

Product Design

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ZW3D[™] V2022 CAD Product Design

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Foreword

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

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

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

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

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

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

What you are learning is **ZW3D CAD Product Design**, a primary tutorial.

Thanks for being our user! The ZW3D Team

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

Requirements	Recommended
Operating System	Microsoft [®] Windows 7_SP1 Microsoft [®] Windows 10
Processor	Intel [®] Core™ 5 or above
RAM	8G or above
Graphic	OpenGL [®] 3.1 or above 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.

🖞 ZW3D2020Eng_x64.exe		Open
		Run as administrator
Fig	ure 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.

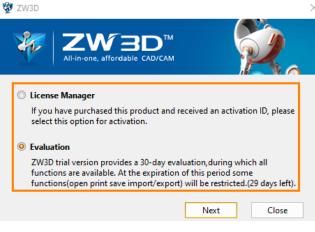


Figure 2 Evaluation

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*.

0	y Online Activation					
<u> </u>	y Offline Activation Activation					
-						
Soft-key O	line Activation					
Activation ID	:	-	-	O Verify		
Jser Inform	ation					
Name:)*	Company:)*
Email:		*	Phone:			
Country:	Select Country/Regior 🔻	*	Industry:	Select Industry	٣	*
(Items with *	can not be empty.)					
System or ai	emember to return the licens ny hardware change for smoo license from the license mana	oth lic				

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.

Manager

fault units?	
Millimeters	Inches
efault user role	
Expert	*
Primary	
Intermediate	
Advanced	
Expert	
Figure 4	User Role Settings

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 **m** 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.

æ (🗅 🖆 🚟	$a \sim \sim 0$:• v 🗧 File	e Edit Vie	ew linsert Att	ributes Inquire	e Tools Utiliti	es Applications	Window	Help					ZW3D	2021 Alpha x64 - [Part001.Z3]						GD 23
											PMI Tools	Visualize	Inquire	Electrode A	pp Mold					⇔ Fir	id a command	ຊ 🤌 😣	_ 8 ×
	getch Block	Extrude Revolve Basic Shape	Sweep Lof		hamfer Draft I	lole Rib Three		Face Shell Offset •	Sh	Jadd Divide S Add Divide S Edit Shape		ace Resolve In SelfX	nlay C)	Bend - Point	ith Wrap to Wrap Pa Faces to Fac Morph	ettern Pattern Geometry	• • • • • • • • • • • • • • • • • • •	*	opy Scale	Datum Plane - Datum	2		
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Sele	ct command	or entity.																					

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.

💯 Configuration		Configuration		
General	General	General	Solid background	
Part		Part	Color	
2D	 One object per file (new files) Show hints 	2D Color	Gradient background	
Color	Save file without display data	Background	Upper left Upper	r right
Background	Ul Language English 🔻	Display	Lower left Lower	r right
Fig	ure 7 Changing UI Language	F	igure 8 Changing Background Color	

0 00 0

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.

 D_BasicCasesStudy		Commands Transfer Hotkey Mouse
Trainings	Ribbon Appearance → Ribbon Tabs →	Command List
	Ribbon Panels 🕨	Type: Part 🔻
	ToolBars >	Group: Shape
	Styles 🕨	
	Customize	Search:

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.

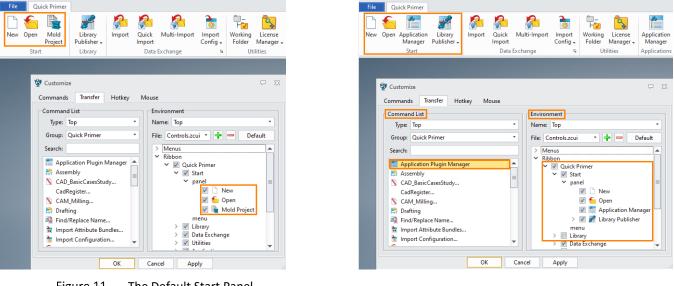


Figure 11 The Default Start Panel

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.

Product Design <<<<</>

🐲 Customize		🧐 Customize				\sim	23	
Commands Transfer Hotkey Mouse		Commands Transfer	Hotkey Mouse	e				
Category: Menus *	Default	Mouse Actions Function Key		Mouse Button				
Search:	Just show items with hotkey	Pan Null	*	+	Middle Mouse Button		•	
 ✓ Menus ✓ &File 		Rotation Null	•	+	Right Mouse Button		•	
L' New	Ctrl+N	Zoom Ctrl	•	+	Middle Mouse Button		•	
Copen Open in ZWCAD	Ctrl+O	Reverse mouse whee	l zoom direction			Default		

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.

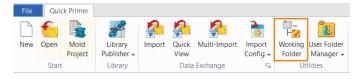


Figure 14 Set Working Folder

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.

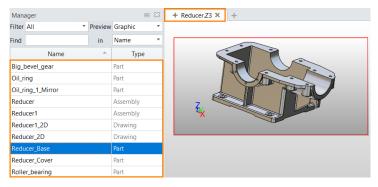


Figure 15 Multi-object File

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.

Configuration		Ω Σ
General	General	
Part	General	
2D	One object per file (new files)	Automatic file locking
	Show hints	Auto open error window
Color	Save file without display data	 Confirm File/Save
Background	UI Language English •	
Display	Default layer name Layer0000	

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.

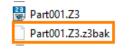


Figure 17 Auto File Backup

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.

V Configuration					Configuration			5
General	-	Default linear units	mm •		Color	*	ZW3D backup folder	C:\Users\THINK\Documents\ZW3D_bak
Part		Default mass units	kg *		Background	2	Save backup files in the same I	ocation as the original
2D		Object tolerance (mm)	0.01				Standard parts folder	
Color		Max file revisions to backup	210	(Backup folder defined on "Files" tab)	Display		Create instance in working file	C:\Users\THINK\Documents\ZW3D\Stand
Background		Create auto-recover file	and the	minutes	Files		Symbol library	
		C Enable detailed tooltip	Detailed toolt	ip timeout (seconds) 1	CAM		Clipboard library	
Display		Reduce assembly memo	ory use		User	-	Paper size definitions	DEF_SHEET_MM
Files	يسلو	Use 3rd xlsx lib to replace	ce Excel	- Annow	PDM	-	Session journal	ers\THINK\AppData\Roaming\ZWSOFT\Z

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.

📄 Part001.2.z3bak
📄 Part001.3.z3bak
📄 Part001.4.z3bak
Part001.5.z3bak
📄 Part001.6.z3bak
👹 Part001.Z3

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.

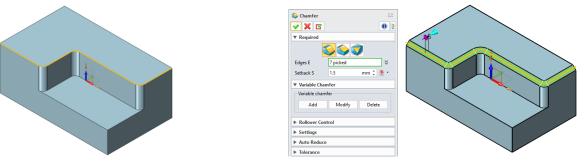


Figure 20 Selecting Connected Lines with Shift

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.

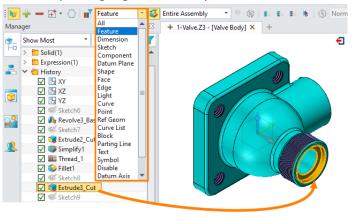


Figure 21 Picking with Filter

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 prehighlighted 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.

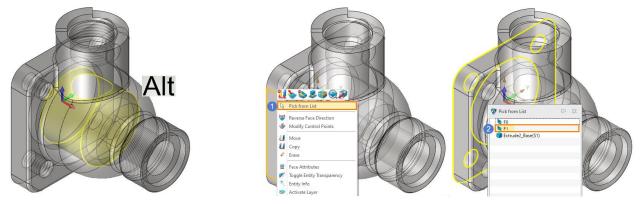


Figure 22 Picking with Alt

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.

Mana	iger
\$ _	Show Most 🔹
-0	A Part001
_	> 🛅 Solid(1)
- B	🗸 📩 History
	🗹 🕌 Default CSYS
	🗹 燧 Sketch1
0	✓ ✓ Strude1 Base
	🖌 🥯 Sketch2
29	MODEL STOP HERE

Corresponding Modeling Feature

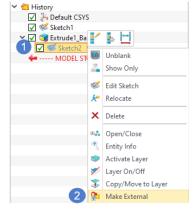


Figure 28 Converting to External Sketches

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.

💯 Create New File			₽ X
Туре			
١		۲	C
Part/Assembly	Drawing Packet	Drawing Sheet	Standalone Sketch
	Σ	5	
CAM Plan	Equation Set	Multi-Object	
	Figure 29	Standalone Sk	

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	
*	Snap Filter	Define the filter for object picking
-	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.

	💯 Configuration	
	General	General
	Part	Grid spacing (mm) 5 Grid style point 🔻
😵 Sketch Settings 🗢 🖾	2D	Drawing standard ANSI •
	Color	✓ Display drawing border
Units mm ·	Background	Highlight modified dimensions
Grid Spacing 5	Display	✓ Auto change clipboard color
Shu spacing 5	Files	Sketch
	CAM	Auto sketch orientation
Enable constraint solver	User	Project edges into sketch 🛛 Auto constrain sketch
Auto constrain new geometry	User	Auto prompt dimension value
	PDM	Enable external snap Add weak dimensions automatically
Auto dimension new geometry	ECAD	Transparent constraint icon 🗹 Add constraints when over-constrained
		Display inactive sketch in grey
		Show center of arc/circle
Reset OK Cancel		Enable constraint conflicting manager Image: Display default XY axis

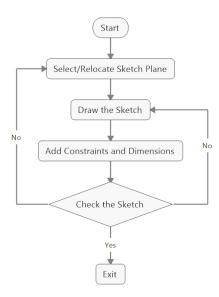
Figure 31 Setting Grid Spacing

Figure 32 More Sketch Settings

3.3 Sketch Flow & Elements

3.3.1 Sketch Flow

Below is the sketch flow of ZW3D.





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 \checkmark to enter the sketch environment.

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	▼ Required			
¶ ⊢□	Plane	Default CSYS_XY		Z
_		Use previous		V
	▼ Orientation			
	Up		🗧 💆 🕶	
9	Origin		🗧 💆 🗧	X

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.

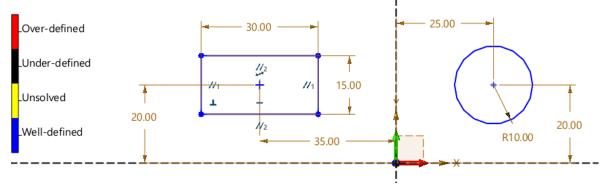


Figure 35 Elements of a Well-Defined Sketch

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.

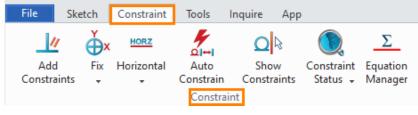


Figure 36 Geometric Constraints

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.

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✓ X	0
▼ Required	
Curves/Points 2 picked	₫
▼ Contraints	
	=

× 11 // 0 Add Points Parallel Auto Constrain Constraints Vertical θ× Fix + Part001.Z3 Ľ۲, Points Horizontal Points Vertical Point to Midpoint Point to Line/Curve Point to Intersection Point Coincident

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.

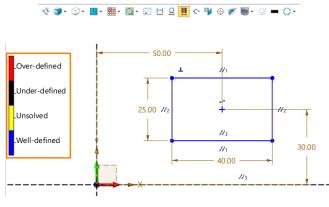


Figure 39 Different Statuses of Constraints Are in Different Colors

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

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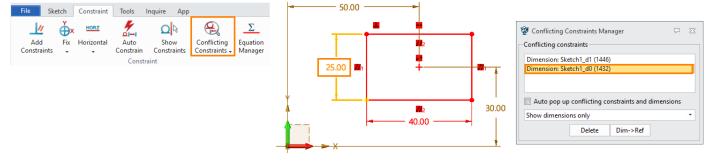


Figure 40 Check and Solve Conflicts of Constraints

3.4.2 Dimensional Constraints

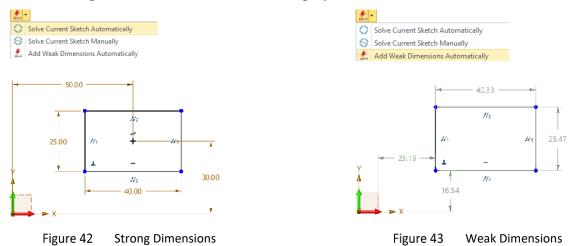
With the Quick Dimension command, you can easily add dimensional constraints.



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.



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.

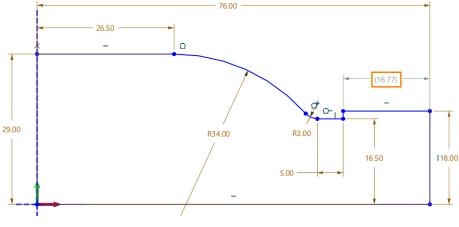


Figure 44 Reference Dimensions

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

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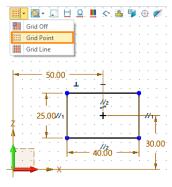
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Dimensions On/Off	Constraints On/Off

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**.



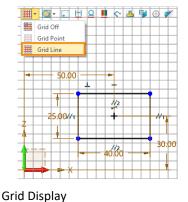
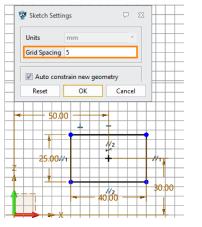
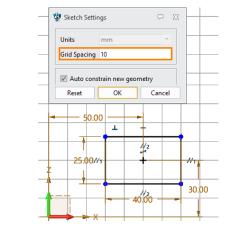


Figure 46 G

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







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.

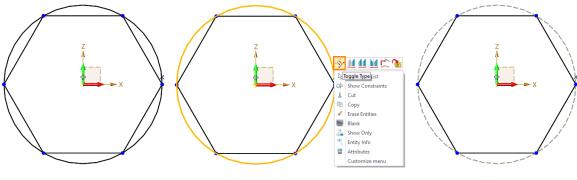


Figure 48 Toggling Types of Geometries

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

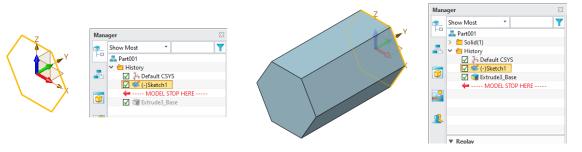
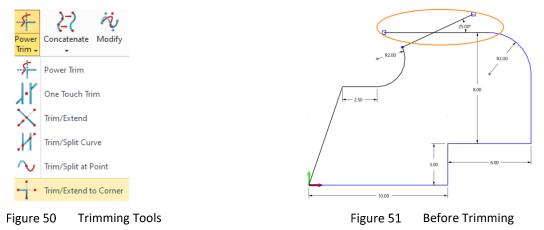


Figure 49 Construction Geometries Will Not Appear in the Modeling Environment

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.



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.

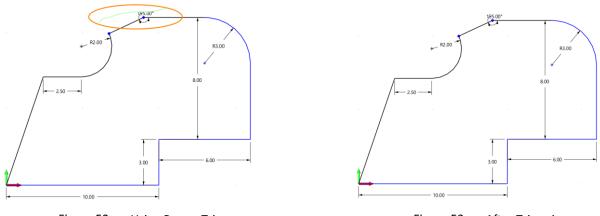


Figure 52 Using Power Trim

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.

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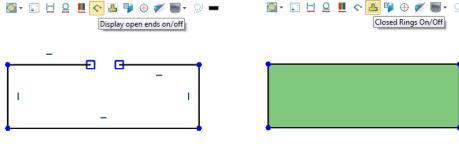


Figure 54 Checking Unconnected Lines

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.

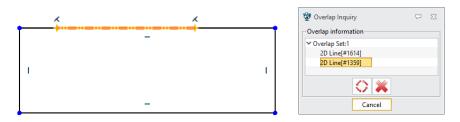


Figure 55 Checking Overlapping Lines

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.

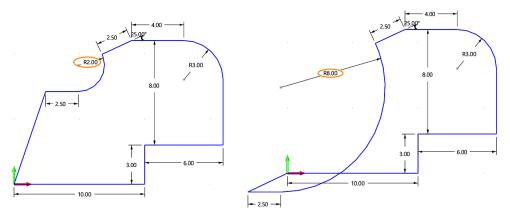


Figure 56 Deformed Sketch Due to Modification of Dimensions

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*.

