 File

**Note:** By default, the first part will be named after the new .Z3 file. If you need to rename it, exit the modeling environment first and then, right-click on the part and select ***Rename*** to rename it in the ***Manager***. As shown in the figure below, the name of the part is changed from Valve to Valve Body.

To return to the modeling environment, simply double-click on the part.

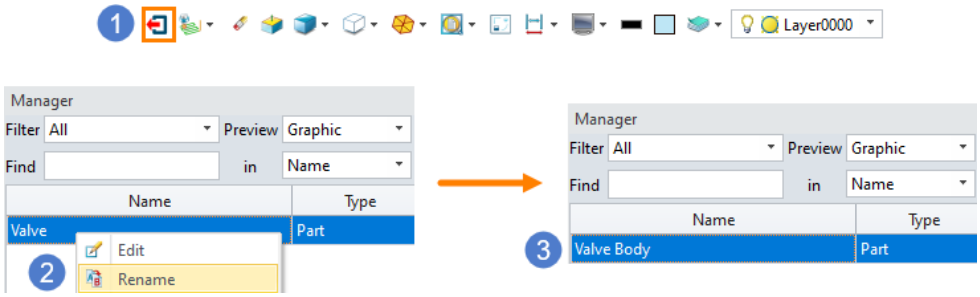



Figure 127 Renaming the First Part

STEP 02 Define the length variable ***Base Length*** in the ***Equation Manager***. In this case, it is set to be 75 mm.

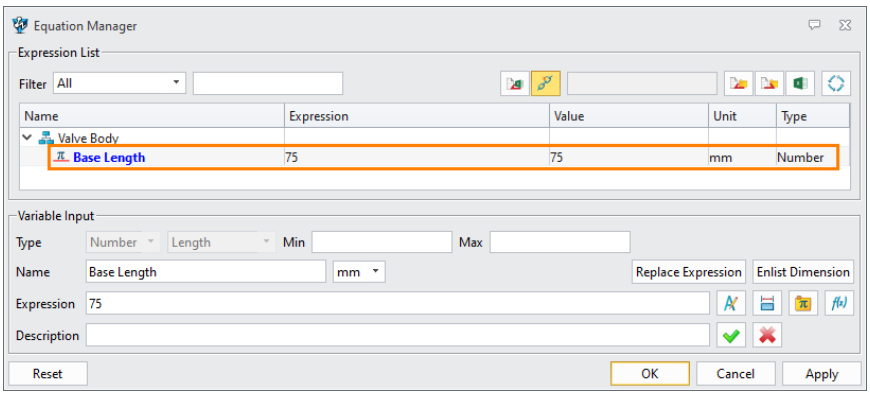



Figure 128 Defining the Base Length in the Equation Manager

STEP 03 Click the ***Extrude*** command, select ***Sketch*** in the ***Profile P*** box, and select ***Datum YZ*** as the sketch plane.

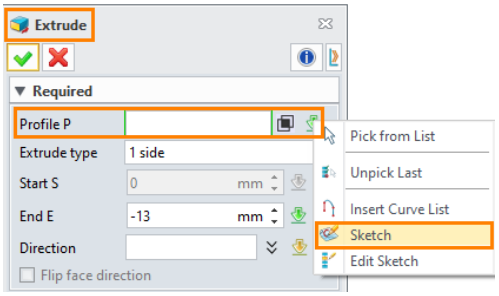



Figure 129 Creating a Sketch

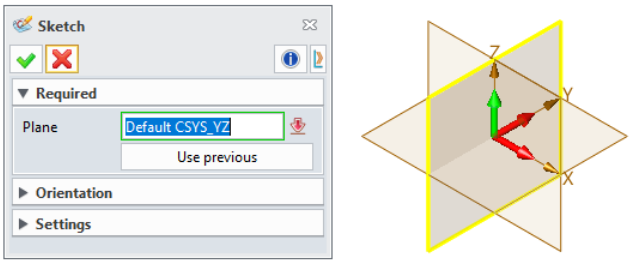



Figure 130 Datum YZ as the Sketch Plane

STEP 04 Draw a rough rectangular profile, as shown in Figure 131.

STEP 05 Delete a dimension and apply the ***Equal*** constraint to the selected lines, as shown in Figure 132.

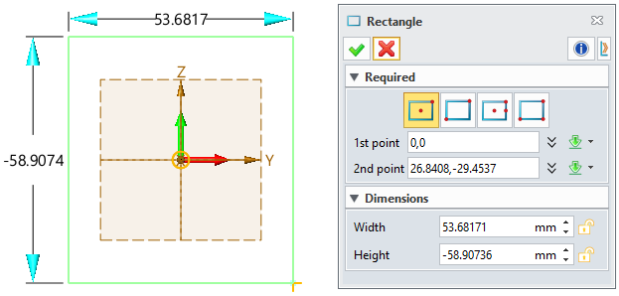



Figure 131 Drawing a Rectangular Profile

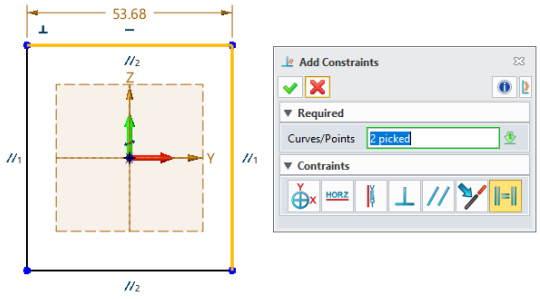



Figure 132 Adding the Equal Constraint

STEP 06 Double-click on the only dimension and assign the ***Base Length*** to it.

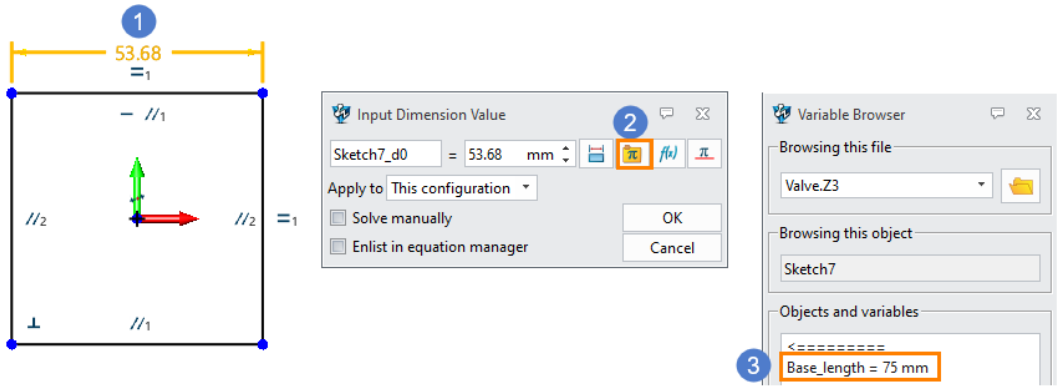



Figure 133 Inputting the Dimension with the Base Length Variable

STEP 07 Exit the sketch environment and invoke the Extrude dialogue box. Extrude the sketch from 0 mm to -13 mm by setting ***Extrude type***, ***Start S*** and ***End E*** according to Figure 134 and click ✔. You will find the ***History Manager*** updated.

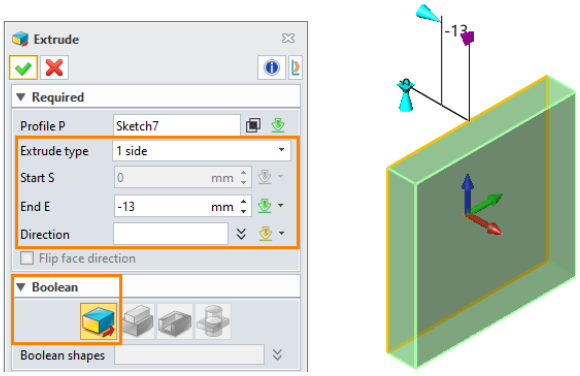



Figure 134 Extruding the Profile

STEP 08 Use the ***Sketch*** command to create an external sketch or ***Revolve*** to create an internal sketch. Refer to **Chapter 3.6.1** for details.

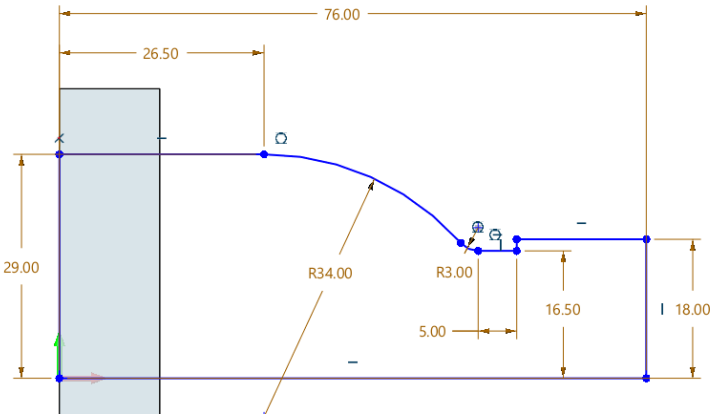



Figure 135 The Sketch of the Revolve Feature

STEP 09 Click the ***Revolve*** command and select the sketch you just created. Specify the ***End angle E*** as 360 degrees, choose the ***Add*** Boolean type, and click .

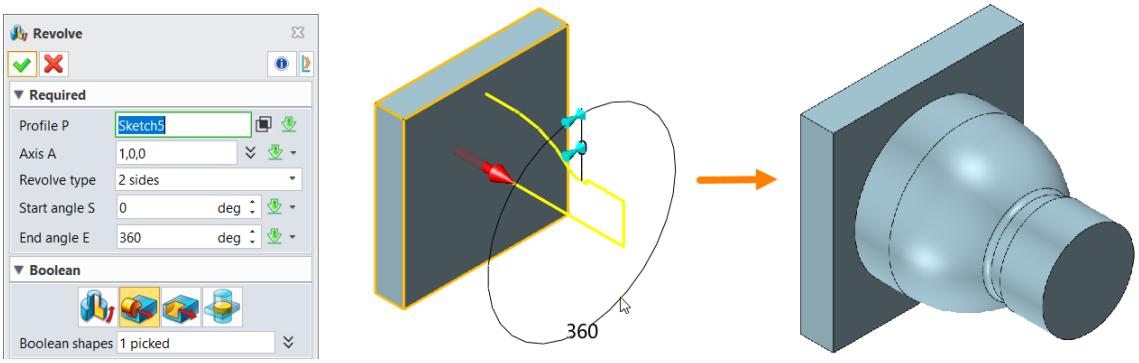



Figure 136 Revolving the Sketch

STEP 10 Sketch a rectangle whose baseline is on the X axis and base point is 3 mm from the origin on the XZ datum plane, as shown in the figure below.

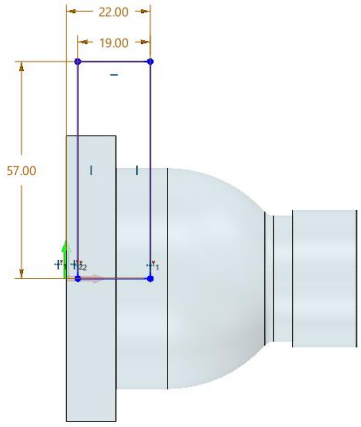



Figure 137 Sketching a rectangle on the XZ Datum Plane

STEP 11 Pick the sketch created in step 10 as the Profile P in the ***Revolve*** dialogue box, choose the ***Add*** Boolean type, and click to revolve the sketch around the default revolve axis.

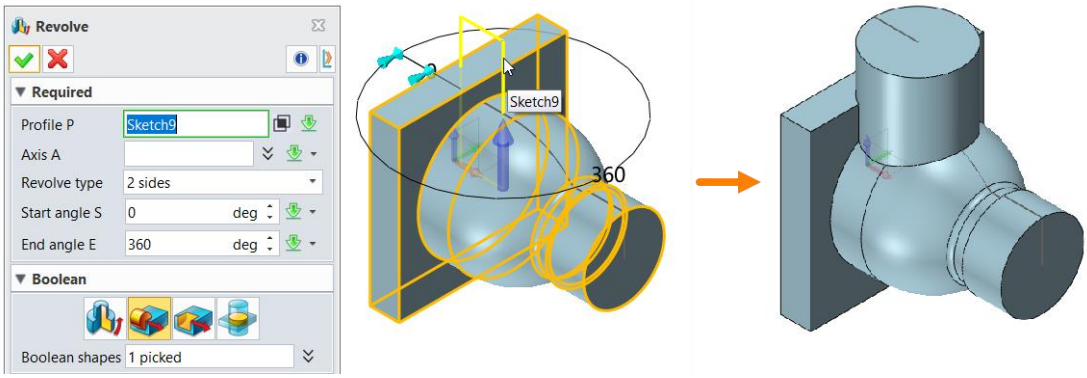



Figure 138 Making Another Revolve Feature

STEP 12 Sketch a revolve-cut profile whose baseline is on the X axis at the origin on the XZ datum plane, as shown in the figure below.

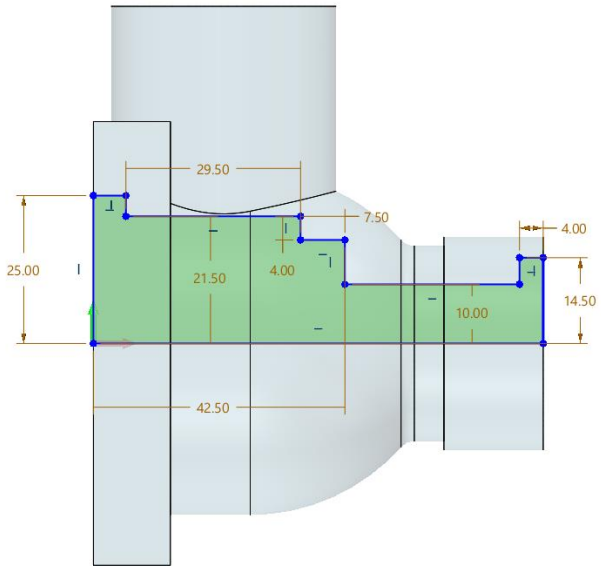
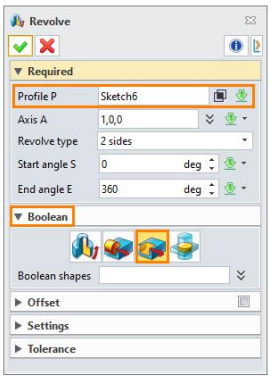



Figure 139 The Revolve-Cut Profile

STEP 13 Pick the sketch created in step 12 as the Profile P in the ***Revolve*** dialogue box, choose the ***Remove*** Boolean type, and click to create the revolve-cut feature.



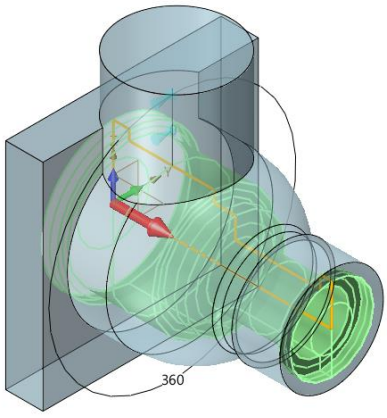




Figure 140 Selecting the Profile and Using the Remove Boolean Type

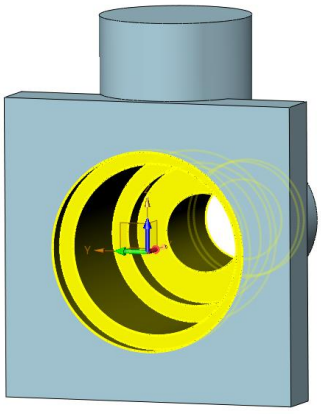


Figure 141 The Revolve-Cut Feature

STEP 14 Sketch a circle (radius=13.00 mm) on the top of this feature, click the ***Extrude*** command, pick the circle, select the ***Remove*** Boolean type, input the depth (4.0 mm), and click to create the first extrude-cut feature.

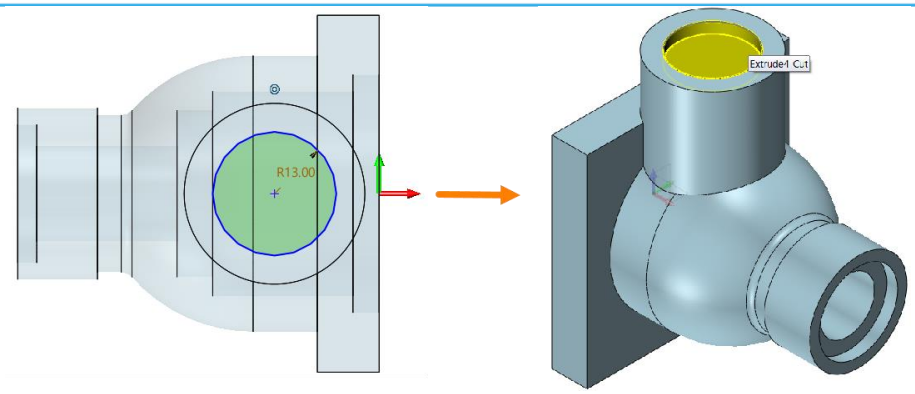



Figure 142 Creating the First Extrude-Cut Feature

STEP 15 Sketch a major sector on the top of this feature, click the ***Extrude*** command, pick the sector, select the ***Remove*** Boolean type, input the depth (2.0 mm), and click ✔ to create the second extrude-cut feature.

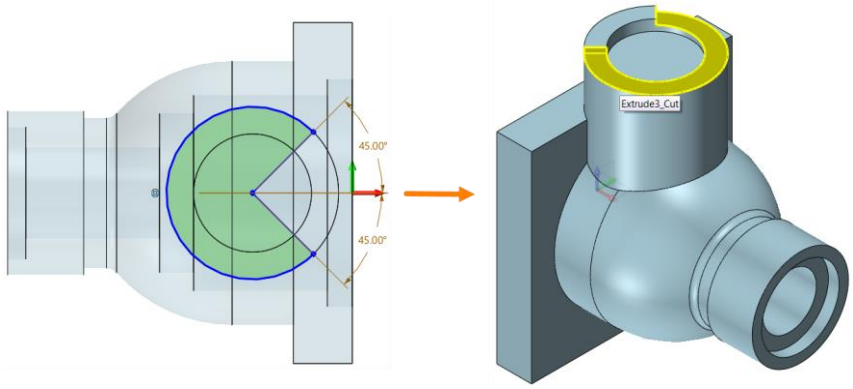



Figure 143 Creating the Second Extrude-Cut Feature

STEP 16 Click the ***Hole*** command, select the ***Thread hole*** type, choose ***Sketch*** in the ***Location*** drop-down list, create four points, and set the parameters according to the figure below, and click ✔ to create screw hole features.

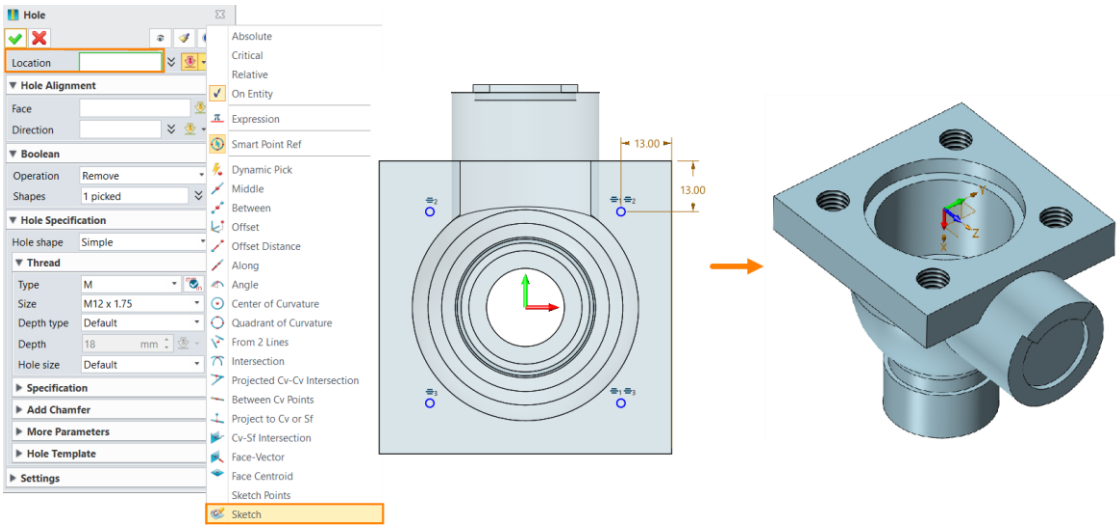



Figure 144 Creating Screw Hole Features

STEP 17 Click the ***Hole*** command, select the ***Thread hole*** type, pick the center point of the surface (created in step 14) in the ***Location*** drop-down list, set the parameters according to the figure below, and click ✔ to create another screw hole feature.

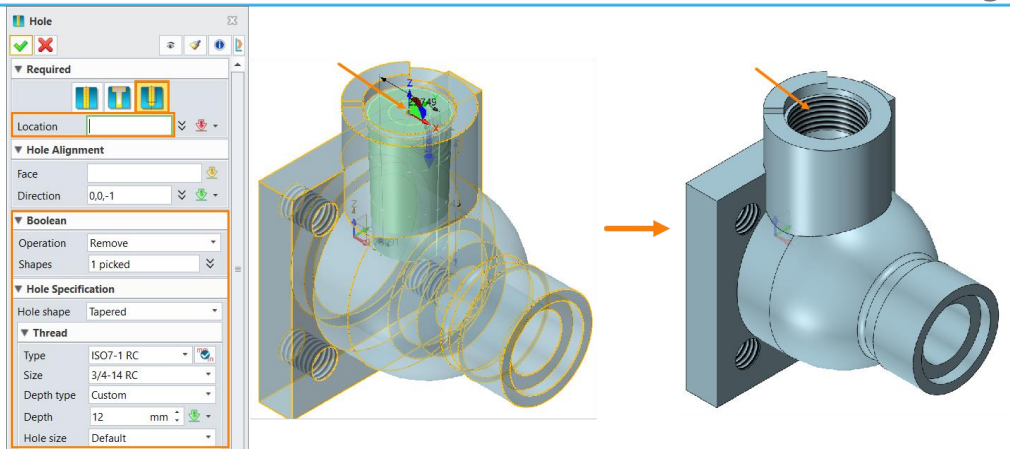



Figure 145 Creating Another Screw Hole Feature

STEP 18 Go to the ***Engineering Feature*** panel of the ***Shape*** tab, click the ***Flag Ext Thread*** command, pick the external revolve surface, set the thread parameters according to Figure 146, and click ✔ to create an external thread feature.

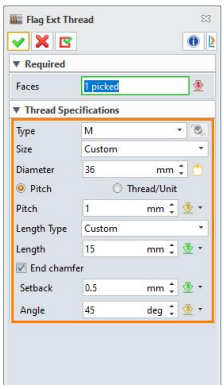
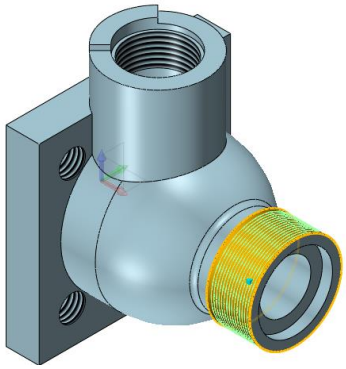



Figure 146 Setting Thread Parameters



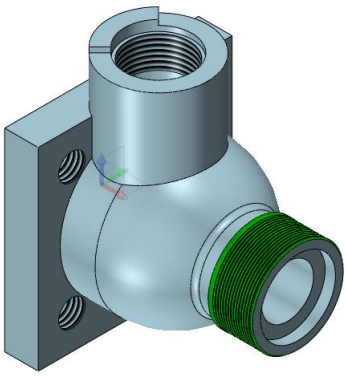


Figure 147 The Thread Feature

STEP 19 Add fillets (R=10.0 mm on the green faces, R=2.0 mm on the yellow faces, R=1.5 mm on the red faces), as shown in the figure below.

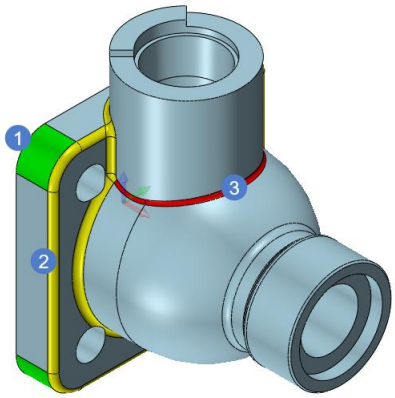



Figure 148 Adding Fillets

STEP 20 Check and optimize the modeling process in the ***History Manager***, and then click the ***Regen*** button to regenerate the whole model.

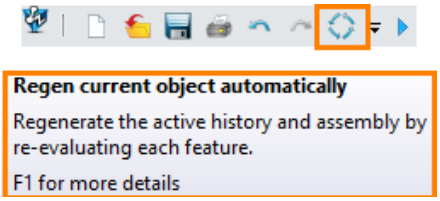



Figure 149 Regenerating the Whole Model

STEP 21 Go to the ***Attributes*** panel of the ***Tools*** tab and set part attributes according to the figures below.

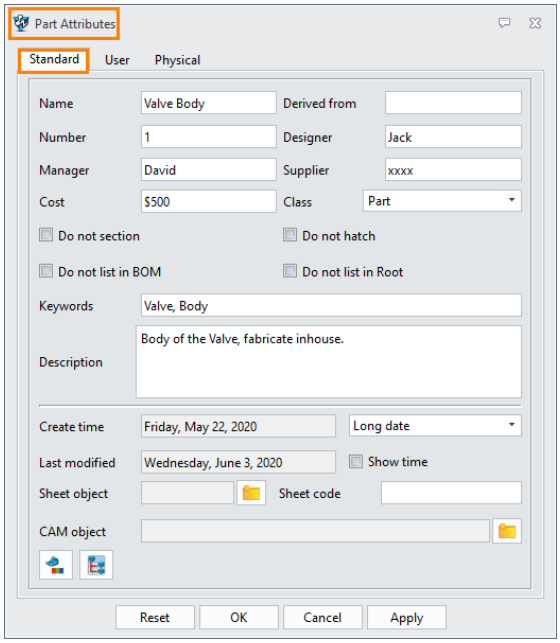



Figure 150 Setting Part Attributes

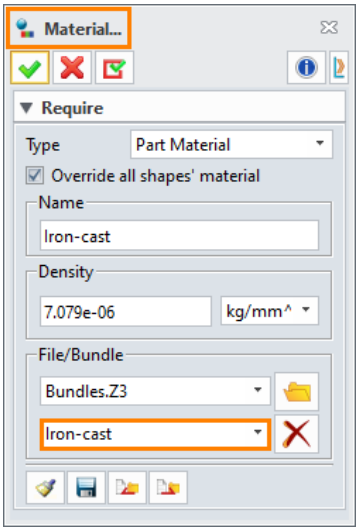



Figure 151 Selecting the Material

STEP 22 Go to the ***Texture*** panel of the ***Visualize*** tab, click the ***Face Attributes*** command, select ***Shape*** in the ***Filter List***, pick the face of the Valve Body, and set parameters according to the figure below so that the appearance of the model is modified.

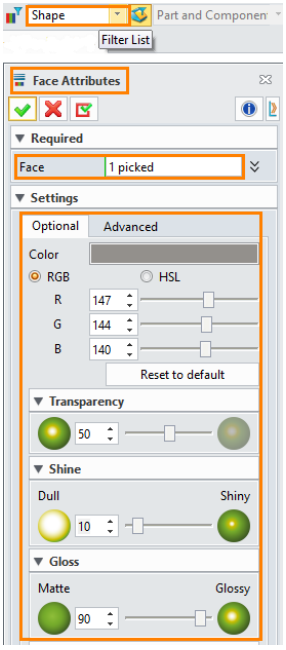



Figure 152 Setting Face Attributes

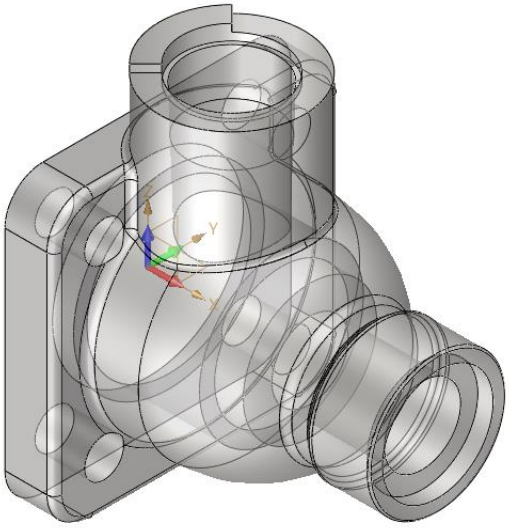


Figure 153 The Final Appearance

STEP 23 Save the file.

### 4.5.2 Case 2 – Wrench

You will learn how to add draft features and other modeling tricks from practicing this case.

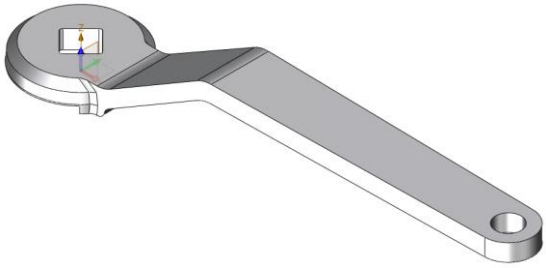


Figure 154 Wrench

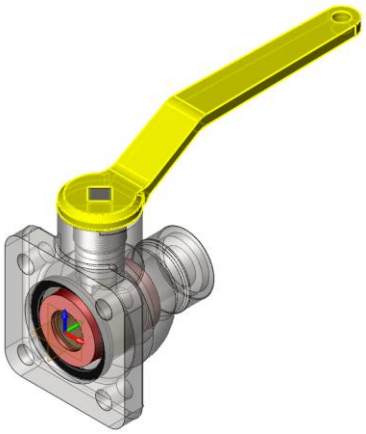


Figure 155 The Wrench in the Valve Assembly

STEP 01 Create a new part file named Wrench in the current Valve.Z3 file, which can be done by directly clicking the plus button on the left side of the file in the modeling environment.

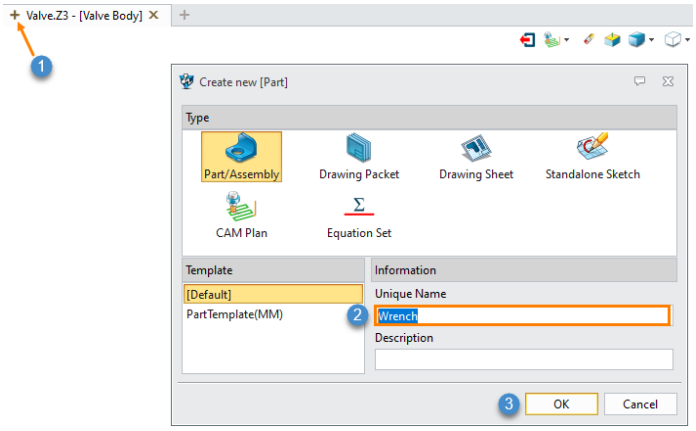
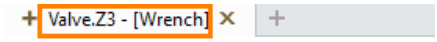



Figure 156 Creating a New Part File Named Wrench

After creating the new part file, the file name will be updated, which can be seen in the modeling environment and the **Z3 Manager** if you clicked the ***Exit*** button.



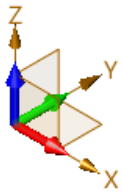




Figure 157 New Part in the Modeling Environment

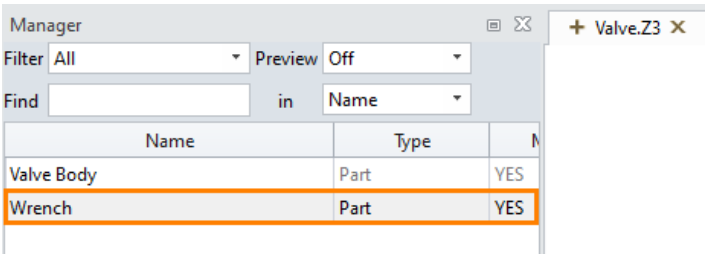



Figure 158 New Part in the Z3 Manager

STEP 02 Draw the main profile on the XZ plane and extrude it symmetrically, as shown in the figures below. For details, you can refer to **3.6.2**.

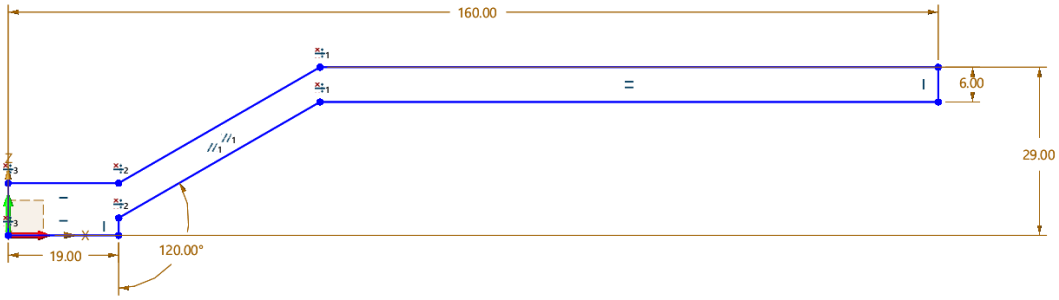



Figure 159 Drawing the Main Profile

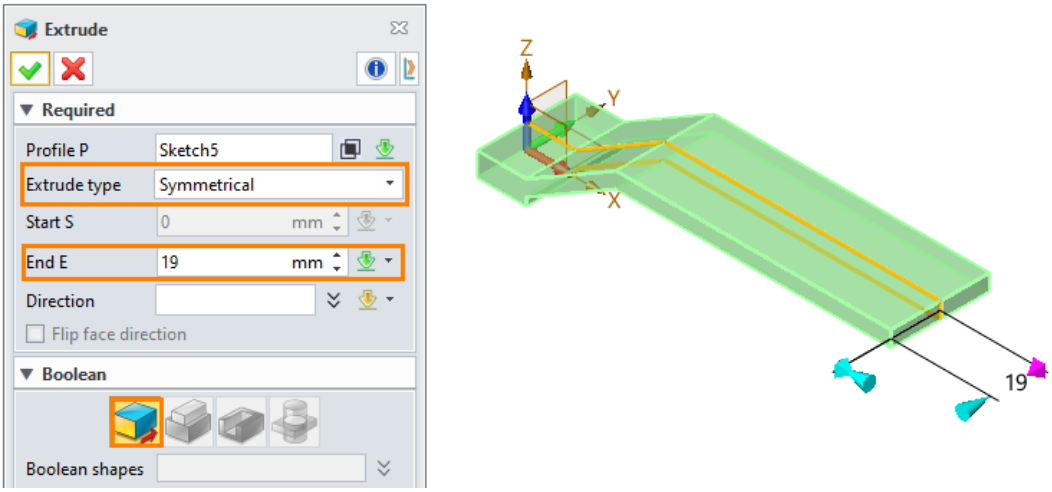



Figure 160 Extruding the Profile

STEP 03 Draw a sketch whose radius and height are equal to the existed edges on XZ datum plane. Click the ***Reference*** command in the ***Reference*** panel, select the target edges and all the dash lines, right-click and select ***Toggle type***, and finally, draw another line to close the whole sketch.

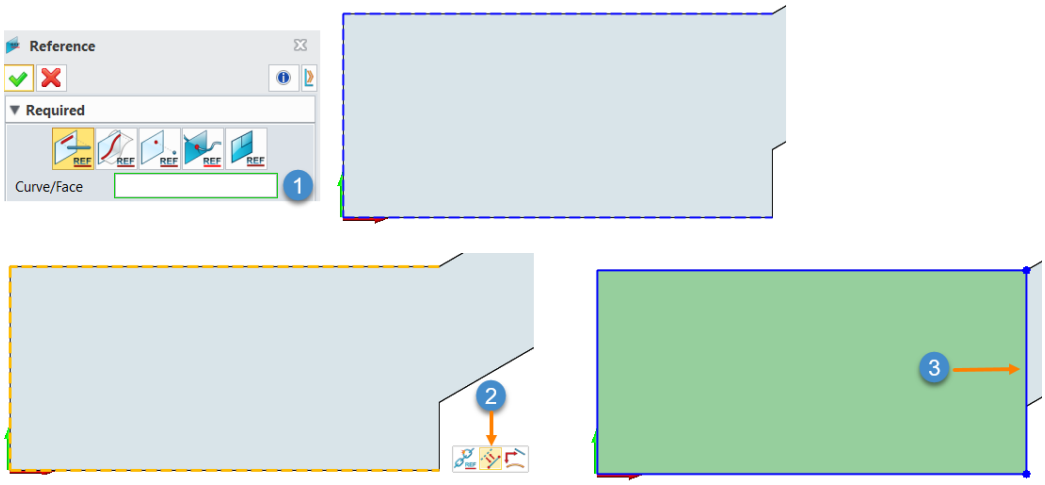



Figure 161 Sketching the Revolve Profile

STEP 04 Click the ***Revolve*** command, select the sketch created in step 3, choose the ***Base*** Boolean type, and click ✔ to create the revolve feature.

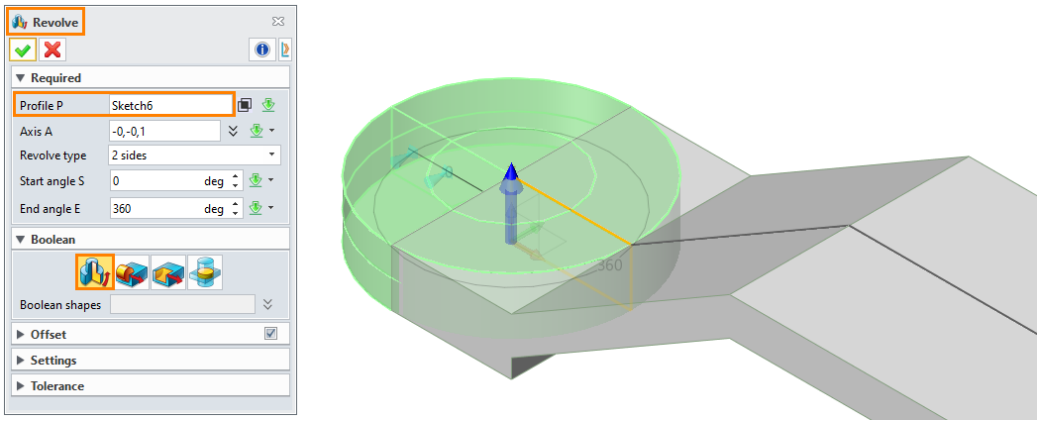



Figure 162 Creating the Revolve Feature

STEP 05 Click the ***Datum Plane*** command, pick the corner point, and click ✔ to create the draft datum on the revolve feature.

