Smooth R Function

3.3 Smooth R function

(Milling (DRO)

Function:

When a milling machine is used, especially in the process of machining a mold, arc often needs to be machined on a work piece. If the arc surface is complex, or a lot of round angles need to be machined, or the arc or round angle needs to be accurately machined, a CNC milling machine should be utilized.

But in the daily machining process, only a simple arc surface or a round angle is needed with no requirements for the precision of the arc or round angle (particularly in the process of machining molds). If there is no CNC milling machine in the production line, the best way is to machine it with a manual milling machine as it saves time and efforts, compared to outsourcing it. In the past, an operator used to calculate the tool positioning in arc machining with a scientific calculator, but this method was time-consuming and liable for errors.

DRO provides a simple and easy positioning function for arc cutting tool, so the operator can perform arc machining in the shortest time. But before you decide to use smooth R function or CNC machining, please bear the following points in mind to make sure smooth R yields the best performance.

The R function group in DRO contains two R functions: smooth R function and simple R function.

Smooth R function:

Smooth R function is a function for full-functional arc machining. The operator can use the smooth R function to machine all types of most complex arc, even an arc to be connected to another arc (commonly known as R-to-R).

Advantages of smooth R function:

Smooth R function can be used to machine the most complex arc or even for complex machining in R-to-R.

Disadvantages of smooth R function:

Operation is complex and the operator needs to know the basic coordinate system in order to calculate the start point, the end point and the center.

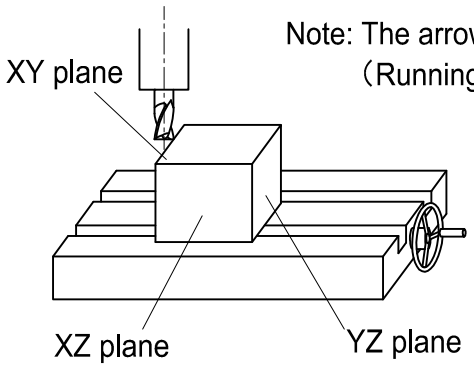
Understand the coordinate system:

An operator who has no CNC programming experience or who has not used the DRO R function before may have difficulty in mastering the concept of coordinate system. Coordinates are a pair of numbers used to determine positions.

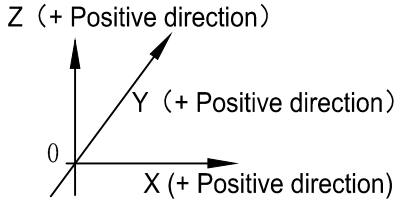
When using the DRO R function, the center coordinates of the arc surface, and the coordinates of the start point and those of the end point must be input to inform DRO about the geometric parameters of the arc surface to be machined.

In the process of installing a DRO, professional customer service installers will generally set the display orientation in the same direction as the machine axis. In a general milling machine, the dial direction is shown as below. Therefore, the DRO display direction will normally be set as follows.

3.3 Smooth R Function

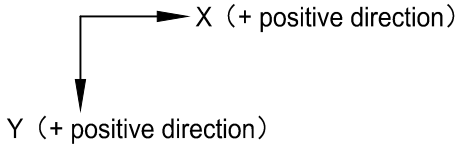


Note: The arrow indicates the positive direction of the coordinate
（Running direction of the tool relative to the table ）

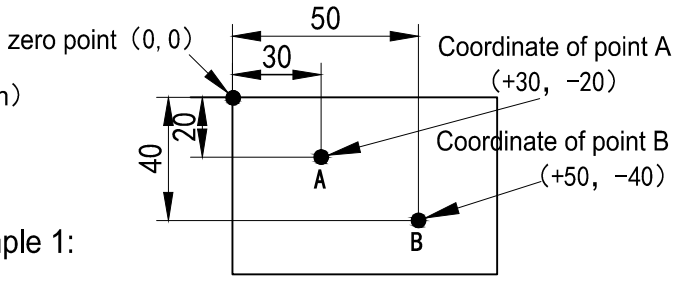


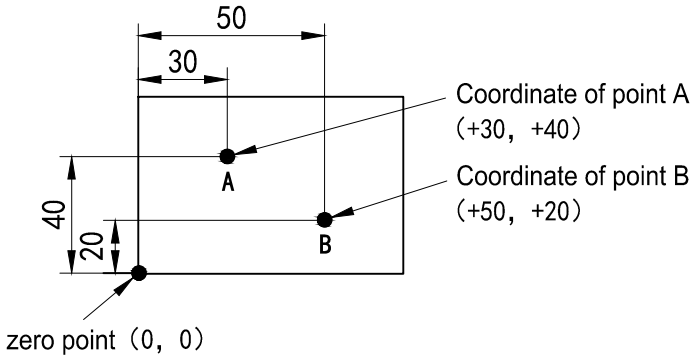
What are coordinates?

Coordinates are used to indicate positions. During plane machining, each set of coordinates contains two values, respectively corresponding to the distances from the zero point on the plane. The following is a simple example.

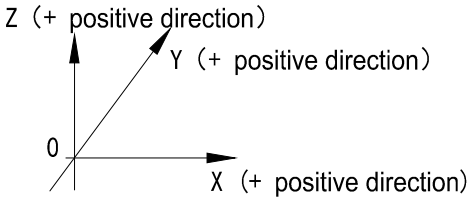


Example 1:



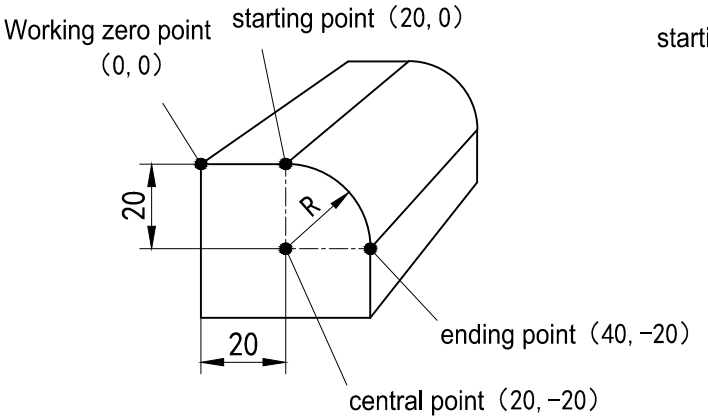


Example 2:

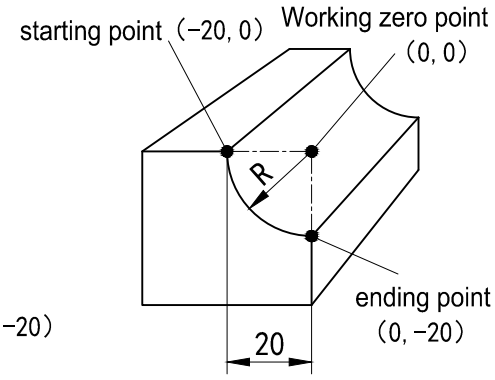


3.3 Smooth R Function

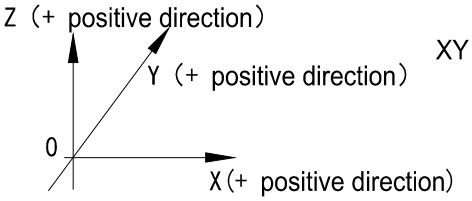
Example 3:

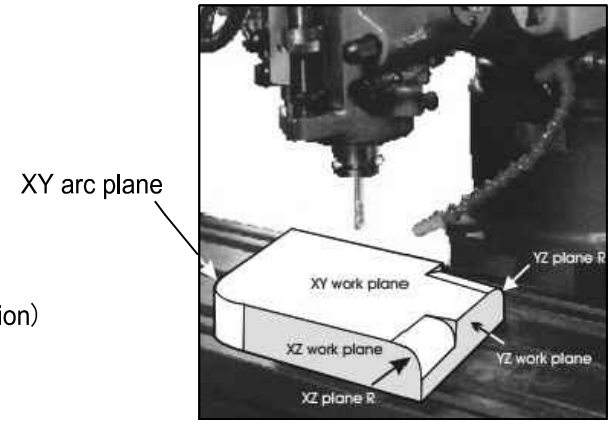


Example 4:

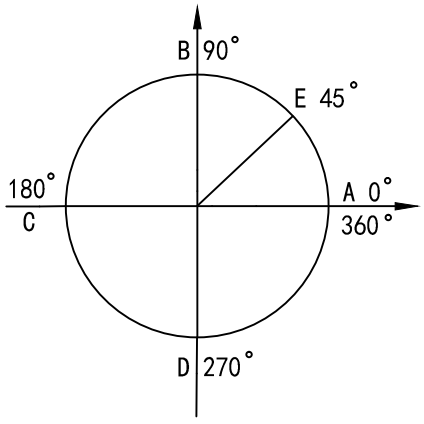


During the machining process, the coordinate of the machine tool are shown in the figure below, and the indication of the machining plane is shown in the figure.





