ZWCAD Mechanical

Cost-effective CAD

Learn ZWCAD Mechanical 2012 in a Day

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Contents

1. Basic drawing, dimension, and symbols

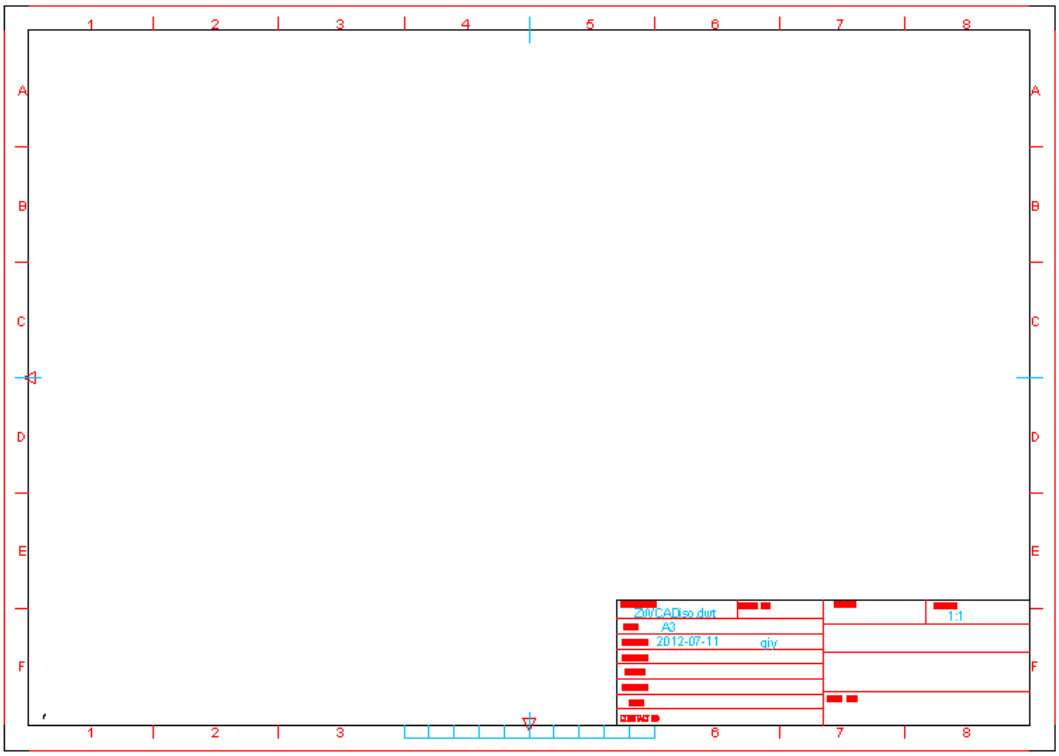
This manual introduces the basic drawing features of ZWCAD MECHANICAL to give users a general understanding of the drawing tools and dimensioning methods.

1.1. Set frame to A3 and 1:1

Click ‘Frame Settings’ under ‘Drawing’ in ZWCADM or enter a ‘TF’ space or carriage return to bring up the dialog box of Frame Settings;

Select ‘A3’ as frame size, and ‘1:1’ as ratio, click ‘OK’,

The command prompt appears: Please choose a new drawing center and the entities for rescale: Select an appropriate location in the drawing environment as the initial position of the frame (the frame will be generated in the origin of coordinates with a directly press of Enter). The Frame Settings are now done. The outcome is as follows:



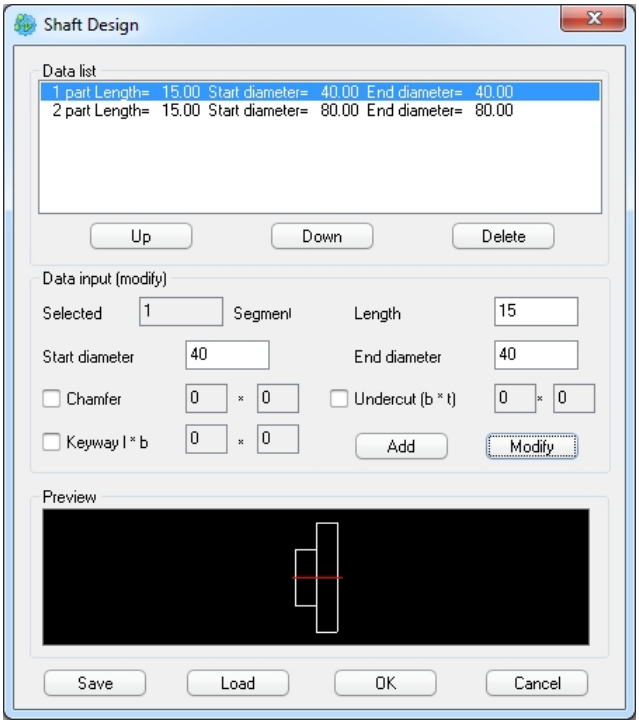
Note: After inputting a command in the command prompt line, press Space or Enter to execute the command, similarly hereinafter without further prompt.

1.2. Drawing a contour

1.2.1. Draw the contour of a part through hole and shaft design

Step 1:

Click the ‘Shaft Design’ in the ‘Part design’ under ‘ZWCADM’ menu, or enter: ZWMSHAFT



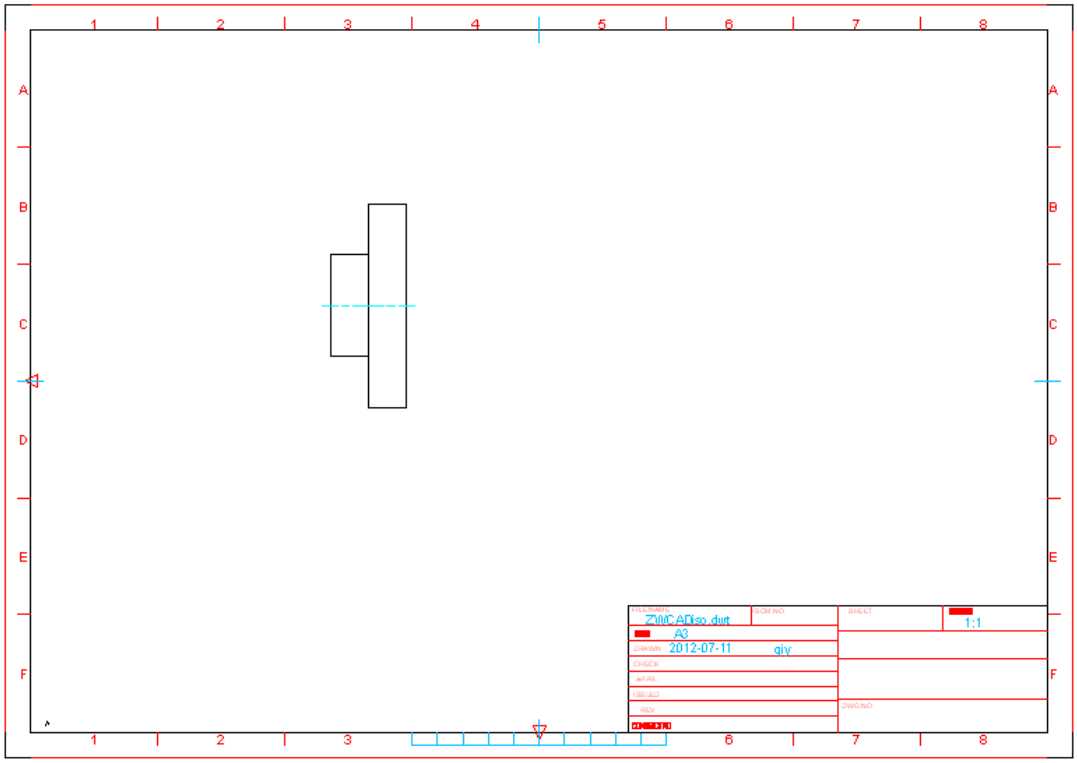
In the displayed dialog box of the ‘Shaft Design’, add new data content in the dialog box:

In case of ‘1’ as the current selection, input data 15 into the ‘Length’, 40 into the ‘Initial Diameter’, 40 into the ‘Termination Diameter’, click the ‘Add’ button. By then the data input for the first shaft is done and the contents of the shaft thus formed can be viewed in the ‘Preview’ box;

In case of ‘2’ as the current selection, input data 15 into the ‘Length’, 80 into the ‘Initial Diameter’, 80 into the ‘Termination Diameter’, click the ‘Add’ button. By then the data input for the second shaft is done and the contents of the shaft thus formed can be viewed in the ‘Preview’ box;

Click ‘OK’ to draw the newly designed shaft;

Select an appropriate location in the drawing for the shaft, as shown:



1.2.2. Two methods to draw a hole

1. Draw a center hole:

Click ‘Hole and Shaft Design’ in the ‘Construction Tools’ under the ‘ZWCADM’ or input: ZWMHOLEAXIS;

When prompted: Input first point of hole or [Draw Shaft(S)/First Diameter (F): 100.00/End Diameter (E): 100.00/Centerline Extended length (L): 3.00/Draw Centerline: No (C)]:

Input: F, set the hole diameter to: 10

Input: Select the centerline position by the left side of part as the starting point of the hole, and that by right side as the end point of the hole, by then the center hole is drawn.

2.Draw a hole around:

Draw the centerline of the hole around

Execute of the ‘offset’ command or execute: OFFSET

Select the centerline of part, input the offset distance of 30 for offset towards both sides respectively to form the centerline of hole around;

Trim the centerline of the hole around

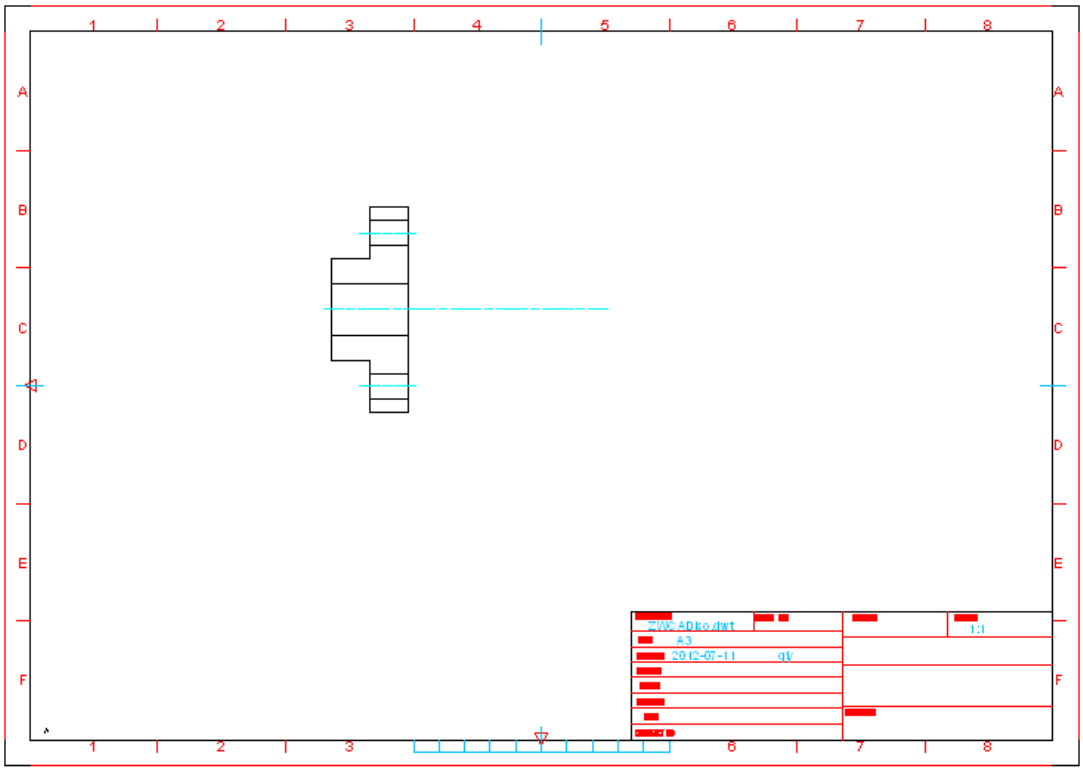
Click ‘Break’ in the ‘Construction Tools’ under ‘ZWCADM’ menu or input: DAD;

Select the centerline around as the target, select appropriate breaking points to break the centerline at the specified location and delete the unwanted part.