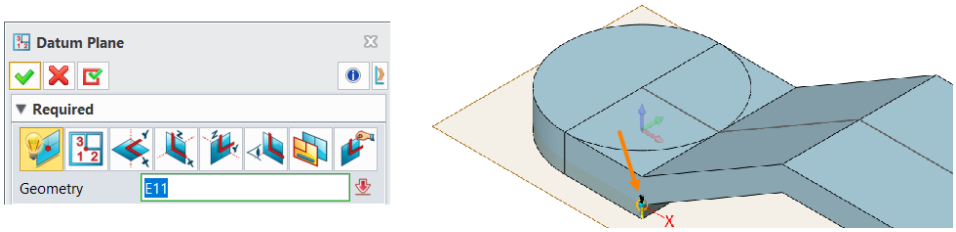



Figure 163 Creating the Draft Datum Plane

STEP 06 Go to the ***Engineering Feature*** panel, click the ***Draft*** command, pick the draft datum created in step 5 in the ***About D*** box, input the draft angle (5 degrees) in the ***About A*** box, select the draft face.

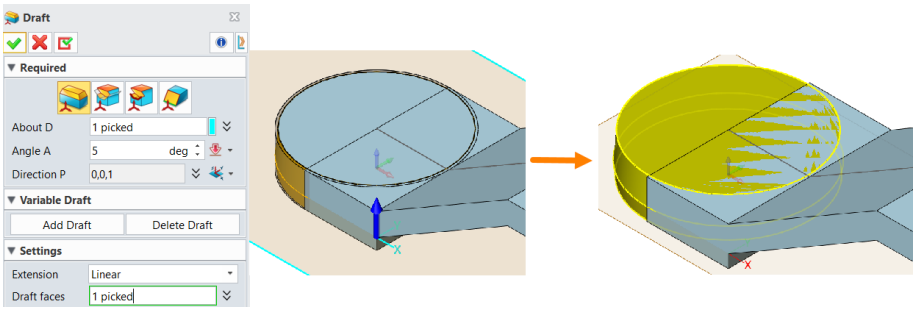



Figure 164 Making the Draft

STEP 07 Sketch a profile on the XY datum plane. As shown in the figure below, its radius is 8.50 and it is tangent to the three connected lines.

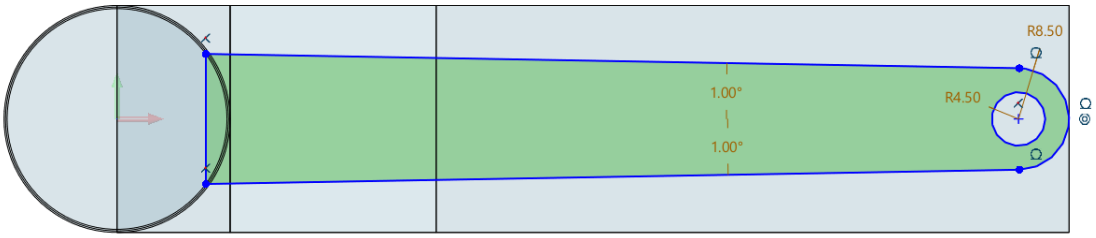



Figure 165 Sketching the Profile

STEP 08 Click the ***Extrude*** command, pick the profile created in step 7, select the ***Intersect*** Boolean type, pick the first extrude feature as the intersect object in the ***Boolean shapes*** box, and click ✔.

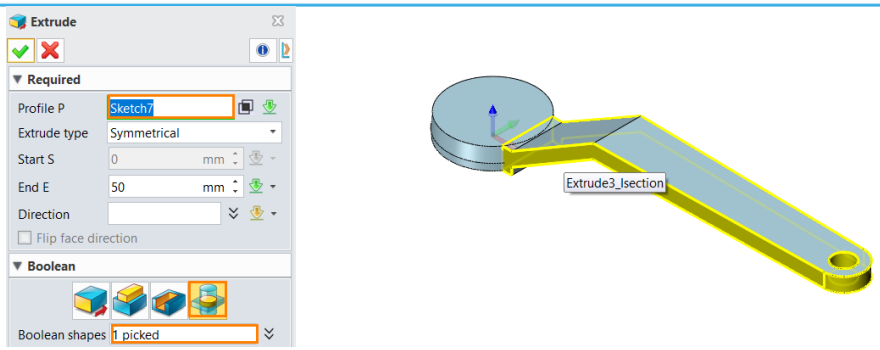



Figure 166 Intersecting the First Extrude Feature

STEP 09 Combine the shapes into one with the ***Add Shape*** command in the ***Edit Shape*** panel of the ***Shape*** tab.

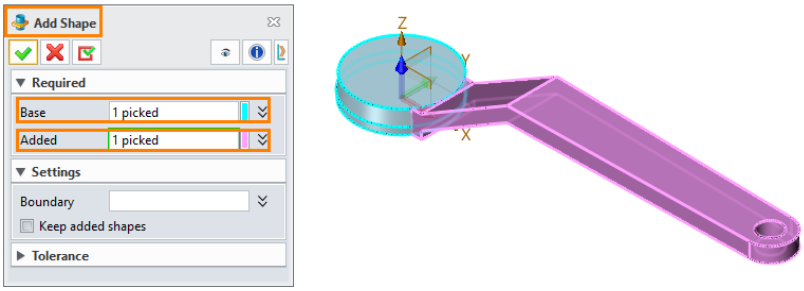



Figure 167 Combining Separated Shapes

STEP 10 Draw a sketch on the XY datum plane, click the ***Extrude*** command, cut the step (Depth=2.5 mm).

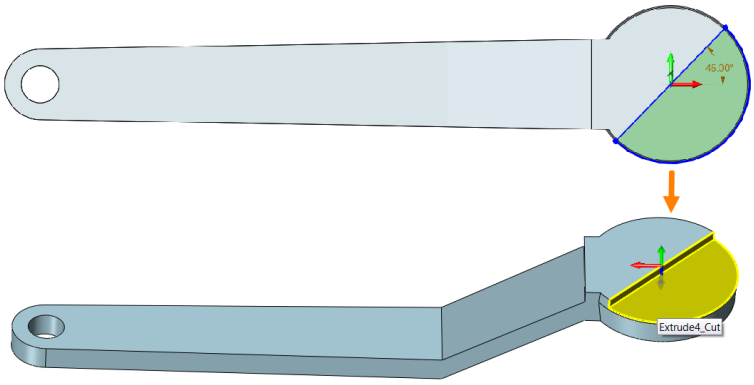



Figure 168 Cutting the Step Feature

STEP 11 Draw a square profile on the XY datum plane, click the ***Extrude*** command to cut the square hole.

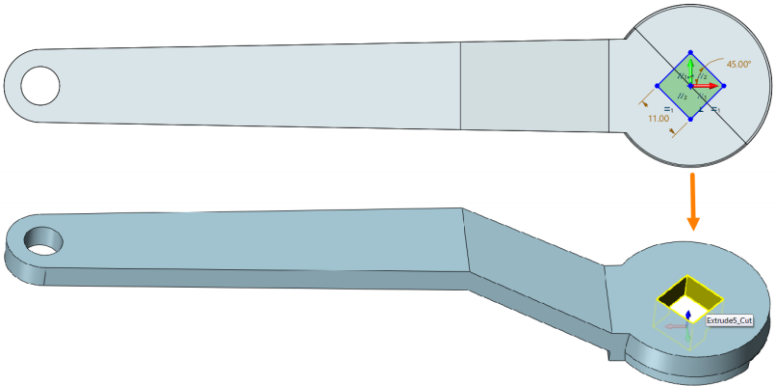



Figure 169 Cutting the Square Hole

STEP 12 Add chamfers and fillets (Chamfer=2.5 mm on the green faces, R=9.0 mm on the yellow faces, R=2.5 mm on the red faces), as shown in the figure below.

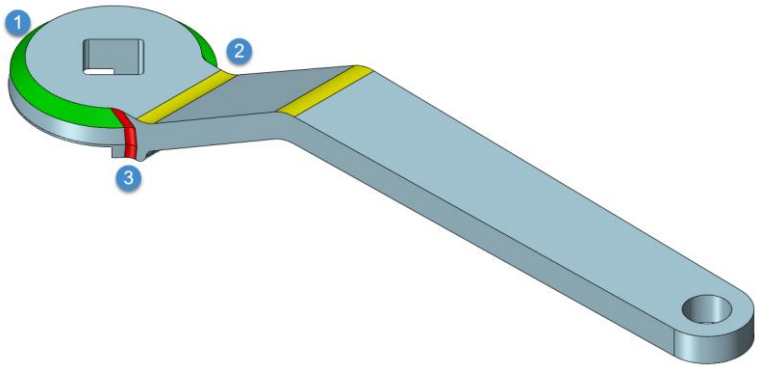
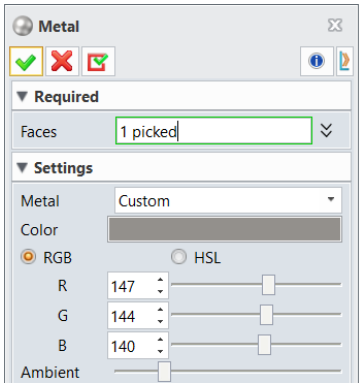



Figure 170 Adding Chamfers and Fillets

STEP 13 Add part attributes and modify the appearance by selecting an appearance in the ***Texture*** panel and applying it on the current model.



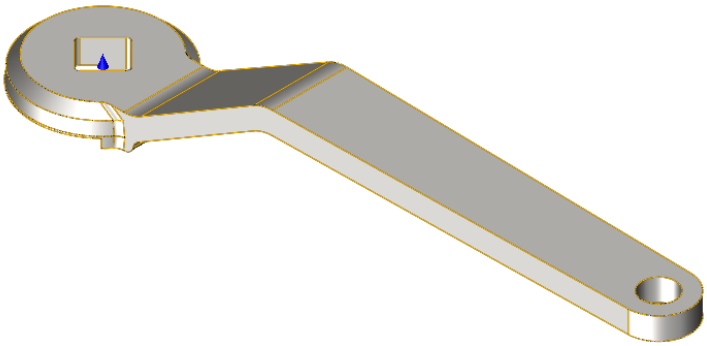



Figure 171 Modifying the Appearance of the Wrench

STEP 14 Check, regenerate, and save the whole model.

### 4.5.3 Case 3 – Valve Core

The **Valve Core** is located inside the **Valve Body** and driven by the **Valve Rod**.

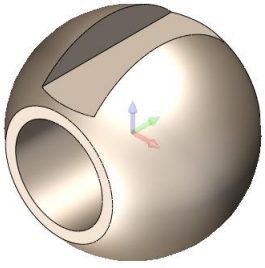


Figure 172 The Valve Core

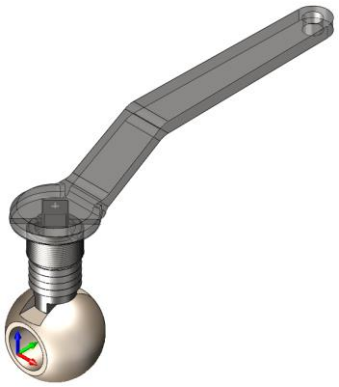


Figure 173 The Valve Core in the Valve Assembly

STEP 01 Create a new part file named Valve Core in the current Valve.Z3 file.

STEP 02 Draw a revolve profile on the YZ datum plane by creating and constraining a vertical line, mirroring it, connecting the two vertical lines, drawing a circle (Radius=21.00 mm), and trimming the sketch. Exit the Sketch environment and return to the modeling environment.

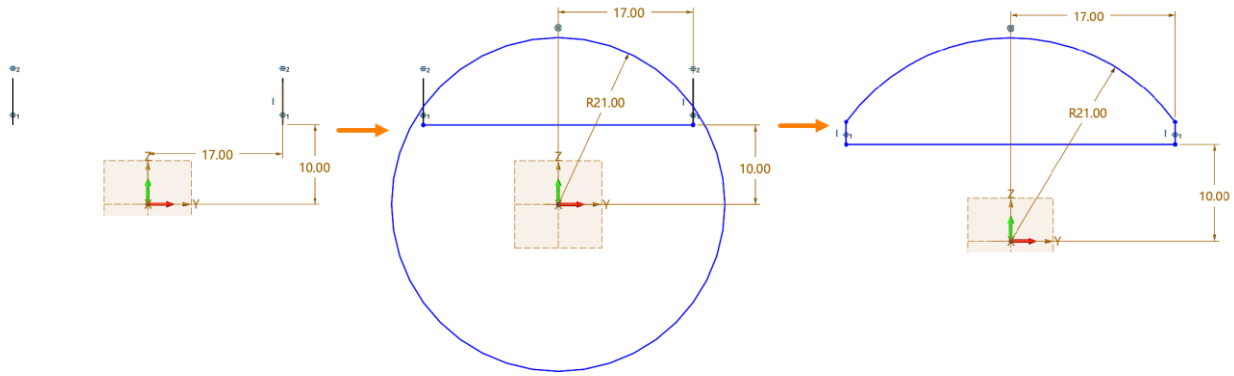



Figure 174 Drawing the Revolve Profile

STEP 03 Click the ***Revolve*** command, pick the profile created in step 2, revolve around the X axis, and click to create the revolve feature.

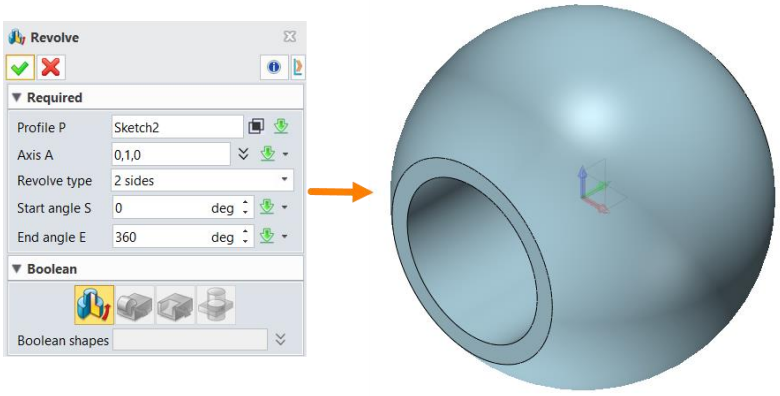



Figure 175 Creating the Revolve Feature

STEP 04 Draw a profile on the XZ plane, as shown in Figure 176. Click the ***Extrude*** command, pick the sketch, cut through the revolve feature, and click to create the slot feature.

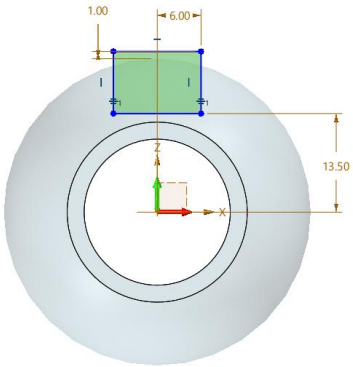



Figure 176 Drawing a Profile

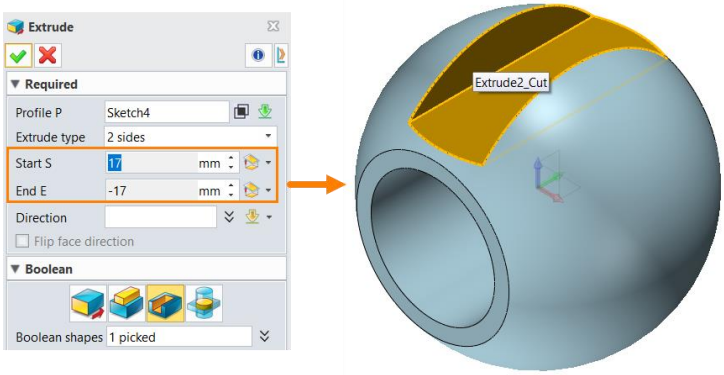



Figure 177 Making the Slot Feature

***Note:*** Make sure that the **Start S** and **End E** are linked to the sides surfaces of this model so that the slot cuts through.

STEP 05 Add part attributes, modify the appearance, and save the file.

### 4.5.4 Case 4 – Valve Rod

The **Valve Rod** is part of the **Rod Sub-assembly**.

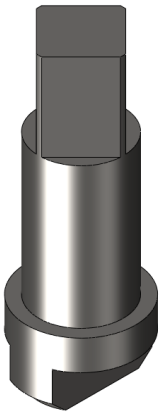


Figure 178 The Valve Rod

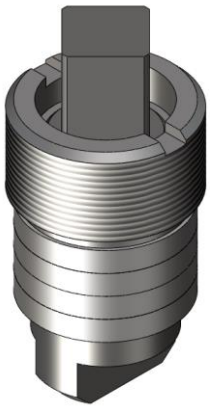


Figure 179 The Rod Sub-Assembly

STEP 01 Create new part file named Valve Rod in the current Valve.Z3 file.

STEP 02 Draw the revolve profile at the origin on the XZ datum plane, constrain it, exit the Sketch environment, and return to the modeling level.

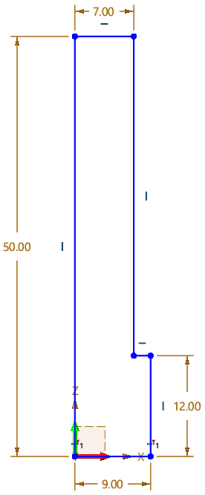



Figure 180 Drawing the Revolve Profile

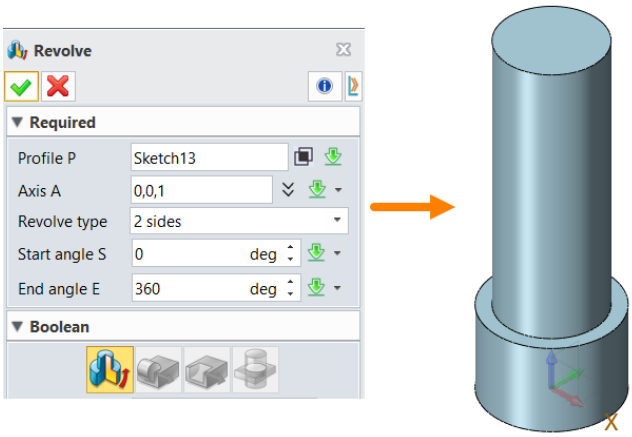



Figure 181 Creating the Revolve Feature

STEP 03 Click the ***Revolve*** command, pick the profile created in step 2, revolve around the Z axis, and click to create the revolve feature.

STEP 04 Sketch a rectangular profile on the XZ datum plane, mirror it along the Z axis, exit the Sketch environment, and return to the modeling environment.


Figure 182 Drawing the Sketch

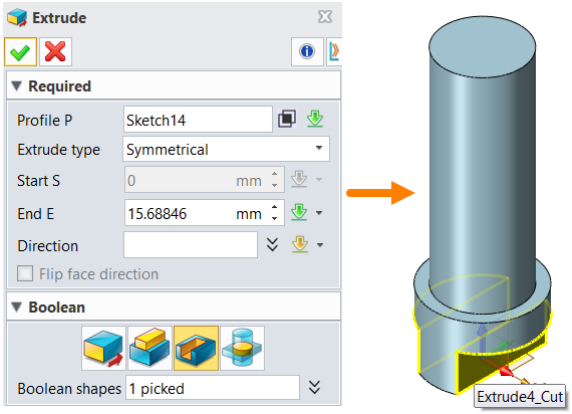



Figure 183 Making the Step Feature

STEP 05 Click the ***Extrude*** command, pick the sketch created in step 4, cut through the revolve feature, and click to create the step feature.

STEP 06 Sketch the square profile on the top of the part, select the outer circle as the reference, right-click, and toggle it into a solid line, exit the Sketch environment, and return to the modeling environment.

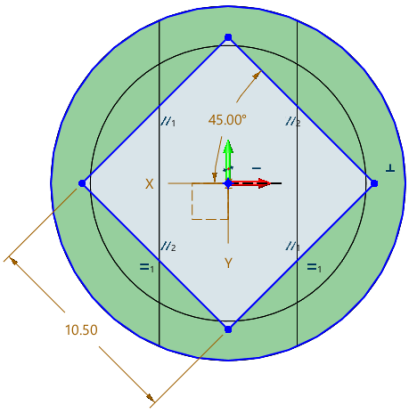



Figure 184 Drawing the Square Profile

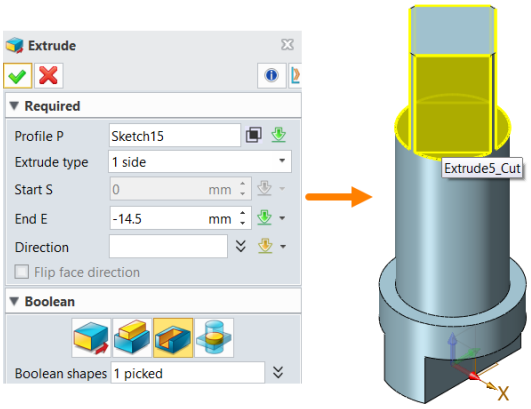



Figure 185 Creating the Square Feature

STEP 07 Click the ***Extrude*** command, pick the square profile, cut the shape (Depth=14.5 mm), and click ✔ to create the square feature.

STEP 08 Add chamfers (2.0 mm) to the model, modify its appearance, and save the file.


Figure 186 Adding Chamfers

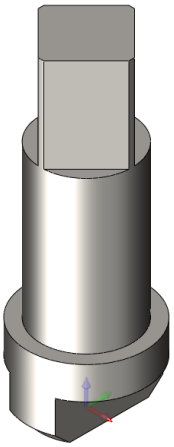


Figure 187 Modifying the Appearance

### 4.5.5 Case 5 – Shim Compressor

The **Shim Compressor** is part of the **Rod Sub-assembly**.

Figure 188 The Shim Compressor

Figure 189 The Rod Sub-Assembly

STEP 01 Create new part file named Shim Compressor in the current Valve.Z3 file.

STEP 02 Sketch a revolve profile whose baseline is on the X axis on the XZ datum plane, exit the Sketch environment, and return to the modeling environment.

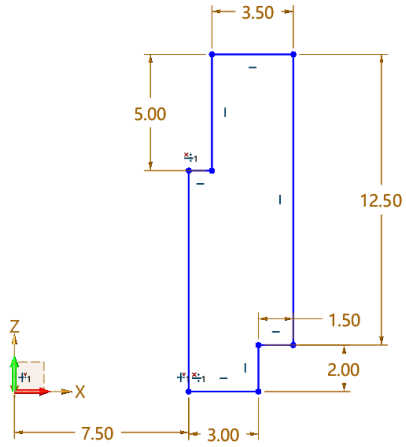



Figure 190 Sketching the Revolve Profile

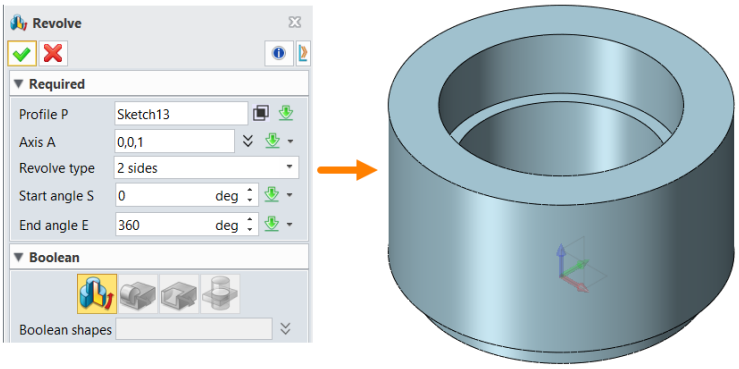



Figure 191 Creating the Revolve Feature

STEP 03 Click the ***Revolve*** command, pick the profile created in step 2, revolve around the Z axis, and click ✔.

STEP 04 Click the ***Flag Ext Thread*** command in the ***Engineering Feature*** panel, pick the outer surface of the cylinder, specify the ***Thread Specifications*** options, and click ✔ to add decorative threads to it.

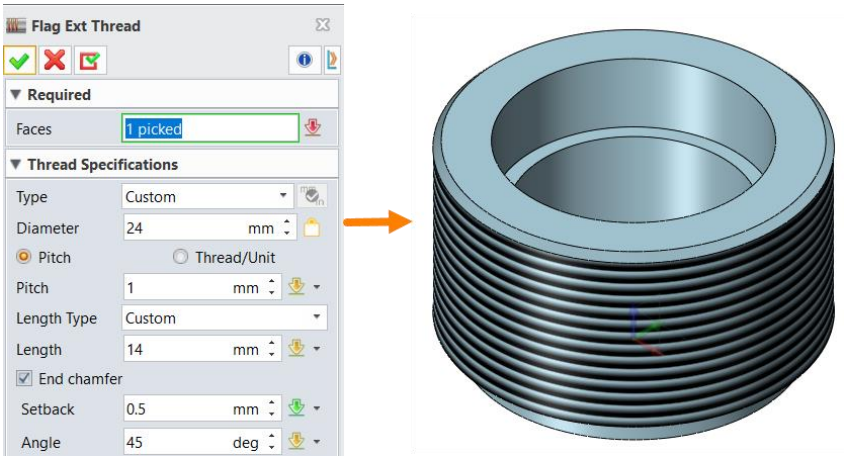



Figure 192 Adding Decorative Threads

STEP 05 Sketch a rectangle whose center is that of the circle on the top of the part.

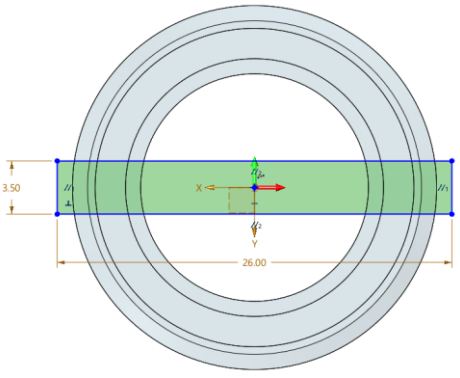



Figure 193 The Rectangular Profile

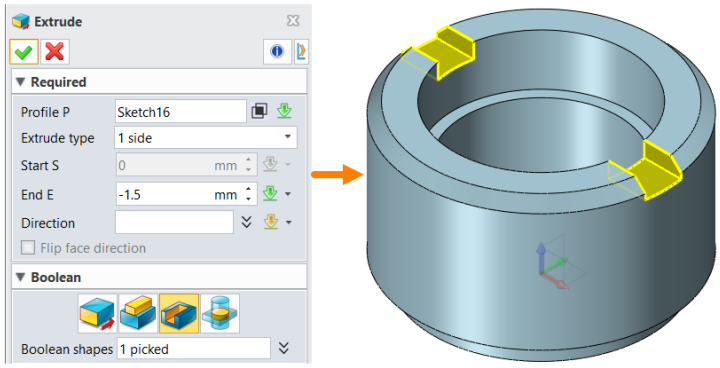



Figure 194 Creating the Slot

STEP 06 Click the ***Extrude*** command, pick the rectangular profile created in step 5, cut the shape (Depth=1.5 mm), and click ✔ to create the slot feature.

STEP 07 Modify its appearance and save the file.

Figure 195 The Shim Compressor

### 4.5.6 Other Parts

To compete the assembly, there should be more parts, which can be created with either the ***Revolve*** or ***Extrude*** command.

- **Bottom Shim**

The **Bottom Shim** is part of the **Rod Sub-assembly**.

To create it, you need to draw two concentric circles (Radii are 7.00 mm and 11.00 mm, respectively) on the XY datum plane, exit the Sketch environment, click the ***Extrude*** command, specify the **End E** option to be 3.38 mm, and click ✔. Modify its appearance and save the file.

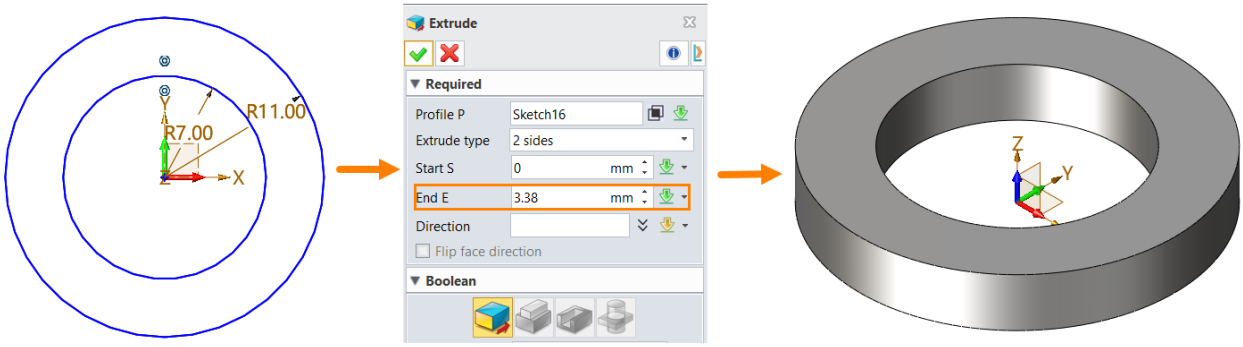



Figure 196 Creating the Bottom Shim

- **Top Shim**

The steps of making the **Top Shim** are similar to those of making the **Bottom Shim**, only the thickness (3.37 mm) is different. The parameters are as the figure below shows.

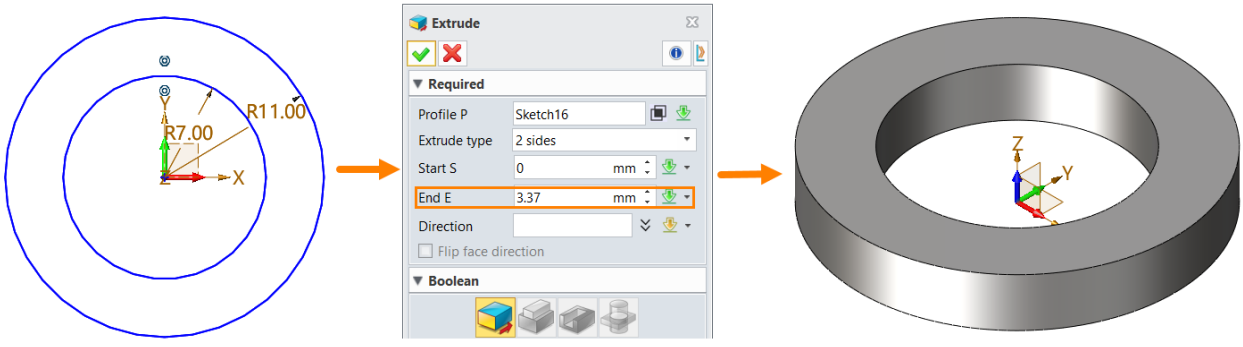



Figure 197 Creating the Top Shim

- **Adjusting Shim**

The **Adjusting Shim** will be used in the final **Valve** assembly.

The steps of making it are similar to those of making the **Bottom Shim**, only that it is drawn in the YZ plane and its thickness is 2.0 mm.

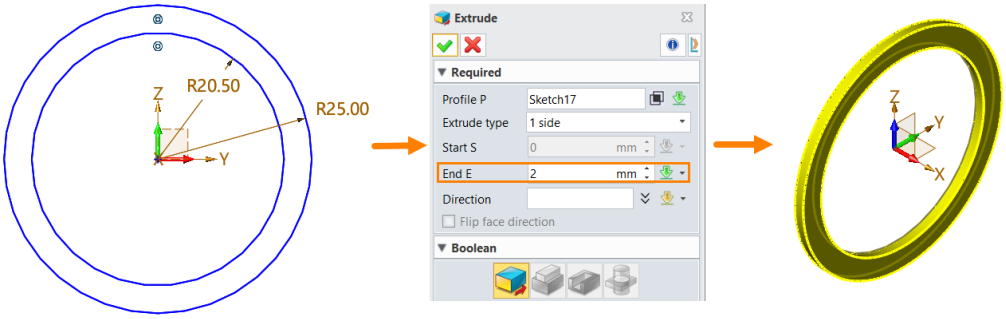



Figure 198 Creating the Adjusting Shim

- **Seal Ring**

Likewise, the **Seal Ring** will be used in the final **Valve** assembly.

Sketch its profile on the XY datum plane, exit the Sketch environment, click the ***Revolve*** command, pick the profile, and click ✔. Modify its appearance and save the file.

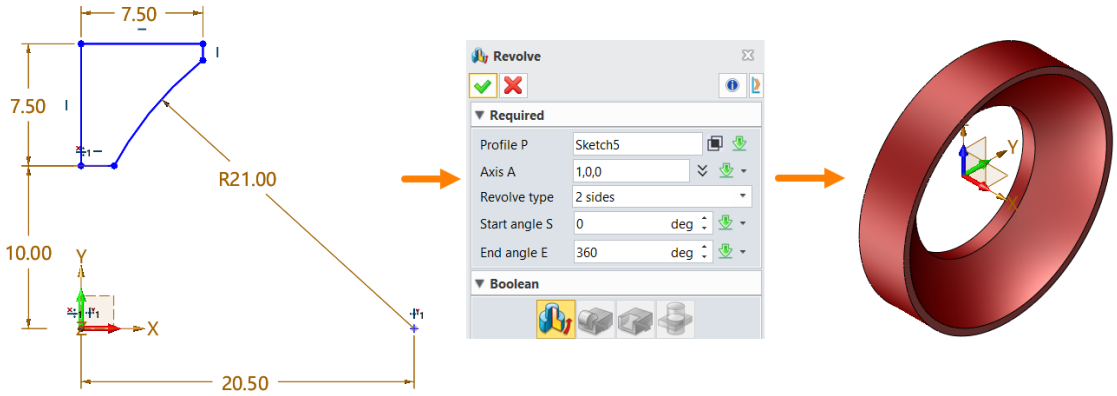



Figure 199 Creating the Seal Ring

# 5 Assembly

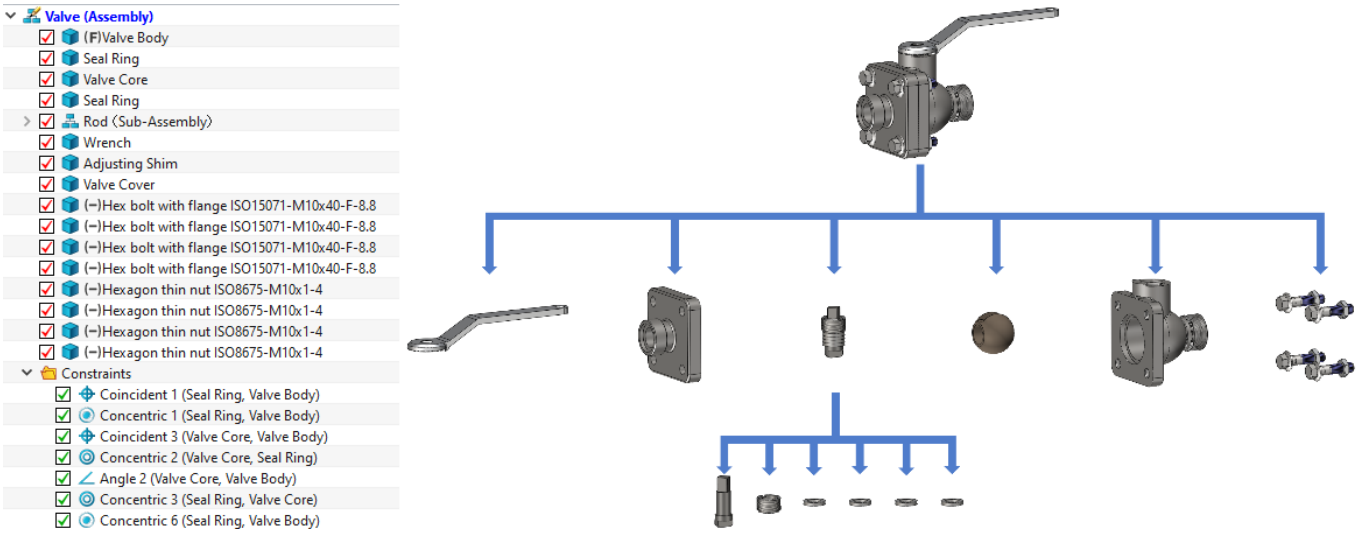



Figure 200 An example of assembly structure

Assembly Modeling is a computer-aided design technology and method which can helps engineer to handle with multifile into an assembly. Then assembly structure, movement, and design relationship can be analyzed by the virtual models, the picture shown below is an example of assembly in ZW3D.

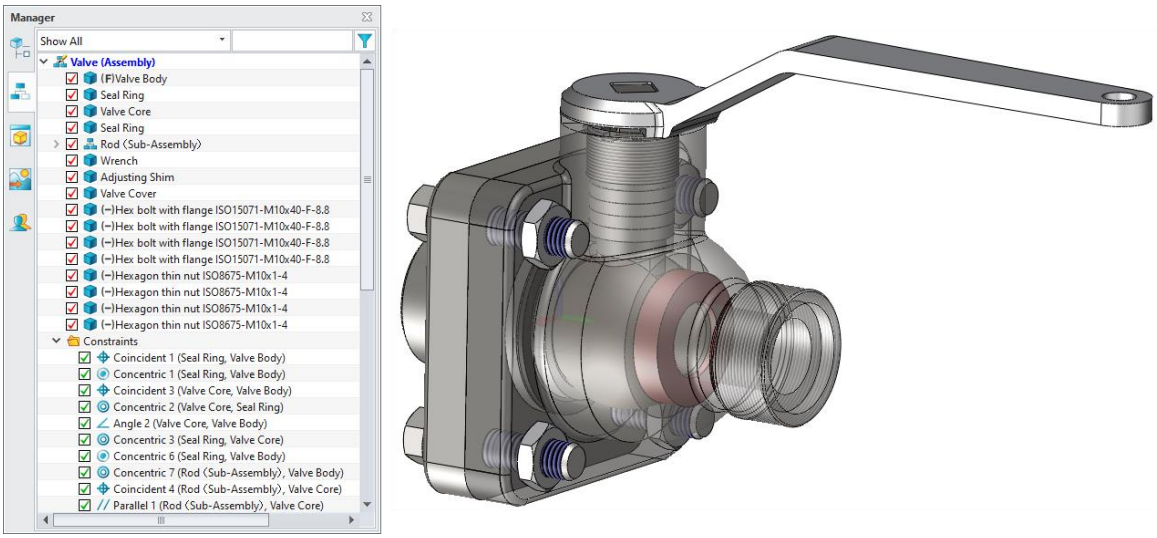



Figure 201 An example of assembly in ZW3D

## 5.1 Introduction to assembly

Any completed product of CAD design is composed of multiple parts, however, in the assembly level, the parts are normally called as the components which means the components assembled into an assembly modeling in CAD software.

Following are the definition of some nomenclatures in CAD technology.

**Part:** The single unit model which is independent. The part is consisting of design variables, geometry, material attribute and part attributes.

**Component:** The most basic unit which consist into sub-assembly or assembly. Besides, the component is a part when it is not in the sub-assembly or assembly.