The definition of the angle and direction:



AB arc (from A to B: the starting angle A is 0°, and the ending angle B is 90°)
(from B to A: the starting angle B is 90°, and the ending angle A is 0°)

ED arc (from E to D: the starting angle E is 45°, and the ending angle D is 270°)
(from D to E: the starting angle D is 270°, and the ending angle E is 45°)

3.3 Smooth R Function

Smooth "R" Arc Function:

Procedure for using the smooth arc machining function:

Load and clamp the work piece, tool setting as shown in figure A, figure B and figure C, and then zero every axis (set the position point of the tool setting to zero).

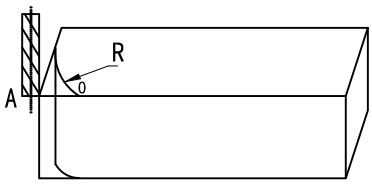


Figure A

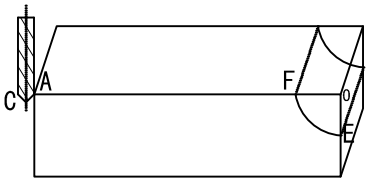


Figure C

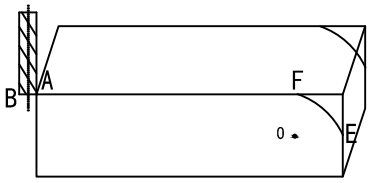


Figure B

Step 1: Press the key to enter the smooth R arc function.

L-LEN:NO.1-NO.4 In Smooth R function,we can set four groups of data.Every group has no interference with each other.

Name - - - - No.1-No.4 the drawing name setting

PLANE - - Plane selection (XY YZ XZ)

CT-POS X - - Center of a circle X coordinates

CT-POS Y - - Center of a circle Y coordinates

DIA - - - - - - - diameter of the tool

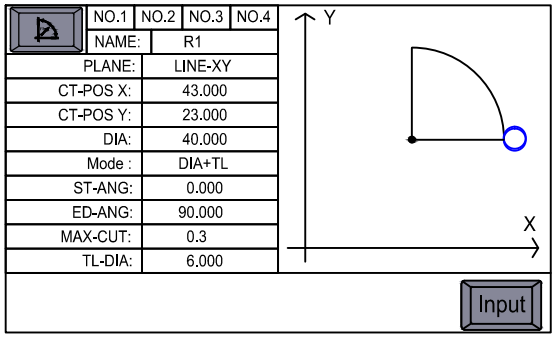
Mode - - - - - Select the machining place to be concave or convex.

ST-ANG - - - starting angle

ED-ANG - - - end angle

MAX-CUT- - maximum cutting amount

TL--DIA - - - - diameter of the tool



Smooth R Function

Example 1 :take the shown in the machining drawing as example

The workpiece size are shown in the below figure.

1.zero the tool setting.

2.select smooth R mode.

3.Select XY plane for machining arc (LINE-XY)

4.Input the coordinate of circle center CT-POS=(43.23)

5.Input arc diameter DIR=40.000

6.Input the arc machining lane

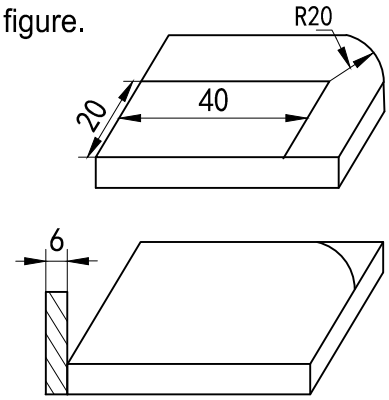
Mode = DIA+TL(Select the convex to machine)

7.Input the ST-ANG=0

8.Input the ED-ANG=90

9.Input the MAX-CUT-0.3

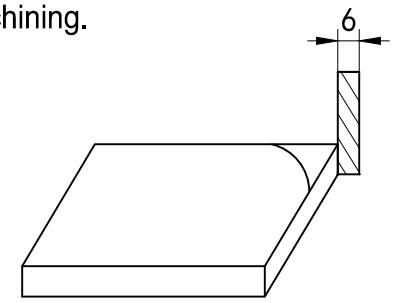
10.Input the TL-DIA=6.000



（Figure A）

3.3 Smooth R Function

The following is the operation figure for smooth R arc machining.



（Figure B）

Step 1: Zero the tool setting

Step 2:Press [△] to enter the Smooth R Function.

Step 3:Choose the machining drawing
No.1-No.4By pressing [←] [→] to choose,
after choosing well,press [↓] to enter
the drawing name setting.

Step 4:Drawing name setting (NAME)
Press [Input] to set the drawing name,
choose the letter by pressing [↑] [↓] [←] [→] .
Choose the digit by press the digit button,then
press [Input] to input.After setting well,press [ENT] to save.

Step 5:Select the XY plane as the machining plane (LINE-XY) Press [Set] to set the machining type.

Step 6:Input the coordinate of circle center.
If setting the tool as figure A CT-POSX=43 CT-POS Y=23
If setting the tool as figure B CT-POS X=23 CT-POS Y=23

Step 7:Input arc diameter DIA = 40, and press [Input] to do the data inputting.

Step 8:Select the convex as the machining plane Mode = DIA+TL
Press [Set] to set the machining type.

Step 9:Input the starting angle ST-ANG=0

Step 10:Input the end angle of the arc ED-ANG=90

Step 11: Input the maximum cutting amount MAX-CUT=0.3

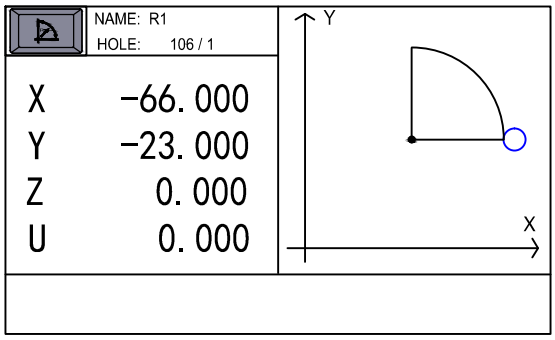
Step 12 :Input the tool diameter TL-DIA=6.000

Step 13:After setting well of above parameter,press [ENT] to enter the machining interface,and it will display the fist point which will be machined.

Step 14: Move the machine tool until the axis displays zero,i.e.the R starting point

Step 15: Press [←] [→] to display the position of each machining point,and move the machine tool until the axis displays zero,i.e.the position of each point of R arc.

Step 16:Press [C] to exit the smooth arc function anytime.



