



**Doosan Infracore**  
Machine Tools

# Operation Manual

EZ Guide-I

Machining Center

MCTOPEEZG41

MCTOPEEZG41

# **OPERATION MANUAL**

---

EZ GUIDE-i

**MACHINING CENTER**



**Doosan Infracore**

# TABLE OF CONTENTS

## **I.General**

<b>1 Overview .....</b>	<b>2</b>
<b>2 How to Start Making New Program?.....</b>	<b>3</b>
<b>3 Outline of Each Step.....</b>	<b>4</b>
3.1 Preparation for Machining - Fixed Start.....	4
3.2 Preparation for Machining - Process Start.....	8
3.3 Preparation for Graphic Display - Blank.....	9
3.4 Preparation for Graphic Display - Tool Specification.....	10
3.5 Machining Cycle Selection.....	14
3.6 Selecting Figures.....	15
3.7 Pre-check for Mistakes in Program.....	16

## **II.Actual Programming**

<b>1 Surface Work by Facing Cycle.....</b>	<b>20</b>
1.1 Main Feature of Facing Cycle.....	20
1.2 Outline of Work Figure Definition for Facing Cycle.....	24
1.3 XY Square Figure for Facing.....	26
1.4 XY Circle Figure for Facing.....	30
1.5 XY Track Figure for Facing.....	32
1.6 XY Free Figure for Facing.....	33
<b>2 Hole Machining.....</b>	<b>44</b>
<b>3 Grooving.....</b>	<b>46</b>
3.1 Main Feature of Grooving Cycle.....	46
3.2 Cutting Condition Set-up in Grooving Cycle.....	47
3.3 Setting Parameters for Grooving Cycle.....	48
<b>4 Pocketing.....</b>	<b>54</b>
4.1 Main Feature of Pocketing Cycle.....	54
4.2 Setting Parameters for Pocketing Cycle.....	56
4.3 Free Figure Pocketing.....	62
<b>5 Contouring.....</b>	<b>73</b>
5.1 Main Feature of Contouring Cycle.....	73
5.2 Setting Parameters of Contouring Cycle.....	74
5.3 Free Figure Outer Wall Contouring.....	79
5.4 How to Set-up Partial Contouring.....	83

# I . GENERAL

# 1.Overview

Please read the manual published by FANUC carefully first, especially from p.3 to p.237 and from p.247 to p.276(ver.B-63874EN/04).

It is recommended for you to be accustomed to the basic operation regarding editing and checking actual machining program first.

Those parts describe general feature of this software and detailed guidance for making new program, selecting tool & workpiece shape for simulation, setting offset data, editing cycles & figures for each menu, etc. And never forget to read & test from p.652 to p.680, for it is one of the most important section which shows good example to make real program.

Then this small guidebook will be of little help to you for catching up how to make, modify, and check programs in actual use, what kind of data to be set or input, and so forth.

It is obviously true ***EZ Guide i*** cannot cover all machining items at the customer's side, and has some limitation for complicated programming(i.e. Mold and Die program).

Please keep in mind that the main purpose of this software is just to support your operator or programmer when his making relatively short process for simple parts only.

And please do not apply tool diameter compensation(G41/G42), for this software is able to calculate and command optimized tool path even without that kind of troublesome programming factor which sometimes causes terrible damage to your workpiece.



### 3.Outline of each step

When a machining program is being edited for operation, the above mentioned procedure is normally expected to be set-up by an operator.

Main feature of each step in **EZ Guide i** (hereinafter EZ-i) is to be explained.

#### **3.1 Preparation for machining - *Fixed Start(Fixed form1)* (START menu set-up)**

A programmer is supposed to establish tool condition, work offset, and spindle rpm before any real-cutting movement blocks.

To maximize efficiency of **EZ-i** by eliminating any trouble in doing so, some basic items need to be set and selected properly in the assigned area.

You can register a series of frequently used sentence to start machining just like the following blocks in a kind of program bank, **Fixed Start**, so it is possible to select suitable repeated data at anytime you create new program.

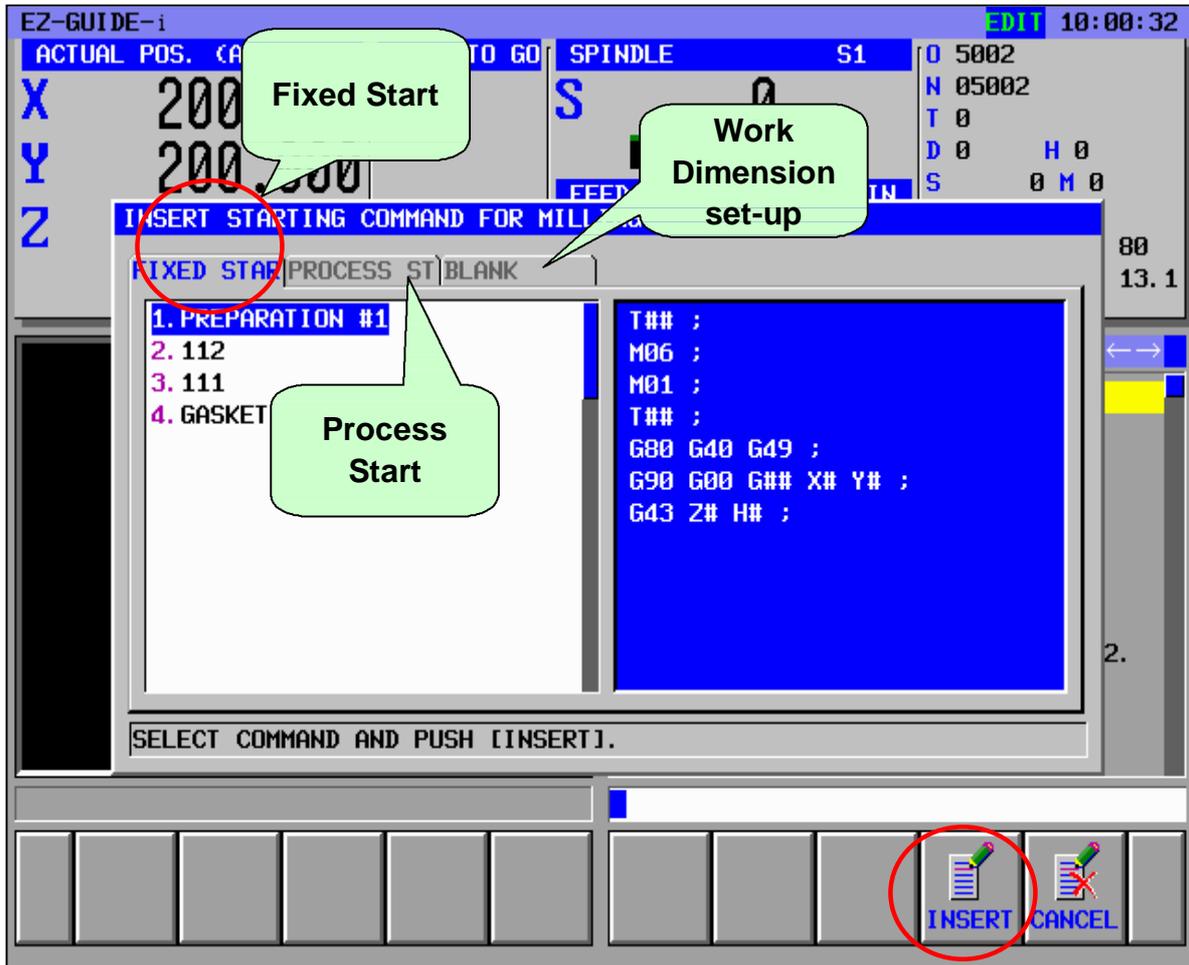
```
T## M06 ;  
G80 G40 G49 ;  
G90 G00 G5# X0 Y0 ;  
G43 H## Z50. ;  
M03 S1000 ;
```

Even though it is also possible to input the above blocks manually or with the next coming **Process Start** menu, **Fixed start** is recommended to save time in many cases.

After pushing main **START** key, it goes to the screen on **Fig.2**.



<Fig.1 Start key to set-up pre-machining condition in edit mode>



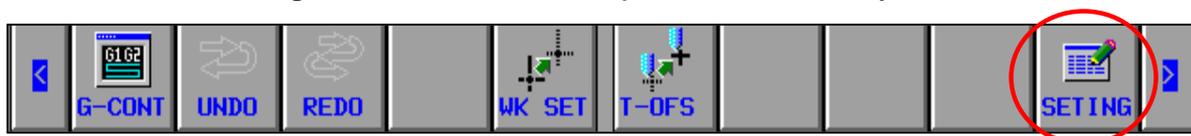
<Fig.2 Starting fixed format example>

At first, look up the format list on left side then choose suitable one for a program then push **Insert** key on the screen.

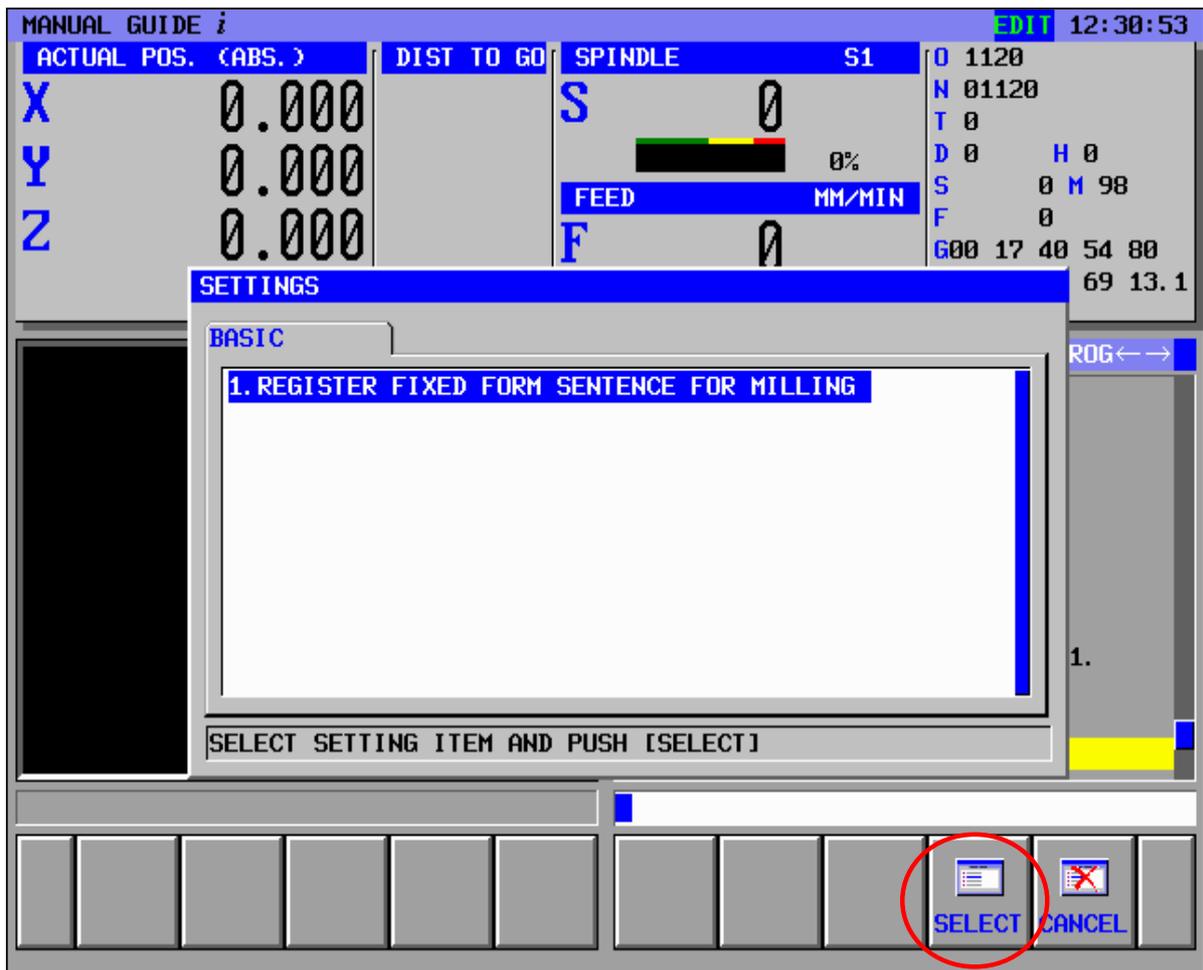
**Fixed Start** is a sub-menu in main **START** key, which has root in program start condition set-up process.

The following steps show how to establish **Fixed Start** format.

- ① Push **Setting** key in edit mode, which is accompanied by the next setting screen. After that, push **Select** key.



<Fig.3 Setting key to edit fixed format in edit mode>



<Fig.4 Setting screen before editing fixed form>

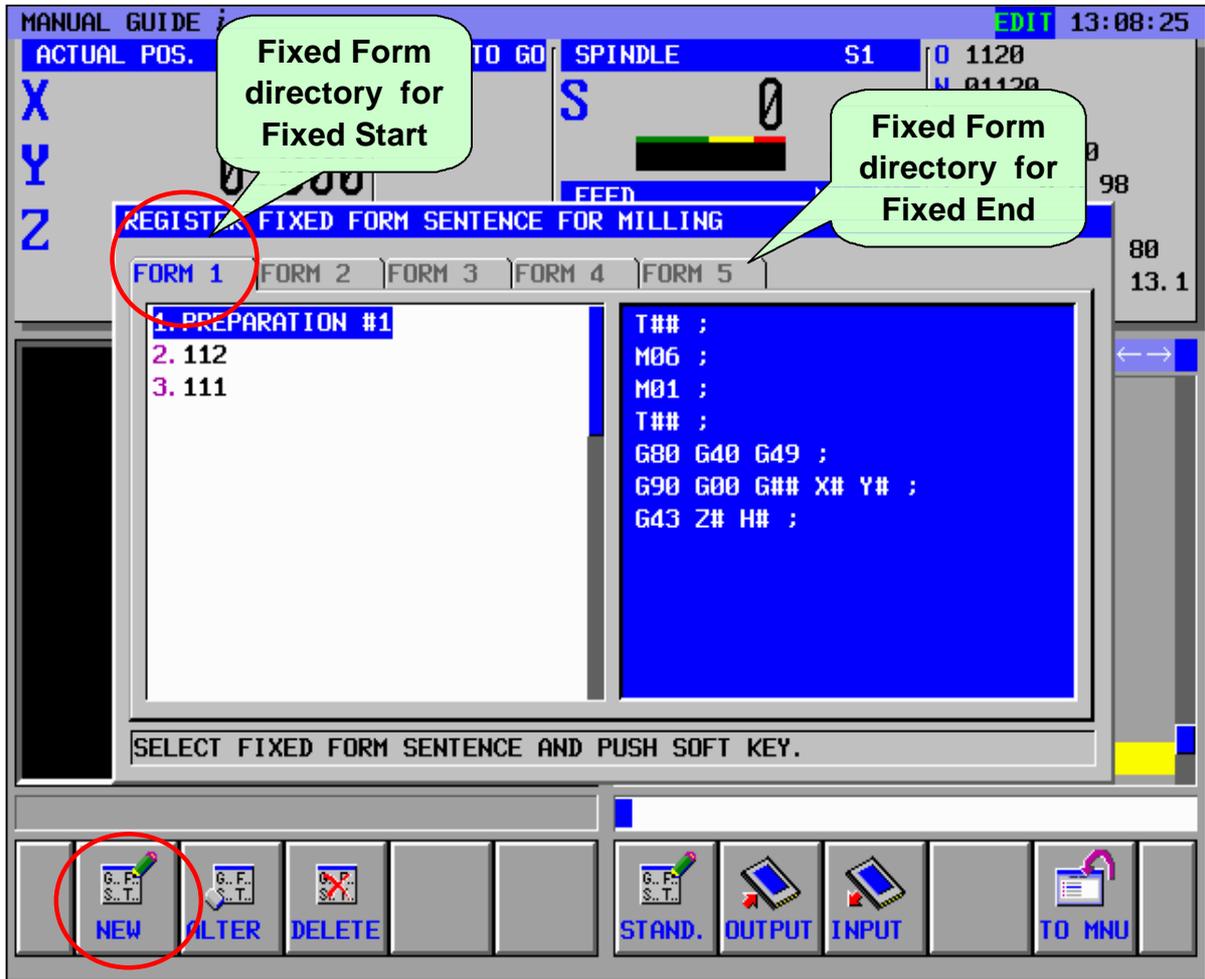
② You can find out 5 cells on the screen(**Fig.5**).

Among them, **Form 1** is directly related to **Fixed Start** and **Form 5** to **Fixed End**. It means all data in **Form 1** can be referred to in **Fixed Start** list.

In the same way, you can set-up **End** menu by inputting frequently used program ending blocks into **Form 5**.

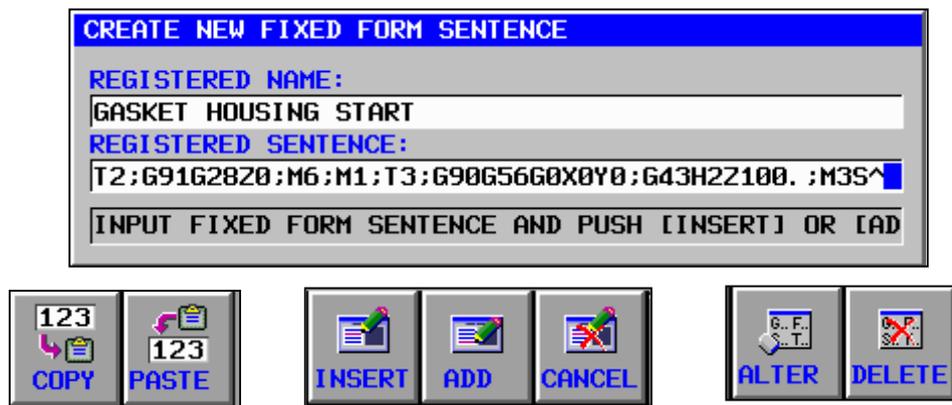
Another 3 cells(**Form 2-4**) is for storage of general program blocks which may be used frequently.

Press **NEW** key.



<Fig.5 Fixed Form registration main screen>

③ Register new name and actual program blocks in need in the blank.



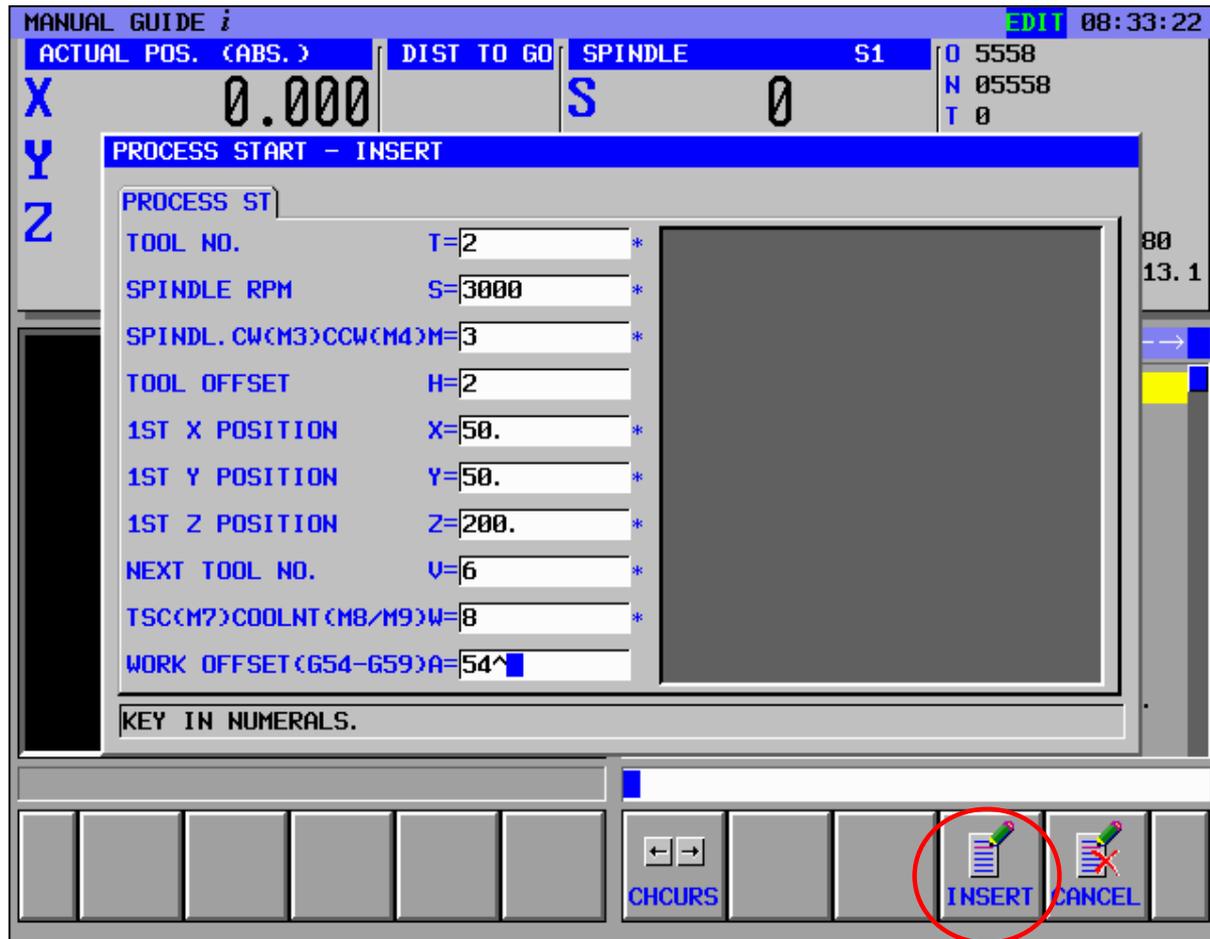
<Fig.6 Actual registration of Fixed Format for Start>

④ Make use of the several function keys above for editing.