Draw the contour of the hole around

Click the ‘Drawing mirror line’ in the ‘Drawing Tools’ under the ‘ZWCADM’ menu or input: DC;

Select the part centerline as symmetry axis, capture the intersection between the centerline around and contour, specify the radius of the hole to be 5 and draw the center hole at both sides.



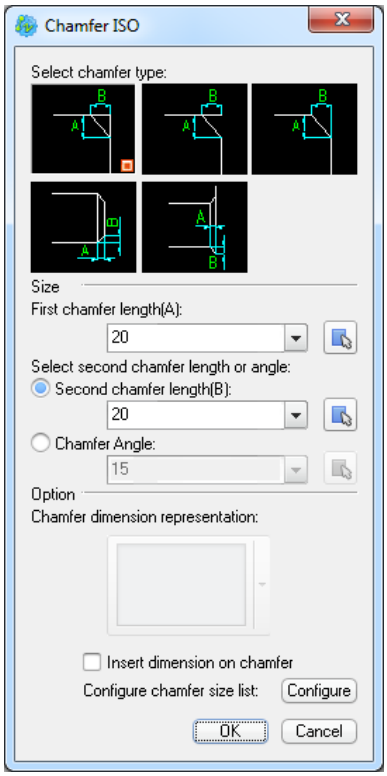
1.2.3. Draw a chamfer

Click the ‘Chamfer’ in the ‘Construction Tools’ under the ‘ZWCADM’ menu or input: DJ,

The prompt appear: (type: No Cut) (Dimension: off) Current chamfer setting = 0.5,45

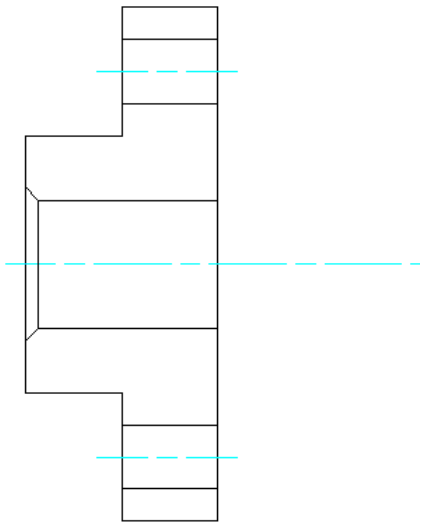
Select first object or [Polyline/Setup/add Dimension]:<Setup>:,

Input: S to bring up a dialog box and select the ‘Chamfering Model’, and set the length of the two chamfers to 2 for both the first and second chamfer (alternatively, select ‘Chamfer Angle’, for instance, set the Chamfer Angle to 45 degree), as shown:



After setting, click ‘OK’ to draw;

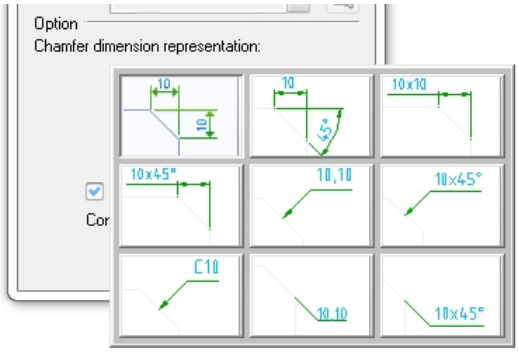
Select the upper edge of the center hole of the part as the first object, and the lower one as the second object; select the left end line of the part. The chamfer of part hole is drawn as shown in the figure:



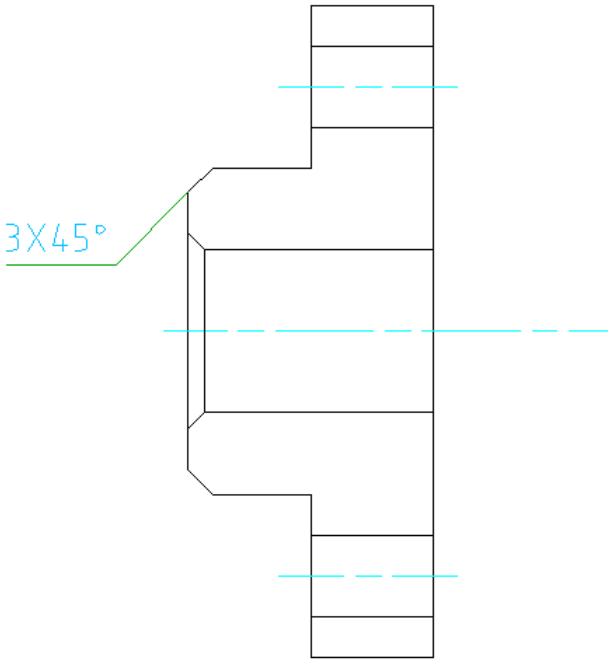
At this point, it is possible to set the chamfer for a shaft according to the above steps except the difference in the setting of chamfer type and size. As it is a sectional view, it is necessary to delete the unwanted straight line.

Note: during the execution of the command, press the ‘ESC’ to exit the command being executed.

It is possible to dimension the chamfer while drawing it. Check the ‘Insert Dimension on Chamfer’ in the ‘Chamfer Setting’ and the chamfer, once drawn, will be automatically dimensioned. Click Chamfer Dimension Representaiton to choose among a variety of representations, as shown in the figure:

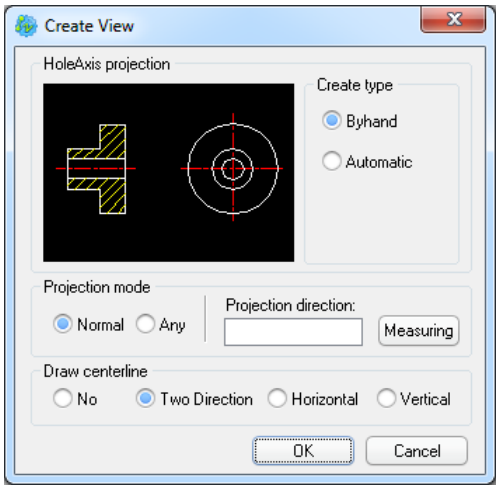


Upon drawing the outcome are shown in figure:



1.2.4. Draw a projection

Click ‘Hole or Shaft Projection’ in the ‘Construction Tools’ under the ‘ZWCADM’ menu or input: HSP to bring up the settings page, as shown:



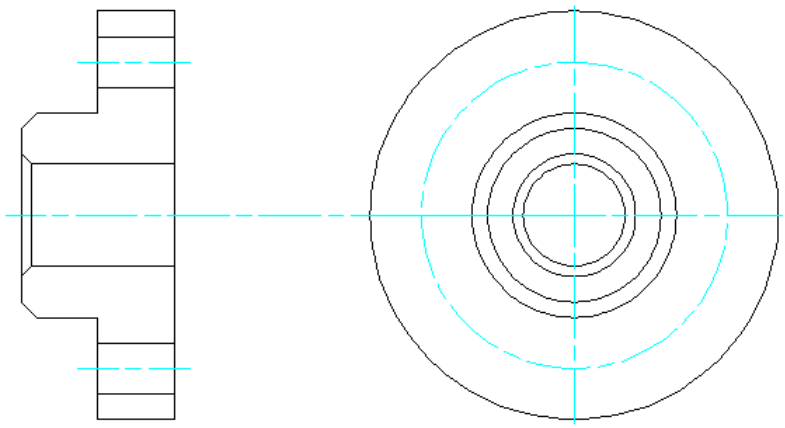
Select ‘Manual’, set the projection direction to ‘Normal’, and the Draw the Centerline to ‘Vertical’ and click ‘OK’;

Prompted : Please select axis; select the centerline of the part;

Prompted: Please specify feature projection point, and select the feature points on the part in turn as projection points, right click to end the selection

Prompted: Input a position point:, select the right side of the part centerline as the insertion position point of the projection, at this point of time, the drawing of the main view of the part is completed,

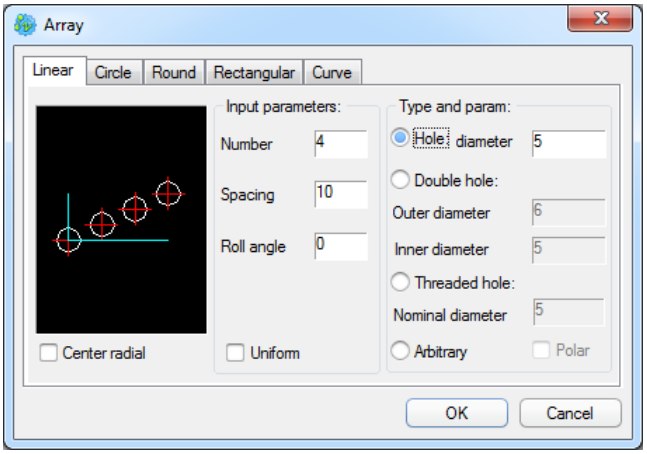
It is shown as the figure:



Note: Since the drawing in question is completely symmetrical, it is possible to draw it by ‘automatic’ projection, only it is generally not recommend.

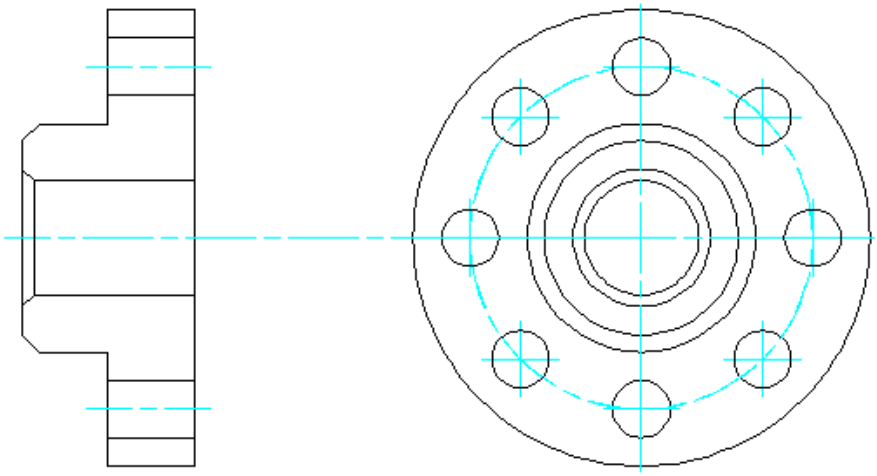
1.2.5. Draw a hole array

Click the ‘Hole Array’ in the Construction Tools under the ‘ZWCADM’ menu or input: KZ to bring up the settings page, as shown:



Select the Circular Array, enter the number: 8, the distribution diameter: 60, the hole diameter: 10, check the ‘Centerline Radial Distribution’, and click ‘OK’;

Prompted: specify the array basis point:, select the center point of the main view of the part as the basepoint, at this point of time the hole on the part is drawn as shown in the figure:



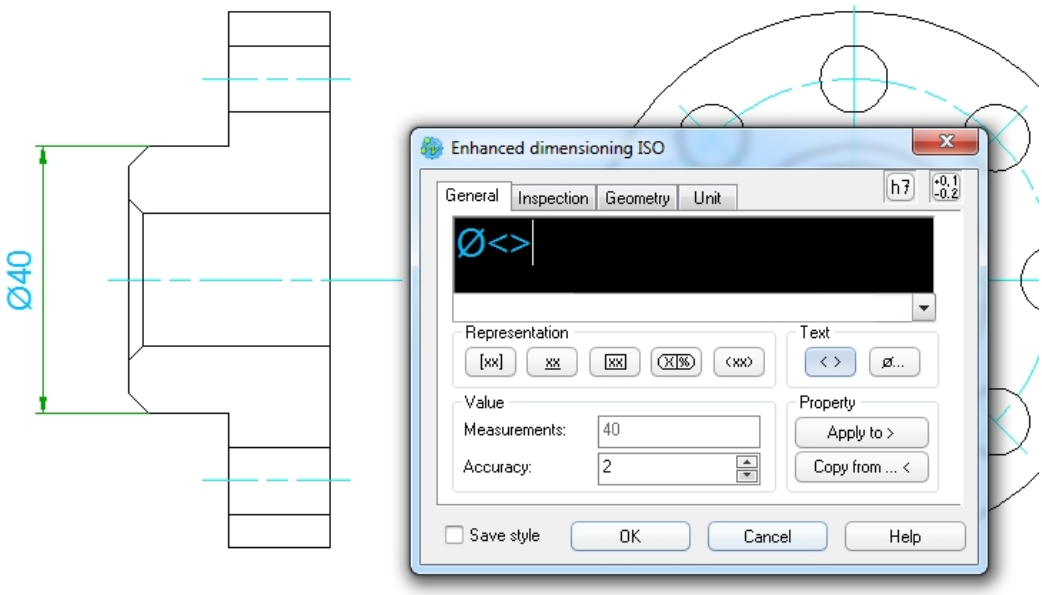
1.3. Dimension

1.3.1. Dimension

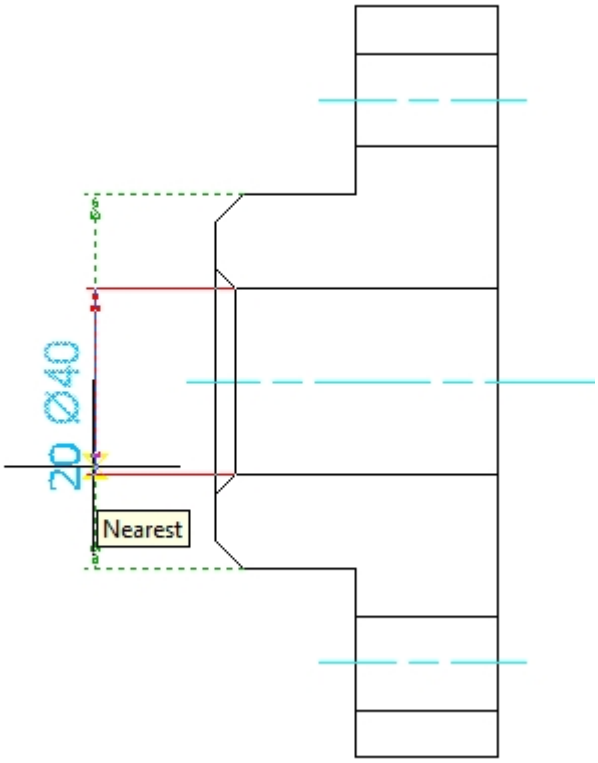
Click the ‘Intelligent Dimension’ in the ‘Dimension’ under the ‘ZWCADM’ menu or input: D

Prompted: (single) specifies the first extension line origin or [Angle (A)/Baseline (B)/Continuous (C)/Select (S)] <Select (S)>: , select two points of the outer contour of the part;

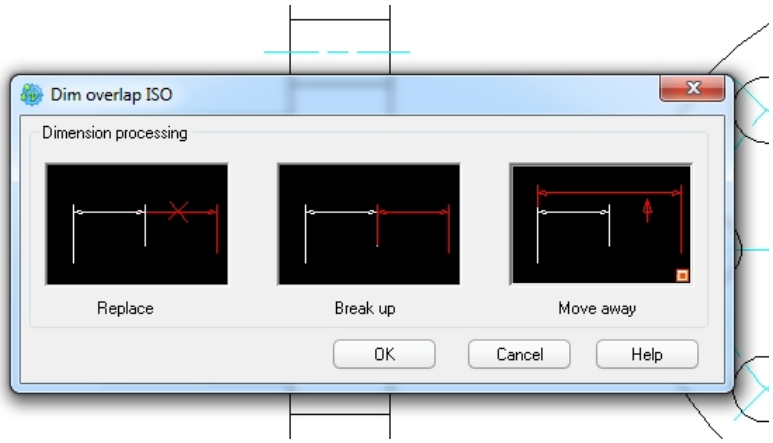
Prompted: Specify dimension line location or [Drag (D)/Horizontal (H)/Vertical (V)/Alignment (A)/Rotated(R)/Pick up the Object Contour (P)/Orientation(O)/Config (C)] , at this point of time the dimension line location will follow the mouse. Once the location is specified, the dialog box of ‘Enhanced Dimension’ pops up for the users to click φ symbol in the ‘Text’area and add a diameter symbol for the size, as shown:



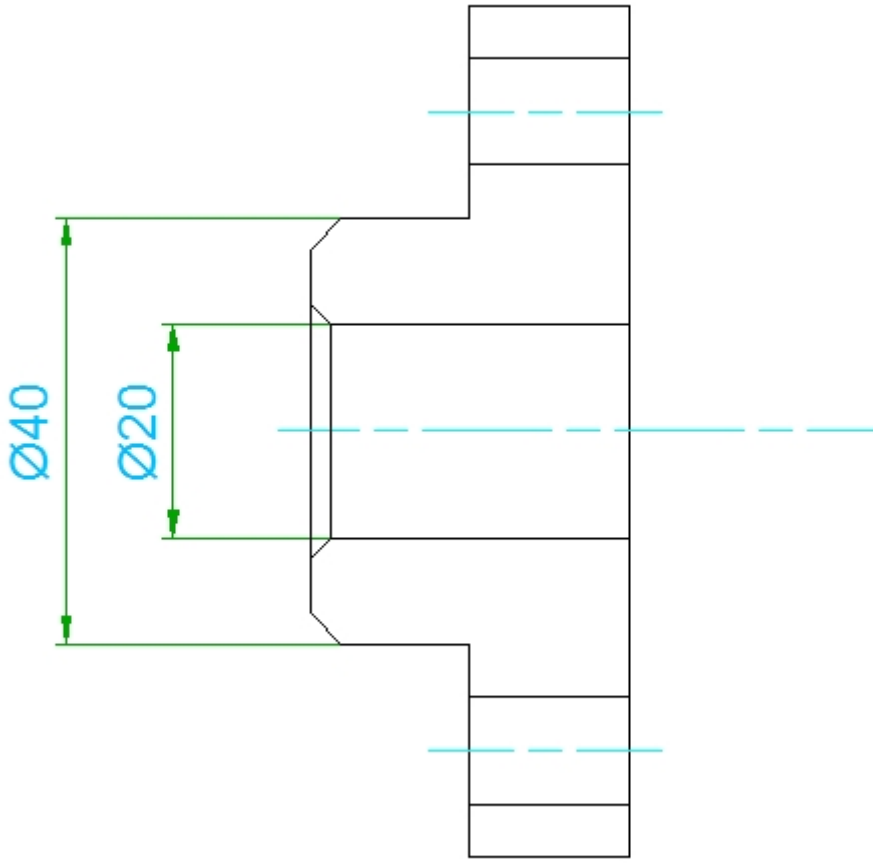
Click ‘OK’ and the dimensioning of diameter of the outer contour is completed. At this point of time the command is in the initial execution status of ‘Intelligent dimension’. Follow the prompt to select two points of the inner contour of the part and move the dimension line to the freshly dimensioned dimension line position, at which moment the freshly dimensioned dimension line is in the status of ‘Contour Pick Up’, as shown:



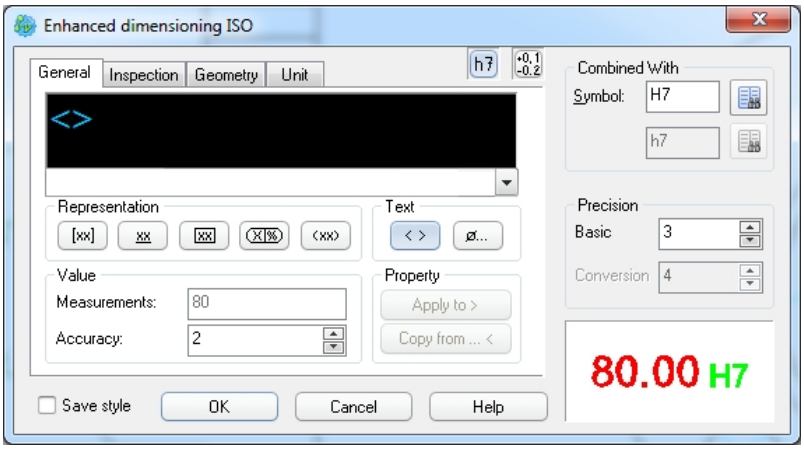
Specify the dimension line to here and bring up the dialog box of ‘Enhanced Dimension’, add the ‘φ’ symbol, click ‘OK’. Since at this time the two dimension lines are at the same location, the dimensions will overlap and a dialog box of ‘Dimension Overlap’ will pop up for the users to select ‘Move Away’ in the ‘Dimension Processing’ and click ‘OK’;



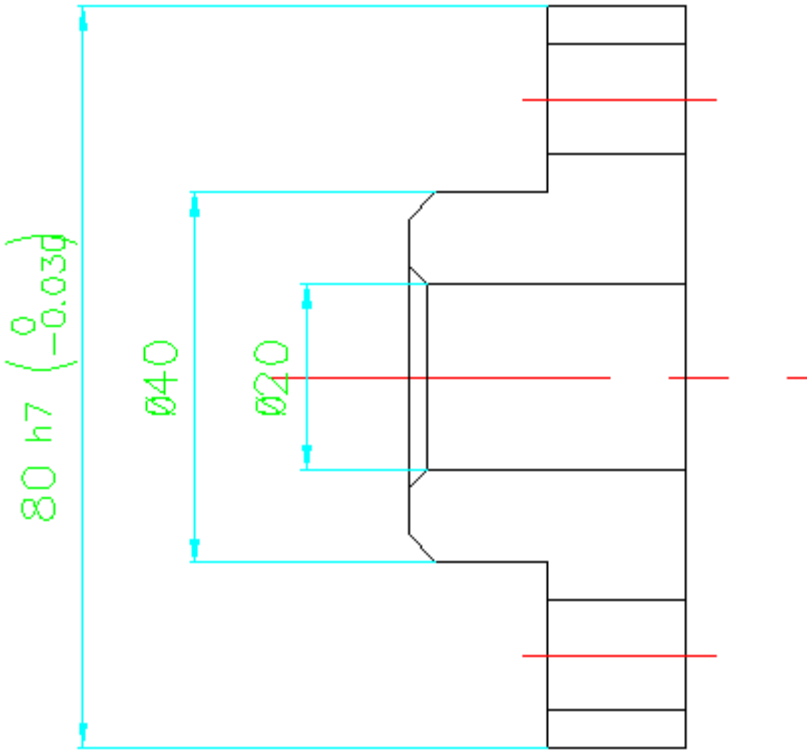
At this point, the dimension of the outer contour will automatically move up while the dimension of the inner contour remains in the original position, as shown:

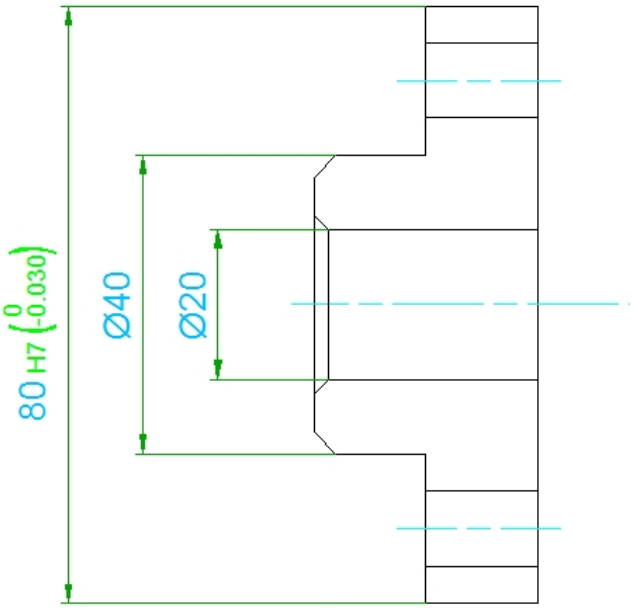


Proceed to dimension the edge contour of the part by adding tolerance to the dimension. When the dialog box of ‘Enhanced Dimension’ pops up after the specification of the dimension line location, click the ‘Mating’ button at the top right, the dialog box at this time will be extended, as shown in the figure:



Click the ‘Select Fit Type’ at the lower-right corner of the dialog box and select a desired type before clicking ‘OK’ to finish the dimensioning. The outcome is shown in the figure:



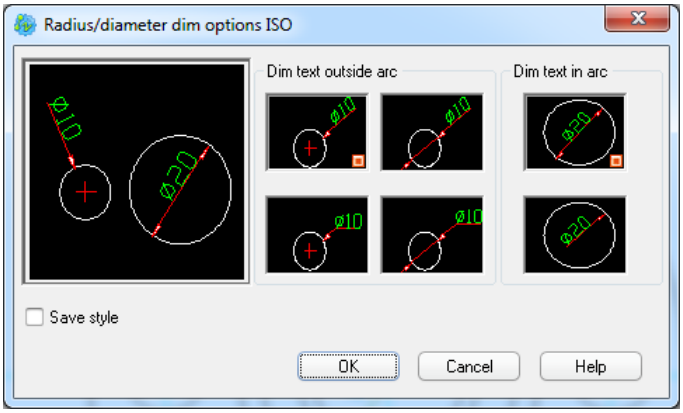


Proceed to dimension other areas of the part. To dimension the radius at the centerline position of the holes in the main view, continue with the ‘Intelligent dimension’,