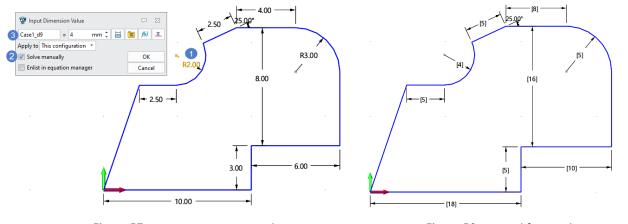
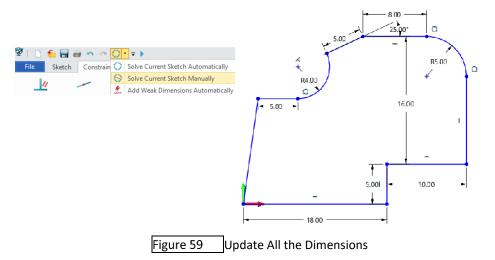


Figure 57 Input Dimension Value

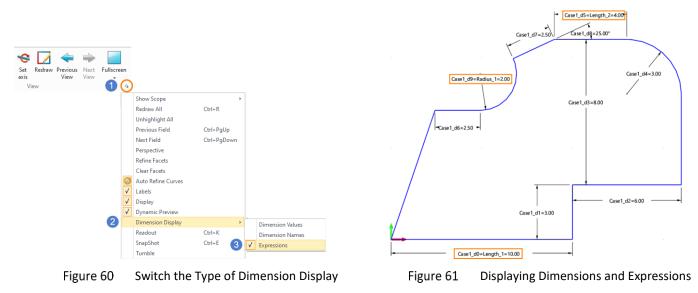
STEP 02 Modify other dimensions, and their values will be in brackets, meaning that they are not updated yet.

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



3.5.6 Dimension Display

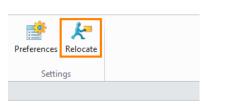
Some dimensions of a parametric sketch are linked with variables or expressions. In this case, you can change the type of dimension display (*Tools* tab -> *View* panel -> the second on -> *Dimension Display* -> *Expressions*), as shown in Figure 60. After that, both dimension values and expressions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



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*.



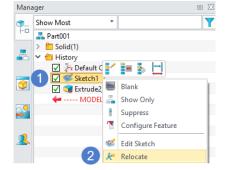


Figure 62 Relocating in the Sketch Environment

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.

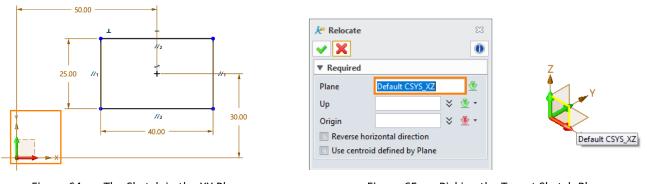
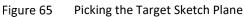


Figure 64 The Sketch in the XY Plane



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

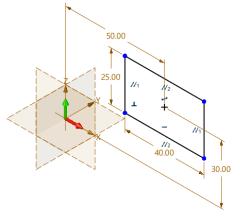


Figure 66 Relocating the Sketch Plane to XZ

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.

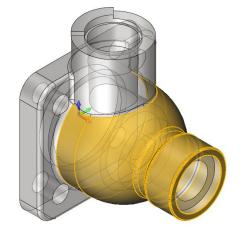
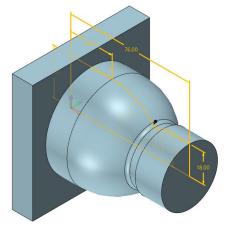
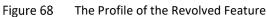
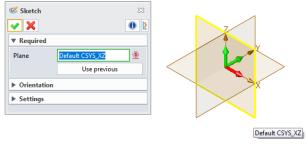


Figure 67 The Revolved Feature of the Valve Body





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

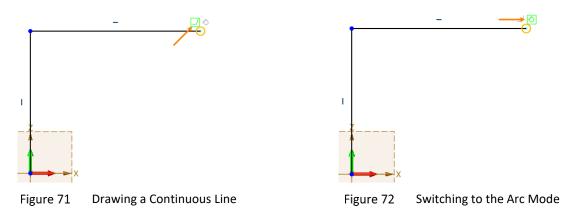


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🗹 Enable "Sr	mart Pick"		
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Figure 69 Select the XZ Plane as the Sketch Plane

Figure 70 Enabling Smart Pick

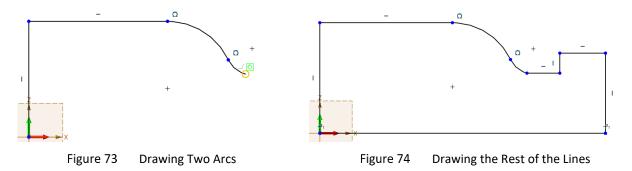
STEP 02 Use the **Draw** command to draw a continuous line vertically from the origin and then horizontally. Make sure **Smart Pick** is enabled in the **Snap Filter** dialogue box for a quicker sketch process.



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.



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.

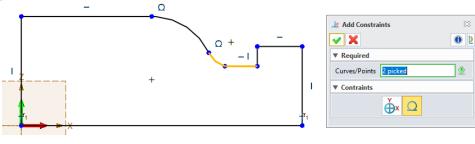
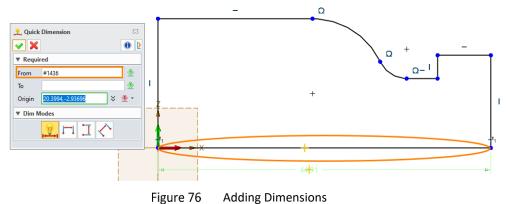


Figure 75 Adding Proper Constraints

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



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.

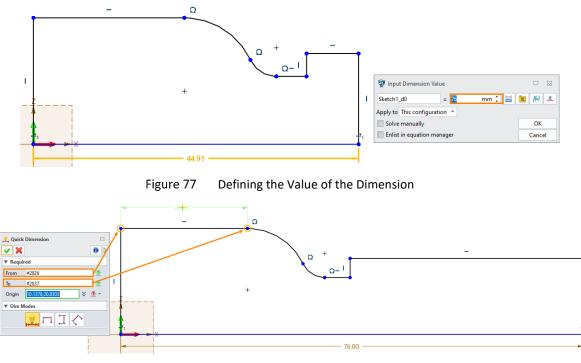


Figure 78 Specifying the Dimension with Points

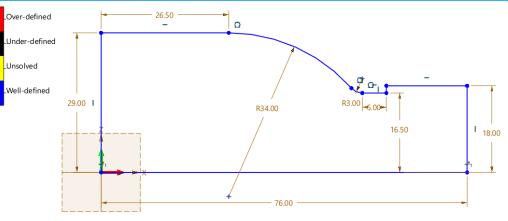


Figure 79 All the Dimensions Are Added

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

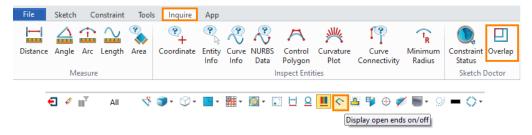


Figure 80 Checking the Profile

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.

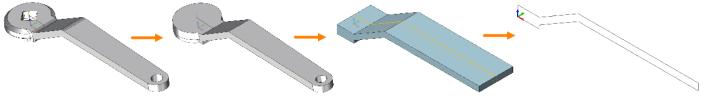


Figure 81 Profile for Extrude Feature of Wrench

- 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.

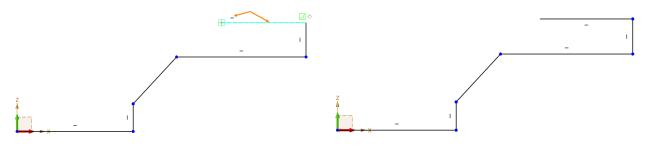


Figure 82 Drawing a Continuous Line

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.

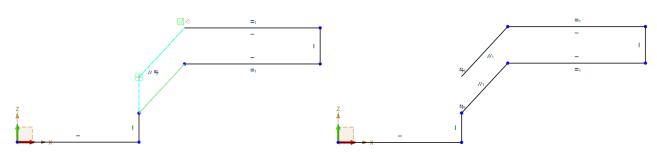


Figure 84 Drawing the Parallel Slanting Line

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

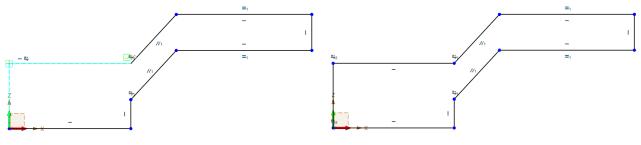


Figure 85 Finishing the Rough Profile

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

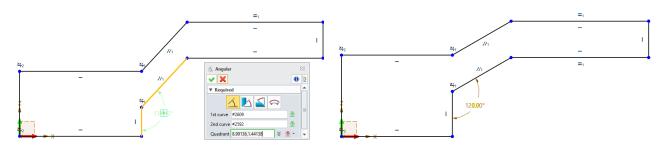


Figure 86 Specifying the Angular Dimension

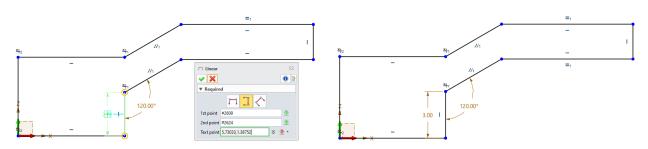


Figure 87 Defining Linear Dimensions with the Quick Dimension or Linear command

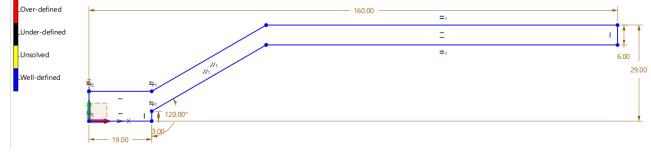


Figure 88 The Final Well-Defined Sketch

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

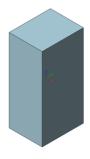
Feature-based modeling means that a feature is regarded as the basic modeling unit and that a 3D model consists of various features. Generally, modeling features can be divided into three types.

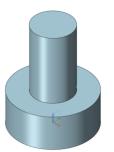
Datum feature

Datum features include datum planes, axes, and point.

Basic feature

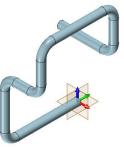
Basic features include extrude, revolve, and sweep features.





Extrude Feature

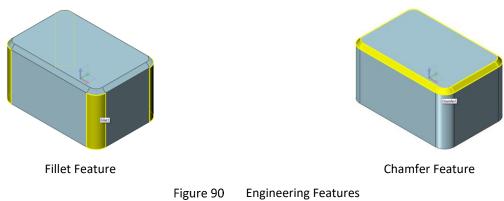
Revolve Feature Figure 89 Basic Features



Sweep Feature

Engineering feature

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



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.

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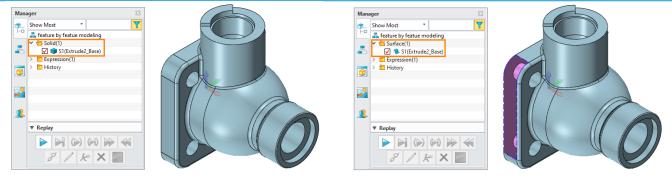
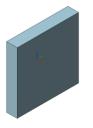


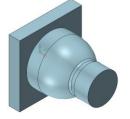
Figure 91 Solid Shape and Surface Shape

Parametric Modeling 4.2

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.









Cut Features



Engineering Features

Extrude Base

Add a Revolve

Add Another Revolve 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.

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✓ 🧐 Sketch6				
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X:A Overall_length	75	75	mm	Number
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Type Number T Le			Replace Exp	
Variable Input Type Number Le Name Expression Description			Replace Exp	

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.

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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.

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	Boolean shapes	, @ @ \$	*		Angle Measure Arc Measure Face Dihedral Expression 2		Browsing this object Modeling Process		
x	 Offset Settings Tolerance 			1			Objects and variables <pre> 4 End_angle = 270 deg Total_length = 15 mn </pre>		

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.

Manager 🛛			
Show Most Show Most Modeling Process			
 E Solid(1) Surface(0) Wireframe(0) 	😵 Create/edit variable 🕞 🖾		
★ Expression(2) <u> <u>π</u> End_angle = 360.00 deg <u> <u>π</u> Total length = 15.00 mm </u></u>	Variable Type Number Y Angle Y	Manager 🛛	⊈∣ ⊡ 🗲 层 🖨 ∾ ~ 🚫 ד ▸
> Part Config(1)	Name End_angle deg ▼ 2 Expression 270 At Image: Compared to the second sec	> Solid(1)	File Shape Free Form Wireframe
Default CSYS Sketch1 My Revolve1_Base	Description	burface(0) Wireframe(0) v	Set Rotation Set Redraw Previous Next
MODEL STOP HERE	Reset OK Cancel		Center axis View View View

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.

🖞 Configuration		\square	23
General	General		A
Part			
2D	Automatic file locking		
	Show hints		
Color	Save file without display data 🔹 🔲 Confirm File/Save		
Background	UI Language English		
Display	Default layer name Layer0000		
Files	Max undo steps 75 🗹 Compact undo/redo Max undo memory (MB) 600		
САМ	Direction key rotate speed		
User	Default linear units mm 🔹		
PDM	Default mass units kg 💌		
PDIVI	Object tolerance (mm) 0.01		
ECAD	Max file revisions to backup 0 (Backup folder defined on "Files" tab)		

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.

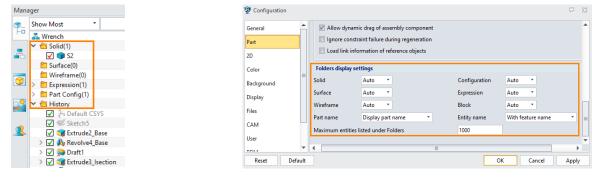


Figure 98 Folders i

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.

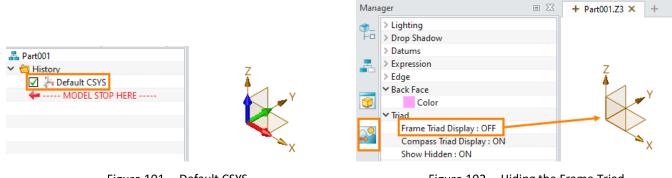
Mana	iger 🕅
\$	Show Most 🔹 🍸
-0	🚣 Wrench 🔺
	> 🛅 Solid(2)
- To	> 🛅 Expression(1) 📃
	🗸 🔁 History
	✓ ↓ Default CSYS
Ţ	🖌 🥨 Sketch5
	🖌 🧊 Extrude2_Base
29	> V In Revolve4 Base
-	🖛 MODEL STOP HERE
	> 🗹 💭 Draft1 🛛 🔻
<u>_</u>	▼ Replay
	8 / k ×

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.

Product Design <<<<<</>



Default CSYS Figure 101

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.

3 1 2 Datum						
Plane 🗸			🧏 Datum Plane		23	
/	Datum Axis		🗸 🗙 🖪		0	
3	Datum Plane	_	▼ Required			
Y z x	Datum CSYS			🗧 💐 🎉 🍕	L 🕼 🖋	
e II	Drag Datum		Offset	10	mm 🗘 垫 *	
\boldsymbol{k}	LCS					

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*.

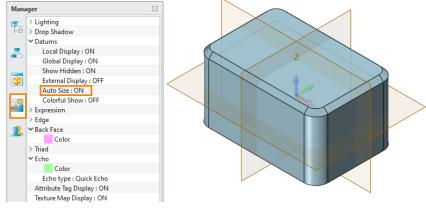


Figure 104 Making the Datum Planes Appear Bigger

> Show or Hide

You can show or hide datum planes.

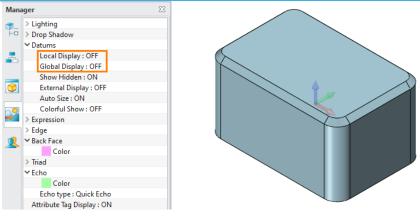


Figure 105 Turning off the Display of Datum Planes

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*.

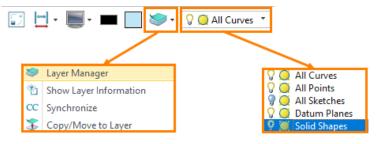


Figure 106 Layer-Related Tools in ZW3D

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.

😨 Layer Manager									\Box	23
All Category	◀ Filter All	* All	•			Select object in I	layer			\bigcirc
✓ ✓ All Category	Active	ID 🔺	Name	On	Frozen	Category Quantity		Desc	Description	
CURVES	- >	0	Layer0000	S	Q		1			\neg
SHEETS		,	1							-
SKETCHES										
SOLIDS										
[No Category]	Δ	Activate New		w		Delete	Import	Import Export		
Category Editor					ОК	Cancel	Apply			

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*.

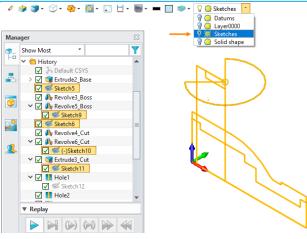


Figure 108 Display All Sketches

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.

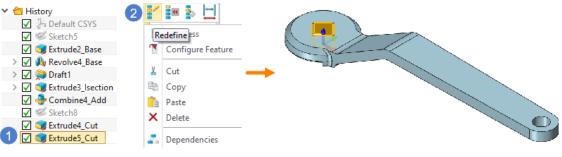


Figure 109 Feature Operations

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.

Mana	iger 🛛	
¶ ⊢□	Show Most 🔹 🍸	
L - 0	The second secon	
-	> 🛅 Solid(1)	
- 6	🛩 🛅 History	
	🗹 🕌 Default CSYS	
7	🗹 🧊 Block2_Base	
	🔽 🌍 Fillet1	
	🔽 🌑 Shell1	
	MODEL STOP HERE	

Figure 110 Make the Shell After Fillet

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

Mana	ager	23	
9 _	Show Most 🔹	Y	
μ	🚠 Engineering features1		
	> 🛅 Solid(1)		
- B	🗸 🔄 History		
	🗹 🕌 Default CSYS		
	Block2_Base		
\bigcirc	🔽 🌑 Shell1		
	🔽 🌍 Fillet1		
	MODEL STOP HERE		
-			

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.