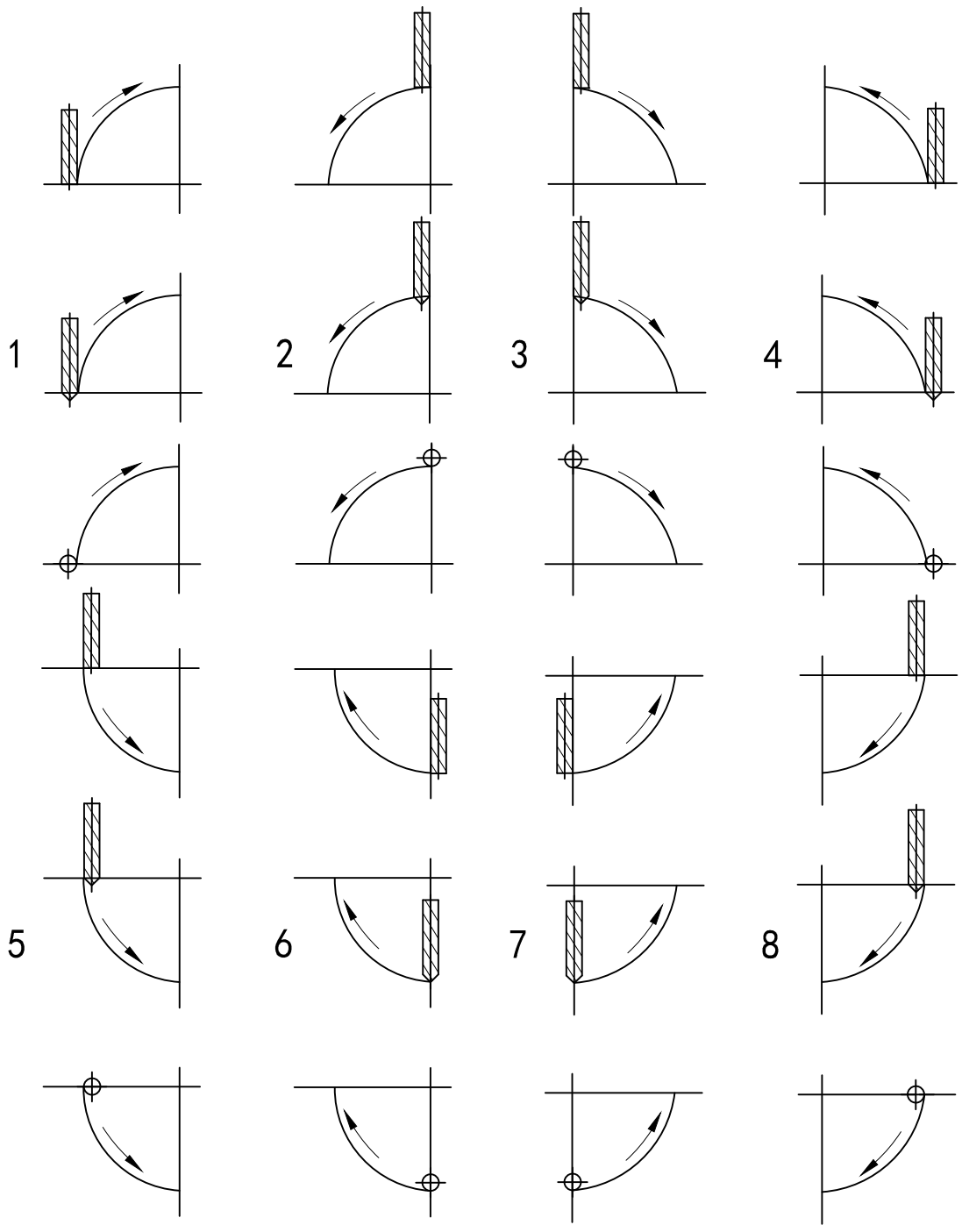
If set the tool as figure B

Simple R function

3.4: Procedures for using simple R function

(Applicable to: Milling DROs)

Function: If you are not familiar with the concept of plane coordinate, you may have difficulty of using the smooth R arc function. If very simple arcs are needed for machining and there is no high requirement for the smoothness, the simple R arc calculation function can be used at this moment. Generally, the arc machining mainly includes the following 8 types, and the flat end milling cutter or arc milling cutter is used for the machining.



3.4 Simple R function

Procedure for using simple R function:

Place the tool directly opposite to the starting point of the arc, and press the key to enter R arc calculation function. Please refer to figure (1) for the method to place the tool directly opposite to the starting point of the arc.

L-LEN:NO.1-NO.4 In Simple R function,we can set four groups of data.Every group has no interference with each other.

Name - - - - No.1-No.4 the drawing name setting

PLANE - - Plane selection (XY YZ XZ)

CT-POS X - - Center of a circle X coordinates

CT-POS Y - - Center of a circle Y coordinates

Radius - - - - - Arc radius

Mode - - - - - Select the machining placeto be concave or convex.

Quadrant - - - Select the machining quadrant

Direction- - - clockwise(0) anticlockwise(1)

MAX-CUT- - maximum cutting amount

TL--DIA - - - - diameter of the tool

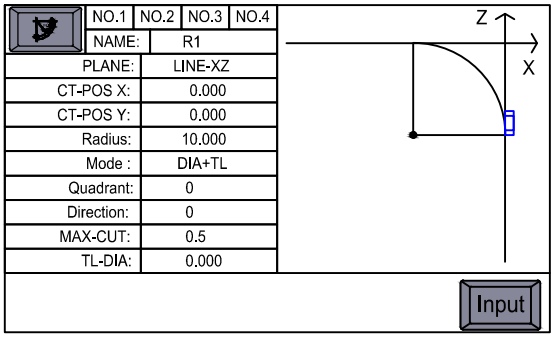


Figure A

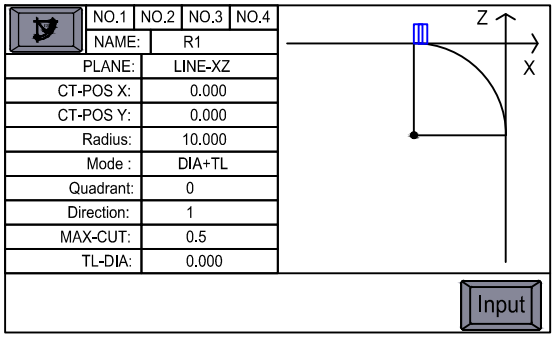
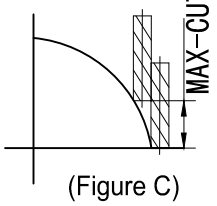
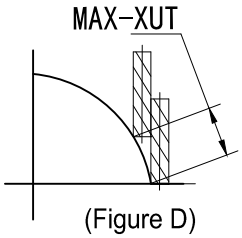


Figure B

In machining the arcs on XZ and YZ planes, "MAX-CUT" in the simple R function refers to the amount of feed for every step as shown in figure (C).The MAX-CUT can be changed during the machining process. When machining the arc on XY plane, "MAX-CUT" refers to the cutting amount of every tool. As shown in figure (D), the cutting amount for every tool is equal.

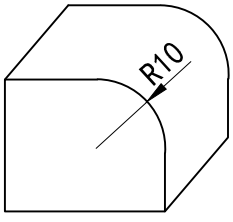


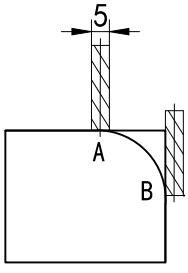
(Figure C)



(Figure D)

Example 1: Take machining the arc shown in the figure as example:





3.4 Simple R function

Step 1: Place the tool directly opposite to the starting point (A or B) of the arc, then press the [ ] key to enter ARC function.

Step 2:Choose the machining drawingNo.1-No.4By pressing [←] [→] to choose,after choosing well,press [↓] to enter the drawing name setting.

Step 3:Drawing name setting (NAME) Press [Input] to set the drawing name, choose the letter by pressing [↑] [↓] [←] [→] .Choose the digit by press the digit button,then press [Input] to input.After setting well,press [ENT] to save.

Step 4:Select the XZ plane as the machining plane (LINE-XZ) Press [Set] to set the machining type.

Step 5:Input the arc radius R=5, and press [Input] to do the data inputting.

Step 6:Select the machining plane as convex Mode = DIA+TL,Press [Set] to set the machining type.

Step 7:Select the machining quadrant Quadrant = 0 ,Press [Set] to set the machining type.

Step8:Press [Set] to choose the simple R machining type

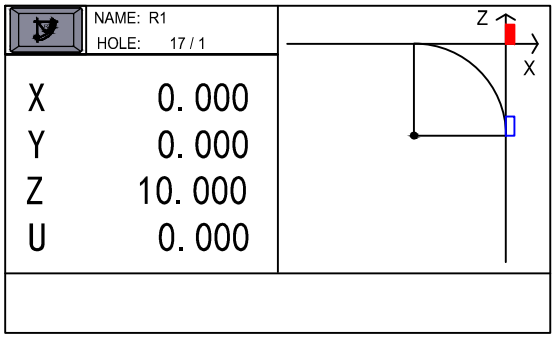
If take the A as the starting point and set the direction=1 to proceed the machining in the clockwise direction(shown in figure A)

If take the B as the starting point and set the direction=0 to proceed the machining in the anticlockwise direction(shown in figure B)

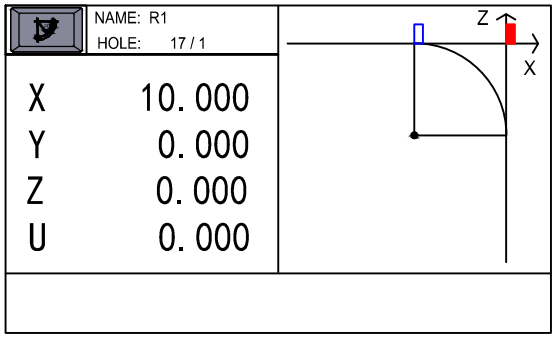
Step 9: Input the maximum cutting amount MAX-CUT=0.5

Step 10: Input the diameter of the tool TL-DIA=0

Step 11:After setting well these parameter,press [ENT] to enter the machining interface.



Take A as the starting point（0，0）



Take B as the starting point（0，0）

Step 12:Press [←] [→] to display the next point for former point.Turn the machine tool until displaying zero.

Step 13: Press [C] to exit simple R function at any time.

Rectangle Chambering

3.5 Rectangle Chambering

(Applicable to: milling machine)

Machining the work-piece chamber shown in figure A,the chambering function may be used.Operators can operate conveniently following the prompts.Shown in figure B,the machining starts from the chamber center and proceeds along the direction indicated by the arrow.The completion of the machining is shown in figure C.