## 3.2 Preparation for machining - *Process Start* (*START* menu set-up)

Press arrow key( → ) on **Fig.2** screen to find the following display.

After inputting necessary parameters, choose **Insert** key.



<Fig.7 Process Start menu in Start screen>

In case you choose the cutting condition as **Fig.7**, the following block is registered in the program you are making.

```
G300 T2 S3000 M3 H2 X50. Y50. Z200. V6 W8 A54
```

It is executed just like normal programming codes in actual process by macro execution as follows;

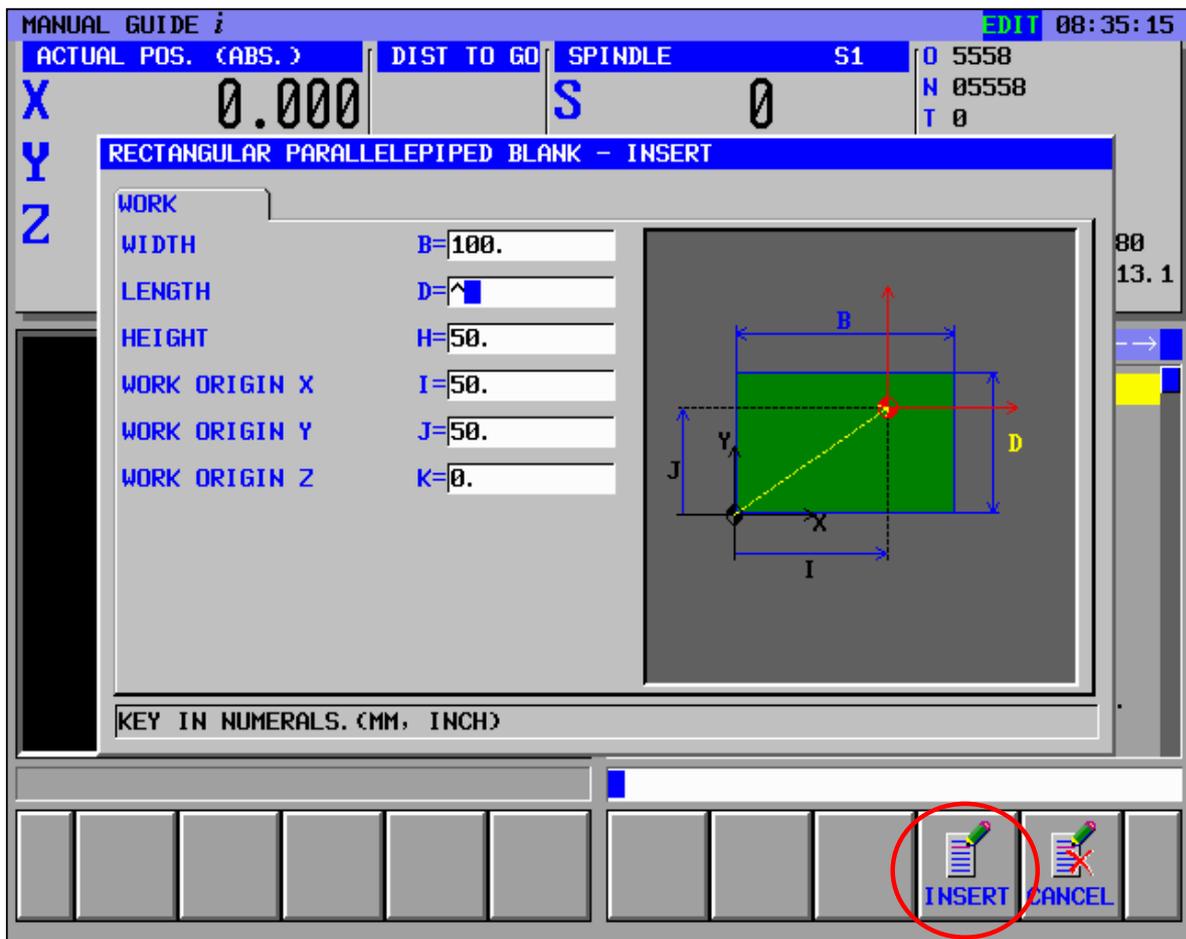
```
T2 M6 ;
G90 G0 G54 X50. Y50. M3 S3000 T6 ;
G43 H2 Z200. M8 ;
```

### 3.3 Preparation for graphic display - *Blank*(Dimension) (*START* menu set-up)

*EZ-i* enables animated simulation or tool path drawing of real part before or during machining for an operator who would like to make sure whether the program in use has any bugs or not.

For that reason, some parameters like general shape, diameter & length of tools and dimension of workpiece should be set for those graphic functions, apart from actual machining program blocks.

Workpiece dimension setting screen is in *Start* menu(**Fig.2**).

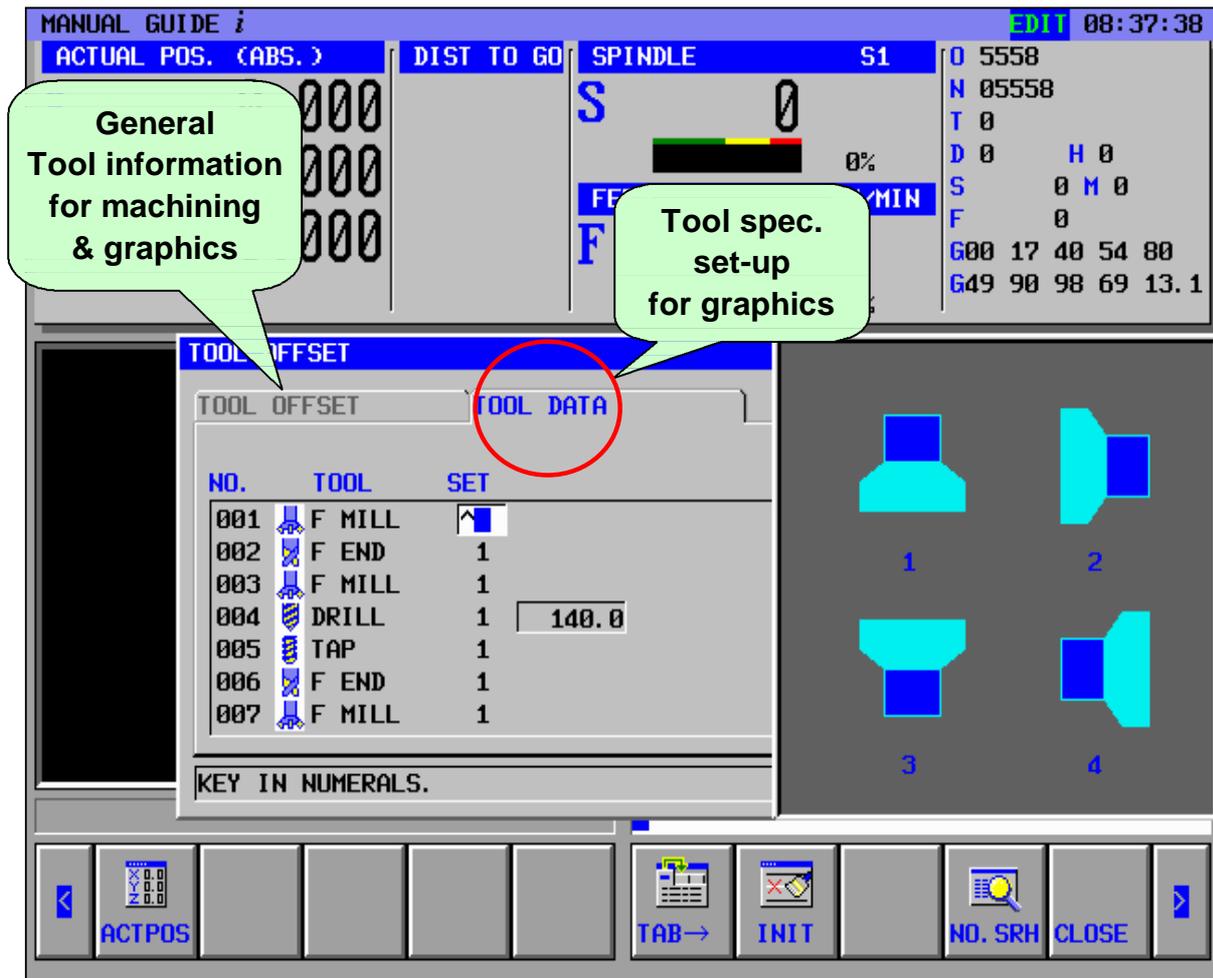


<Fig.8 Workpiece dimension setting for graphic display>

Press arrow key( → ) on **Fig.2** screen to find the above display, and after inputting adequate parameters, press *Insert* key.

### 3.4 Preparation for graphic display - Tool specification

Regarding tool data input for graphic display, Doosan Infracore recommends **Tool Data Base function**, which has convenient concept for application.

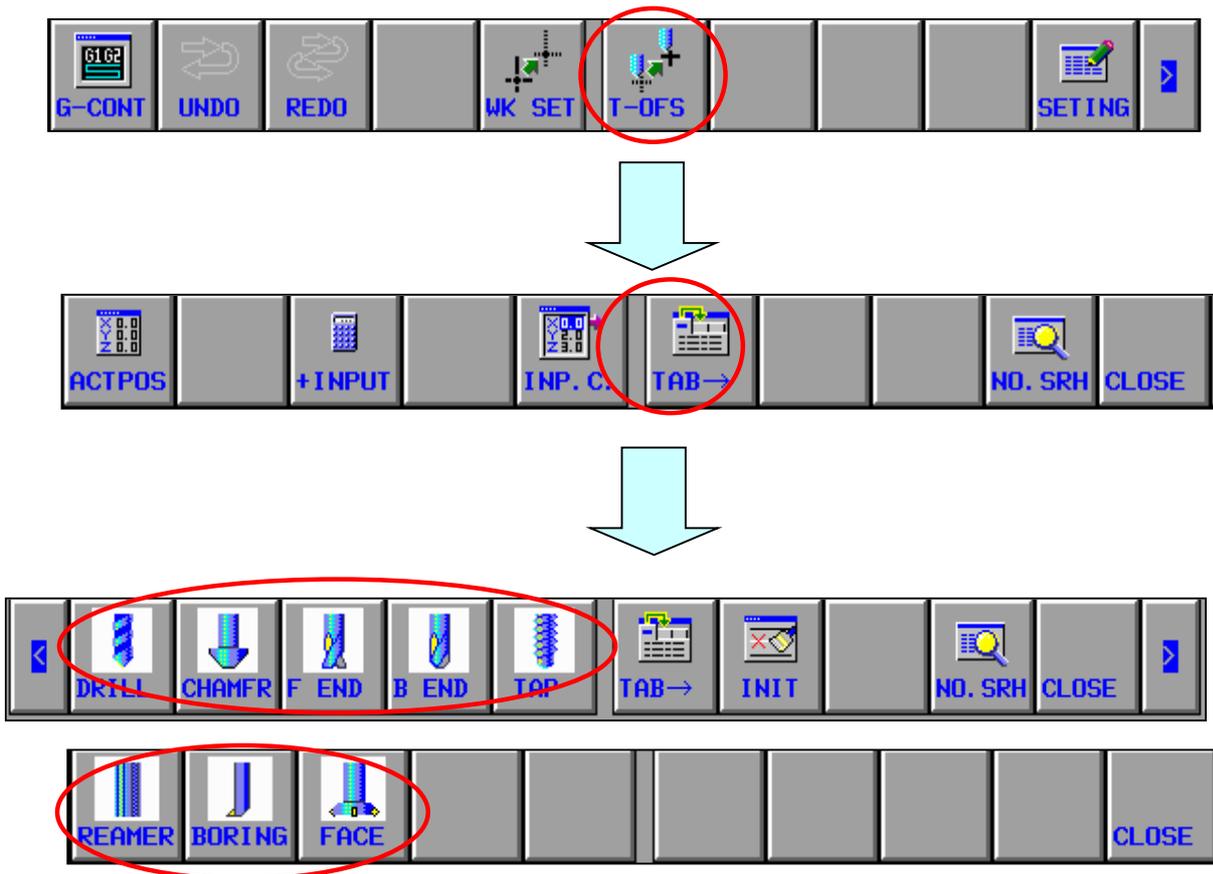


<Fig.9 Tool Data Base screen in tool offset >

#### NOTE

1. Tool Data Base function is enabled when bit 0 of parameter No.14850 = 0.
2. For more effective and easier operation, set bit 0 of parameter No.27002 = 0.

**Fig.10** shows how to find out and set graphic condition with regard to tool specification.



*<Fig.10 Tool type setting sequence for graphic display>*

Once you establish tool length, radius, type and direction, there is **NO** need to set-up them for simulation or tool path drawing in every cycle. In addition, from the point of tool path's view, the radius value input will cover all milling cycles which need careful consideration of radius-directional cutting movement, by being the datum of calculation for the way ahead without G41/G42 compensation command.

It means, as mentioned before, programmers do not have to give so much consideration to tool diameter compensation(G41/G42) when using this software.

**EZ-i** is absolutely able to calculate and command optimized accurate tool path, even without that kind of troublesome programming factor which sometimes causes terrible damage to your workpiece.

For that reason, input the exact size of cutter in radius geometry column for both real machining and graphic check.

If you have reasonable experienced data for tool wear and would like to control any accuracy of **mass production work** which has tight tolerance, do not hesitate to make use of wear compensation.

Side flutes of the endmill in use will try to find out the same route as initial condition by internal calculation.

In normal cases, especially when you would like to set-up any new part, you may need to modify cutting condition parameters before starting main machining. Generally operators would change offset value when meet size troubles. For minute shift of machined size with **EZ-i**, however, operators should not change radius offset value but modify the program by altering cycle and figure configuration described in 3.5 and 3.6.

TOOL OFFSET				
TOOL OFFSET			TOOL DATA	
NO.	TOOL LENGTH COMP.		CUTTER COMPENSATION	
	GEOMETRY	WEAR	GEOMETRY	WEAR
001	▲	0.000	50.000	0.000
002	100.000	0.000	10.000	0.000
003	120.000	0.000	40.000	0.000
004	80.000	0.000	4.000	0.000
005	80.000	0.000	8.000	0.000
006	120.000	0.000	2.500	0.000
007	0.000	0.000	10.000	0.000

KEY IN NUMERALS.

<Fig.11 Tool information setting cell for graphic display & machining>

The following example shows how to activate tool definition for graphics.

<b>Start</b> condition set by <b>Fixed Start</b> or <b>Process Start</b>	<b>T2 M6 ;</b>  <b>G90 G0 G54 X50. Y50. M3 S3000 T6 ;</b>  <b>G43 H2 Z200. M8 ;</b>
<b>Shape</b> configuration	<b>G1902 B200. D200. H50. I100. J100. K0 ;</b>
<b>Tool set</b> for graphics	<b>D2 ; (Manually input)</b>
<b>Main cycle</b> definition	<b>G1060 T20. S5 .....</b>

**NOTE**

1. If tool data for graphics is not defined by D# parameter in single block(i.e D3 ; ), alarm 3064(Tool Data is short.) appears.
2. For all kind of graphics, do not forget to insert D# single block after Shape configuration(G1902) manually.

### 3.5 Machining cycle selection

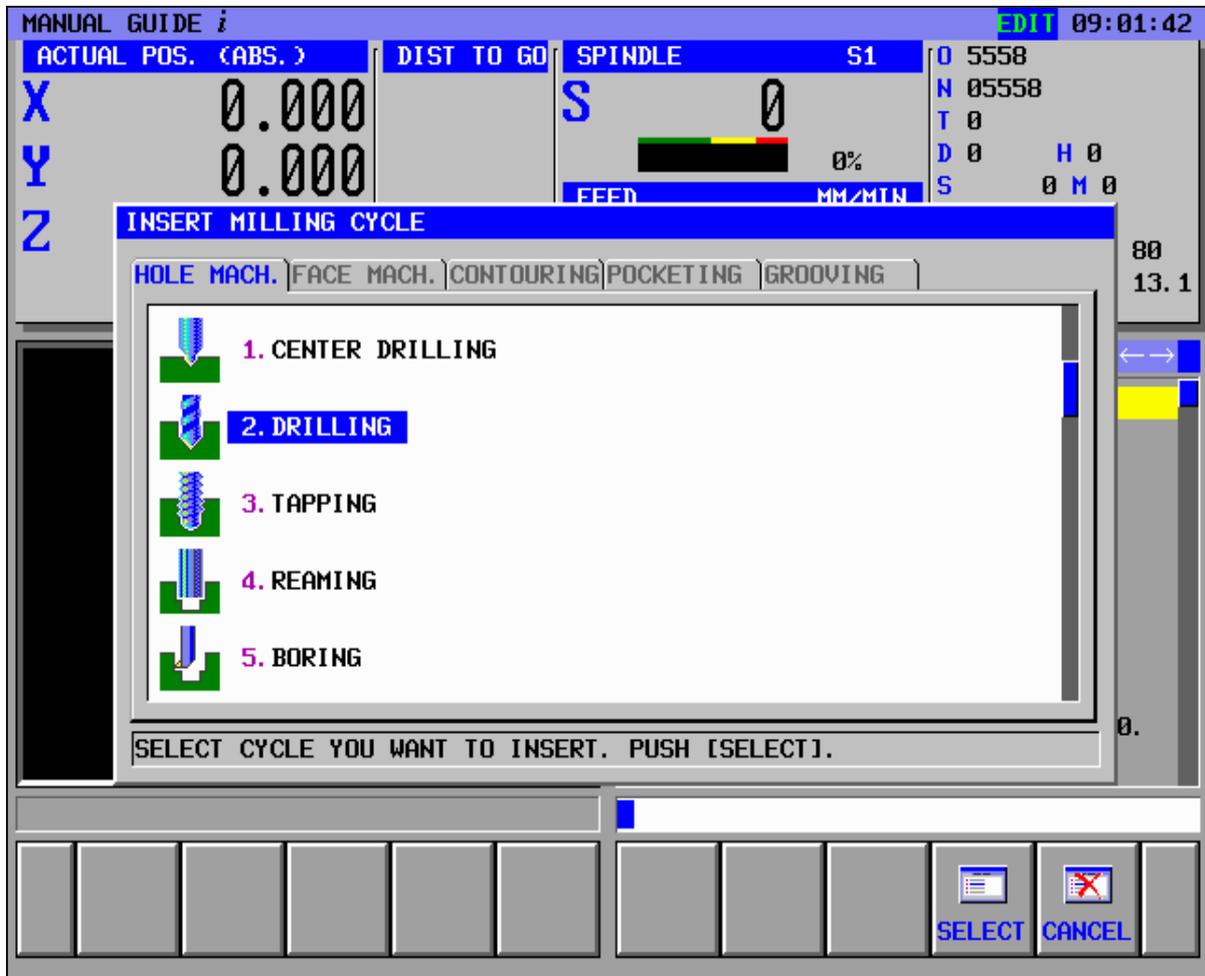
Two kinds of machining types are expected to be in general use for any parts at the customer's side.

