



Home Page <http://www.autonc.jp>

Manual of Base Plane Machining



————— Product basis —————

In the base plane machining, by the Product basis, You enter a base size and software calculate the position of the tool automatically.

————— Tool basis —————

In the base plane machining, by the Tool basis, You enter the position of the first tool and the position of the last tool. This position is always accepted.

The middle becomes an equal pace by the software.

[2 Way] [2 Way+]

You machine a base plane by the going up and down with the face-mill cutter.

A depth of cuts and the number of times to the direction of Z can be optionally set.

A width of cuts is computed to become an equal pace from the condition.

NC data for roughing cut and finishing cut is created.

[Spiral] [Spiral +]

You machine a base plane to inside in the swirl from outside with the milling-cutter. Without changing height, all can be cut with the down cutting. A depth of cuts and the number of times to the direction of Z can be optionally set. A width of cuts is computed to become an equal pace from the condition. NC data for roughing cut and finishing cut is created.

[Keyway]

You create NC data for the roughing cut and finishing cut, which machines a keyway. For roughing cut, it cuts deeply at the equal pace in Z. For finishing cut, it turns around the same height several times.

[U Slot]

You create NC data for the roughing cut and finishing cut, which machines a U Slot. For roughing cut, it cuts deeply at the equal pace in Z. For finishing cut, it turns around the same height several times.

[U Slot (Chamfer)]

You create NC data for the roughing cut and finishing cut, which machines a U Slot with chamfer. For roughing cut, it cuts deeply at the equal pace in Z. For finishing cut, it turns around the same height several times.

[Side Cutter]

You machine a surface of the U Slot by Side-cutter from the direction of the base.

[Japanese]

When pushing this button, it becomes Japanese display.

[Parameter]

You specify an editor for the editing. You start up the editor at the "EDIT" button of the program. At present, you aren't using the rapid move speed of the machine tool.

Parameter Set

Editor for Use:

C:\WINDOWS\system32\notepad.exe

Rapid Move Speed of NC Machine

X axis	<input type="text" value="5000"/>	mm/min
Y axis	<input type="text" value="5000"/>	mm/min
Z axis	<input type="text" value="3000"/>	mm/min

[2 Way - Product basis -]

When pushing this button, a light-blue entry area is cleared.

You put a machining plane maximum and the minimum value coordinates.
If not needing below the decimal point, omit a decimal point.

Expand-Length is the opening of the machining plane and the tool.
The Y-axis moves in G00.

You put rapid move height.
You start from the position of X0Y0Z (rapid move Z height).

You put a depth of cut.

You choose Yes or No at the <G92>.

When you choose Yes, G90G92X0Y0Z (the rapid move z height) is stored.

When it is No, G90X0Y0; G01Z (the rapid move z height) is stored.

(For the details, refer to the NC data).

The screenshot shows a CNC control interface with two main sections: 'TOOL' and 'CUTTING HEIGHT LIST'.
In the 'TOOL' section, there is a 'G92' label followed by two radio buttons: 'Yes' (unselected) and 'No' (selected). Below this are several input fields:

- Diameter: 200 mm
- Std. Y Interval: 180 mm (highlighted with a red box)
- Limet Y Interval: 190 mm
- Feed Rate: 200 mm/min
- Spindle Speed: 130 rev/min

The 'CUTTING HEIGHT LIST' section features a vertical scrollable list of numbers: 15, 13, 11, 9, 7, 5, 3, 1, 0.2. To the right of the list is a button labeled 'LIST READ TEST'. Below the list, the text 'For Roughing' is written in red.

<The thinking way of cutting width>

It cuts off a quotient with $((Y_{max} - Y_{min}) / \text{Std Y Interval})$ and it decides the cut number of times.

If it doesn't exceed Limit Y Interval, it moves a tool at the pace.

When exceeding a Limit Y Interval, it divides by the cut number of times +1 once more and it decides a pace.

You check whether or not it is possible to read a "cutting height list" and you prevent from an entry mistake.

Put a cutting height pattern in turn.
(Not to enter the space line. big → small)

For this example, if depth is over 15mm, first it cuts 15 mm, and cuts at the 2 mm pace until it becomes a 3 mm leaving, next, it cuts 1mm leaving and last cuts the 0.2 mm leaving.
(When putting 0 last, you finish the plane with same tool.)

You cut to the bottom from the list value that is smaller than depth.
When depth is 9 mm, you cut with 7, 5, 3, 1, 0.2 leaving.

"Select" At the button,
it selects a write folder.

It is possible to
display the NC data.

Write Folder:
C:\

SELECT

Roughing File: TeimenAra EDIT Check File: TeimenCheck EDIT

Finish File: TeimenStage EDIT 1 Time cut. Height 0

Graphic Display

CHECK

ROUGHING

FINISH

START

CANCEL

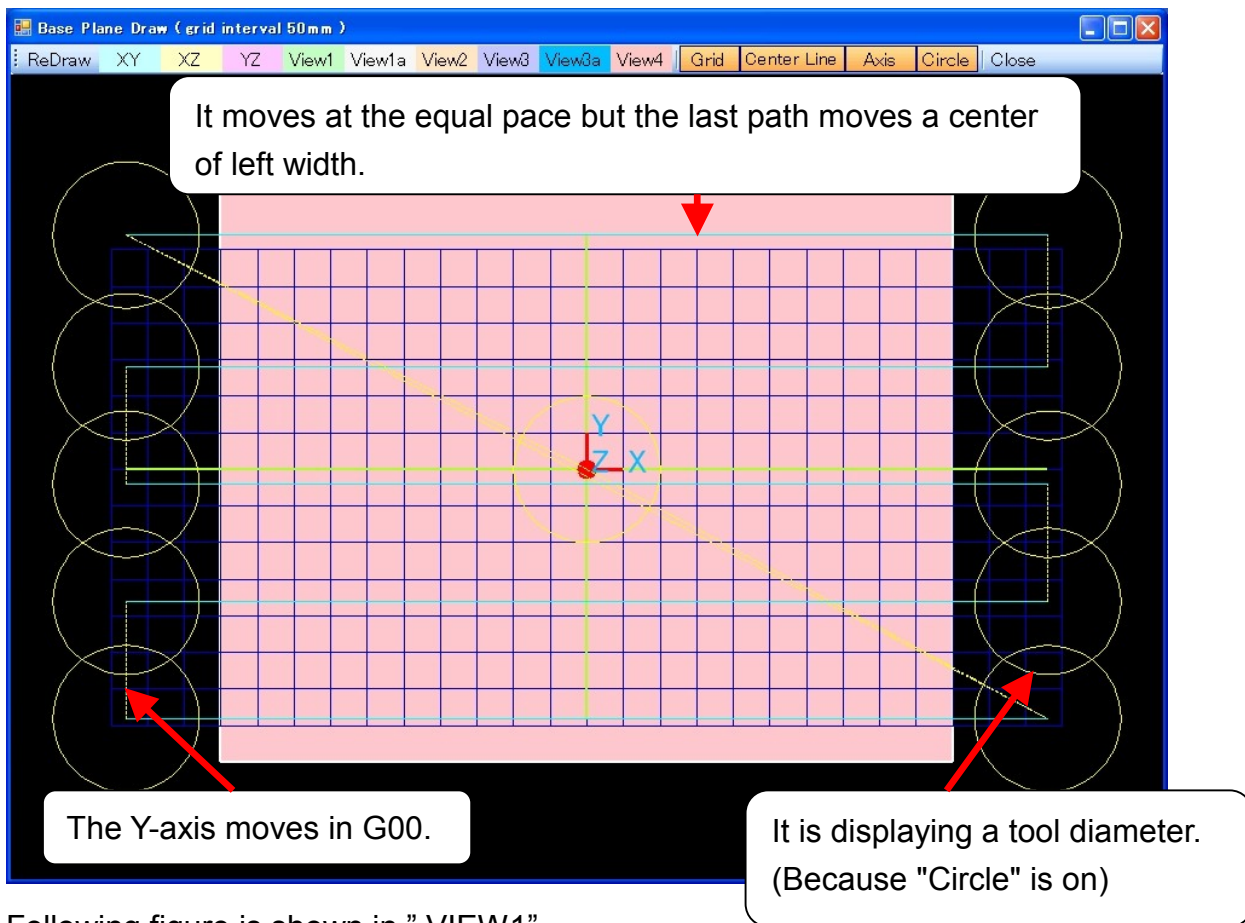
You enter a Roughing file name, a Finish file name, a Check file name.
The Finish file creates the data to cut once at height Z0.
The Check file moves to confirm a cut range and height.
" EDIT " When pushing the button, the editor starts up and opens a file.

When pushing the "Start" button after fill in all data, the NC data of the file name is created in folder.

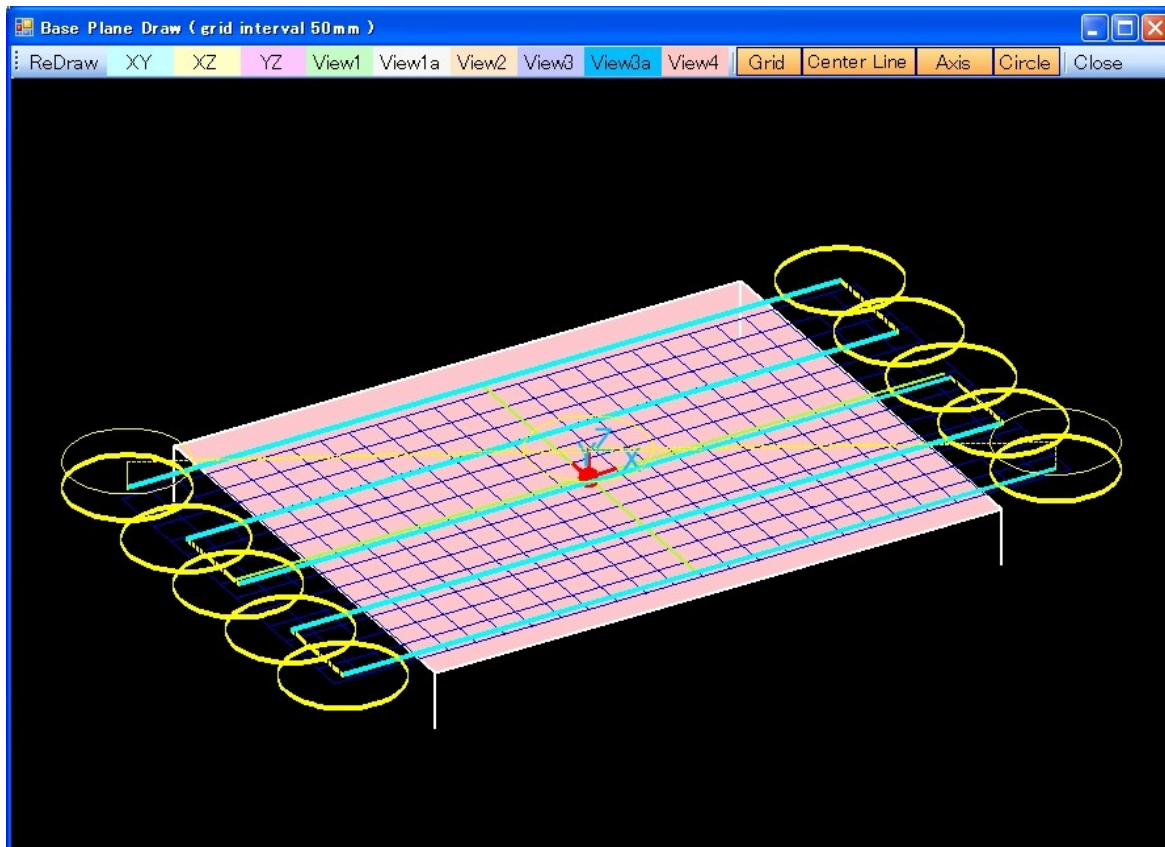
When pushing the "Cancel" button, it ends.
Input data is saved and is restored in the next time.