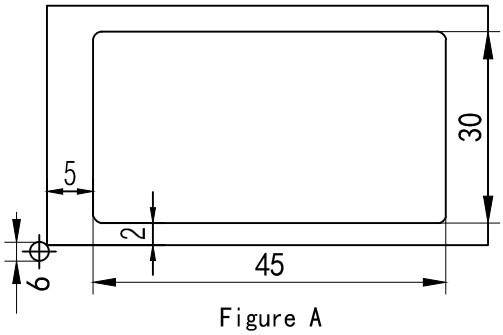


Figure A

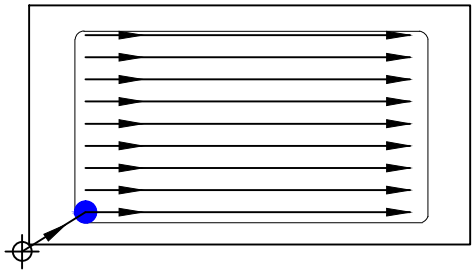


Figure B

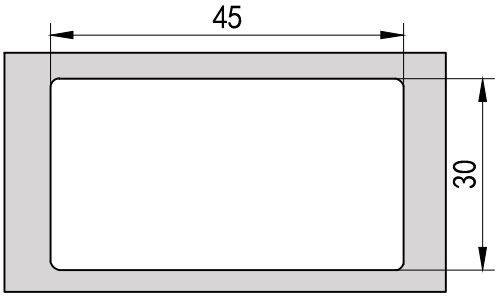


Figure C

L-LEN:NO.1-NO.4 In Rectangle Chambering function,we can set four groups of data.Every group has no interference with each other.

Name - - - - No.1-No.4 the drawing name setting

PLANE - - Plane selection (XY YZ XZ)

CT-POS X - - X axes coordinate of chamber position

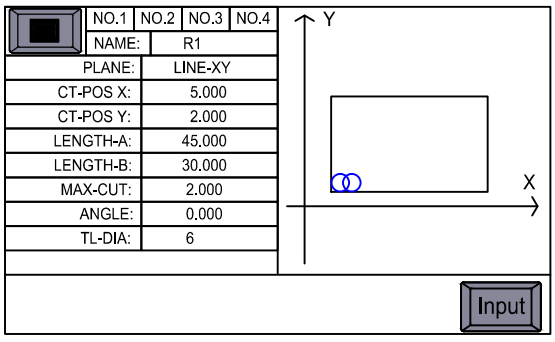
CT-POS Y - - Y axes coordinate of chamber position

LENGTH-A--The side length A of the rectangle

LENGTH-B--The side length B of the rectangle

ANGLE --The rotated angle of rectangle

TL--DIA - - - - diameter of the tool



The parameter setting of rectangle chambering

3.5 Rectangle Chambering

Example: For machining the work piece shown in figure A, the operating procedures are as follows

Step 1: Set the tool according to the position shown in figure A, and zero, then press the key to enter the chambering function.

Step 2:Choose the machining drawingNo.1-No.4By pressing ← → to choose,after choosing well,press ↓ to enter the drawing name setting.

Step 3:Drawing name setting (NAME) Press Input to set the drawing name, choose the letter by pressing ↑ ↓ ← → .Choose the digit by press the digit button,then press Input to input.After setting well,press ENT to save.

Step 4:Select the XY plane as the machining plane (LINE-XY) Press Set to set the machining type.

Step 5:Set the X axes coordinate of chamber position to be CT-POS X=5,press Input to input.

Step 6:Set the Y axes coordinate of chamber position to be CT-POS Y=2,press Input to input.

Step 7:Set the side length A of the rectangle to be LENGTH-A=45,press Input to input.nput.

Step 8:Set the side length B of the rectangle to be LENGTH-B=30,press Input to input.nput.

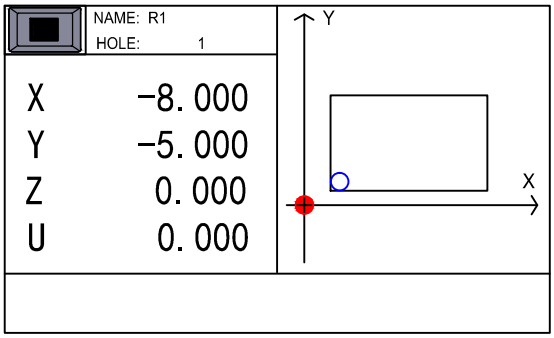
Step 9:Set the maximum cutting to be MAX-CUT=2,press Input to input.

Step 10:Set the rotated angle of rectangle to be ANGLE=O,press Input to input.

Step 11:Set the diameter of the tool to be TL=DIA=6,press Input to input.

Step 12:After inputting well,press ENT to enter the machining state.

Step 13:Press ← → to display the next machining postion,and move the machine tool following the prompts until X axes and Y axes display zero.Press C to exit the chambering function.

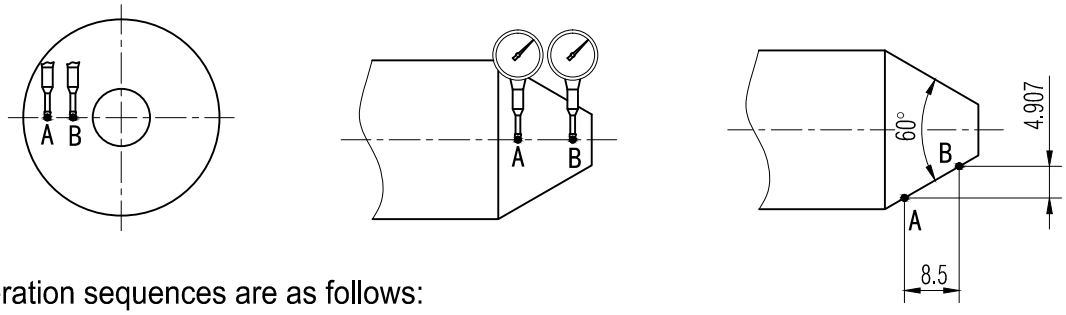


The interface of rectangle chambering

Taper measuring

3.6 Taper measuring

The function is used to turn tapered work piece and could measure the taper of the work piece in machining.



Operation sequences are as follows:

As shown in the figure, make the contact of the lever indicator to contact position A on the work piece surface and press until the lever indicator points to zero.

Step 1: Press [△] key to enter taper measuring function.
Step 2:Select the plane to be LINE-XY by pressing [←] [→] .
Step 3:Move the table to make the measuring tool such as the dial indicator touch the point A on the work-piece until the dial indicator turn to be zero.Press [ENT] to find the position of point B.
Step 4:Move the table to make the measuring tool such as the dial indicator touch the point B on the work-piece until the dial indicator turn to be zero
Step 5:Press [ENT] to calculate.(Shown in figure C)

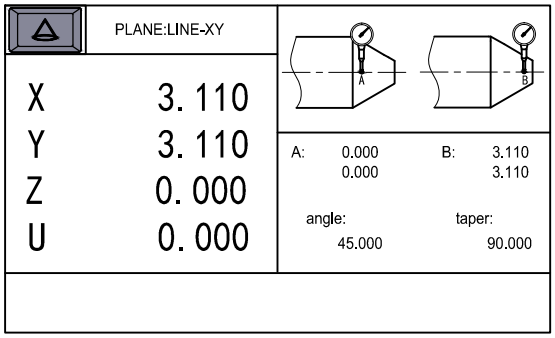


Figure C

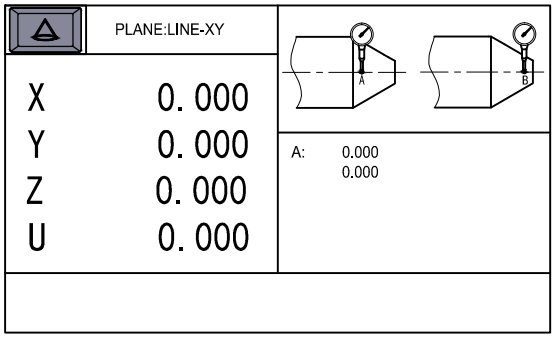


Figure B

Point A - - The coordinate of point A

Point B - - The coordinate of Point B

Step 6:Press [C] to exit the taper measuring function.