Roughly speaking, one is hole making and the other is milling.

Face trimming, pocketing, grooving, outside endmilling (this is called as contouring in **EZ-i**) and chamfering come within the category of milling.

Also hole making covers drilling, tapping (normally Rigid Tapping), boring, reaming, and so on.

For further information, refer to the following chapter II.

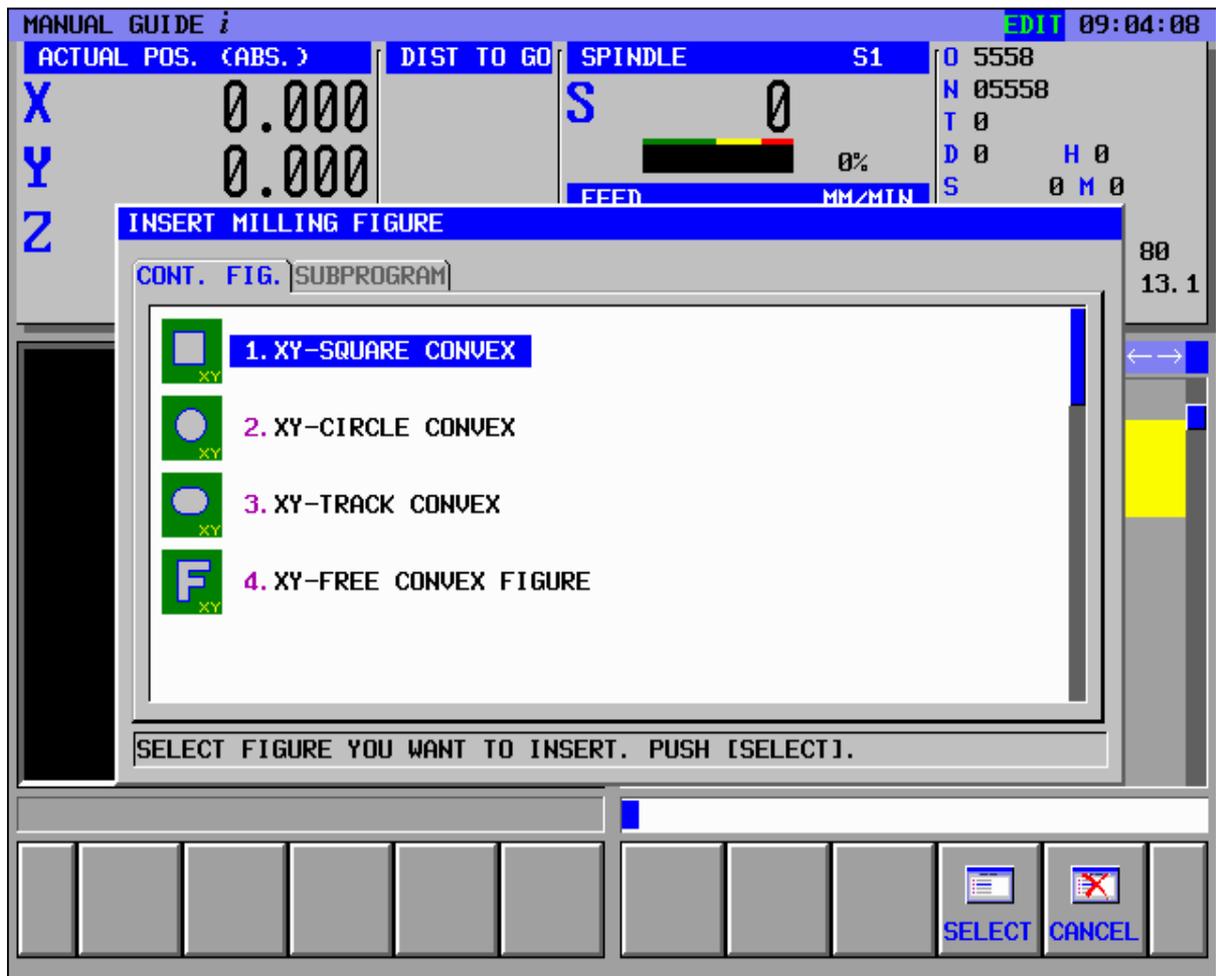


<Fig.12 Machining type selection in cycle menu>

### 3.6 Selecting Figures

In normal use, selecting a cycle makes the screen display proper figure automatically which is exclusively used for the already chosen cycle menu.

You can select fixed pattern figure or free one according to the program. For further information, refer to the following chapter II.



<Fig.13 Figure selection - coming right after cycle set-up>

### 3.7 Pre-check for mistakes in program

Machining programs can be checked without actual cutting, using animated simulation or realistic tool path drawing in **EZ-i**.

Both two ways are non-cutting methods, so you do not have to wait for a finished product or inspection result for finding bugs.

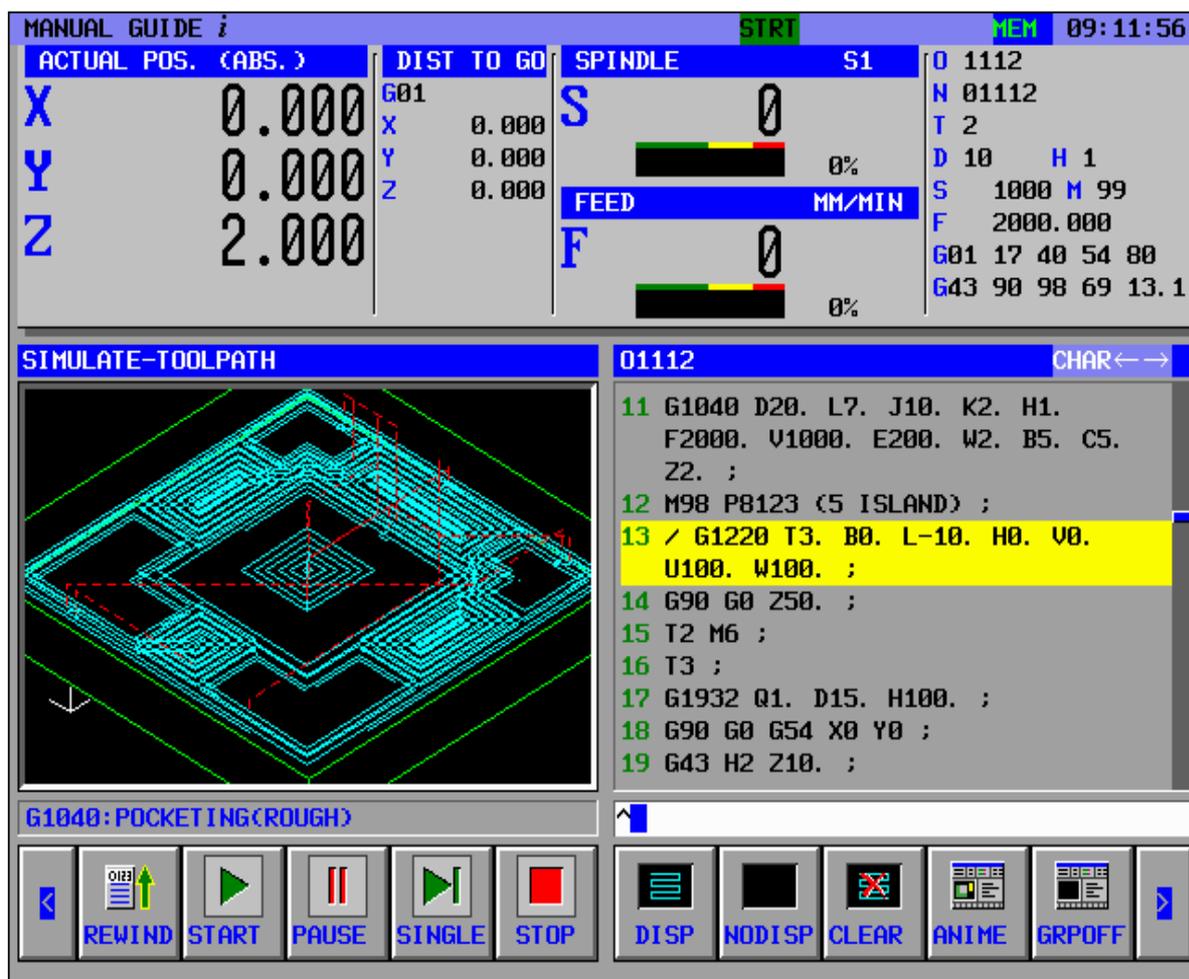
In addition, it takes less time to carry out the process than real machining.



<Fig.14 2D/3D Simulation (animation or tool path drawing) for pre-check>

#### NOTE

1. **EZ-i** provides only 2 types of material shape for **Animation** function. One is the rectangular and the other is the cylindrical.
2. For free style parts(i.e. automotive accessories) which need special clamp device, **Tool Path Drawing** is supposed to be used.



<Fig.15 Tool path drawing for pre-check>

In some cases, real time tool path confirmation may be more useful for operators. It shows both the frame of workpiece and the movement of tool tip while real machining is in operation.

You can execute the function pushing **MCHDRW** menu key on board. In this menu, the same screen displays during operation as **Tool Path Drawing** method.

Repeatedly speaking, **Animation** functions only in case of rectangular and cylinder type work for all machining cycles.

For arbitrary shape machining of **Facing** and **Outer Wall Contouring**, please make use of **Tool Path Drawing** or **MCHDRW** menu.

Another free figure embodiment on rectangular and cylinder work, however, can be pre-checked by **Animation** method.

## NOTE

1. Simulation(***Animation & Tool Path Drawing***) or drawing during machining function works only in memory mode.
2. With regard to scaling, rotating and moving graphic display, refer to the chapter II-9 in FANUC manual.

## **II . ACTUAL PROGRAMMING**

# 1.Surface work by *Facing* cycle

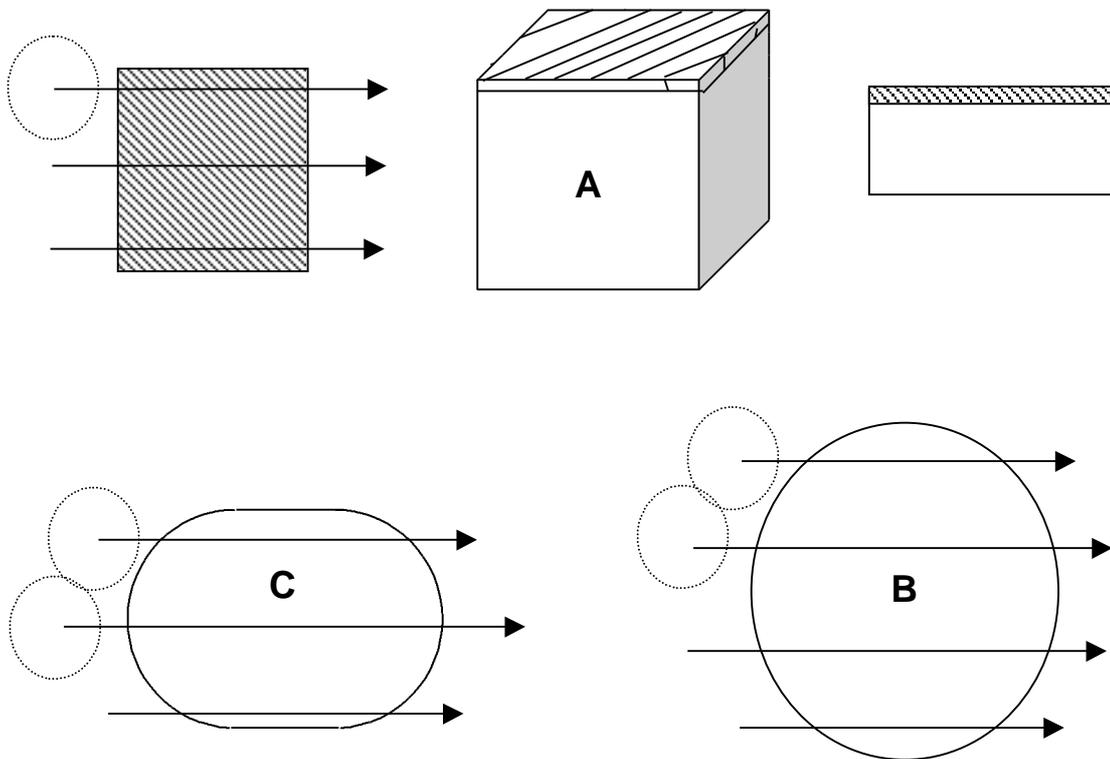
## 1.1 Main feature of *Facing* cycle

By using this cycle, the operator can carry out any routine trimming of whole face, with the shortest tool path.

Whole face is the key word to programming, for the function of the cycle is the chip removal of entire surface area to any commanded depth.

This cycle consists of rough and finish cut.

Regarding material shape, it provides 4 types for programming - 3 fixed(XY square, circle, track) & free shape facing.



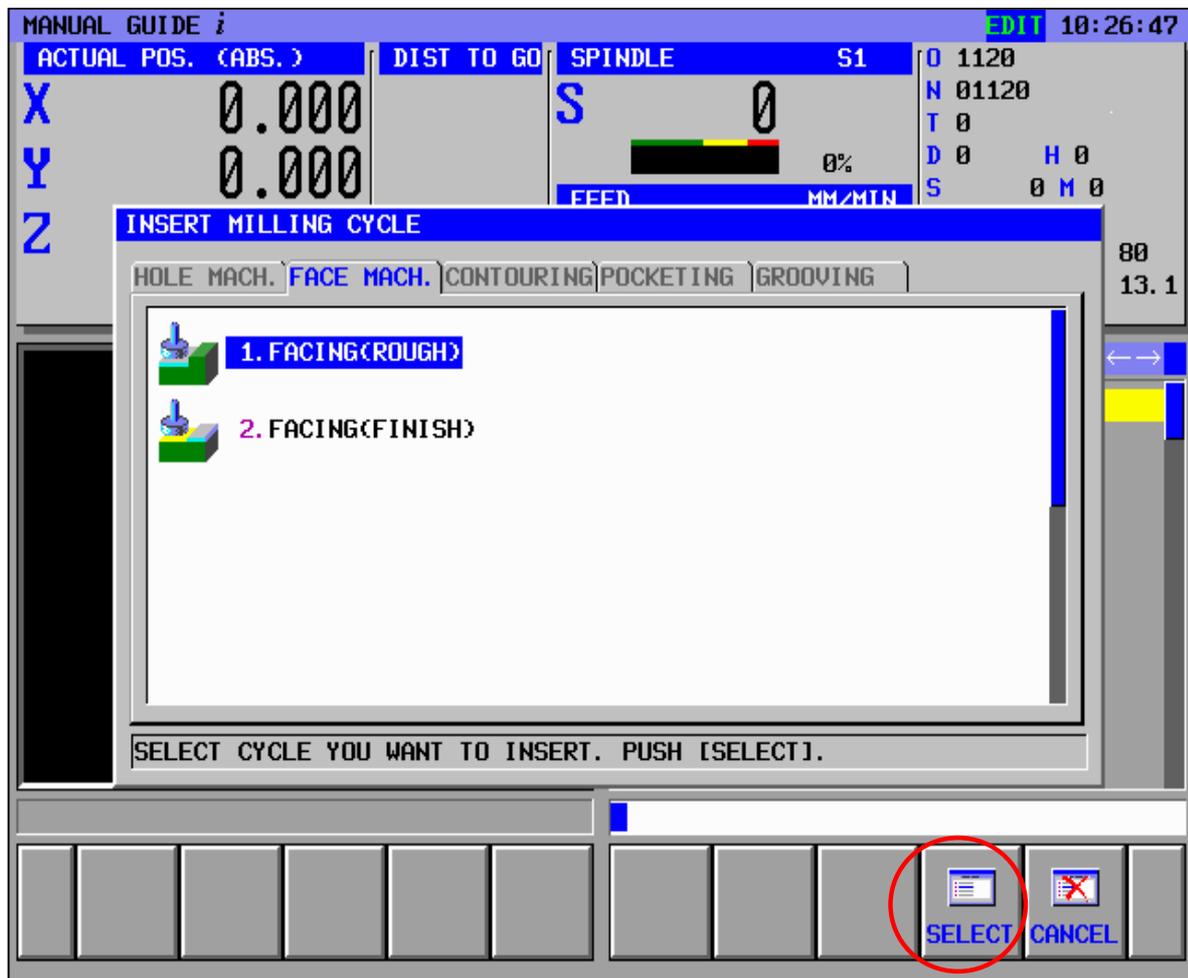
<Fig.16 Part shape in Facing cycle - fixed figure>

**Fig.16** shows how a face-cutter in use goes to remove the aimed depth of material which has fixed shape.

If you are going to trim the surface of a square or circle shaped part, it is possible you check the program made by **EZ-i** with animated simulation. But, the operator is supposed to do so with **Tool Path Drawing** menu in the case of track and free figure facing.

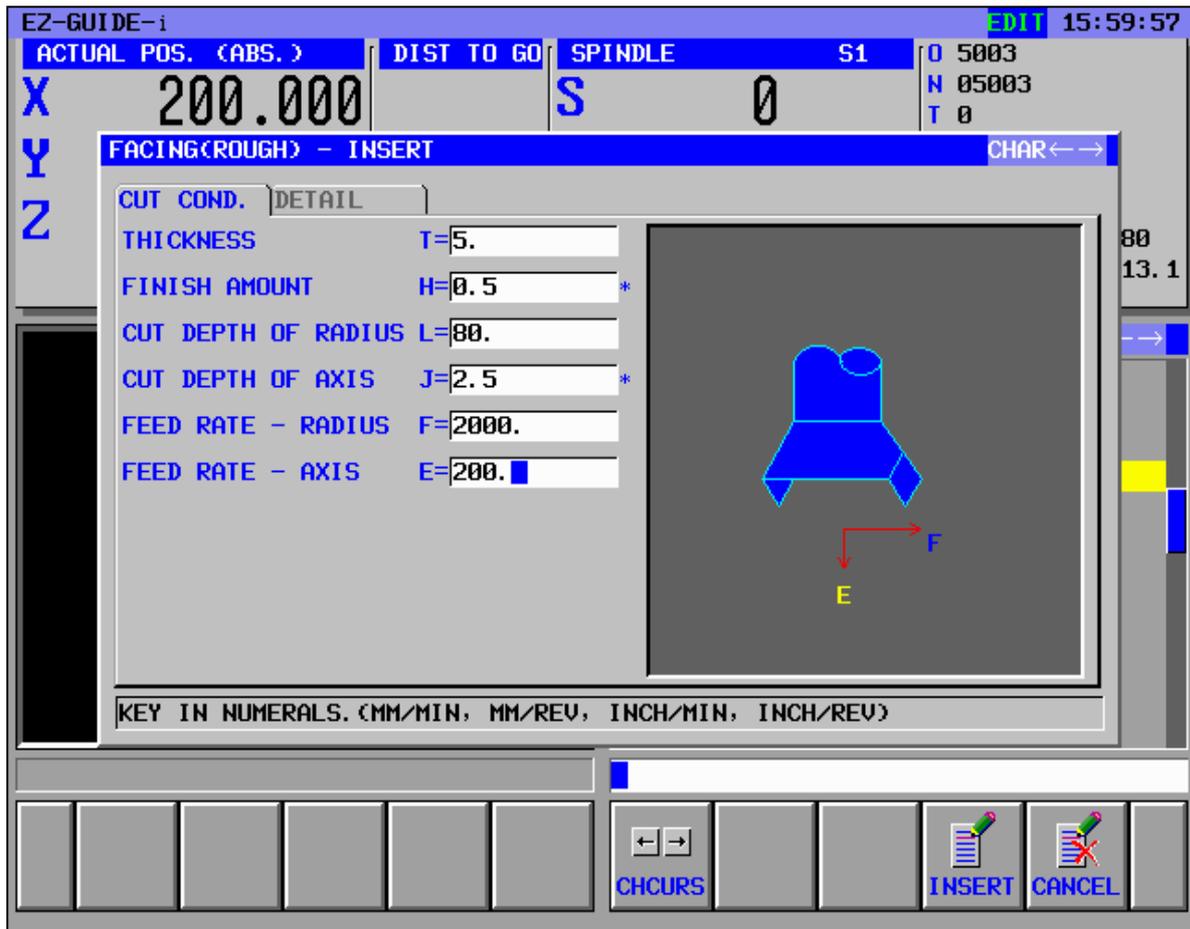
As indicated above **Fig.16**, **EZ-i** tries to make the shortest tool path by approaching face cutter as closely as possible to workpiece.