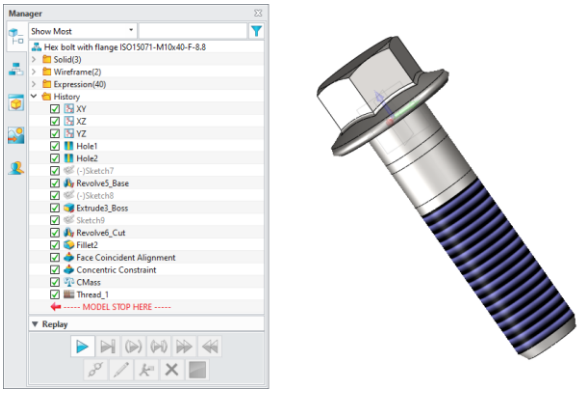


Figure 202 Part (with history manager)

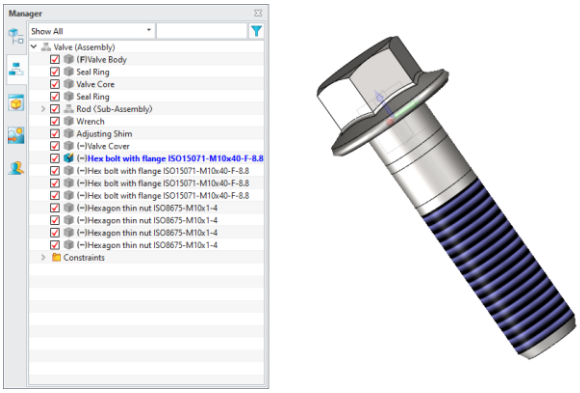
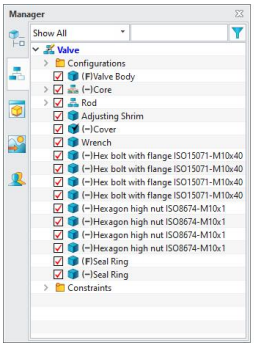


Figure 203 Component (with assembly manager)

**Assembly:** The final objective of assembly modeling which also can be called product and it is composed of different sub-assembly or component with constraints.

**Sub-assembly:** Literally it is the secondary assembly which compose into assembly and composed of different secondary sub-assembly or component with constraints.



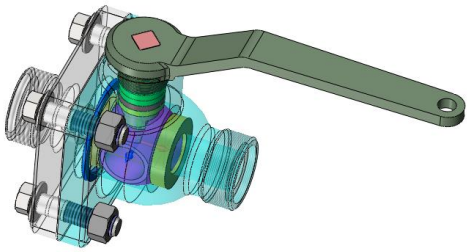
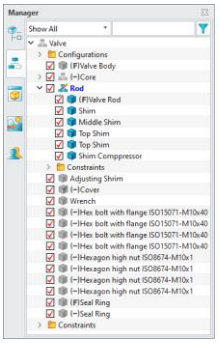


Figure 204 Assembly



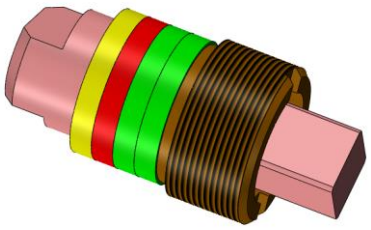


Figure 205 Sub-assembly

**Constraint:** In assembly modeling, the spatial position and relative motion of the components can be defined by constraint, then you can analyze whether there is interference between components and whether they are moving normally.

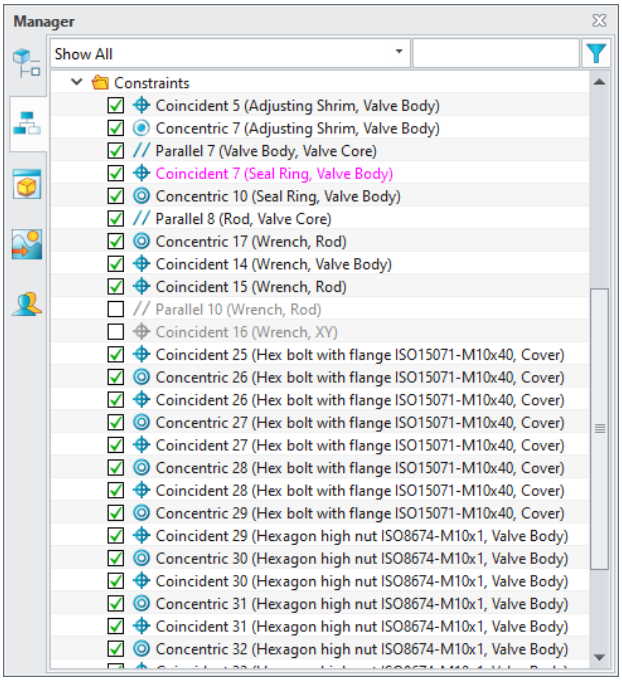



Figure 206 Constraints in assembly manager of ZW3D

To better understand the hierarchical relationship between various components of an assembly, an assembly tree is demonstrated below.

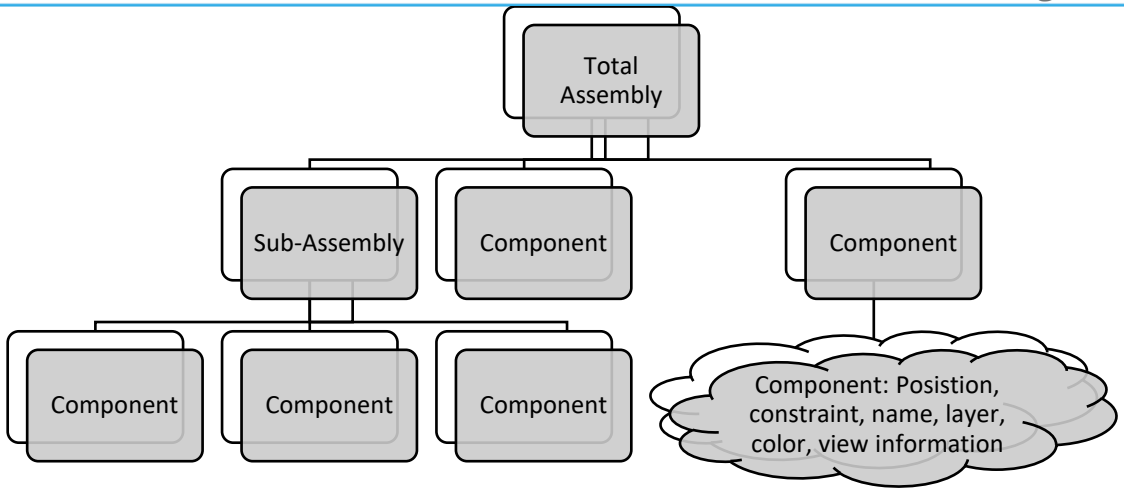



Figure 207 Assembly tree

As shown in the assembly tree above, an assembly can be divided into several sub-assemblies and components in different levels, and each of the sub-assemblies is also composed of different components. In the assembly tree, each of the leaves represent individual components or sub-assemblies. On the top of the tree with the highest hierarchy is total assembly.

## 5.2 Introduction to assembly approaches

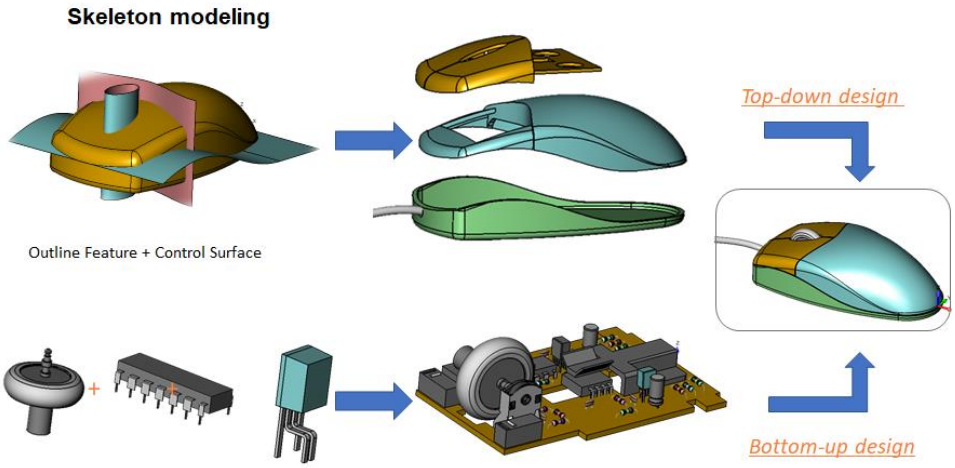



Figure 208 Top-down design and bottom-up design

Basically, there are two fundamental methods in assembly design, they are Bottom-Up design and Top-Down design respectively, each of them can meet different design requirements. If the design order is: Finish the independent part design first and then apply those parts into assembly, we call it bottom-up design, if the assembly or product shape is first completed and then design the associative parts in assembly, this is called as top-down design. Both 2 methods can meet various needs at design, the detailed introduction is as follows.

- **Bottom-Up design**

The bottom-up design is the most common used method, which is the traditional way to make the assembly. The individual parts are created separately and then assemble them to form the assembly of a product.

Since the geometry of the components is independent, changes to any component do not affect other components. Moreover, it is easier to maintain the relationships and regeneration behaviour of components in bottom-up design. If all the components have already been created and are ready to use, bottom-up design is more appropriate.

- **Top-Down design**

Top-down design is an associative design approach. In the top-level assembly, driving parameters, control sketch and product shape both can be defined. Then design process progresses are from the top assembly to the individual parts. If driven geometry or parameters are changed, the related components are affected.

In the software, the associated updates can be done automatically. In some assemblies that consists some associative components or during the R&D process of the products, the top-down design will be more ideal to manage the design.

ZW3D provides both bottom-up design and top-down design and these two methods can be combined in designing according to design purpose.

## 5.3 Attentions in assembly

### 5.3.1 Introduction to assembly manager

The assembly manager is a tab which can be accessed from the ZW3D data manager. It displays all the components inserted into the active assembly, the parent/child relationships of components and the constraints of the components. It is used to manage the whole process of assembly work. To show the assembly manager, you can select the manager button from tools menu in the bottom right of ZW3D and then select the assembly manager tab. Besides, you can also right-click the blank area of graphics windows to open the ZW3D manager.

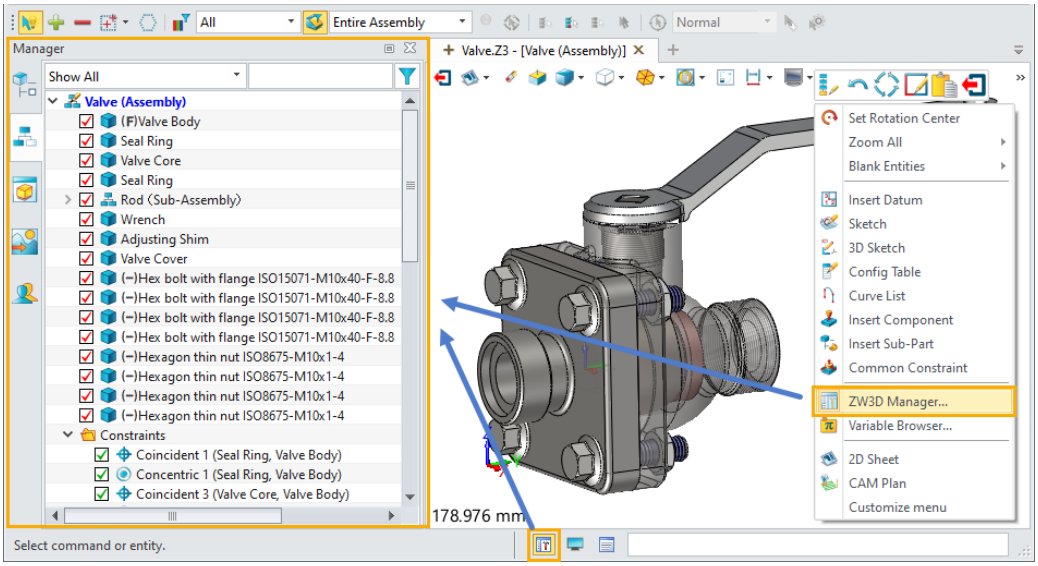



Figure 209 Turn on/ off Manager

The picture above is an example of the assembly manager, it normally includes, components, and constraints.

In assembly manager, there are some commonly used options which are introduced below.

- **Filter**

The filter can select whether show only components or constraints, or both as shown below.

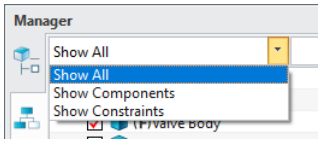



Figure 210 Filter

Below is the definition of each of the options in assembly manager filter.

Show All: The manager shows all information, including the inserted components and alignments.

Show Components: The manager only shows the components.

Show Alignments: The manager only shows the alignments.

- **Select/Preview the Component**

The assembly manager displays information about the active assembly, when you move the cursor on the specific component, it will be highlighted in the graphics window which can help you to locate the component in the complex assembly.

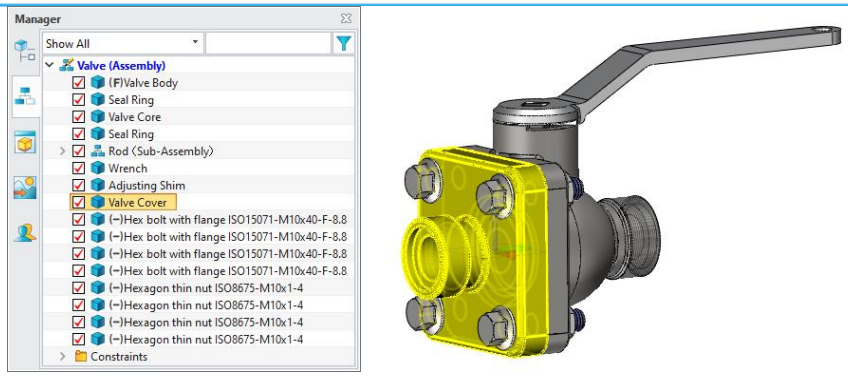



Figure 211 Select/Preview the component

- **Blank/Unblank the Component**

From right menu of the component in assembly manager or document aware toolbars, you can choose to blank or unblank the component, shown as the picture below. Besides, you can quickly blank the component by checkbox in the assembly tree.

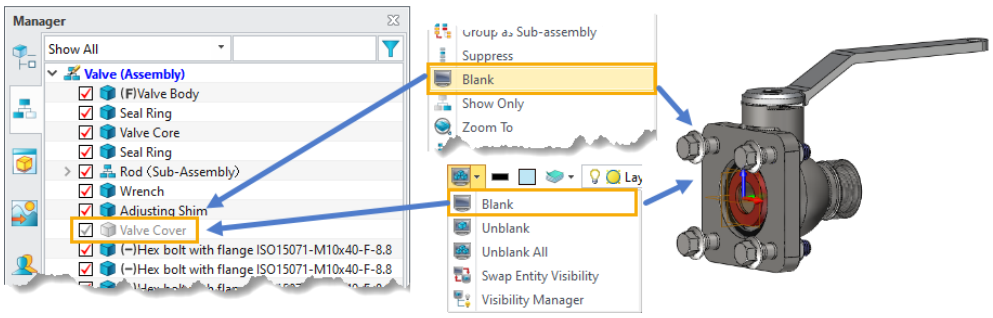



Figure 212 Blank the Component

- **Suppress/Unsuppress the Component**

From right menu of the component in assembly manager, you can choose to suppress of unsuppress the component, if the component is suppressed, the constraints which are related to the suppressed component will be unavailable.

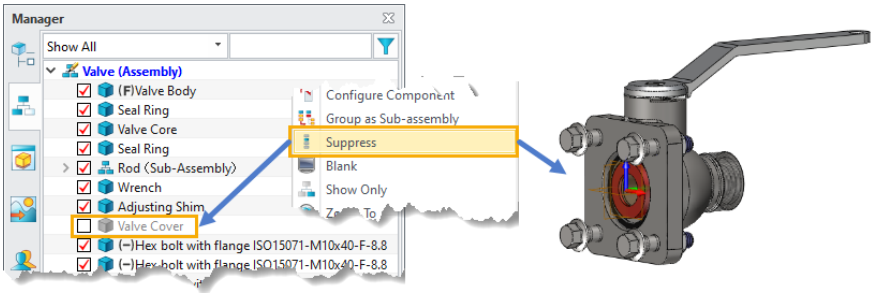



Figure 213 Suppress the Component

- **Disable/Enable the Constraints**

When right click on the certain constraint, in the right menu, user can disable\Enable the specific constraint. After disable or tick off the constraint, the constraint will not take effect until user enables it.

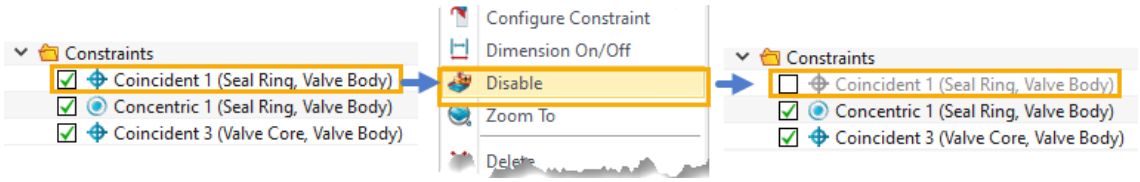



Figure 214 Disable the Constraints

- **Display Mode of the Components and Constraints**

There are two types of display mode in ZW3D assembly manager, **separated mode** and **combined mode**, with this two modes, users can display the constraint with different locations, with separated mode, all the components and the constraints are displayed separately, with combined mode, each components and related constraints are displayed together, the picture below the separated mode(Left) and combined mode(Right), you can switch between this two modes by right menu of assembly manager.

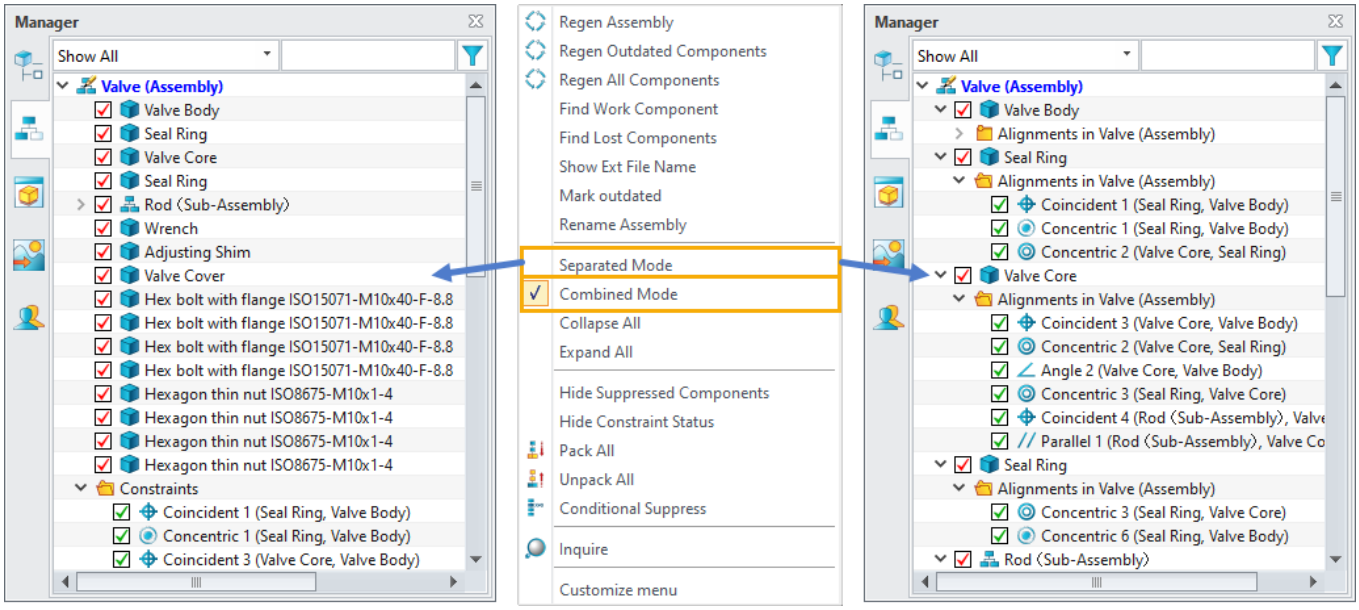



Figure 215 Separated Mode (Left) and Combined Mode (Right)

- **Inquire Parent Assembly**

In assembly manager, you can also view the parent/child relationships of components and the constraints relationships between components in the active assembly or right-click the component to select the command ***Display Parent*** from the menu.

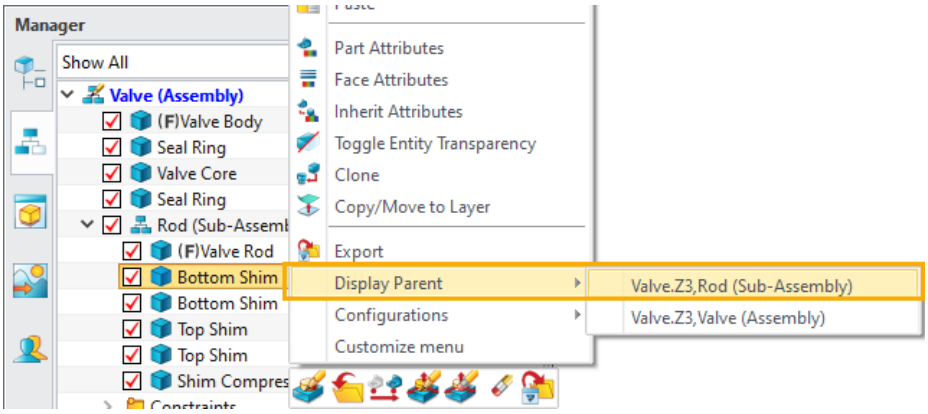



Figure 216 Display Parent

The detailed introduction of right menu can refer to the Assembly Manager Description in ZW3D help documents.

### 5.3.2 Insert Component

In ZW3D, before you start a new assembly design, you could create a new assembly file or add a new assembly object in the existing file (such as*.z3), then assemble parts into the assembly.

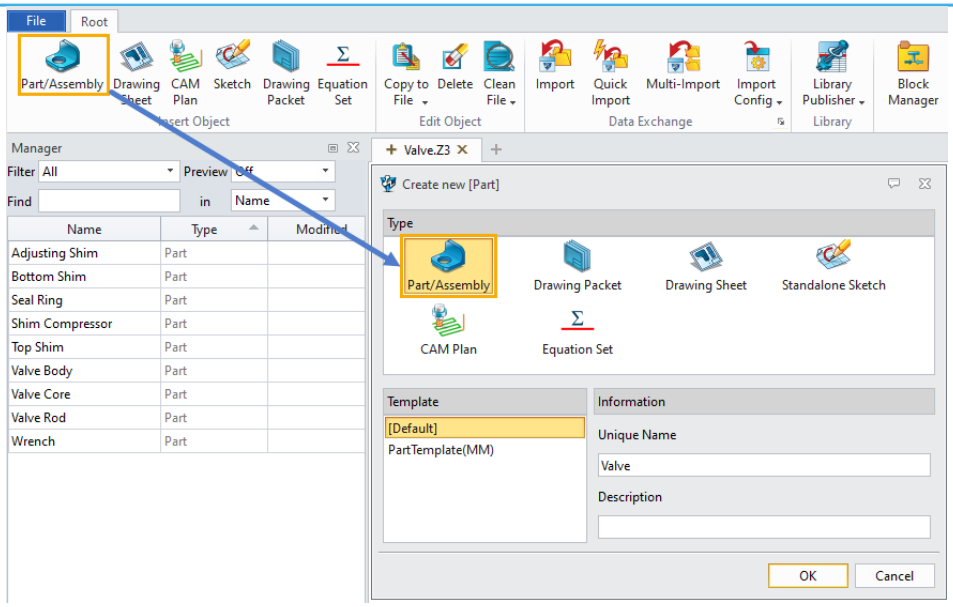



Figure 217 Create a new assembly

In assembly module, there are various tools in the ribbon bar. in first step, you need to insert the first component by ***Insert*** command. From the ribbon tab or the right menu when clicking the space of graphics area, you both can access to ***Insert*** command, as shown in the image below.

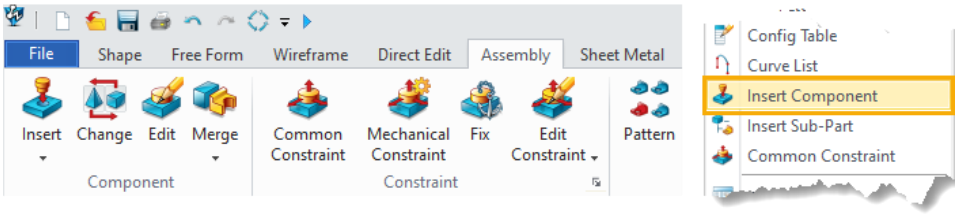



Figure 218 Insert command

To easily find the component in the File/Part, you can select the ***Graphics*** type from preview, afterwards, input the insert location, you can either click the point in the graphics window. moreover, tick ***Fix component*** option is highly recommended for the first component thus following components can refer to this fixed one.

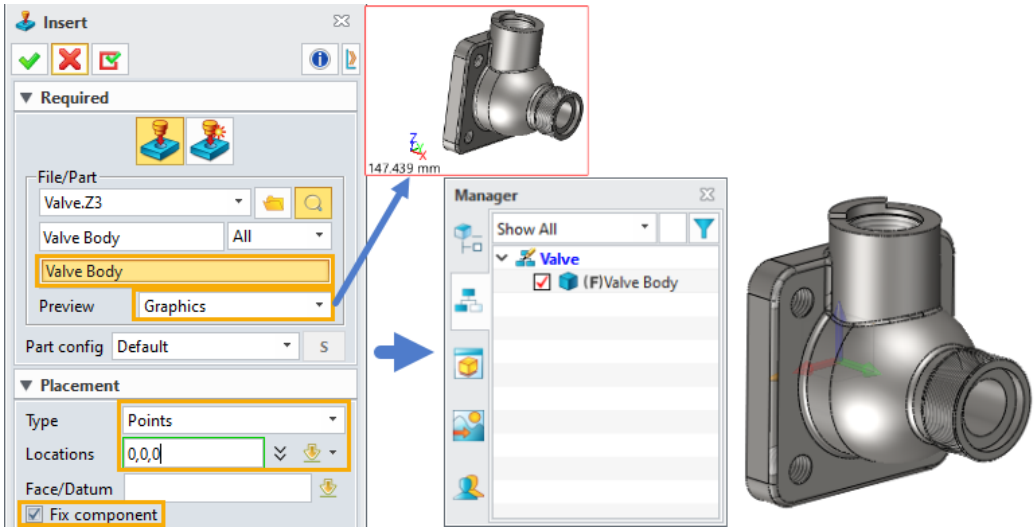



Figure 219 Insert the component

- **Multi Insert**

Apart from insert the component one by one, ZW3D have a more convenient and efficient way to insert batch components by using ***Multi insert***.

With ***Multi Insert*** you can insert all the needed components by just one-time operation, similar as single insert, you can use the ***Preview*** to help you find out the needed components and then input the insert location. The picture below shows the dialog box of ***Multi Insert***.

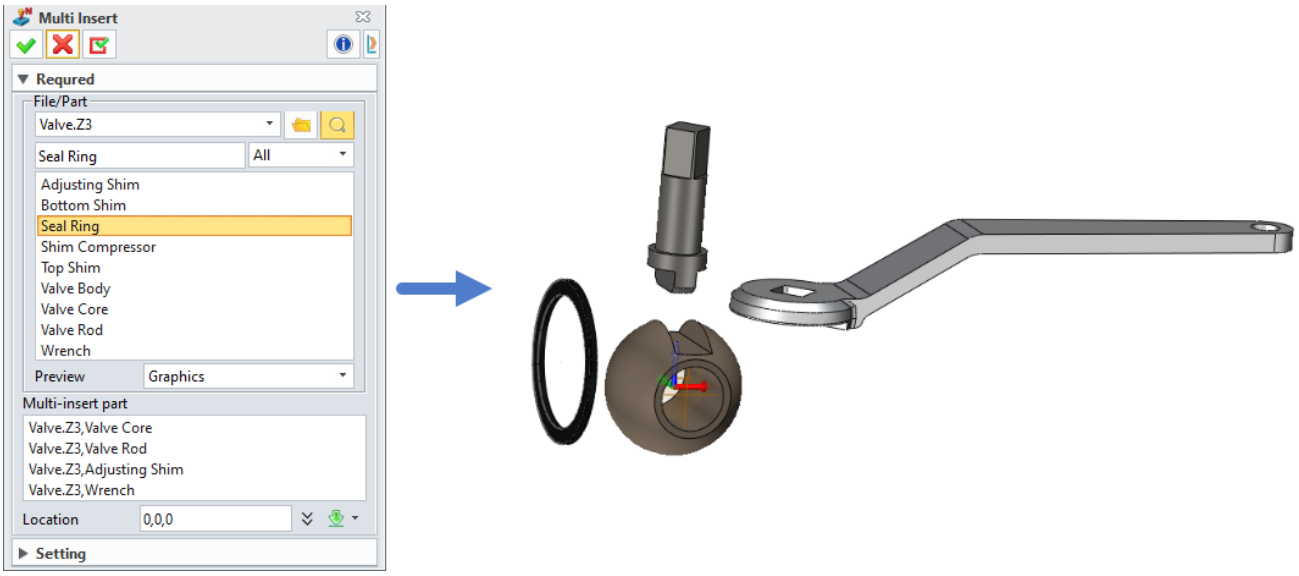



Figure 220 Multi insert

### 5.3.3 Constraints Definition

After inserted the components, how to fix the relative location or set the motion range between different components? Except for adding the component, proper constraints are necessarily to be added for the completed assembly. Next part, we will introduce the constraints in ZW3D and how we can easily add reasonable constraints for each of the components.

In ZW3D assembly ribbon tab, you can find several commands about constraints, as shown in the picture below. They are common constraint, mechanical constraint, fix and edit constraints respectively, In this section, we will focus on Common Constraint.

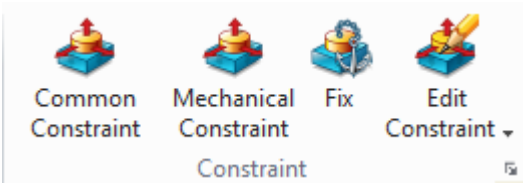



Figure 221 Constraint ribbon panel

During adding the constraints, compared with the part geometry, the datum of each component is highly recommended to use as prior reference, because it will not be affected if the component is changed.

Now, let take an example to show you how to add the constraint for the valve core with datum faces.

First, enable ***Unblank External Datum*** by right click the component as shown below. Then the external datum will be displayed.

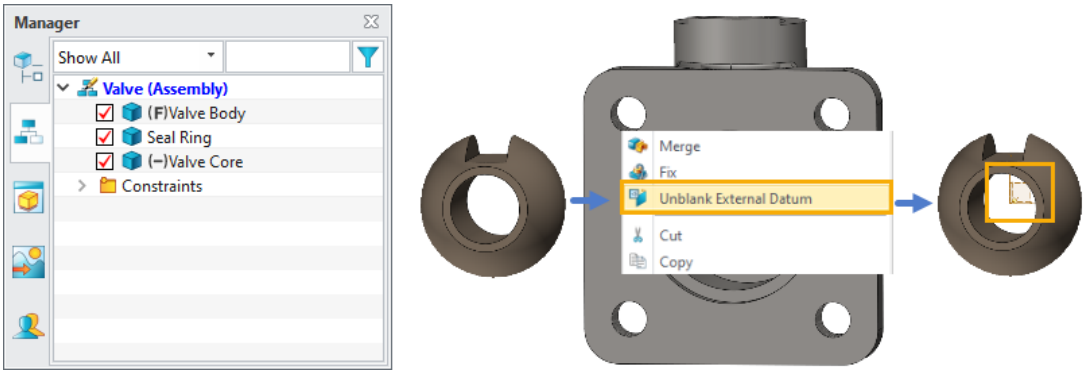



Figure 222 Unblank External Datum

For some components, if you think the datum size is too small to add the constraints, please go to ***Visual Manager*** -> ***Datum***, then turn on ***Auto Size***.

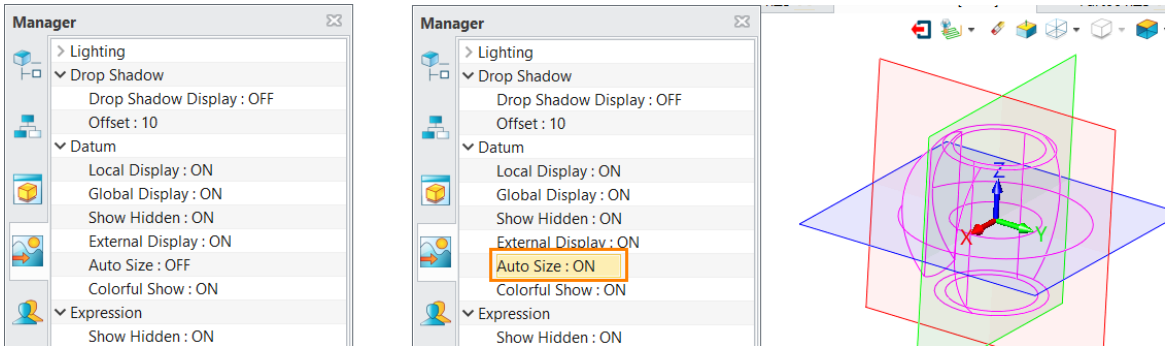



Figure 223 Turn on Auto Size

ZW3D provides various kinds of constraints by ***Common Constraints***, let us take the seal ring as an example.

STEP 01 Insert the **Seal Ring**, and select ***Common Constraint*** from ***Assembly*** ribbon tab, then select the **coincident** constraint.

STEP 02 Select the bottom face of **Seal Ring** as the 1st entity and the face inside **Valve Body** as 2nd entity as shown in the picture below.

STEP 03 Set the offset or range value according to the needs. In this case, set the offset to **0** mm.

STEP 04 If the direction of the constraints is not the needed one, you can click ***Flip Direction*** button.

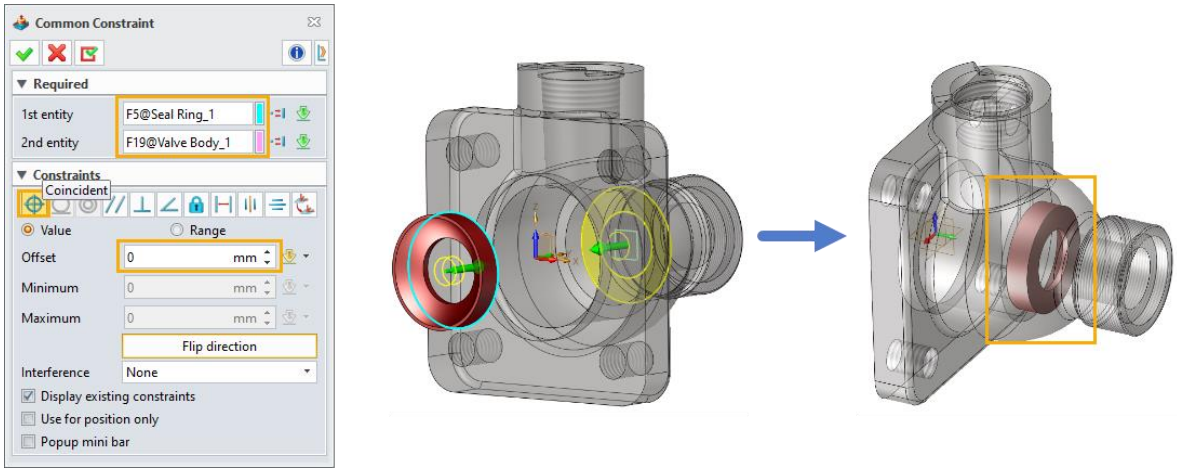



Figure 224 Add coincident constraint

In this case, parallel constraint is applied between valve core XY datum and valve body XZ datum as shown in the image below. If the direction of the constraints is not the needed one, you can click the flip direction button.

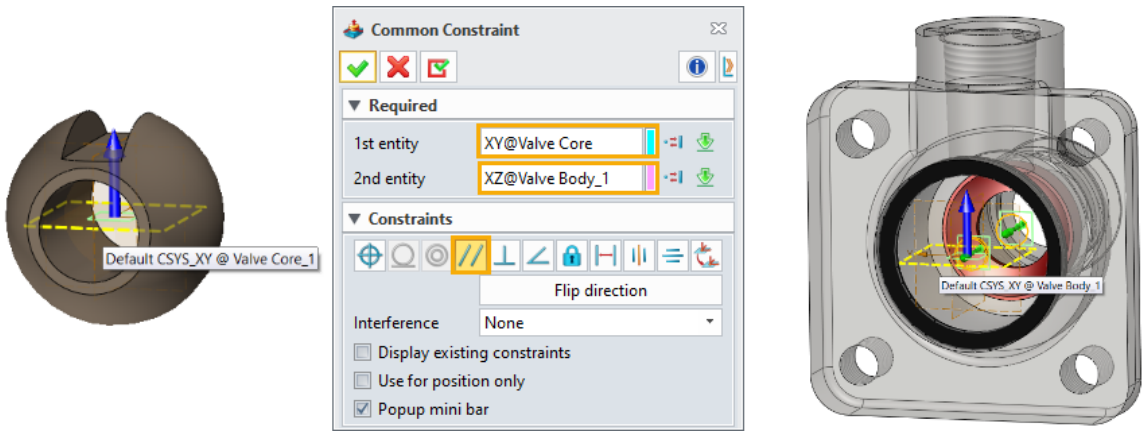



Figure 225 Add parallel constraint

Until now, we have learned how to add the basic constraints for components, afterwards, you could try to learn more.

### 5.3.4 Edit Constraints

If you not only want to check the constraint status but also you want to redefine the constraint, under this circumstance, you can right click the component from assembly manager or graphic windows and then click ***Edit Constraint*** to open the edit window, you can either find the **Edit Constraint** from ***Assembly*** ribbon tab and then pick the component.

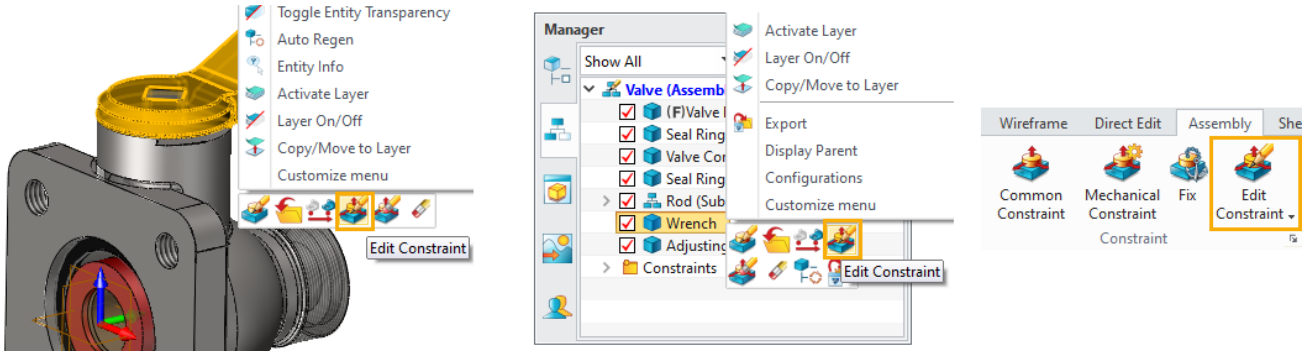



Figure 226 Edit Constraints

With edit constraint window, you can find all the relevant constraints of the picked components and then edit them if needed.