The Display of the NC data

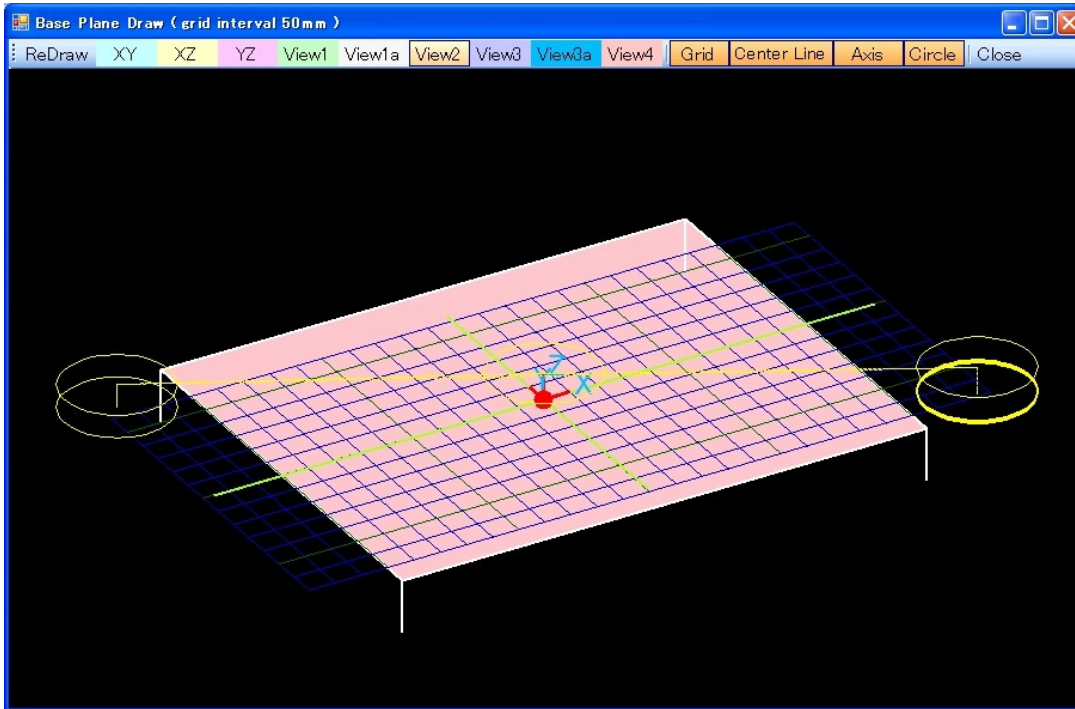
The following shows by "ROUGHING" button. This figure is shown in "the XY Plane".



Following figure is shown in "VIEW1".



The following figure is displaying a confirmation file. You can check machining range and height.



The Display menu



You can select the display direction.

XY plane , XZ plane ,YZ plane, View1, View1a, View2, View3, View3a, View4.

View is the show that was seen from the diagonal top.

View1, View2, View3, View4 are the show which was seen from 30 degrees above from just beside.

View1a, View3a are the show, which was seen from 60 degrees above from just beside.

The show button of Grid, the centerline, the Axis, Circle becomes on.

When making "Grid" off, the grid of blue 50 mm disappears.

When making "Centerline" off, the olive-green X-axis, the Y-axis, the Z-axis disappear.

When making "Axis" off, the coordinate system of the X, the Y, Z disappears.

When making "Circle" off, the tool diameter display disappears.

The part can be displayed in the expansion when clicking with the mouse and dragging.

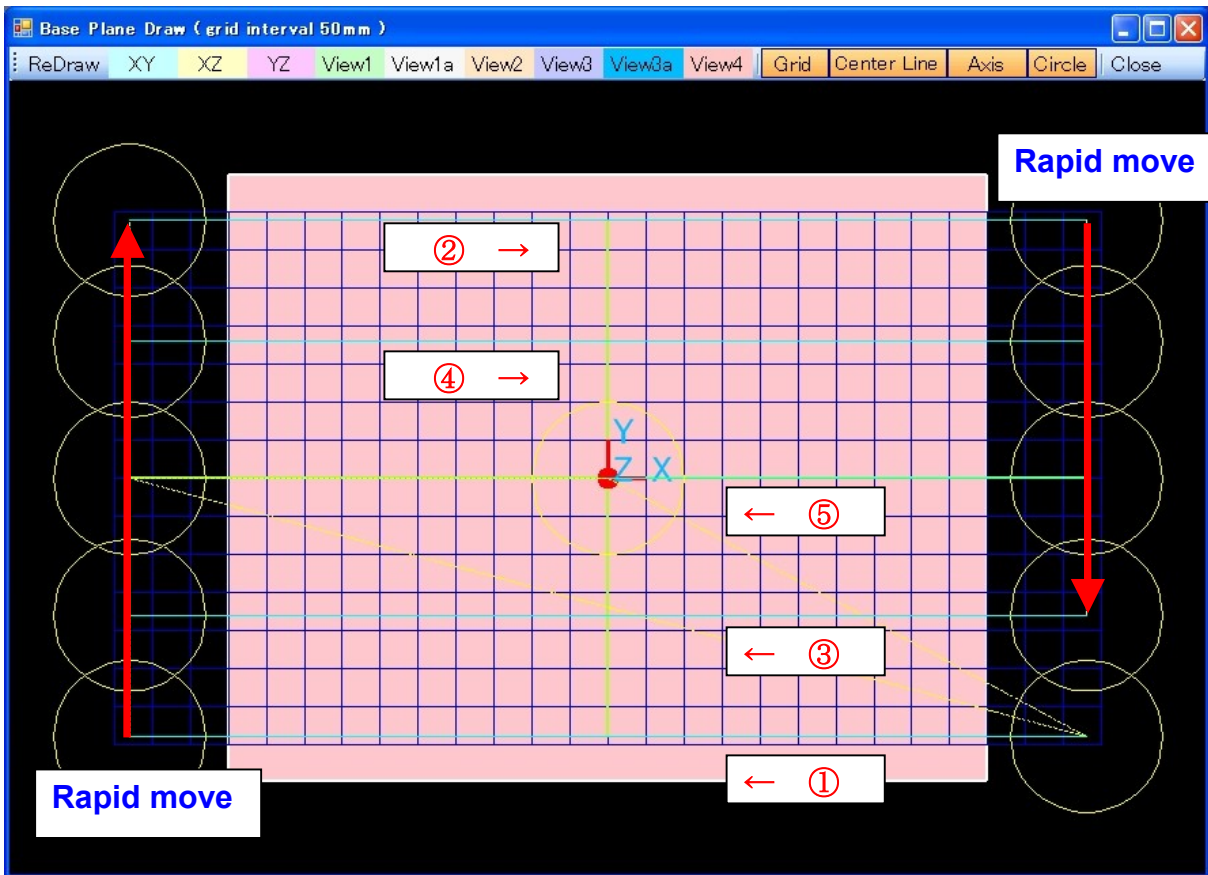
It returns to the ex-screen by "Close".

[Spiral - Product basis -]

Only a difference from 2Way is described. "Revision" is always "No" in the first. When "Revision" is "Yes", three pieces of data are added in the end of the finishing data. It prevents from line's being left in Central when making a "Std. Y Interval" small extremely.

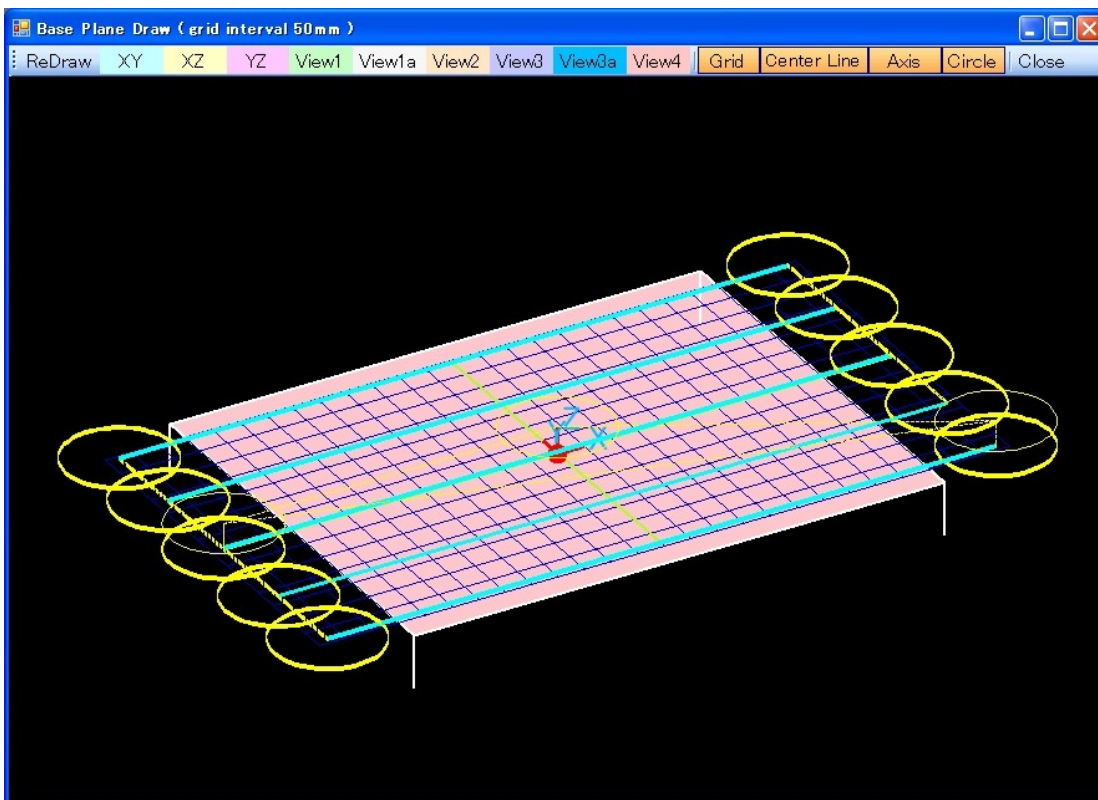
The Display of the NC data

The following shows by "ROUGHING" button. This figure is shown in "the XY Plane".