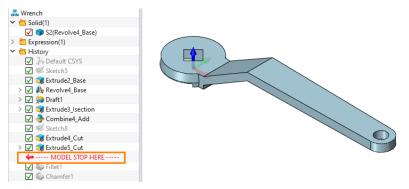


Figure 112 Insert Features

4.4.5 Display Modes and View Types

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

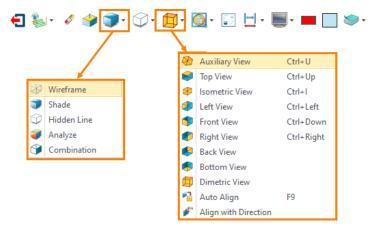
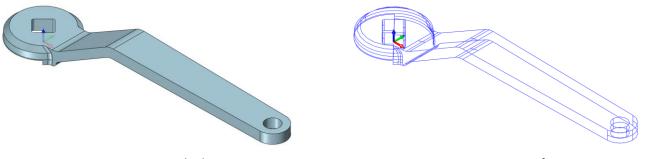
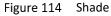


Figure 113 Display Modes and View Types

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





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.

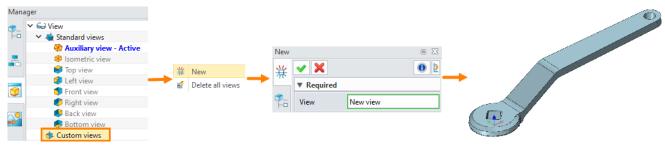


Figure 116 Creating a Custom View

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.

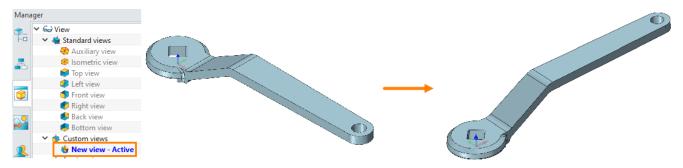


Figure 117 Change to the Custom View

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.

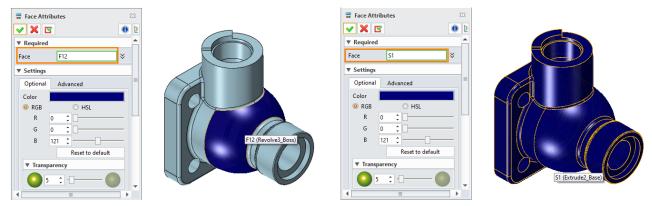


Figure 118 Setting Face Appearance

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.

Metal •	Metal Met (brushed) - (cast		Bronze		🔘 Metal (d	ast)			23		1	Cont.		
	Metal	-	- 16 18		~ X	3			0 2					
	Aluminium		0 16		▼ Require	d								
	Bronze				Faces	1 pi	icked		≈	1				
	Copper				▼ Settings	-				N				
	Gold				Metal Color	Cus	tom		-			X		
	Nickel				RGB		⊙н	SL					1	
	Platinum				R	147	-						11	1
\bigcirc	Silver				G B	144 140	:				1 il			
	Steel				Ambient Diffuse		-0						X	
	Titanium				Specular					1				
Fi	igure 119	Text	ure Li	st				Fi	igure 120	Тех	turizing	the Pa	rt	

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.

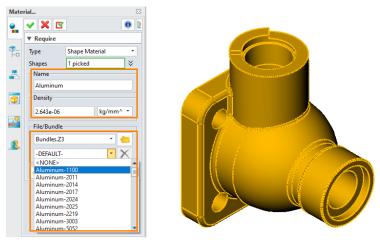


Figure 121 Setting the Material

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**.

💇 Part Attributes				Π 23
Standard User	Physical			
Name	Valve Body	Derived fro	m	
Number		Designer	David	
Manager		Supplier	ZWSOFT	
Cost		Class		•
Do not section	1	🔲 Do not I	hatch	
Do not list in E	юм	🔲 Do not l	list in Root	
Keywords	Each keyword sho	ould be separated b	y a "," comma	
Description				
Create time	12/27/2019		Short date	•
Last modified	5/16/2020		Show time	
Sheet object		Sheet code		
CAM object				
1				
	Reset OI	K Cancel	Apply	

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*.

👰 Part Attribu	ites 1		Γ Σ	3	🐲 Material Attributes 🗢 🛛
Standard	User Physical				Name
Source	All in current	•	Update data on save		Density
Material	Aluminum	2	Decimal 0.00 🔻		2.643e-06 kg/mm' *
Density	2.64e-06	kg 🔻 / mm 🔻	^3		File/Bundle
Mass	N/A	kg 🦯			Bundles.Z3 DEFAULT-
Area	N/A	mm^2 🥖			V R Cancel
Volume	N/A	mm^3 🦯			

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.

Part Attribute	s						\Box	23
Standard l	Jser Physic	al						
Source	All in current	•			🔲 Up	date data	on save	
Material	Aluminum			late	Decim	al 0.00	•	
Density	2.64e-06		kg ₹	mn	n ▼ ^3			
Mass	0.32		kg		1			1
Area	35198.47		mm^2		1			L
Volume	122049.37		mm^3		/			L
Size L	76.00	w	94.50	н	75.00	mm		L
Centroid X	3.23	Υ	5.72	z	9.08e-05	mm		
Stock size] 🥥					

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.

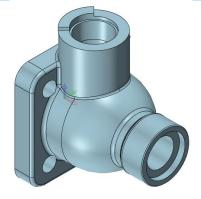


Figure 125 Valve Body

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

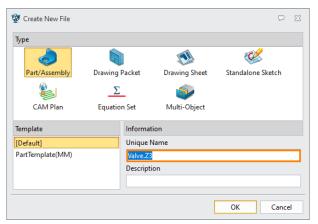


Figure 126 Creating a New .Z3 File

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.

	(1 🕄 🕷	<u>s</u> ▼	ø	4) -	•	8	• 🔘 •	57	⊢ -		-		🤝 -	8	🕽 Layer000	• 00	
Man	ager																		
Filter	All		-	Previ	ew (Graphi	c	-				Man	ager						
	[Filter	All			,	Preview	Graphic	*
Find				in		Name		*									-		
		Name					Туре		_			Find					in	Name	•
		Name					type								Nam	ne			Туре
Valve						Part										-			
											3	Valve	e Body	/				Part	
	2) 📓	Rename																	

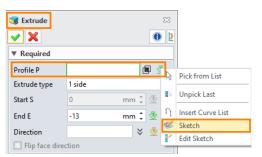
Figure 127 Renaming the First Part

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

Filter All	•		2	🔜 ն ն 🔇
Name		Expression	Value	Unit Type
🗸 🚣 Valve				
<u>π</u> B	ase Length	75	75	mm Number
Variable Inp	ut			
	ut Number * Length	× Min	Max	
Туре		• Min I		place Expression Enlist Dimension
Variable Inp Type Name Expression	Number 🝸 Length			place Expression Enlist Dimens

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.



Sketch	×3	
Plane	Default CSYS_YZ	
Flane	Use previous	
Orientatio	n	X
Settings		

Figure 129 Creating a Sketch

rigule 150 Datum 12 as the Sketch Plane	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.

 	- 53.68 =1				
	- //1		🐲 Input Dimension Value	2 🖓 🕱	😵 Variable Browser 🛛 🖓 🖄
			Sketch7_d0 = 53.68 mm 🗘 🔚	<u>π</u> f(x) <u>π</u>	Browsing this file
			Apply to This configuration *	01	Valve.Z3 *
1/2		//2 =	 Solve manually Enlist in equation manager 	OK Cancel	Browsing this object
					Sketch7
Т	//1				Objects and variables
•					3

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.

Extrude		23		-13
 X 		0		
Required				
Profile P	Sketch7	🗖 🔮		
xtrude type	1 side	· ·		
itart S	0	mm 🗘 🕭 🐑		
ind E	-13	mm 🗘 🕭 👻		
Direction		🗧 🕹 🗸		
Flip face dire	ction			
Boolean				
	904	\$		
Boolean shapes		\approx		

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.

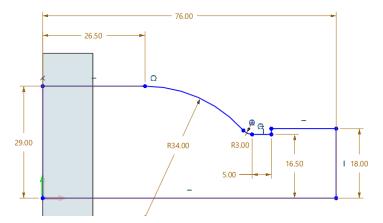


Figure 135 The Sketch of the Revolve Feature

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 .

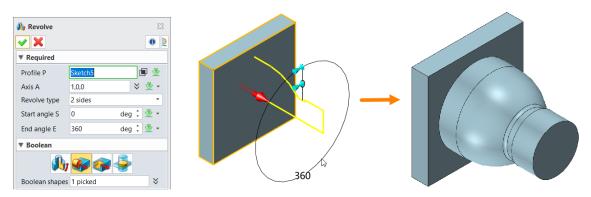


Figure 136 Revolving the Sketch

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.

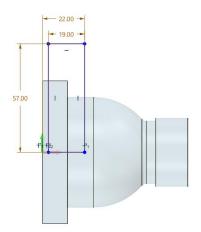


Figure 137 Sketching a rectangle on the XZ Datum Plane

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 v to revolve the sketch around the default revolve axis.

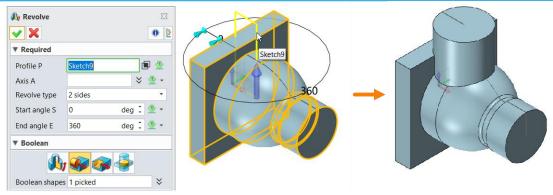


Figure 138 Making Another Revolve Feature

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.

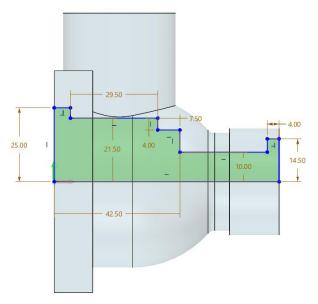


Figure 139 The Revolve-Cut Profile

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.

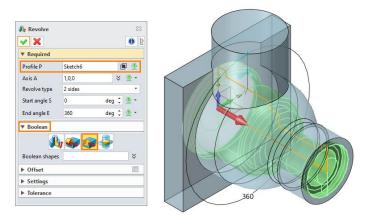




Figure 140 Selecting the Profile and Using the Remove Boolean Type

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 v to create the first extrude-cut feature.

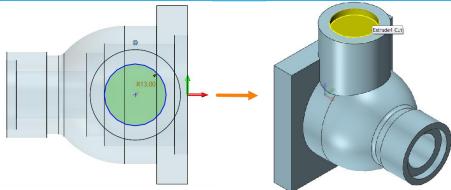


Figure 142 Creating the First Extrude-Cut Feature

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 v to create the second extrude-cut feature.

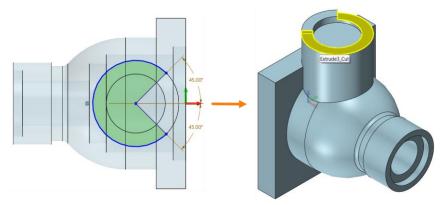


Figure 143 Creating the Second Extrude-Cut Feature

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 v to create screw hole features.

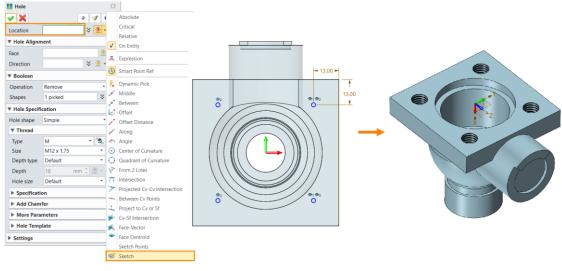


Figure 144 Creating Screw Hole Features

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 v to create another screw hole feature.

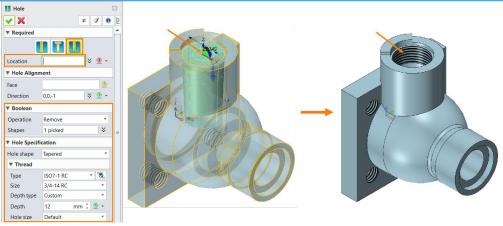


Figure 145 Creating Another Screw Hole Feature

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.

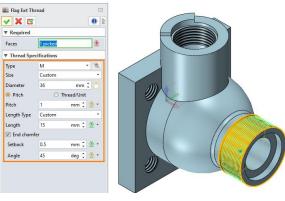


Figure 146 Setting Thread Parameters

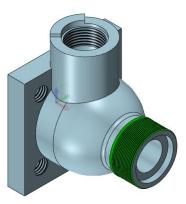


Figure 147 The Thread Feature

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.

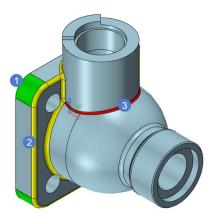


Figure 148 Adding Fillets

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

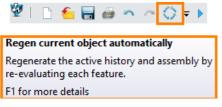


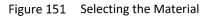
Figure 149 Regenerating the Whole Model

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

	Physical		
Name	Valve Body	Derived from	
Number	1	Designer	Jack
Manager	David	Supplier	хххх
Cost	\$500	Class	Part 👻
Do not sectio	n	Do not ha	tch
🔲 Do not list in	BOM	🔲 Do not list	in Root
Keywords	Valve, Body		
Description	Body of the Valve, fabri	cate inhouse.	
Create time	Friday, May 22, 2020	L	ong date 🔹
Create time Last modified	Friday, May 22, 2020 Wednesday, June 3, 20		ong date *
Last modified		20	

Figure 150 Setting Part Attributes

 Material ✓ X Ľ Require 		23 ()
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Iron-cast Density		
7.079e-06		kg/mm^ ▼
File/Bundle		
Bundles.Z3		
Iron-cast		
🧭 🔒 🗅		



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.

Shape Part and Component *
Face Attributes
▼ Required
Face 1 picked 🗸
▼ Settings
Optional Advanced
Color
R 147 🗘 —————
G 144 🗘 ——————————————————————————————————
B 140 🗘 ———————————————————————————————————
Reset to default
▼ Transparency
50 \$
▼ Shine
Dull Shiny
0 10 0
▼ Gloss
Matte Glossy
90 0

Figure 152 Setting Face Attributes

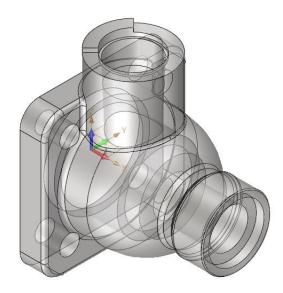
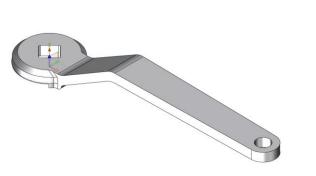


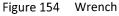
Figure 153 The Final Appearance

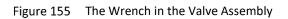
STEP 23 Save the file.

4.5.2 Case 2 – Wrench

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







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.

Valve.Z3 - [Valve Body] 🗙	+						
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0	😨 Create new [Part]					$\overline{\nabla}$	23
	Туре						
				1	¢	\$	
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		Σ					
	CAM Plan	Equation	Set				
	Template		Informatio	n			
	[Default]		Unique Na	me			
	PartTemplate(MM)	2	Wrench				
		_	Description	n			
				3	ОК	Cance	
				O	UK	Cance	

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.

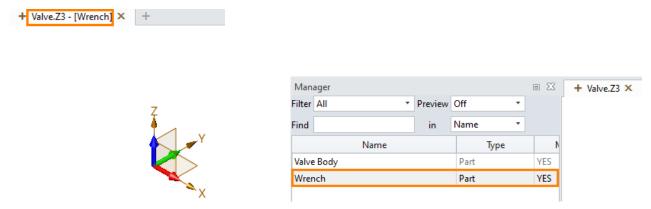


Figure 157 New Part in the Modeling Environment

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**.

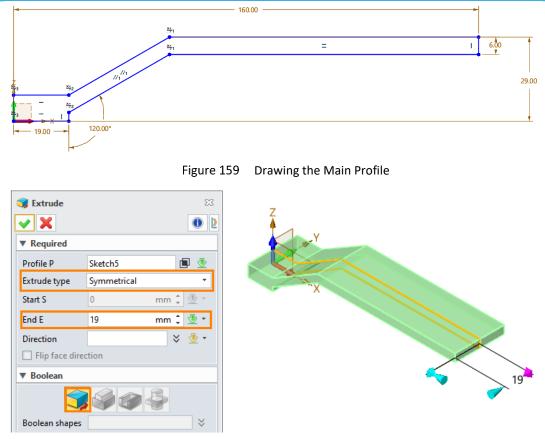


Figure 160 Extruding the Profile

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.