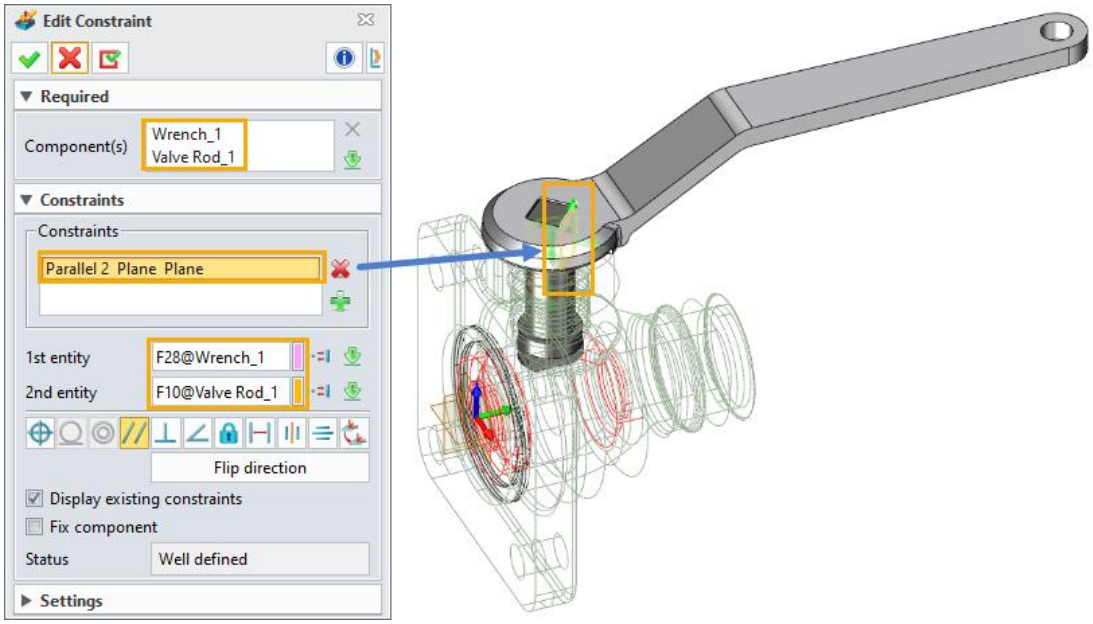



Figure 227 Edit constraint dialog box

### 5.3.5 Check Constraint Status

After the constraints has been added for all the components, you may wonder whether there are any missing constraints, in other words, whether assembly is well-constrained. Now, the way to check the status of constraints will be introduced in the following.

In ZW3D, it is very easy to check the constraint status between different components, the most convenient way to check the constraints status is view assembly manager. As can be seen from the picture below, there is a symbol (F)/ (-)/ (+) on the left of component.

***(F)*** means this component is fixed.

***(-)*** means this component is under-constrained. The appropriate constraints can be added for it.

***(+)*** means this component is over-constrained. There are redundant constraints that create conflicts.

***No symbol*** means this component is well-constrained.

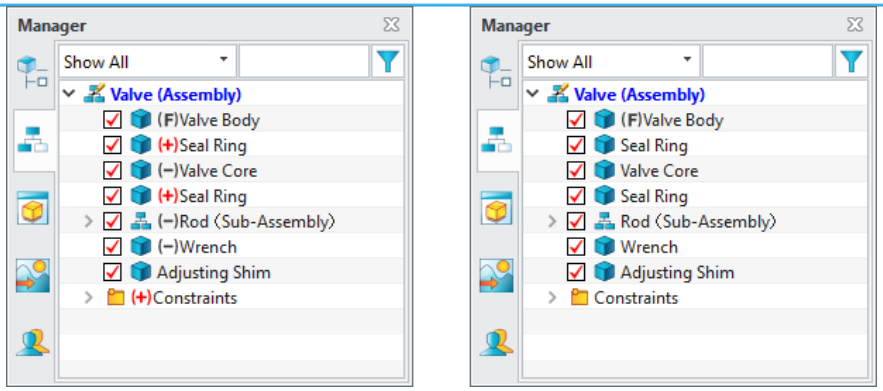



Figure 228 Constraint status

Apart from view the symbols in the assembly manager, you can use ***Constraint Status*** command to check the current constraint status for components.

The picture below is the constraint status checking windows in graphic interface. As can be seen in the graphic interface, the constraints status was expressed with different color. It is intuitive for user to check the status.

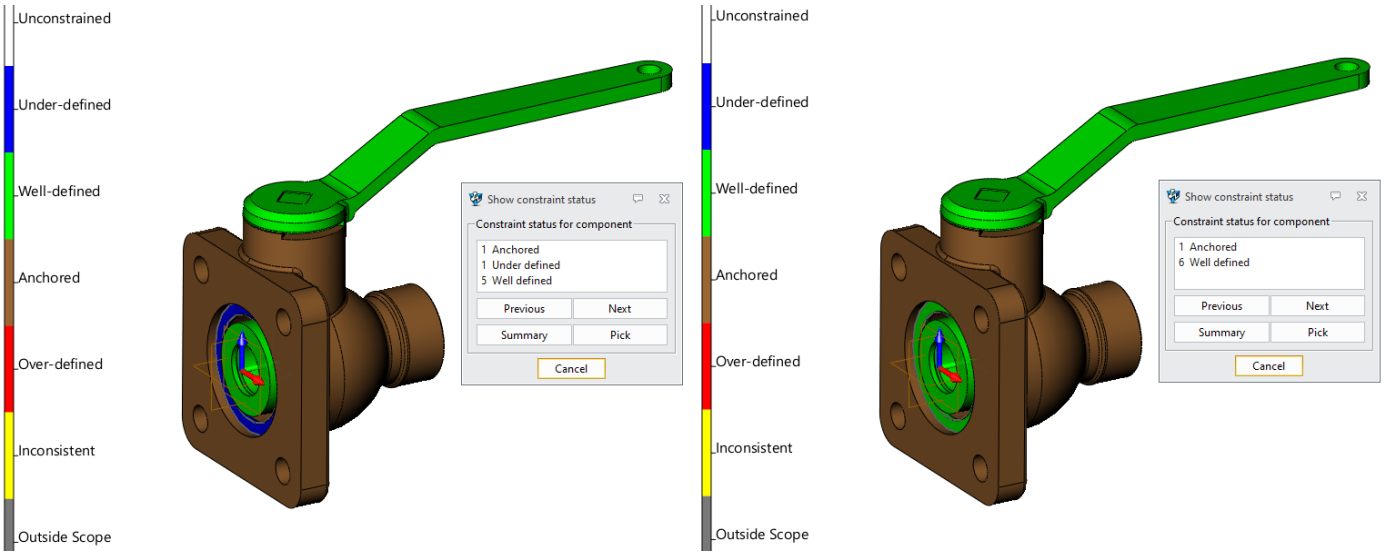



Figure 229 Check constraint status

### 5.3.6 Check Assembly Motion

After finishing all the assembly work, if you want to check the motion status of components, with the ***Drag*** or ***Rotate*** function, you can move or rotate the component. or check the motion status. The picture below shows the drag and rotate command.

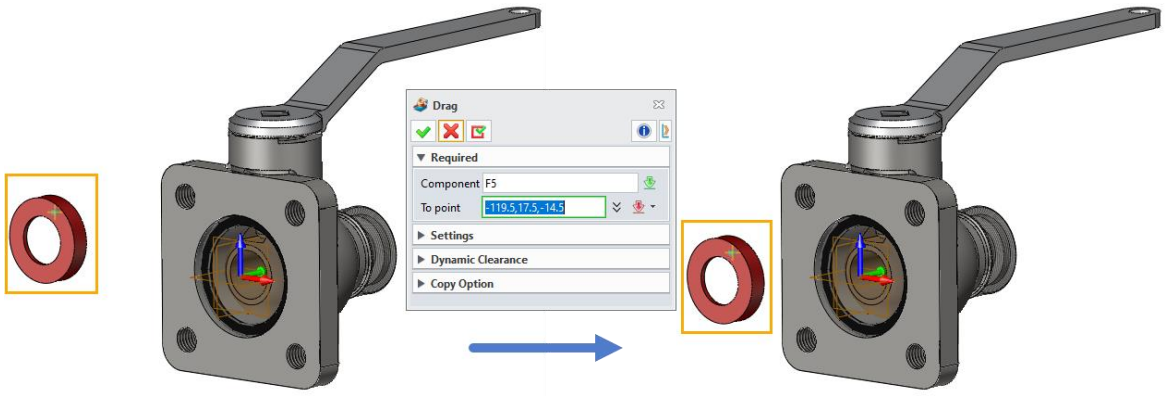



Figure 230 Drag command

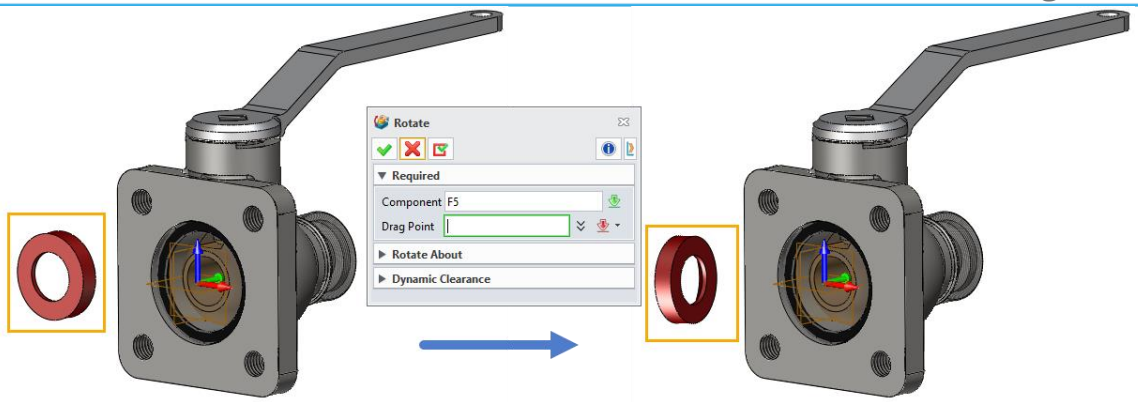



Figure 231 Rotate command

Also, if you just want to simply check the component motion, you can directly select the component from the graphic window and drag or rotate it by **left mouse button**.

### 5.3.7 Interference Check

In the complex assembly, it is hard to visually inspect whether there is interference inside the assembly, however, in ZW3D you can use the ***Interference Check*** function to inquire interference, which you can find from the ***Assembly*** ribbon tab -> ***Inquire*** panel.

***Interference Check*** function provides two different methods to define checking scope. They are ***Only among the picked*** and ***With other components*** respectively, below is the definition for these two methods.

- **Only among the picked:**

Only check the interference between the picked components.

- **With other components:**

Check the interference between the picked components and other unpicked components.

The picture below is an example of ***Only among the picked***. First select the components which needs to check the interference, and then you will get the result after click ***Check*** button. As can be seen in the picture below, the interference results are shown in the result window and in the graphics window.

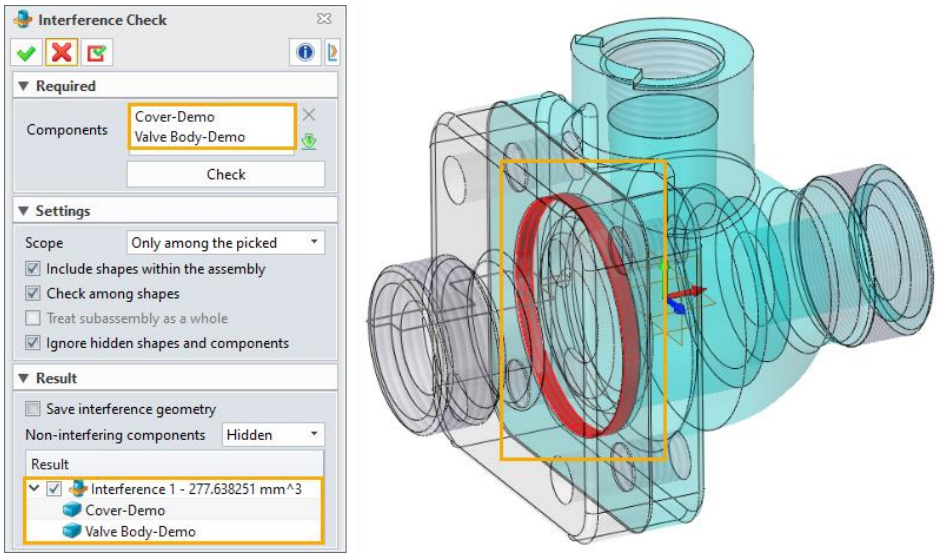



Figure 232 Interference Check

***Dynamic Section View***

If you want to get more clear view of the interference area, go to ***Inquire*** ribbon tab->***Section*** tool is recommended, which delivers dynamic and visual interference check. The picture below shows the section view of the interference part.

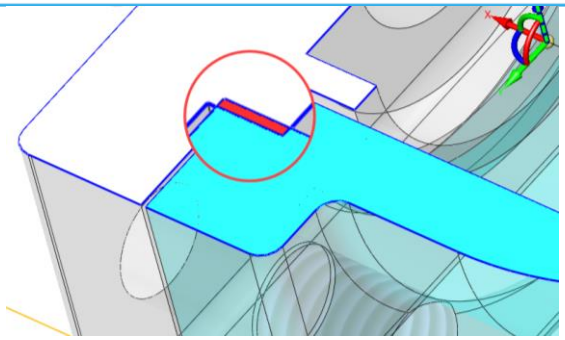


Figure 233 Dynamic section view

### 5.3.8 Exploded View

To make sure other person could easily understand the internal details and assemble process of the assembly, ***Exploded View*** command from ***Assembly*** ribbon tab can help us go into a separate working area to create the exploded view in ZW3D, as shown in the image below.

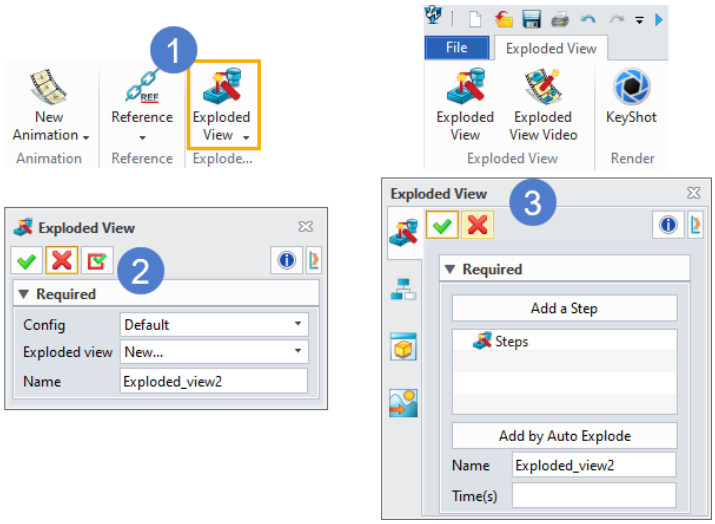



Figure 234 Exploded view working area

There are two ways to create the exploded view, the first one is adding every explosion step manually by ***Add a step*** and another one is to click ***Add by Auto Explode*** button to generate self-explosion steps automatically. We recommend adding the step manually to create expected result more precisely.

When you ***Add a step*** manually, the ***Move*** command is activated to move the component. There are 6 different ways to create the exploded step. The picture below shows an example of ***Move entities along a direction***, you can define the moving direction and the distance.

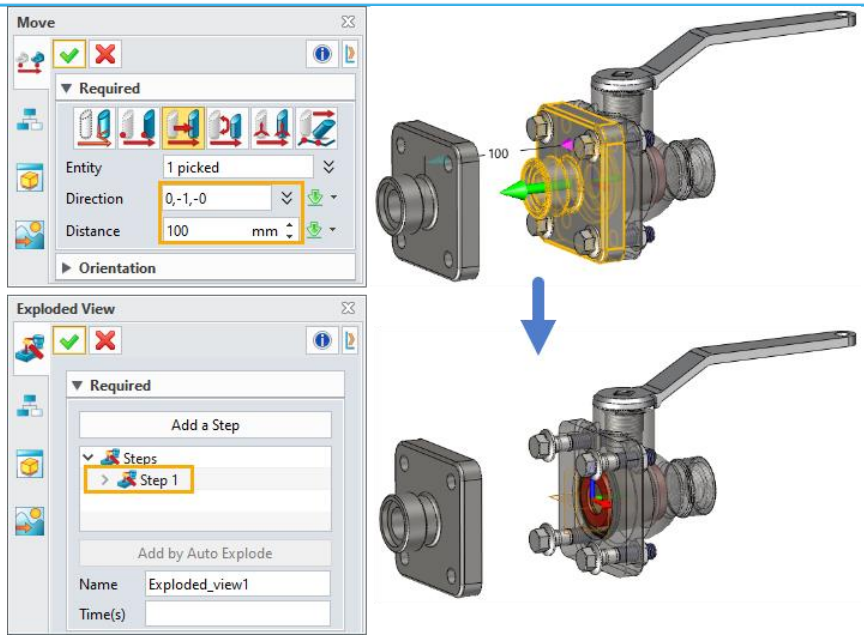



Figure 235 Move entities along a direction

After creating all the needed steps, you can save the exploded view as an AVI video by ***Exploded View Video***. When you back to assembly level, you can check the ***Exploded view*** from configuration of assembly manager.

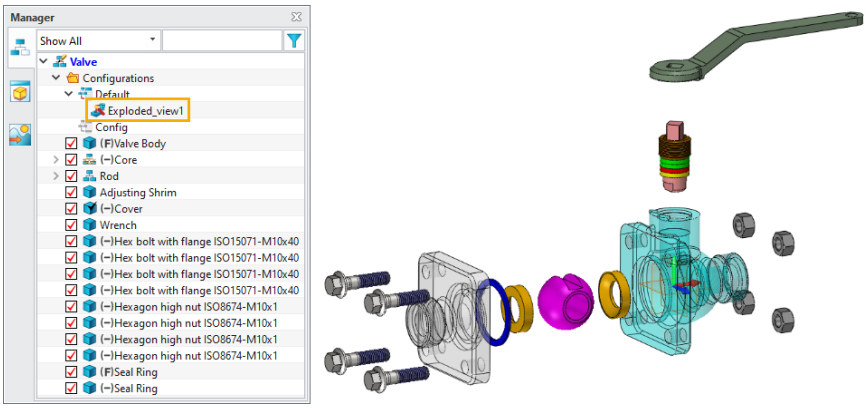



Figure 236 Exploded view

## 5.3.9 Associative Reference

Associative Reference is mostly used in top-down design. With ***Assembly*** ribbon tab-> ***Reference*** function, you can create a reference geometry that linked with the external geometry from other components.

Now, let us look at the reference function.

There are five different types of reference in assembly, they are **Curve**, **Plane**, **Point**, **Face**, and **Shape** respectively. Before adding the associative reference, you need to active the related part in assembly and then select the ***Reference*** command. The picture below shows the **Face** reference setting, and the highlighted face on the ball valve is the selected reference face.


Figure 237 Example of associative reference

After finished creating the reference, you can use the reference geometry to create another geometry, the picture below is using the reference surface to extrude a solid.

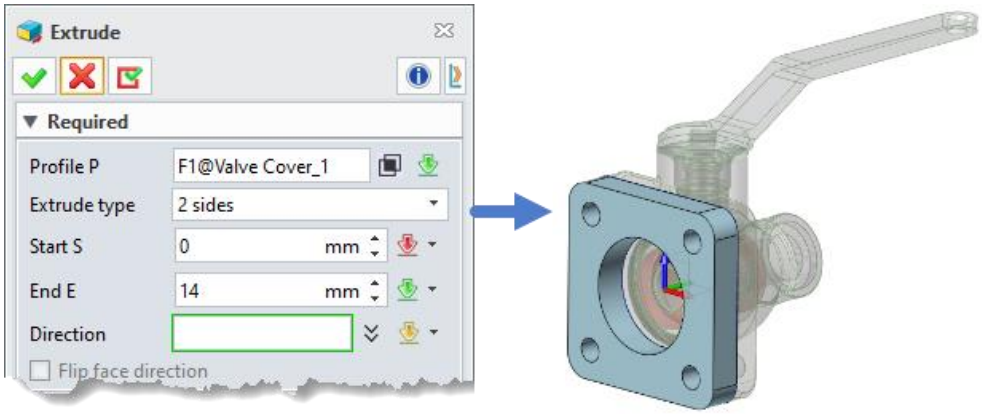



Figure 238 Extrude a solid from the reference surface

As can be seen in the figure above, the associative copy is checked in association setting, this option is used to create reference geometry which associates with the referenced external geometry. The reference geometry will be re-evaluated each time when referenced geometry is regenerated. If not checked, reference geometry created by this option is a one-time static copy. The picture below shows the comparison of the associative reference with/without associative copy.

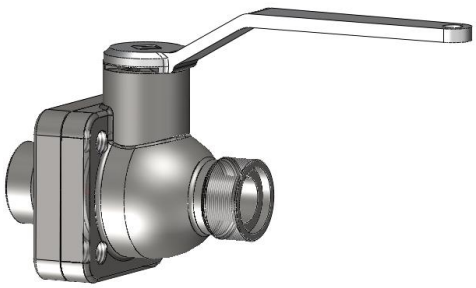
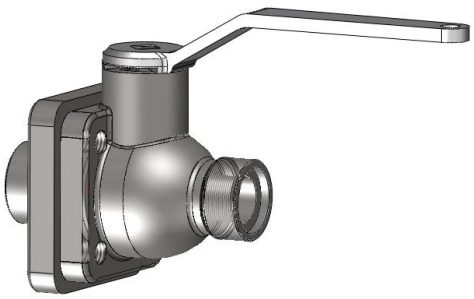
| Reference-Associative copy (**checked**) | Reference-Associative copy (**un-checked**) |
|---|---|
|  |  |

Figure 239 Associative reference with (Left)/without (Right) associative copy after changing the valve body size

### 5.3.10 Standard parts in ZW3D

- **Build-in Library**

In ZW3D, there are various kinds of standard parts which you can find in ***Reuse Library*** tab, as shown below.

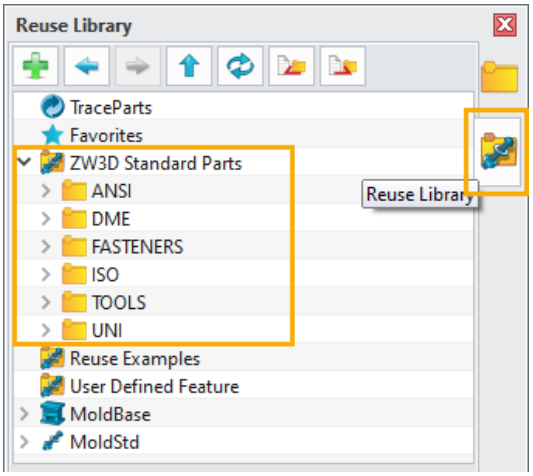



Figure 240 Reuse library

With these large amounts of different kinds of standard parts database, you can directly use it in your assembly, the picture below shows an example of using the ISO standard hexagon head bolts in ball valve assembly.

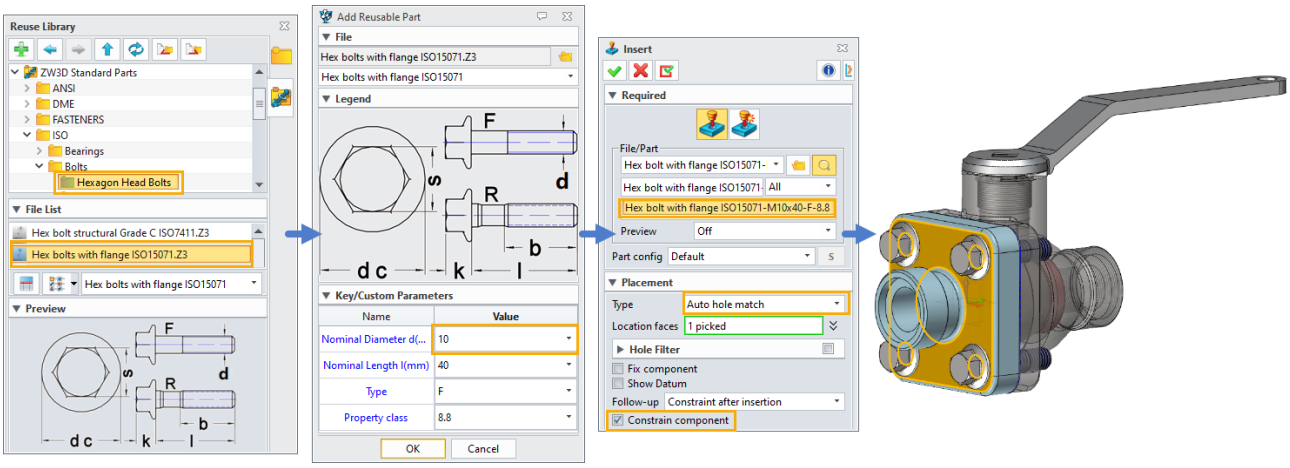



Figure 241 Insert ISO standard hexagon head bolts in ball valve assembly

- **PARTsolutions Library**

Apart from ZW3D standard parts, ZW3D also has free third-party standard part library PARTsolutions, you can access to it from ***APP*** ribbon tab, however, you need to download and install it before using it.

The picture below shows the catalogs of PARTsolutions and the interface of model parameters. Various kinds of mainstream standard parts are provided, you can use them to meet your design requirements.

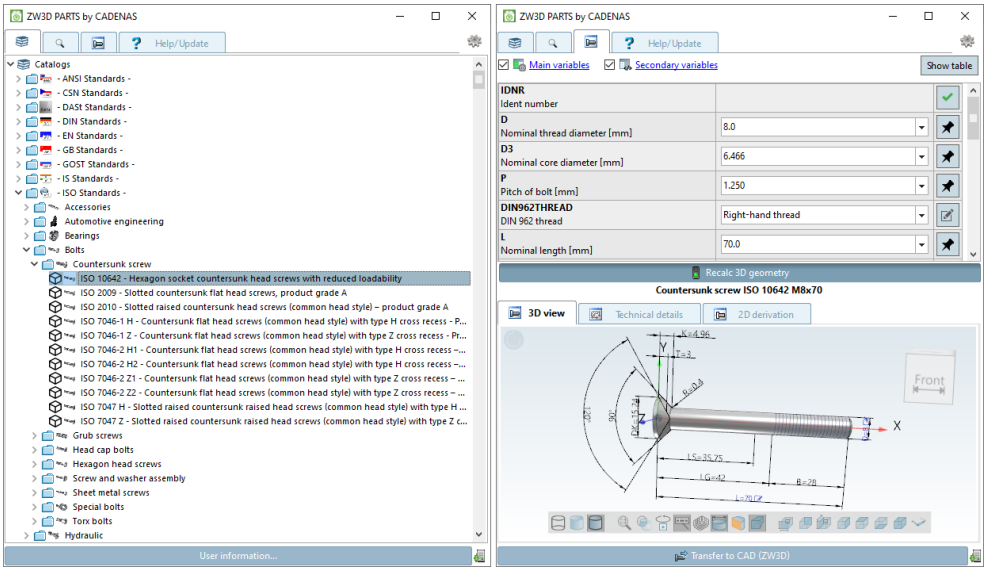



Figure 242 PARTsolutions

## 5.3.11 Rename Assembly

Change the object name or file name are very common after finishing the assembly design, with ***Rename Assembly*** command in ZW3D, you can rename any assembly or component and keep the associative relationship between them. ***Rename assembly*** command can be accessed from the right menu from the assembly manager.

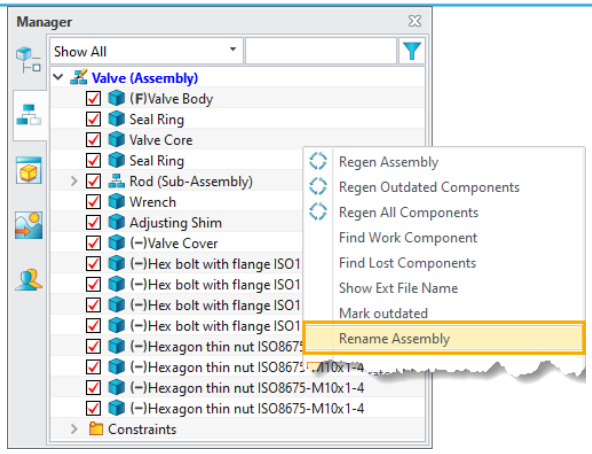



Figure 243 Rename assembly command

ZW3D can automatically generate a new name according to the naming rules. By setting a prefix or suffix and changing the file name in batches. Press Ctrl/Shift to multiple objects.

You can change old object name and old file name by just double-click and directly enter a new name in the marked area.

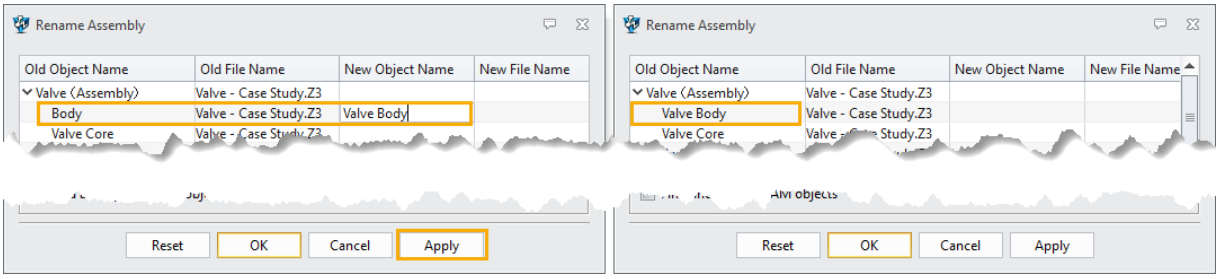



Figure 244 Change the object name in rename assembly

***Note:*** *For the single object file (*.z3prt),the object name is the same with the file name. Whether the object name or the file name is changed, they will be updated synchronously.*

## 5.4 Assembly designing case

Now we have finished the basic study of assembly module, in this section, we will take a ball valve as an example to teach you how to finish a whole assembly.

The picture below shows the completed assembly of a ball valve.

As you can see in the assembly manager, the ball valve consists of kinds of components and a subassembly. Based on the previous modeling design, let us finish the assembly design.

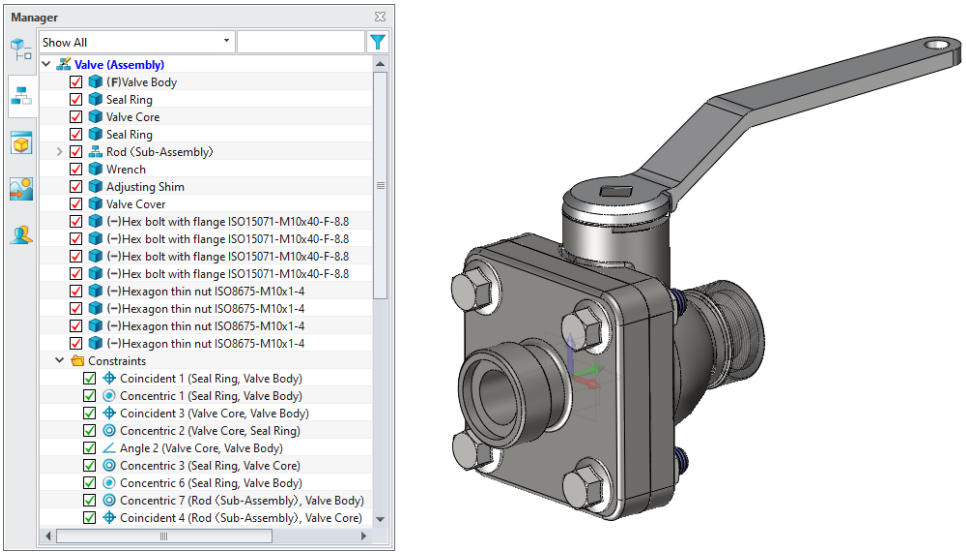



Figure 245 Completed assembly of a ball valve

### 5.4.1 Create a new assembly file

STEP 01 Create a new assembly object in the existing Z3 file.

Open the existing **Valve.z3** file which was created in the previous chapter and create a new ***Part/Assembly*** object from ribbon panel. In this step, we will create an assembly (**Rod**). Then we will enter the part/assembly level of Rod.

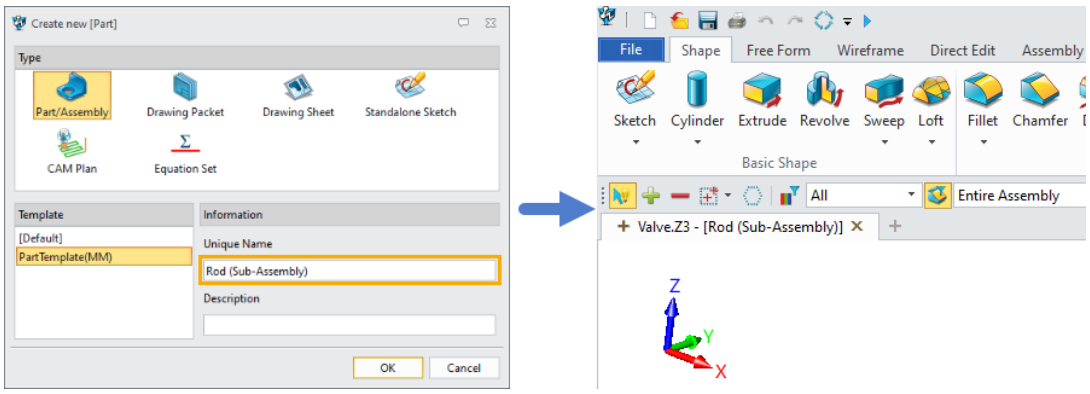



Figure 246 Create a Rod assembly

STEP 02 Insert the first component **Valve Rod**.

Select ***Insert*** command from ***Assembly*** ribbon tab.

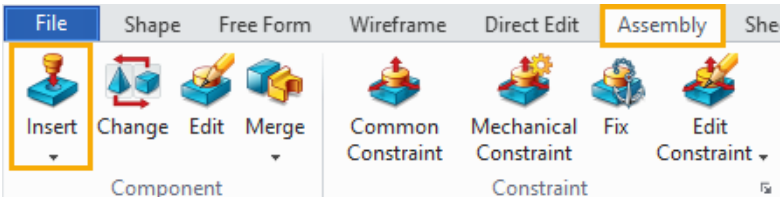



Figure 247 Insert command

Select **Valve Rod** to insert, then select the origin of coordinates as the insert location and **XY datum** as the insert face. After that, select ***Fix component*** option because this is the first component of the assembly, it would be better to fix it after inserting it.

Finally, click ***Ok*** ***（√）*** to fix the first component.

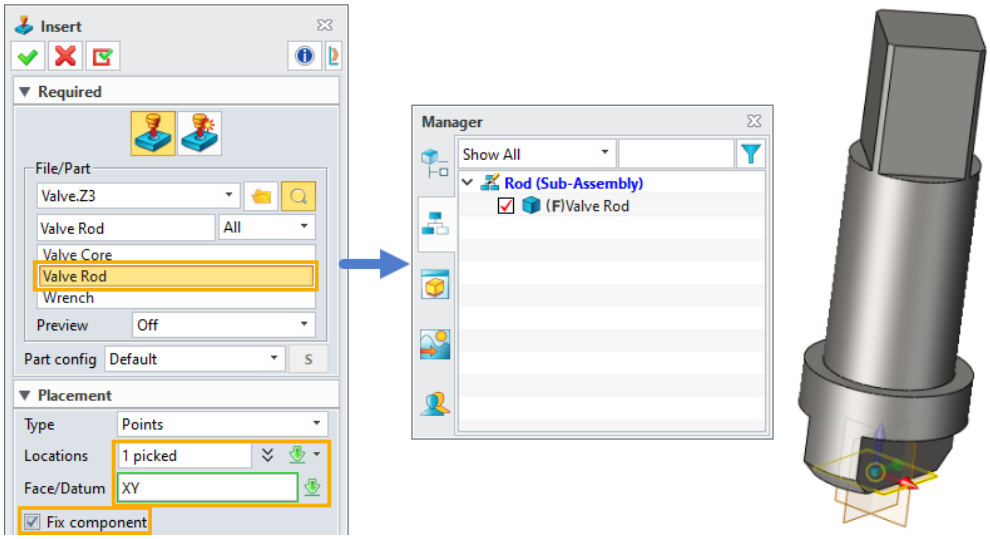



Figure 248 Insert first component

STEP 03 Insert the second component **Bottom Shim**.

Select ***Insert*** command and select **Bottom Shim**, then choose any of the location to insert and untick ***Fix component***. Click ***Ok ( √)*** button to insert the component, then common constraint dialogue box will pop-up.

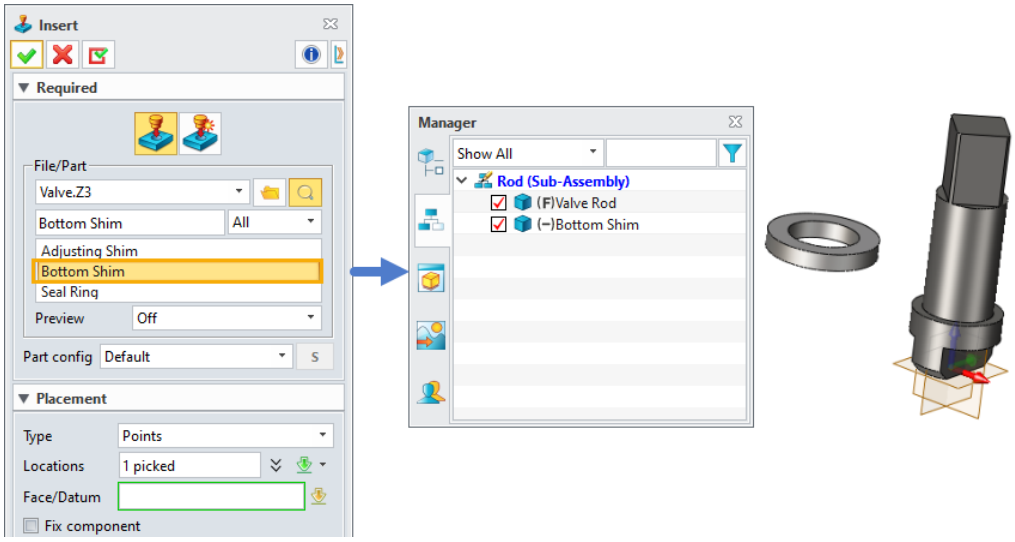



Figure 249 Insert bottom shim

STEP 04 Add the coincident constraint.

Select **Coincident** constraint and then select the bottom face of **Bottom Shim** as the 1st entity and the face on the **Rod** as 2nd entity, as shown in the picture below, after that, set **0mm** in offset and click ***Apply***.