Pompted: (single) Specifies the first extension line origin or [Angle (A) /Baseline (B)/Continuous (C)/Selection (S)] <Selection (S):, input: Space, or S;

Prompted: Select arc, line, circle or dimension: to select the centerline of the hole in the main view,

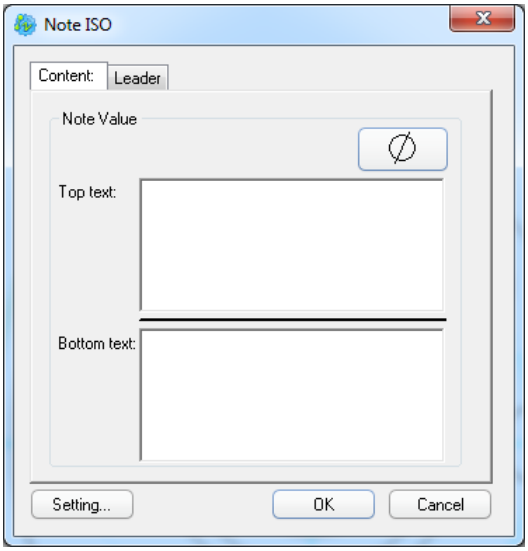
Prompted: Specify dimension line location or [Linear (L)/Radius (R)/Jogged Radius (J)/Option (O)/Config(C)] <Config (C)>: (now it is possible to quickly convert the dimension style to other types, as shown: Linear, Radius, and Jogged Radius Dimension. Since we are dimensioning the diameter of the centerline, we can ignore other types), Input: O to bring up the dialog box of 'Radius/Diameter Dimension Options', as shown in figure:



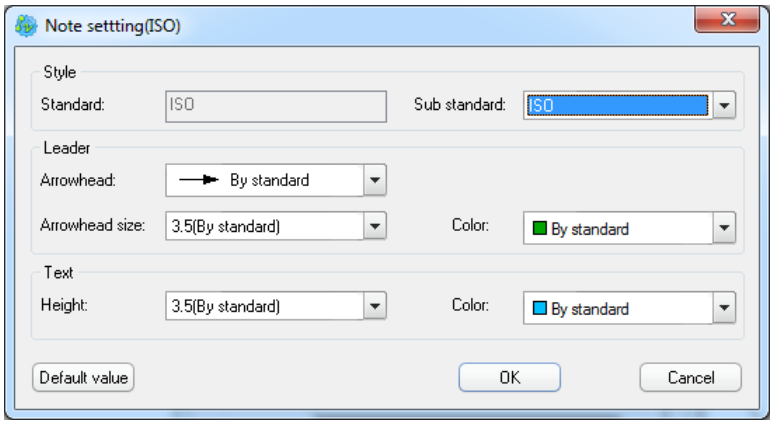
In the dialog box, select the desired dimension style, click 'OK', and select an appropriate dimension line position to finish dimensioning the diameter of the centerline.

1.3.2. Leader Dimension

Click Leader Dimension in the Dimension under the 'ZWCADM' menu or input: NO to bring out the dialog box of 'Leader Dimension', as shown:



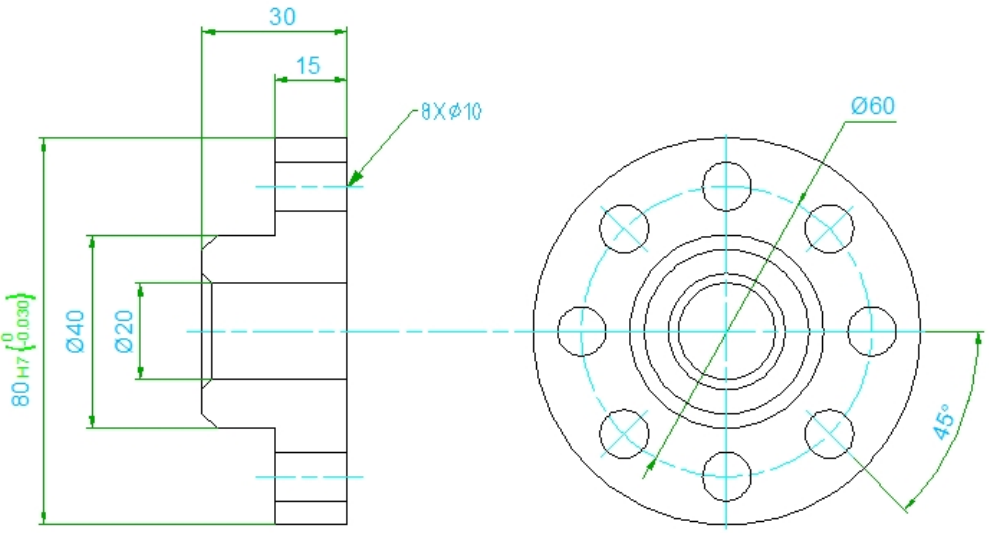
Added 'Top Text' and 'Bottom Text' in the Leader Dimension Contents, or click the 'Symbols' button on the top to add symbols, click the 'Settings' at the lower left corner to set the leader dimension symbol, as shown in the figure:



Set the leader arrow style, size, color and text height, color, etc. in the ‘Settings’, at the end of which Click OK;

Prompted to: select the object or lead point: select the desired position to be dimensioned;

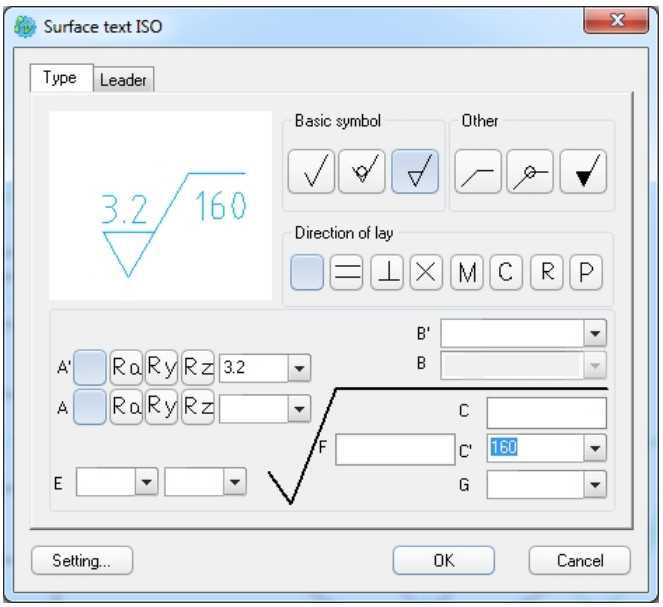
Prompted to: Next point or [Config (C)/Polyline Leader (P)/No leader(N)/Rotation Orientation(D)] <Config (C):, users can opt operation for polyline, no leader, or rotation orientation as appropriate, as shown:



1.3.3. Symbol Dimension

(1) Surface Text (SF)

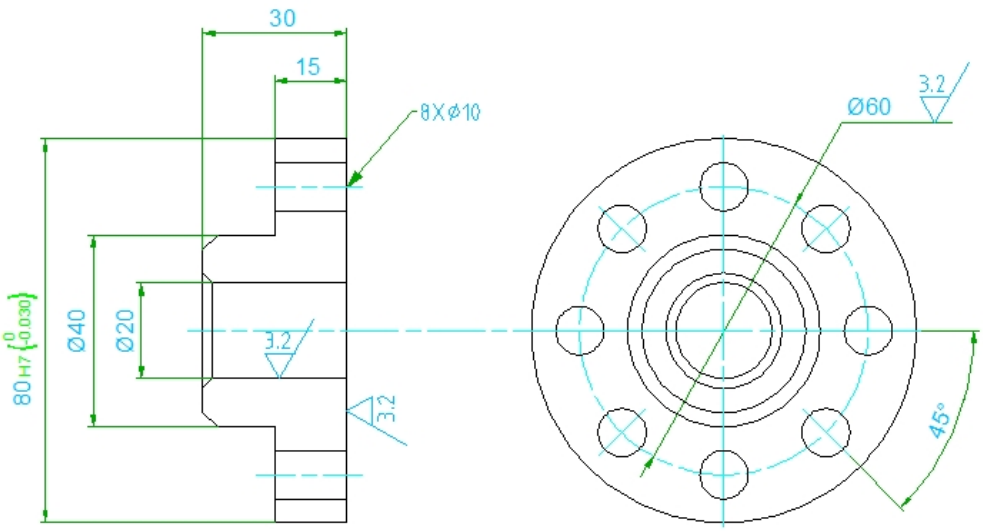
Click ‘Surface Text’ in the ‘Symbol Dimension’ under the ‘ZWCADM’ menu or input: SF to bring up the dialog box of ‘Surface Text’, select Material Removal Prohibited in the Basic Symbols and enter in ‘A’ or select from the drop-down list the appropriate value (this step also applies to other parts). Now the desired roughness symbol can be seen in the preview, as shown:



After editing, click ‘OK’

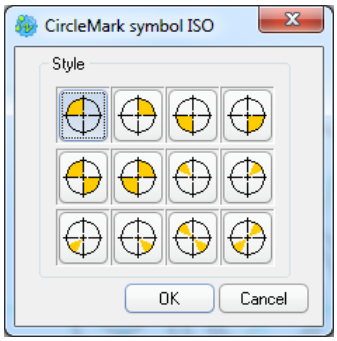
Prompted: Select the object to attach or: , select the inner contour in the left view of the part as the target, at this point it is possible to move the roughness symbol along the selected target, or select a direction (similarly in other Symbol Dimensioning situations) ;

Prompted: Specify the insert point or [press the CTRL key to add leader/Config (C)] <Config (C), now it is possible to select the final location. Follow the prompt to press the CTRL key and see draw a leader from the selected point, select an appropriate location to dimension the surface text in the desired locations. The outcome of dimensioning is as shown the figure:



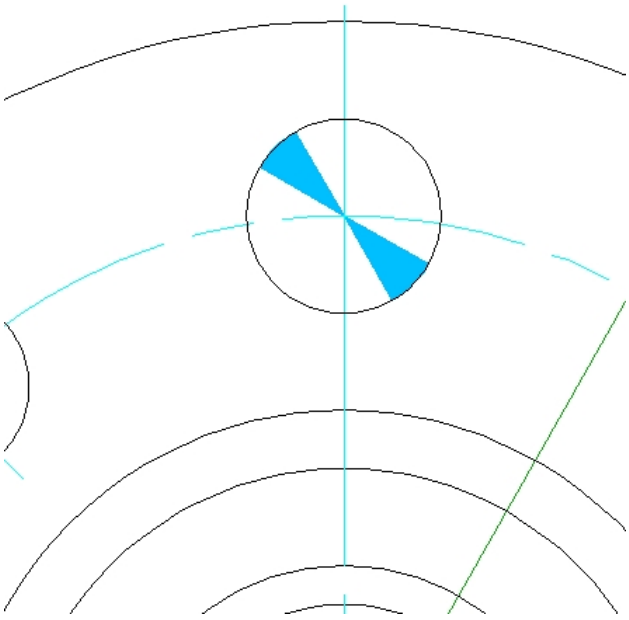
(2) Circlemark

Click the ‘Circle Mark’ in the ‘Symbol Dimension’ under the ‘ZWCADM’ menu or input BJ to bring up the dialog box of ‘Circlemark’. A total of 16 circlemarks are provided, as shown:



Select the desired circlemark, and click ‘OK’;

Pompted: select hole:,select the desired graphic for circlemark in the drawing. After selection of the circlemark is done, as shown:

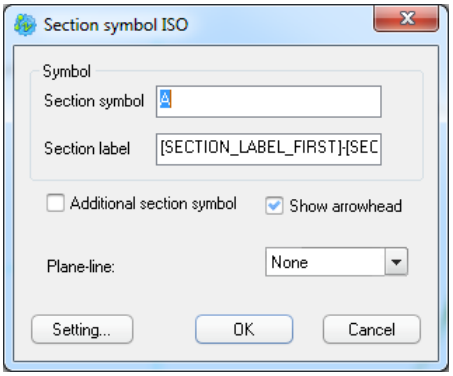


1.3.4. View Dimension

Click the ‘Section Line’ in the ‘Create View’ under the ‘ZWCADM’ menu point or input: STL,