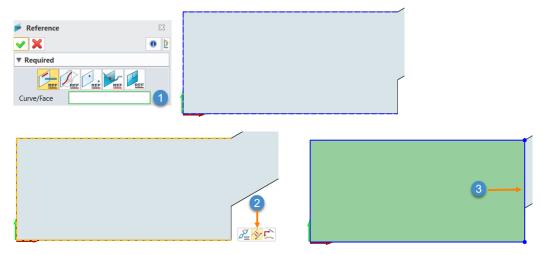


Figure 161 Sketching the Revolve Profile

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

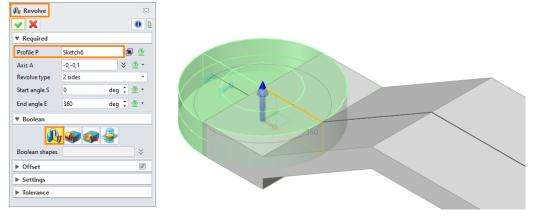


Figure 162 Creating the Revolve Feature

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

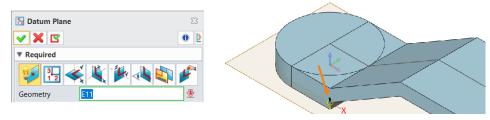


Figure 163 Creating the Draft Datum Plane

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.

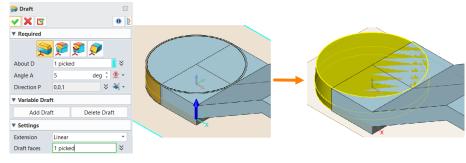


Figure 164 Making the Draft

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 .

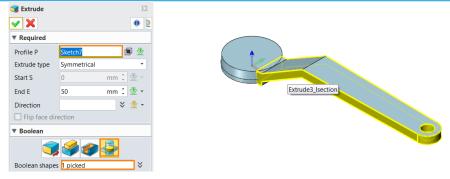


Figure 166 Intersecting the First Extrude Feature

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

Add Shape	Z Y
▼ Required	
Base 1 picked 🗸	
Added 1 picked 🛛 🕹	X
▼ Settings	
Boundary 🗧	
Keep added shapes	
▶ Tolerance	Sec. and

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).

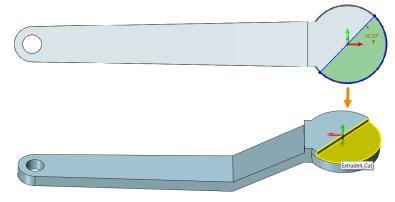


Figure 168 Cutting the Step Feature

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

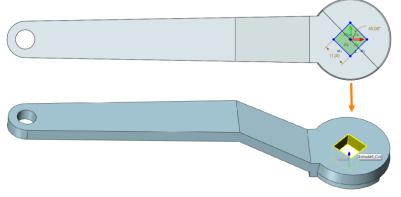


Figure 169 Cutting the Square Hole

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.

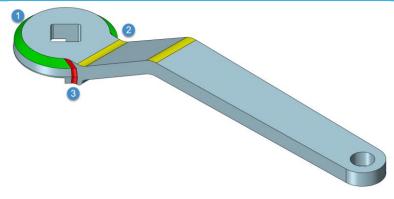


Figure 170 Adding Chamfers and Fillets

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

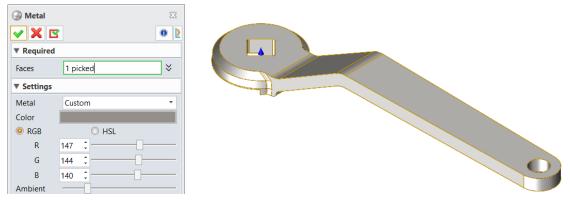


Figure 171 Modifying the Appearance of the Wrench

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

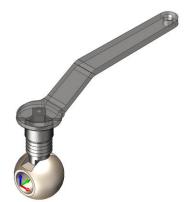


Figure 173 The Valve Core in the Valve Assembly

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.

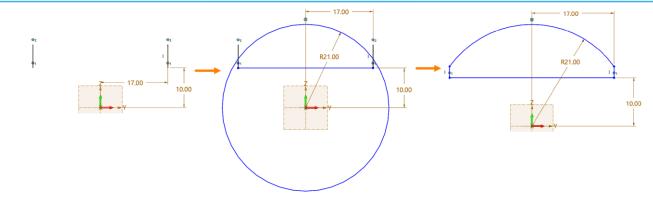


Figure 174 Drawing the Revolve Profile

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

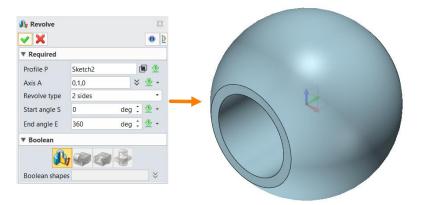


Figure 175 Creating the Revolve Feature

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.

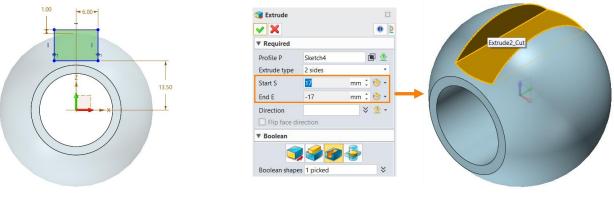


Figure 176 Drawing a Profile

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.

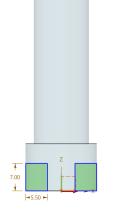
STEP 02 Draw the revolve profile at the origin on the XZ datum plane, constrain it, exit the Sketch environment, and return to the modeling level.



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.



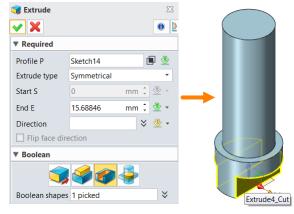


Figure 182 Drawing the Sketch

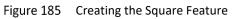
Figure 183 Making the Step Feature

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

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	✓ Required Profile P Extrude type Start S End E Direction Flip face dir Veolean	Required Profile P Sketch15 Extrude type 1 side Start S 0 End E -14.5 Direction * 14.5 Flip face direction V Boolean

Figure 184 Drawing the Square Profile



STEP 07 Click the *Extrude* command, pick the square profile, cut the shape (Depth=14.5 mm), and click v 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 188 The Shim Compressor



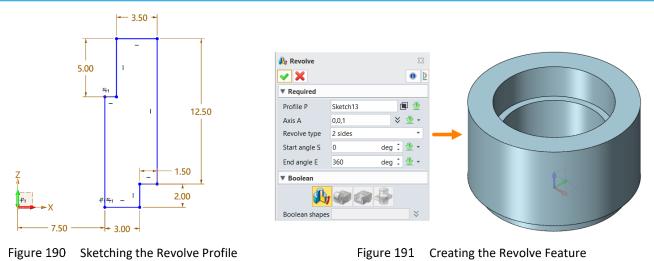
Figure 187 Modifying the Appearance



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.



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.

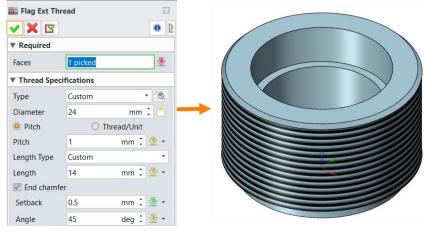
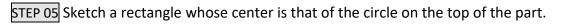


Figure 192 Adding Decorative Threads



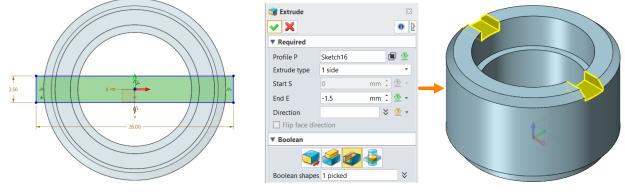


Figure 193 The Rectangular Profile

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 v 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 \checkmark . Modify its appearance and save the file.

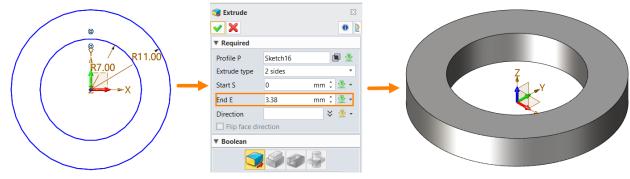


Figure 196 Creating the Bottom Shim

> 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.

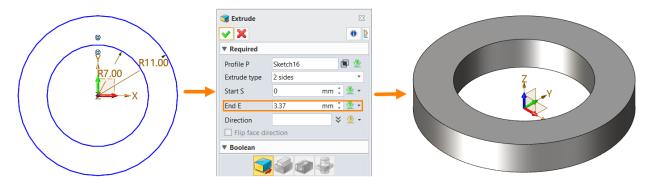


Figure 197 Creating the Top Shim

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.

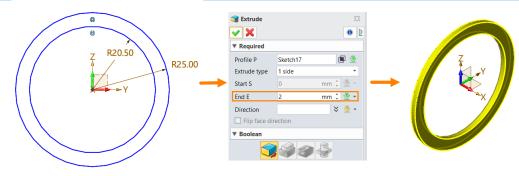


Figure 198 Creating the Adjusting Shim

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 \checkmark . Modify its appearance and save the file.

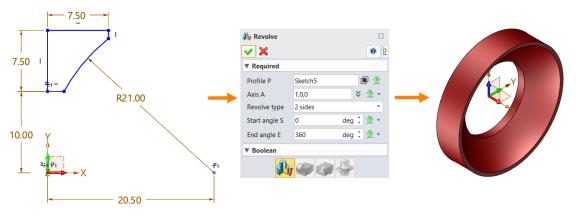


Figure 199 Creating the Seal Ring

5 Assembly

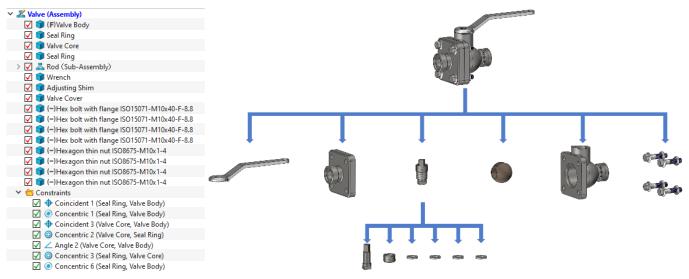


Figure 200 An example of assembly structure

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.

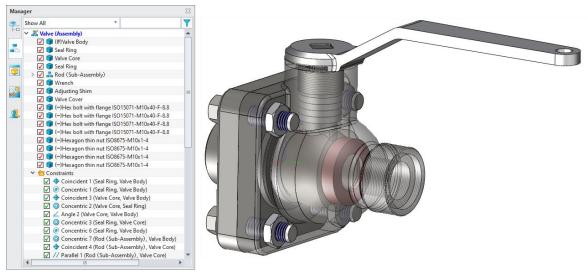


Figure 201 An example of assembly in ZW3D

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.

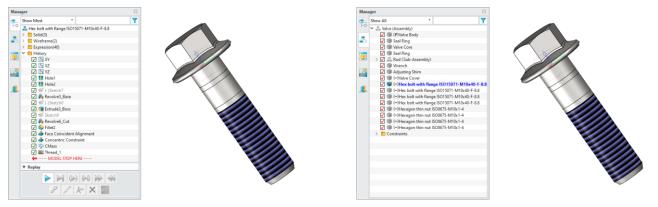


Figure 202 Part (with history manager)

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.

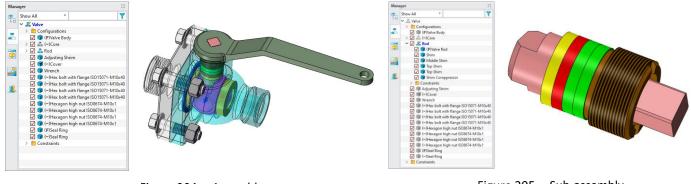


Figure 204 Assembly

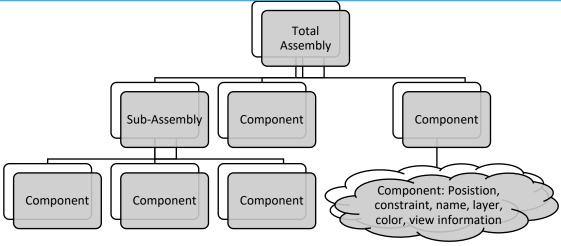
Figure 205 Sub-assembly

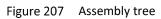
Constraint: In assembly modeling, the spatial position and relative motion of the components can be defined by constraint, then you can analyze whether there is interference between components and whether they are moving normally.

nag	er	Σ
S	Show All	
	✓	4
1	🗹 💠 Coincident 5 (Adjusting Shrim, Valve Body)	
	🗹 💿 Concentric 7 (Adjusting Shrim, Valve Body)	
	Image: Parallel 7 (Valve Body, Valve Core)	
	🗹 💠 Coincident 7 (Seal Ring, Valve Body)	
	🗹 🥥 Concentric 10 (Seal Ring, Valve Body)	
	Image: Parallel 8 (Rod, Valve Core)	
1	🗹 🥥 Concentric 17 (Wrench, Rod)	
	🗹 💠 Coincident 14 (Wrench, Valve Body)	
	🗹 💠 Coincident 15 (Wrench, Rod)	_
•	// Parallel 10 (Wrench, Rod)	
	Coincident 16 (Wrench, XY)	
	Coincident 25 (Hex bolt with flange ISO15071-M10x40, Cover)	
	Concentric 26 (Hex bolt with flange ISO15071-M10x40, Cover)	
	Coincident 26 (Hex bolt with flange ISO15071-M10x40, Cover)	
	Concentric 27 (Hex bolt with flange ISO15071-M10x40, Cover)	
	Coincident 27 (Hex bolt with flange ISO15071-M10x40, Cover)	
	Concentric 28 (Hex bolt with flange ISO15071-M10x40, Cover)	
	Coincident 28 (Hex bolt with flange ISO15071-M10x40, Cover)	
	Concentric 29 (Hex bolt with flange ISO15071-M10x40, Cover)	
	Coincident 29 (Hexagon high nut ISO8674-M10x1, Valve Body)	
	Oconcentric 30 (Hexagon high nut ISO8674-M10x1, Valve Body)	
	Coincident 30 (Hexagon high nut ISO8674-M10x1, Valve Body)	
	Ocncentric 31 (Hexagon high nut ISO8674-M10x1, Valve Body)	
	Coincident 31 (Hexagon high nut ISO8674-M10x1, Valve Body)	
	Ocncentric 32 (Hexagon high nut ISO8674-M10x1, Valve Body)	

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.





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

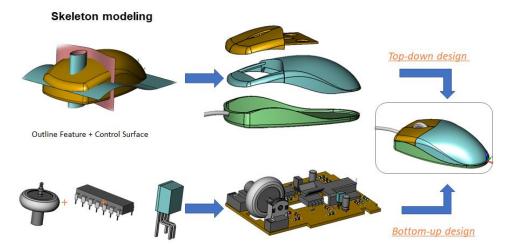


Figure 208 Top-down design and bottom-up design

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.

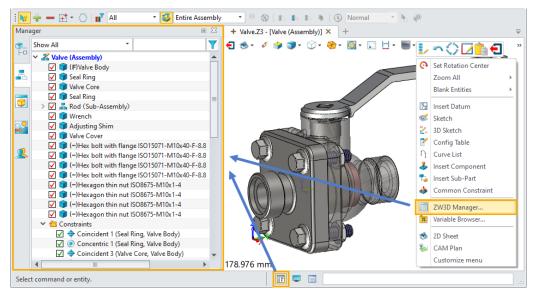


Figure 209 Turn on/ off Manager

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.

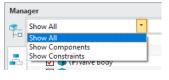


Figure 210 Filter

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.

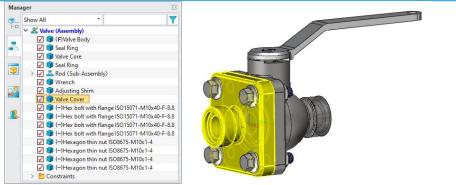


Figure 211 Select/Preview the component

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.

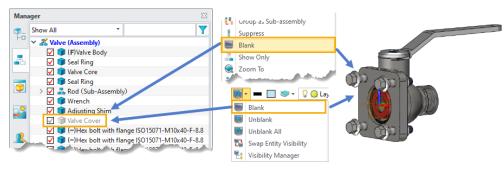


Figure 212 Blank the Component

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.

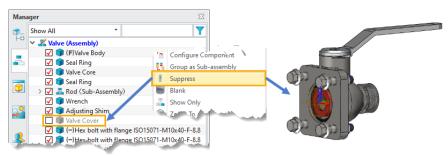


Figure 213 Suppress the Component

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.

	1	Configure Constraint		
✓ ⁴ Constraints	H	Dimension On/Off	~	🔁 Constraints
🔽 💠 Coincident 1 (Seal Ring, Valve Body) 🛶	4	Disable	\rightarrow	Coincident 1 (Seal Ring, Valve Body)
🗹 🧿 Concentric 1 (Seal Ring, Valve Body)	۵	Zoom To		🗹 💿 Concentric 1 (Seal Ring, Valve Body)
🗹 💠 Coincident 3 (Valve Core, Valve Body)	~			Coincident 3 (Valve Core, Valve Body)
	1	Delete		

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.