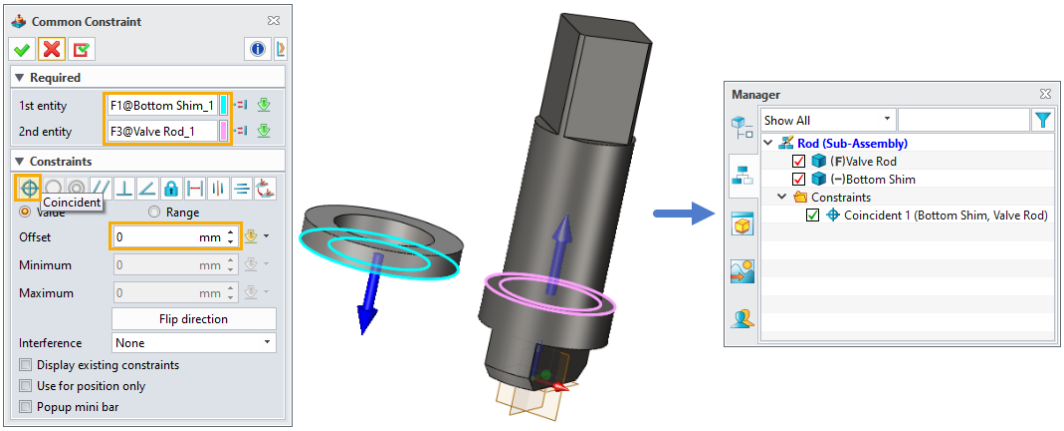



Figure 250 Add the coincident constraint

STEP 05 Add the concentric constraint.

Select **Concentric** constraint and then select the inner ring of **Bottom Shim** as the 1st entity and cylinder profile of rod as 2nd entity, after that, tick ***Lock rotation*** and click **Apply**, after that, we can find the **Bottom Shim** is well constrained in assembly manager.

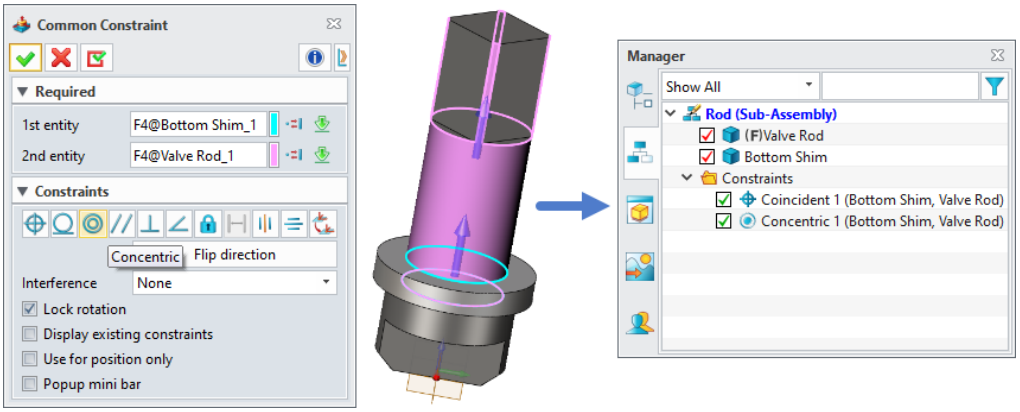


Figure 251 Add the concentric constraint

STEP 06 Use the same method as the previous step to insert another **Bottom Shim** and **Top Shim** and **Shim Compressor**, the result will be same as the picture below.

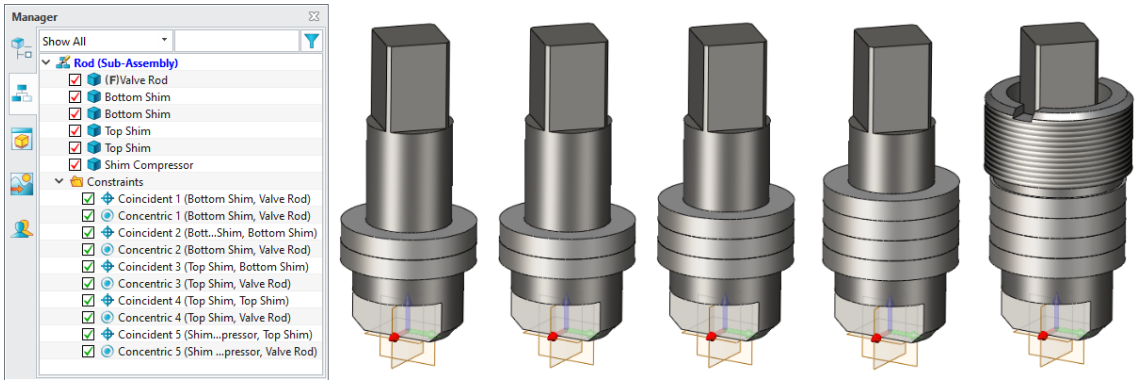



Figure 252 Insert the remaining components

Now, we have finished the assembly work of sub-assembly (Rob). In the next part, we will assemble the completed product valve.

### 5.4.2 Create the final assembly

➢ **Part 1: Create the top assembly**

STEP 01 Exit the Rod (Sub-Assembly) and move back to root manager.


Figure 253 Back to root manager

STEP 02 Create a new ***Part/Assembly*** object from ribbon panel, in this step, we will create the **Valve (Assembly)**. Then, we will enter the part/assembly level of **Valve (Assembly)**. In the next part, we will insert the components into this assembly.

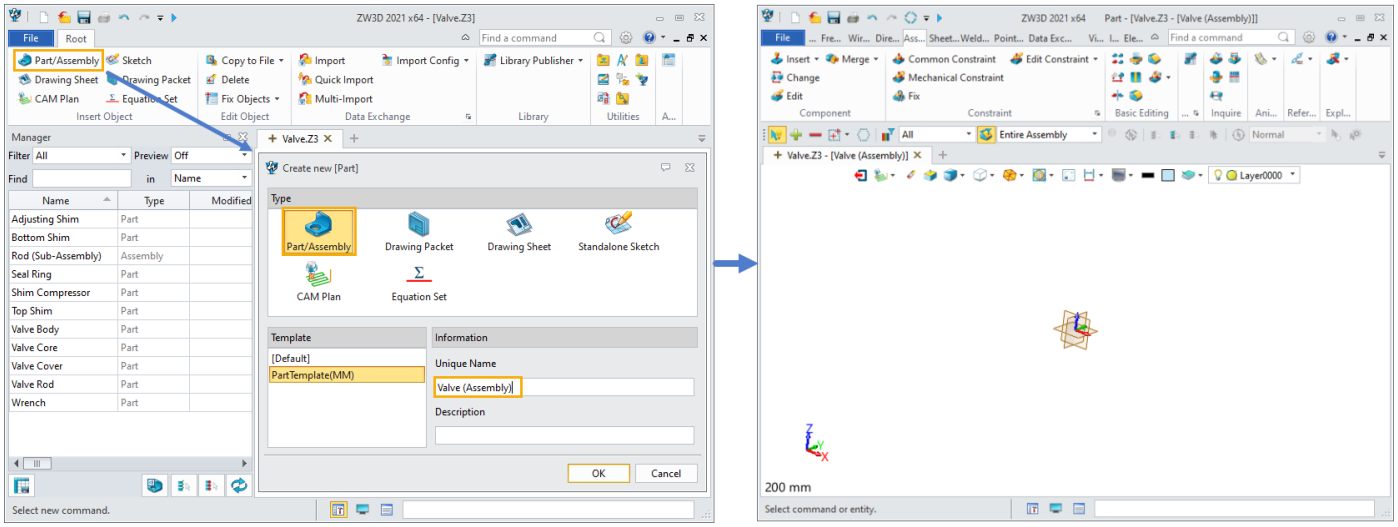



Figure 254 Create the final assembly

➢ **Part 2: Insert the first component (Valve Body) and fix it**

STEP 03 Insert **Valve Body** as the first component, select the **origin of coordinates** as the insert location, then ***Rotate*** the orientation to make the **Valve Body** same as the picture below. Besides, because this is the first component of the assembly, it would be better to fix it after inserting it.

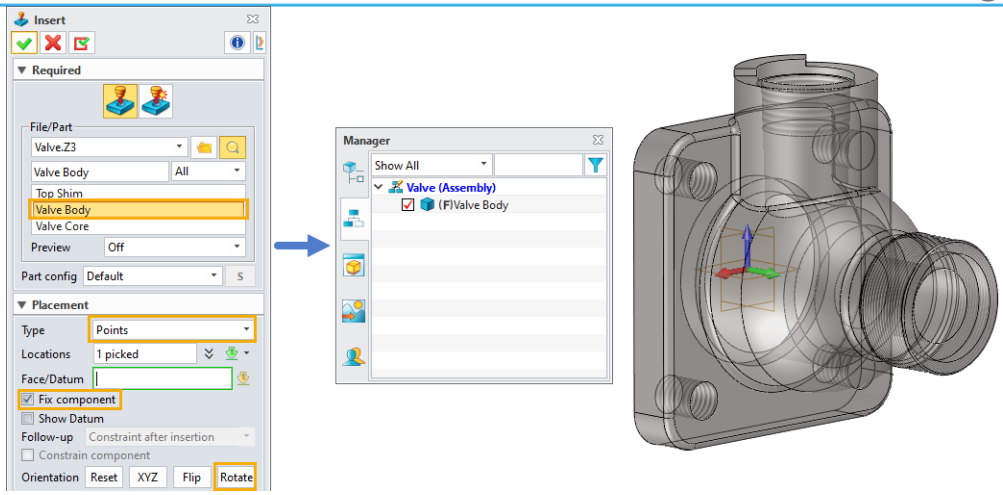



Figure 255 Insert valve body

➢ **Part 3: Insert the component (Seal Ring) and add the constraints.**

STEP 04 Insert **Seal Ring** and choose any of the location point and untick ***Fix component*** option to insert.

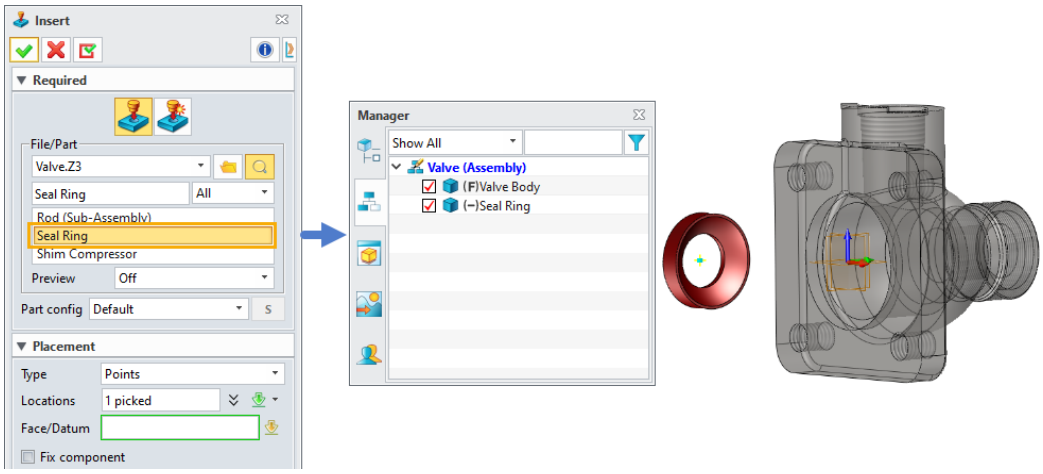



Figure 256 Insert seal ring

STEP 05 After Insert **Seal Ring**, select **Coincident** constraint and then select the bottom face of **Seal Ring** as the 1st entity and the face inside valve body as 2nd entity as shown in the picture below, after that, set **0mm** in offset and click ***Apply***.

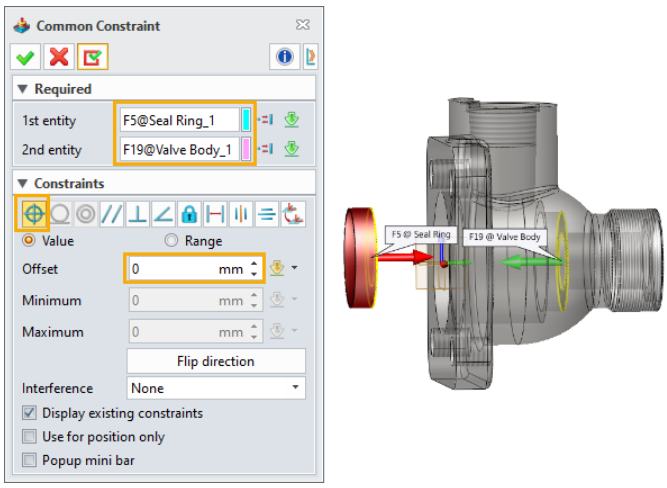



Figure 257 Add the coincident constraint

STEP 06 Select **Concentric** constraint and then select the cylinder profile of **Seal Ring** as the 1st entity and cylinder profile of inner **Valve Body** as 2nd entity, after that, tick ***Lock rotation*** and click ***Apply***.

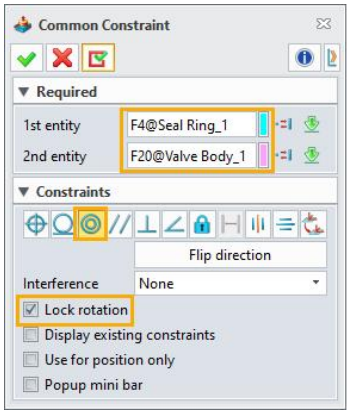
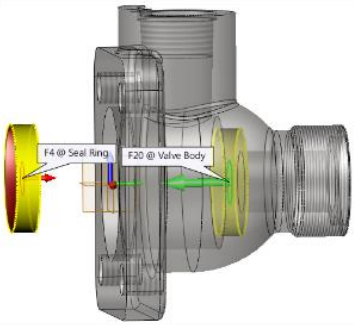



Figure 258 Add the concentric constraint



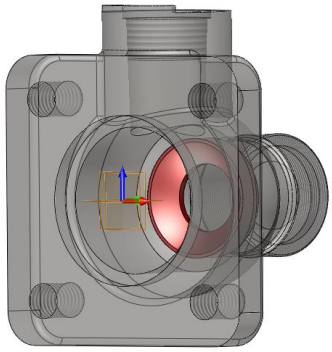



Figure 259 Well-defined seal ring

➢ **Part 4: Insert the component (Valve Core) and add the constraints.**

STEP 07 Insert **Valve Core**, then right-click the component, click ***Unblank External Datum*** from right menu to show the datum of the component. Afterwards, repeat the same step for **Valve Body**.

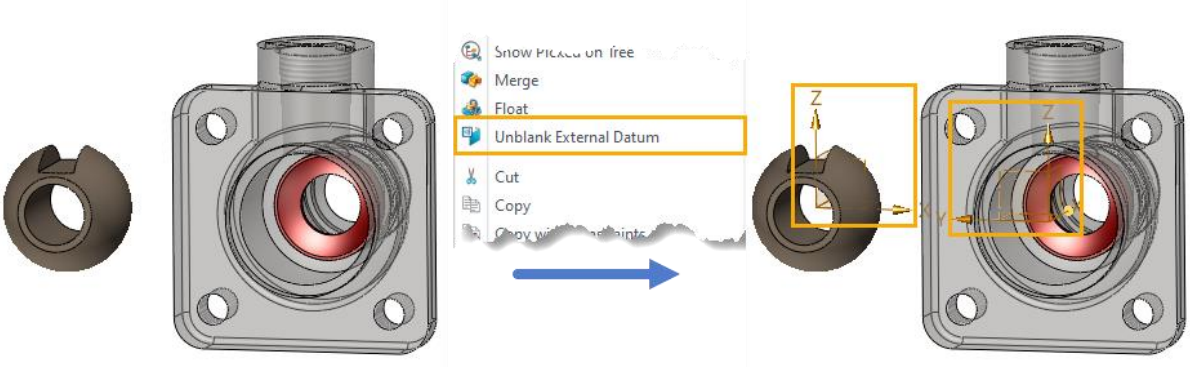



Figure 260 Unblank external datum

STEP 08 Select **Coincident** constraint and then select the ***XY*** datum of **Valve Core** as the 1st entity and ***XY*** datum of **Valve Body** as 2nd entity as shown in the picture below, after that, set **0mm** in offset and click ***Apply***.

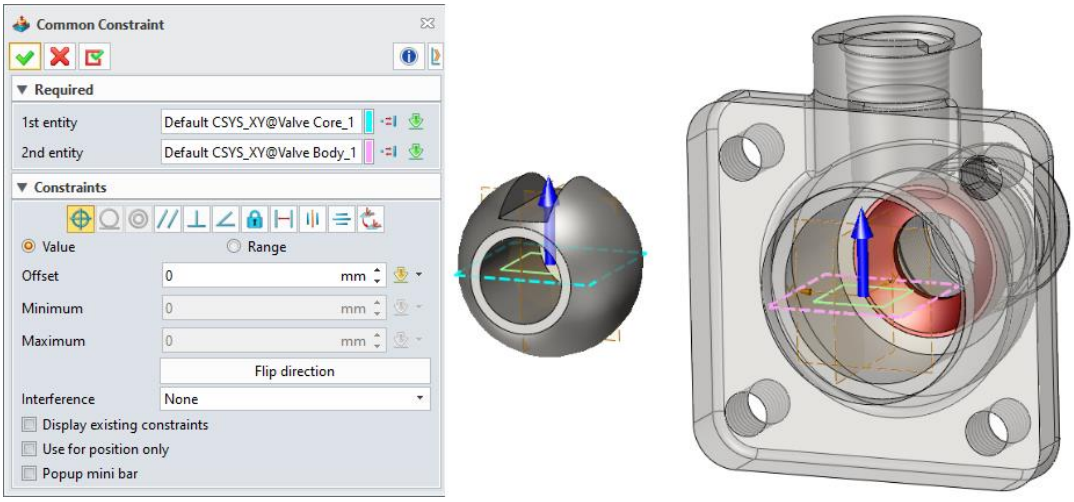



Figure 261 Add the coincident constraint

STEP 09 Select **Concentric** constraint and then select the sphere of **Valve Core** as the 1st entity and curved surface of **Seal Ring** as 2nd entity and click **Apply** as shown in the picture below.

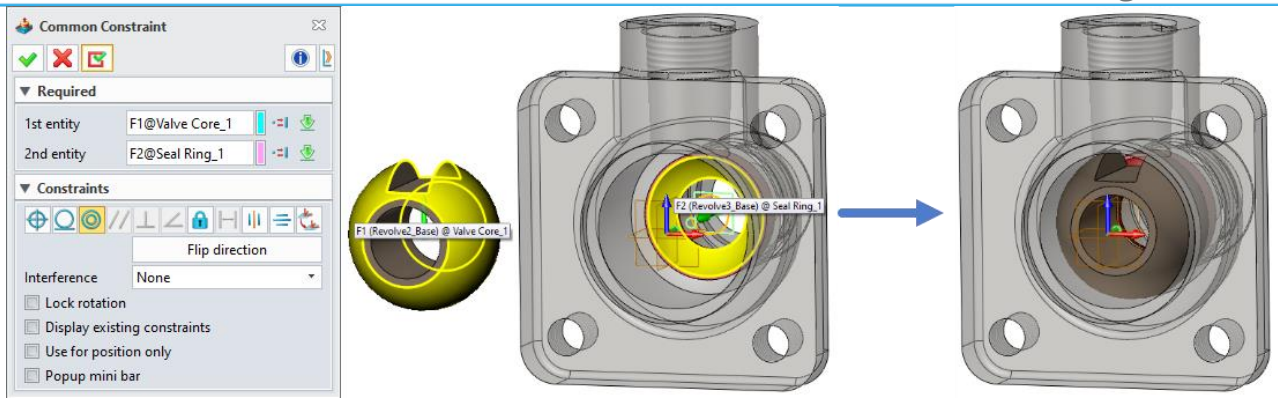



Figure 262 Add the concentric constraint

STEP 10 Select at **Angle** constraint, and then select the ***YZ*** datum of **Valve Core** as the 1st entity and ***XZ*** datum of **Valve Body** as 2nd entity, then set the angle range from **0 to 90 degree** and click **Apply** as shown in the picture below**.**

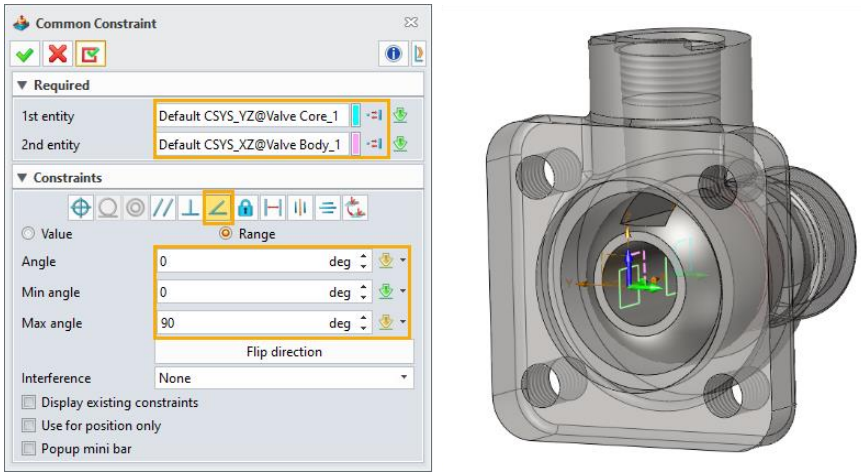



Figure 263 Add the angle constraint

The picture below shows the **Valve Core** rotate according to angle constraints.

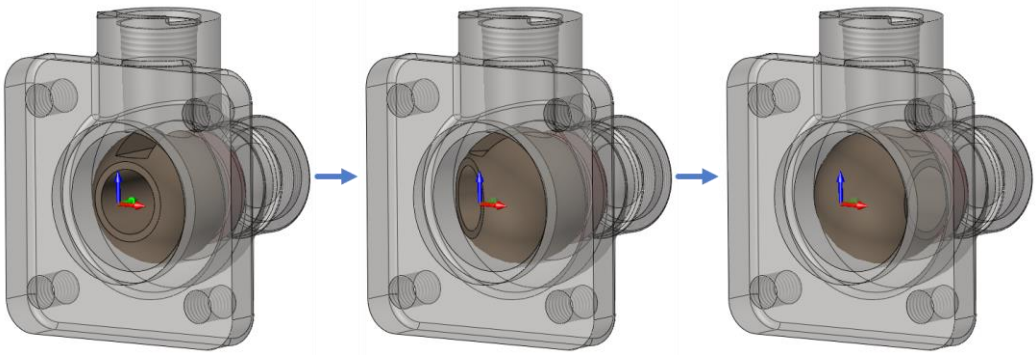


Figure 264 Rotate the Valve Core

➢ **Part 5: Insert the component (Seal ring) and add the constraints.**

STEP 11 Insert another **Seal Ring**, for more clear assembly work, you can blank the **Valve Body** from right menu.

Then select **Concentric** constraint and then select the sphere of **Seal Ring** as the 1st entity and curved surface of **Valve Core** as 2nd entity and click **OK,** as shown in the picture below.

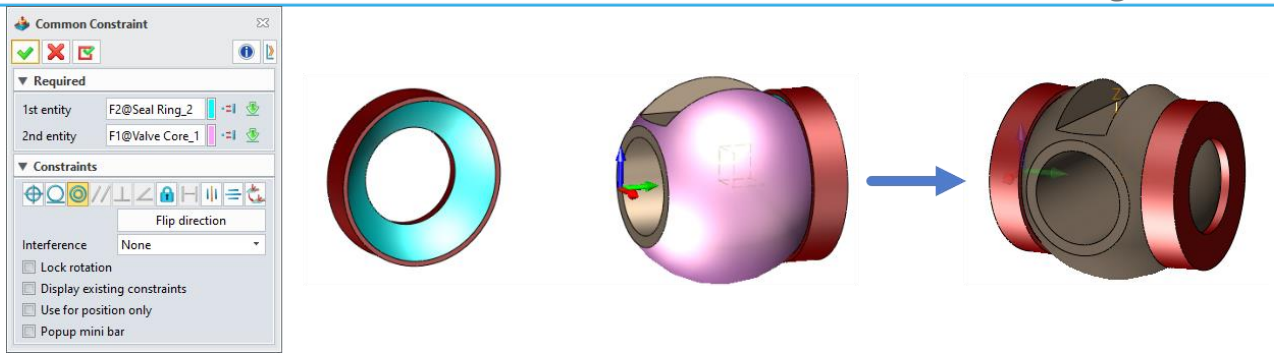



Figure 265 Add the first concentric constraint for seal ring

STEP 12 Unblank the **Valve Body** and select **Concentric** constraint and then select cylinder profile of **Seal Ring** as the 1st entity and cylinder profile of inner **Valve Body** as 2nd entity, after that, tick **Lock rotation** and click ***Apply*** as shown in the picture below.

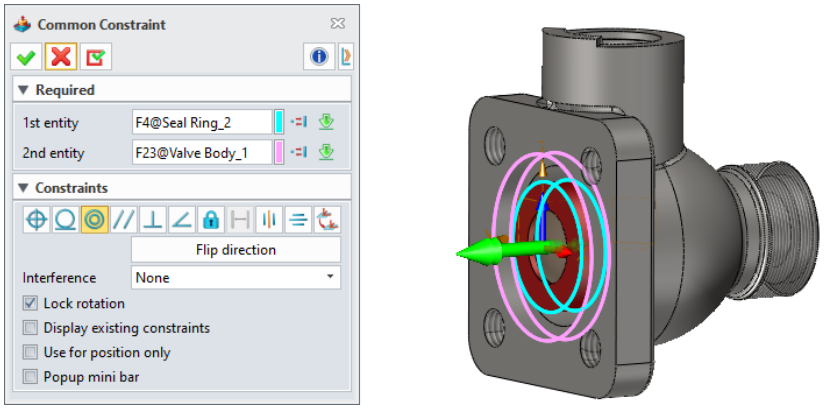



Figure 266 Add the second concentric constraint for seal ring

- **Part 6: Insert the Rod (Sub-Assembly) and add the constraints.**

STEP 13 Insert **Rod (Sub-Assembly)**, then select **Concentric** constraint and select the cylinder profile of **Rod** as the 1st entity and cylinder profile of inner **Valve Body** as 2nd entity and click ok as shown in the picture below.

STEP 14 Select **Coincident** constraint and select the bottom face of **Rod** as the 1st entity and top face of **Valve Core** as 2nd entity as shown in the picture below, after that, set **0mm** in offset and click ***Apply***.

STEP 15 Choose **Parallel** constraint and select the highlighted face of **Rod** and **Valve Core** as shown in the picture below and click **OK**.

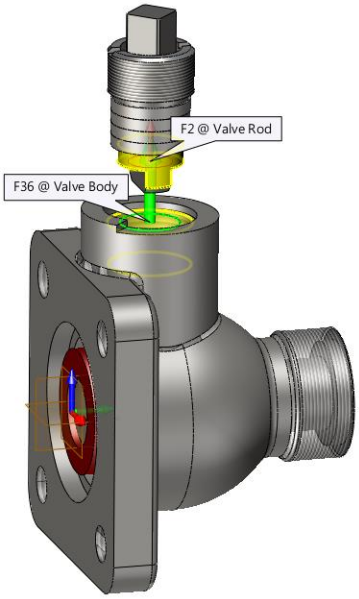



Figure 267 Concentric constraint

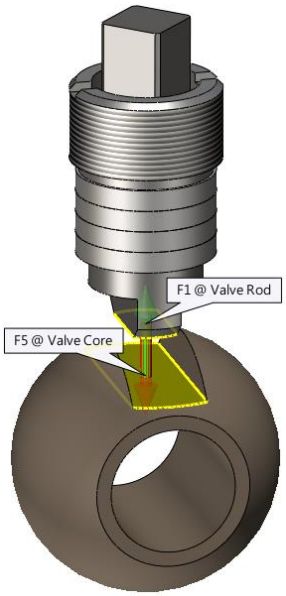



Figure 268 Coincident constraint

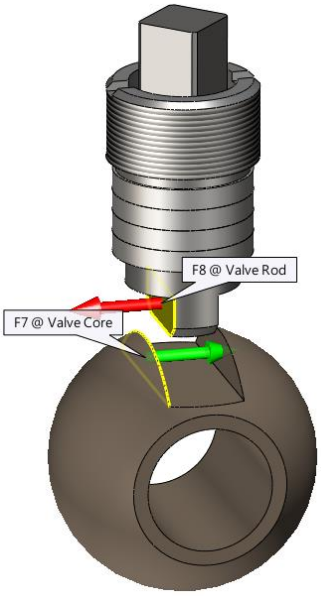
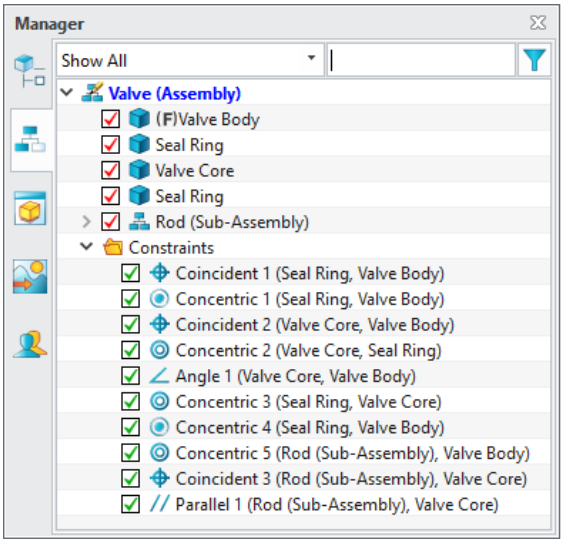



Figure 269 Parallel constraint

Now, the Rod (Sub-Assembly) is well constrained as shown in the assembly manager below.



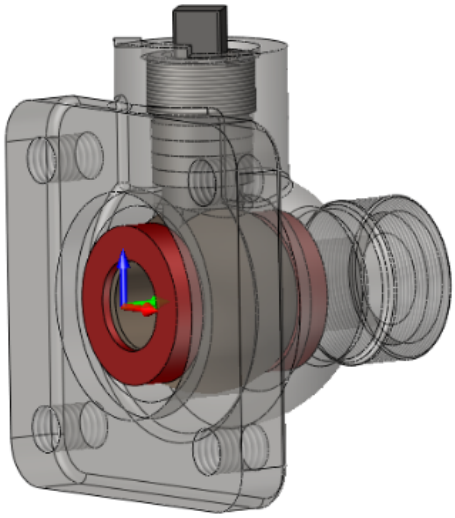



Figure 270 Constraint status in assembly manager

➢ **Part 7: Insert the component (Wrench and Adjusting Shim) and add the constraints.**

STEP 16 Insert **Wrench**, then choose **Coincident** constraint and select highlighted face of **Wrench** and **Valve Body** as shown in the picture below, after that, set **0mm** in offset and click **Apply**.

STEP 17 Choose **Concentric** constraint and select the cylinder profile of **Wrench** as the 1st entity and cylinder profile of **Valve Body** as 2nd entity and click ok as shown in the picture below.

STEP 18 Choose **Parallel** constraint and select the highlighted face of **Wrench** and **Valve Body** as shown in the picture below and click **OK**.

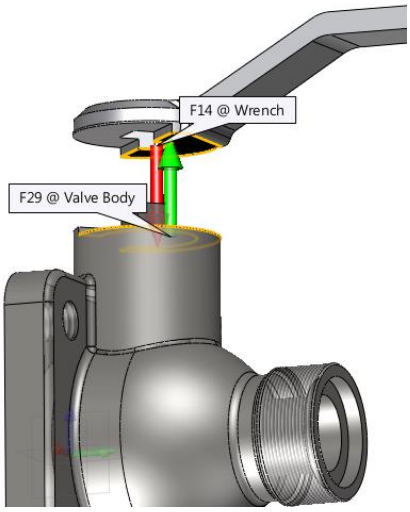



Figure 271 Coincident constraint

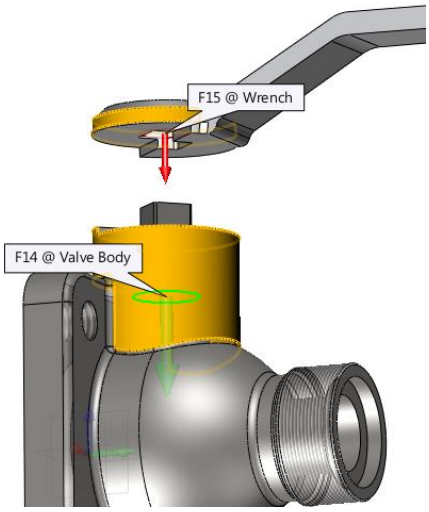



Figure 272 Concentric constraint

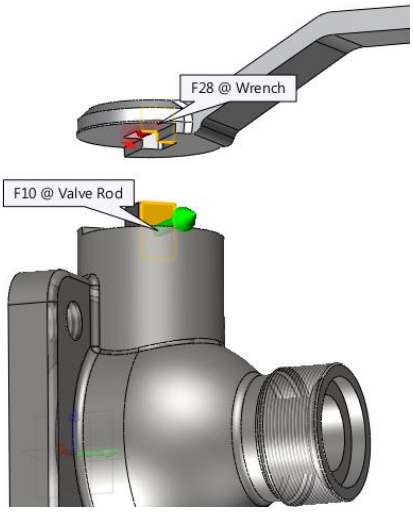



Figure 273 Parallel constraint

STEP 19 Insert **Adjusting Shim**, then choose **Coincident** constraint and select highlighted face of **Adjusting Shim** and **Valve Body** as shown in the picture below, after that, set **0mm** in offset and click **Apply**.

STEP 20 Choose **Concentric** constraint and select the cylinder profile of **Adjusting Shim** as the 1st entity and cylinder profile of inner **Valve Body** as 2nd entity as shown in the picture below. after that, tick **Lock rotation** and click **Apply**.

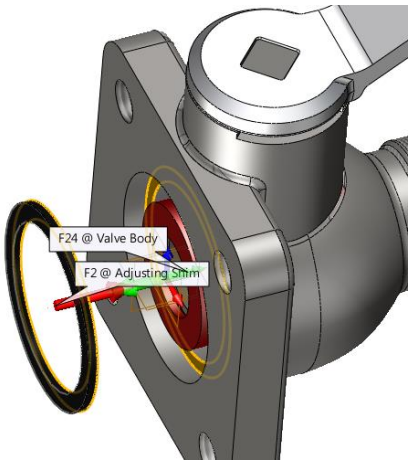



Figure 274 Step 19

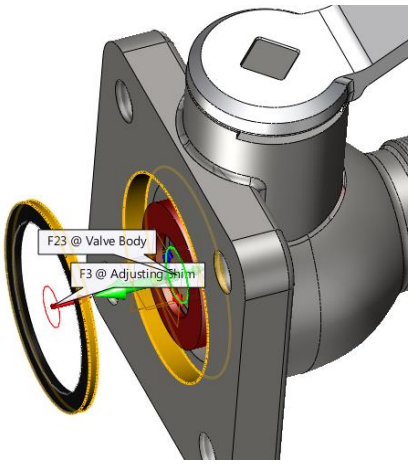
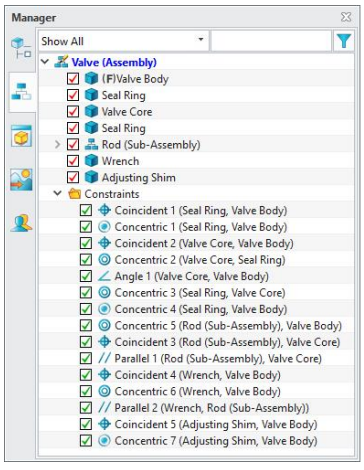



Figure 275 Step 20

Until now, we have completed most of the valve assembly, as can be seen in the picture below, all the components are well constrained.



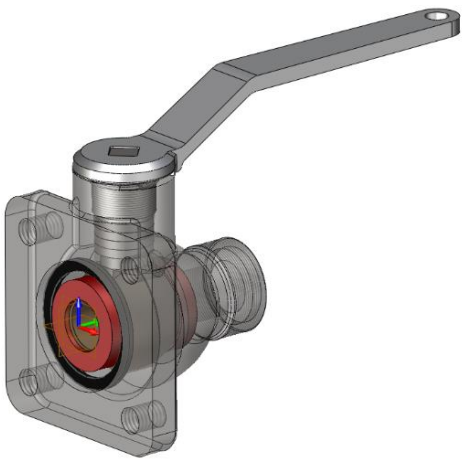



Figure 276 Valve assembly

In the next part, we will continue the reference design for valve cover.

### 5.4.3 In-Context Reference Design

For a whole valve assembly, there is a valve cover part not finished. Next, we will refer to the **Valve Body** part to design the cover part. Generally, this design is called in-context assembly design. And this cover part refers to the valve body. The reference design has been introduced in **Chapter 5.3.9**.

Now, let us follow these steps to do this cover part modeling in the assembly level.

➢ **Part1: Directly create a new component in an assembly**

STEP 01 Open **Valve (Assembly)**, click ***Insert*** command, directly input create a new component **Valve Cover** as a new component name, then click **OK**. Then the new component **Valve Cover** is created and activated**.**

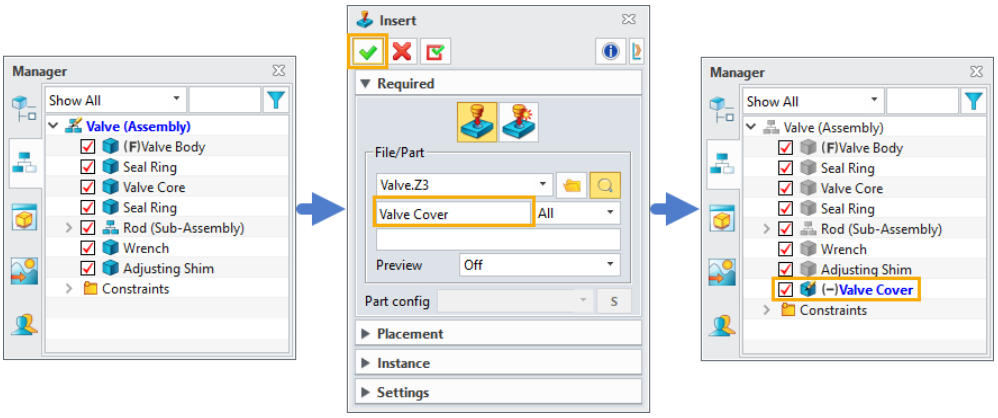



Figure 277 Create the Valve Cover in Insert panel

➢ **Part 2: Create the reference geometry**

STEP 02 Select ***Reference*** command from ***Assembly*** ribbon tab and select the **Face** reference, then pick the face from **Valve Body** as shown below and click **Associative copy** and **Apply**.

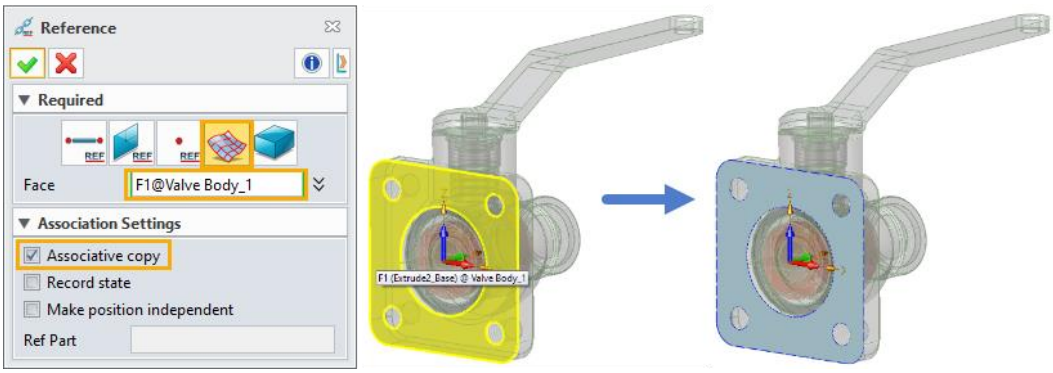



Figure 278 Create face reference

➢ **Part3: Solid Modeling**

STEP 03 Select ***Extrude*** command from ***Shape*** ribbon tab and select the reference face as **Profile P**, then extrude it from **0.5mm** to **14.5mm**, as shown below.