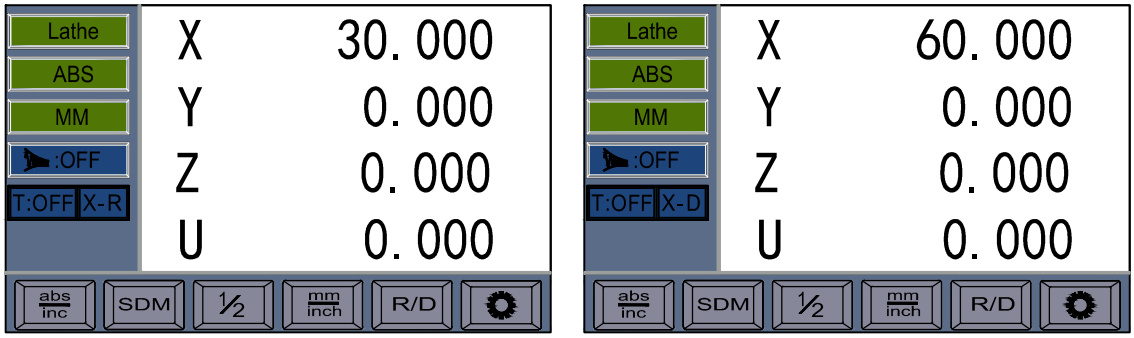
Diamete/radius Conversion

3.7 Diamete/radius Conversion

(Applicable to: Lathe)

Function Introduction:

When the DRO has been set as the lathe mode,press R/D to change the X axes to be diameter.Press R/D again to change the X axes to be radius



Radius Diameter

Note：When the DRO is used as a lathe meter, only the X axis has radius/diameter conversion and Y and Z axis don't have this function.

Vectoring

3.8 Vectoring(suitable for lathe which has the rotatable tool post)

Vectoring function is used for displaying combined movement of either X – Z axis pair or Y– Z axes pair taking into consideration angle between Y and Z i.e. α. The resulting combined movement is displayed on X and Y axis.

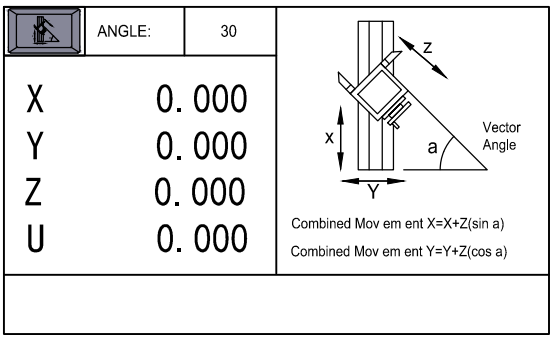
Step 1,Press  to enter the vectoring function.

Step 2,Press 

to input the rotated angle of the tool post.

Step 3:Move the Z axes,then the combined movement of XZ will display on X axis,the combined movement of YZ will display on Y axes.

Step 4,Press [C] to exit the vectoring function.



The interface of the vectoring function.

TOOL Storeroom

3.9. TOOL Storeroom

Function introduction:

Various tools are needed to turn different work pieces or their surface, so we have to load/unload tools and set tools. To save the operator's time, the lathe function of the DRO is provided with the function of 16 sets of tool magazine.

Note: The function of 16 sets of tool magazine could only be used together with a tool post on the lathe. Don't use this function without a tool post to avoid errors in machining.

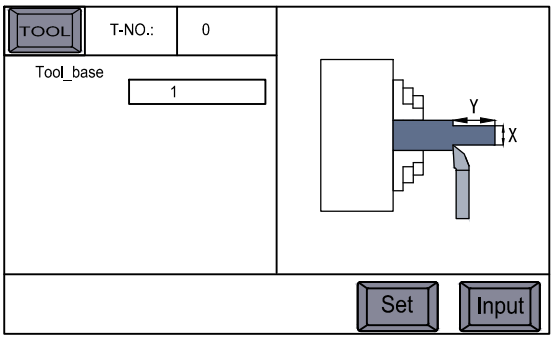


Figure A

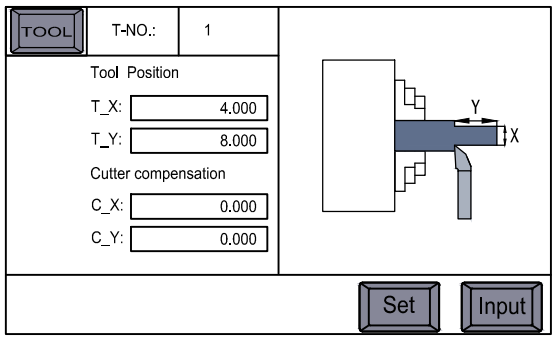


Figure B

Basic settings:

1.The parameter of tool setting.

T-NO. - - Tool No.(Shown in figure A)(It can save 16 tools as maximum.When the T-NO.=0,it indicates the DRO entering the setting of base tool and turn off the tool function.

Tool-base - -checking and setting the base tool(shown in figure A)

T-X- - the X axes of the tool from the center point of the work-piece(shown in figure B)

T-Y- - the Y axes of the tool from the center point of the work-piece(shown in figure B)

C-X- - the compensation of tool,this is the compensated value after tool abrasion.

C-Y- - the compensation of tool,this is the compensated value after tool abrasion.

2.Set a base tool on the center point in INC coordinate.

Here we need to fix the work-piece on the holder first,and enter the TOOL function to select one tool as the base tool(shown in figure A).Use the base to machine the work-piece to be smooth,and then measure out the value of X and Y for work-piece.And input this value to the base tool T-X and T-Y(shown in figure C).Then the base tool setting is done.

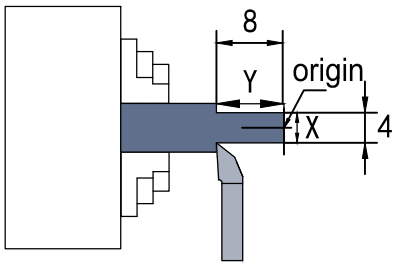


Figure C

3. Set the second tool according to the base tool.Input the measured value of T-X and T-Y of the second tool(shown in figure E).We can use the same way of the base tool to measure the second tool.This setting apply to the following tool.

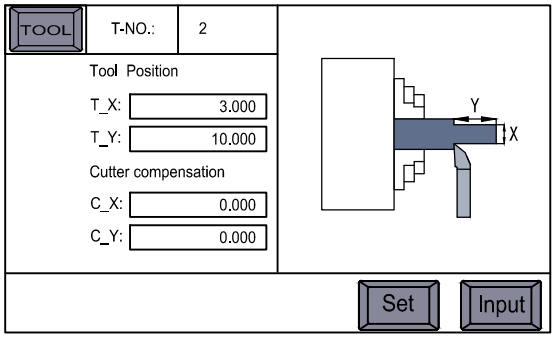


Figure D

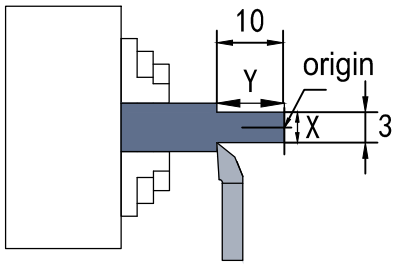


Figure E

4. Save the tool No. and the corresponding distance which is relative to the base tool into the DRO.
5. During the machining,operator can input the tool No. which will be used,then the DRO will display the distance between the target tool and the base tool.Move the corresponding axes until the value of X and Y to be zero.
6. When need to machine another batch of work-piece,there is no need to determine the tool again.Fix the work-piece to the holder,and set the origin of the base tool again.Then other tools will be adjusted automatically.
7. Change the base tool.If any err of the base tool happened during the machining,we can select another tool as the base tool.There is no need to determine the position of other tools.Replace the former base tool,and determine the position of this new tool,then this new tool will be taken as a normal tool instead of the base tool.

8.Tool compensation.

If there is any damages on the tool during the machining.We can measure out the damaged value of the tool,and input this value to C-X and C-Y.The compensation will be made by the DRO at then.

Tool setting

Step 1:Press TOOL to enter the tool function.

Step 2:Base tool setting.Set the first tool as the base tool.

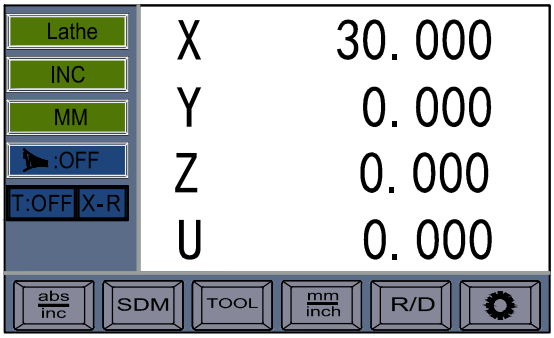
Press ← → to set the tool No. as 0(T-NO.=0).Then enter the base tool setting,Press Input to enter the tool setting.Press Set to input the tool No. .After inputting well,press ENT to save.(shown in figure A).