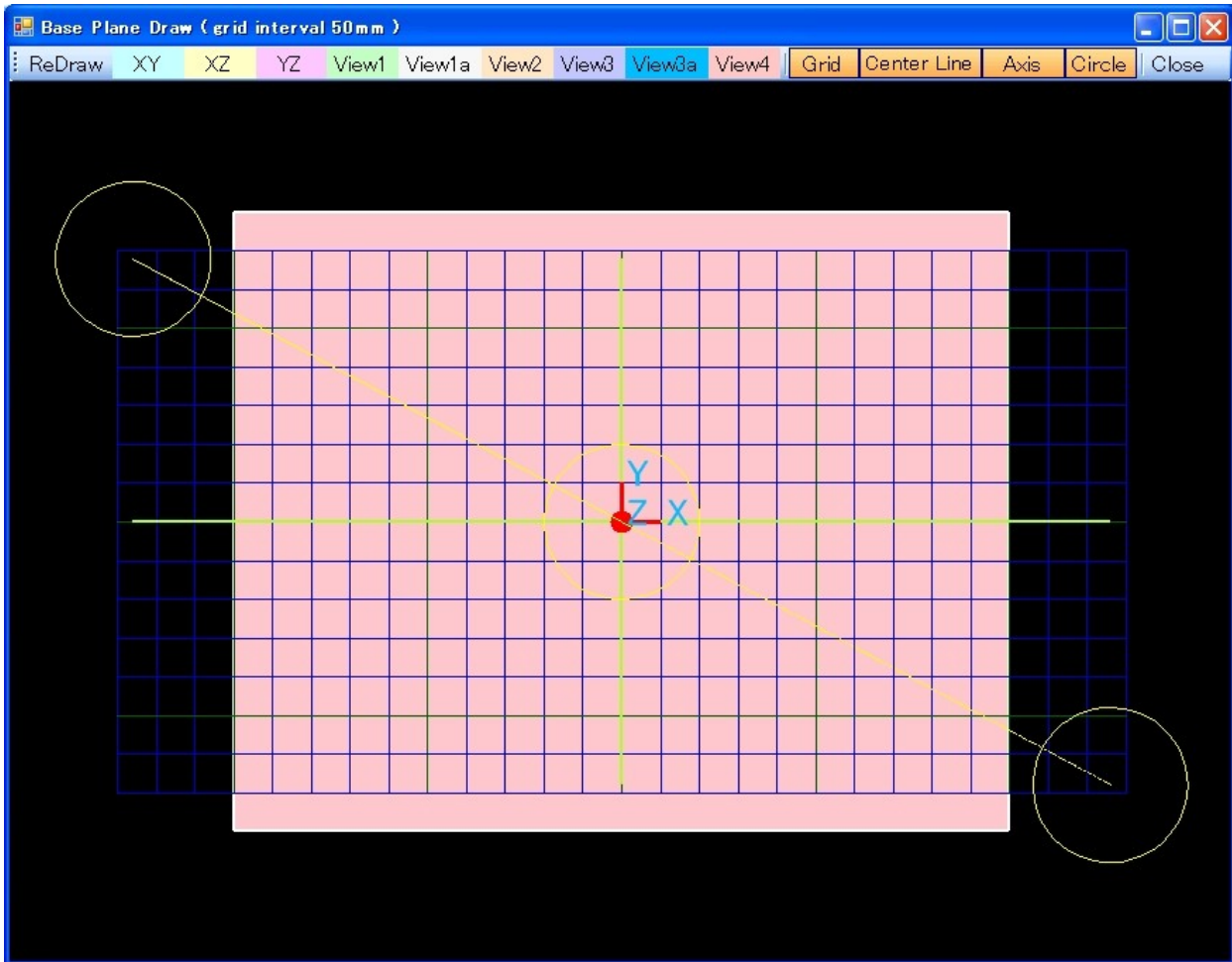


Following figure is shown in "VIEW1".

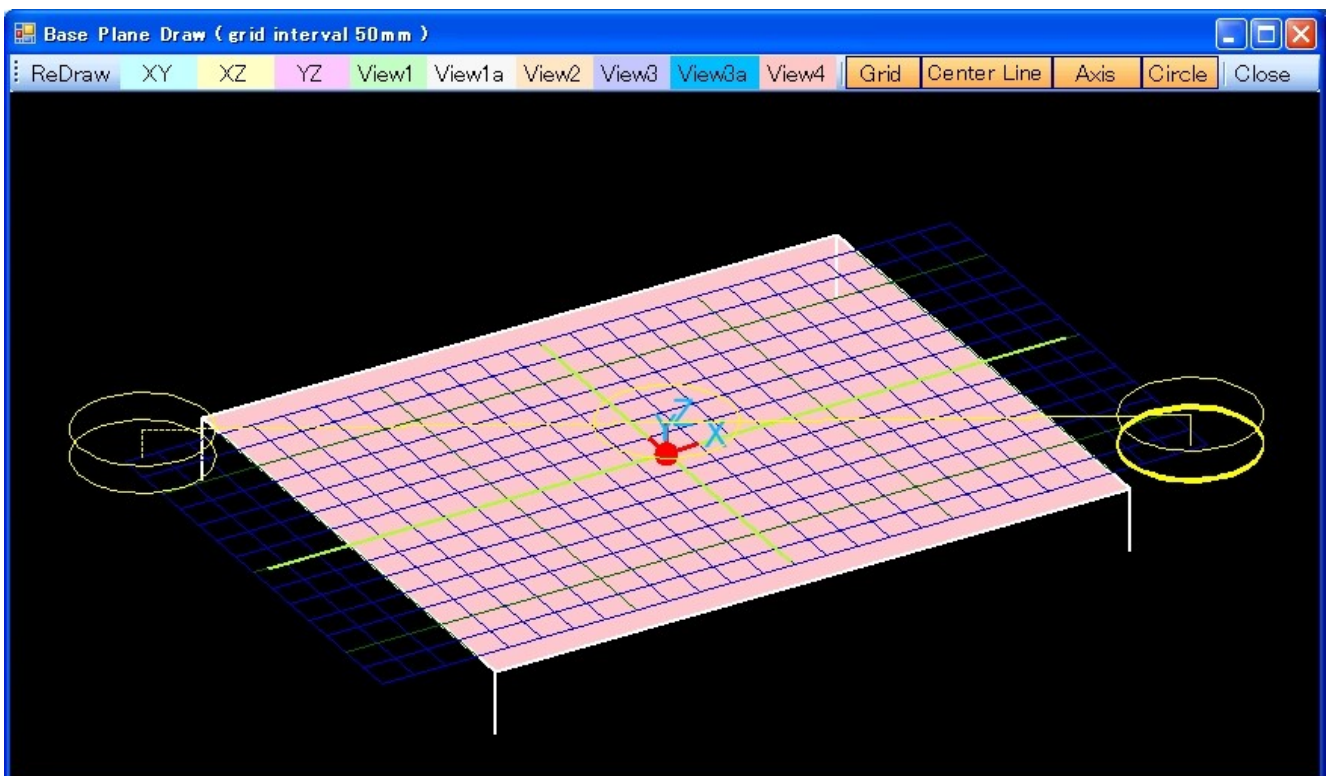
As for the "FINISH" button too, it is shown in the same way.



The following figure is displaying a confirmation file. You can check machining range and height. It is displaying in the XY plane.



Following figure is shown in "VIEW1".



[2 Way - Tool basis -]

BASE PLANE 2 WAY CUTTING ----- Tool basis -----

BASE PLANE 2 WAY CUTTING CLEAR COORDINATE

X Minim: -500
Y Maxim: 400

TOOL: G92 Yes No

Diameter: 200 mm
Std. Y Interval: 180 mm
Limet Y Interval: 190 mm
Feed Rate: 200 mm/min
Spindle Speed: 130 rev/min

CUTTING HEIGHT LIST

- 15
- 13
- 11
- 9
- 7
- 5
- 3
- 1
- 0.2

LIST READ TEST

-- Graphic Display --

CHECK

ROUGHING

FINISH

Write Folder: C:\Documents and Settings\takayuki kamasuka\デスクトップ

SELECT

Roughing File: TeimenAra EDIT Check File: TeimenCheck EDIT

Finish File: TeimenSiage EDIT 1 Time cut. Height 0

START

CANCEL

BASE PLANE 2 WAY CUTTING CLEAR COORDINATE

X Minim: -500
Y Maxim: 400

Tool Pitch 0

Rapid Move Height 50

Cutting Volume 9

X Maxim: 500
Y Minim: -400

The machining starting-point of the tool becomes the lower right. The upper left becomes a machining breadth-maximum. This position is always accepted.

If not needing below the decimal point, omit a decimal point.

The others are the same as the 2 Way in product basis.

[Spiral - Tool basis -]

The machining starting-point of the tool becomes the lower right.

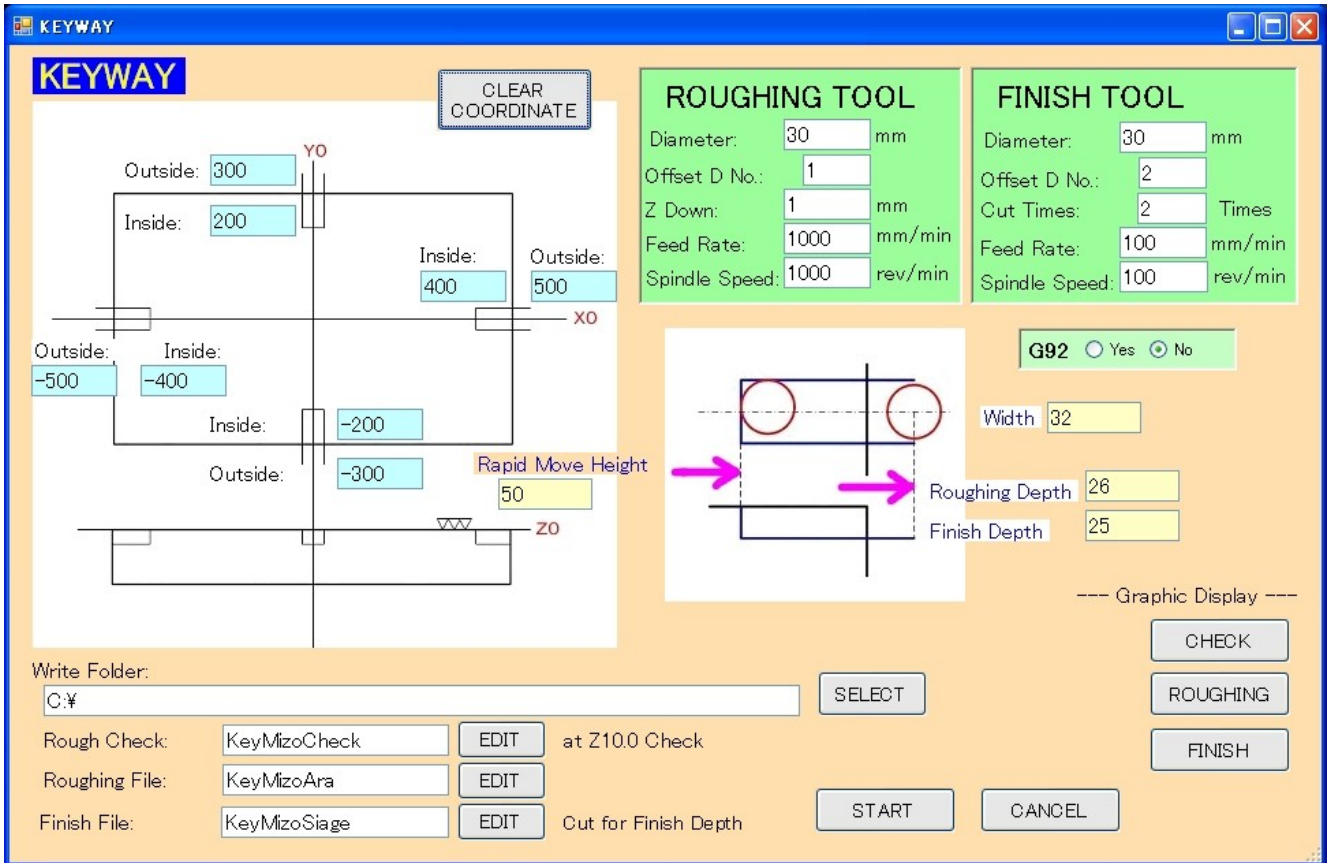
The upper left becomes the 2nd machining starting-point.