You can designate the approach and escape gap in the menu.



<Fig.17 Rough & finish mode selecting screen in Facing cycle>

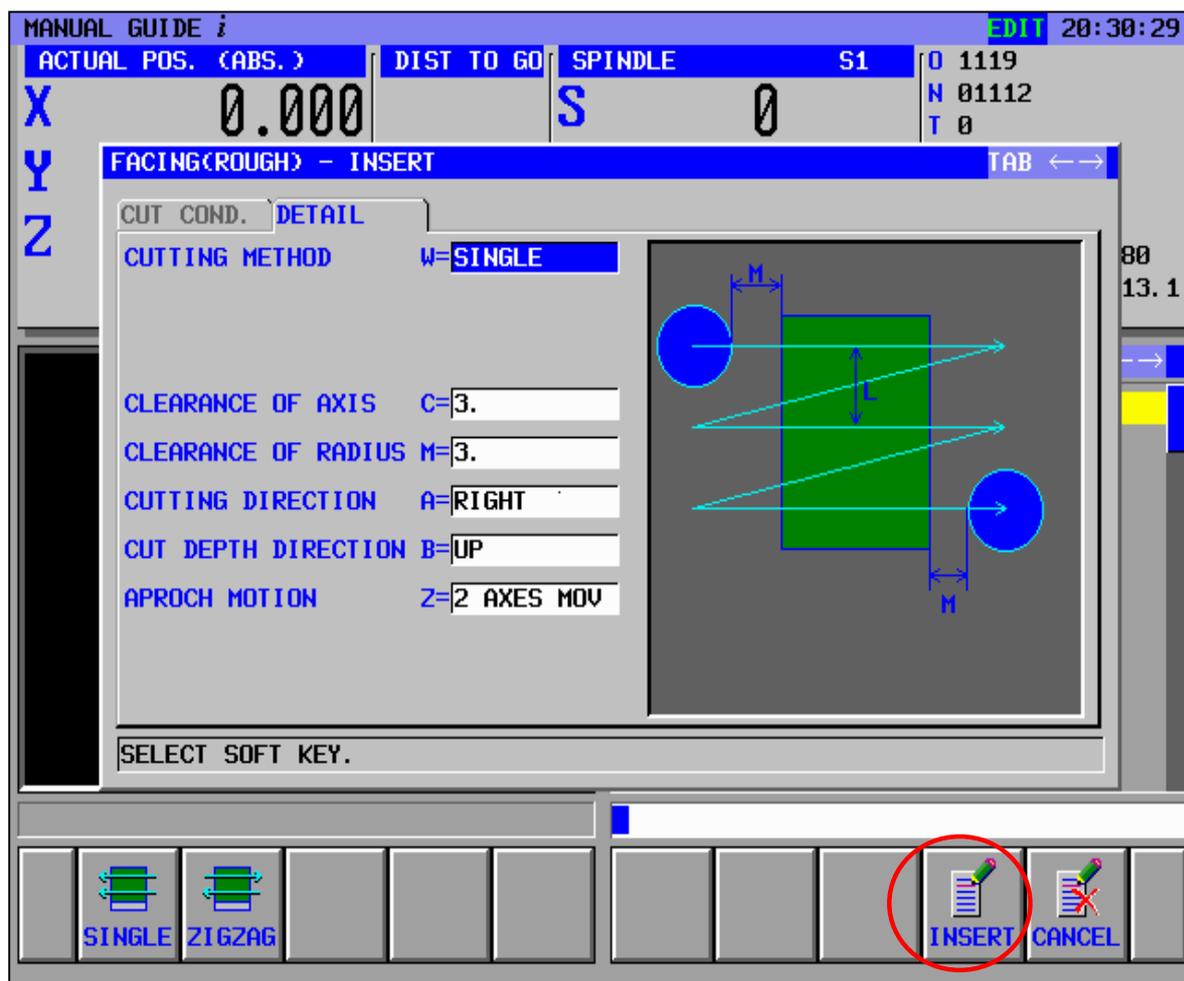
It is also recommended to set cut depth, width, feedrate, direction, and approaching method from an initial position to a part to be machined.



<Fig.18 Cut condition select screen in rough Facing cycle>

Parameter		Remark
T	Thickness	Whole cut depth(from surface to final bottom including finish amount)
H*	Finish Amount	In case you want finishing cut, leave adequate amount of depth.
L	Cut Depth of Radius	For input, take the hardness of material, rigidity of cutter and power of machine into careful consideration.
J*	Cut Depth of Axis	Set in case T value is too big for one path. For example, T=10mm/ J=5mm results in 2 times cut movement.
F	Feed Rate-Radius	Actual machining feed rate for cutting on surface
E	Feed Rate-Axis	When approach to work is performed by G01, set adequate value for safety.

Press arrow key(→ ) on **Fig.18** screen to find the following display.

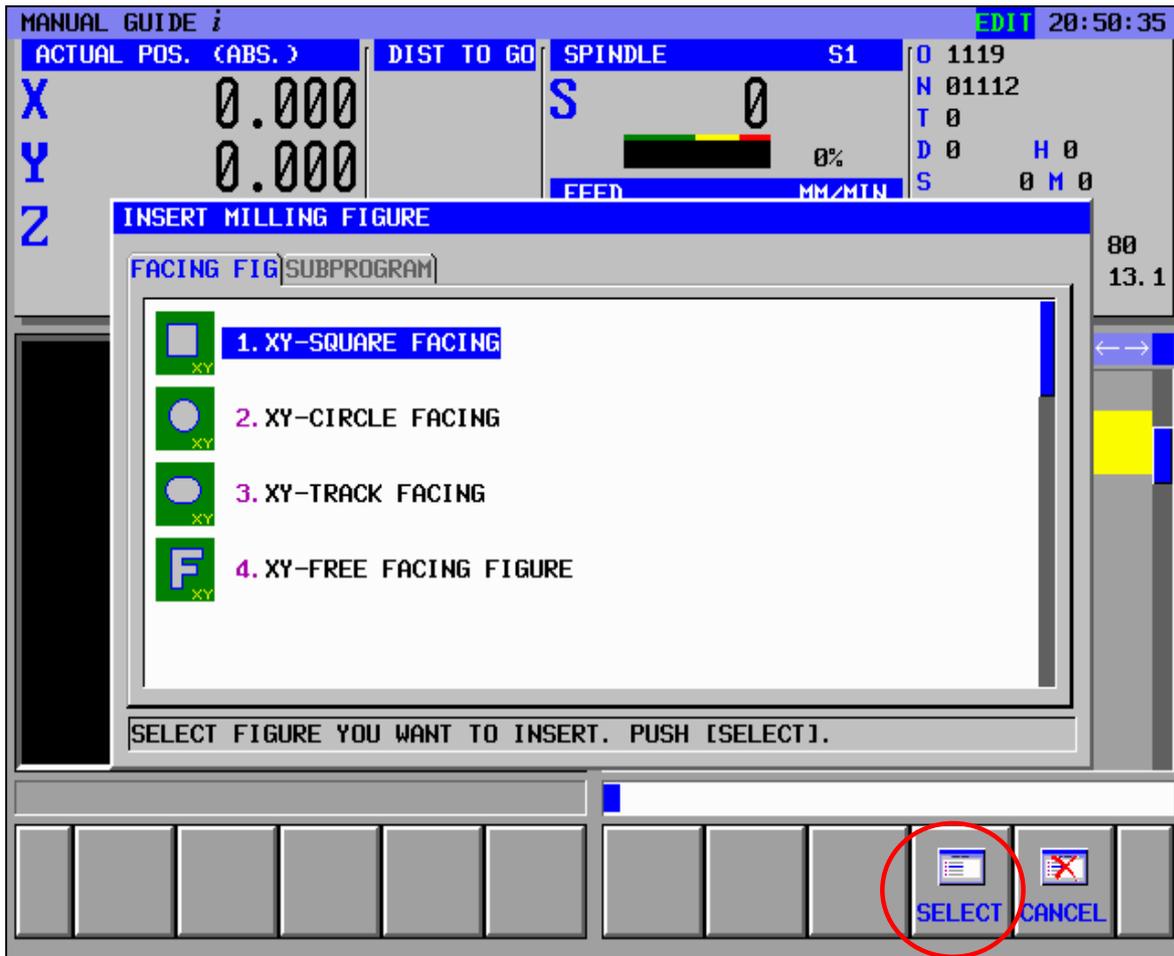


<Fig.19 Detail cut condition select screen in rough Facing cycle>

Parameter		Remark
<b>W</b>	Cutting Method	For better roughness and tool life, please choose single method.
<b>C</b>	Clearance of Axis	Approaching clearance from the start point in Z direction.
<b>M</b>	Clearance of Radius	Approaching clearance from the start point in XY plane.
<b>A</b>	Cutting Direction	Select which directional G01 movement to be performed.
<b>B</b>	Cut Depth Direction	Select which directional G00 shift for radius direction to be performed.
<b>Z</b>	Approach Motion	To prevent collision with work, set 2 axis movement for approaching.

## 1.2 Outline of work figure definition for *Facing* cycle

In all cases, entering a cycle cut condition displays continuously the following figure selection menu to be used exclusively for the already chosen *Facing* cycle machining type.



<Fig.20 Facing cycle figure selecting screen>

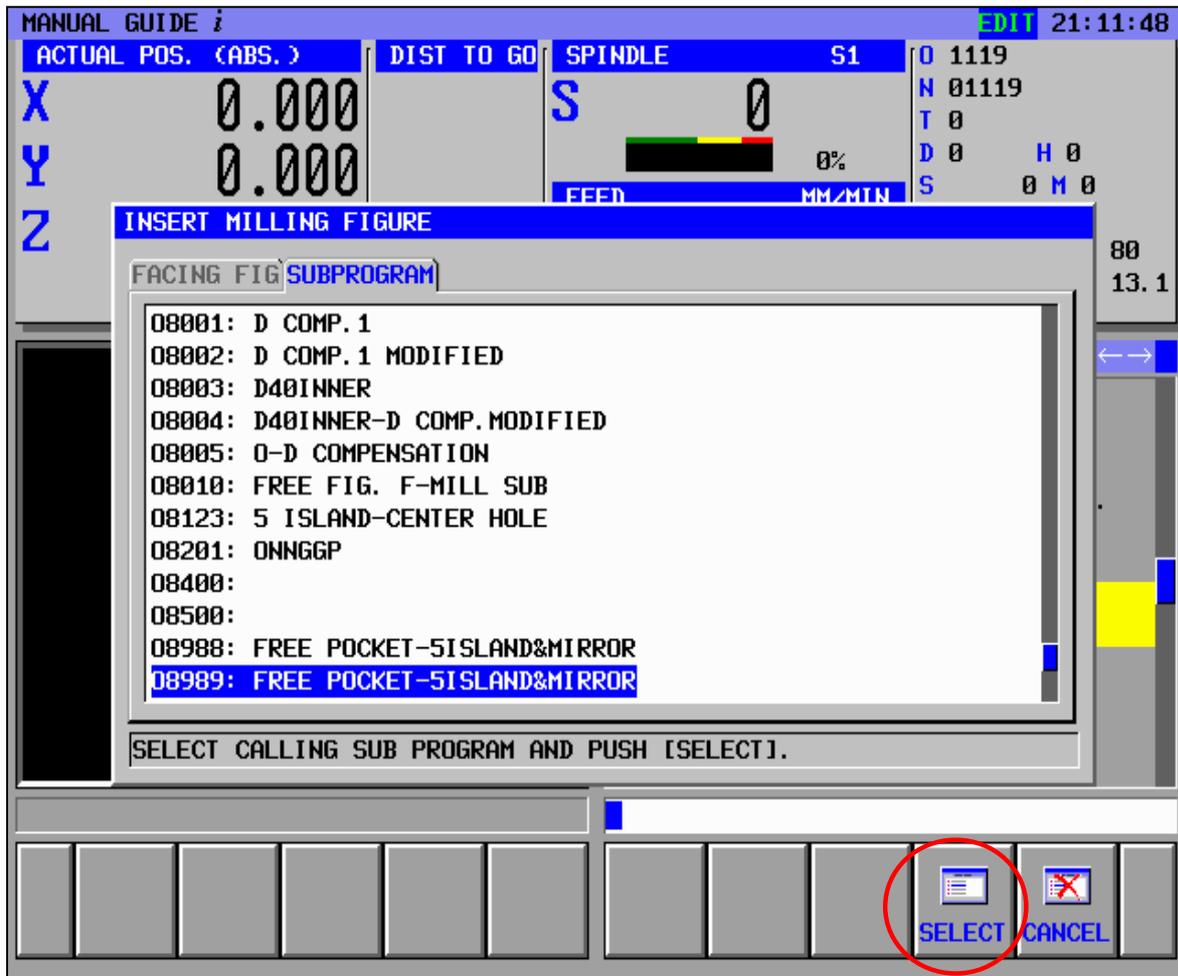
As mentioned before, *Facing* cycle provides 4 types for programming - 3 fixed(XY square, circle, track) & free shape facing.

Among them, when trimming of a square or circle shaped part, it is possible you check the program made by *EZ-i* with animated simulation. But, the operator is supposed to do so with *Tool Path Drawing* menu in the case of track and free figure facing, for animation functions only in case of cylinder or rectangular parallel piped blank.

Any frequently used shape can be stored as **Subprogram** in the directory prepared for the customer.

The number of such programs for identification is limited by parameter setting from #14722 to #14723.

Doosan Infracore recommends O8000-O8999 for machining center.

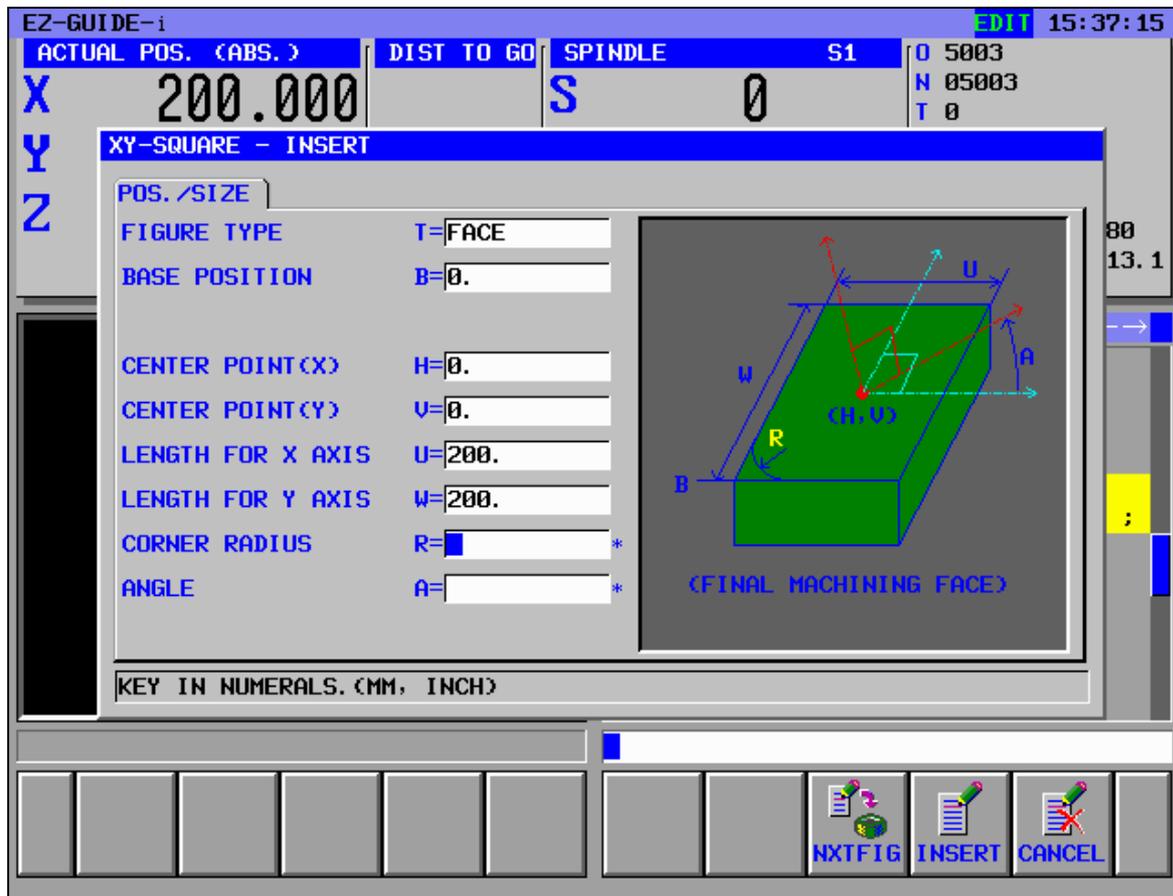


<Fig.21 Facing cycle figure selecting screen-using subprogram>

For free figure machining - not only **Facing** but also endmilling & even hole working, it is much convenient to store the shape in subprograms by creating a new one in the menu.

Detail method is explained in the following example of section 1.6

### 1.3 XY Square figure for Facing



<Fig.22 XY square figure setting screen in Facing cycle>

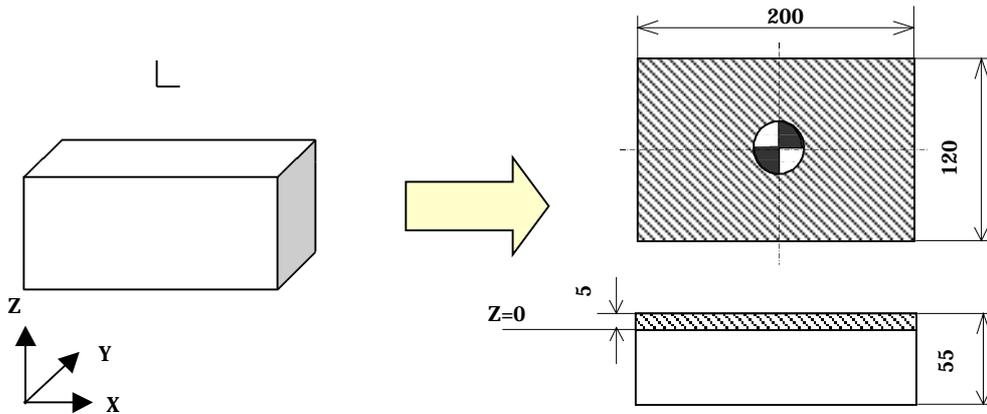
Parameter		Remark
<b>T</b>	Figure Type	Always fixed as <b>Face</b> (because Facing cycle is already selected).
<b>B</b>	Base Position	Z absolute coordinate of the <b>final machined surface</b>
<b>H / V</b>	Center Point(X / Y)	X / Y absolute coordinate of the center position of the figure
<b>U / W</b>	Length of X/Y axis	Length and width of the work

**Fig.22** shows an example parameter set-up for 200\*200\*50 square type workpiece whose center point is absolutely X=0/Y=0.