Step 3:After the setting of the base tool,press ← → to select the No.1 tool.Press Set to proceed the setting.Select the corresponding parameter by pressing ↑ ↓.And press Input to proceed the setting. (Shown in figure B)

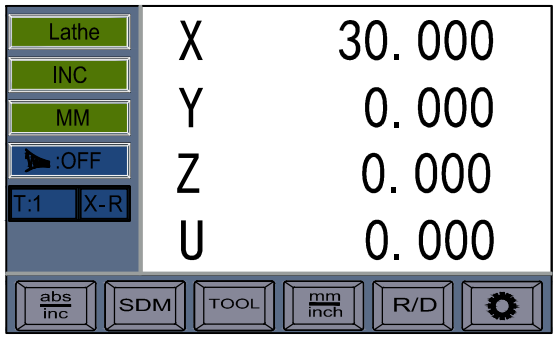
Step 4:No.2 tool setting.Press ← → to switch the tool to No.2 and then input the corresponding parameter(Shown in figure D).

Step 5:After setting well all the tool step by step,then user can switch the tool very fast.
Press TOOL to enter the tool function. Press the digits to select the target tool,then the DRO will switch the tool to the target one.Meanwhile it can be switched by press Input to input the tool No..

Step 6:Log off the tool function.
Press TOOL to enter the tool function.And press 0 or press Input and then 0 to Log off the tool function.



Tool function off



Tool function off

Note:Operator can zero the value when the base tool is under using in INC coordinate.It cannot be zeroed for any other tool.

Congruous Output Function of EDM

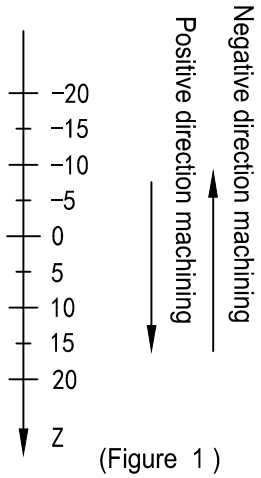
3.10 Congruous Output Function of EDM

(Applicable to EDM DRO)

1. Function introduction:

This function is used for the specialized machining by the electric discharge machine (i.e. EDM). When the target value on Z axis of the EDM equals the current value, the DRO will output a switch signal to control the EDM to stop the depth machining.

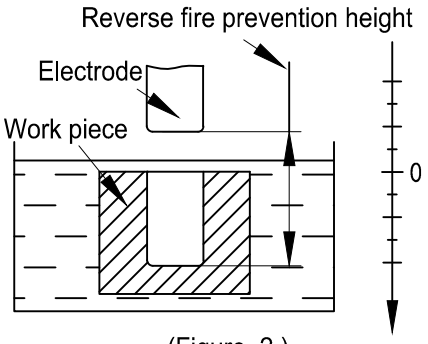
The setting for Z axis direction of D60-3E type DRO is shown in figure 1, i.e., the depth is larger, the coordinate value displayed by Z axis is larger. Since the machining is started, as the depth increases gradually, the value displayed by Z axis increases gradually.



(Figure 1)

According to the set direction on the Z axis, the machining direction includes the positive direction and negative direction. When the electrode drops, the machining direction is from upper to lower part, and the DRO value will increase. We call the machining direction as "positive direction machining" which is the normal direction. When the electrode rises, the machining direction is from lower to upper part, and the DRO value will decrease. We call the machining direction as "Negative", namely, "negative direction machining" (as shown in figure1).

The D80 DRO with EDM function also has the function of "reverse fire prevention height" which is not offered by other similar DROs. This function is one kind of intelligent safety protection device of position following and detecting. When the carbon deposition occurs on the electrode surface in the process of positive direction machining, especially in the long-time machining or round-the-clock machining without supervision by people, the carbon deposition will increase gradually along the reverse direction without being cleared up by people. Once the electrode exceeds the liquid level, the fire may tend to break out to cause the damage. This “reverse fire prevention height” function is set for this problem. If the “reverse fire prevention height” is set, the DRO will give a warning and an alarm when the height enhanced by the electrode exceeds the height (i.e., the reverse fire prevention height) between electrode and the machined plane depth. Meanwhile, the output signal will shut down the EDM automatically to completely eradicate the chance of fire breaking out. (See figure 2)



(Figure 2)

3.10. Congruous Output Function of EDM

2. Specific Operations:

1: Before machining, set the parameters of "reverse fire prevention height", "exit mode", "machining direction" and "EDM mode".

2: Firstly move the main axis electrode of Z axis to make it touch the work piece reference, and then zero Z axis or set the number.

3: Press the Input key, and input the depth value for machining (the depth value will be displayed on X axis), such as 10.00, then press the ENT key to confirm. After the confirmation, press the ENT again to exit "DEPTH" and enter "EDM" state for machining.

4: "The target value of the machining depth" will be displayed on X axis. will be displayed on "Dpeth" axis. The value on "Dpeth" axis is the machined depth value of the work piece.will be displayed on Z axis. Note: The value on Z axis is the value of position where the main axis electrode of Z axis is located.

5: After the machining is started, the value displayed on "Dpeth" axis will get on for the target value gradually. If the electrode rises and drops repeatedly at this moment, the value displayed on Z axis will change accordingly. However, the value displayed on "Dpeth" axis will not change and always indicate the machined depth value.

6: When the value displayed on Z axis equals the target value, the limit switch will close, and the EDM will stop machining, also, the information screen will display "FINISH". According to the setting made by the operators, there have two exit modes: I. Automatic mode. Exit the EDM machining state automatically and restore the displaying state before machining. II. Pause mode. The screen always shows " END", and it need to press the C key to exit and restore the original displaying state.

3.EDM parameter introduction

Zero all axis' value before entering EDM function.Then press EDM to enter the parameter setting interface.

L-LEN:NO.1-NO.4 In EDM function,we can set four groups of data.Every group has no interference with each other.

Name - - - - No.1-No.4 the drawing name setting

DIR - - - Machining direction

POSITIVE indicates the positive machining direction.

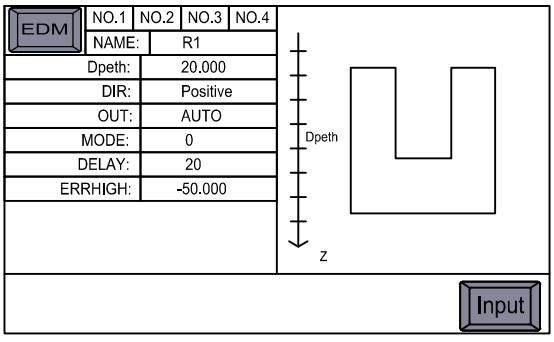
NEGATIVE indicates the negative machining direction.

Press Set to set.

OUT - - Setting for exit mode

STOP indicates the stop mode.

AUTO indicates the automatic mode Press Set to set



EDM Setup interface

3.10. Congruous Output Function of EDM

MODE - - - EDM machining mode.

Press the Set key to select MODE 0. The output states of the relay in MODE 0 are as follows:

a. When the power is off, the relay coil is OFF.
b. When the CPU is not initialized, the relay coil is OFF.
c. When the normal state output of booting is 1, the relay coil is ON.
d. When EDM function outputs 0 in operation, the relay coil is ON.
e. When EDM outputs 0 in depth, the relay coil is OFF.

Press the Set key to select MODE 1. The output states of the relay in MODE 1 are as follows:

a. When the power is off, the relay coil is OFF.
b. When the CPU is not initialized, the relay coil is OFF.
c. When the normal state output of booting is 0, the relay coil is OFF.
d. When EDM function outputs 1 in operation, the relay coil is ON.
e. When EDM outputs 0 in depth, the relay coil is OFF.

DELAY - -Post-set time,the unit is 0.1 second(when the model is set as the exit automatically OUT-AUTO).This time can be set.Press Input to proceed the setting.

ERRHIGH - reverse fire prevention height.

The machining defaulted by the DRO is the positive direction machining. As in example 1 and example 2, you should set the positive direction machining as the machining at first in the positive direction machining; as the work piece shown in the machining figure (F), you should set the negative direction machining as the machining direction before machining in the negative direction machining. Otherwise, after entering the machining, the DRO will identify that the machining has been completed and exit the machining.

4. Examples of Positive Direction Machining

Example 1: Machining the work piece as shown in Figure (A) , The work piece and electrode are shown in figure (B). Please set the positive direction machining as the machining direction at first.