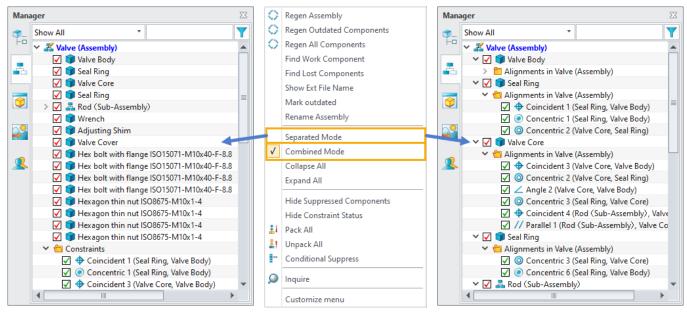


Figure 215 Separated Mode (Left) and Combined Mode (Right)

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.

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Valve Core	2 2	Clone Copy/Move to Layer			
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✓ ♥ Bottom Shim		Configurations	×	_	Valve.Z3,Valve (Assembly)
👤 🧊 Top Shim		Customize menu		Γ	
Constraints	۷	🎦 🗳 🖑 🔮			

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.

Part/Assembly Drawing CAM Sketch Drawing Equation Copy to Delete Clean Import Quick Multi-Import Import Library Blo	File Root							
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Valve Rod Part [Default] Unique Name Wrench Part PartTemplate(MM) Valve	Valve Core	Part	Templat	e		Information		
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Valve	Wrench	Part		-		Unique Name		
Description						Valve		
						Description		
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OK Cancel							OK	Cancel

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.

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Insert Change Edit Merge	Common Mechanical Fix	Edit Pattern	🎭 Insert Sub-Part
• • •	Constraint Constraint	Constraint 🗸	Common Constraint

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.

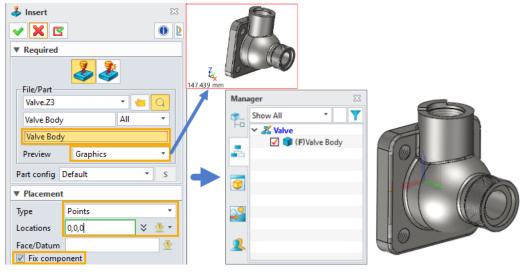


Figure 219 Insert the component

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**.

Image: Wildling to the second seco	
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	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.

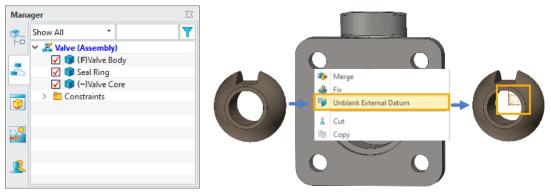


Figure 222 Unblank External Datum

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*.

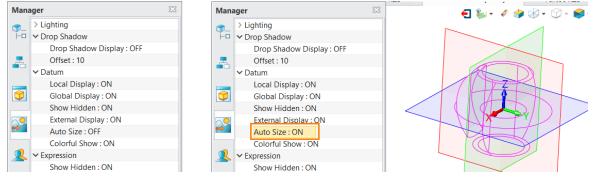


Figure 223 Turn on Auto Size

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.

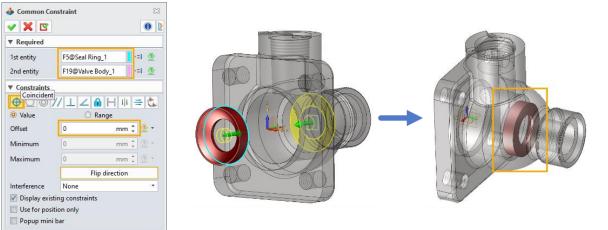


Figure 224 Add coincident constraint

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.

	🕹 Common Cons	straint 🖾			
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		Flip direction			
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	Display existing	ng constraints			
	Use for position	on only			
	🗹 Popup mini b	ar			



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.

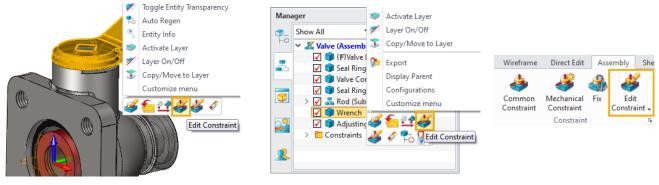


Figure 226 Edit Constraints

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

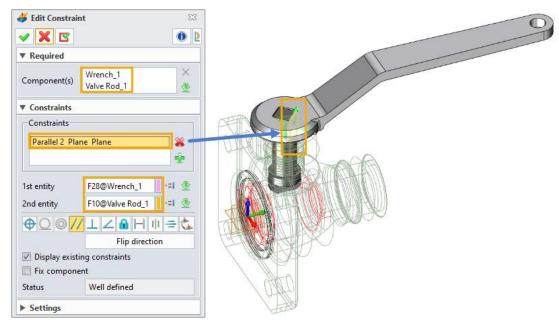


Figure 227 Edit constraint dialog box

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.

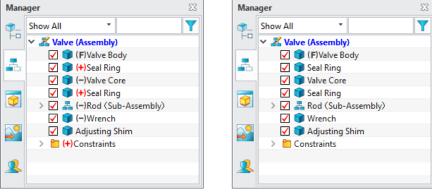
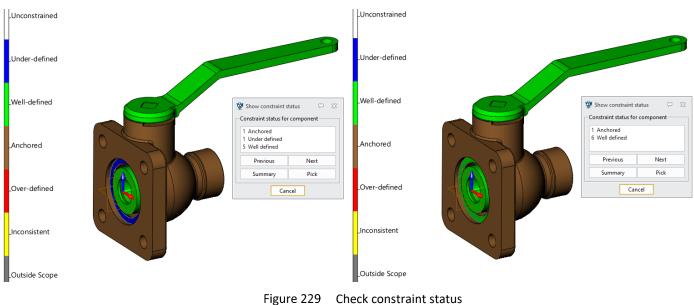


Figure 228 Constraint status

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.



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.

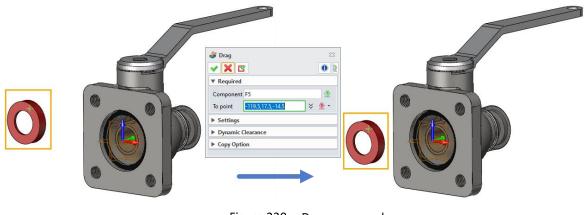


Figure 230 Drag command

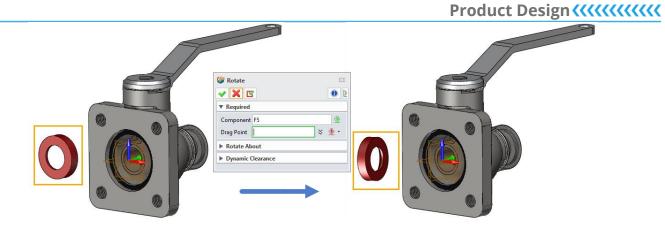


Figure 231 Rotate command

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.

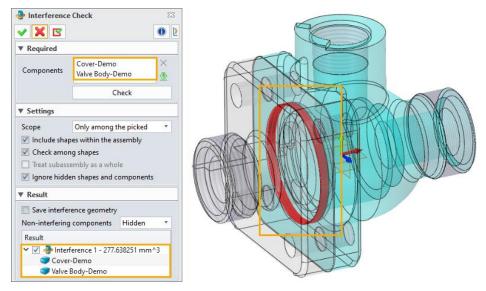


Figure 232 Interference Check

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.

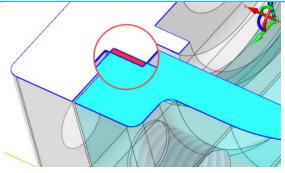


Figure 233 Dynamic section view

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.

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		Name Exploded_view2		
		Time(s)		

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.

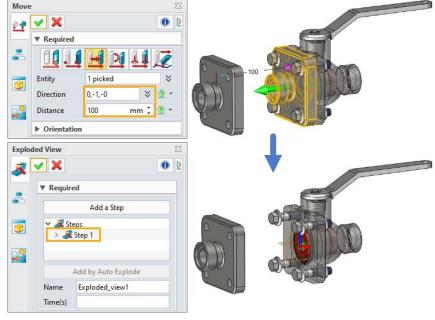


Figure 235 Move entities along a direction

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.

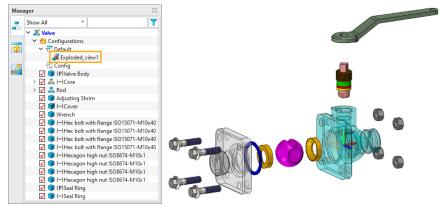


Figure 236 Exploded view

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.

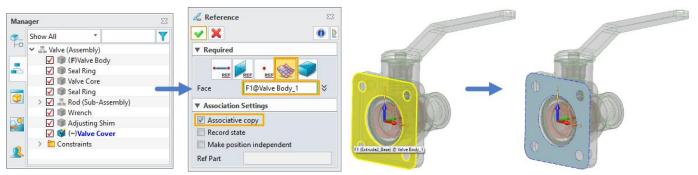


Figure 237 Example of associative reference

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.

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Required		
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Extrude type	2 sides 🔹	
Start S	0 mm 🌲 👲 *	
End E	14 mm 🗘 👲 🔻	
Direction	× 🐠 -	

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.

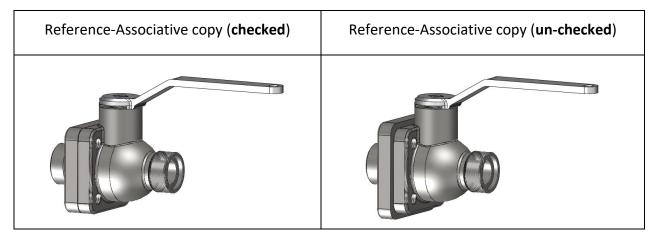


Figure 239 Associative reference with (Left)/without (Right) associative copy after changing the valve body size

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.

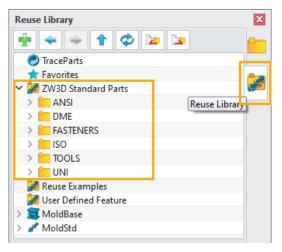
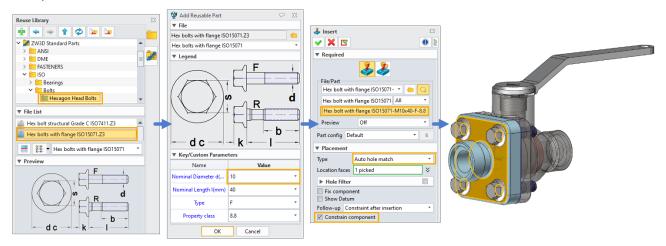
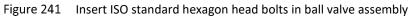


Figure 240 Reuse library

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.





> 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.

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> 💼 😓 - CSN Standards -			IDNR		[~
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> i Automotive engineering			DIN 962 thread			
>			L	70.0	T-	-
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 → 150 2009 - Slotted countersumk flat head screws, product grade A → 150 2010 - Slotted side clountersum k flat head screws (common head style) - product gr → 150 7054-11 - Countersum k flat head screws (common head style) with type H corss flow screws (common head style) with type H corss flow screws (common head style) with type H corss flow screws (common head style) with type H corss flow screws (common head style) with type H corss flow screws (common head style) with type H corss flow screws (common head style) with type H corss flow screws (common head style) with type H corss flow screws (common head style) with type H corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss flow screws (common head style) with type Z corss (common head style) with	ccess - P ccess - Pr reccess eccess eccess eccess type H type Z c		Countersunk	screw ISO 10642 M8x70	4	t N
¹ → 150 2009 - Slotted countersumk flat head screws (common head style) - product gr ¹ → 150 7045-11 - Countersumk flat head screws (common head style) with type I cross i ¹ → 150 7045-11 - Countersumk flat head screws (common head style) with type I cross i ¹ → 150 7045-11 - Countersumk flat head screws (common head style) with type I cross i ¹ → 150 7045-12 - Countersumk flat head screws (common head style) with type I cross i ¹ → 150 7045-12 - Countersumk flat head screws (common head style) with type I cross i ¹ → 150 7045-21 - Countersumk flat head screws (common head style) with type I cross i ¹ → 150 7045-21 - Countersumk flat head screws (common head style) with type I cross i ¹ → 150 7045-22 - Countersumk flat head screws (common head style) with type I cross i ¹ → 150 7047 I - Sotted raised countersumk (scale head screws (common head style) with ¹ → 150 7047 I - Sotted raised countersumk raise head screws (common head style) with ¹ → 150 7047 I - Sotted raised countersumk raise head screws (common head style) with ¹ → 150 7047 I - Sotted raised countersumk raised head screws (common head style) with ¹ →	ccess - P ccess - Pr reccess eccess eccess eccess type H type Z c	* 1	Countersunk	screw ISO 10642 M8x70	4	t H

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.

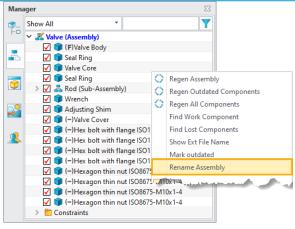


Figure 243 Rename assembly command

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.

顰 Rename Assembly			⊽ ∑		💯 Rename Assembly			₽ %
Old Object Name	Old File Name	New Object Name	New File Name		Old Object Name	Old File Name	New Object Name	New File Name 📥
✓ Valve (Assembly)	Valve - Case Study.Z3				✓ Valve (Assembly)	Valve - Case Study.Z3		
Body	Valve - Case Study.Z3	Valve Body			Valve Body	Valve - Case Study.Z3		=
Valve Core	Valve - Case Study 73	the Contract	A grant and	Ņ.	Valve Core	Valve - Study.Z3	5	
Avi dis Am objects								
Rese	et OK C	Cancel Apply			Reset	ОК	Cancel Apply	

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.

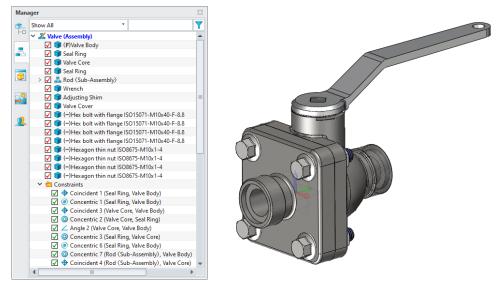


Figure 245 Completed assembly of a ball valve

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.

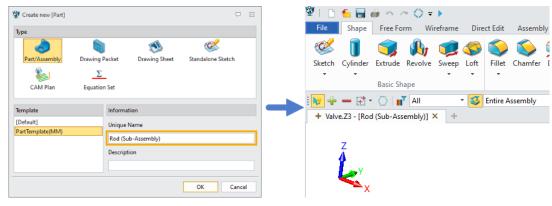


Figure 246 Create a Rod assembly

STEP 02 Insert the first component Valve Rod.

Select Insert command from Assembly ribbon tab.

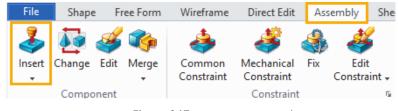


Figure 247 Insert command

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 ($\sqrt{}$) to fix the first component.

↓ Insert ⊠ ✓ X II 0 ↓	
Required File/Part Valve.Z3 Valve.Rod All Valve Rod Valve Rod Vrench Preview Off Part config Default	Manager
▼ Placement Type Points ▼ Locations 1 picked & ③ ▼ Face/Datum XY ④ ▼ Fix 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** ($\sqrt{}$) button to insert the component, then common constraint dialogue box will pop-up.

Required										
	7 7	¥		Man	ager			23		
-File/Part				₽ _	Show All	•		Y		
Valve.Z3					🗠 🔏 Rod (S					
						(F)Valve R				
Bottom Shi		All			V 🔰	(–)Botton	n Shim			- 618
Adjusting										
Bottom Sh	iim			9						
Seal Ring										d li
Preview	Off		-	~						
Part config	Default		• S	•					<	
▼ Placement	t			2						\square
Туре	Points		-		L					
Locations	1 picked	*	: 👲 🕶							

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 Omm in offset and click Apply.

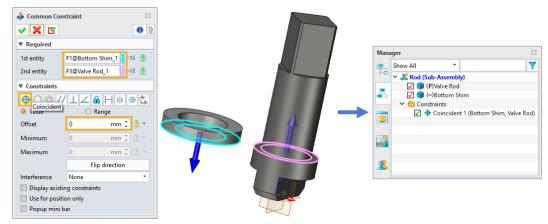


Figure 250 Add the coincident constraint

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.

🕹 Common Constraint 🖂		
✓ X I	Ma	inager 🛛
▼ Required		Show All 🔹 🍸
1st entity F4@Bottom Shim 1		✓ ∡ Rod (Sub-Assembly)
1st entity F4@Bottom Shim_1		🗹 🇊 (F)Valve Rod
2nd entity F4@Valve Rod_1 📑 👲	1	🚽 🗊 Bottom Shim
		✓
▼ Constraints		
		Concentric 1 (Bottom Shim, Valve Rod)
Concentric Flip direction		
Interference None •		
Lock rotation		
Display existing constraints		<u>_</u>
Use for position only		
Popup mini bar		

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.