Z absolute coordinate of the surface was 5(initial material), and final machined face is 0. Compare with Fig.18

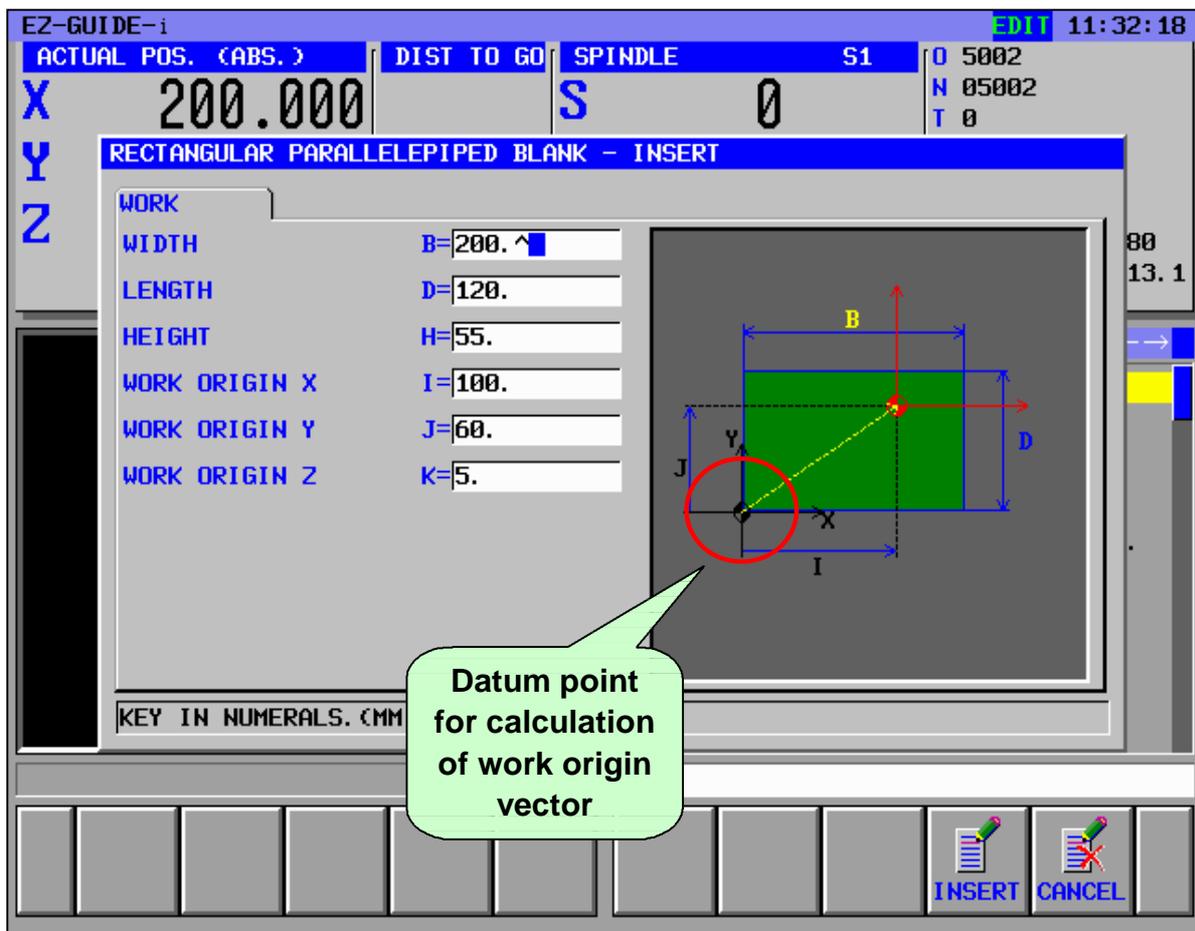
If there is any need to change those parameters, press **Alter** key in **Fig.1**.

The following example can be of help to understand the relationship between **Blank** and cycle figure set-up.



<Fig.23 Example material for Facing with 200\*120\*55 rectangular shape>

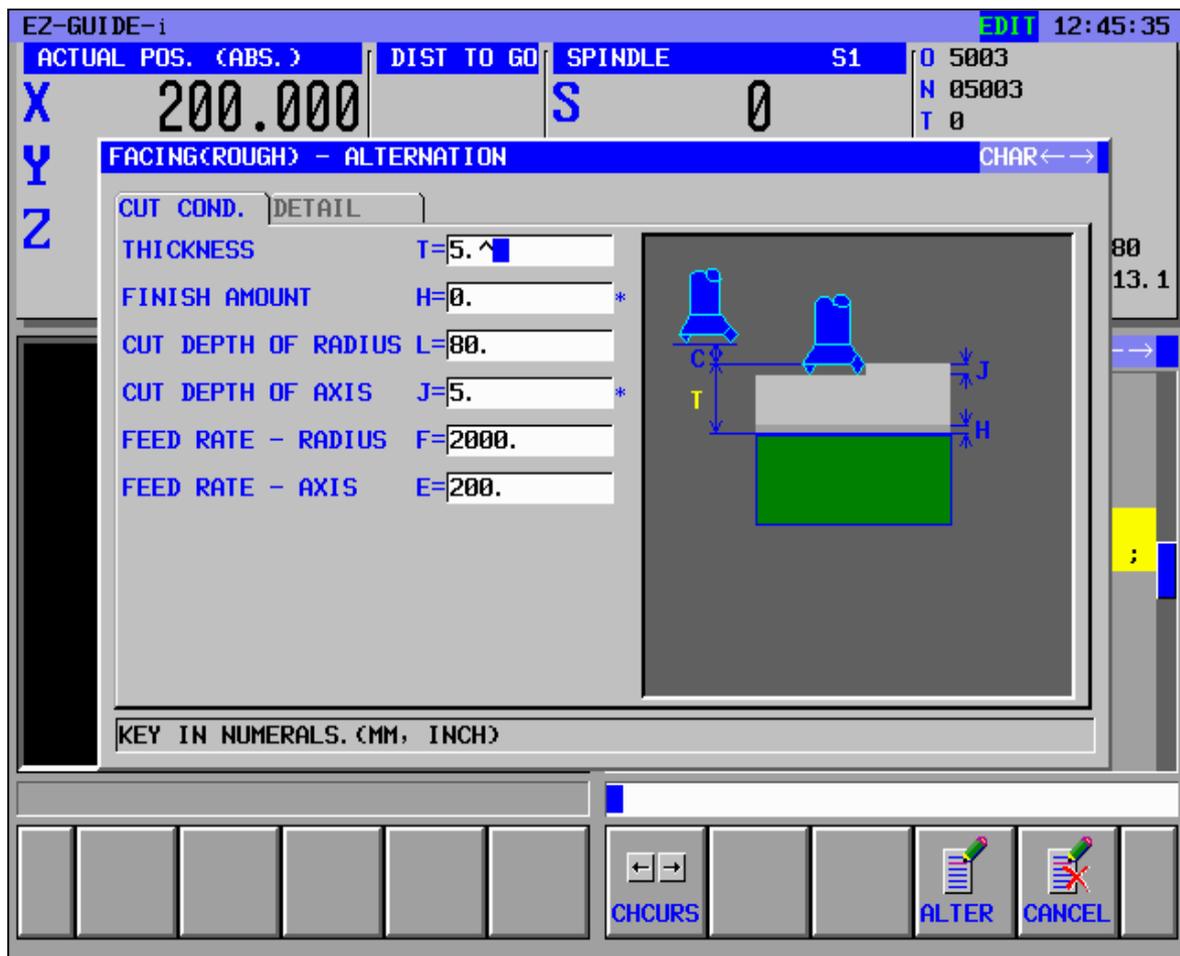
Raw cast material has 200\*120\*55 dimension and the cut depth is to be 5mm in Z direction, so final height of workpiece is 50mm.



<Fig.24 Setting parameters of Shape configuration for graphics>

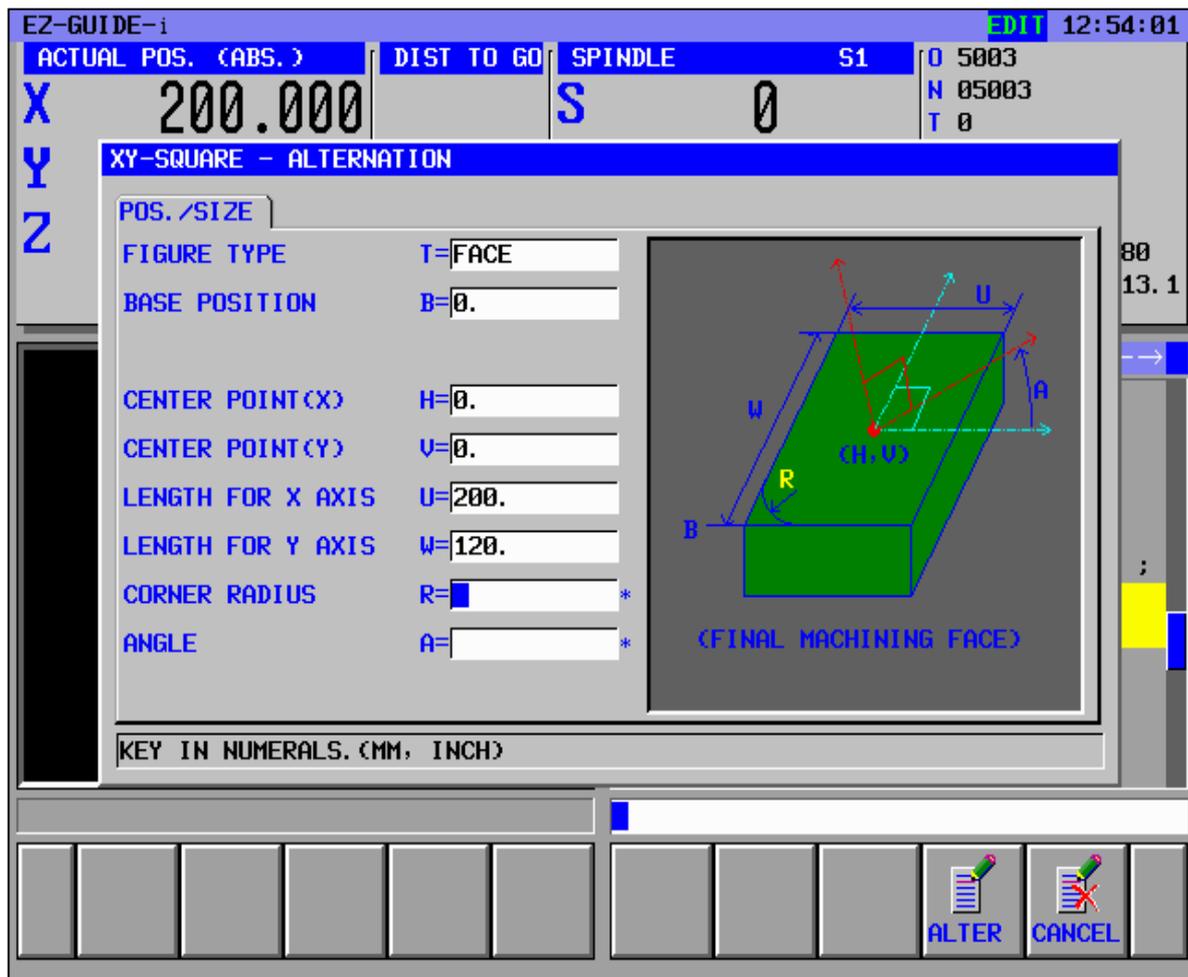
Parameter		Remark
B	Width	Width of workpiece (X direction)
D	Length	Length of workpiece (Y direction)
H	Height	Height of <b>RAW</b> workpiece (Z direction)
I	Work Origin X	X Vector amount of work offset 0 point from datum (refer to <b>Fig.24</b> )
J	Work Origin Y	Y Vector amount of work offset 0 point from datum (refer to <b>Fig.24</b> )
K	Work Origin Z	Z Vector amount of work offset 0 point from raw surface

After setting the above **Blank** parameters, you are supposed to define **Facing** cycle menu. Please refer to **Fig.18** & **Fig.19** for information. With regard to cycle cutting condition set-up, you are to pay attention to the Z axial cutting depth.



<Fig.25 Setting parameters of Facing cycle>

And final step is the **Figure** definition.



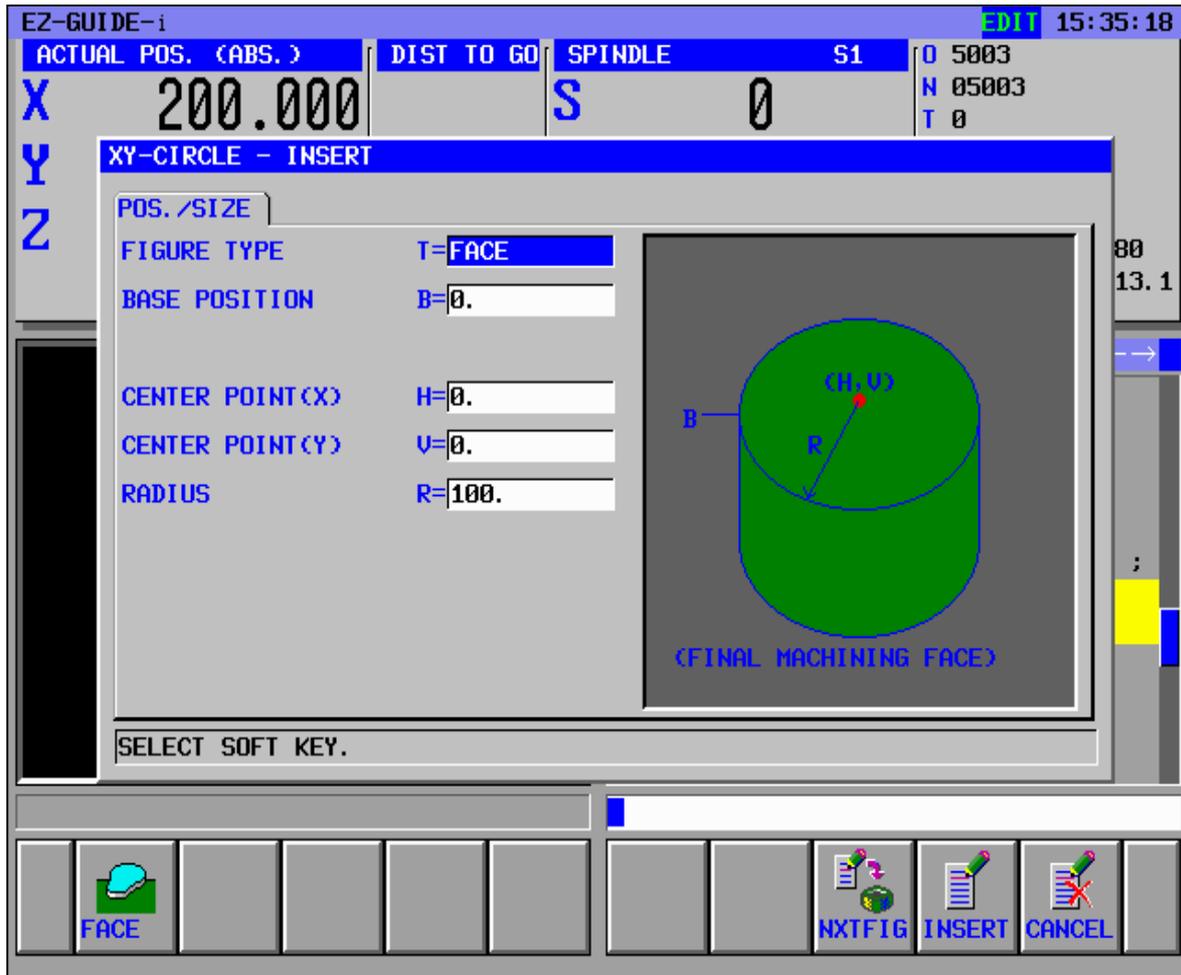
<Fig.26 Setting parameters of Facing XY square figure>

**Fig.24-Fig.26** show how to make actual face milling program from the point of work shape's view.

#### NOTE

1. Repeated data input of work shape configuration is going to be improved by FANUC for the customer's convenience when programming.

## 1.4 XY Circle figure for *Facing*



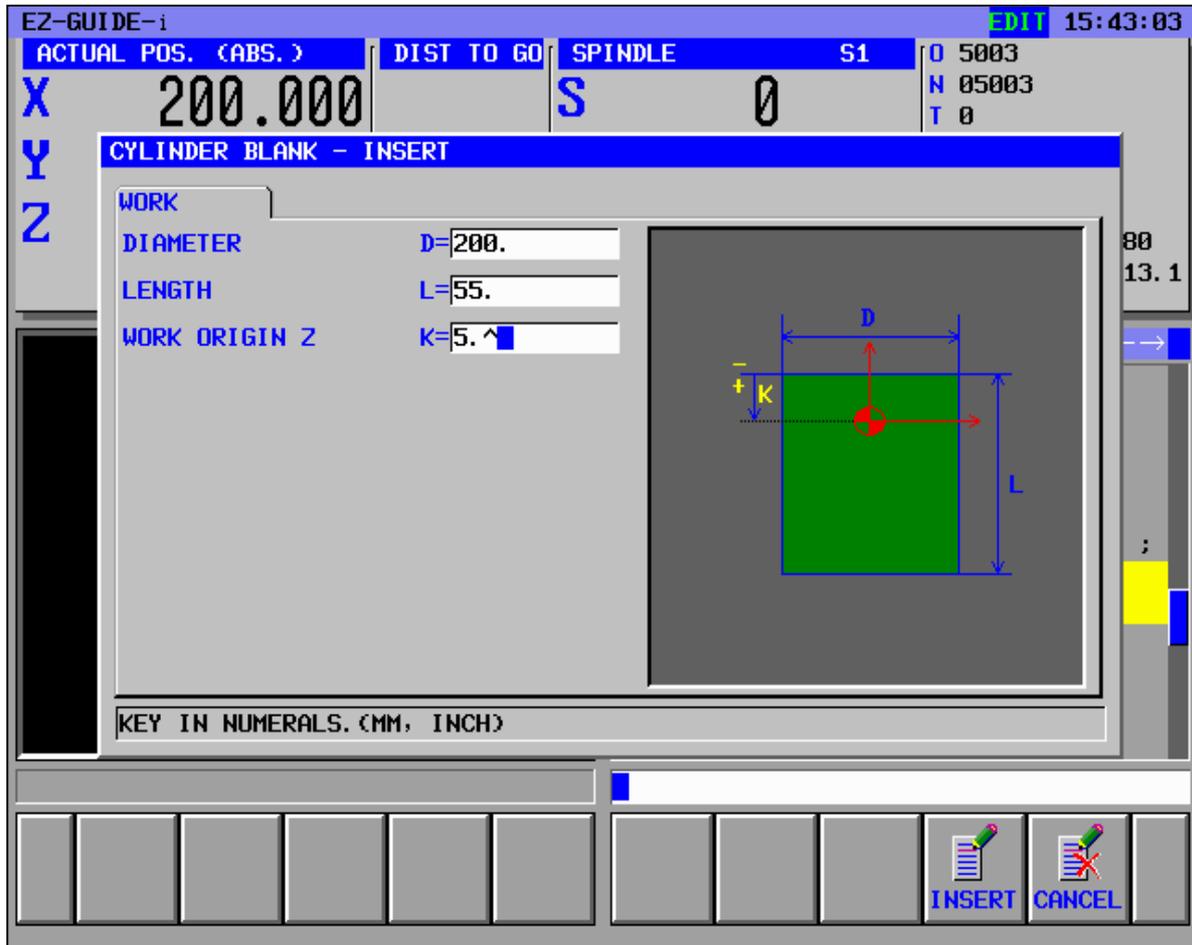
<Fig.27 XY Circle figure setting screen in Facing cycle>

Parameter		Remark
<b>T</b>	Figure Type	Always fixed as <b>Face</b> (because Facing cycle is already selected).
<b>B</b>	Base Position	Z absolute coordinate of the <b>final machined surface</b>
<b>H / V</b>	Center Point(X / Y)	X / Y absolute coordinate of the center position of the figure
<b>R</b>	Radius	Radius of work

**Fig.27** results from the cylinder type **Blank** set-up in **Start** menu.

Before the above setting, cycle definition just like **Fig.22** & **Fig.25** is necessary, too.