If not needing below the decimal point, omit a decimal point.

The others are the same as the spiral in product basis.

When "Revision" is "Yes", three pieces of data are added in the end of the finishing data. It prevents from line's being left in Central when making a "Std. Y Interval" small extremely.

[Keyway]



It digs a keyway on the centerline. It begins from the left side, and you can put a key in the finishing process and adjust the tool radius offset easily. (2009.06.02)



For roughing cut, it cuts deeply at the equal Z pace.

For finishing cut, it turns around the same height several times.

You choose Yes or No at the <G92>.
 When you choose Yes, G90G92X0Y0Z (the rapid move z height) is stored.
 When it is No, G90X0Y0; G01Z (the rapid move z height) is stored.
 (For the details, refer to the NC data).

When pushing this button, a light-blue entry area is cleared.

KEYWAY

CLEAR COORDINATE

Outside: 300
Inside: 200
Inside: 400
Outside: 500
Outside: -500
Inside: -400
Inside: -200
Outside: -300
Rapid Move Height: 50

Y0
X0
Z0

The outside is the position that the tool goes down to.
The inside is the wall that the tool is tangent to.

Inside Outside

Width: 32
Roughing Depth: 26
Finish Depth: 25

In the light-blue entry area, the blank place is skipped.
Maximum four keyways can be entered.
If erasing vertical data, it machines the two horizontal keyways.

You define the surface Z0 that digs a keyway, and enter a Roughing-Depth and a Finish-Depth.
Enter the positive depth.

Write Folder: C:\

Rough Check: KeyMizoCheck EDIT at Z10.0 Check

Roughing File: KeyMizoAra EDIT

Finish File: KeyMizoSiage EDIT Cut for Finish Depth

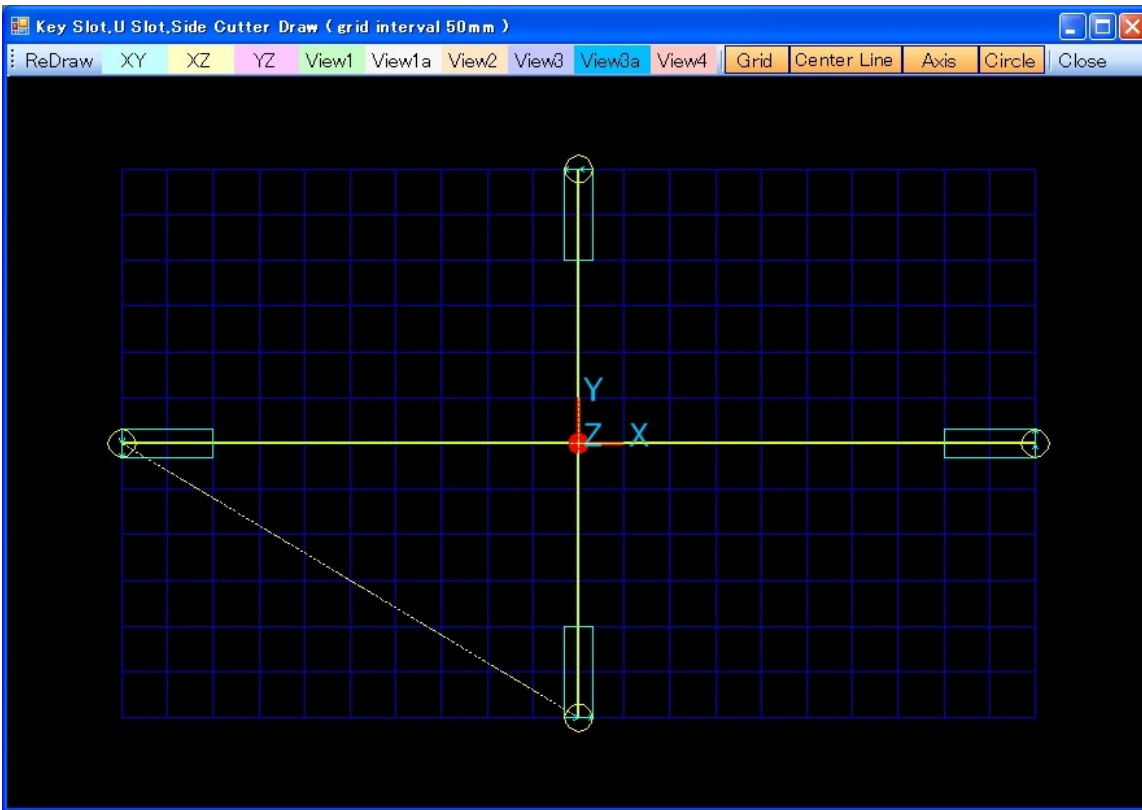
CHECK
ROUGHING
FINISH
START
CANCEL

You enter a Roughing file name, a Finish file name, and a Rough Check file name.
The Check file first moves to lowest depth and moves to confirm a path at Z10.0mm.
" EDIT " When pushing the button, the editor starts up and opens a file.

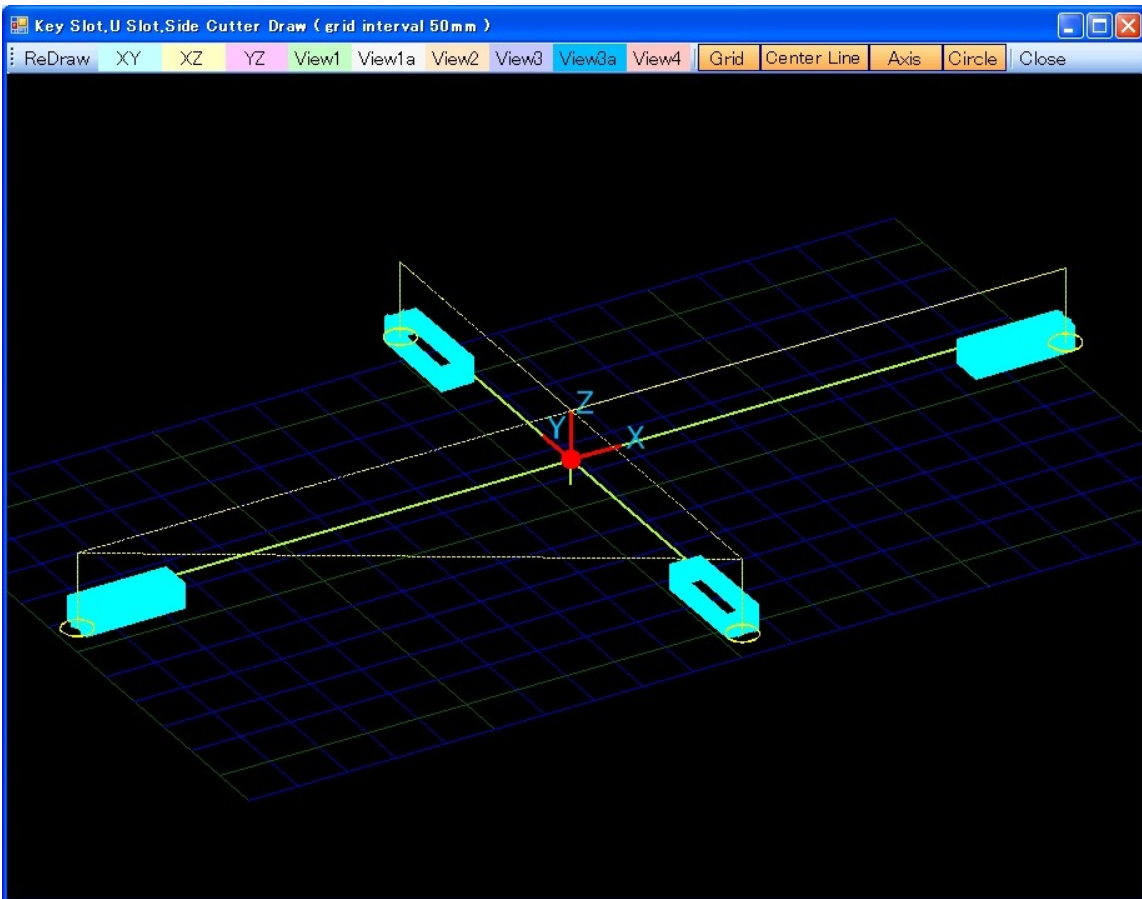
It is possible to display the NC data.

The Display of the NC data

The following shows by "ROUGHING" button. This figure is shown in "the XY Plane".