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.

👻 🗋 🌜 🔚 🧀	🗠 🗠 🕈 🕨		ZW3D 2021 x	64 - [Valve.Z3]	2	6	👻 🗋 🌜 🥁 🗠	~ 🔿 🔻 🕨 ZW3D 2021 x	64 Part - [Valve.Z3 - [Valve (Assembly)]]	- = 23
File Root				 Find a comman 	d 🔾 🛞 🙆 = 🗗	<	File Fre Wir D	Dire Ass SheetWeld Point Data Exc	Vi I Ele A Find a command C) 💮 😧 - æ×
Part/Assembly	Sketch	🕵 Copy to File	🔹 🔗 Import 🛛 👌 Imp	ort Config 👻 📝 Library Publ	isher 🕶 😫 🥂 🔽 🛅		insert 🔹 🦚 Merge 🔹	🕹 Common Constraint 🛛 🗳 Edit Constra	int 🕶 式 👼 🥥 🖉 🦉 👙 🗞 🕶	£ • 😹 •
👏 Drawing Sheet 🐚	Prawing Packet	😴 Delete	🍖 Quick Import		🖾 🐾 🦅		🔁 Change	ቆ Mechanical Constraint	😂 🛄 🛷 👘 🚽 🗮	
الله 🕹 CAM Plan	Equation Set	1 Fix Objects	 Multi-Import 		🗟 🌭		🍜 Edit	I Fix	+ 😜 🛛 🕂	
Insert Obj	ject	Edit Object	Data Exchange	5 Library	Utilities A		Component	Constraint	Basic Editing S Inquire Ani	Refer Expl
Manager		- 22 +	Valve.Z3 × +			;	1 😿 🕂 🗕 🖽 • 🔿 I	👔 All 🔹 🔇 Entire Assembly	🔹 🕾 🎼 🏗 👘 🚯 Normal	* h ₀ ₁ 0
Filter All	* Preview Off	•				1	+ Valve.Z3 - [Valve (Asse	mbly)] × +		Ŧ
Find	in Name	¥	Create new [Part]		□ 23		E 🗧	🎍 🖌 🤌 🌒 e 🖓 e 🧶 e 🔯 e 📰	💾 🕶 🔚 👻 🖛 🔲 🐲 🔹 💡 🥥 Layer0000	*
Name 🔺	Type	Modified 1	ype							
	Part				<u>د</u>					
	Part			J 🖓						
Rod (Sub-Assembly)	Assembly		5 m	ig Packet Drawing Sheet	Standalone Sketch					
Seal Ring	Part		- Lei -	Σ			1			
Shim Compressor	Part			tion Set						
Top Shim	Part							R	R	
Valve Body	Part		emplate	Information				4	\$	
Valve Core	Part		Default1					\checkmark	2	
Valve Cover	Part		PartTemplate(MM)	Unique Name						
Valve Rod	Part		arciemplace(wild)	Valve (Assembly)						
Wrench	Part			Description						
							z			
							Ly.			
4					OK Cancel		~ ×			
	ی 😓	15 Ø			OK Cancel		200 mm			
Select new command.			📊 💻 🗐				Select command or entity.	II 🗢 I	3	

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.

▼ Required		
File/Part	Manager 🔀	
Valve Body All *	Show All	
Top Shim	Valve (Assembly)	
Valve Body	F)Valve Body	
Valve Core		the states
Preview Off -		
Part config Default S		
▼ Placement		
Type Points 🔻		
Locations 1 picked 💝 🥸 🔻	2	
Face/Datum		
Fix component		
Show Datum		
Follow-up Constraint after insertion *		H
Constrain component		
Orientation Reset XYZ Flip Rotate		

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.

🕹 Insert 🛛	
▼ Required	
33	Manager 🔯
File/Part	
Valve.Z3 👻 🖕 📿	Valve (Assembly)
Seal Ring All 🔻	(F) Valve Body (-) Seal Ring
Rod (Sub-Assembly)	Image: A state of the state
Seal Ring	
Shim Compressor	
Preview Off •	
Part config Default S	
▼ Placement	
Type Points -	
Locations 1 picked 💝 🖑 🕶	
Face/Datum	
Fix component	

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 Omm in offset and click Apply.

🚸 Common Con	straint 🛛	
🗸 🗶 🖪	0	
▼ Required		
1st entity	F5@Seal Ring_1	
2nd entity Constraints	F19@Valve Body_1	
	(⊥∠⋒Hѱ=&	
Value	🔘 Range	F5 Ø Seal Ring - F19 @ Valve Body
Offset	0 mm 🗘 垫 🔻	
Minimum	0 mm 🗘 🥸 👻	
Maximum	0 mm 🗘 🥸 👻	
	Flip direction	County of the second
Interference	None 🔻	
🔽 Display existi	ng constraints	
Use for positi	on only	
Popup mini b	bar	

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*.

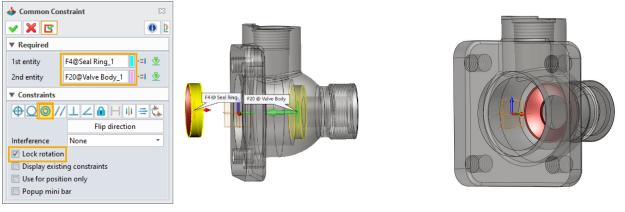


Figure 258 Add the concentric constraint

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.

Common Cons	straint	23		
🗸 🗶 🖪		0	2	
Required]	
1st entity	Default CSYS_XY@	Valve Core_1 📘 📲 👲		A A A A A A A A A A A A A A A A A A A
2nd entity	Default CSYS_XY@	Valve Body_1 🔄 💷		
▼ Constraints				
0	◎ // ⊥ ∠ 🔒	⊢⊪ = 🐍		
Value	🔿 Rang			
Offset	0	mm 🗘 💆 🔻	+	
Minimum	0	mm 🛟 💇 *		
Maximum	0	mm 🗘 🖄 *		
	Flip	direction		
Interference	None	170		
Display existin	ng constraints			
Use for position	on only			
Popup mini b	ar			

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.

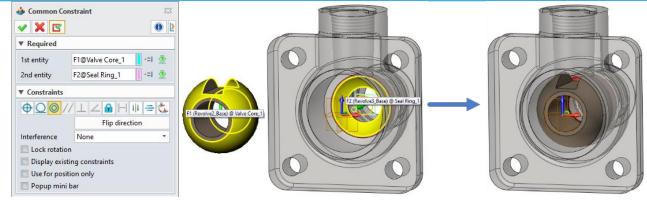


Figure 262 Add the concentric constraint

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.

👍 Common Constrai	nt		83			-		
🗸 🗶 🖸			0 2					
▼ Required								
1st entity	Default CSYS_YZ	@Valve Core_1 📘 🕫	1 🕸				-	
2nd entity	Default CSYS_XZ	@Valve Body_1	1 🕸		400			
▼ Constraints							CA E	5
⊕ Q©	//⊥∠ ⋒	H III = 🐍				T	1 and 1	11
O Value	© Ra		_			F		
Angle	0	deg 🗘	٠ 🖄					
Min angle	0	deg 🗘	٠ 💆					
Max angle	90	deg 🌲	🕸 •			IC		-
	F	lip direction					11/X	
Interference	None		•			1	20	
Display existing c	onstraints			T	Tex	X		-
Use for position o	nly				()			1
🔲 Popup mini bar					\sim			

Figure 263 Add the angle constraint

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

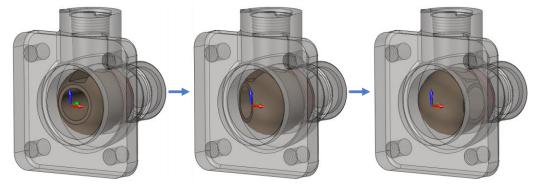


Figure 264 Rotate the Valve Core

> 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.

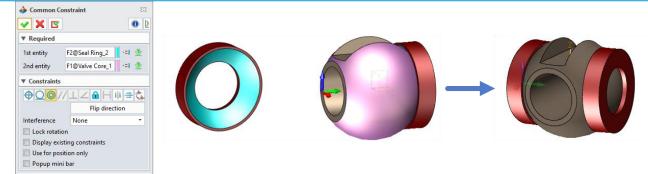


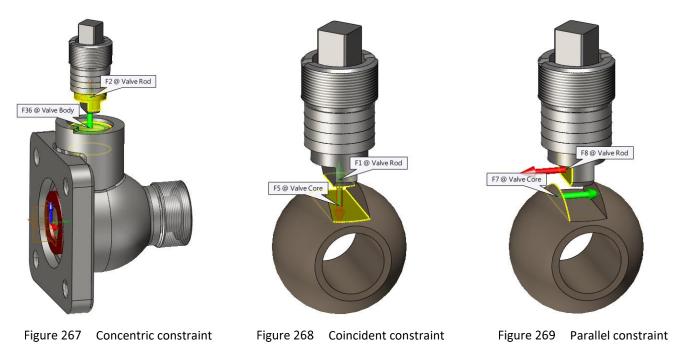
Figure 265 Add the first concentric constraint for seal ring

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.

🕹 Common Cons	straint 🖾	
Required 1st entity	F4@Seal Ring_2	and the second s
2nd entity	F23@Valve Body_1	
▼ Constraints	/⊥∠⋒⊢⋓≡‱	
Interference	Flip direction	
 Lock rotation Display existin Use for position Popup mini b 	ng constraints on only	

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 **Omm** 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**.



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

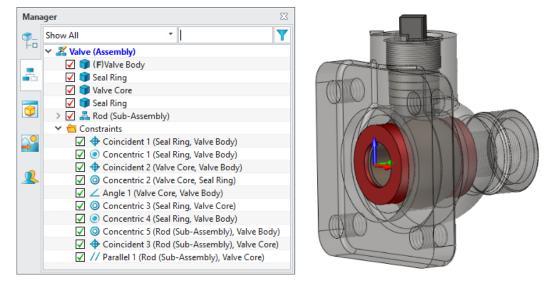
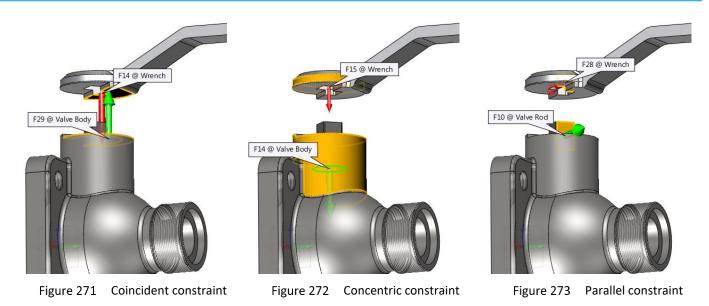
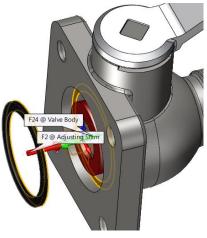


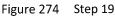
Figure 270 Constraint status in assembly manager

- > 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.



- 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 Omm 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.





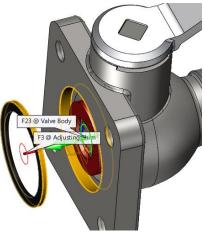


Figure 275 Step 20

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

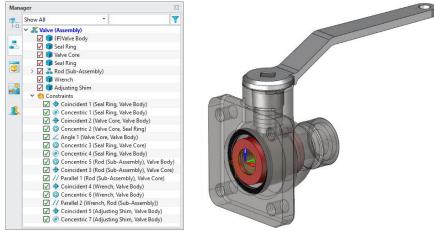


Figure 276 Valve assembly

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.

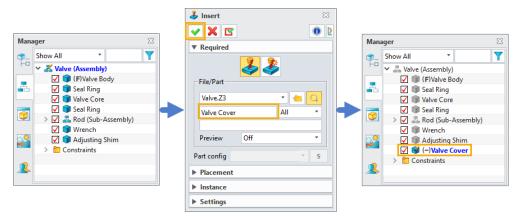


Figure 277 Create the Valve Cover in Insert panel

> 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.

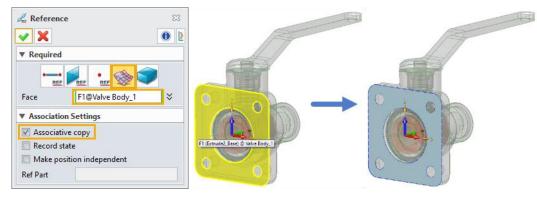


Figure 278 Create face reference

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.

🕃 Extrude	× 1	
▼ Required		
Profile P	F1@Valve Cover_1 🔳 👲	
Extrude type	2 sides 🔹 👻	
Start S	0.5 mm 🌲 👲 *	
End E	14.5 mm 🗘 💆 🕶	
Direction	× 👲 -	
Flip face di	rection	

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.

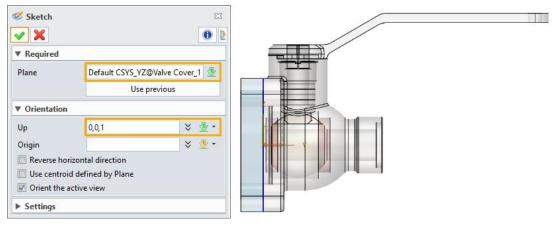


Figure 280 Create a sketch in Valve Cover

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

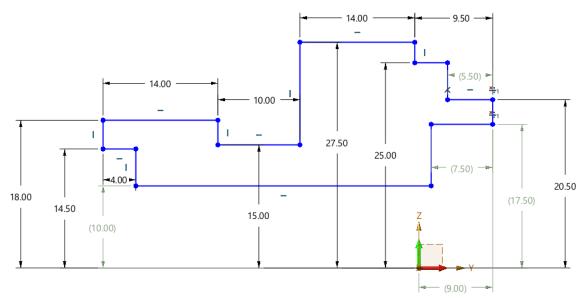


Figure 281 Create a sketch on Valve Cover

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**.

Ny Revolve	83	
🗸 🗙 🖪	0	
▼ Required		
Profile P	Sketch1@Valve Cover_1 🔳 👲	
Axis A	-0,-1,-0 🛛 💥 🔮 🕶	
Revolve type	2 sides 🔹	
Start angle S	0 deg 🗘 🔮 👻 🕶	
End angle E	360 deg 🗘 🔮 👻	
▼ Boolean		
Í.	h a a a	
Boolean shapes	S2@Valve Cover_1 🗸 🗸	

Figure 282 Revolve the sketch

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

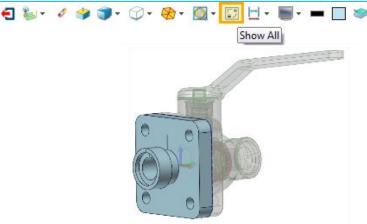


Figure 283 Show all

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.

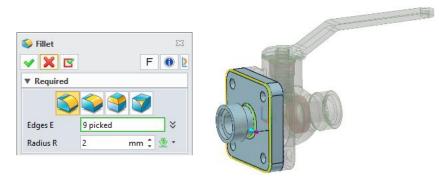


Figure 284 Add the fillet

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.

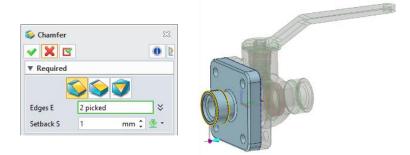


Figure 285 Add the chamfer

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.

Mana	ager 🛛
¶_ ⊢□	Show All Value (Assembly)
	V 🗊 (F)Valve Body
÷.	🗹 🇊 Seal Ring
	🗹 🇊 Valve Core
1	🗹 🇊 Seal Ring
	> 🗹 🚣 Rod (Sub-Assembly)
	🗹 🇊 Wrench
2	🗹 🧊 Adjusting Shim
	☑ 🇊 (−)Valve Cover
2	✓
	Coincident 1 (Seal Ring, Valve Body)
	Concentric 1 (Seal Ring, Valve Body)
	Coincident 2 (Valve Core, Seal Ring)
	Coincident 2 (XY, Valve Core)
	Coincident 3 (Seal Ring, Valve Core)
	Concentric 3 (Seal Ring, Valve Body)
	Coincident 4 (Rod (Sub-Assembly), Valve Core)
	 Concentric 4 (Rod (Sub-Assembly), Valve Body) // Parallel 1 (Rod (Sub-Assembly), Valve Core)
	Angle 1 (Valve Core, YZ)
	Coincident 5 (Wrench, Valve Body)
	✓ Concentric 5 (Wrench, Valve Body)
	V Parallel 2 (Rod (Sub-Assembly), Wrench)
	Coincident 6 (Adjusting Shim, Valve Body)
	Oncentric 6 (Adjusting Shim, Valve Body)

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.

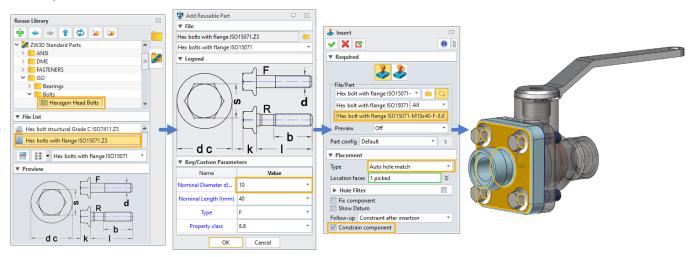


Figure 287 Insert hexagon head bolts from reuse library

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.