And regarding **Blank** set-up, next coming **Fig.28** shows how to do it.



<Fig.28 Cylinder Blank setting for XY Circle figure in Facing cycle>

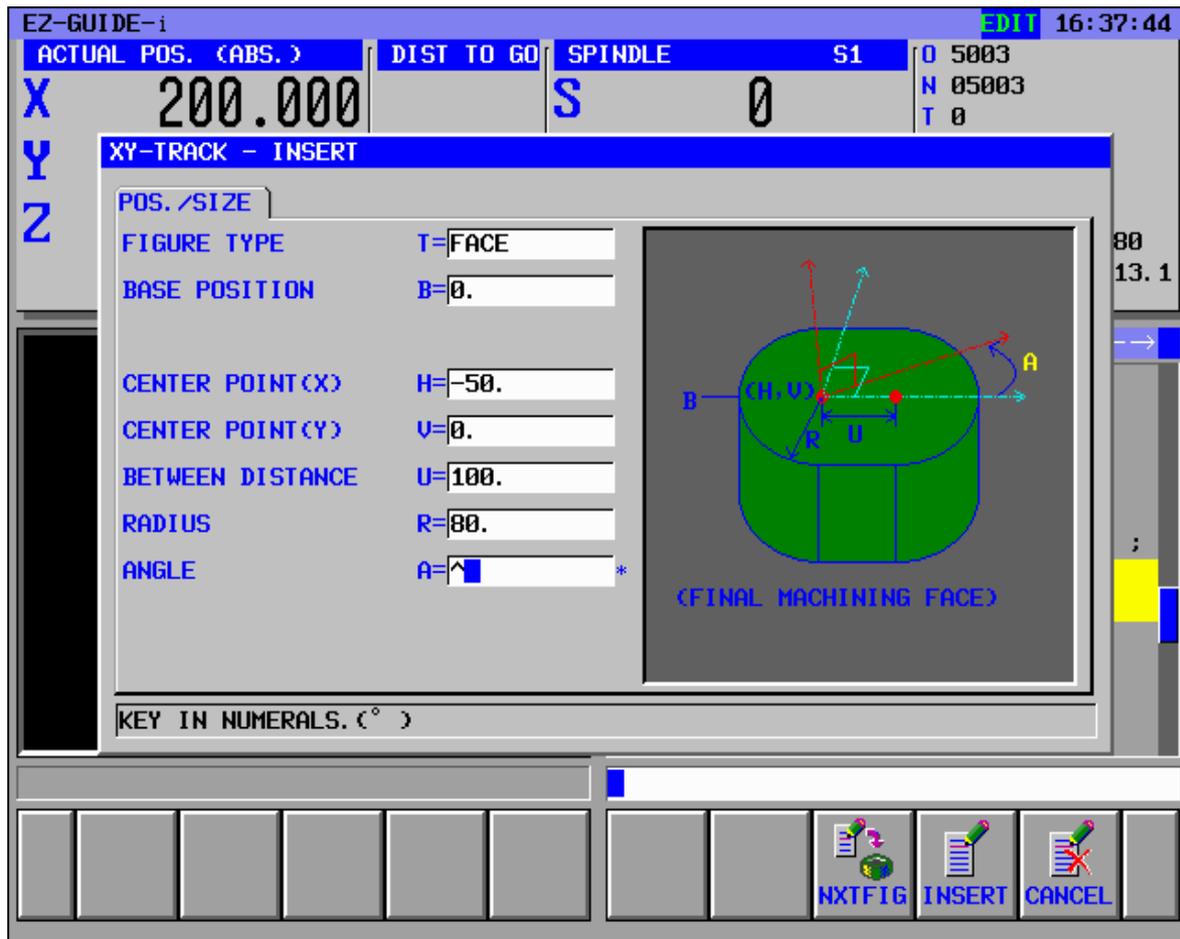
Parameter		Remark
D	Diameter	Diameter of cylinder type work
L	Length	Height of cylinder
K	Work Origin Z	Z Vector amount of work offset 0 point from raw surface

As mentioned before, when rectangular or cylinder type work is to be machined with **Facing** cycle, graphical **Blank** configuration is absolutely necessary for 3D animated pre-check of programming error. You must set related parameters properly on assigned screen for accurate simulation. In case of **Track** type fixed shape or **Free** figure, however, it is impossible to check by animation so that you have to make use of **Tool Path Drawing** or **MCHDRW** function.

## 1.5 XY Track figure for *Facing*

**Fig.18** C shows how *EZ-i* tries to optimize the tool path by shortening feed movement in track-cylinder type work face milling.

After set-up cycle cutting condition as **Fig.22** & **Fig.25**, you are supposed to fulfill the following parameter setting.



<Fig.29 XY track figure setting in Facing cycle>

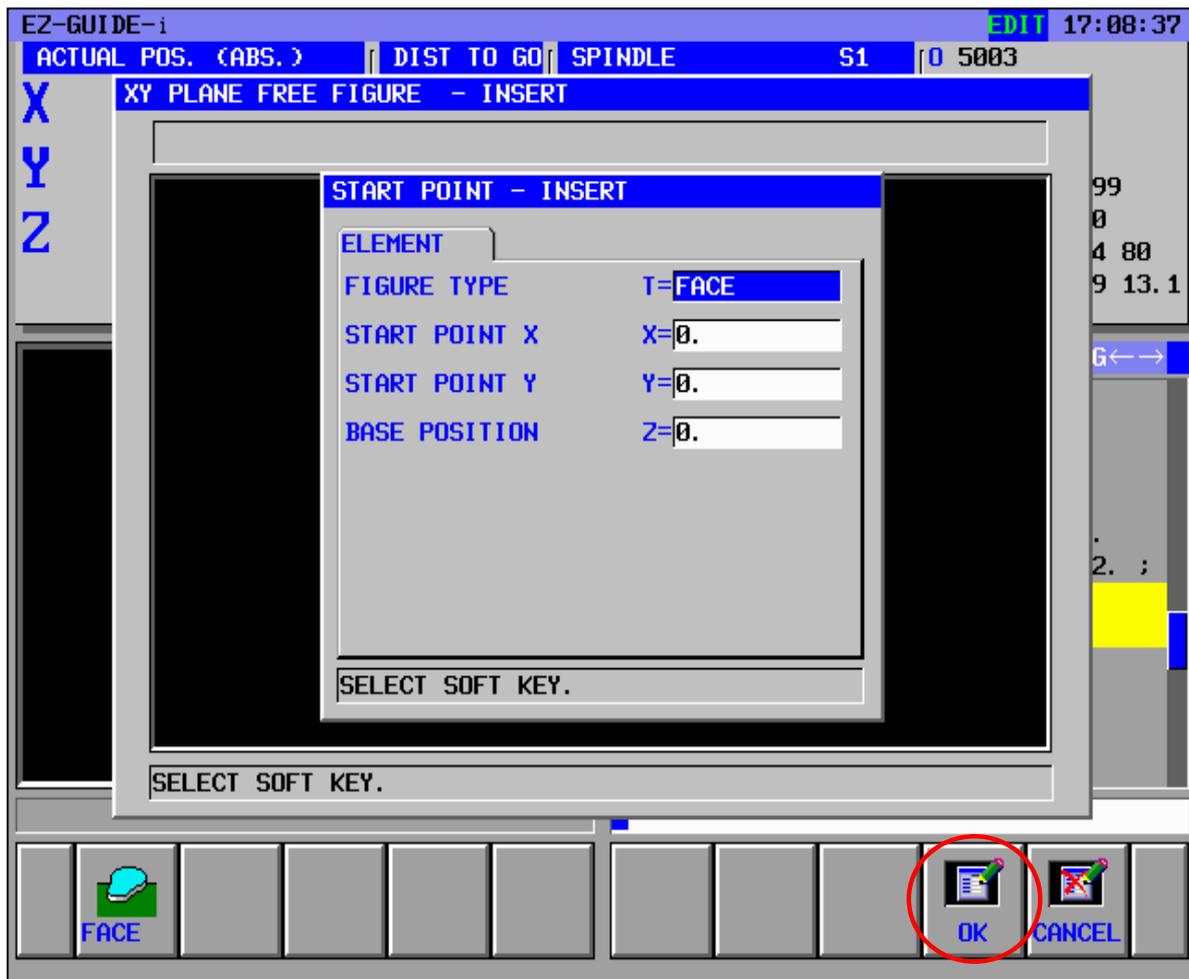
Parameter		Remark
<b>T</b>	Figure Type	Always fixed as <b>Face</b> (because Facing cycle is already selected).
<b>B</b>	Base Position	Z absolute coordinate of the <b>final machined surface</b>
<b>H / V</b>	Center point X / Y	X/Y coordinate of the left side center position
<b>U</b>	Between Distance	Distance between the left and right semi-circles
<b>R</b>	Radius	Radius of both semi-circle

## 1.6 XY Free figure for *Facing*

For milling cycles, a free figure can be specified by inputting data such as start point and linear / circular movement vector.

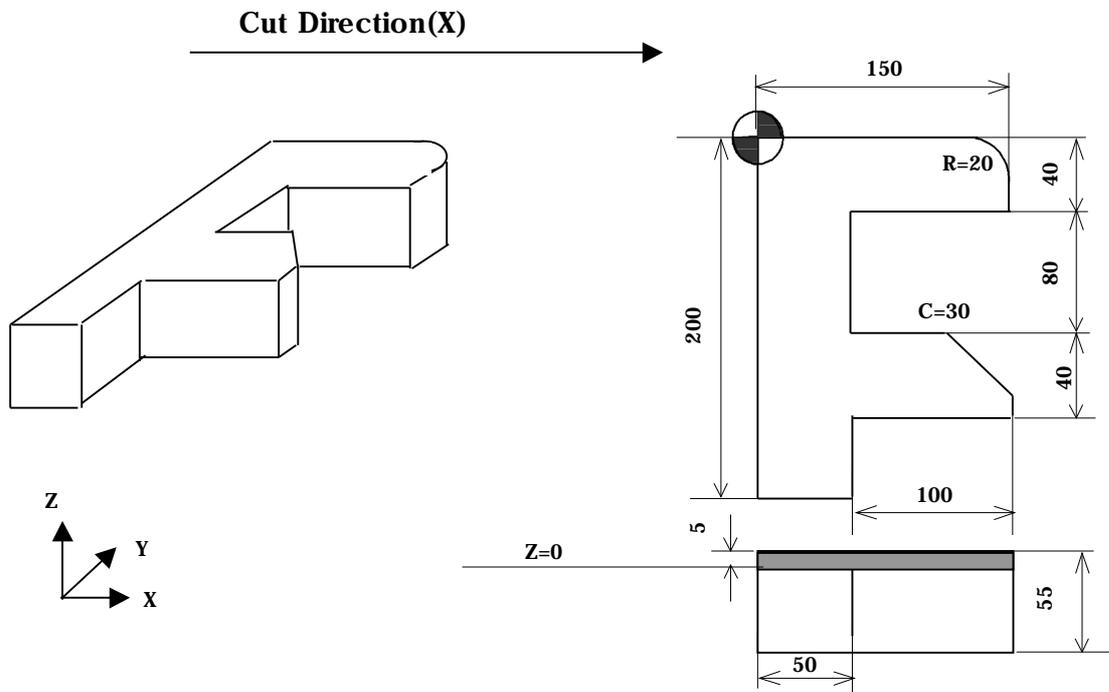
All cycles using milling cutter have similar concept to make arbitrary shape of work, only except for free figure **Pocket** with **Island**.

In that case, please refer to II-3 **Pocket** cycle.



<Fig.30 XY Free figure setting in Facing cycle - Start Point setting>

Parameter		Remark
T	Figure Type	Always fixed as <b>Face</b> (because Facing cycle is already selected).
X / Y	Start Point X / Y	X / Y absolute coordinate of the start point of an arbitrary figure
Z	Base Position	Z absolute coordinate of the <b>final machined surface</b>



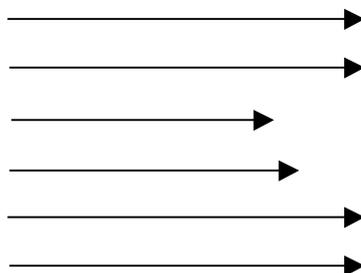
<Fig.31 XY Free figure example in Facing cycle - Tool path outline>

For the example above, 50mm face cutter is used and 5mm is to be machined to make Z reference surface( $Z=0$ ).

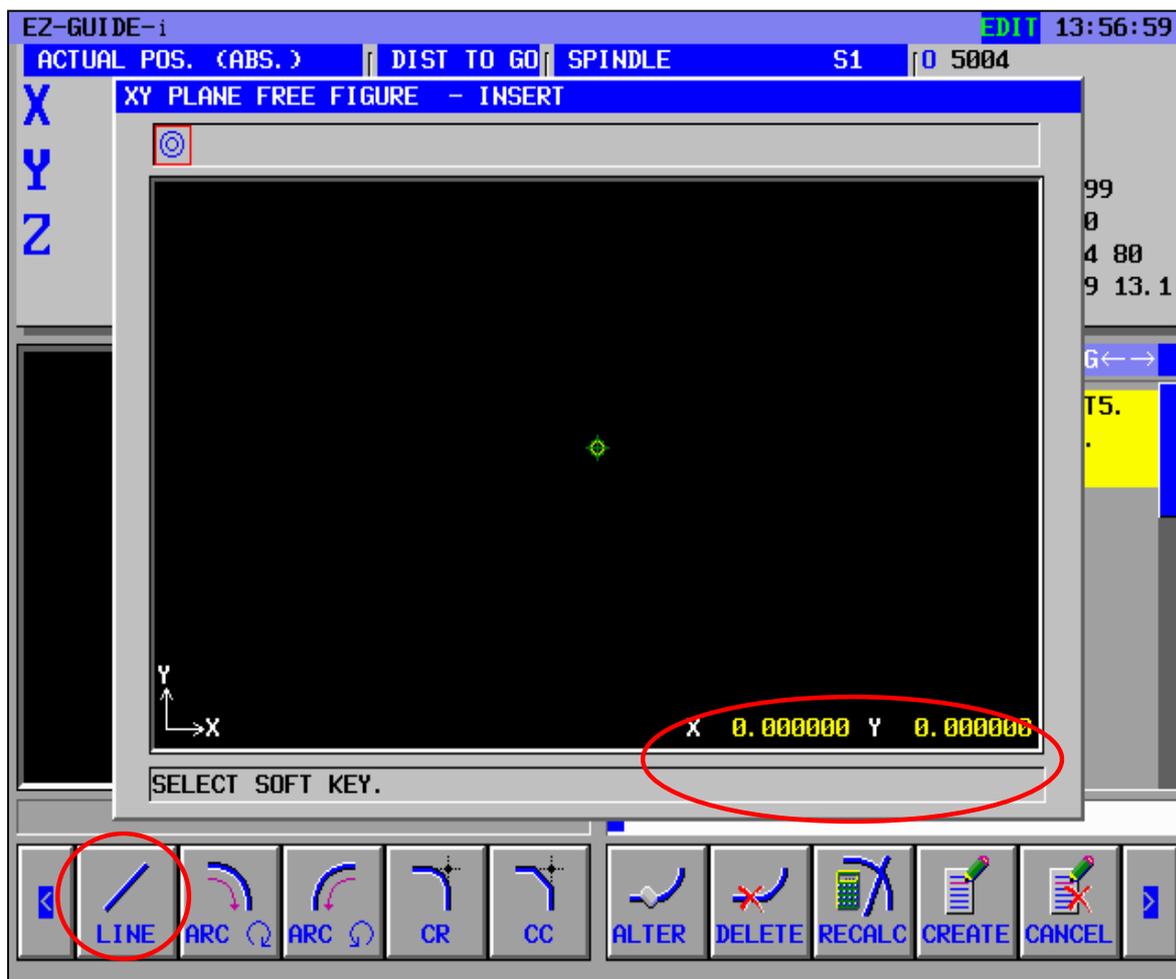
In this case, you are to follow the same procedure as shown on **Fig.18**, **Fig.19**, **Fig.20** and **Fig.30**, which display the necessary steps from cycle selection and cut condition set-up to free figure start point setting.

For a pre-check process of free shape, the operators are supposed to use **Tool Path Drawing** or **MCHDRW** function.

If you set up the cut depth of radius(L) parameter on Fig.25 as 40mm, tool path is fixed as following **Fig.32**.



<Fig.32 Actual tool path for the part of Fig.31>



<Fig.33 XY Free figure setting in Facing cycle - Start Point display>

Pushing **Line** key results in the following continuous steps(**Fig.34**).

LINE - INSERT TAB ← →

ELEMENT ATTRIBUTE

LINE DIRECTION D=RIGHT

END POINT X X=150.\*

LAST CONNECTION L=NOTHING

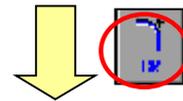
NEXT CONNECTION M=TANGENT

SELECT SOFT KEY.



XY PLANE FREE FIGURE - INSERT

SELECT SOFT KEY.



LINE - INSERT TAB ← →

ELEMENT ATTRIBUTE

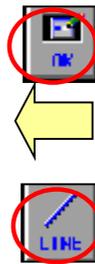
LINE DIRECTION D=DOWN

END POINT Y Y=-40.\*

LAST CONNECTION L=TANGENT

NEXT CONNECTION M=TANGENT

SELECT SOFT KEY.