Prompted: Select point: select a point in centerline of the main view;

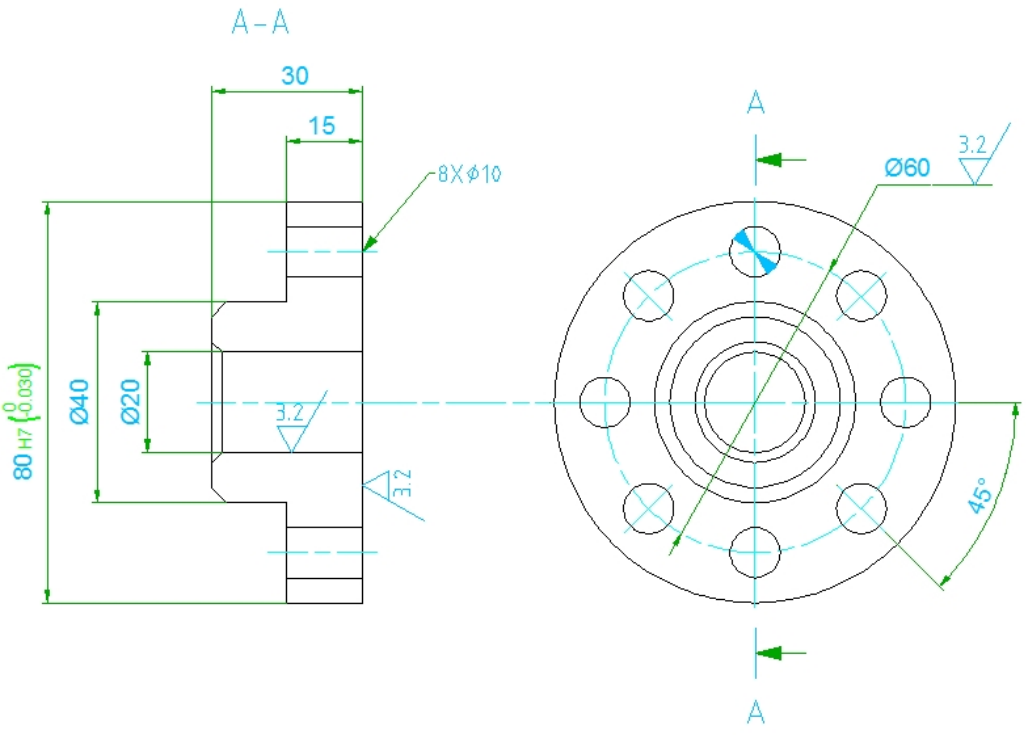
Prompted: The specified next section line point or [Config (C)] <Config (C)>:, select a lower point; right-click to the end the selection and bring up the dialog box of ‘Section Symbol’, as shown in the figure:



Set ‘Section Symbol’s, ‘Section Label’ in the dialog box and click ‘OK’;

When prompted to: specify side of section, or [Config (C)] <Config (C)>:,direct the left view;

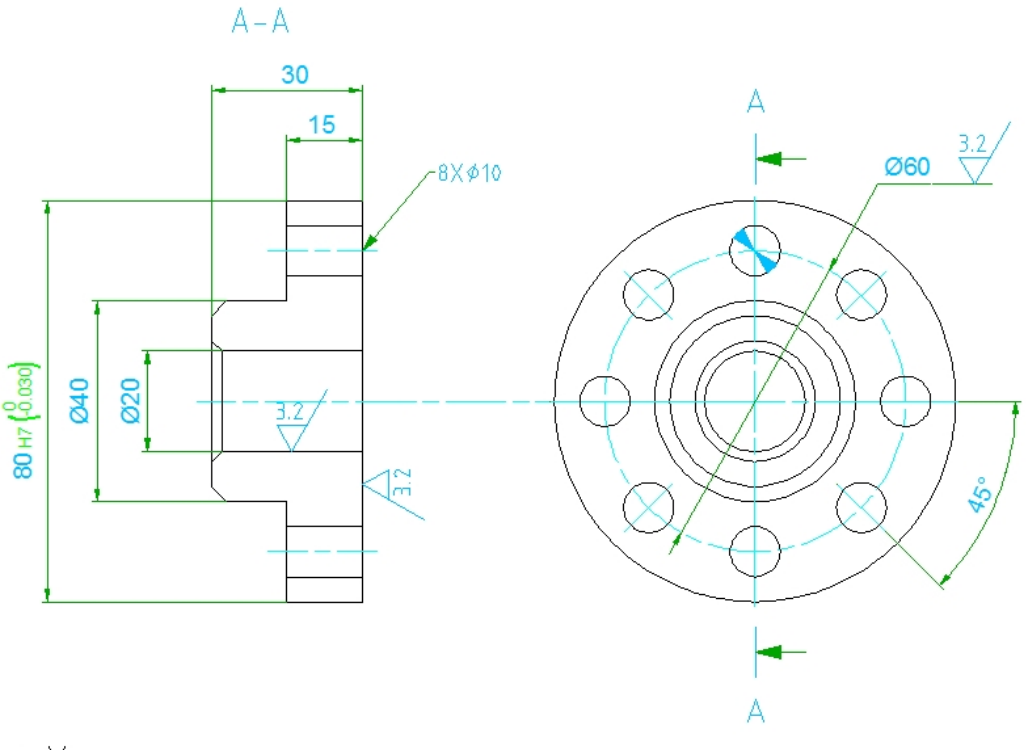
When prompted to: specify origin of section view or [Config (C)] <Config (C)>:, place the view name at the upper part of left view. As the section line drawing is done, the outcome is shown in the figure:



Section line

Click the ‘Section Line’ in the ‘Drawing Tools’ under the ‘ZWCADM’ menu or input: H;

Prompted: Specify internal point or [Properties(P)/Select Object (S)/Drawing Boundary (W)/Delete Boundary (B)/Advanced(A)/Drawing Order (DR)/Origin Point(O)]:, Select the desired area of the main view to add the section line, right click to execute carriage return. The section line drawing is completed as shown in the figure:



Super Edit

Double-clickV in the drawing or select the Dimension, Symbol Dimension etc. to edit and modify these objectives, for instance, double-click the surface text symbol and modify the surface text symbol in the original position by changing the settings in the setting page of surface text.

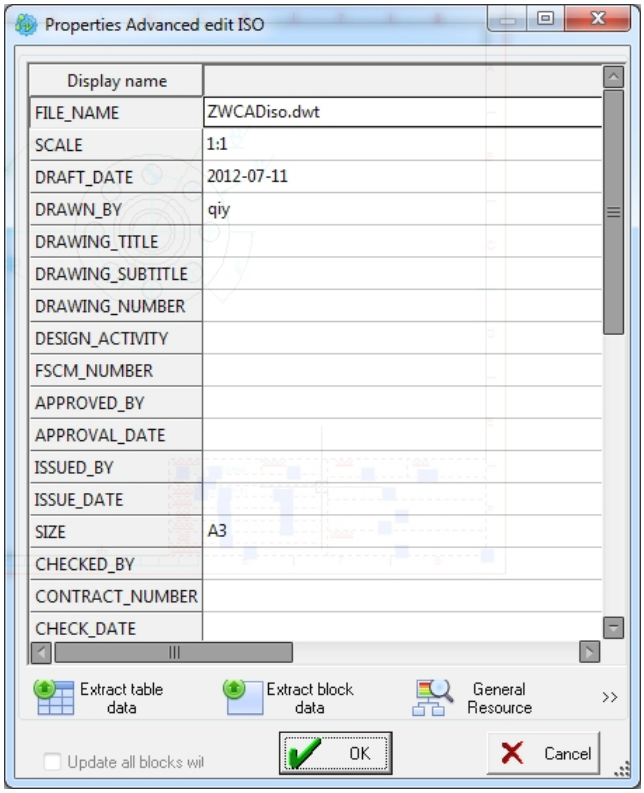
2. Operational procedure of ZWCAD MECHANISM

Here is the brief description of the general steps for users to mechanical designs with ZWCAD MECHANICAL and for them to develop an overall understanding of ZWCAD MECHANISM.

2.1. Fill in the title

Execute: TF, set the drawing to A3 and the ratio to 1:1;

Fill in the title, and execute: ZWMTITLEEDIT or double-click the title to fill in or edit the title, as shown in figure:



2.1.1. Manual completion:

Click any item and fill in or change the content.

2.1.2. Fill in single line through resource operations

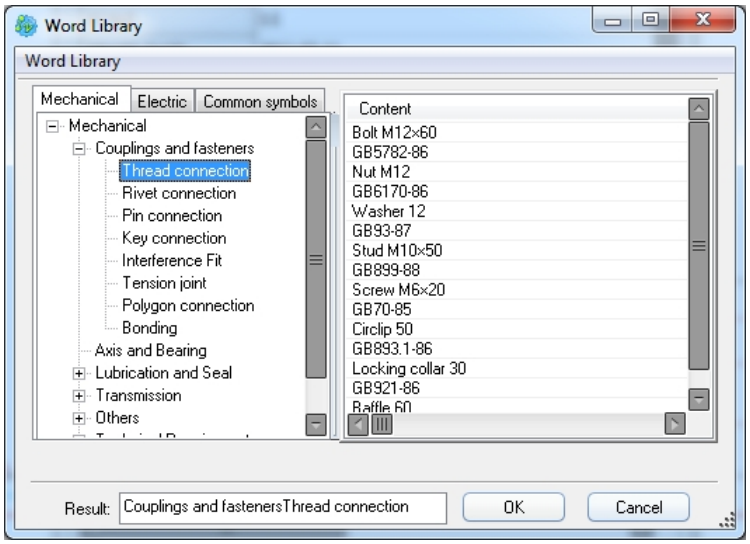
Right-click a row to pop up the menu, and invoke resource operation to fill in the title. The resources available for invocation include such three kinds: word libraries, engineering calculator, and extract page text.

For example: invoke text from the word library

Select an item from the title so that it is open to input or modification;

Execute: CTRL + W to invoke the word library;

Pop up the invocation dialog box of ‘Word Library’, as shown in the figure:



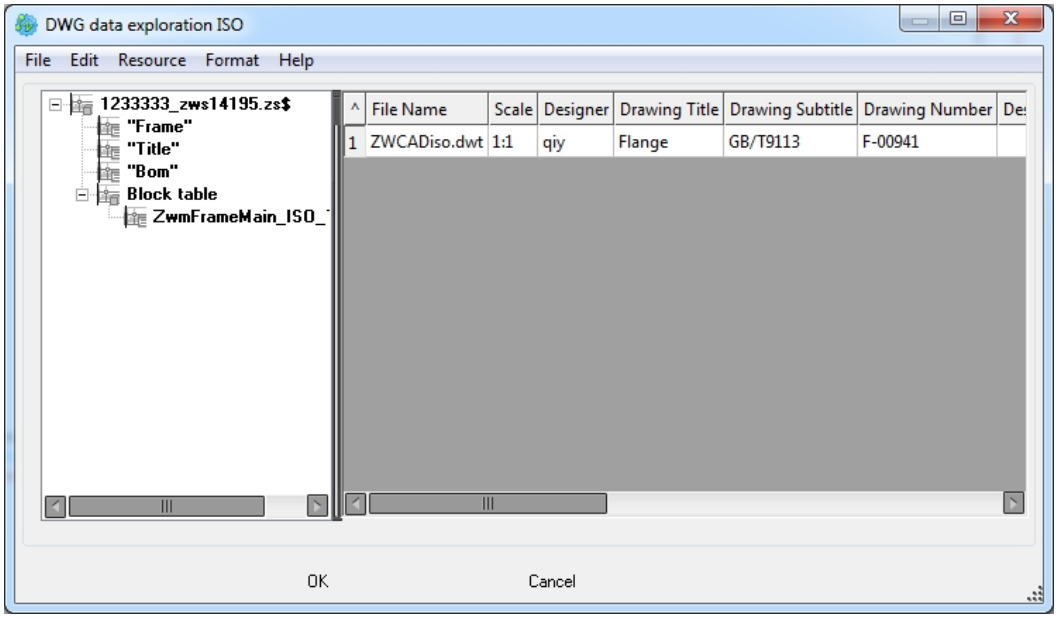
Click the option on the left side ‘Directory and Target Tree Branch’, double-click the desired ‘Content’ in the content box on the right side, as the selected contents is added to the ‘Outcome’, click ‘OK’, the content selected from the word library is added to the selected box.