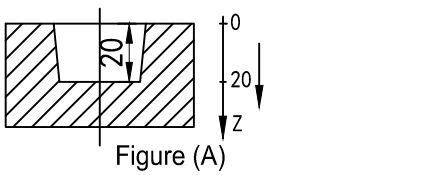


Figure (A)

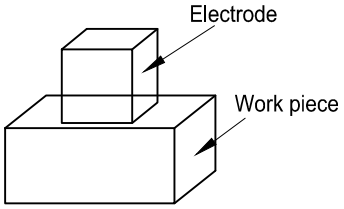
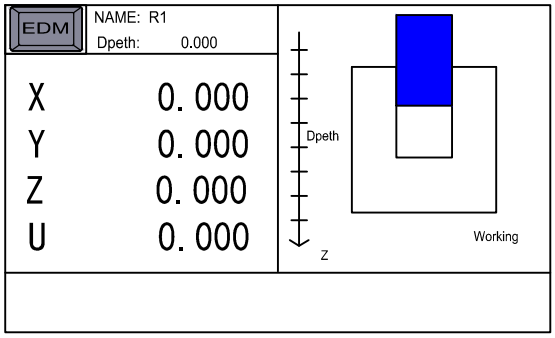


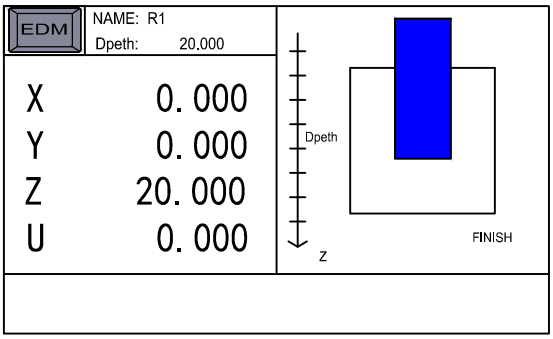
Figure (B)

3.10. Congruous Output Function of EDM

Step 1: As shown in figure (B), move the main axis electrode to make it touch the work piece and then press the [X], [Y] and [Z] keys to zero.
Step 2: Press the [EDM] key to enter the machining.
Step 3: Set the machining depth. DEPTH = 20
Step 4:Set the machining direction as positive DIR=POSITIVE
Step 5:Set the exit mode as the automatic OUT=AUTO
Step 6 :Set the EDM mode MODE=0
Step 7:Set the post-set time as DELAY=20
Step 8 :Set the reverse fire prevention height ERRHIGH=-50
Step 9:After setting well of the above parameter,press [ENT] to enter the machining interface.
Step 10: When the value displayed on Z axis equals the target value, the limit switch will close; the information window on the right will display " FINISH" for 2 seconds, then back to the state before machining.



EDM machining interface



EDM machining finish

Example 2: Machining the work pieces as shown in Figure (C)
Please set the positive direction machining as the machining direction at first.

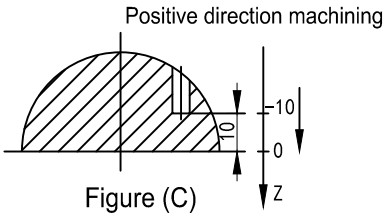


Figure (C)

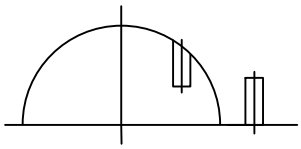


Figure (D)

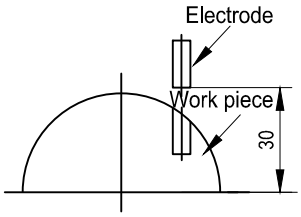
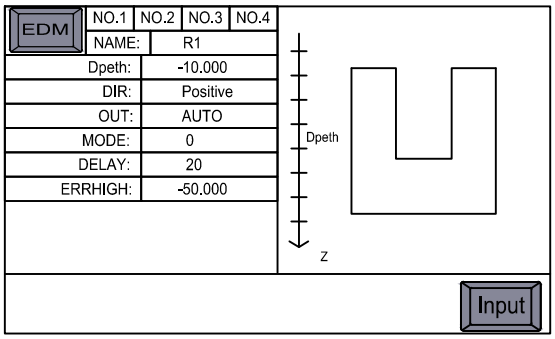


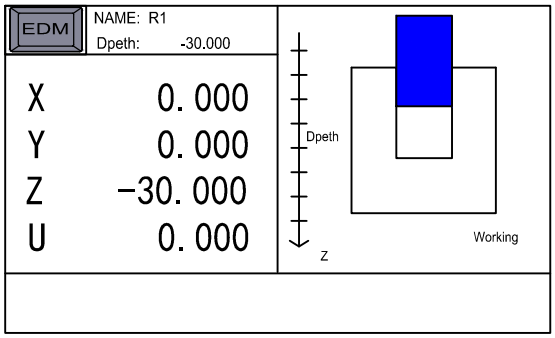
Figure (E)

Step 1: As shown in figure (B), move the main axis electrode to make it touch the work piece and then press the [X], [Y] and [Z] keys to zero.
Step 2: Press the [EDM] key to enter the machining.
Step 3: Set the machining depth. DEPTH = -10
Step 4:Set the machining direction as positive DIR=POSITIVE
Step 5:Set the exit mode as the automatic OUT=AUTO
Step 6 :Set the EDM mode MODE=0
Step 7:Set the post-set time as DELAY=20
Step 8 :Set the reverse fire prevention height ERRHIGH=-50
Step 9:After setting well of the above parameter,press [ENT] to enter the machining interface.

Step 10: When the value displayed on Z axis equals the target value, the limit switch will close; the information window on the right will display " FINISH" for 2 seconds, then back to the state before machining.



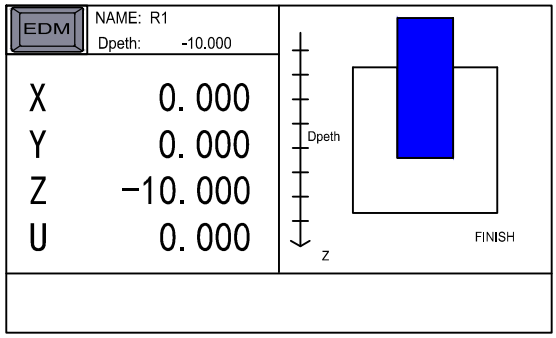
EDM setting interface



EDM machining interface

5. Examples of Negative Direction Machining

Example 3: Machining the work pieces as shown in Figure (F) ,Please set the negative direction machining as the machining direction.



EDM machining finish

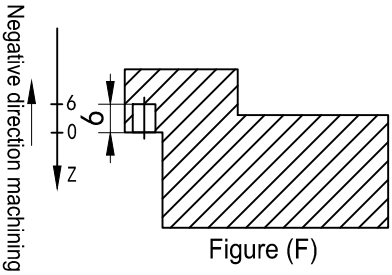


Figure (F)

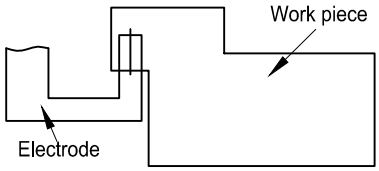
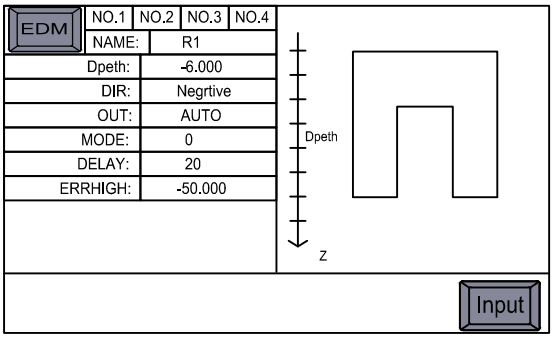


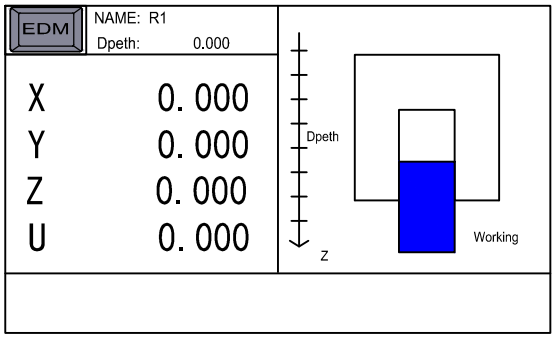
Figure (G)

Step 1: As shown in figure (B), move the main axis electrode to make it touch the work piece and then press the [X], [Y] and [Z] keys to zero.
Step 2: Press the [EDM] key to enter the machining.
Step 3: Set the machining depth. DEPIH = 6
Step 4:Set the machining direction as positive DIR=NEGRTIVE
Step 5:Set the exit mode as the automatic OUT=AUTO
Step 6 :Set the EDM mode MODE=0
Step 7:Set the post-set time as DELAY=20
Step 8 :Set the reverse fire prevention height ERRHIGH=-50
Step 9:After setting well of the above parameter,press [ENT] to enter the machining interface.
Step 10: When the value displayed on Z axis equals the target value, the limit switch will close; the information window on the right will display " FINISH" for 2 seconds, then back to the state before machining.