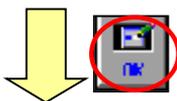


CORNER R - INSERT CHAR ← →

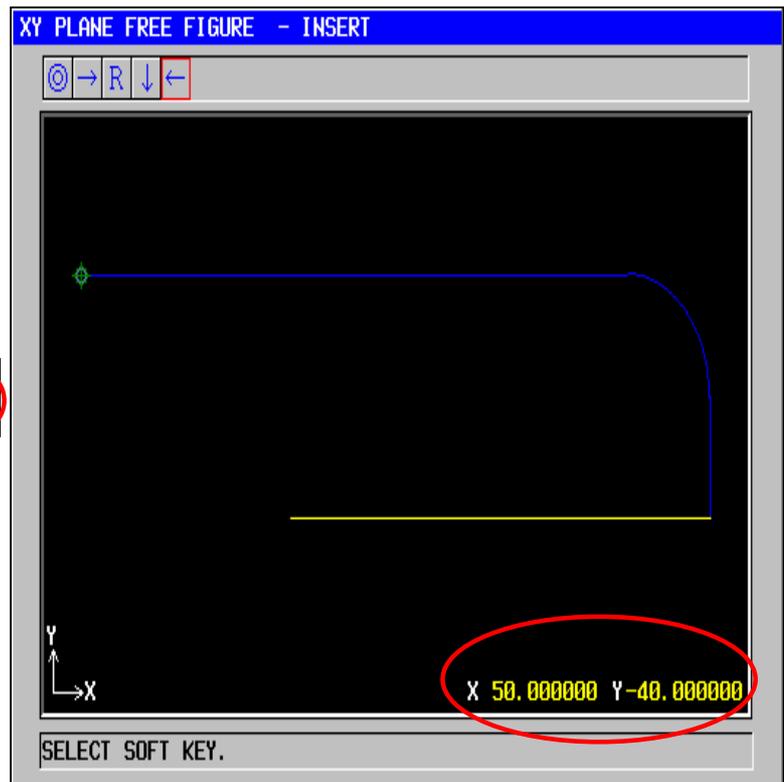
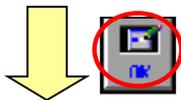
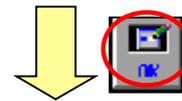
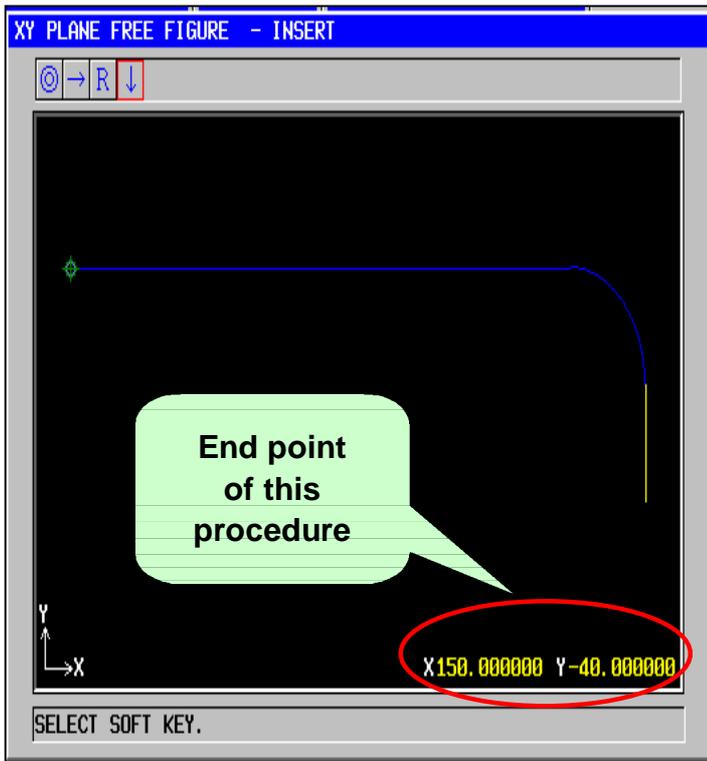
ELEMENT ATTRIBUTE

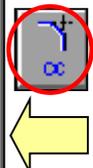
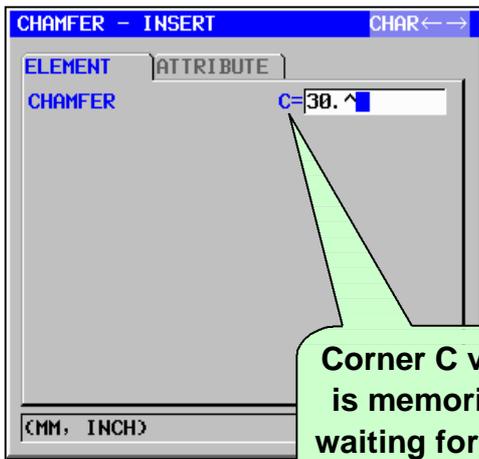
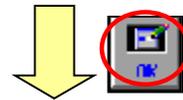
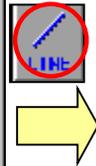
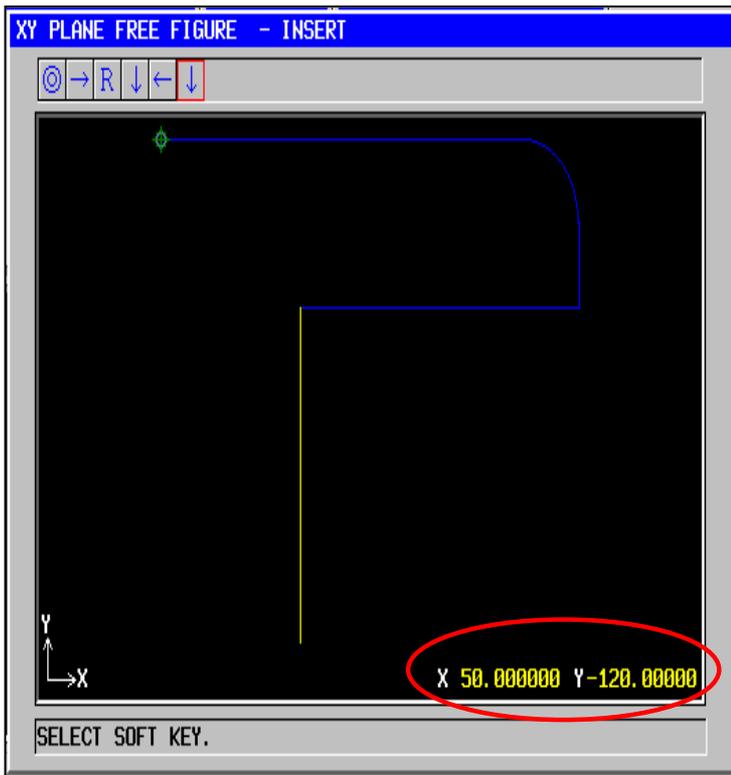
CORNER RADIUS R=20^

(MM, INCH)

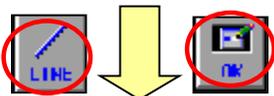
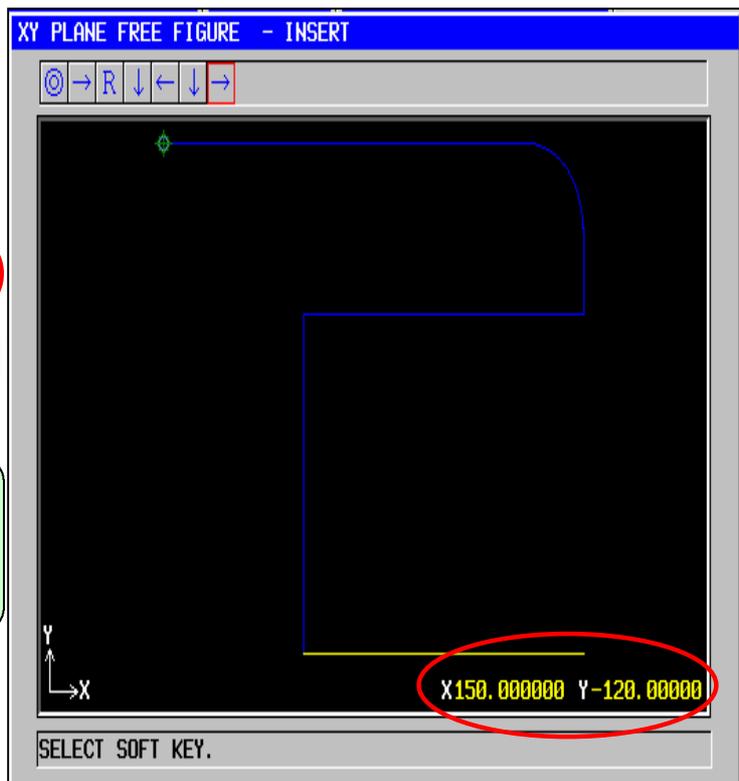


Corner R value is memorized waiting for next Line command.





Corner C value is memorized waiting for next Line command.



**LINE - INSERT** TAB ←→

ELEMENT ATTRIBUTE

LINE DIRECTION D=DOWN

END POINT Y Y=-160.\*

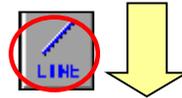
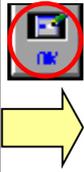
NEXT CONNECTION M=TANGENT

SELECT SOFT KEY.

**XY PLANE FREE FIGURE - INSERT**

⊙ → R ↓ ← ↓ → C ↓

SELECT SOFT KEY.



**XY PLANE FREE FIGURE - INSERT**

⊙ → R ↓ ← ↓ → C ↓ ←

SELECT SOFT KEY.



**LINE - INSERT** TAB ←→

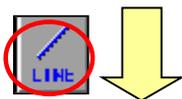
ELEMENT ATTRIBUTE

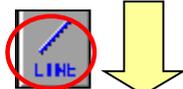
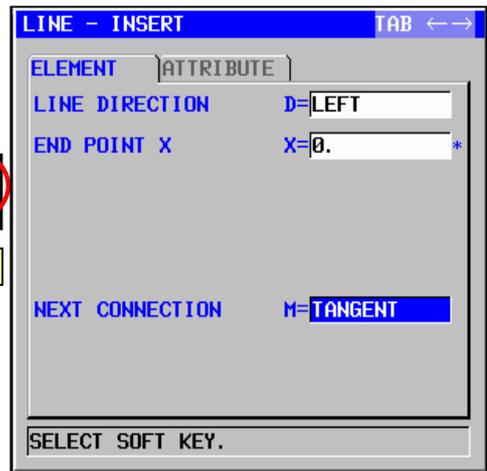
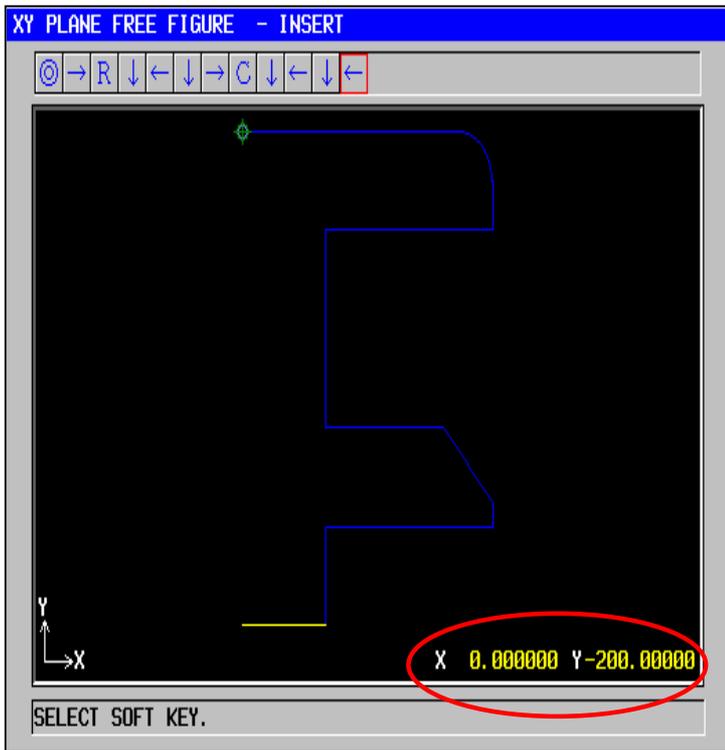
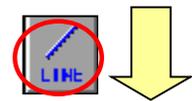
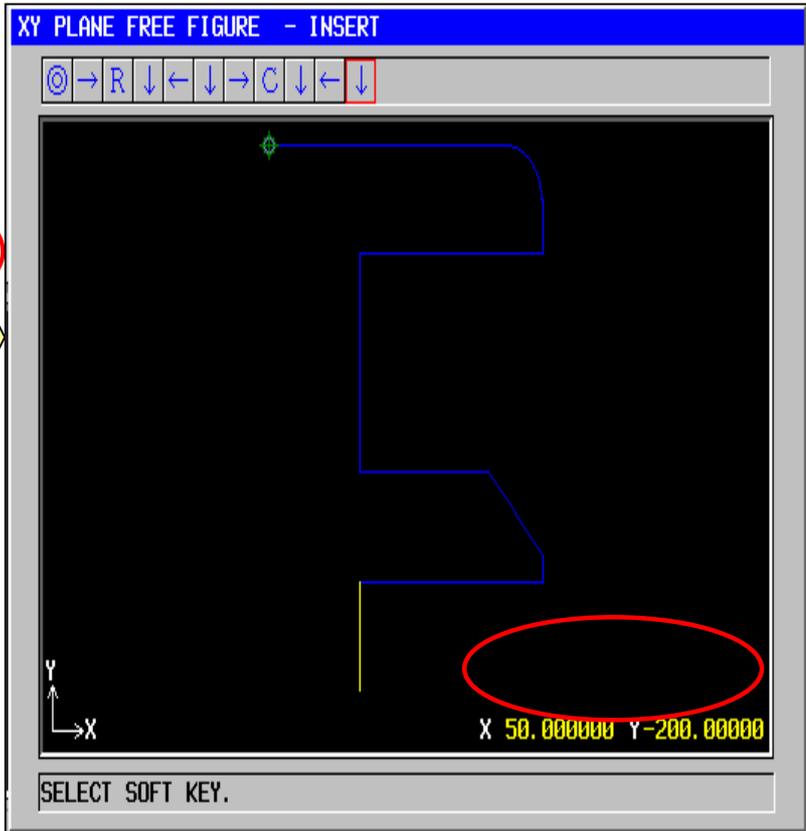
LINE DIRECTION D=LEFT

END POINT X X=50.\*

NEXT CONNECTION M=TANGENT

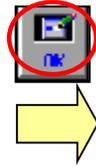
SELECT SOFT KEY.





ELEMENT	ATTRIBUTE
LINE DIRECTION	D=UP
END POINT Y	Y=0.
NEXT CONNECTION	N=TANGENT

SELECT SOFT KEY.

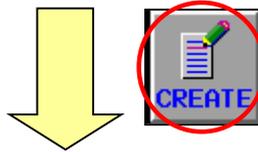


XY PLANE FREE FIGURE - INSERT

⊙ → R ↓ ← ↓ → C ↓ ← ↓ ← ↑

X 0.000000 Y 0.000000

SELECT SOFT KEY.



METHOD OF FREE FIGURE CREATION

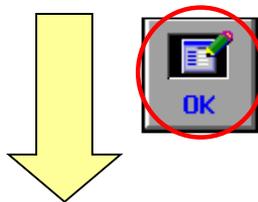
INSERT IN CURRENT PROGRAM

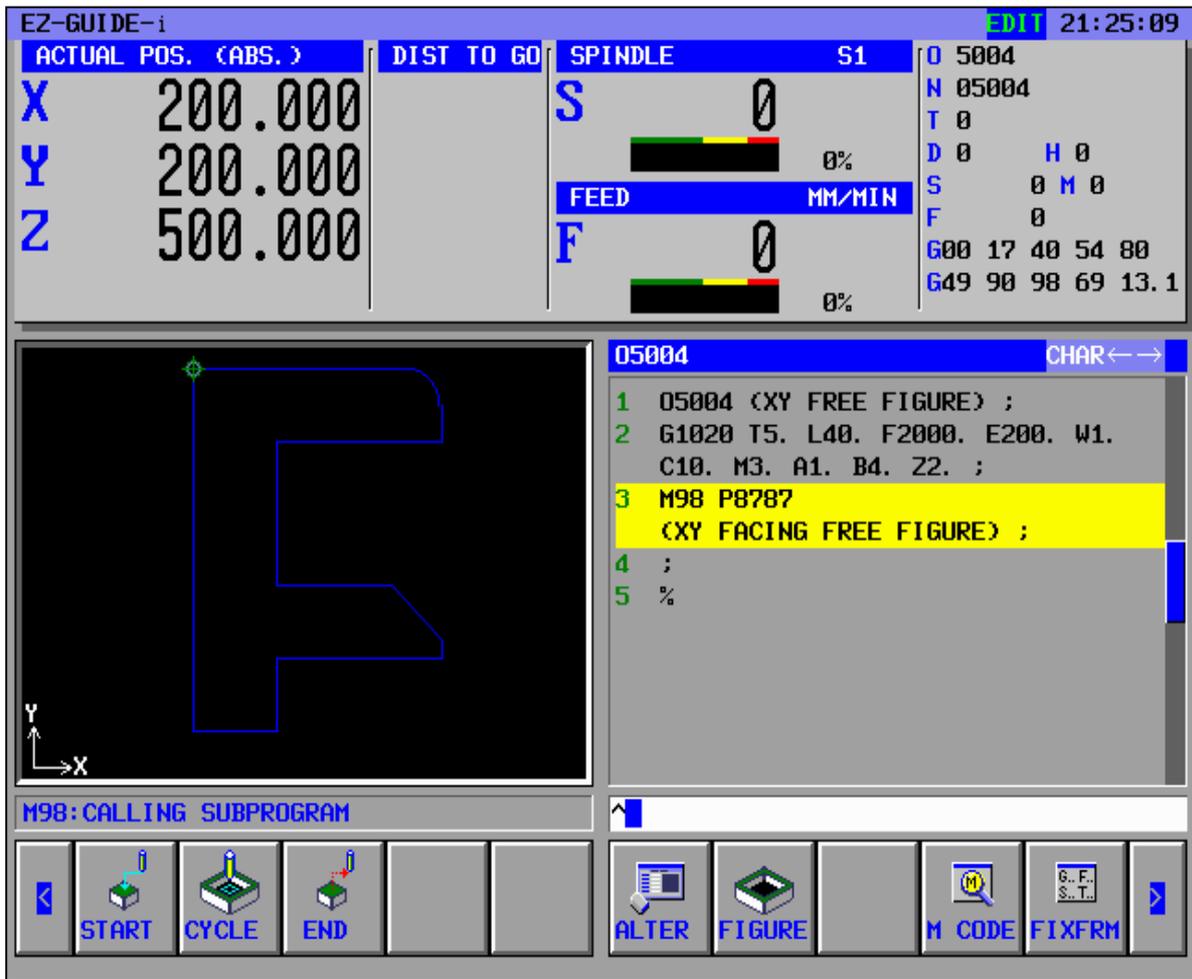
CREATE AS SUB PROGRAM

SUBPRO NO. : 8787

FIGURE NAME: XY FACING FREG FIGURE^

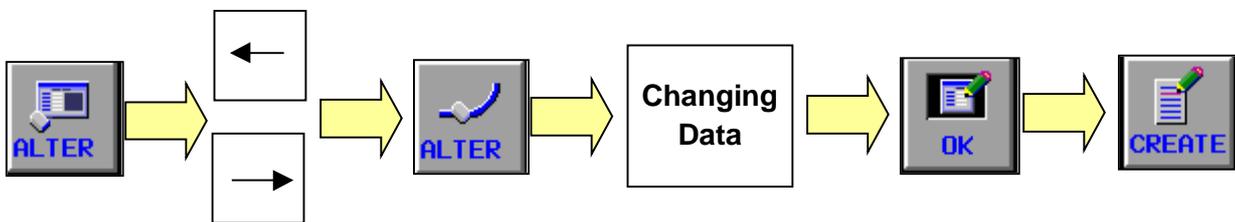
INPUT SUBPRO NO. AND FIGURE NAME.



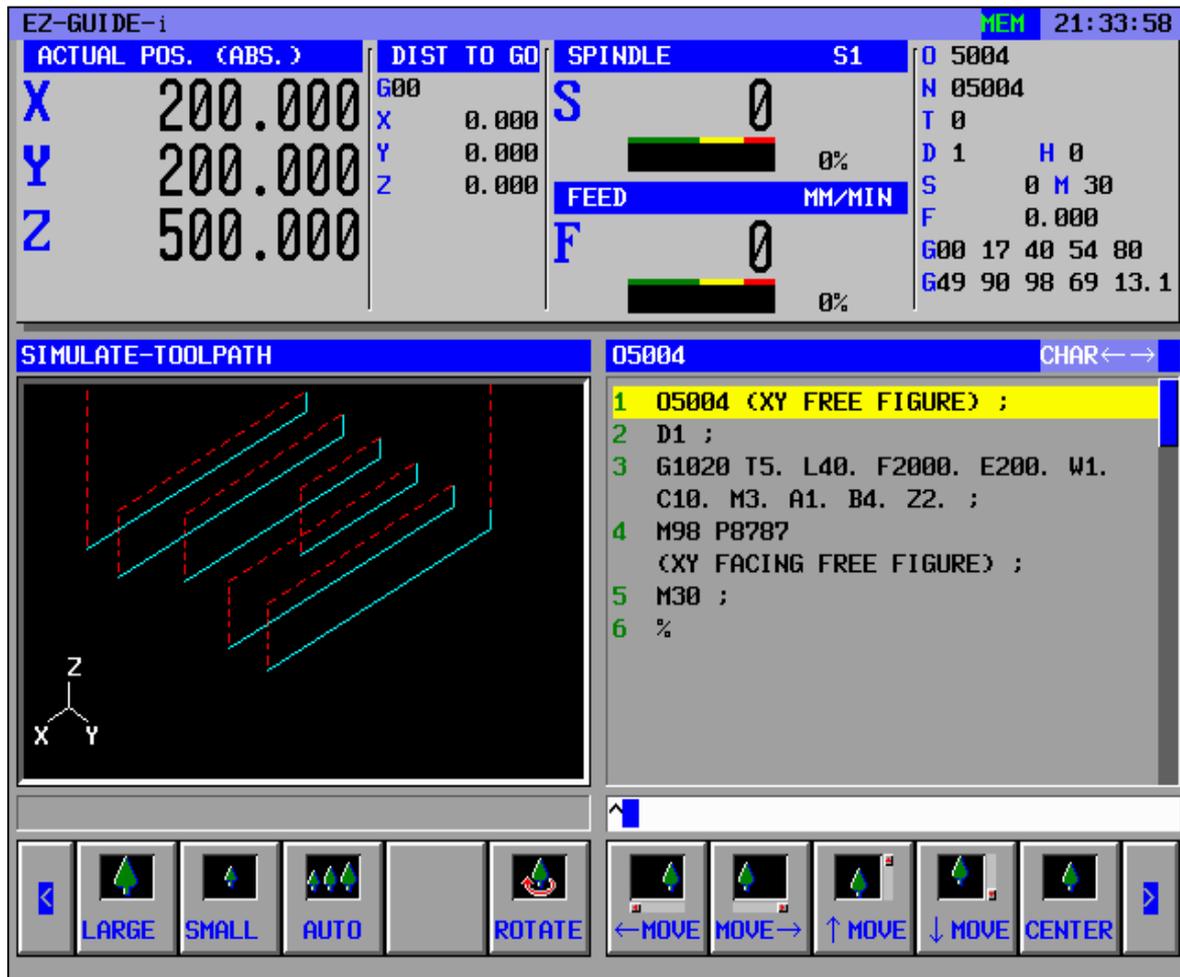


<Fig.34 Continuous flow of XY Free figure set-up in Facing cycle>

When you need to modify any figure element, press **Alter** and cursor as follows ;



<Fig.35 Alternation of free figure element>



<Fig.36 Tool Path Drawing pre-check >



Figure	Remarks
Random Points	-.General random positioning is performed according to data input.
Linear Points	-.Points in a line with same or different interval -.Omitting any points is allowed.
Grid Points	-.Points in a grid -.Omitting any points is allowed.
Rectangle Points	-.Points in a rectangular line -.Omitting any points is allowed.
Circle Points	-.Points in a circle line -.Omitting any points is allowed.
Arc Points	-.Points in an arc with same or different interval -.Omitting any points is allowed.