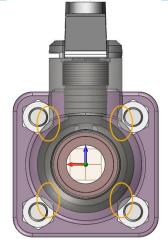


Figure 289 Interference between standard parts and valve body

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.

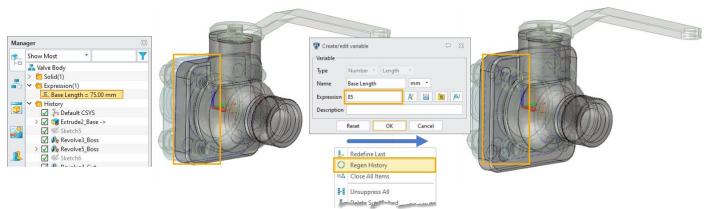


Figure 290 Change the base length of valve body

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.

lana	iger		23
	Show All		Y
1-0	✓ ∡ Valve (Assembly)		
*	 ✓ (F)Valve Body ✓ (F)Valve Body ✓ Seal Ring ✓ Valve Core ✓ (P) Seal Ring 	Ö	Regen Assembly Regen Outdated Components Regen All Components
T	 And (Sub-Assembly) Wrench Adjusting Shim 		Find Work Component Find Lost Components Show Ext File Name

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.

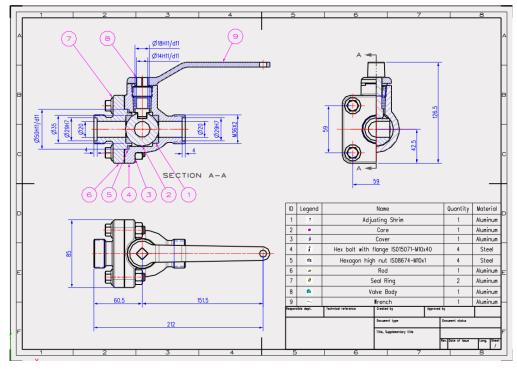


Figure 292 2D drawing in ZW3D

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.

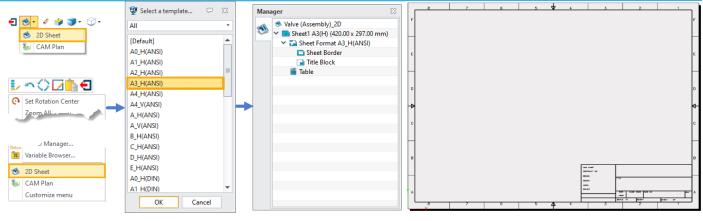


Figure 293 Create a new 2D sheet

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.

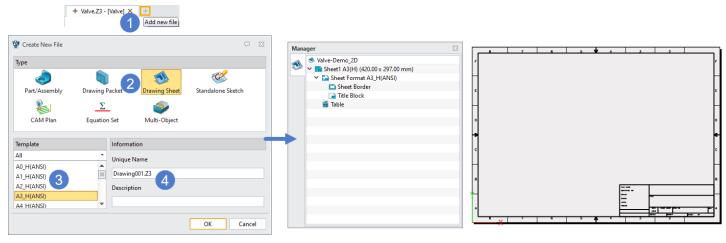


Figure 294 Create a new 2D sheet

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.

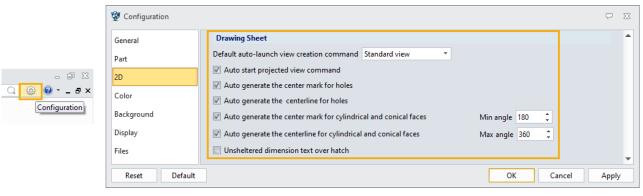


Figure 295 Configuration

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.

		Drawing Setting:	5	~	23
		Units	mm		-
		Mass units	kg		٠
Dimension Drawing Tools In	nquire	Grid spacing	5		
	π	Type of projection	By Standard		٠
Origin Step Size	Variables Browser	Projection tol	0.01		
Settings	browser	Reset	OK Canc	el	

Figure 296 Preferences

Tools ribbon tab -> Attributes Panel -> Style Manager

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

	😵 Style Manager		23 Q
	Current Document * All	Drafting Standard	
	V & Standards	Activate	New Save Reset
	✓	General View	
	✓ ★ Point	Style Type Availal	ble Styles (One for the standard)
sion Drawing Tools Inquire	Default - Active ✓ (Line Default - Active ✓ Text Style (ANSD - Active ✓ Hatch Hatch Style (ANSD - Active ✓ (Line Active) ✓ (Line Activ	Leader label Leader Style (AN) Balloon label Leader Style (BO) Datum fasture Leader Style (BO) Datum target Feature control symbol Centerline/center mark Leader Style (GB) Linear Angular Radial/diametric Arc length Charmfer Hole callout User Table BOM, Table Hole, Table Ebb BM, Table Structural BOM	
		Weid Table Base View Projection View Auxiliary View Section View Detail View Break Line Weid Caterpilar End treatment V	
	Import Export all		Apply Cancel OK

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.

Mana	ager	23	Sheet Name Sheet1 Scale 1/1
1	 Valve (Assembly)_2D ✓ Sheet1 A3(H) (420.00 × 297 ✓ Sheet Format A3_H(ANS Sheet Border Title Block Table 		Description Display paper color Display sheet shadow Start label Next section view label A Next detail view label A Next datum label
	<u> </u>		Associated model -DEFAULT- none OK Cancel

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.

Valve_2D Sheet1 A3(H) (420.00 x 297.00 Sheet Format A3_H(ANSI) Sheet Border Title Block) Add Sheet Border	X	Template A0_H(ANSI) A1_H(ANSI) A1_H(ANSI) B1 A2_H(ANSI) B1 A4_H(ANSI) B1 A4_H(ANSI) A4_H(ANSI)	Custom Sheet Size A3(H) (420.00 × 29 Width 420 Sheet Format Configuration <custom></custom>		t 297	14
Valve_2D Sheet1 A3(H) (420.00 x 297.00 Sheet Format A3_H(ANSI) Sheet Border			X	A1_H(ANSI) A2_H(ANSI) A3_H(ANSI) A4_H(ANSI) A4_V(ANSI) A_V(ANSI) A_H(ANSI)	Width 420 Sheet Format Configuration <custom></custom>			1.4
Valve_2D Sheet1 A3(H) (420.00 x 297.00 Sheet Format A3_H(ANSI) Sheet Border				A2_H(ANSI) A3_H(ANSI) A4_H(ANSI) A4_V(ANSI) A_H(ANSI)	Sheet Format Configuration <custom></custom>	Heigh		1.64
Sheet1 A3(H) (420.00 x 297.00 Sheet Format A3_H(ANSI) Sheet Border				A4_H(ANSI) A4_V(ANSI) A_H(ANSI)	Configuration <custom></custom>			1.1
Sheet Format A3_H(ANSI)			-	A4_V(ANSI) A_H(ANSI)			1 🖬 3	1.14
Sheet Border		Add Sheet Border		A_H(ANSI)				
		Add Sheet Border			Sheet Border			1.04
📄 Title Block				A_V(ANSI) B_H(ANSI)	Use Border			
		Add Title Block		C_H(ANSI)				
🎬 Table		Add Additional List Block		D_H(ANSI) *	Bound Custom Marg	in		
					Trimming Mark		Centering I	Mark
		Add Code List Block		Preview	Partition			
		Sheet Format Attributes			Horizontal	8	Length 52.500	
		Sheer Format Attributes			Vertical	6	Length 49.500	
		Expand Subitems			Insert block			
		Collapse Subitems			Tala	Tab	a blash/ANSI man)	
					licite	10	e biock(Arks(_mm)	
	_	Customize menu		^	Code List	<n< td=""><td>one></td><td></td></n<>	one>	
					Additional List	<n< td=""><td>one></td><td></td></n<>	one>	
				Width: 420.0mm				
			Add Code List Block Add Code List Block Sheet Format Attributes	Add Code List Block Sheet Format Attributes Expand Subitems Collapse Subitems	Add Code List Block Add Code List Block Sheet Format Attributes Expand Subitems Collapse Subitems	▲ Add Code List Block ▲ Sheet Format Attributes Expand Subitems Collapse Subitems Customize menu Height: 297.0mm	Add Code List Block Collapse Subitems Collapse Subitems Customize menu Height: 297.0mm	▲ Add Code List Block ▲ Sheet Format Attributes Expand Subitems Collapse Subitems Customize menu Height: 297.0mm

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 dropdown list and set other parameter such as scale.

🧧 Standard 🛛 🖾	View RIGHT •	
	Location 186.884,336.792 😵 🔮 🔻	
▼ Required	▼ Settings	
File/Part	Advance	
Valve.Z3 🔹 🤄 📿		
Valve (Assembly) All 🔻	Shov. sc	┟╴┲╶╢╴╼╫╬╫╶┝╴┾╢╶╫╏╸┙
Rod (Sub-Assembly)	Scale type Use custom scale 🔻	
Valve (Assembly)	◎ X/Y 1 ‡ : 1.5 ‡	└ ╢ ╬╬ <mark>╞╼╬</mark> ┠┙

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.

Projection 🖾							
Required							
Base view	#875	65	₫				
Location	376.0	67,331.218	> 👲 🔹				
▼ Settings							
Projection		3rd angle	-				
Dimension	type	Projected	•				

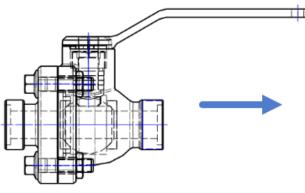
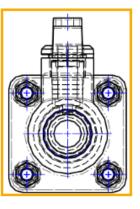


Figure 301 Create the projection view



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.

E Layout	▼ Settings	General Label Lines Comp	Inherit PMI
✓ X	Optional Advanced		Show scale
▼ Required	Projection 3rd Angle		Scale type Use custom scale 🔻
File/Part	Layout	🗣 🊫 🎝 IH	● X/Y 1 ÷ 3 ↓ ○ X.X 0.333333 ↑
Valve.Z3 🔹 🦕			 ○ X . X 0.333333 ↓ ✓ Synchronize sheet scale
Valve (Assembly) All 🔻		🕑 🖤 🖤 🗖 🗛 👘	Show label
Rod (Sub-Assembly)		$\sim \downarrow \ll $	Label
Valve (Assembly)			

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.

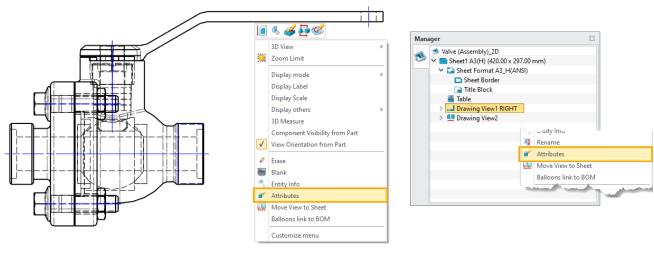


Figure 303 View attributes

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

💯 View Attributes 🗢 🖂	Scale type	Use custom scale	-	🖞 View Attributes 🖓 🖄 🖞 View Attributes 🖓 🖄
Style <from standard=""> *</from>	◎ X/Y ○ X.X	1 1: 1.5	5	Style <from standard=""></from>
General Label Lines Text Comp	Synchronize :			General Label Lines Text Comp General Label Lines Text Comp
🛛 🖓 🇊 🐉	Show label			Individual Lines
	Label	RIGHT	π	Visible Show shape
🔶 🏠 🎝 IH				Hidden V 🚣 Valve (Assembly)
				Tangent 📝 🗊 Valve Body
	All Off	Teggle	On	Tangent Hidden
	AILOIT	Toggle All	on	Color 🛛 🐨 🐨 Valve Core
				Seal Ring
│	Layer La	ayer0000	-	Line type V V Assembly
				Line width 0.25mm Valve Rod
Inherit PMI				Cine width Shim
Show scale	1	OK Cancel	Apply	Layer Layer0000 Component visibility from part

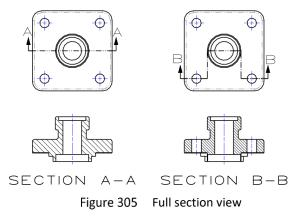
Figure 304 View attributes

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.



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.

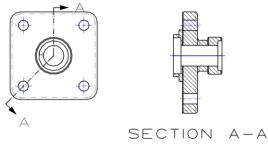
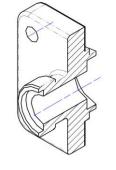


Figure 306 Aligned section view

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.



SECTION A-A

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.

File	Layout	Dimension	Drawin	g Tools	Inquire			
Dimension				Linear Chamfer	Angular	✓ Radial/Diametric	Arc Length	Hole Callout
				Dimensi	ion			

Figure 311 Dimension ribbon tab

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

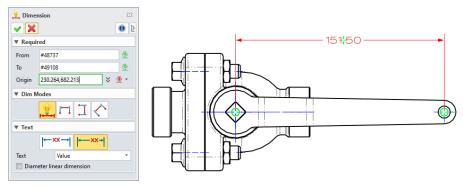
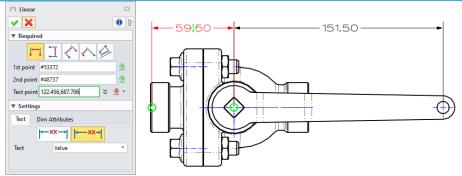


Figure 312 Quick dimension





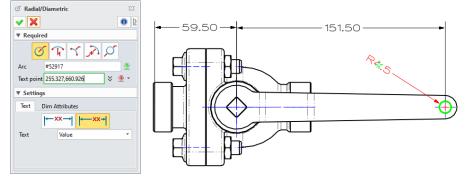


Figure 314 Radial/Diametric dimension

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.

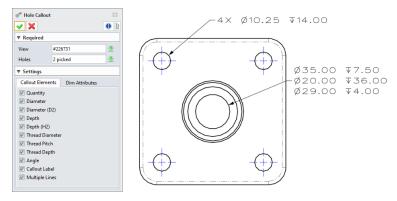


Figure 315 Hole Callout dimension

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.

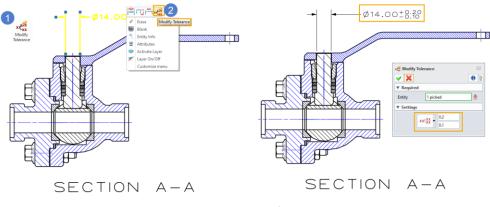


Figure 316 Modify tolerance

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.

Ribbon Appearance Ribbon Tabs) 			. Ø R M	X.8XX X [*] XX [*]
Ribbon Panels	F			Modify Tolerance	
ToolBars	•	Document Aware Toolbars	⊬		1
Styles	Þ	Document Aware Float Tip	►	. 🗸 🗙 🕕 🕑 🖢	
Customize	\checkmark	DimTool		▼ Required	
		Layout		Entity 1 picked 👲	
		Dimension			
		Drawing		▼ Settings	
		Attributes		xx±xx - 0.20	
		Tools		XX-XX 0.10	
		Inquire			

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.

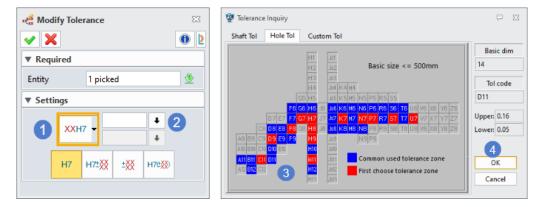


Figure 318 Tolerance inquiry

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.

Center Mark	23	$(- \oplus$	\bigcirc
✓ X	0		
▼ Required		(((−	+
Arc #225284	₫		
Settings		\square	\oplus
		(+	Υ

Figure 320 Center mark

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

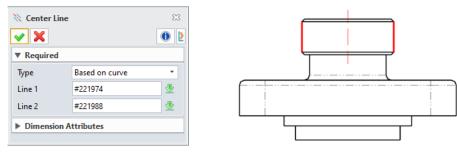


Figure 321 Center line

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

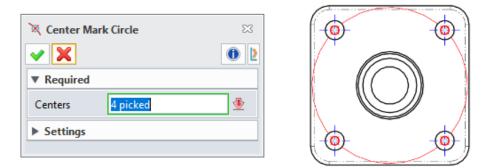


Figure 322 Center mark circle

Datum Feature:

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



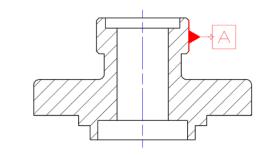


Figure 323 Datum feature

Feature Control Symbol:

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

	🐲 Feature Control Symbol Editor.		\Box X]
	Notation Text1:		+	
✓ X	SymbolTolerance 1	Tolerance 2	Datum	////
i	⊥ - • 0.01 • F	F	A L& B L& L&	
Required		• F	LS LS LS	
FCS text [VxX]x0.01[VxX]xA[VxX]xB[Box]				
Location 2 picked 💝 🖑 🔻				
▼ Leader Points	N ∠ ⊕ 2:		•	
Leader pts 🛛 💝 👻 🔻	◎ =			
Auxiliary location 🛛 🗧 👻 🔹				A
Dimension Attributes	1 11			
			OK Cancel	



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.