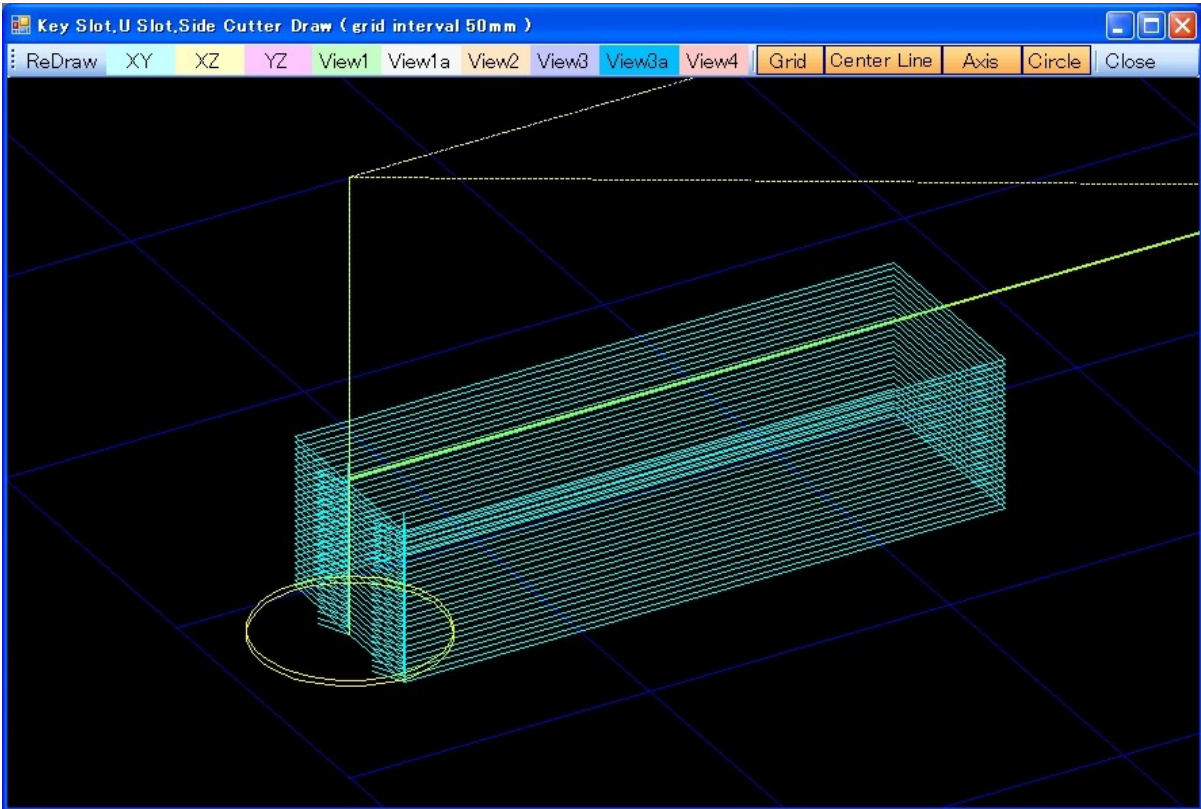


Following figure is shown in "VIEW1".

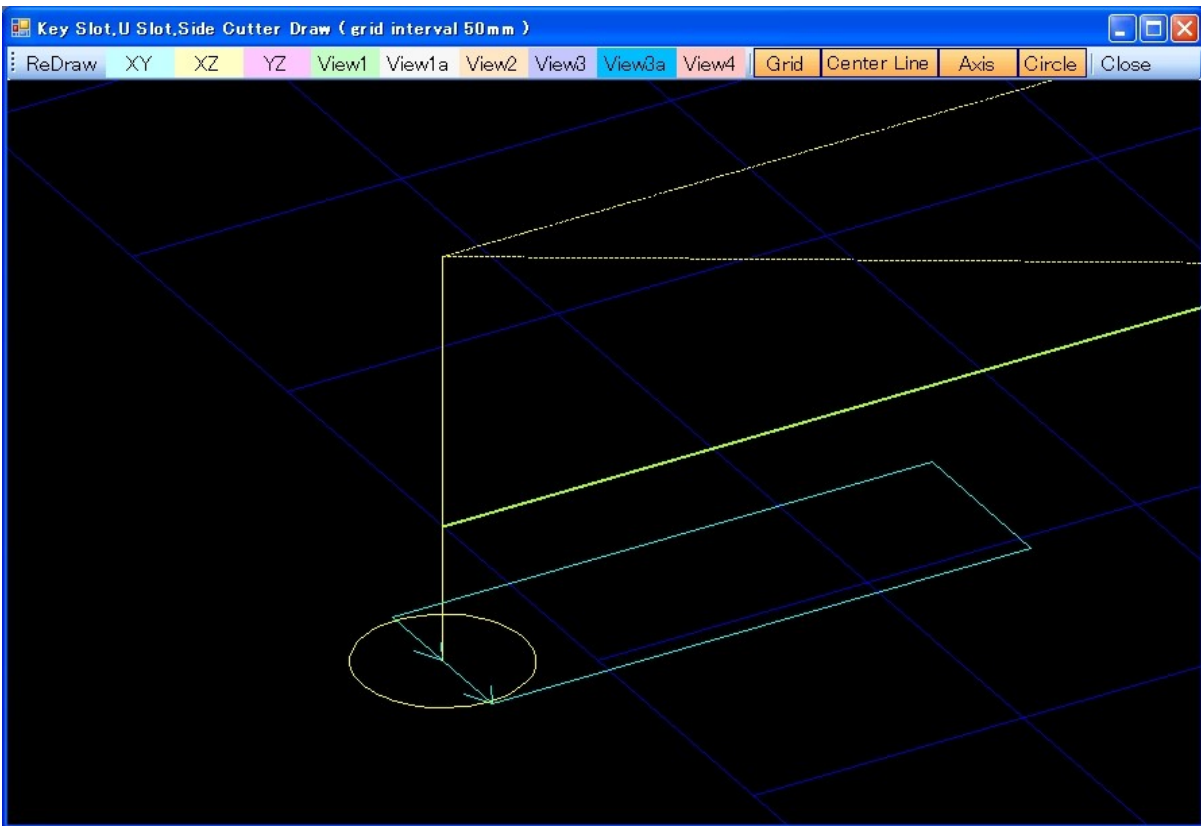


Following figure is shown in "VIEW1".

It is partially expanded with clicking and dragging by the mouse.

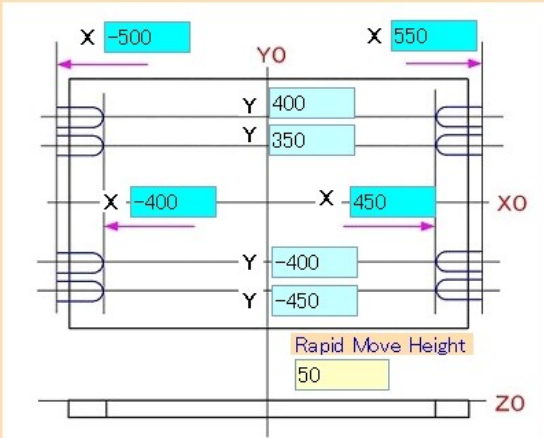


The following shows by "FINISH" button. This figure is shown in "VIEW1".



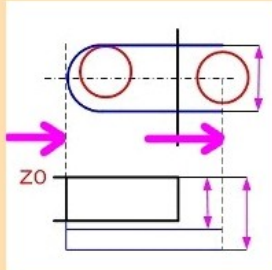
[U Slot]

U SLOT if Data in light blue Area is blank,it's Skipped.



ROUGHING TOOL
Diameter: 25 mm
Offset D No.: 1
Z Down: 1 mm
Feed Rate: 1000 mm/min
Spindle Speed: 1200 rev/min

FINISH TOOL
Diameter: 25 mm
Offset D No.: 2
Cut Times: 2 Times
Feed Rate: 200 mm/min
Spindle Speed: 300 rev/min



Width 30
Roughing Depth 40
Finish Depth 60

G92 Yes No

CLEAR COORDINATE

Write Folder: C:\¥

Rough Check: UmizoCheck EDIT at Z10.0 Check

Roughing File: UmizoAra EDIT

Finish File: UmizoStage EDIT

SELECT START CANCEL CHECK ROUGHING FINISH

--- Graphic Display ---

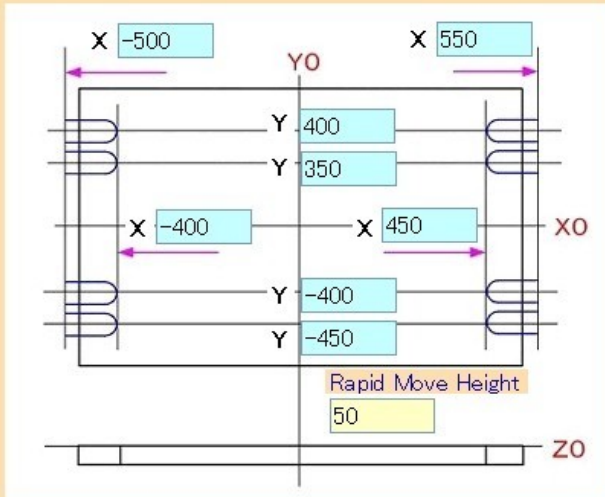
The setting of the tool and the file output are the same as the keyway.

If you use a Roughing-Check file, you can find mistakes of input.

In the light-blue entry area, the blank place is skipped.

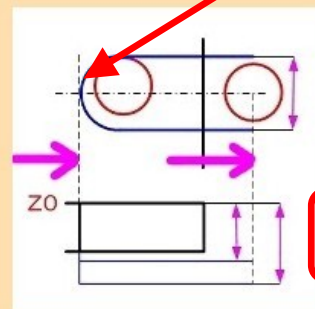
U SLOT

if Data in light blue Area is blank, it's Skipped.



CLEAR
COORDINATE

When putting width, it makes an arc with the diameter that is the same as the width.



Width 30

Roughing Depth 40
Finish Depth 60

When pushing this button, a light-blue entry area is cleared.

You define the surface Z0 that digs a U slot, and enter a Roughing-Depth and a Finish-Depth. Enter the positive depth.

The outside of X-axis is the position, which the tool goes down to.
The inside is the wall, which the arc is tangent to.
The Y value is a center position at the U slot.

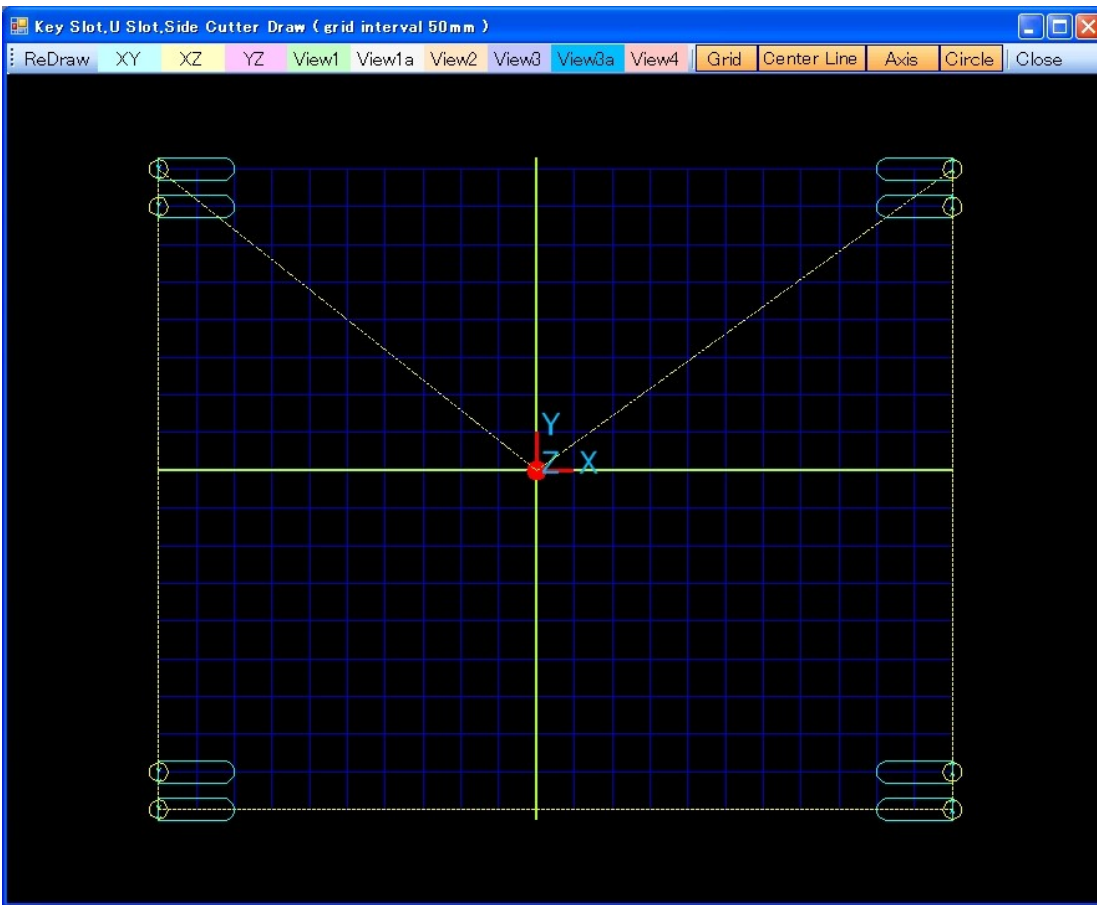
Only the place where a position is decided in the combination of the X and the Y machines a U slot.