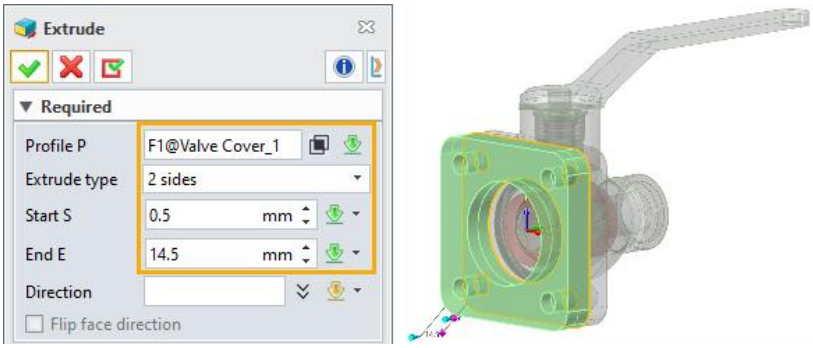



Figure 279 Extrude the reference face

After creating the reference solid, let us create the sketch of the **Valve Cover** revolve part. If you want working area to be clear, you can blank other components by ***Show target*** command from ***Document Aware Toolbars***.

STEP 04 Select ***Sketch*** command from ***Shape*** ribbon tab, then select **YZ** datum of **Valve Cover** as the sketch plane and **Z** axis as up orientation as shown below.

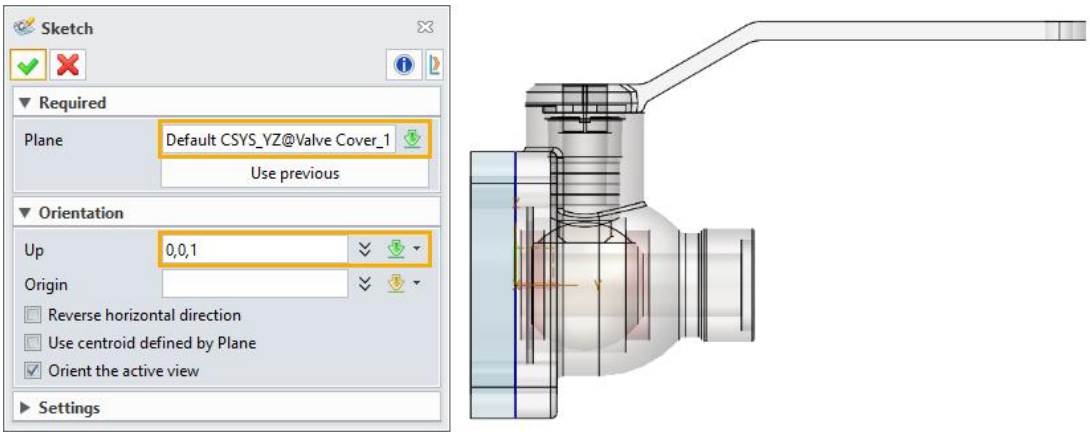



Figure 280 Create a sketch in Valve Cover

STEP 05 Draw the sketch with the shape and dimension as shown in the picture below and then back to modeling level.

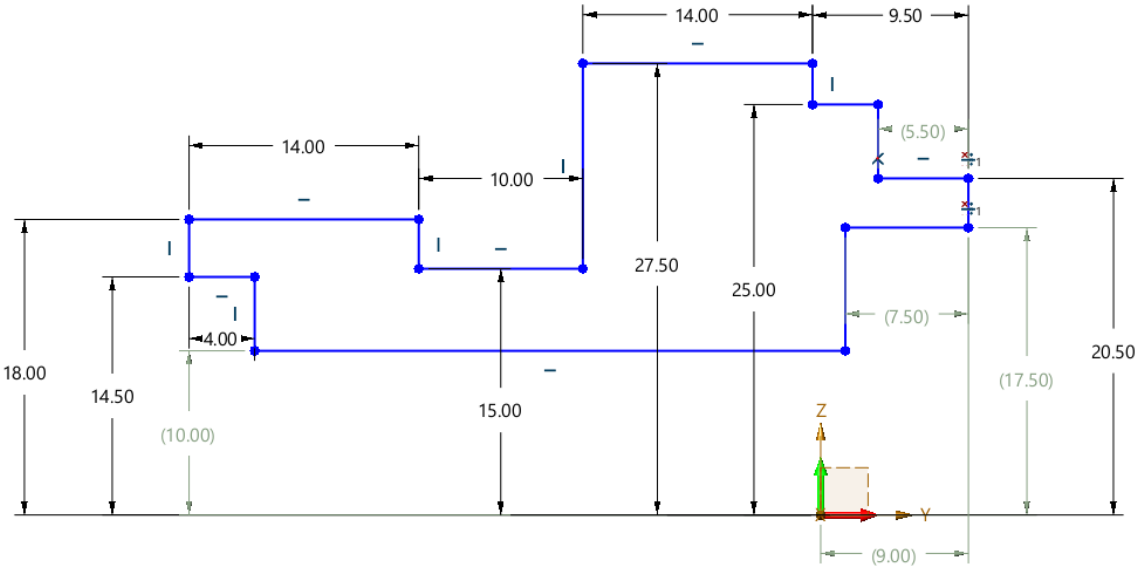



Figure 281 Create a sketch on Valve Cover

STEP 06 Select ***Revolve*** command from ***Shape*** ribbon tab and set the parameter as shown below, then select **Add** in **Boolean** and select the extruded solid which was created in step 3 as the **Boolean shapes**.

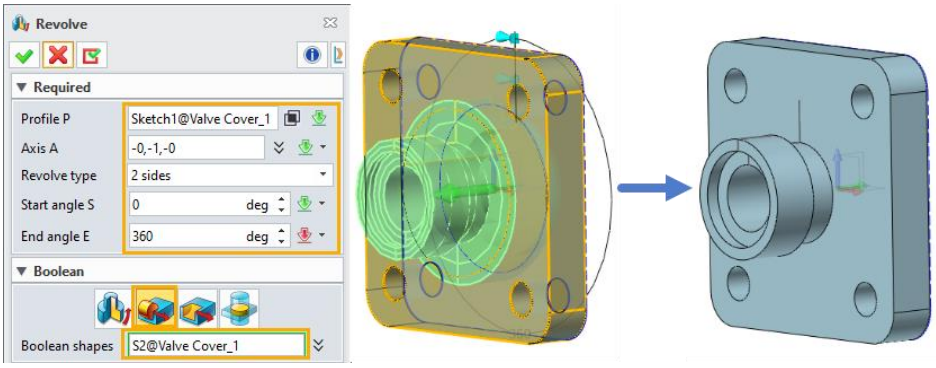



Figure 282 Revolve the sketch

If you want to view the whole assembly, click ***Show All*** from ***Document Aware Toolbars.***

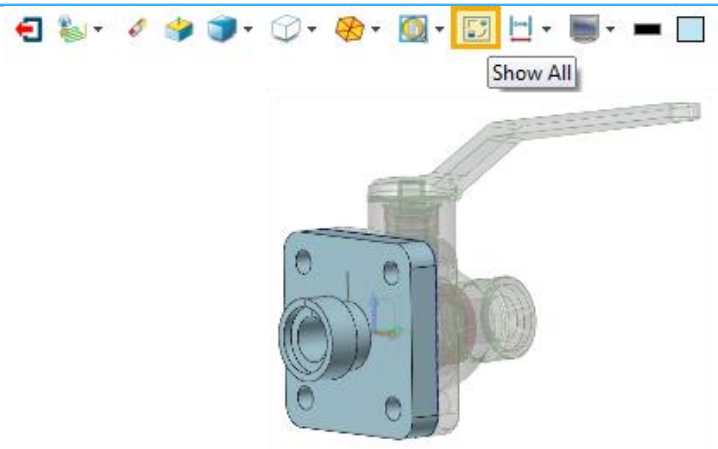



Figure 283 Show all

STEP 07 Select ***Fillet*** command from ***Shape*** ribbon tab and add the fillet for the edge as shown in the picture below with **2mm** radius.

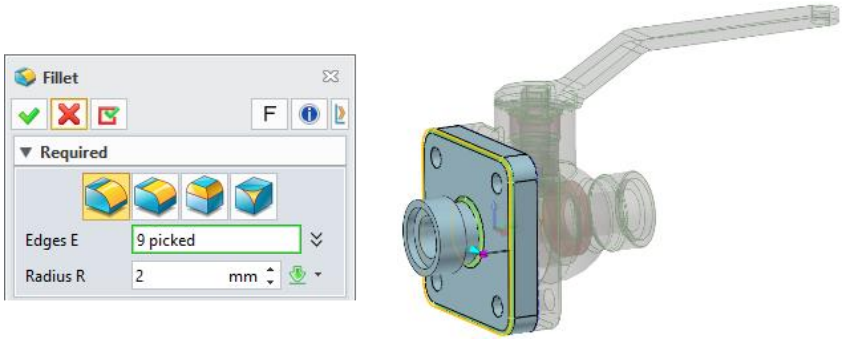



Figure 284 Add the fillet

STEP 08 Select ***Chamfer*** command from ***Shape*** ribbon tab and add the chamfer for the edge as shown in the picture below with **1mm** setback.

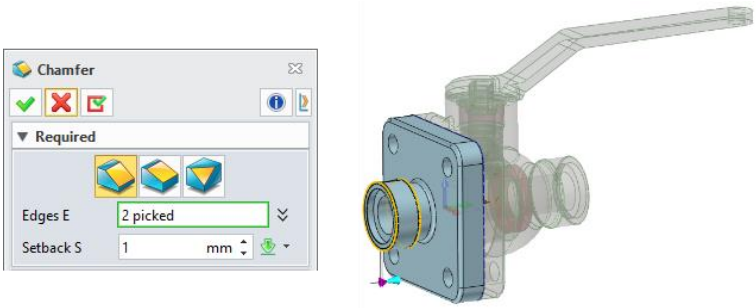



Figure 285 Add the chamfer

Now, we have finished the valve cover modeling. Double click the top assembly to view the whole product assembly. Next, we will insert the standard bolts and nuts for the valve assembly.

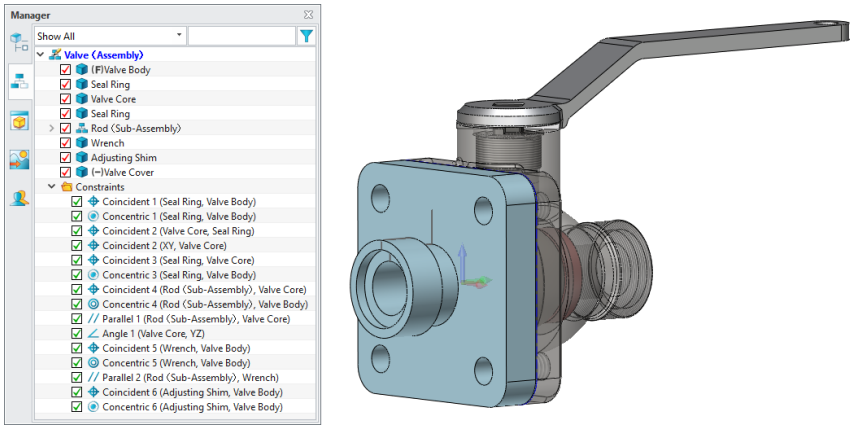



Figure 286 Top assembly

### 5.4.4 Insert Standard Parts

In this part, we will insert the **ISO** standard **Bolts** and **Nuts** from ZW3D **Reuse Library** into the valve assembly.

STEP 01 In **Valve (Assembly)**, Open **Reuse Library** from ZW3D, then select **Hexagon Head Bolts** from **ZW3D Standards Parts** -> select **Hex bolts with flange ISO15071.Z3** from file list -> select **10mm** in diameter -> select **Auto hole match** in placement of insert -> Tick **Constrain component**, as shown in the picture below.

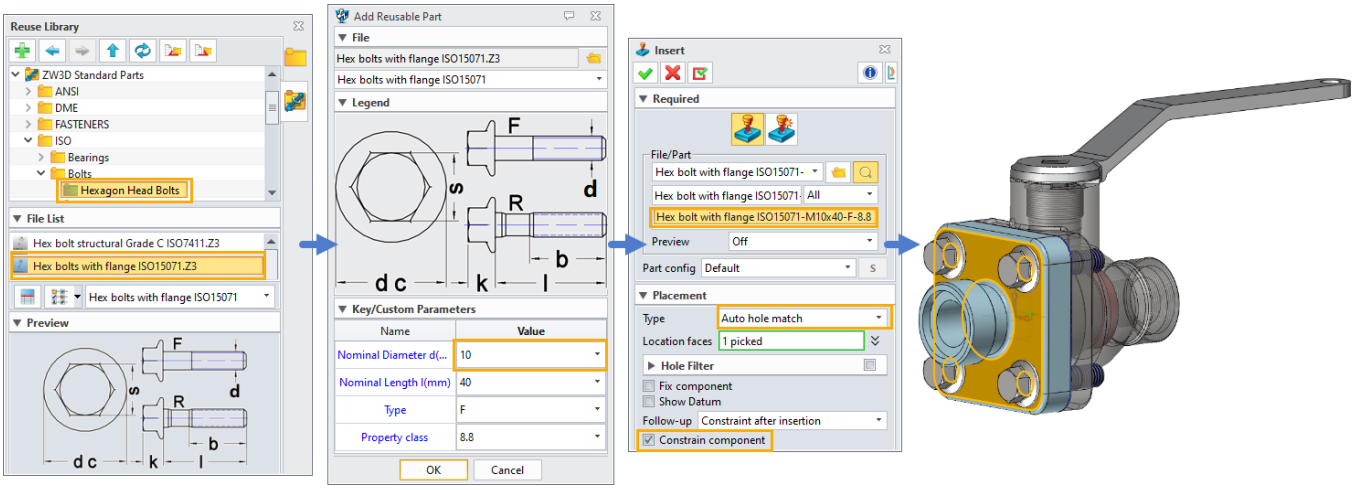



Figure 287 Insert hexagon head bolts from reuse library

STEP 02 Use the same way to insert the Iso standard **Hexagon Nuts(10mm)** as shown in the picture below.

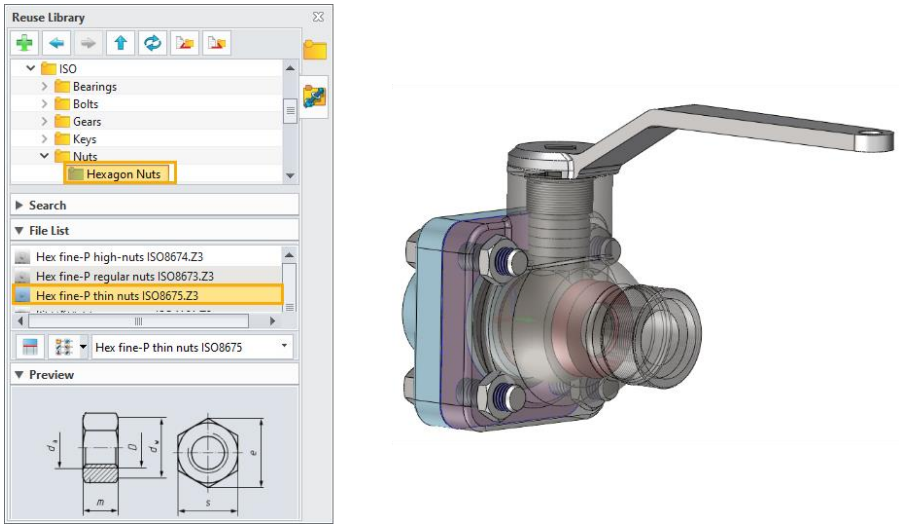



Figure 288 Insert hexagon nuts bolts from reuse library

Now, we have finished the standard parts inserting.

### 5.4.5 Verify the Correction of the Whole Assembly

When checking the whole assembly, we can find there is some interference between the standard parts and valve body as shown in the picture below.

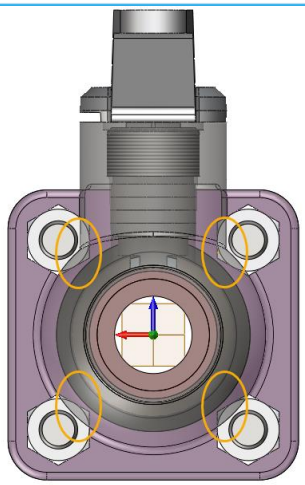


Figure 289 Interference between standard parts and valve body

Both the overall dimension of valve body and valve cover need to be modified. Since the associative reference design is made between these two components. Therefore, we just need to change the dimension of valve body, the valve cover component will be updated

STEP 01 Double click the **Valve Body** to activate it. Go to history manager, then double click the expression **Base Length** and change the value of **Base Length** from **75mm** to **85mm.**

STEP 02 After parameter modification, the model part is marked as **Outdated.** Then right-click the blank area of history manager, select ***Regen History*** to regen the model. Or directly select ***Regen*** command from Title bar.

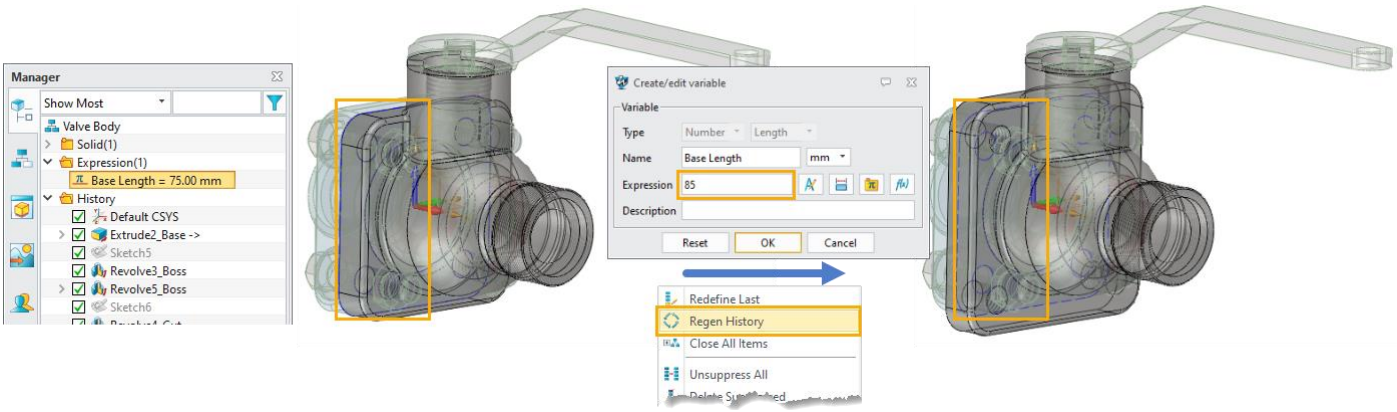



Figure 290 Change the base length of valve body

STEP 03 Back to top assembly **Valve (Assembly)** and then regen all components in right menu of assembly manager, then the **Valve Cover** model and the position of standard parts will be updated automatically.

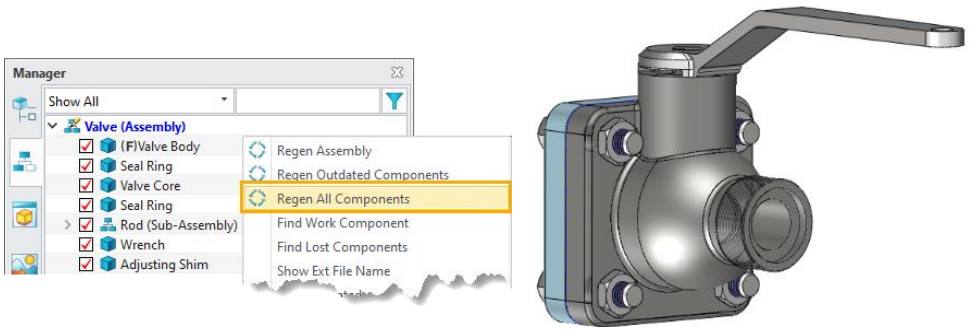



Figure 291 Regen all components

# 6 2D Drawing

The 2D drawing is used to show the engineering information of the object, including part/assembly view, dimensions, symbols and annotations, text, table and so on. In the process of product design and manufacturing, 2D Drawing is an important and widely-used documents even the 3D model in ZW3D are sufficiently intuitive and visual. The picture shown below is an example of 2D drawing in ZW3D.

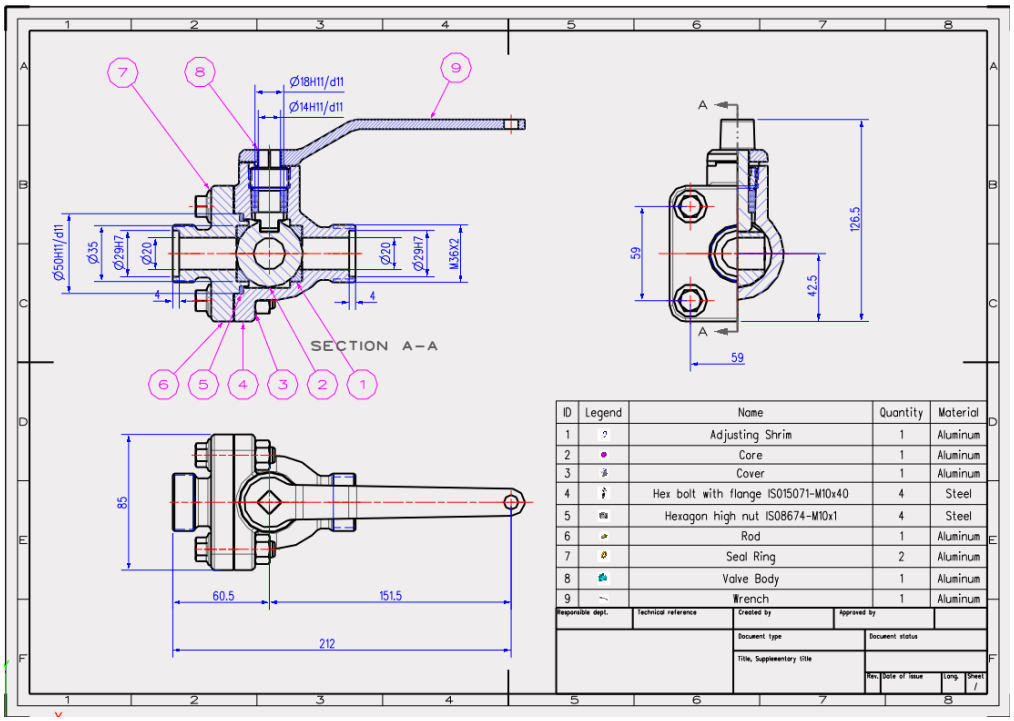



Figure 292 2D drawing in ZW3D

After the 3D model is produced in ZW3D, the associative 2D drawing can be created conveniently, and it can be changed automatically and simultaneously according to the modifications of the 3D model.

## 6.1 Main Elements of 2D Drawing

Generally, the 2D sheet of a part consists of three main parts. Below are the explanations of them.

**View:** Include Standard view (Top, Front, Right, Left, Bottom, Back and Isometric view),

Projection view, Section view, Detail view and so on.

**Dimension:** Include dimension (Shape dimension & Position dimension), Tolerance (Dimension tolerances,

Form tolerances & Position tolerances), Datum symbol, Surface finish symbol and Text.

**Sheet format:** Include sheet border, title block and so on.

As for assembly, 2D sheet includes various views, assembly dimension, fit dimension, BOM table and so on.

## 6.2 Create a New 2D Drawing in ZW3D

There are 2 commonly used methods to create a new 2D sheet in ZW3D.

**Method 1:** In the modeling environment, insert a new 2D sheet by clicking ***2D Sheet*** from ***Document Aware Toolbars*** or right click the working area and then select a template to create a new 2D drawing. Meanwhile, the Standard view command will be activated automatically.

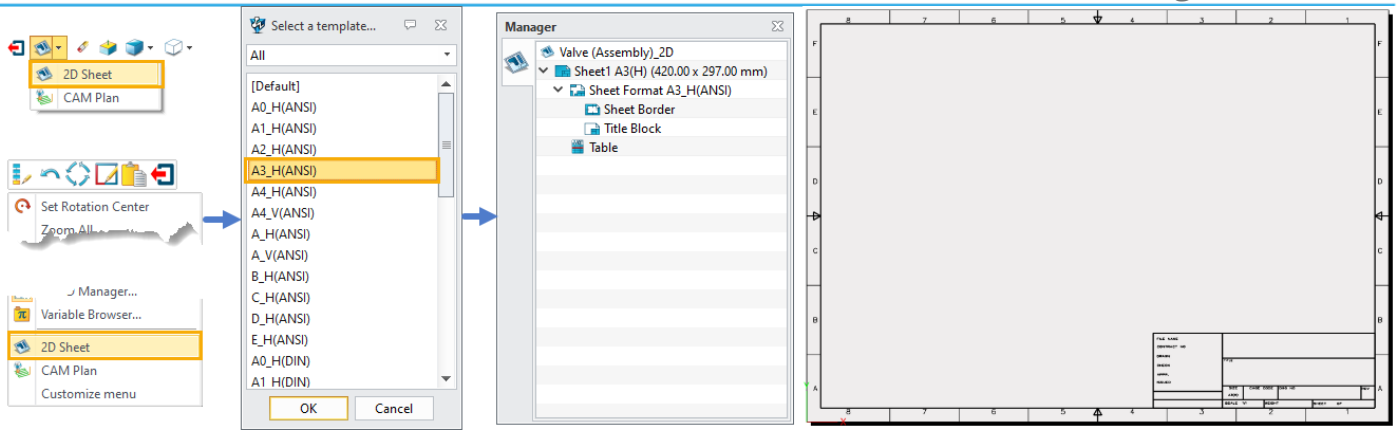



Figure 293 Create a new 2D sheet

**Method 2:** Click ***+*** button (***Add new file*** command) from file title tab, then select the ***Drawing Sheet*** in file type and select Template, input the name of drawing, and click ***OK*** button. After that, the new 2D drawing is created.

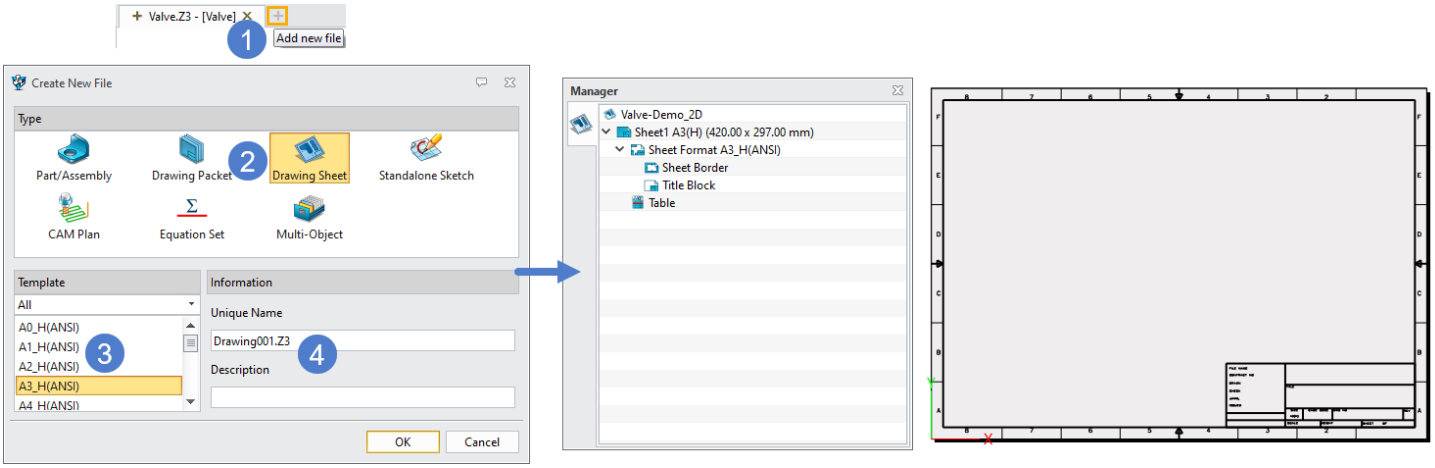



Figure 294 Create a new 2D sheet

## 6.3 General Settings for 2D drawing

This section will introduce some commonly used settings for 2D drawing.

***Top right* -> Configuration**

In the Configuration dialogue box, some default parameters of 2D drawing can be modified. See the image below.

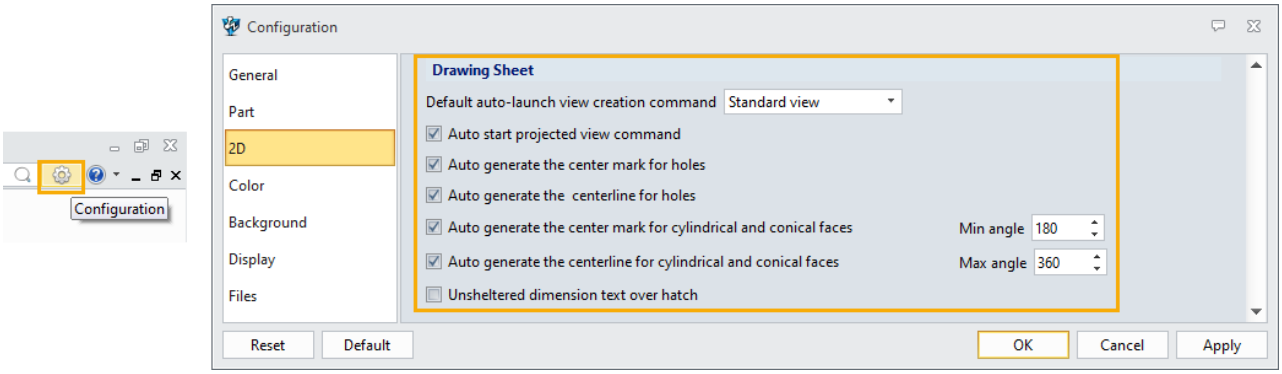



Figure 295 Configuration

***Tools*** ribbon tab ***-> Settings*** Panel ***-> Preferences -> Drawing Settings***

With this command, you can change the drawing settings including Units, Mass units, Grid spacing, Type of projection and Projection tolerance.

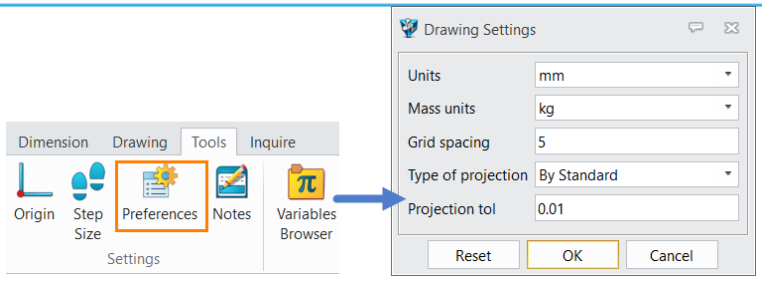



Figure 296 Preferences

***Tools*** ribbon tab ***-> Attributes*** Panel ***-> Style Manager***

With Style Manager, you can customize the drawing as the picture below shows.

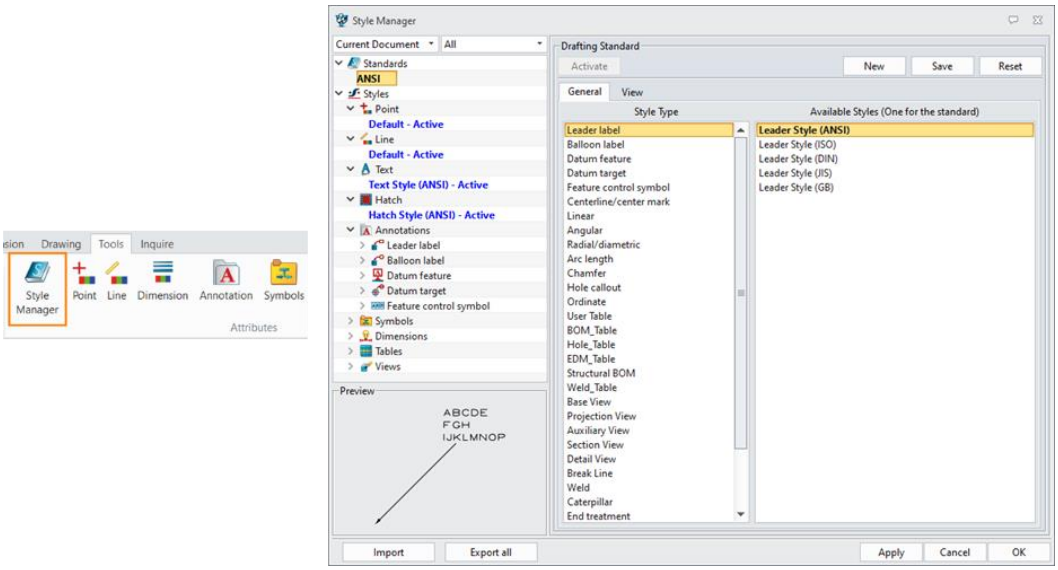



Figure 297 Style manager

***Sheet Manager -> Sheet1 -> Right-click menu -> Attributes***

Sheet Attributes is used for setting sheet name, scale, paper color and other basic attributes for the selected sheet.

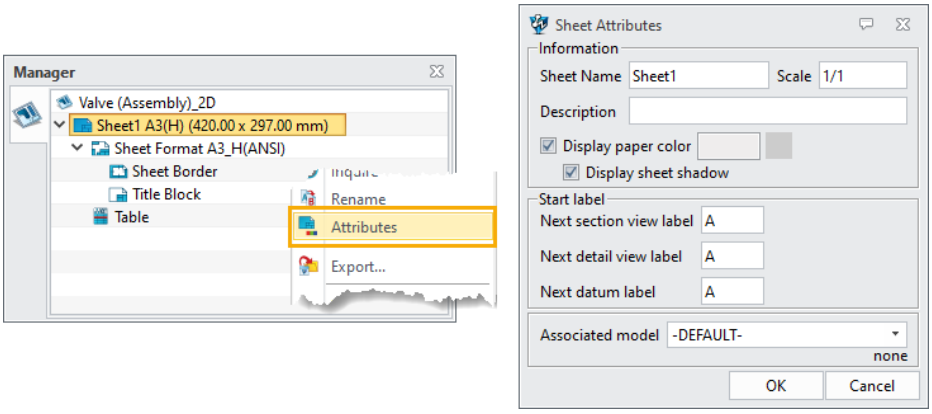



Figure 298 Sheet attributes

**Sheet Manager -> Sheet Format -> Right-click menu -> Sheet Format attributes**

With Sheet Format Attributes, you can redefine or customize the sheet format according to different requirements.

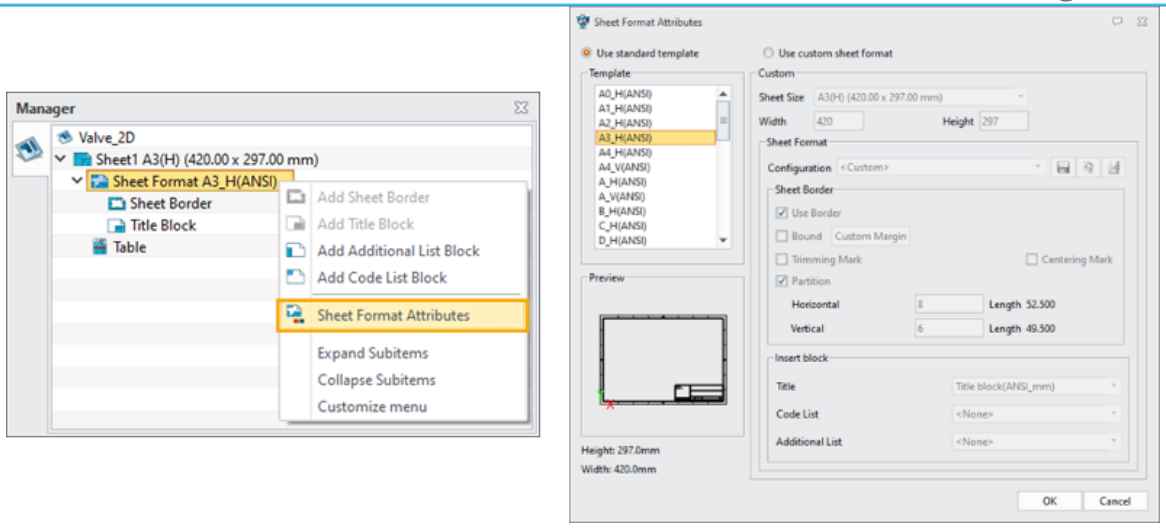



Figure 299 Sheet format attributes

## 6.4 Engineering Drawing

### 6.4.1 Create Standard View and Projection View

As mentioned in **Chapter 6.1**, the view includes standard view, projection view, section view, detail view and so on. This section will introduce how to create the standard view.

After finish creating the new 2D drawing in ZW3D, the standard view command will be activated automatically, you can either select the ***Standard*** command from ***Layout*** ribbon tab to create a standard view from a 3D part.

Before creating the standard view, select the part from the **file/part**, then select the view from the drop-down list and set other parameter such as scale.

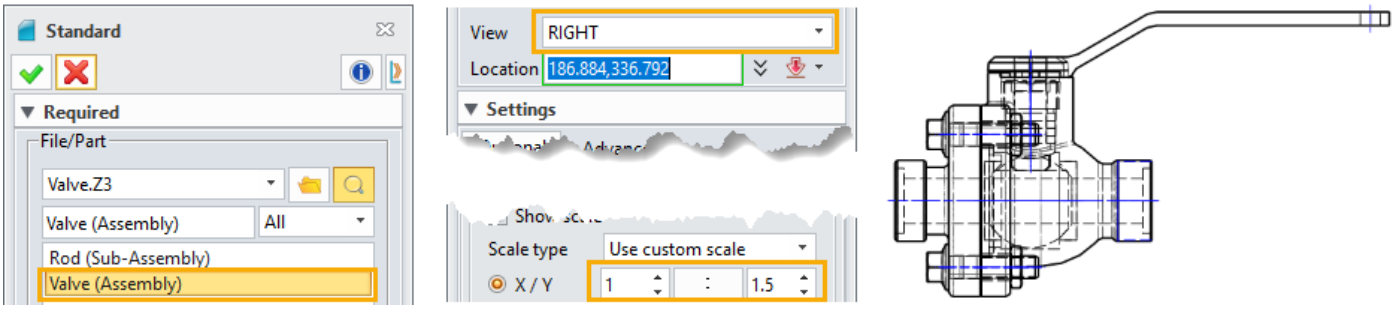



Figure 300 Create the standard view

After creating the standard view, we can select the **Projection** command to create a view projected from the existing standard view.

Before creating the projection view, select the base view and define the location of the projection view and set other attributes.