2.1.3. Overall completion of the title:

It is possible to fill in the title on an overall basis by invoking resources operations at the labe, such three kinds as: read text file, read DWG file, extract tabular data.

The steps for filling in the title by reading DWG file data

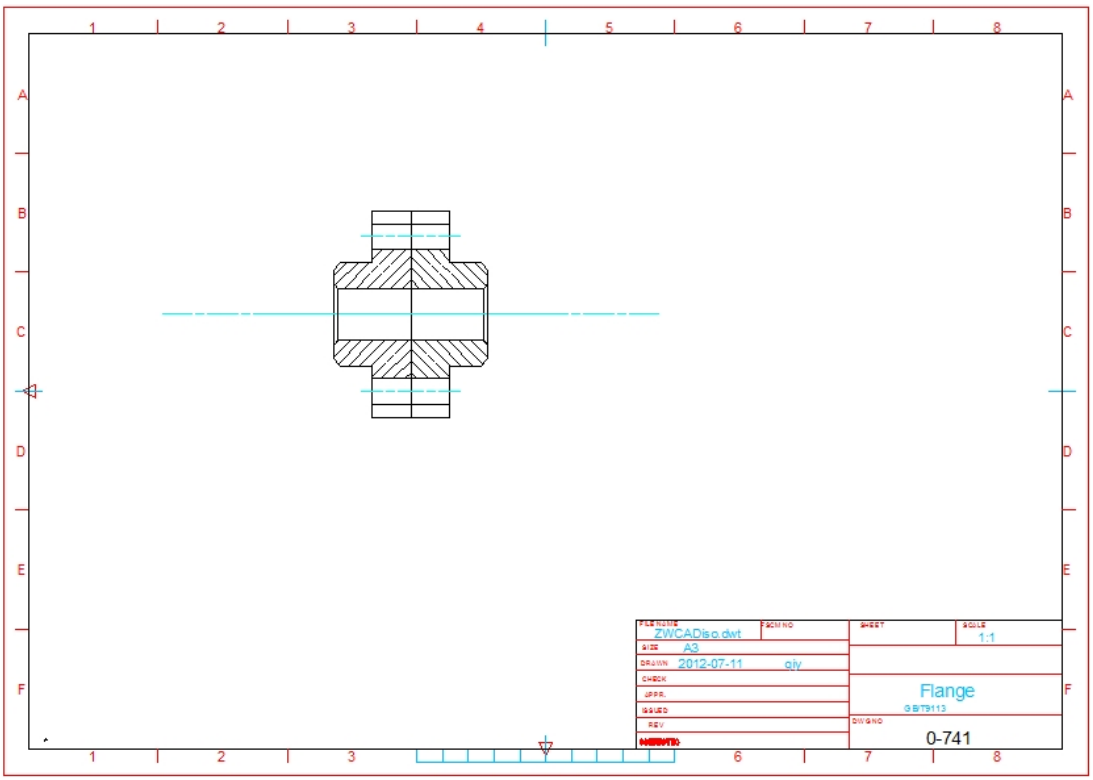
Right-click the label, select ‘Read DWG file’ from the resources and selected the desired DWG files. The dialog box of ‘DWG Data Browsing’ will pop up at the end of selection, as shown in the figure:



Make a selection in the ‘Objectives and Branch’ on the left side, select the data of a particular item on the right side, and click OK to automatically add the selected content to the title.

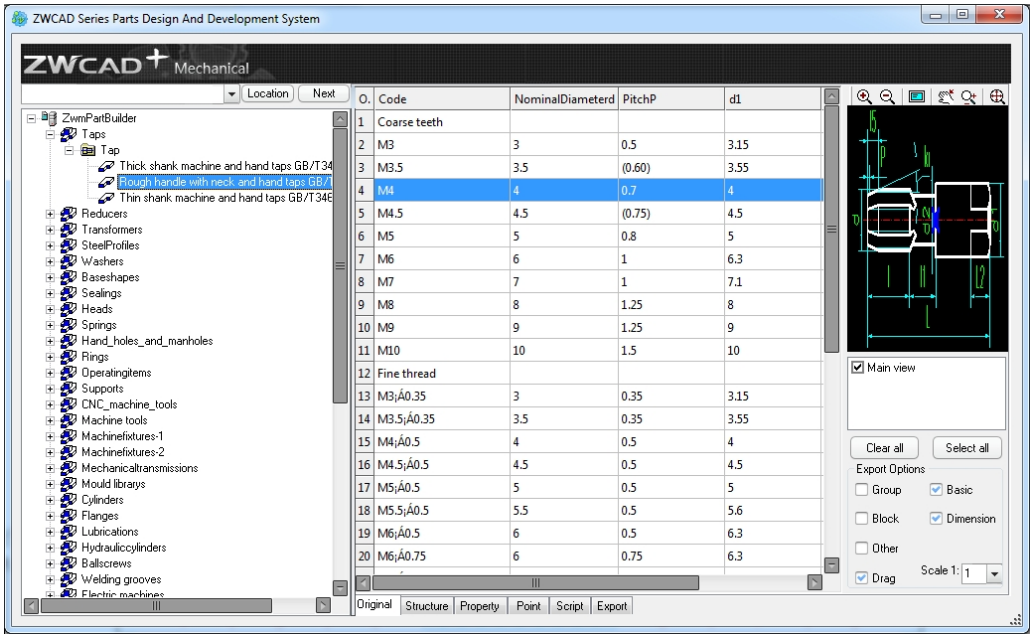
2.2. Part Builder for drawing standard parts:

2.2.1. Draw basic graphics



2.2.2. Add standard fasteners for mechanism

Execute: XH, design system of serial parts to bring up the interface, as shown in the figure:



In the parts list on the left, select the ‘Bolt’→ ‘Hex Bolt’→ ‘Hex Head Bolt’→ ‘Hex Head bolts Class A GB/T5780 -2000’, select ‘M12’ for model and ‘55’ for L/2-level. In graphic preview on the right view the thumbnails of the parts and dimension, view option switch, export options etc., Select ‘Main View’, do not check ‘Dimension’. Click ‘Drawing parts’ and a bolt is drawn in a few seconds.

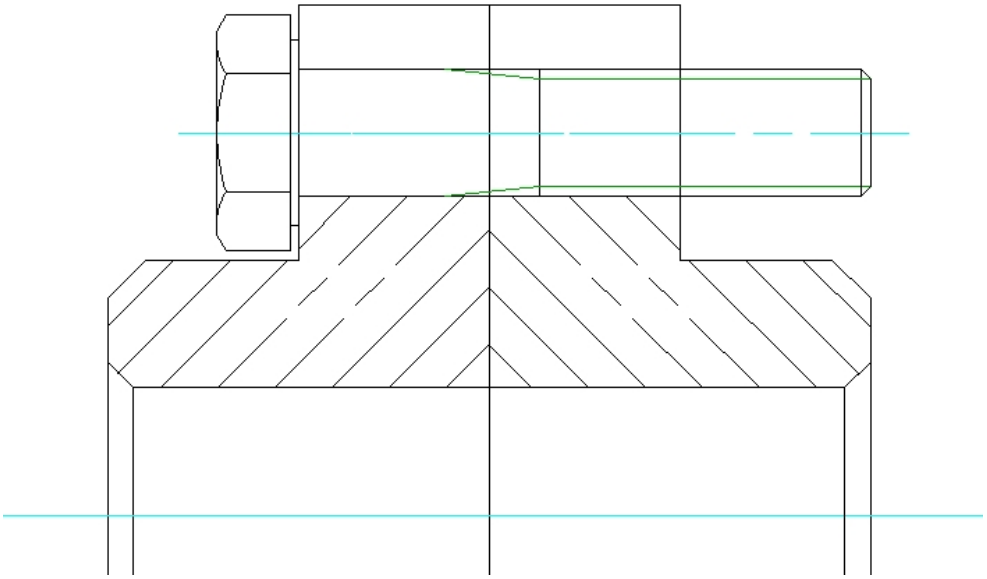
Prompted: Please specify a destination location;

Input: Select the center of the hole at the upper area of the part;

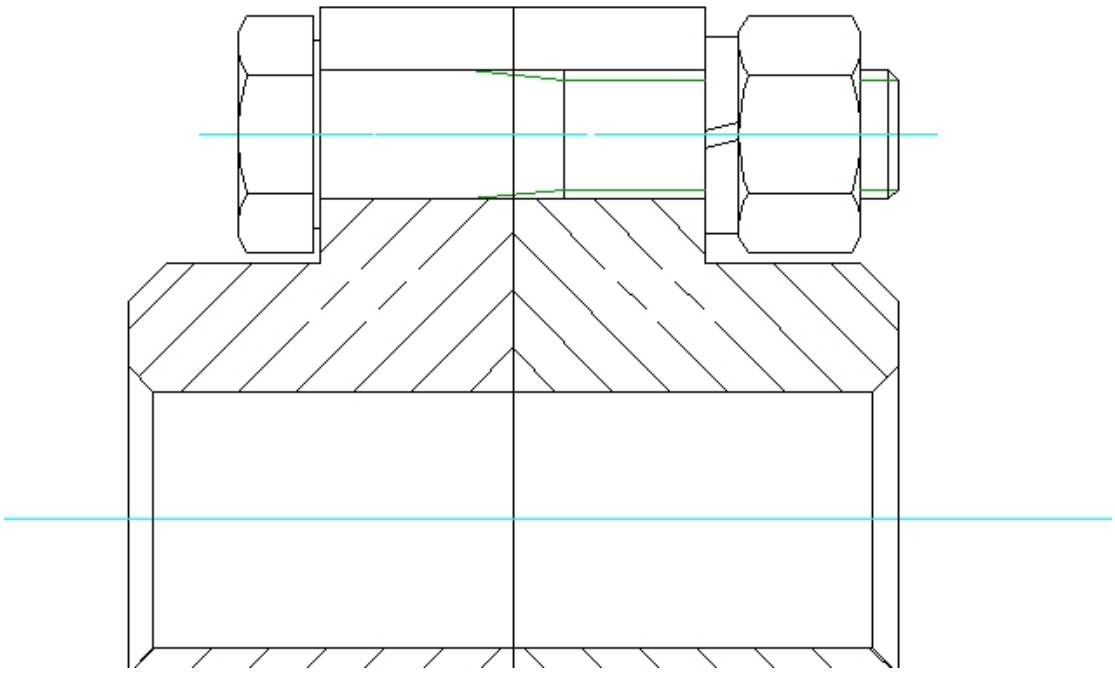
Prompted: Specify rotation angle or [Reference (R)]:

Input: Use the mouse to specify the right direction;

At this point, the bolt is successfully drawn and generated in the specified location, as shown in the figure:



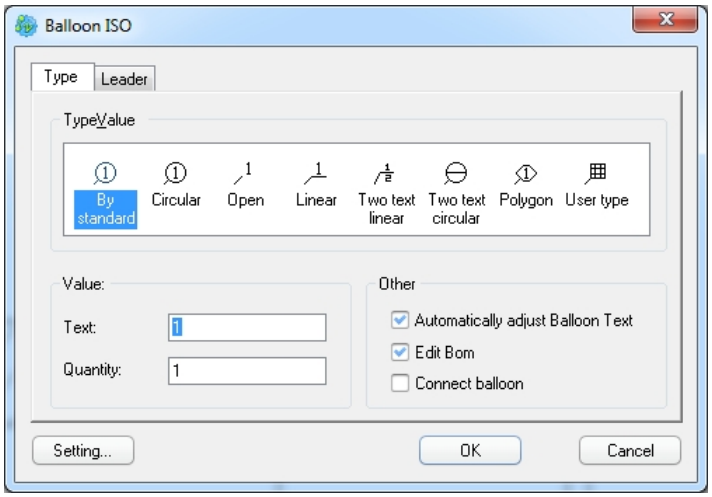
In accordance with the steps above, draw a washer and a nut in order, trim the related line type. The final outcome is shown in the figure:



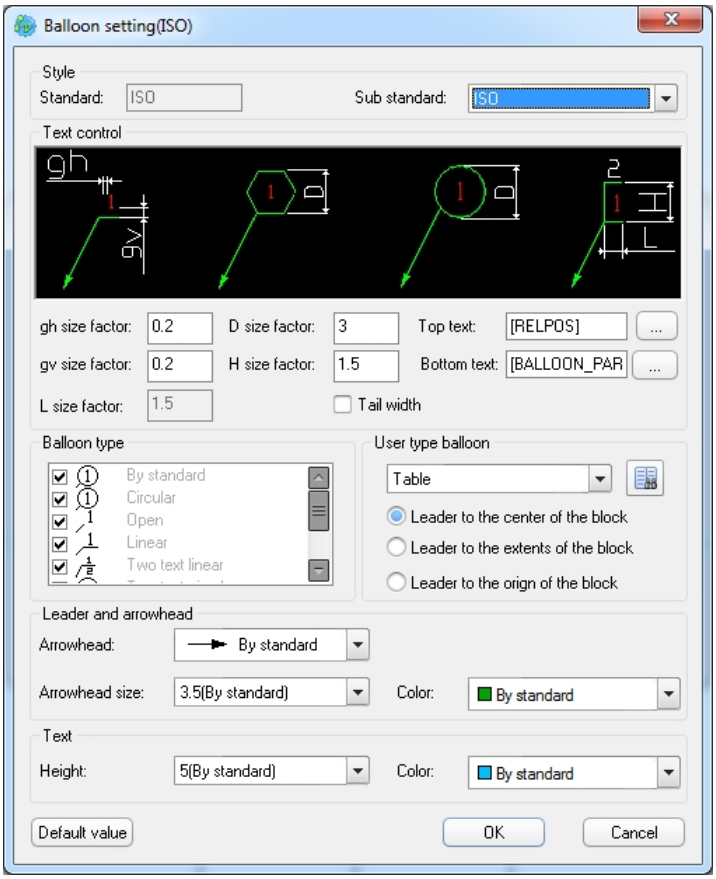
2.3. Balloons dimension and fill out the BOM

2.3.1. Dimension the balloons

Execute: XH, dimension the index for the parts on the drawing. Bring up ‘Balloons’ dialog box, as shown in the figure:



User can select ‘Balloons Type’, ‘Balloons Automatic Adjustment’, ‘Fill in BOM Content’, ‘Connect Balloons’, and set the graphics and text of balloons, as shown in the figure:



After setting, click OK to dimension the balloons;

Prompted: Select the object or lead point:

Input: Set lead point where balloons is to be dimensioned;

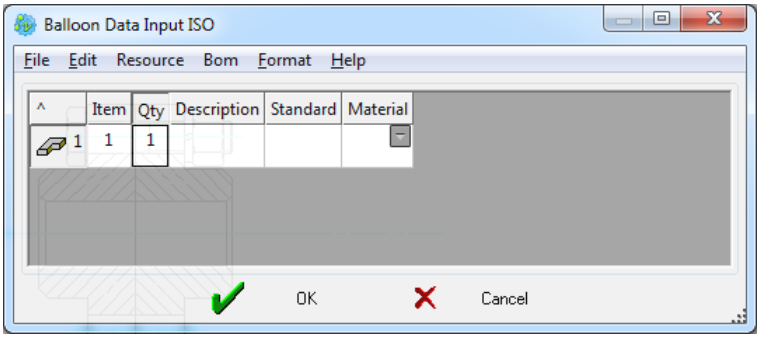
Prompted: Next point or [Config (C)/Balloons Auto Orientation(A)/Towards Left (L)/Towards Right (R)/Polyline (P)/Select Balloons to Align (Q)/No Leader(N)] <Config (C)>:

Input: Specify the end position of balloons dimension;

The balloons dimension is done.

2.3.2. Fill in BOM

If check ‘Fill In BOM Content’ on the balloons settings page, then the contents of filled in BOM will pop up at the end of every balloons dimension, as shown in the figure:

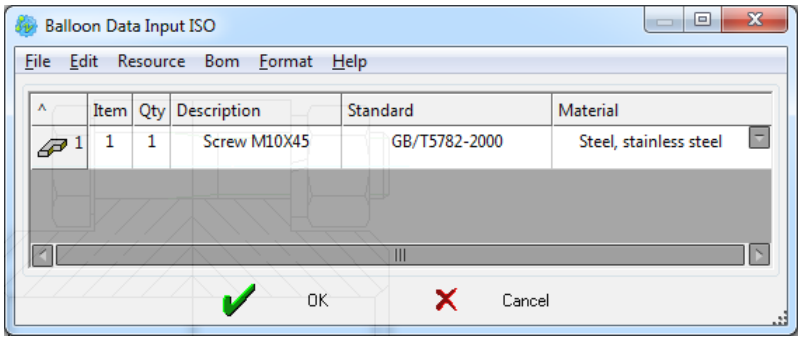


During the fillout process, a variety of operations is at option. If the content for dimension is a standard part, it is possible to ‘Extract Standard Part’ as available in the ‘Resource Operations’;

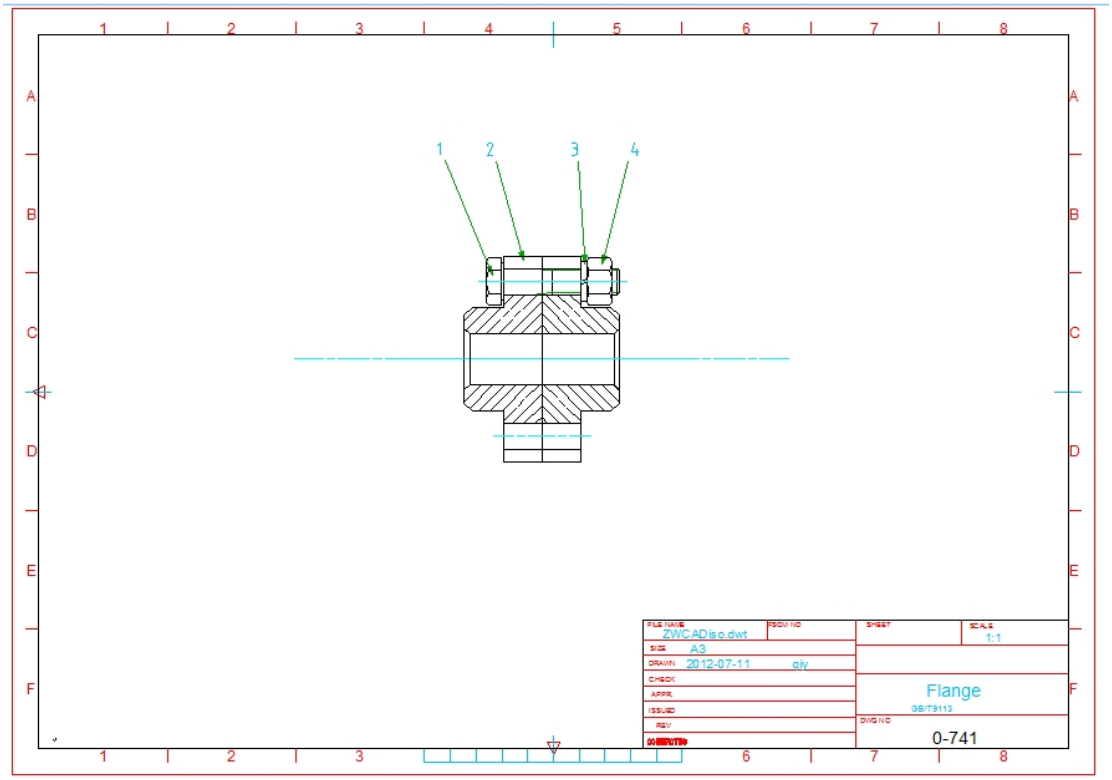
Prompted: Please select a standard part:

Input: Select bolt among graphics;

At this point, the information on the standard part is automatically filled in the BOM,, as shown in the figure:

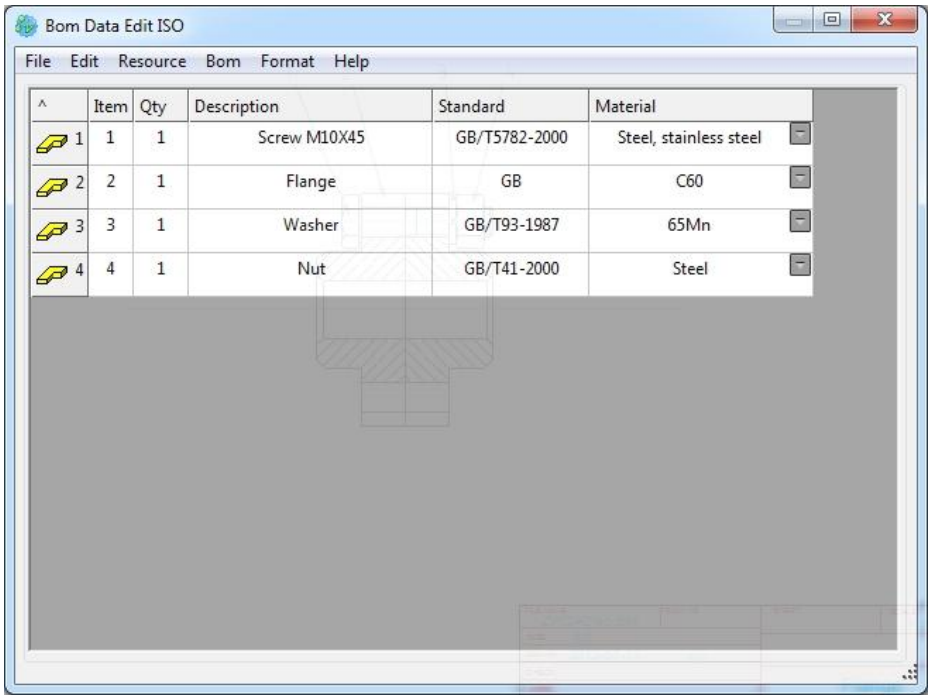


Dimension the balloons the parts by the steps above. The outcome upon the completion of dimensioning is as shown in the figure:



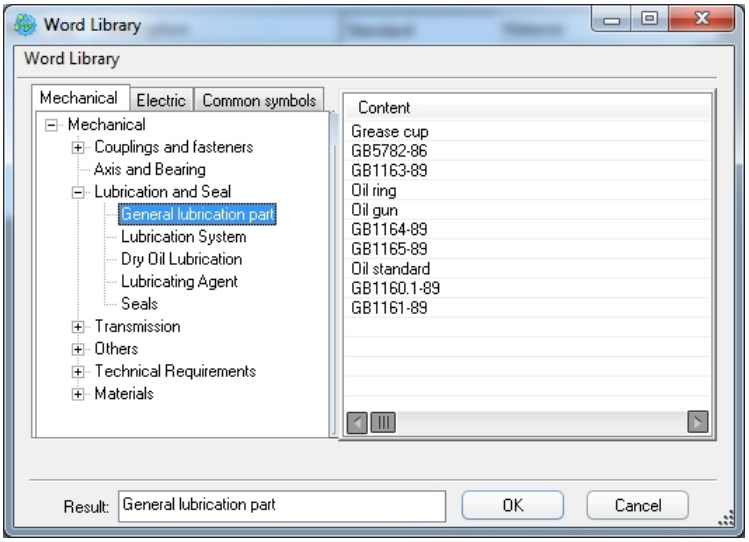
2.3.3. Generate BOM

Execute: PLTB, the edit command, to bring up the ‘Balloons Input’ dialog box, and all contents filled out for the balloons dimension will be displayed, as shown in the figure:



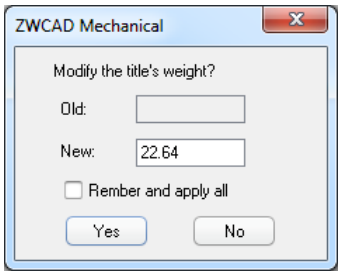
^	Item	Qty	Description	Standard	Material
1	1	1	Screw M10X45	GB/T5782-2000	Steel, stainless steel
2	2	1	Flange	GB	C60
3	3	1	Washer	GB/T93-1987	65Mn
4	4	1	Nut	GB/T41-2000	Steel

At this time can it is still possible to modify the contents, to add a note through the ‘Professional Word Library’ in a part. When the note box process is in the input status, press ctrl + left to invoke the professional word library, as shown in the figure:



Double-click the entry in the ‘Content’ to access ‘Outcome’, click OK to fill in the note.