You use the 90-degree rotation, when The U slot is at the vertical-direction.

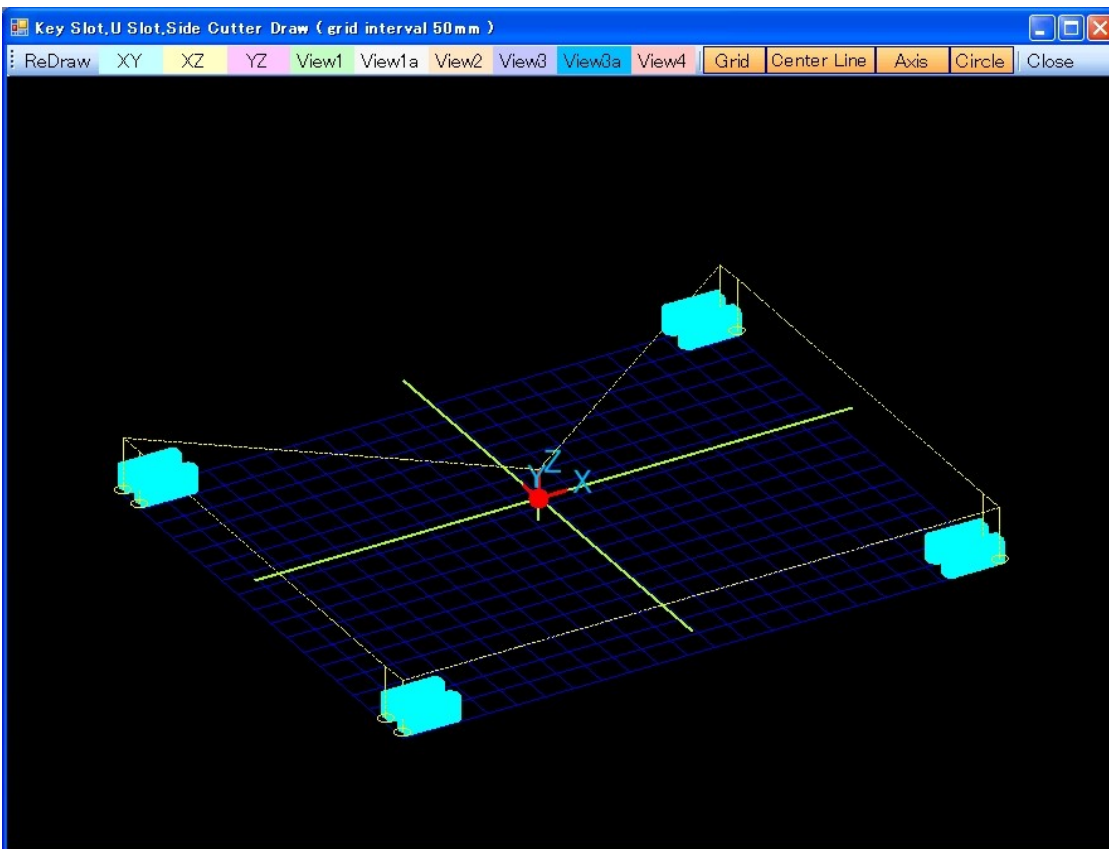
In the homepage of AutoNC,
There is free software of 90-degree rotation.

The Display of the NC data

The following shows by "ROUGHING" button. This figure is shown in "the XY Plane".

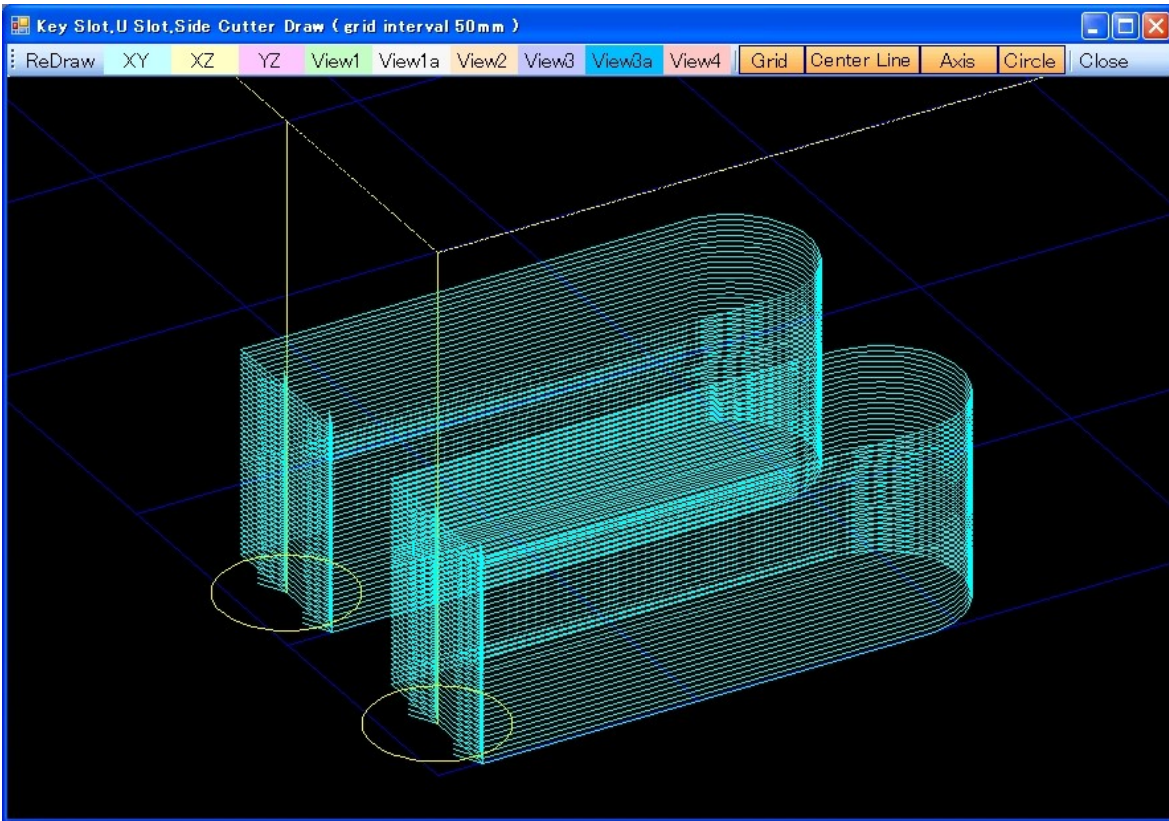


Following figure is shown in "VIEW1".

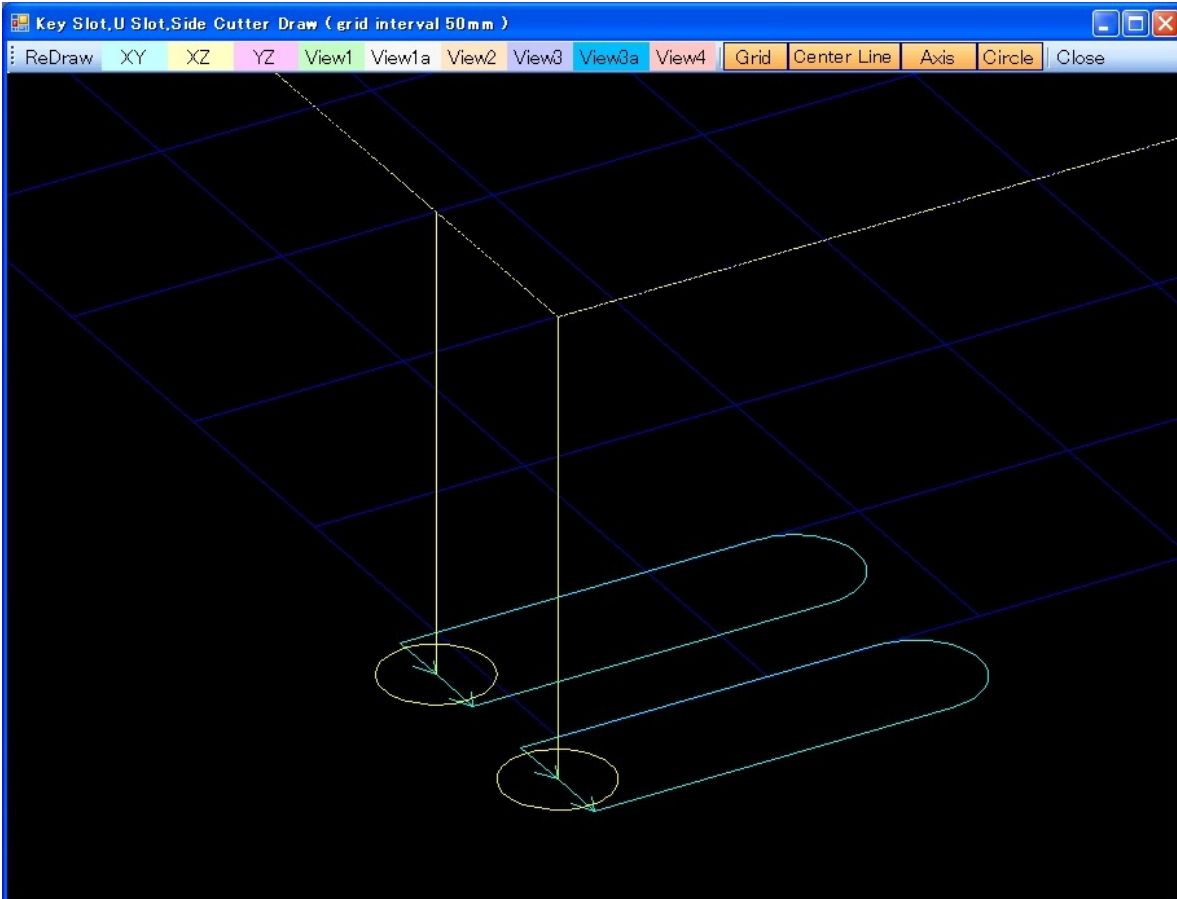


Following figure is shown in "VIEW1".