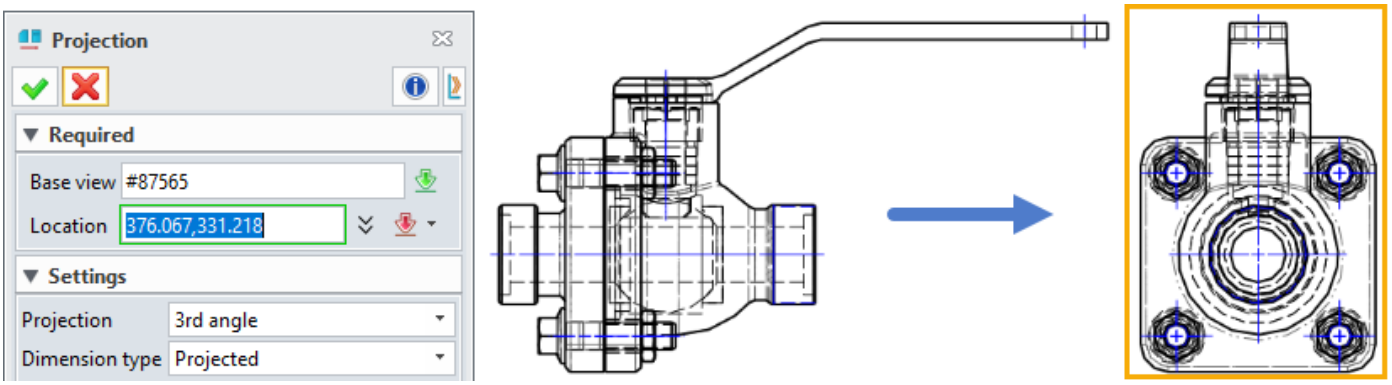



Figure 301 Create the projection view

Besides, you can also use ***Layout*** command which can be accessed from ***Layout*** ribbon tab to create to view by define the layout view and other parameter, such as view label, line attribute.

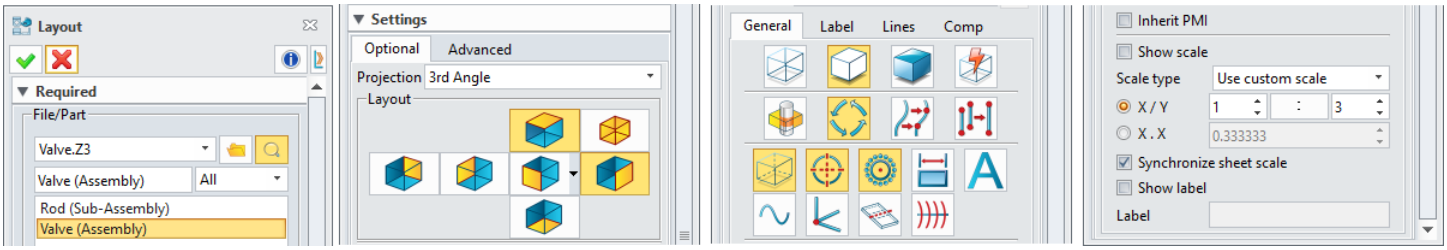



Figure 302 Create the view from layout

### 6.4.2 Modify the view attributes

After creating the view, you can redefine the view attribute by 2 different methods.

**Method 1**: Right menu

Directly right-click the view or right-click the view name from sheet manager, then select the attribute command to modify the view attributes.

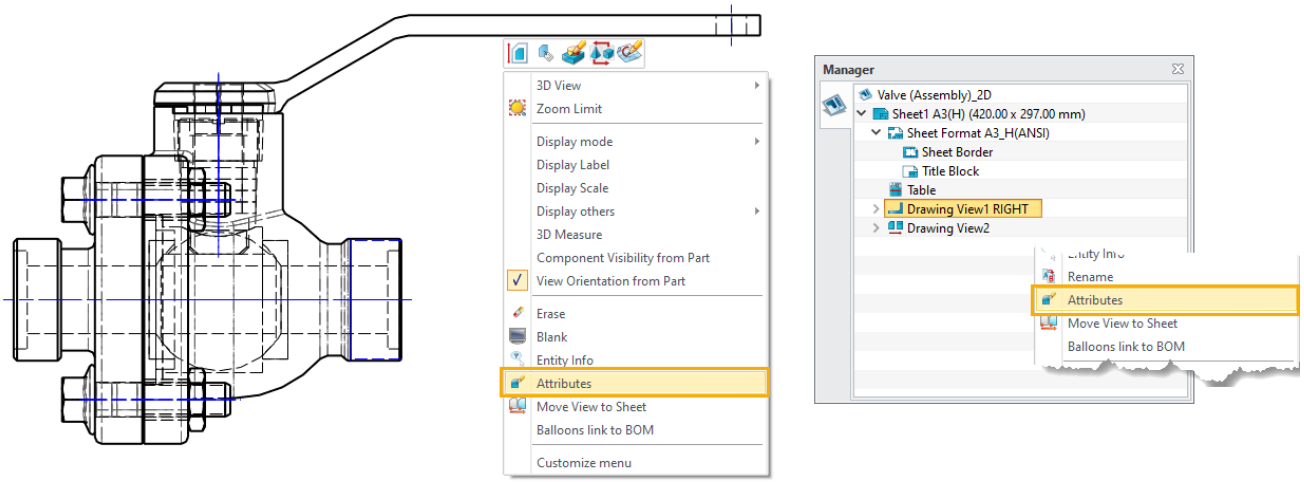



Figure 303 View attributes

**Method 2**: ***View Attributes*** from ***Layout*** ribbon tab.

Select view attributes command and choose the view, then middle click to confirm.

In view attributes, you can change the view parameter as below.

1. Show hidden lines/center lines/threads.
2. Show dimensions/text/3D curves/3D zero point from part.
3. Show available PMI on related views.
4. Show scale and label.
5. Set line attributes
6. Set component visibility

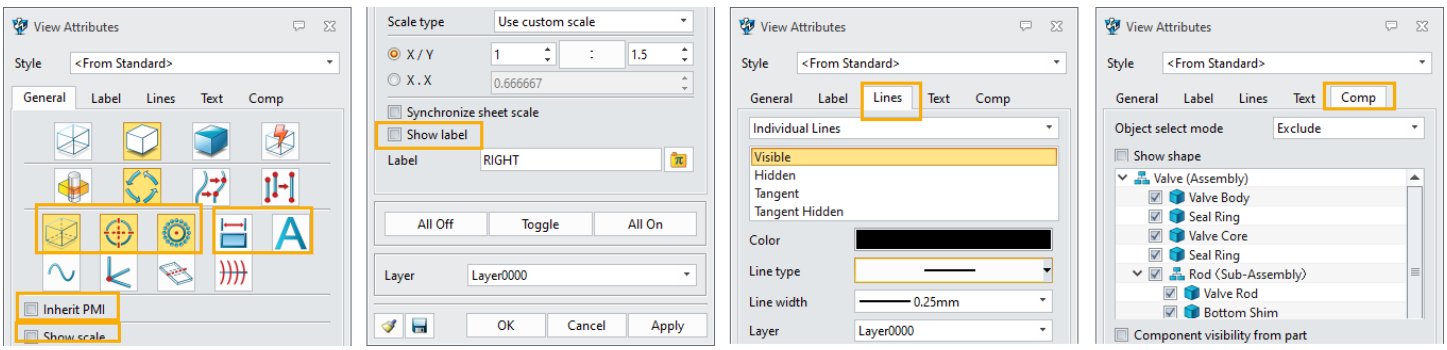



Figure 304 View attributes

### 6.4.3 Create Section View

In ZW3D, several different section view can be created, such as full section view, aligned section view, and isometric section view.

**Full section view:**

With full section command, you can create variety of section view of a 3D layout view by define the section position. If two points are picked and full section line through the model, the full section view is created. If more points are picked, the stepped section view could be created. The picture below shows the two different full section view.

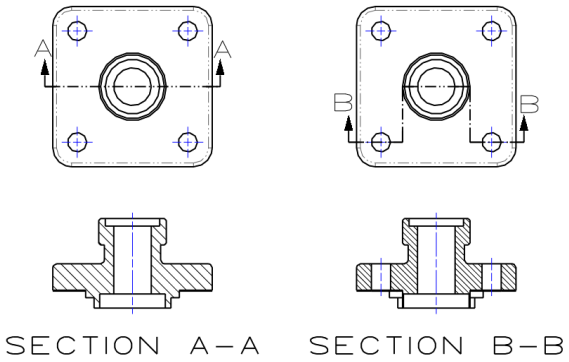



Figure 305 Full section view

**Aligned section view:**

With this command, you can create the section view in two directions, the picture below is an example of aligned section view.

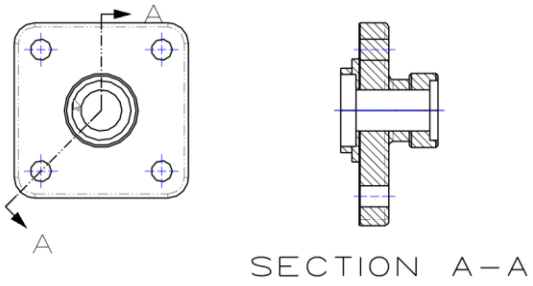



Figure 306 Aligned section view

**Isometric section view:**

In this command, the section line should be defined in the part by **Named Section** command in **Wireframe** ribbon tab, and the sketch line should be open. The picture below is an example of isometric section view.

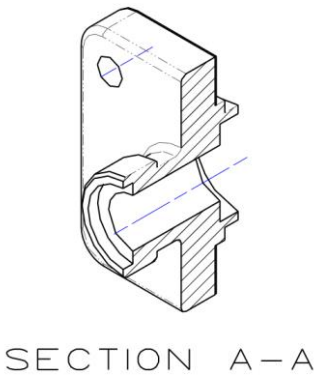



Figure 307 Isometric section view

For more other commands in view, please refer to **From entry to master tutorial**.

### 6.4.4 Edit the section view attribute

After creating the view, you can redefine the section view by right menu of section view or select the **Redefine Section** command from layout ribbon tab.

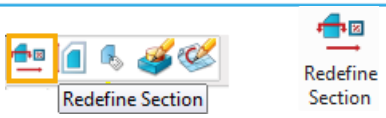



Figure 308 Redefine section

If you want to edit the section line which is created by **Full Section** command, the section line can be edited after creating the view.

You can select the **Insert Step** option from right menu of section line and then point to insert the step by dragging and dropping the insert point to the suitable position to get a new section line.

Or if you want to reverse the section line direction, you could also select the **Reverse Direction** option from right menu of section line.

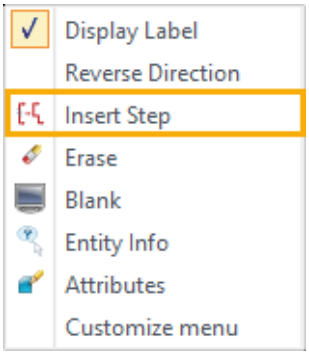



Figure 309 Insert step

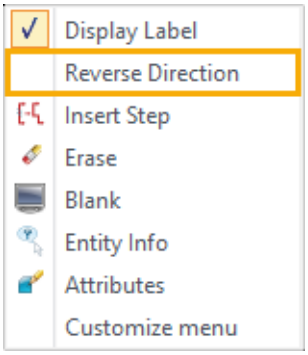



Figure 310 Reverse direction

### 6.4.5 Create the dimension

After finish creating the view, the next step is adding the dimensions.

In ZW3D, you can use quick dimension tool or other dimension tools to create the required dimensions.

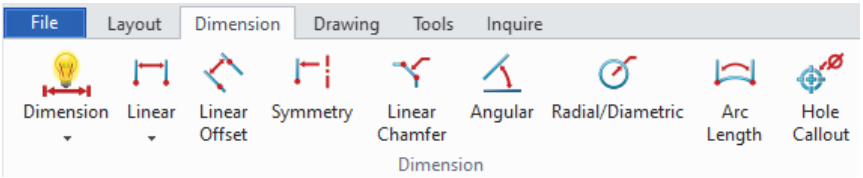



Figure 311 Dimension ribbon tab

The pictures below show the example of the adding the common dimension.

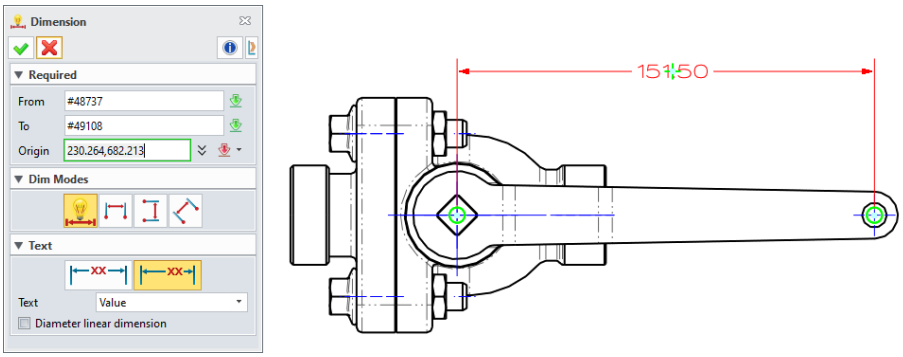



Figure 312 Quick dimension

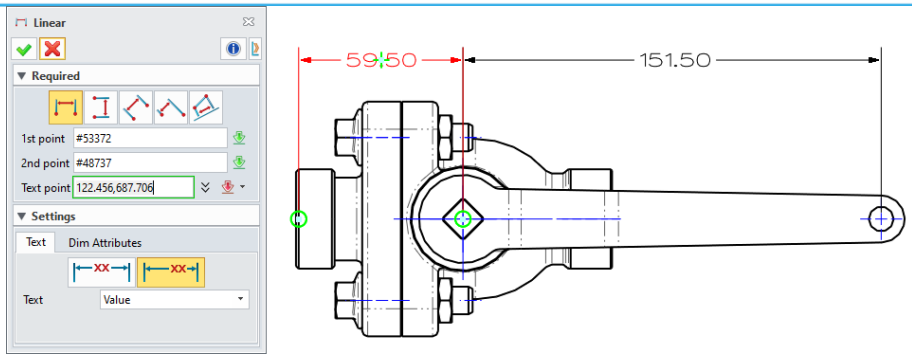



Figure 313 Linear dimension

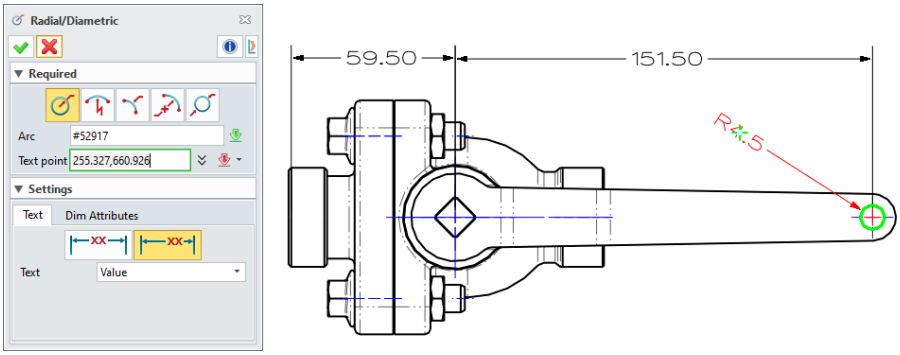



Figure 314 Radial/Diametric dimension

Moreover, for the hole dimension, you can use the **Hole Callout** command to create one or more hole callout dimensions, first select the layout view and then select the holes to add callouts.

The picture below shows an example of hole callout dimension.

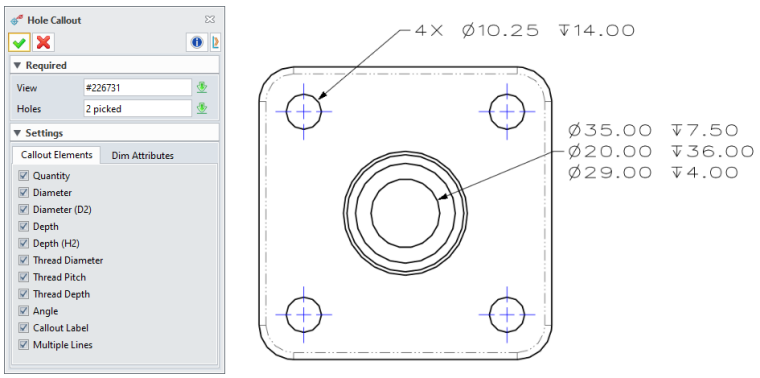



Figure 315 Hole Callout dimension

### 6.4.6 Add the Tolerance

There are several methods to add the tolerance, you can either use the **Modify Tolerance** command from ***Dimension*** ribbon tab or right click the dimension and then select **Modify Tolerance** command to edit the tolerance.

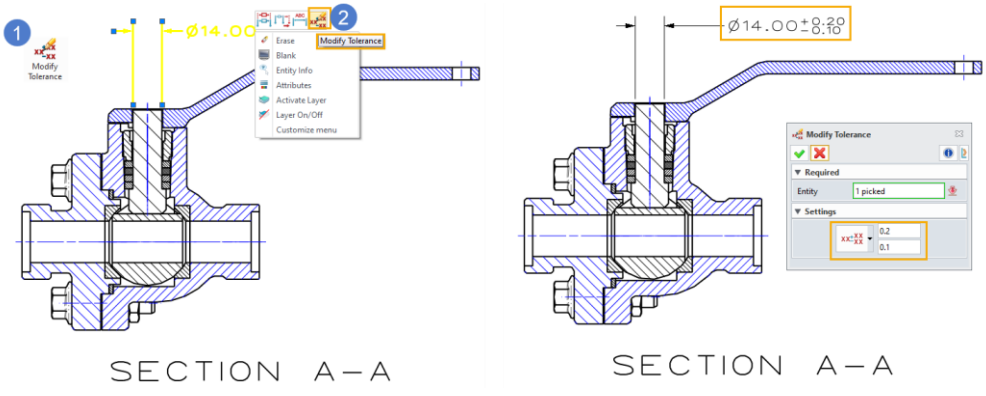



Figure 316 Modify tolerance

Besides, you can also us the right click the blank area in the ribbon toolbar to open quick dim tool the modify the tolerance, the picture below shows the operation steps of this method.

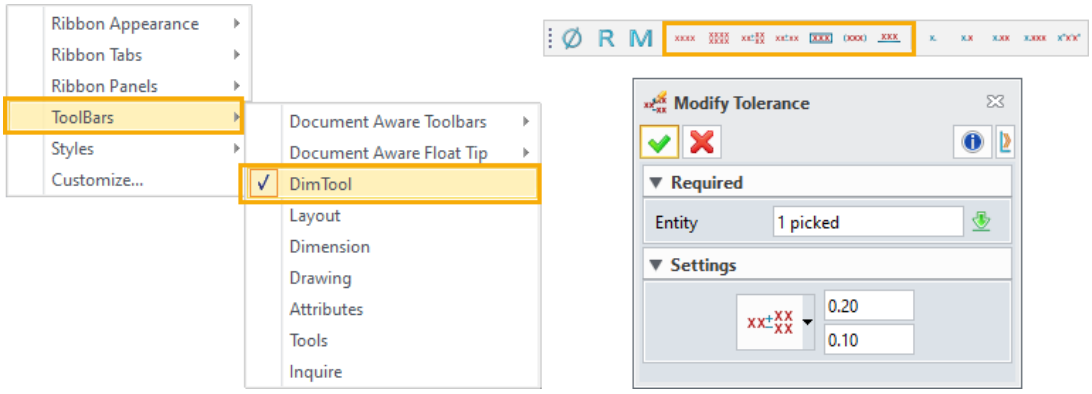



Figure 317 Modify tolerance by DimTool

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

If you want to add tolerance zone, you can select **Tolerance zone** type and open tolerance inquiry to select suitable tolerance zone as shown in the picture below.

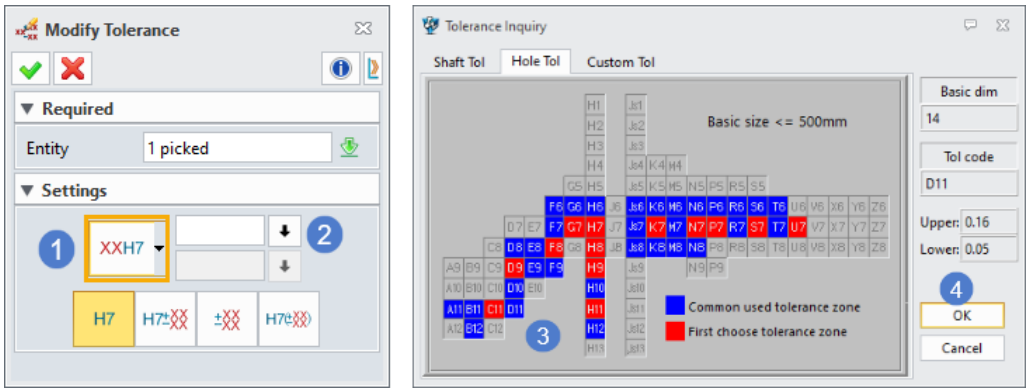



Figure 318 Tolerance inquiry

### 6.4.7 Annotation and symbol

The picture below shows annotation and symbol tools in ZW3D.

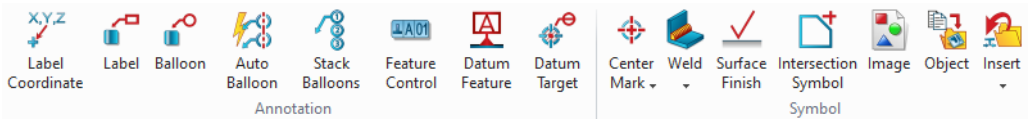



Figure 319 Annotation and symbol tools in ZW3D

In this section, we will introduce the most common used tools of annotation and symbol.

**Center Mark/Line:**

With ***Center Mark*** command, you can create a center mark at an arc or circle.

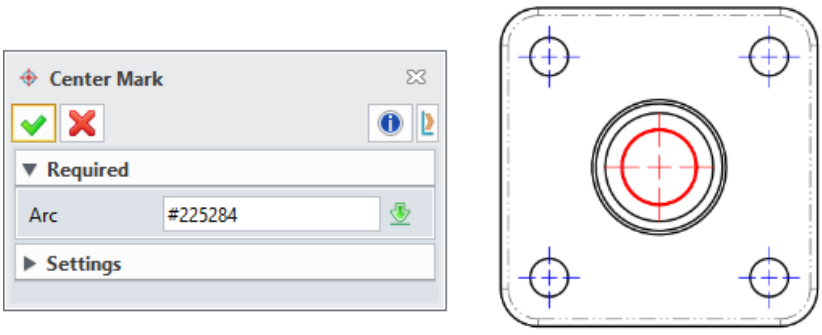



Figure 320 Center mark

With ***Center Line*** command, you can create a center line mark between lines, arcs or circles.

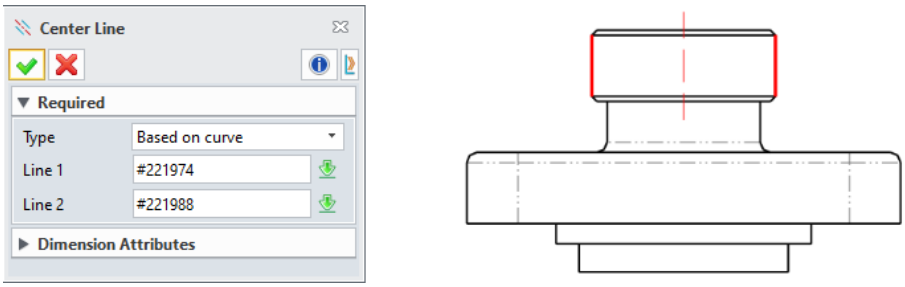



Figure 321 Center line

With ***Center Mark Circle*** command, you can place a center line through a circular pattern.

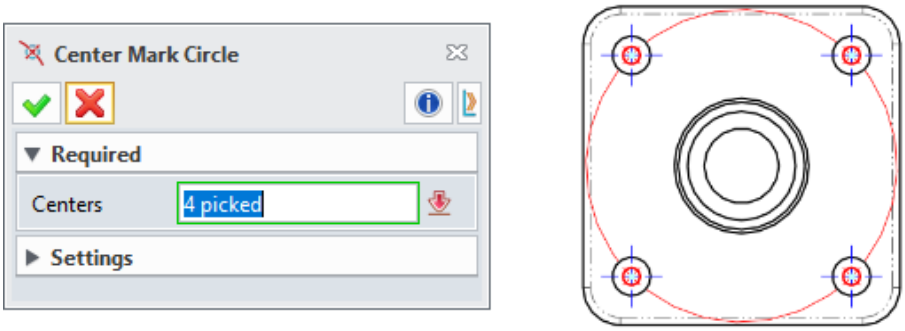



Figure 322 Center mark circle

**Datum Feature:**

With ***Datum Feature*** command, you can create a datum feature.

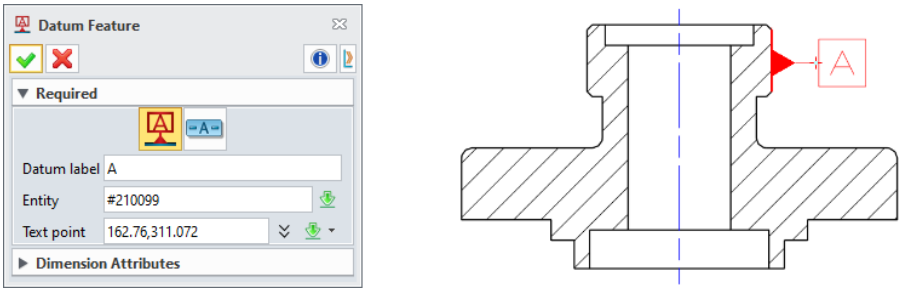



Figure 323 Datum feature

**Feature Control Symbol:**

With this command, you can create a feature control symbol.

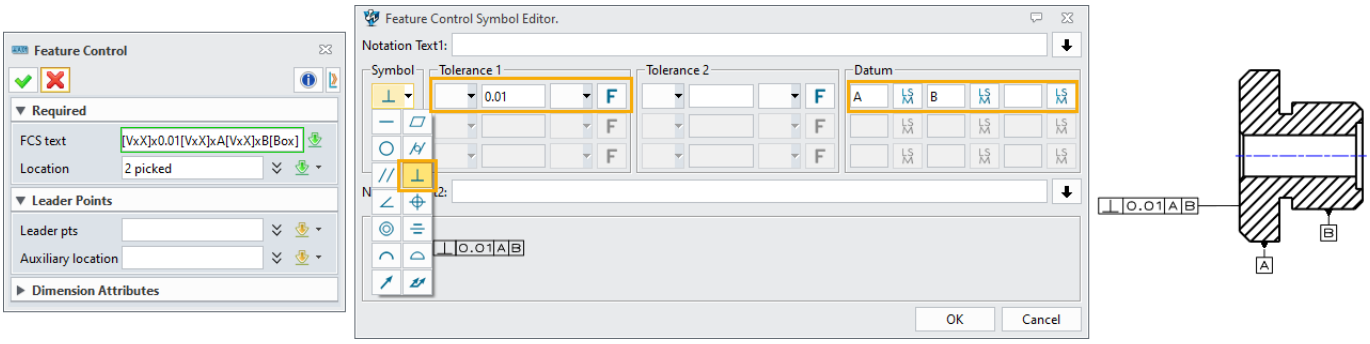



Figure 324 Feature control

**Surface Finish Symbol:**

***Surface Finish*** represents the quality of part surface machining therefore in 2d view, it I s required to select the edge to define the surface finish symbol.

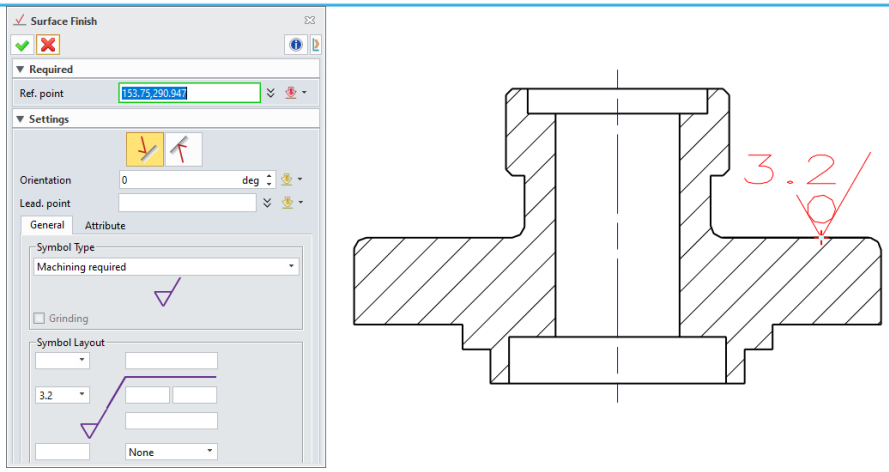



Figure 325 Surface finish

**Label/Text:**

With this command, you can create a label manually.

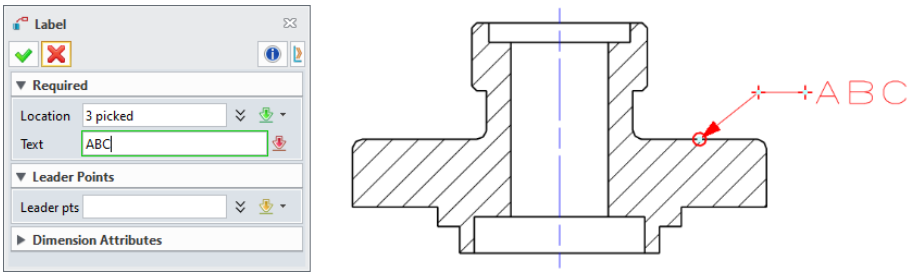
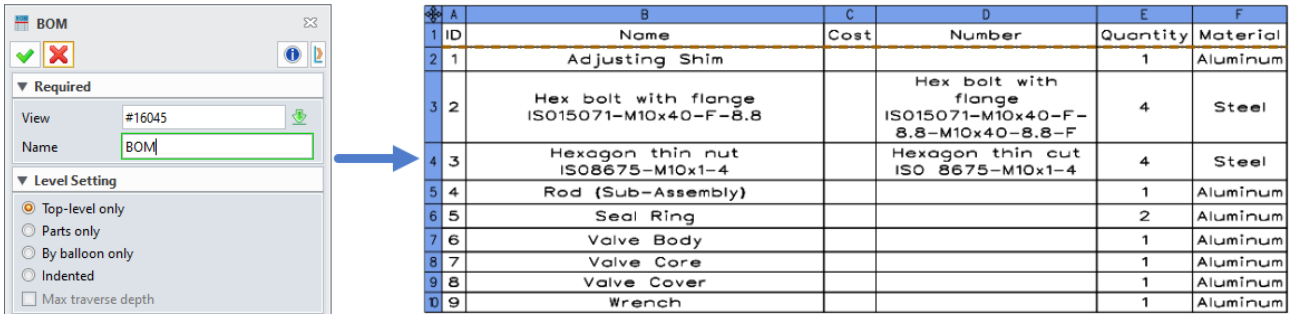



Figure 326 Label

## 6.4.8 Create a BOM table

Select the ***BOM*** command from ***Layout*** ribbon tab and select the view to create a BOM table, then add the name for BOM.




| | A | B | C | D | E | F |
|---|---|---|---|---|---|---|
| 1 | ID | Name | Cost | Number | Quantity | Material |
| 2 | 1 | Adjusting Shim | | | 1 | Aluminum |
| 3 | 2 | Hex bolt with flange ISO15071-M10x40-F-8.8 | | Hex bolt with flange ISO15071-M10x40-F-8.8-M10x40-8.8-F | 4 | Steel |
| 4 | 3 | Hexagon thin nut ISO8675-M10x1-4 | | Hexagon thin cut ISO 8675-M10x1-4 | 4 | Steel |
| 5 | 4 | Rod (Sub-Assembly) | | | 1 | Aluminum |
| 6 | 5 | Seal Ring | | | 2 | Aluminum |
| 7 | 6 | Valve Body | | | 1 | Aluminum |
| 8 | 7 | Valve Core | | | 1 | Aluminum |
| 9 | 8 | Valve Cover | | | 1 | Aluminum |
| 10 | 9 | Wrench | | | 1 | Aluminum |

Figure 327 Create a BOM table

Below is the definition of commonly used option in level setting of BOM table.

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

**Parts only:** Only list out all parts including the one from all sub-assemblies, but not list sub-assembly.

Each subassembly component is an individual item.

In table format, you could use the left/right arrow to add or delete the attribute and use up/down arow to rearrange the sequence of the attribute.

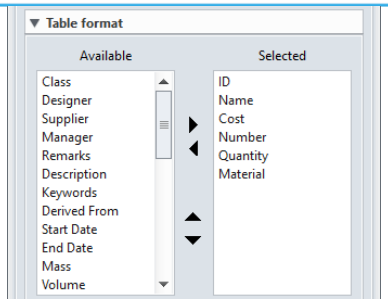



Figure 328 Table format

## 6.5 2D Drawing Case of a Part

Now we have finished the basic study of 2D drawing. In this section, we will take a **Valve Cover** as an example to teach you how to create a 2D drawing for a part.

The picture below shows the completed 2D drawing of a valve cover.

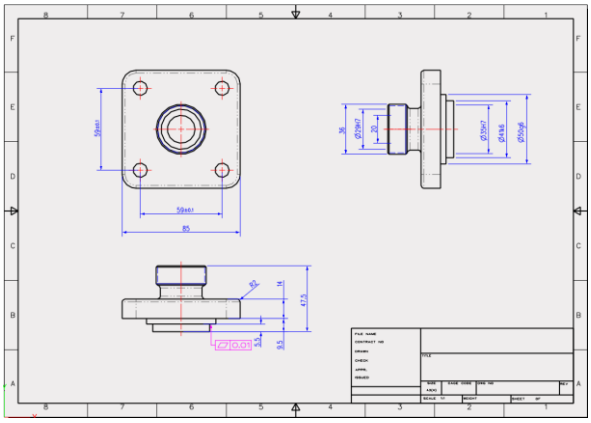


Figure 329 2D drawing of a valve cover

### 6.5.1 Create the View

STEP 01 Open **Valve Cover** part, right click the blank area of the graphic area and select **2D Sheet** from right-click menu, then select **A4_H(ANSI)** as the template.

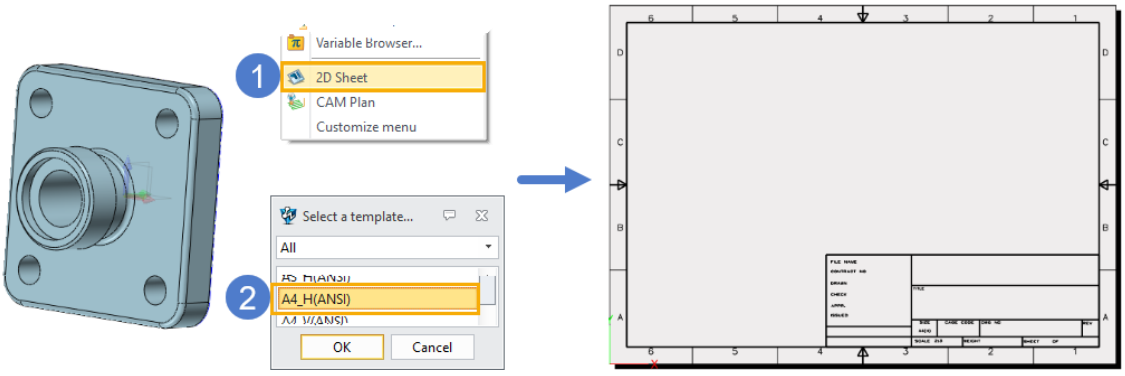



Figure 330 Create the 2D sheet for valve cover

STEP 02 Select ***Standard*** view command from ***Layout*** ribbon tab and select **Front** view of **Valve Cover**, then set the scale to **1:1.5**.

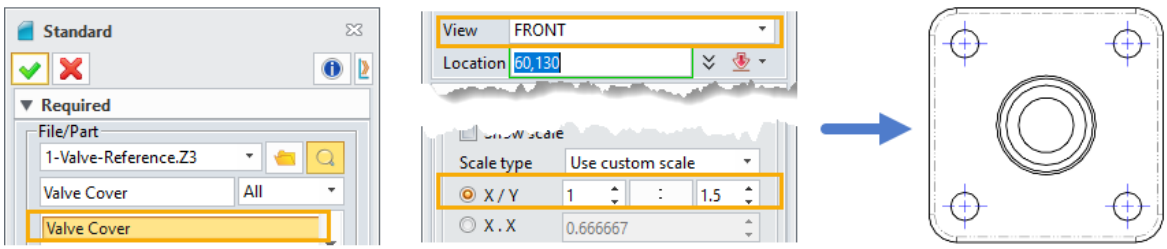



Figure 331 Create the front view of valve cover

STEP 03 Select ***Projection*** view command to create other two views of **Valve Cover** as shown below.

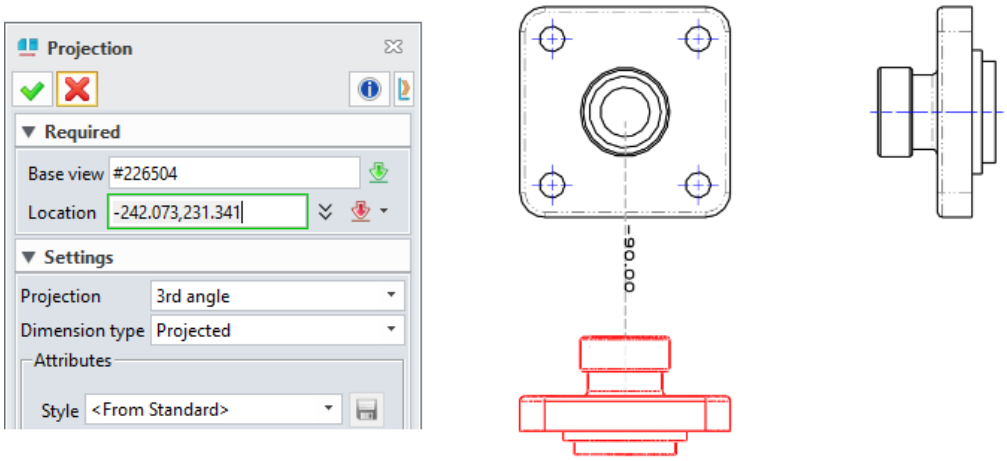



Figure 332 Projection view of valve cover

## 6.5.2 Add Annotation and Symbol

STEP 01 Select ***Quick Dimension*** tool from ***Dimension*** ribbon tab and add the dimension between two bolt holes of **Valve Cover** in the front view,

Then set the precision to **0.1** from **DimTool** and the tolerance from **Modify Tolerance** as shown below.