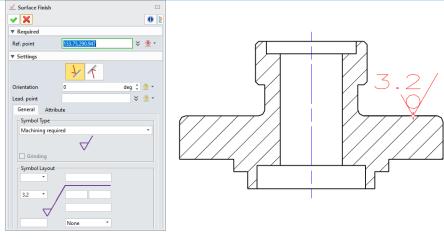


Figure 325 Surface finish

Label/Text:

With this command, you can create a label manually.

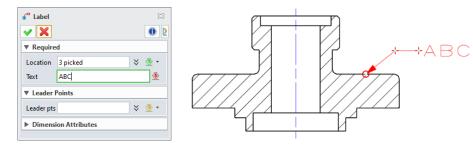


Figure 326 Label

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.

BOM	÷	Α	В	С	D	E	F
	1	ID	Name	Cost	Number	Quantity	Material
	2	1	Adjusting Shim			1	Aluminum
▼ Required View #16045 ③	3	2	Hex bolt with flange IS015071-M10x40-F-8.8		Hex bolt with flange IS015071-M10x40-F- 8.8-M10x40-8.8-F	4	Steel
Name BOM	4	3	Hexagon thin nut IS08675-M10x1-4		Hexagon thin cut ISO 8675-M10x1-4	4	Steel
	5	4	Rod (Sub-Assembly)			1	Aluminum
Top-level only	6	5	Seal Ring			2	Aluminum
O Parts only	7	6	Valve Body			1	Aluminum
O By balloon only	8	7	Valve Core			1	Aluminum
Indented	9	8	Valve Cover			1	Aluminum
Max traverse depth	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.

Available		Selected
Class Designer Supplier Manager Remarks Description Keywords Derived From Start Date		ID Name Cost Number Quantity Material
End Date Mass		
Volume	-	

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.

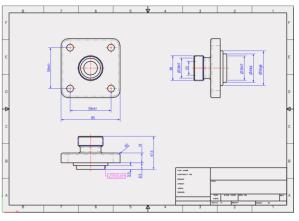


Figure 329 2D drawing of a valve cover

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 rightclick menu, then select A4_H(ANSI) as the template.

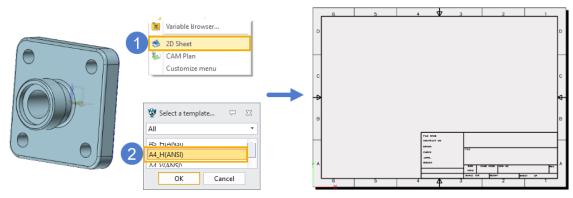


Figure 330 Create the 2D sheet for valve cover

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.

🧧 Standard	View FRONT	
✓ X	▶ Location <u>60,130</u> > <u>●</u> ·	$\varphi \qquad \varphi$
▼ Required		
File/Part	🔄 🔄 unuw state	
1-Valve-Reference.Z3 🔹 🦕	Scale type Use custom scale 🔻	
Valve Cover All	◎ X/Y 1 ‡ : 1.5 ‡	Φ
Valve Cover	○ X.X 0.666667 ‡	Ý Ý

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.

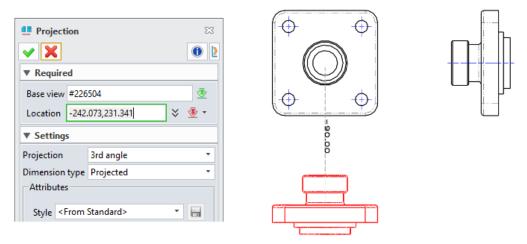


Figure 332 Projection view of valve cover

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.

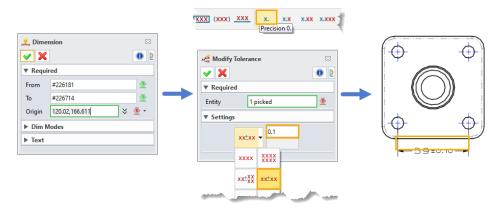


Figure 333 Add the dimension and annotation

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.

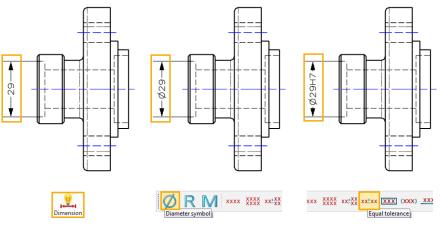


Figure 334 Add the dimension and annotation

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.

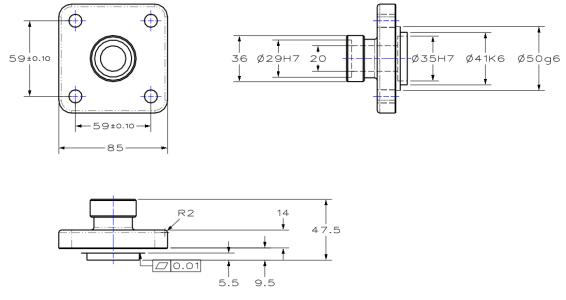


Figure 336 Remaining dimension and symbol

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

🖉 Style Manager						\Box					
Current Document * All	*	Batch edit for dimension style									
> ⊕ [⊕] Datum target		For active style O For all of the style O	displayed styles								
Feature control symbol											
✓ Is Symbols		General Line/Arrow	Text								
> 📉 Centerline/center mark		Layer		Tolerance			Batch edit f	or dimension	styles		
> 📥 Weld		A ative law			Upper		For activ	e style 🔘 For	all displayed s	yles	
>) Caterpillar		Active lay	er *	Туре 🗙 🛪 🔻			General	Line/Arrow	Text		
End treatment		Display			Lower		General	Line/Arrow	lext		
> 🖌 Surface finish		Extension lines	Side 1 and Side 2 ×	Inspection			Use text st	/le	<custom></custom>	. .	
Intersection symbol				Tolerance precision	X.X		-Text Attri	butes			Text Shape
V 😥 Dimensions		Dimension lines	Side 1 and Side 2 🔻	Zero suppression			Foot 7047	D Simplex Ror		↓ B <u>U</u>	
> I ¹ Linear	_	Linear Precision	X.X *	Leading zero	Trai	ling zero		D Simplex Ko	nan		2
> 🔬 Angular		Angular Precision	х.х -	County 2010		ing zero	Sample				→1 ← 0.8
> 🝼 Radial/diametric		Show unit		Zero tol. display	x ^{+0.01}	•		Aae	BbYyZz		ŢXX+
Arc length				Zero toi, display	~-0.00						TXXT 1
> 🌱 Chamfer		Scale factor	1	Alt. Units			Color				→ - 0.3
> 💞 Hole callout		Zero suppression		Use alternate units			Justificati			E.	
> III Ordinate		Leading zero	Trailing zero				Justificatio			=	<u>→1</u> t−− 0
✓ III Tables		Text Position		Position		Right *					
> 🖬 User Table				Unit precision		X.XX -					
> 🛗 BOM_Table			223	Tolerance precision		X.XX -					
> 🔠 Hole_Table				Alternate unit		Millimeters *					
Preview			→-xx -	Alternate unit		Ivillimeters *					
Fleview		Arrow Position	1 1 m			0					
		Force show dimension line	ne	Show unit							

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.

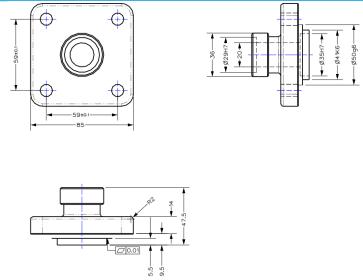


Figure 338 The 2D drawing of valve cover

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.

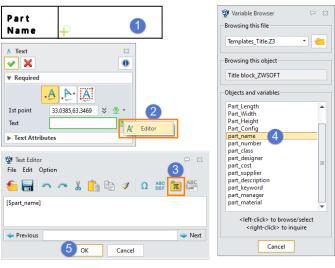


Figure 339 Add text variable on title block

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.

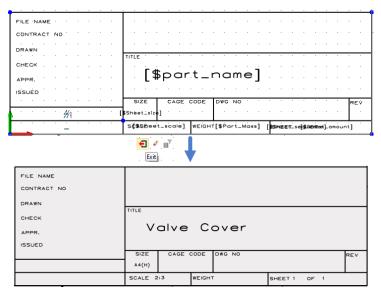


Figure 340 The result of the title block

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.

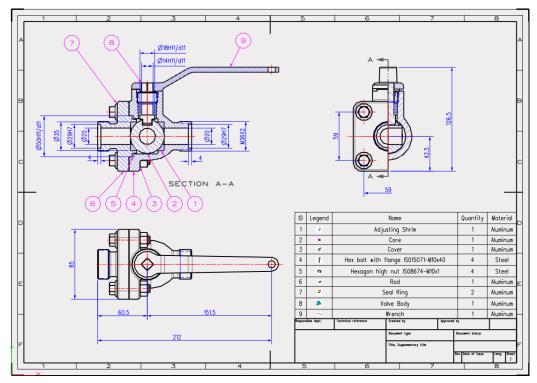


Figure 341 The 2D drawing of the ball valve assembly

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.

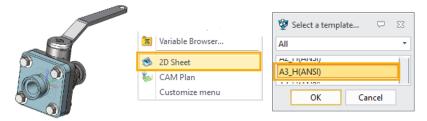


Figure 342 Create the 2D sheet of Valve (Assembly)

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.

Standard		×2		/iew FRO	242,182.435	×	▼ <u>●</u> ▼			
▼ Required File/Part				an a start and		, alternative, e	10-00 K	\rightarrow	\bigcirc	\bigcirc
			nati	v						1
Valve.Z3	•	Q		Scale type	Use custo	om scale	Ŧ			/
Valve (Assembly)	All	-		◎ X / Y	1 🗘	: 1.5	÷ ‡		0	\bigcirc
Rod (Sub-Assembly)				© X.X	0.666667		÷		\square	U)
Valve (Assembly)										

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.

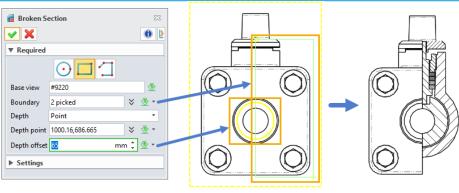


Figure 344 Create the broken section

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).

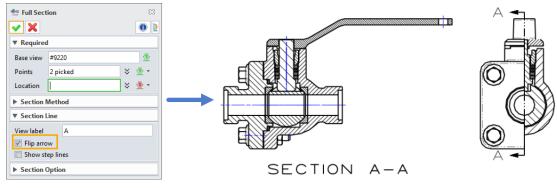


Figure 345 Create the full section

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).

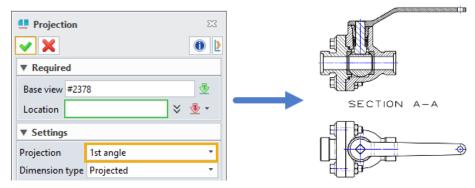


Figure 346 Create the projection view

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.

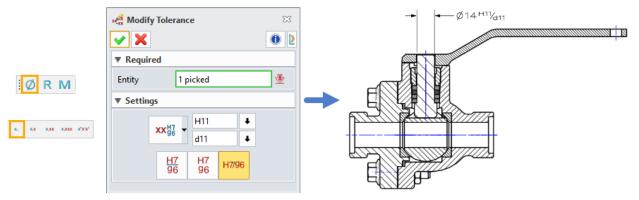


Figure 347 Add the dimension and tolerance

126.5

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

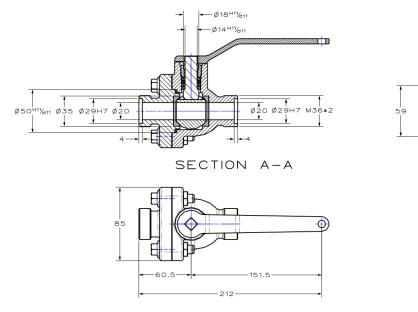


Figure 348 Add remaining dimensions

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

Batch edit for dimension styles		
I For active style \bigcirc For all displayed styles		
General Line/Arrow Text		
Layer Active layer Display	Tolerance Type XX ^{1/7} Upper + Lower +	General Line/Arrow Text Use text style <custom> *</custom>
Extension lines Side 1 and Side 2 * Dimension lines Side 1 and Side 2 * Linear Precision X.X * Angular Precision X.X * Show unit Scale factor Scale factor 1 Zero suppression Image: Constraining zero Leading zero Image: Constraining zero Text Position Trailing zero	H7 96 H796 H796 H796 H796 H796 H796 H796 H796 Leading zero Zero suppression Zero tol. display X.X. At. Units	Visite Kristiolets Font ZW3D Simplex Roman ↓ B ↓ Sample A □ B □ Y Y Z 2 Color Justification 0 0
Arrow Position	Use alternate units Position Right * Unit precision X.XX * Tolerance precision X.XX * Alternate unit Millimeters * 0 Show unit	

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.

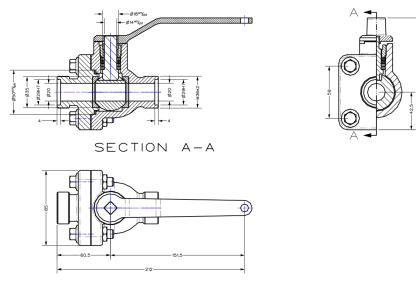


Figure 350 The 2D drawing of ball valve

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.

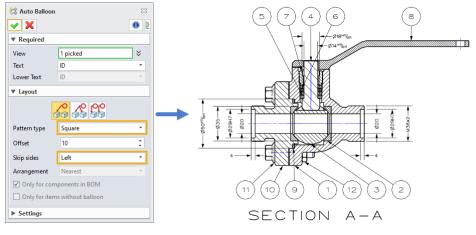
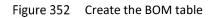


Figure 351 Auto balloon

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.

BOM 23					
▼ Required		Legend	Name	Quantity	Material
View #16045 👲	1	0	Adjusting Shim	1	Aluminum
Name BOM	2	2 1	Hex bolt with flange IS015071-M10x40-F-8.8	4	Steel
Loud and a second	3	5 ന	Hexagon thin nut IS08675-M10x1-4	4	Steel
Inhert 3D . M	4	- 0	Rod (Sub-Assembly)	1	Aluminum
▼ Table format	5	5 0	Seal Ring	2	Aluminum
Available Selected	e	a 6	Valve Body	1	Aluminum
Number	7	7 🔶	Valve Core	1	Aluminum
Class	8	3 4	Valve Cover	1	Aluminum
Designer E Name Cost Quantity	ç)	Wrench	1	Aluminum
Supplier Manager					



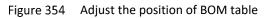
STEP 02 Select all the text in the BOM table and the Cell setting dialog will pop-up, then click More cell attributes and set the Text Shape as shown below.

		Yes Cell #5 *5 # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # # #<	Normal More	cell attributes	
β A	В	C	D	E	Text Shape
1 ID	Legend	Name	Quantity	Material	a.5
2 1		Adjusting Shim	1	Aluminum	→ ← 0.5
32		Hex bolt with flange IS015071-M10x40-F-8.8	4	Steel	
4 3		Hexagon thin nut IS08675-M10x1-4	4	Steel	
5 4		Rod (Sub-Assembly)	1	Aluminum	
5 5		Seal Ring	2	Aluminum	
6		Valve Body	1	Aluminum	
3 7		Valve Core	1	Aluminum]
8		Valve Cover	1	Aluminum	
0 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.

Ŷ	Α	В	С	D	E	
1	ID	Legend	Name	Quantity	Material	L
2	1	0	Adjusting Shim	111	Aluminum	
3	2		Hex bolt with flange IS015071-M10x40-F-8.8	4	Steel	
4	3		Hexagon thin nut IS08675-M10x1-4	4 .	Steel	
5	4		Rod (Sub-Assembly)	· · 1 · ·	Aluminum	
6	5		Seal Ring	2 .	Aluminum	
7	6	<u>\$</u>	Valve Body	1	Aluminum	
8	7		Valve Core	1111	Aluminum	B
9	8	👷	Valve Cover	1	Aluminum	
10	9	1	Wrench	111	Aluminum	1
	FILE	NAME				
	CÓN	TRACT NO				
	DRA	WN				÷
	CHE	ск	TITLE			
	APP	R				
	เร่รม	ép i i i				
			SIZE CAGE CODE DWG NO		REÝ	A
			A3(H)	· · · · ·		
			SCALE 2:3 WEIGHT	SHEET 1 OF	1	
			.3 2		1	



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

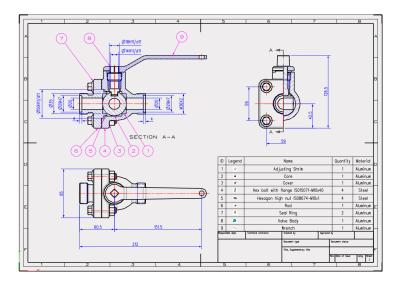


Figure 355 The 2D drawing of the ball valve assembly