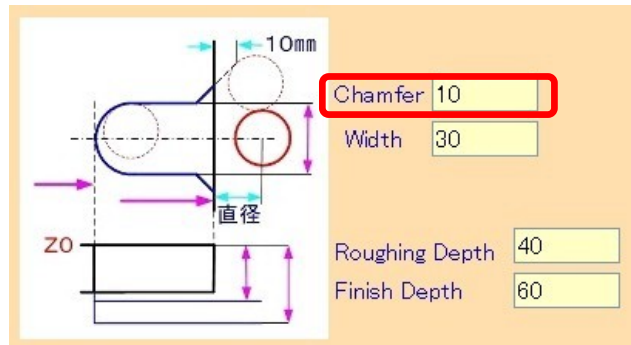
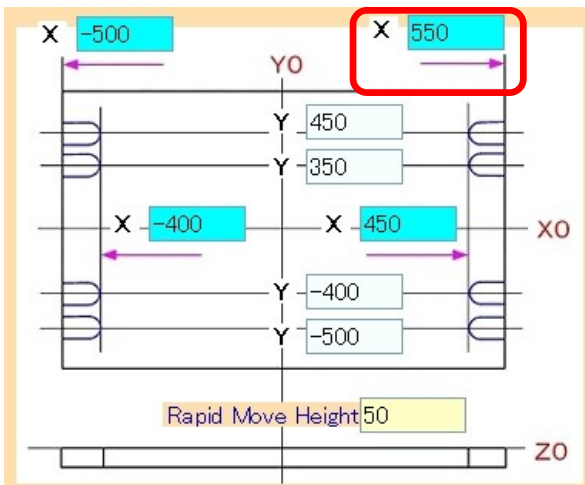
It is partially expanded with clicking and dragging by the mouse.



The following shows by "FINISH" button. This figure is shown in "VIEW1".



[U Slot (Chamfer)]



The outside of X-axis is the position of the end face.

The inside is the wall that the arc is tangent to.

The Y value is a center position at the U slot.

The difference of the previous U slot is to indicate chamfer.

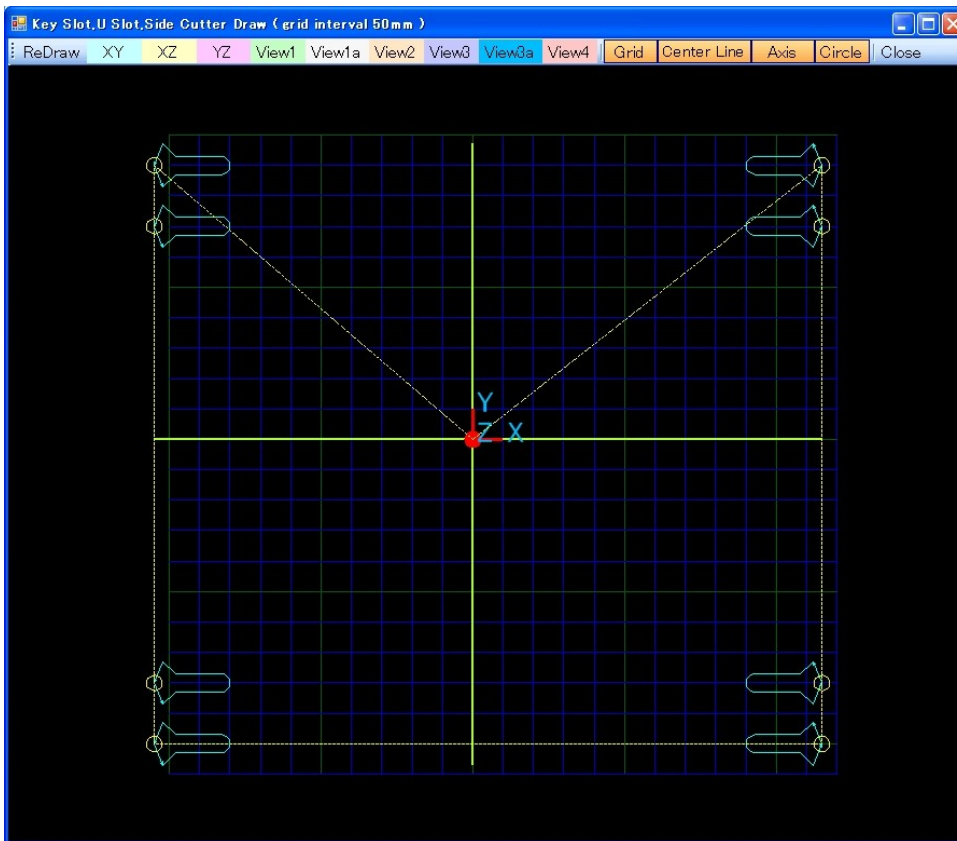
The position that the tool goes down is the position that is the tool diameter distance from an end face.

The approach position is the point of intersection of the line that extended an end face by 10 mm and the line which extended chamfer.

You use the 90-degree rotation, when The U slot is at the vertical-direction.

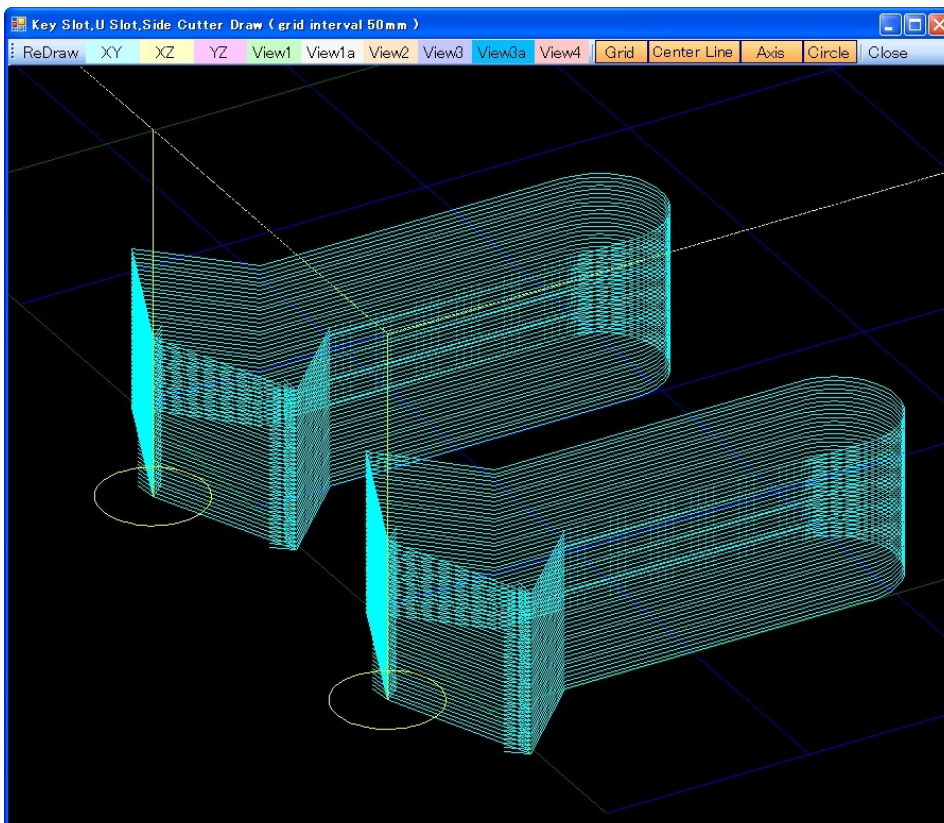
The Display of the NC data

The following shows by "ROUGHING" button. This figure is shown in "the XY Plane".



Following figure is shown in "VIEW1".

It is partially expanded with clicking and dragging by the mouse.



[Side Cutter]

The input method is the same as the U slot, but the side-cutter machining adds a data entry at the vertical-direction, too.

It is possible to do the machining of eight places in the maximum.

All coordinate-values are the position of the U slot center.

In the necessity, it is the position that the tool goes down and the ending position of the machining.

The tool goes and returns to the position which tool goes down, to the ending position. (It made the twice return-speed of the cut-speed. 2009.06.02)

In the figure above, it is 2 mm at Z down, 50 mm of the depth, 10 mm of cutting volume. In this case, it goes first with -58 mm of Z, -56, -54 and to -50mm.

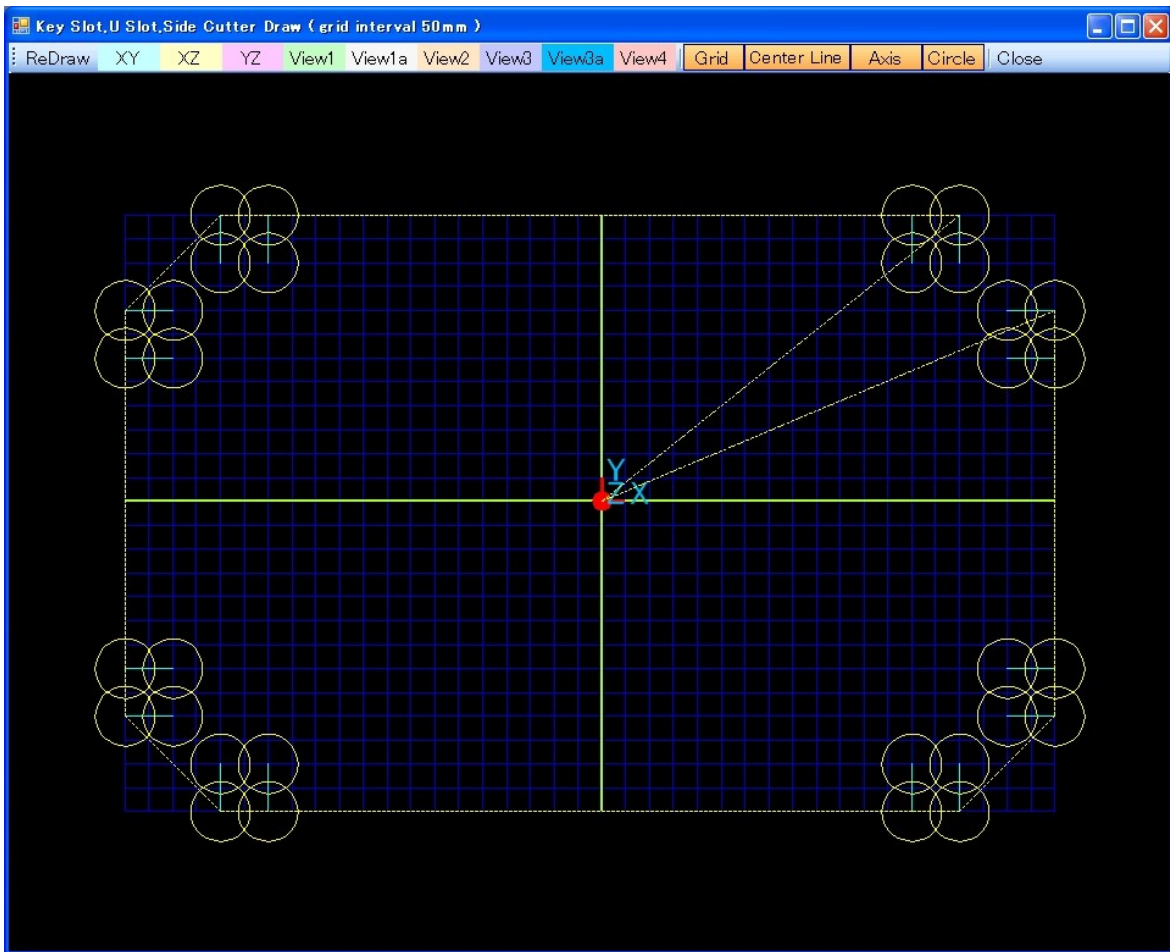
The G01 mode height is the change height of G00 to G01.