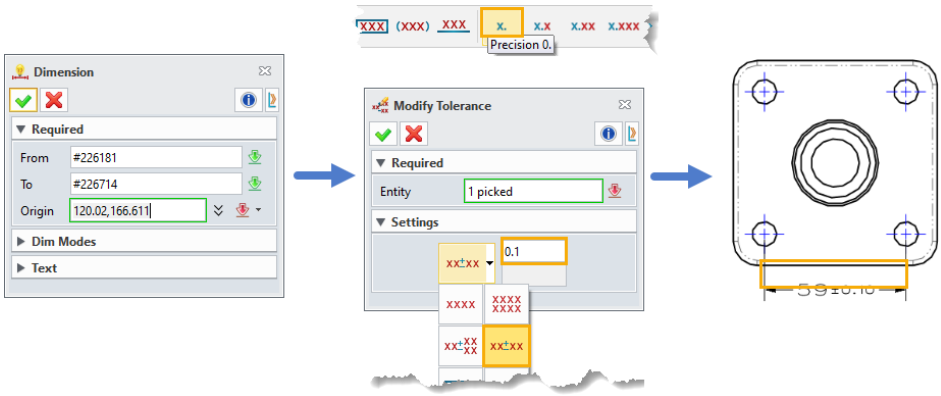



Figure 333 Add the dimension and annotation

STEP 02 Select ***Quick Dimension*** tool from ***Dimension*** ribbon tab and add the dimension of **Valve Cover** port in the front view, add the diameter symbol and tolerance from quick dim tool as shown below.

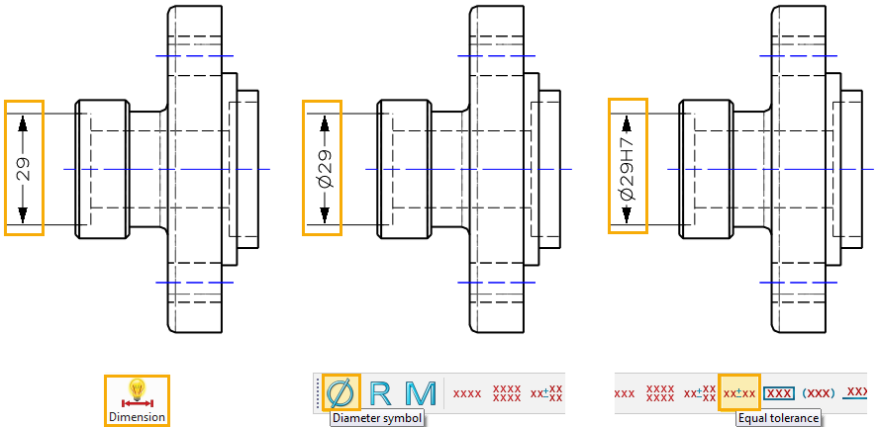



Figure 334 Add the dimension and annotation

STEP 03 Select ***Feature Control Symbol Editor*** tool from ***Dimension*** ribbon tab and set the parameters as shown below.

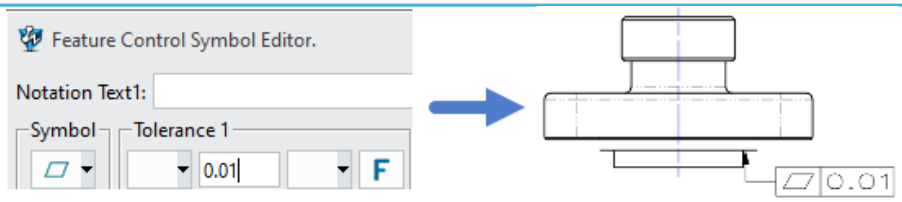



Figure 335 Add feature control symbol

STEP 04 Finish all the remaining dimension and symbol with same method, and the results are shown as the picture below.

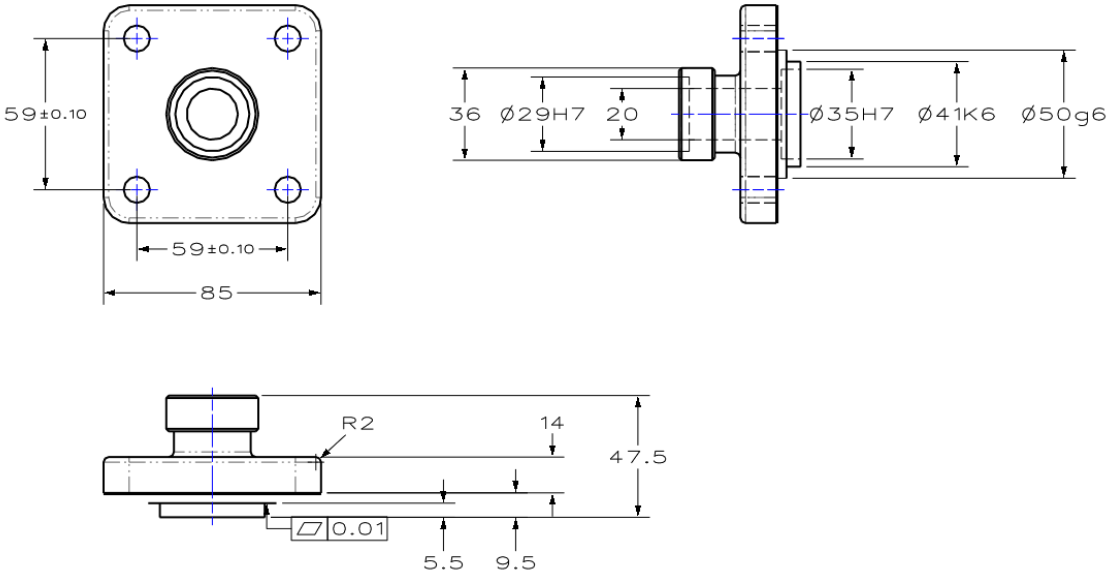



Figure 336 Remaining dimension and symbol

STEP 05 Open ***Style Manager*** from ***Tool*** ribbon tab, select **Dimensions** and set the parameters in **General** and **Text** tabs as shown below.

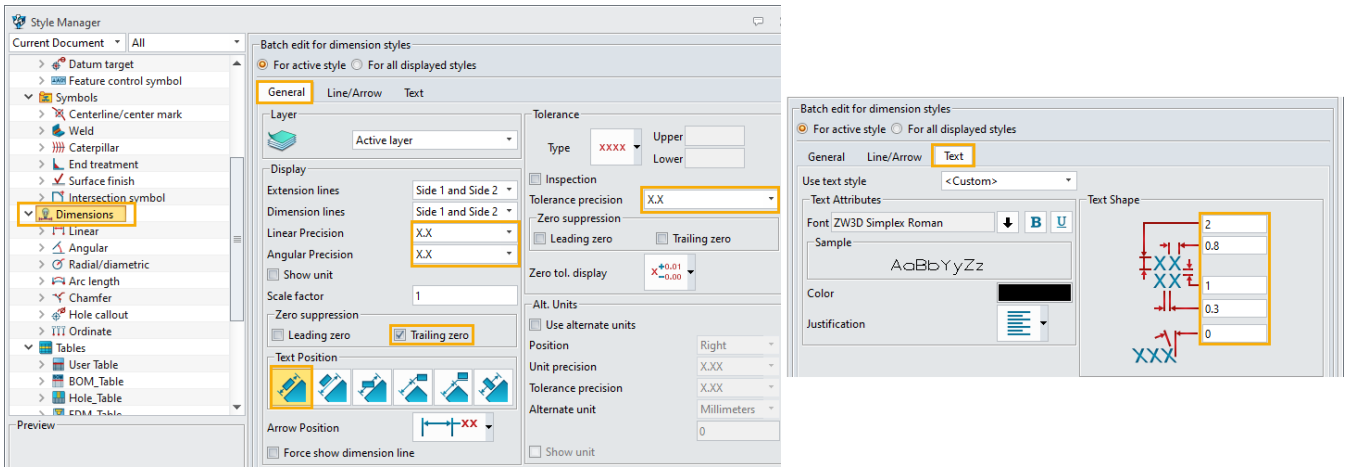



Figure 337 Set the parameter in the General and Text tabs in Style Manager

STEP 06 Adjust the position of annotation and symbols, and you will get the results as below.

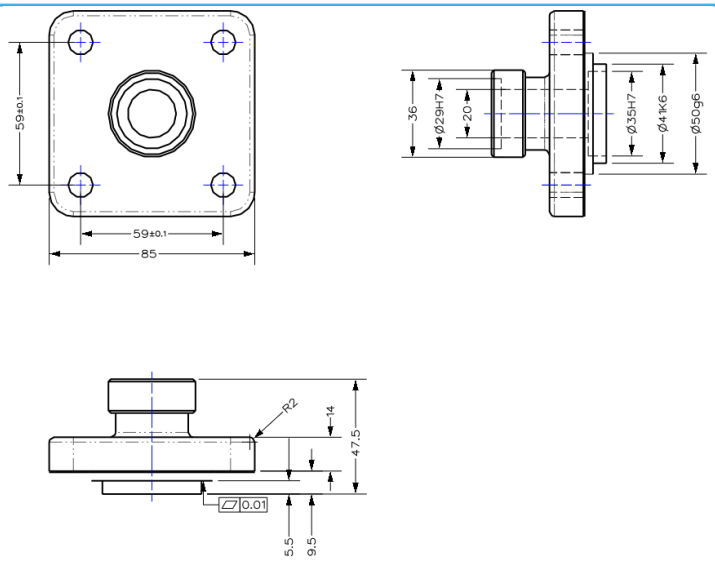



Figure 338 The 2D drawing of valve cover

### 6.5.3 Create the Part Attribute on Title Block

STEP 01 Select ***Text*** command from ***Sketch*** ribbon tab and add it on the **Title** cell, then open the **Text Editor** and add the link with Variable **Part_Name**, see the image below.

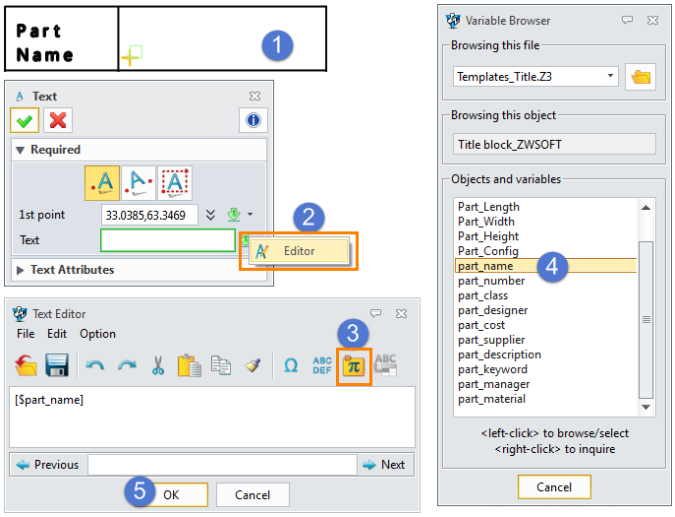



Figure 339 Add text variable on title block

STEP 02 After finishing the definition of the variable link and the back to the 2D sheet level you will get the result as shown below.

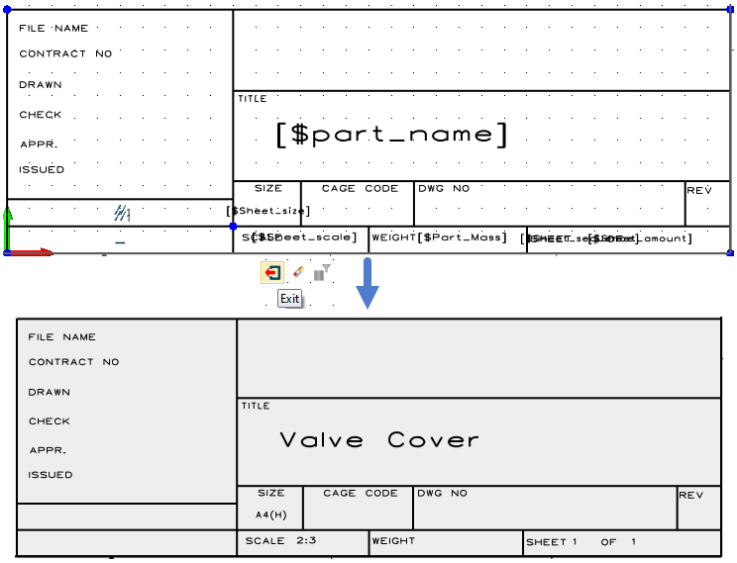



Figure 340 The result of the title block

## 6.6 2D Drawing Case of Assembly

Now we have finished the 2D drawing of a valve cover, in this section, we will take a ball valve as an example to teach you how to create a 2D drawing for an assembly.

The picture below shows the completed 2D drawing of the ball valve assembly.

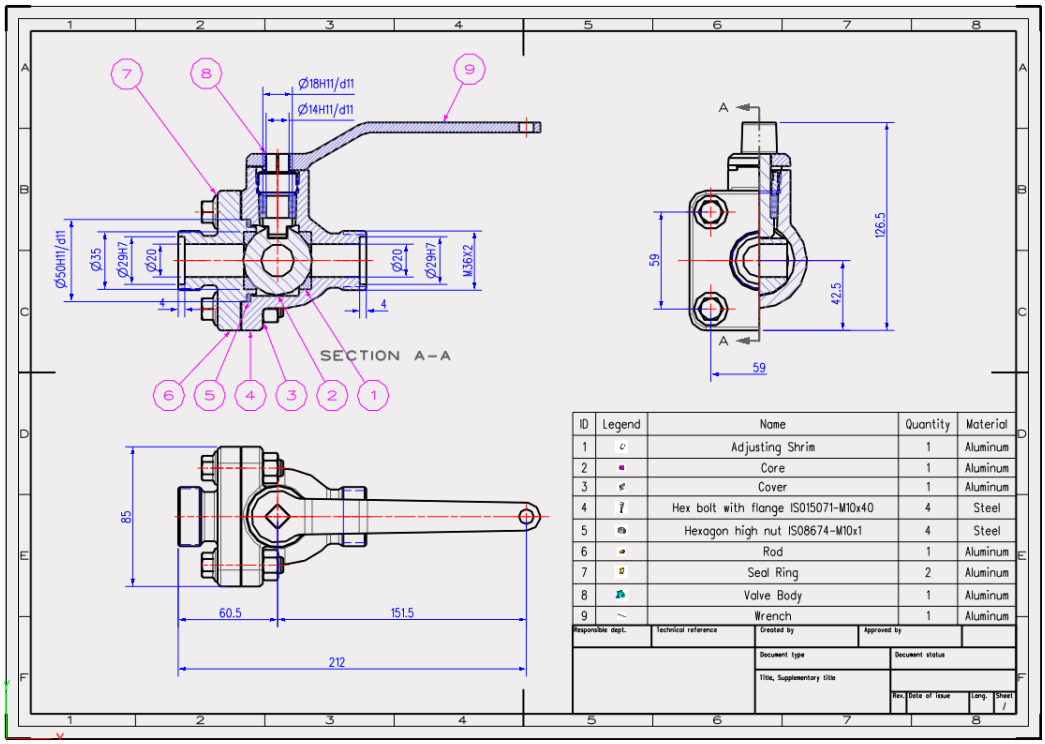



Figure 341 The 2D drawing of the ball valve assembly

### 6.6.1 Create the View

STEP 01 Open **Valve (Assembly)** and select **2D Sheet** from right-click menu. Then select **A3_H(ANSI)** as the template.

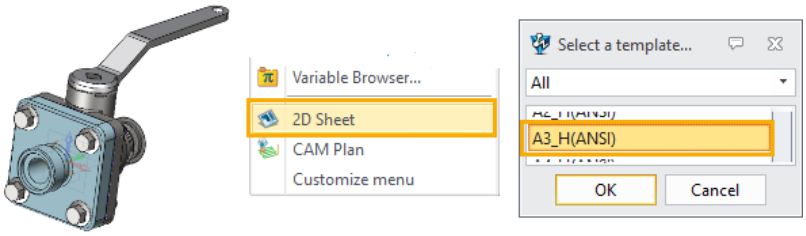



Figure 342 Create the 2D sheet of Valve (Assembly)

STEP 02 Select ***Standard*** view command from ***Layout*** ribbon tab and select the front view of **Valve (Assembly)**. Then set the scale to **1:1.5**.

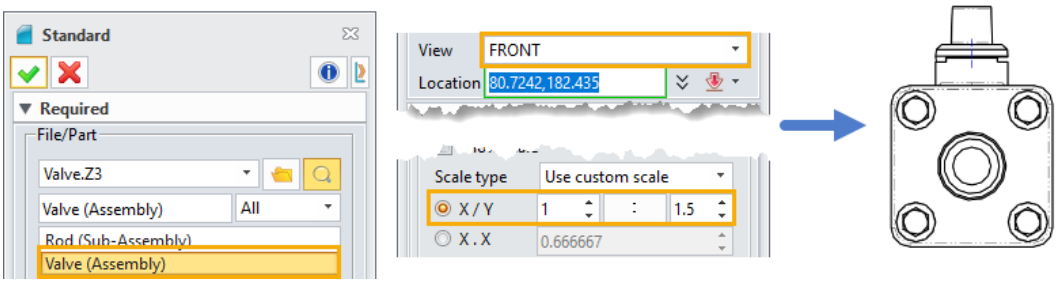



Figure 343 Created the front view of Valve (Assembly)

STEP 03 Select ***Broken Section*** command from Layout ribbon tab to create the broken section for the front view of Valve (Assembly) according to the setting below.

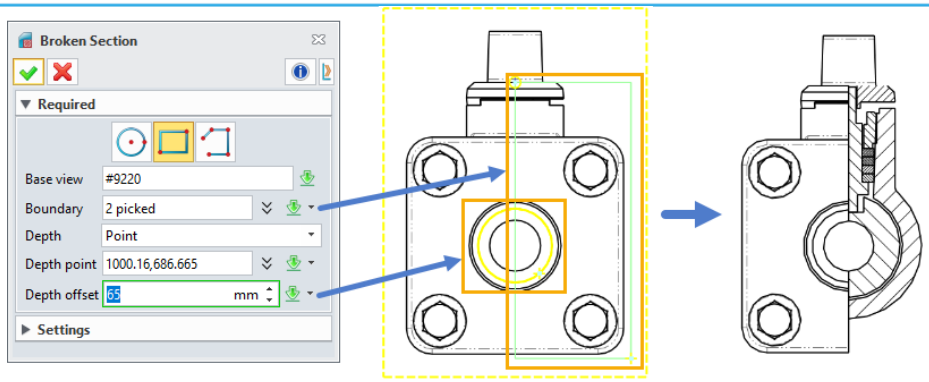



Figure 344 Create the broken section

STEP 04 Select ***Full Section*** command from ***Layout*** ribbon tab and create the section view from front view (Tick the **Flip arrow** in Section Line).

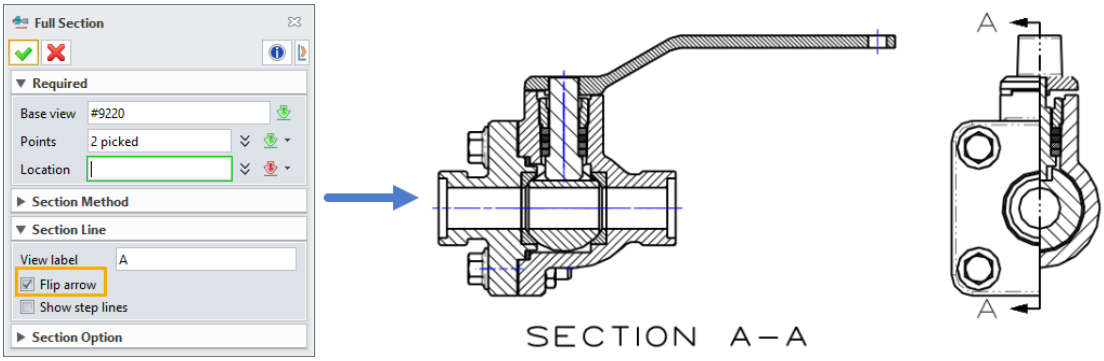



Figure 345 Create the full section

STEP 05 Select ***Projection*** view command from ***Layout*** ribbon tab to create the projection view of the section view which was created in the last step (Select **1st angle** for Projection in Settings).

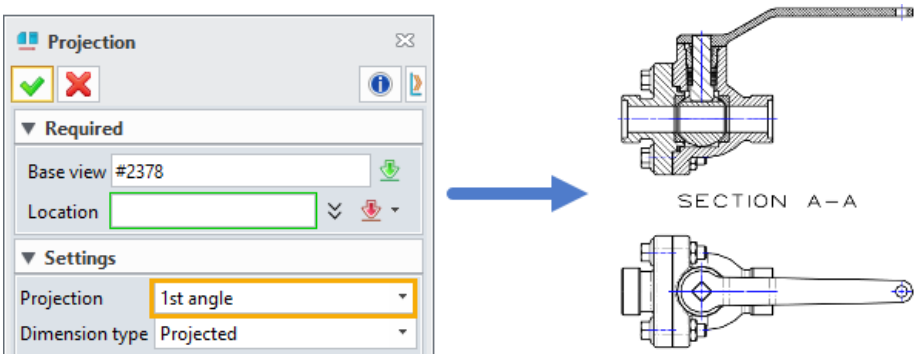



Figure 346 Create the projection view

## 6.6.2 Add Annotation and Symbol

STEP 01 Select ***Quick dimension*** tool from ***Dimension*** ribbon tab and add the dimension for **Valve Rod** in the section view, and the diameter symbol, and set the precision to **0** and the tolerance as shown below.

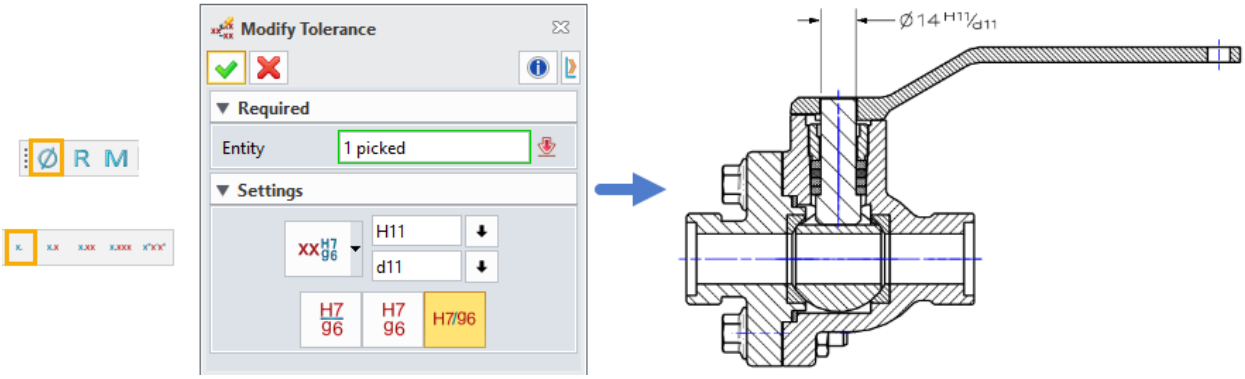



Figure 347 Add the dimension and tolerance

STEP 02 Finish all the remaining dimensions with same method, as shown by the picture below.

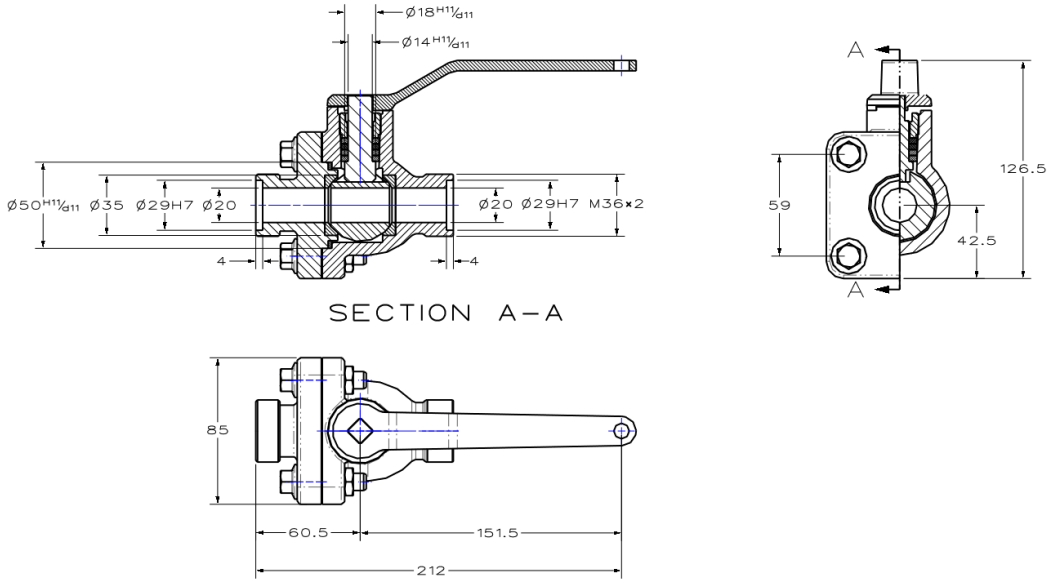



Figure 348 Add remaining dimensions

STEP 03 Open ***Style Manager*** from ***Tools*** ribbon tab, select **Dimensions** and set the parameters in **General** and **Text** tabs as shown below.

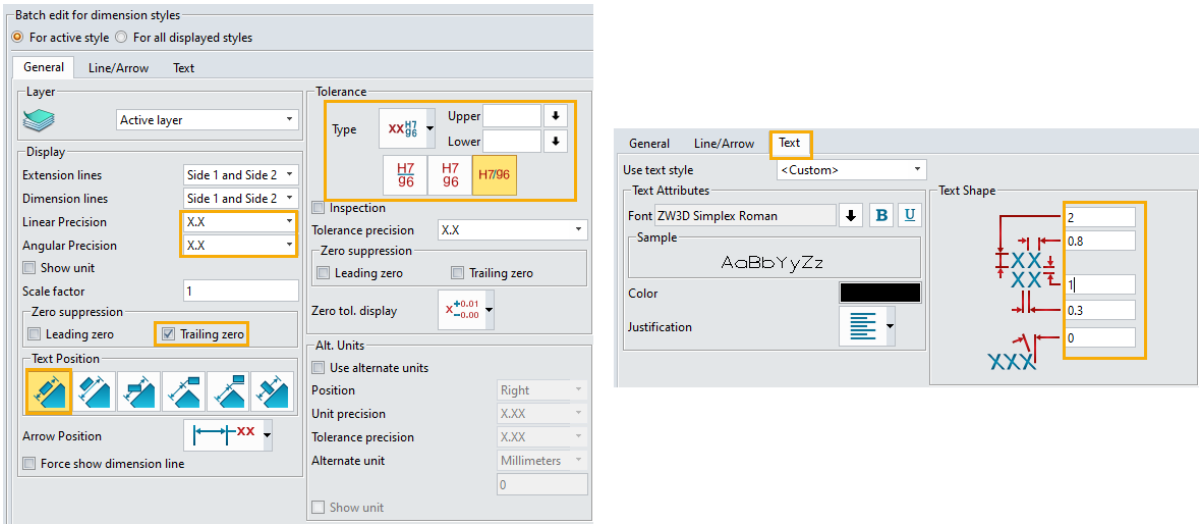



Figure 349 Set the parameters in General and Text tabs in Style Manager

STEP 04 Adjust the position of annotations and symbols, then you will get the results as below.

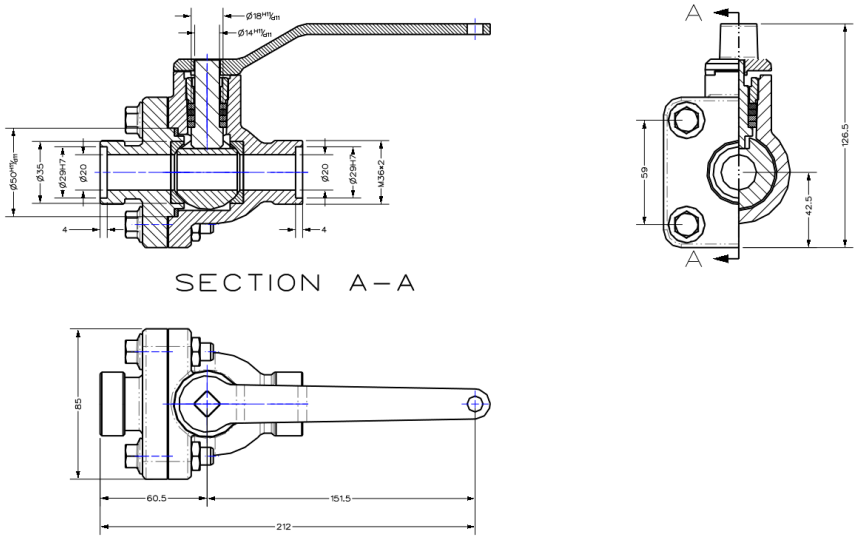



Figure 350 The 2D drawing of ball valve

### 6.6.3 Add Balloon

STEP 01 Select ***Auto Balloon*** tool from ***Dimension*** ribbon tab and add the balloon for section view of the valve assembly, Then select square as the pattern type and skip left side in Layout setting as shown below.

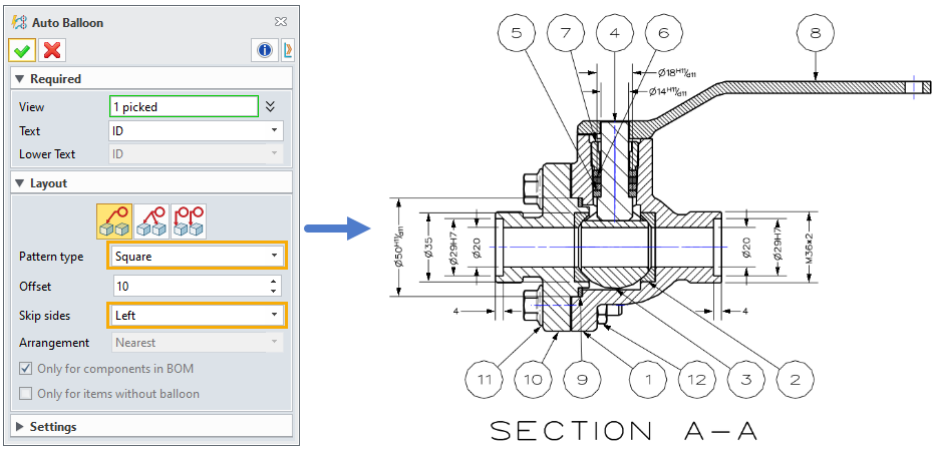
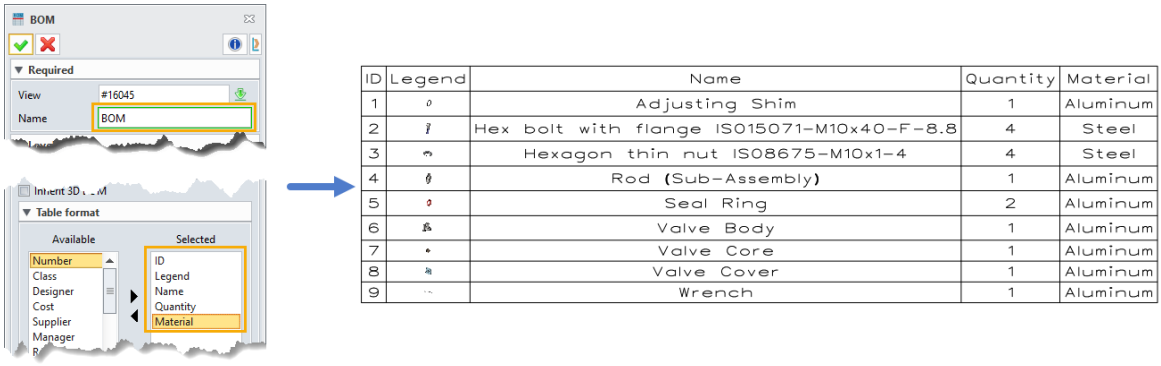



Figure 351 Auto balloon

### 6.6.4 Create a BOM Table

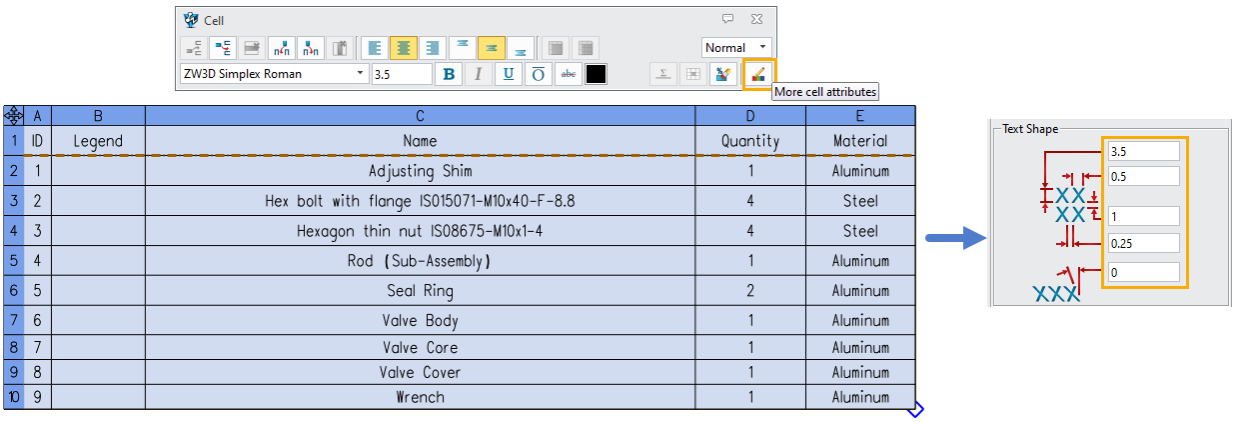
STEP 01 Select ***BOM*** command from ***Layout*** ribbon tab and select the section view of the valve assembly, then add the name and define the table format as shown below.




| ID | Legend | Name | Quantity | Material |
|---|---|---|---|---|
| 1 | | Adjusting Shim | 1 | Aluminum |
| 2 | | Hex bolt with flange ISO15071-M10x40-F-8.8 | 4 | Steel |
| 3 | | Hexagon thin nut ISO8675-M10x1-4 | 4 | Steel |
| 4 | | Rod (Sub-Assembly) | 1 | Aluminum |
| 5 | | Seal Ring | 2 | Aluminum |
| 6 | | Valve Body | 1 | Aluminum |
| 7 | | Valve Core | 1 | Aluminum |
| 8 | | Valve Cover | 1 | Aluminum |
| 9 | | Wrench | 1 | Aluminum |

Figure 352 Create the BOM table

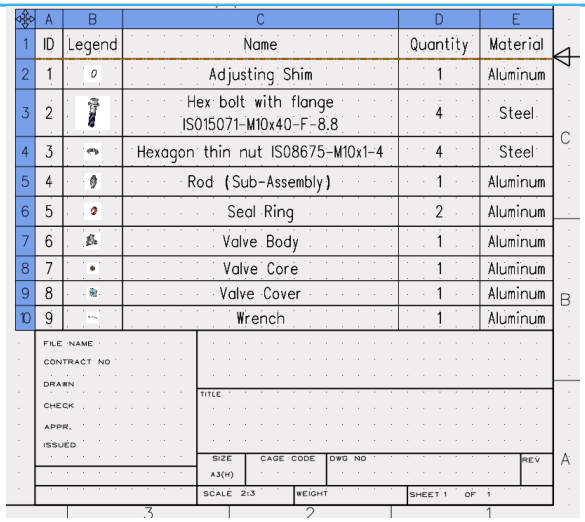
STEP 02 Select all the text in the **BOM** table and the **Cell** setting dialog will pop-up, then click **More cell attributes** and set the **Text Shape** as shown below.




| | A | B | C | D | E |
|---|---|---|---|---|---|
| 1 | ID | Legend | Name | Quantity | Material |
| 2 | 1 | | Adjusting Shim | 1 | Aluminum |
| 3 | 2 | | Hex bolt with flange ISO15071-M10x40-F-8.8 | 4 | Steel |
| 4 | 3 | | Hexagon thin nut ISO8675-M10x1-4 | 4 | Steel |
| 5 | 4 | | Rod (Sub-Assembly) | 1 | Aluminum |
| 6 | 5 | | Seal Ring | 2 | Aluminum |
| 7 | 6 | | Valve Body | 1 | Aluminum |
| 8 | 7 | | Valve Core | 1 | Aluminum |
| 9 | 8 | | Valve Cover | 1 | Aluminum |
| 10 | 9 | | Wrench | 1 | Aluminum |

Figure 353 Set the text shape

STEP 03 Adjust the **BOM** table to the appropriate size and drag it to the bottom right corner to match the template.

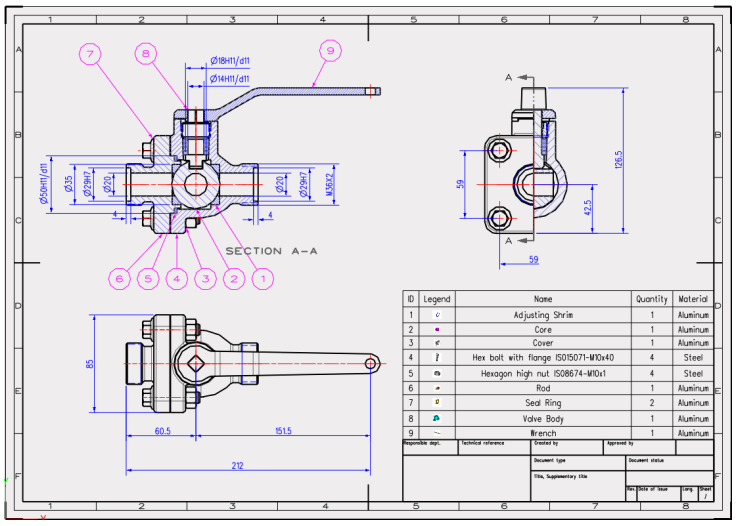


| ID | Legend | Name | Quantity | Material |
|---|---|---|---|---|
| 1 | | Adjusting Shim | 1 | Aluminum |
| 2 | | Hex bolt with flange ISO15071-M10x40-F-8.8 | 4 | Steel |
| 3 | | Hexagon thin nut ISO8675-M10x1-4 | 4 | Steel |
| 4 | | Rod (Sub-Assembly) | 1 | Aluminum |
| 5 | | Seal Ring | 2 | Aluminum |
| 6 | | Valve Body | 1 | Aluminum |
| 7 | | Valve Core | 1 | Aluminum |
| 8 | | Valve Cover | 1 | Aluminum |
| 9 | | Wrench | 1 | Aluminum |

FILE NAME
CONTRACT NO
DRAWN
CHECK
APPR.
ISSUED
TITLE
SIZE A3(H) | CAGE CODE | DWG NO | REV
SCALE 2:3 | WEIGHT | SHEET 1 OF 1

Figure 354 Adjust the position of BOM table

Now we have finished all the steps of creating the 2D drawing of the ball valve assembly.




| ID | Legend | Name | Quantity | Material |
|---|---|---|---|---|
| 1 | | Adjusting Shim | 1 | Aluminum |
| 2 | | Core | 1 | Aluminum |
| 3 | | Cover | 1 | Aluminum |
| 4 | | Hex bolt with flange ISO15071-M10x40 | 4 | Steel |
| 5 | | Hexagon high nut ISO8674-M10x1 | 4 | Steel |
| 6 | | Rod | 1 | Aluminum |
| 7 | | Seal Ring | 2 | Aluminum |
| 8 | | Valve Body | 1 | Aluminum |
| 9 | | Wrench | 1 | Aluminum |

Figure 355 The 2D drawing of the ball valve assembly