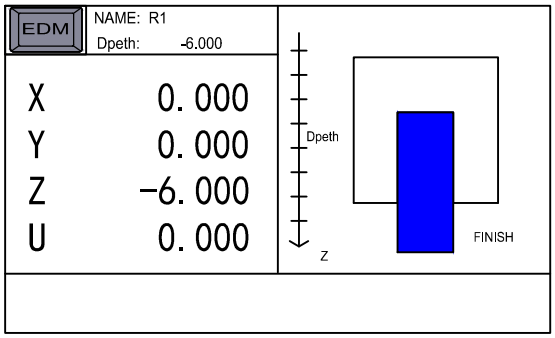
3.10. Congruous Output Function of EDM



EDM setting interface



EDM machining interface



EDM machining finish

6. Use PCD Function together with EDM Function

In PCD function, the DRO can call the EDM function to complete the EDM machining for the PCD. The specific operation procedures are as follows:

1) Press the key to enter the PCD function to set parameters (please refer to the PCD function setting). After setting all parameters, press the EDM key to enter the PCD machining. When displaying the position coordinate of the first machining, move the table to make the electrode aligning the first machining hole.

2） Press the EDM key to input the EDM parameter setting and machining state (refer to the EDM parameter setting for the EDM parameter setting method), and input the machining depth for EDM machining. After the machining is completed, press the EDM key to exit EDM machining and enter the PCD machining. Press the key to display the position coordinate of the second hole. Move the table to make the electrode aligning the next machining hole.

7. Use PLD Function and EDM Function Cooperatively

In PLD function, the DRO can call the EDM function to complete the EDM machining for the PLD. The specific operation procedures are as follows:

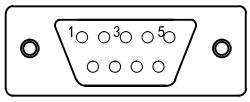
1) Press the key to enter the PLD function to set parameters (please refer to the PLD function setting). After setting all parameters, press the ENT key to confirm entering the PCD machining. The position of first machining hole is displayed in coordinate. Then move the table to make the electrode aligning the first machining hole.

2） Press the EDM key to enter the EDM parameter setting and machining state (refer to the EDM parameter setting for the EDM parameter setting method), and input the machining depth for EDM machining. After the machining is completed, press the EDM key to exit EDM machining and enter the PLD machining. Press the ← key to display the position coordinate of the second hole. Move the table to make the electrode aligning the next machining hole.

8. EQUAL OUT port of rear base plate

The output of EQUAL OUT is the relay output and the contact capacity is: 1.0A30VC, 0.5A125VAC, 0.3A60VDC.

9-pins Socket Pin Number	Signal type	9-pins lead
1	OFF (NC Port)	Black
3	COM (Common Port)	Yellow
5	NO (NO Port)	Red

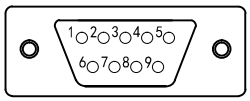


4. Appendix

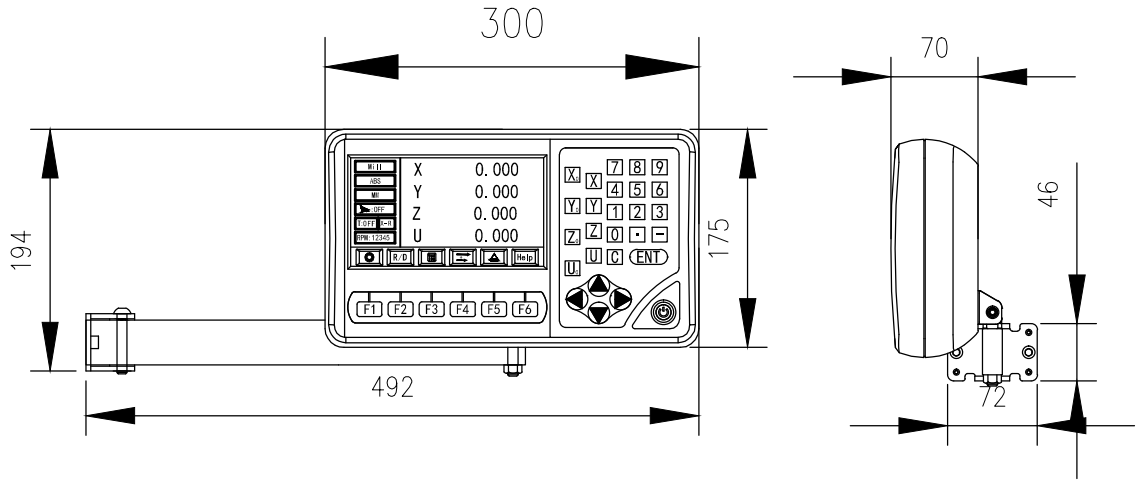
4.1 Notices for Usage:

1. Supply voltage: AC 80 V - - 260 V, 50 - - 60 Hz
2. Power: 15 W
3. Display mode:7 inches true color LCD screen.
4. Operating temperature: -10℃- - 60℃
5. Storage temperature: -30℃- - 70℃
6. Relative humidity (RH): <90% (25)>
7. Axis to be displayed : 1 axis, 2-axis , 3-axis , 4-axis , 5-axis
8. Input signal allowed by the DRO: TTL square wave / RS422
9. Allowable input signal frequency: < 2 MHz
10. Length resolution: 0.1 um, 0.2 um, 0.5 um, 1 um, 2 um, 2.5 um, 5 um and 10 um
11. Minimum resolution of angle display: 0.0001/ pulse
12. Weight: 2.2 KG
13. Volume size: 300 x175 x 70 (mm)
14. Interface definition of the grating ruler: (DB 9-pins socket)

Pin	1	2	3	4	5	6	7	8	9
signal	A-	0V	B-	Shield	R-	A	+5V	B	R



4.2 Installation Figure



4.3 Troubleshooting

The following troubleshootings are just the preliminary methods. If the problems still exist, please do not dismantle the DRO by yourself, but contact our company or the dealers for help in time.

Faults	Fault Causes	Solutions
The DRO doesn't display anything	1. The power is not on ? 2. The power switch is not closed ? 3. The supply voltage is not appropriate 4. The internal supply of the grating ruler is in short circuit.	1. Check whether the power line and power plug are plugged in. 2. Close the power switch. 3. Make sure the supply voltage between 85V-265V. 4. Pull out the connector of the grating ruler.
One axis of the DRO doesn't count	1. Operate the machine after swapping with the grating ruler of another axis. 2. Some special functions of the DRO are being used.	1. If counting, it's the fault of the grating ruler; if not, it's the fault of the DRO. 2. Exit the special function
The counting of DRO is not accurate (it can't zero)	1. The grating ruler isn't installed according to the requirements or the accuracy is not enough. 2. After being used for a long time, the vibration of the machine tool makes the fixed reading head or the screws loosen. 3. The accuracy of the machine tool is not good. 4. The DRO resolution isn't consistent with the grating ruler.	1. Reinstall the grating ruler and adjust the level. 2. Tighten all the fixed screws. 3. Overhaul the machine tool. 4. Reset the DRO resolution.
The counting of DRO is in error,The displayed operation distance isn't consistent with the actual distance	1. The machine tool and the DRO shell are not connected to earth. 2. The accuracy of the machine tool is not good. 3. The running speed of the machine tool is too fast. 4. The grating ruler isn't installed according to the requirements and the accuracy is not enough. 5. The DRO resolution isn't consistent with the grating ruler. 6. The operating size unit is not consistent with the displayed Metric/British units. 7. The linear error compensation setting of the DRO is not appropriate. 8. The grating ruler exceeds the operating range of length or the read head is broken.	1. Connect the machine tool and the DRO shell to earth. 2. Overhaul the machine tool. 3. Reduce the running speed of the machine tool. 4. Reinstall the grating ruler and adjust the level. 5. Reset the DRO resolution. 6. Switch the displayed Metric/British units. 7. Reset the linear error compensation of the DRO. 8. Repair the grating ruler.

Faults	Fault Causes	Solutions
The grating ruler doesn't count	1. The grating ruler exceeds the operating range of length or the read head is broken. 2. The read head of grating ruler rubs the ruler shell leading to the aluminum scraps accumulated. 3. The gap between the read head of grating ruler and the ruler body is too wide. 4. The metal tubes of the grating ruler are damaged, which causing the short circuit or disconnection in internal wiring.	1. Repair the grating ruler 2. Repair the grating ruler 3. Repair the grating ruler 4. Repair the grating ruler
The grating ruler doesn't count sometimes	1. The small box of the grating ruler is separated from the steel ball. 2. The grating glass in the read head of the grating ruler is abraded. 3. There is dirt on the grating glass in the shell of the grating ruler. 4. The elasticity of small box spring in the read head of the grating ruler is not enough.	1. Repair the grating ruler 2. Repair the grating ruler 3. Repair the grating ruler 4. Repair the grating ruler