After completing the BOM, press ctrl + s to generate the BOM in the specified location meanwhile, pass the total weight in the generated list to the weight of the title, as shown in the figure:



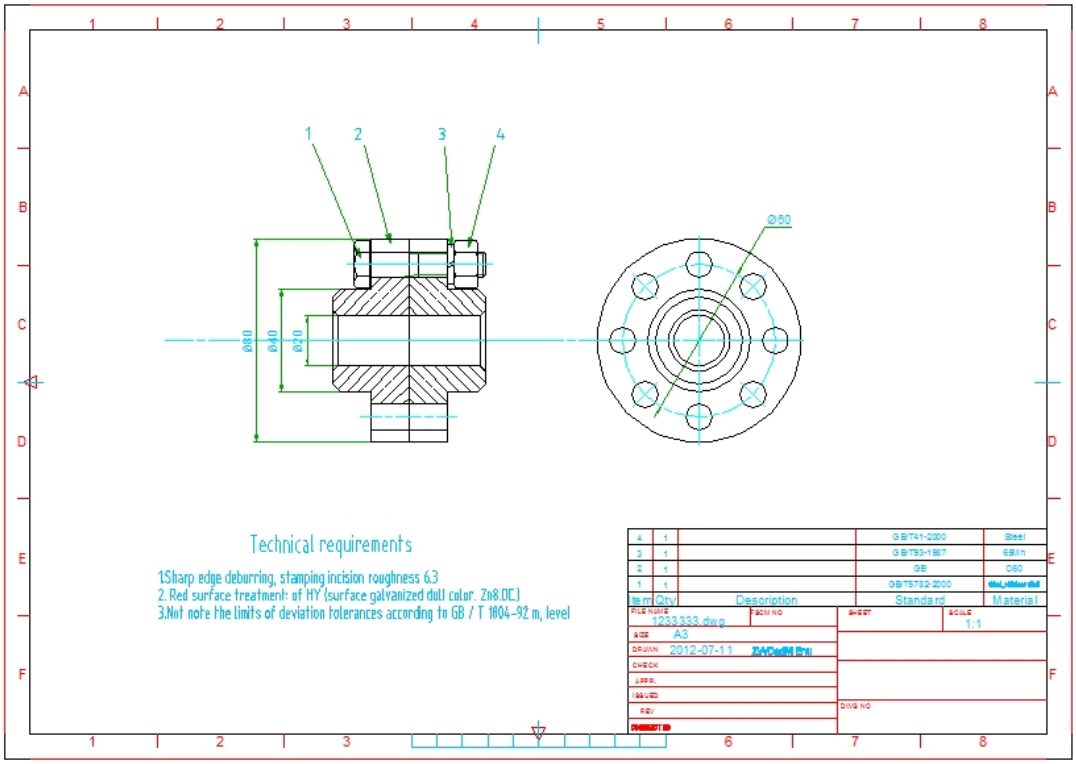
Save the drawing, the drawing is completed.

2.4. Add technical requirements

Adding technical requirements for drawings is also one of the important elements in mechanical design.

Click the ‘Technical Requirements’ in the ‘Text Processing’ under the ‘ZWCADM’ menu or input: TRT;

After specifying the area for the adding of technical requirements as prompted, in the displayed dialog box of ‘Technical Requirements’, select the technical requirements to be added from the ‘Technical Library’ or ‘Read File’ click on ‘OK’ to complete the addition.


1
2
3
4
Ø60
Ø80
Ø40
Ø20
Technical requirements
1.Sharp edge deburring, stamping incision roughness 6.3
2. Red surface treatment: of HY (surface galvanized dull color. Zn8.0C.)
3.Not note the limits of deviation tolerances according to GB / T 1804-92 m, level
4 1 GB/T41-2000 Steel
3 1 GB/T93-1987 65Mn
2 1 GB Q60
1 1 GB/T5782-2000
Item Qty Description Standard Material
FILE NAME 1233333.dwg
SIZE A3
DRAWN 2012-07-11
CHECK
APPR.
ISSUED
REV
SHEET
SCALE 1:1
DWG NO

3.ZWCAD MECHANICAL 2012 Command/abbreviated command list

Operation name	Execution command	Simplified command (shortcut)s
Drawing		
Drawing setting	ZWMFRAMEINIT	FRA
Filling the title	ZWMTITLEEDIT	
Filling the additional list	ZWMFJLEDIT	
Fill ing the parameter list	ZWMCSLEDIT	
Changing a frame	ZWMSWITCHFRAME	
Change scale	ZWMSWITCHSCALE	
Changeing title	ZWMSWITCHTITLE	
Change BOM	ZWMSWITCHBOM	
Changinge code block	ZWMSWITCHDHL	
Changeing additional list	ZWMSWITCHFJL	
Changing parameters list	ZWMSWITCHCSL	
Adding a revision list	ZWMREVISIONLIST	
Multi-frame setting	ZWMFRAMEINIT2	FRA2
Changing standard	ZWMSTDANDARDC	SDC
2. Balloons/BOM		
Balloons	ZWMBALLOON	BAL
Style editing	ZWMEDITBALLOONSHSPLE	
Data modification	ZWMEDITBOMROW	
Balloon alignment	ZWMALIGNBALLOON	
Renumbering balloon	ZWMRENUMBERBALLOON	
Hidinge balloons	ZWMHIDEBALLOON	
Show Balloons	ZWMSHOWBALLOON	
Combine Balloons	ZWMCOMBINEBALLOON	
Adding leader	ZWMADDLEADER	
Removing leader	ZWMADDLEADER	
Generate a BOM	ZWMPARTLIST	PLT
Processing BOM	ZWMTOTALBOMEDIT	TBE
3.Dimension		
Intelligent dimensioning	ZWMPOWERDIM	D
Multi-dimension	ZWMAUTODIM	DAU
Length Dimensioning	ZWMLINEARDIM	
Horizontal dimensioning	ZWMHORIZONTALDIM	
Vertical dimensioning	ZWMVERTICALDIM	
Aligned dimensioning	ZWMALIGNEDDIM	
Half-section dimensioning	ZWMHALFALIGNDIM	
Diameter dimension	ZWMDIAMETERDIM	
Radius dimension	ZWMRADIUSDIM	
Jogged radius dimensioning	ZWMJOGGEDRADIUSDIM	
Coordinate dimensioning	ZWM_ZB	
Arc length dimensioning	ZWMARCLENGTHDIM	

Chain dimensioning	ZWMCHAINDIM	
Baseline dimensioning	ZWMBASELINEDIM	
Center mark	ZWMCENTERDIM NIL	
Angular dimensioning	ZWMANGULARDIM	DAN
Leader dimensioning	ZWMANGULARDIM	NO
Chamfer dimension	ZWMCHAMFERSYM	CHS
Dimension combination	ZWMDIMJOIN	
Inserting dimension	ZWMDIMINSERT	
Dimension alignment	ZWMDIMALIGN	
Dimensional check	ZWMDIMCHECK	
Tolerance for query	ZWMDIMTOLQUERY	
Dimension style	DDIM	DD
4.Symbol		
Surface text	ZWMSURFSYM	SF
FCFrame	ZWMFCFRAME	FCF
Datum identifier	ZWMDATUMID	DTD
Feature Identifier	ZWMFEATID	
Datum target	ZWMDATUMTGT	
Taper & slope symbol	ZWMTAPERSYM	TPS
Center hole	ZWMCENTERHOLE	CTH
Circlemark	ZWMCIRCLEMARK	BJ
Break symbol	ZWMBREAKSYMBOL	BRS
Elevation symbol	ZWMELEVSYM	EF
Welding symbol	ZWMWELDING	WE
5.Creating view		
Section Line	ZWMSECTIONLINE	STL
Direction symbol	ZWMVIEWDIRECTION	
Detail view	ZWMDETAIL	
6. Text processing		
Dimension text	ZWMDIMTEXT	DTT
Technical requirements	ZWMTECHREQUEST	TRT
7 Drawing tools		
Intelligent line drawing	ZWMINTELLIGENTLINE	SS
Drawing a circle with a known circle center	ZWMCIRCLEBYC	CBC
Draw a circle with known endpoint	ZWMCIRCLEBY3P	CBP
Drawing arc with a known circle center	ZWMARCBYC	ASF
Drawing arc with a known end point	ZWMARCBY3P	ABP
Drawing mirror line	ZWMMIRRORLINE	DMRL
Hatch	BHATCH	H
Parallel line	ZWMPARALLELLINE	PAL
Vertical line	ZWMVERTICALLINE	VL
Tangent	ZWMTANGENTLINE	QTL
Common tangent	ZWMCOMMONTANGENT	CTT
Pipeline	ZWMPIPELINE	PPL
Perpendicular bisector	ZWMPERPBISECTOR	PBS
Angle line	ZWMANGLELINER	AN
Bisector (angular bisector)	ZWMANGLEBISECTOR	AB
Ray	ZWMRADIATION	
Center line	ZWMCENTERLINE	CL
Zigzag line	ZwmZigzagLine	
Waviness line	ZWMWAVILNESSLINE	WL

Rectangle	ZWMRECTANGLE	REC
8.Construction Tools		
Formula curves	ZWMFCURVE	FC
Fillet	ZWMFILLETAC	F
Chamfer	ZWMFILLETLC	CHA
Rupture line	ZWMSECTIONSYMBOL	SES
Inserting break symbol	ZWMBREAKSYMBOL1	BRS1
Breaking entities	ZWMBREAKENTIHSP	BRE
Dynamic extension	ZWMDYNAMICEXTEND	DY
Construction Line		
Construction Line	ZwmconstLines	CLIN
Automatic construction line	ZwmAutocLines	
Horizontal	ZwmConstHor	
Vertical	ZwmConstVer	
Cross	ZwmConstCrs	
Two points or an angle	ZwmConstHB	
Form a relative angle with a straight line	ZwmConstHW	
All-distance parallel	ZwmConstPar	
Half distance parallel	ZwmConstPar2	
Perpendicular to the connection between two points	ZwmConstLot2	
Perpendicular to a straight line	ZwmConstLot	
Bi-section line	ZwmConstHM	
Through point ray	ZwmConstxRay	
Through point straight line	ZwmConstxLine	
Z direction	ZwmConstZ	
Construction circle		
Construction circle	ZwmConstCircle	
Parallel construction line tangent to a circle	ZwmConstTan	
Construction line tangent to two circles	ZwmConstTc	
Construction line concentric with a circular	ZwmConstCc	
Construction line of end line of axis	ZwmConstCcrea	
Construction circle tangent to a straight line	ZwmConstC2	
Construction circle tangent to two straight lines	ZwmConstKr	
Rectangular construction line externally tangent to a circle	ZwmConstCircli	
Hole&Axis projection	ZWMHSPROJECTOR	HSP
Hole&Axis design	ZWMHOLEAXIS	
Intersecting line	ZWM_INTER	INT
Technical groove construction	ZWMCONSTRECESS	CSR
Single hole	ZwmSingleHole	SHO
Hole array	ZWMARRAYHOLE	AH
9. Aids		
Super Edit	ZWMSUPEREDIT	V

Tabular data pickup	ZWMTABLEDATAPICKUP	TB
Batch data pickup	ZWMDWPPLATAPICKUP	
Batch text search	ZWMDWPPLATAFINDREPLACE	
DWG data exploration	ZWMDWPPLATAVIEW	
Automatic alignment drawings	ZWMJIGSAWPRINT	JS
Layer transform tool	ZWMCHGLAYER	
Engineering Calculator	ZWMBASCALC	BSC
Style library synchronization	ZWMUPDATE	
11.Series Parts Design System		
Series Parts Design System	ZWM_SPART_OUT	PB
12.Super-symbol library		
Invocation of super-symbol library	ZWM_SYMOUT	FH
Hydraulic and pneumatic symbol library		YQFH
Electrical symbol library		DQFH
Mechanism symbol		JGFH
Metal structural part		JSFH
13.System maintenance tools		
Style configuration	ZWMSHSPLEMANAGER	
Word library	ZWMWORDLIBMNG	
Usertype title	ZWMTITLEDEFINE	
Usertype additional list	ZWMFJLDEFINE	
Custom paralist	ZWMCSLDEFINE	
Usertype drawing code	ZWMREVERSEDEFINE	
Definition of superblock	ZWMATTBLOCKDEF	
Usertype BOM head	ZWMBOMHEADDEFINE	
Usertype BOM body	ZWMBOMBODYDEFINE	
Irregular table pickup	ZWMTBLDATAPICKUPTITCONFIG	
2D regular table pickup	ZWMTBLDATAPICKUPBOMCONFIG	

[Note] Proficiency with the commands above will greatly help your work efficiency. It is suggested to grasp one or two-letter commands before go for the longer ones. This is also a good opportunity to exercise the left-handed applications and operations.

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