Because the Confirmation file is the 50mm of depth, 10 mm of cutting volume, it moves -60 mm of Z only once.

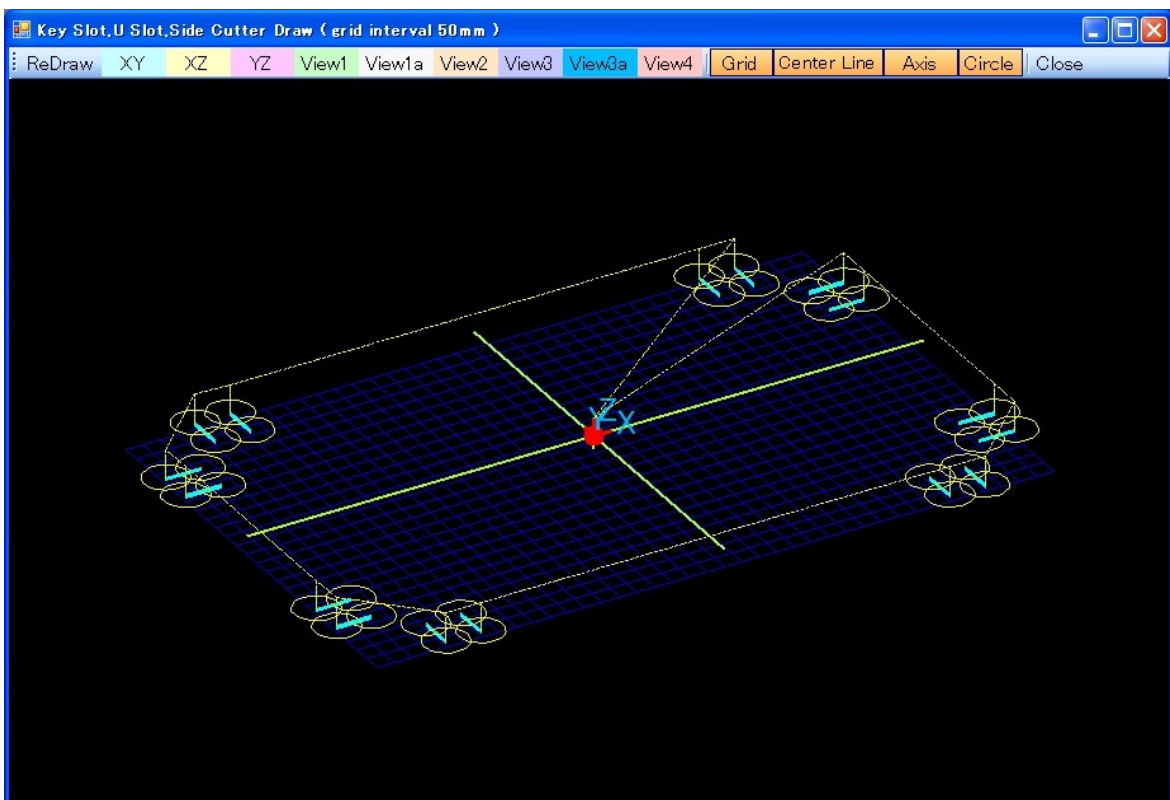
When checking interference by this, actual machining can be automated.

The Display of the NC data

The following shows by "DISPLAY" button. This figure is shown in "the XY Plane".

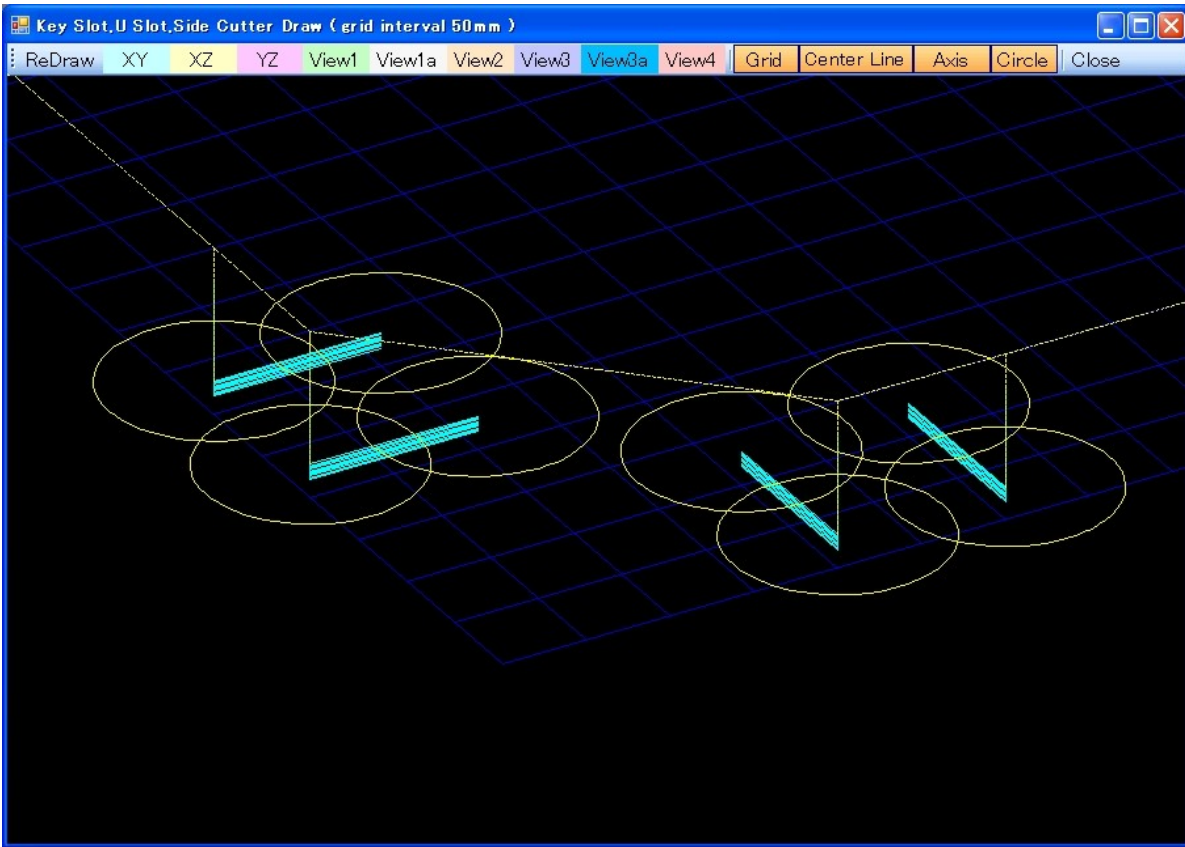


Following figure is shown in "VIEW1".

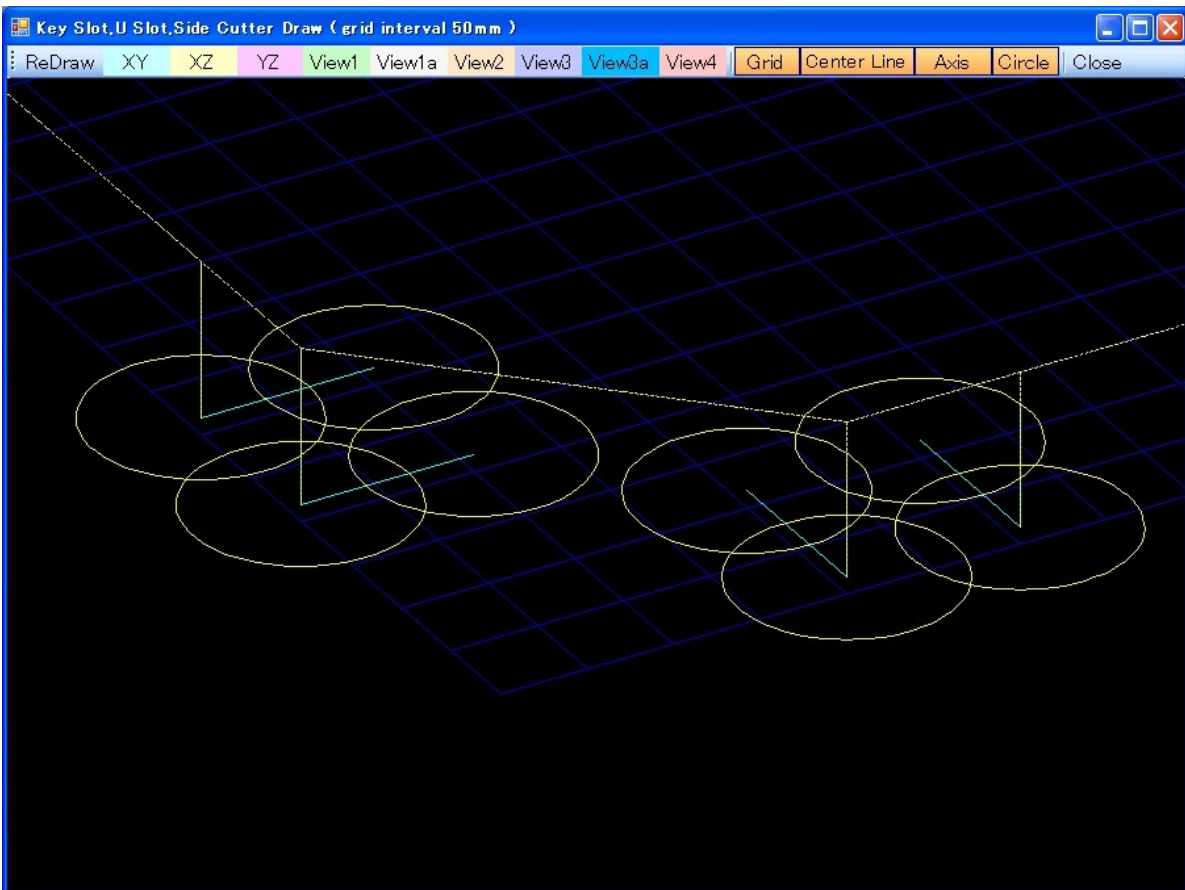


Following figure is shown in "VIEW1".

It is partially expanded with clicking and dragging by the mouse.



The following shows by "CONFIRM" button. This figure is shown in "VIEW1".



The above