# 3. Grooving

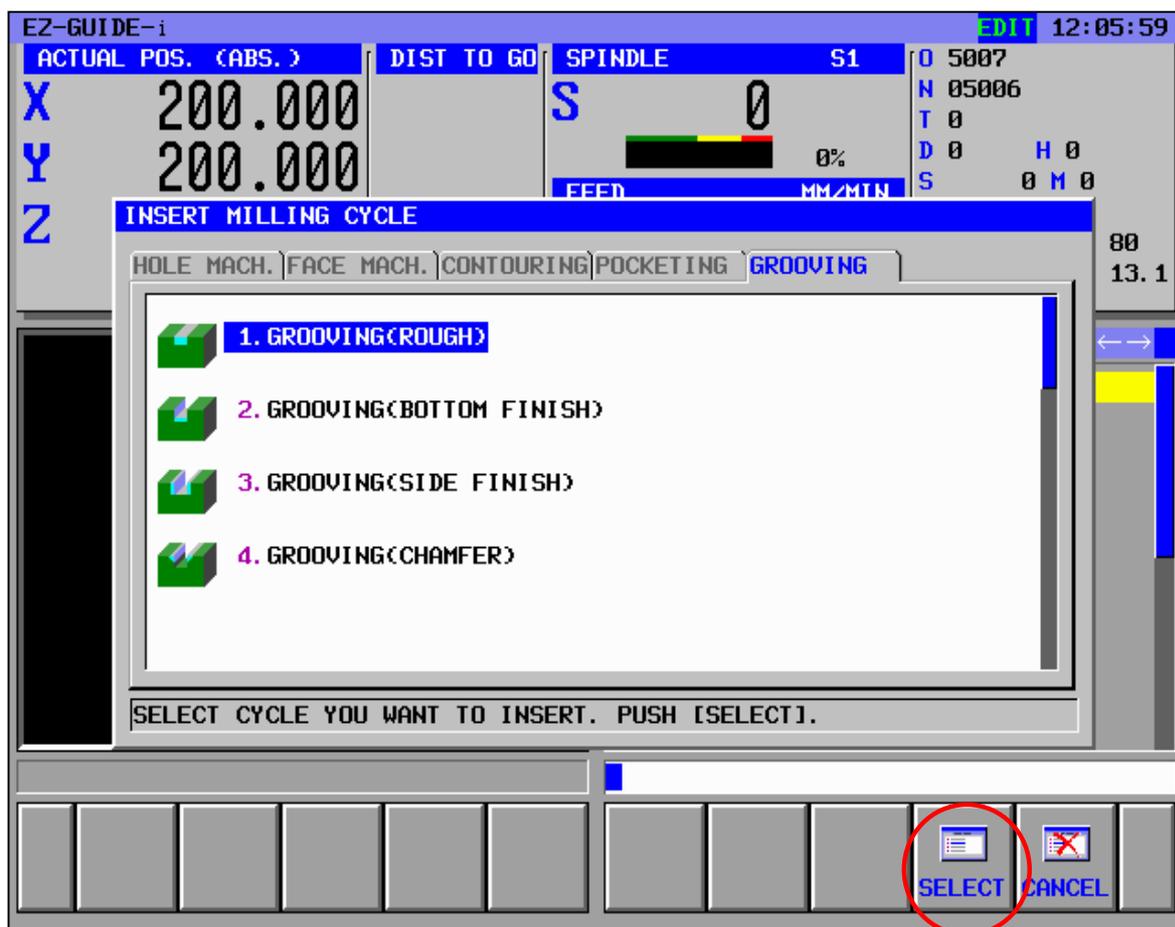
## 3.1 Main feature of Grooving cycle

By using this cycle, the operator can carry out any routine making of key slot or groove.

For definition of cycle, you are supposed to set cutting condition including cut depth, finish amount and feedrate.

With regard to figure selection, **Grooving** cycle supplies 4 fixed (**Square**, **Circle**, **Track** and **Radiate** groove) and **Free Figure**.

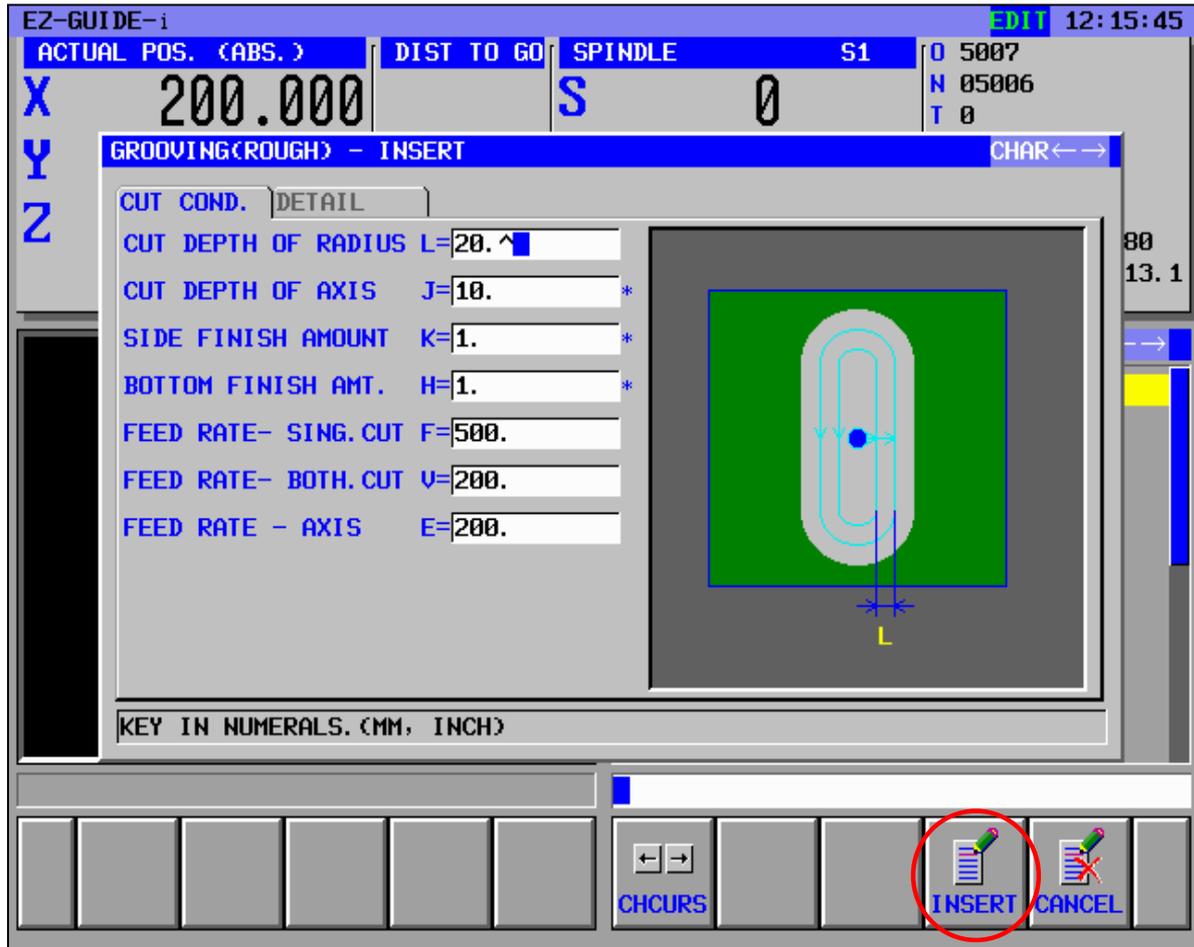
Among them, the start and end point of Square, Circle and Track is always same. In figure set-up, depth, width and length of the groove should be specified.



<Fig.37 Track cycle configuration screen >

### 3.2 Cutting condition set-up in Grooving cycle

Pushing **Select** key in *Fig.37* results in the following screen.

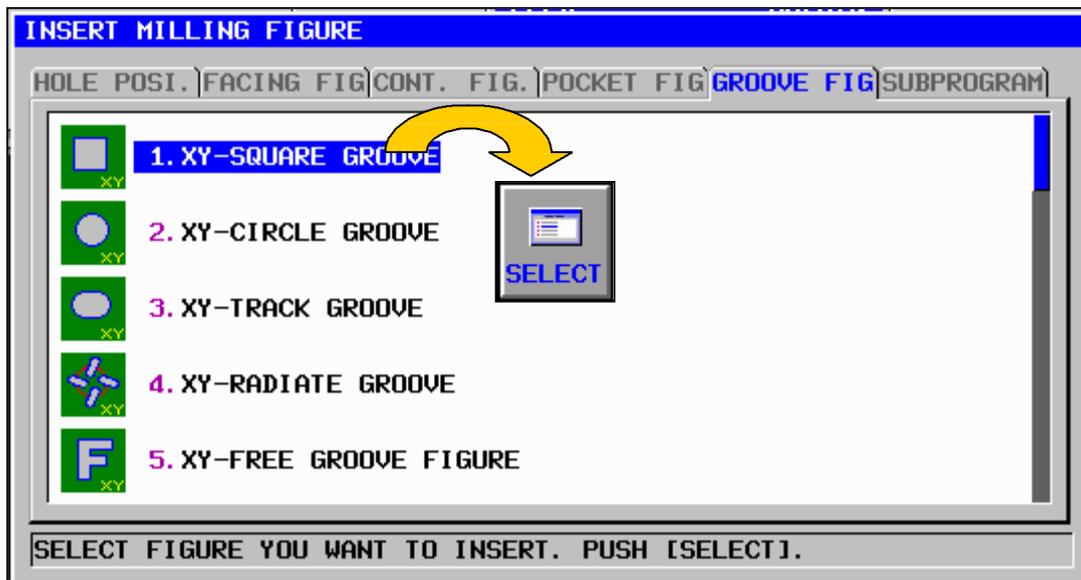


<Fig.38 Cutting condition set-up screen >

Item		Remarks
Cut Depth of Radius	Common	-.Take the bottom surface roughness into consideration regarding overlap lines.
Side Finish Amount	Bottom Finish	-.In setting bottom finish grooving, let the side finish amount remained for final side process.
Feedrate-Both.Cut	Common	-.Take the material hardness and tool spec. into consideration to prevent damage on tip.

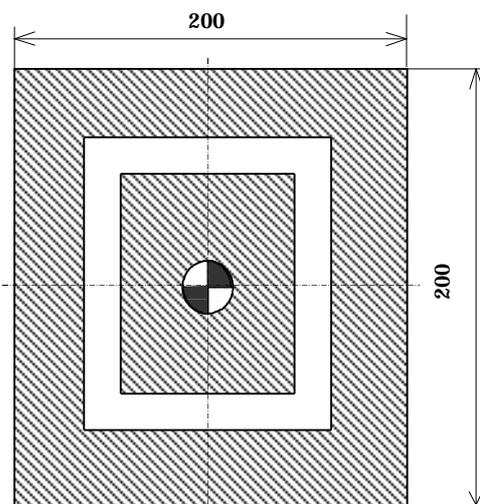
### 3.3 Setting parameters for *Grooving* cycle

As mentioned several times, entering a cycle cut condition displays continuously the following figure selection menu to be used exclusively for the already chosen **Grooving** cycle machining type.



<Fig.39 Track cycle figure selecting screen >

For operator's proper understanding, a square type groove example on rectangular piped work (**Fig. 40**) is to be explained.



**Width of Groove : 25mm**

**Depth of Groove : 20mm**

**Length of Groove in X / Y axis : 120mm**

▶ **Tool Center Distance**

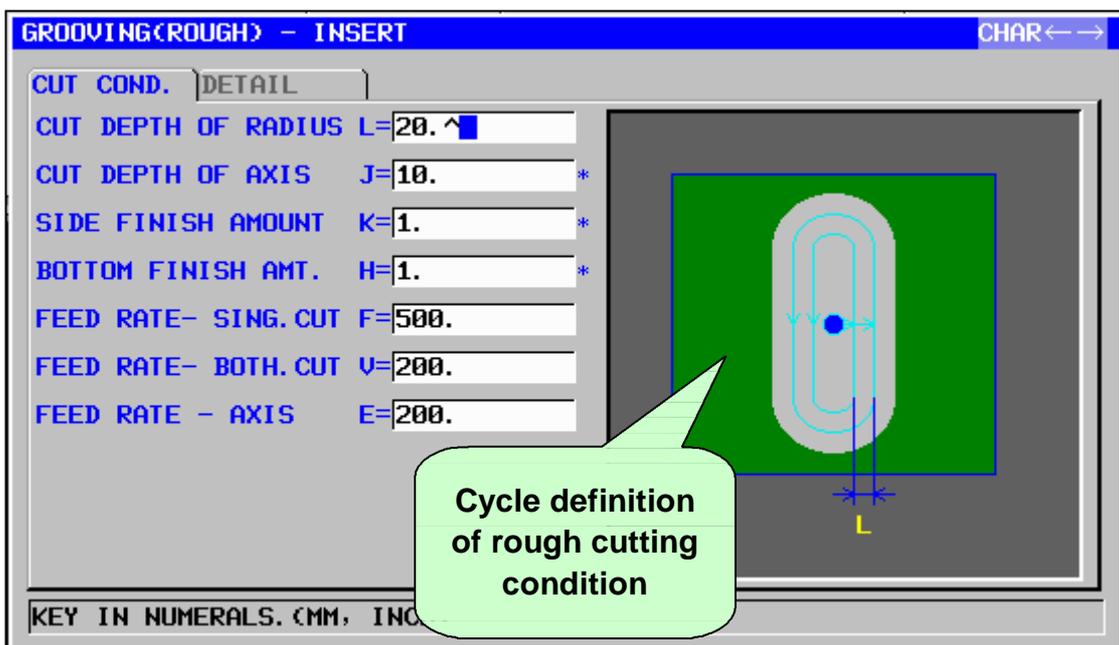
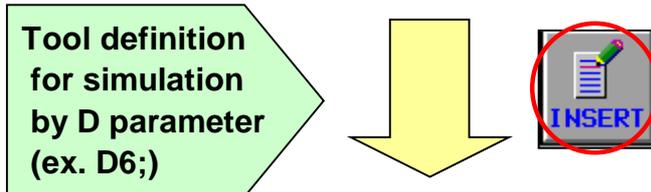
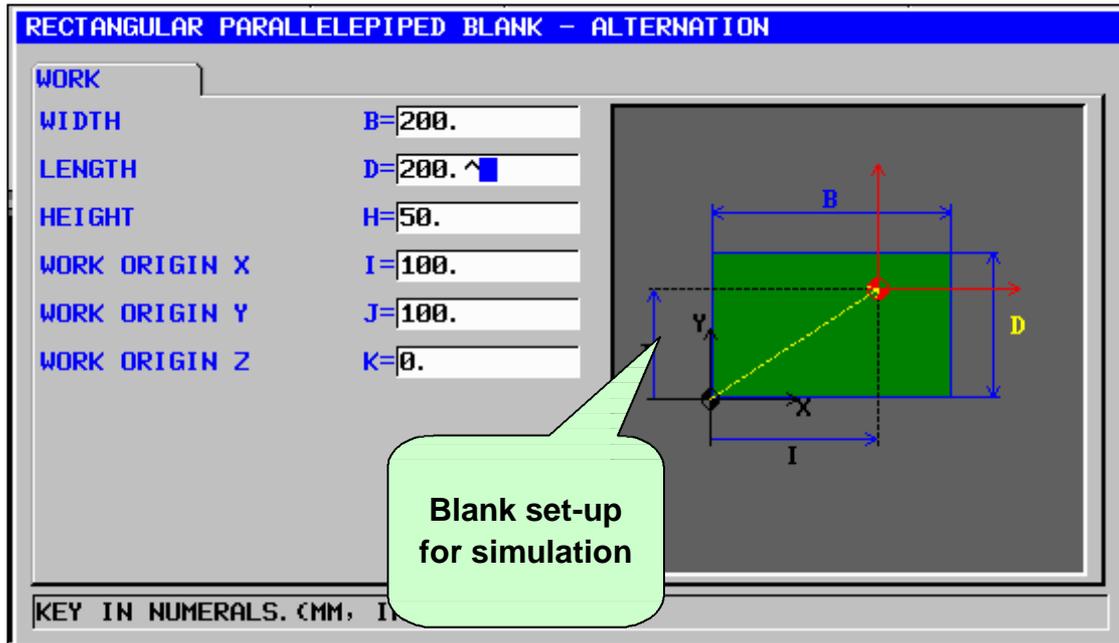
**Z Coordinate of Top Surface : 0**

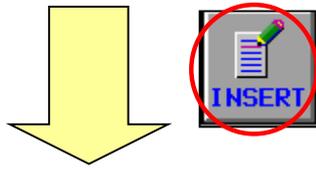
**Rough & Bottom Fin. Endmill = D20 / 2flute**

**Side Fin. Endmill = D10 / 4flute**

<Fig.40 Square Groove example - Dimension & cut condition>

**Fig.41** shows how to make the specified groove above according to the designated process.



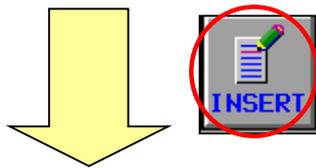


**XY-SQUARE - INSERT**

POS. / SIZE	
FIGURE TYPE	T= <b>GROOVE</b>
BASE POSITION	B= 0.
HEIGHT / DEPTH	L= -20.
CENTER POINT (X)	H= 0.
CENTER POINT (Y)	V= 0.
LENGTH FOR X AXIS	U= 120.
LENGTH FOR Y AXIS	W= 120.
CORNER RADIUS	R=
ANGLE	A=
GROOVE WIDTH	D= 25.

SELECT SOFT KEY.

**Configuration of groove figure**



**METHOD OF FIGURE PROGRAM CREATION**

INSERT IN CURRENT PROGRAM

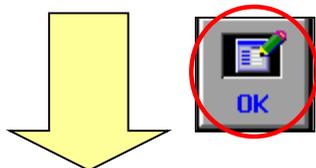
**CREATE AS SUB PROGRAM**

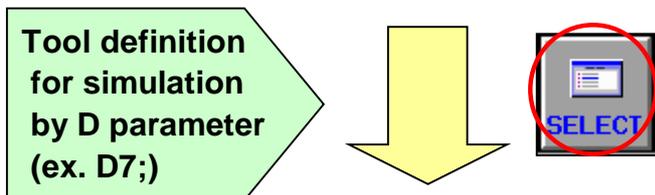
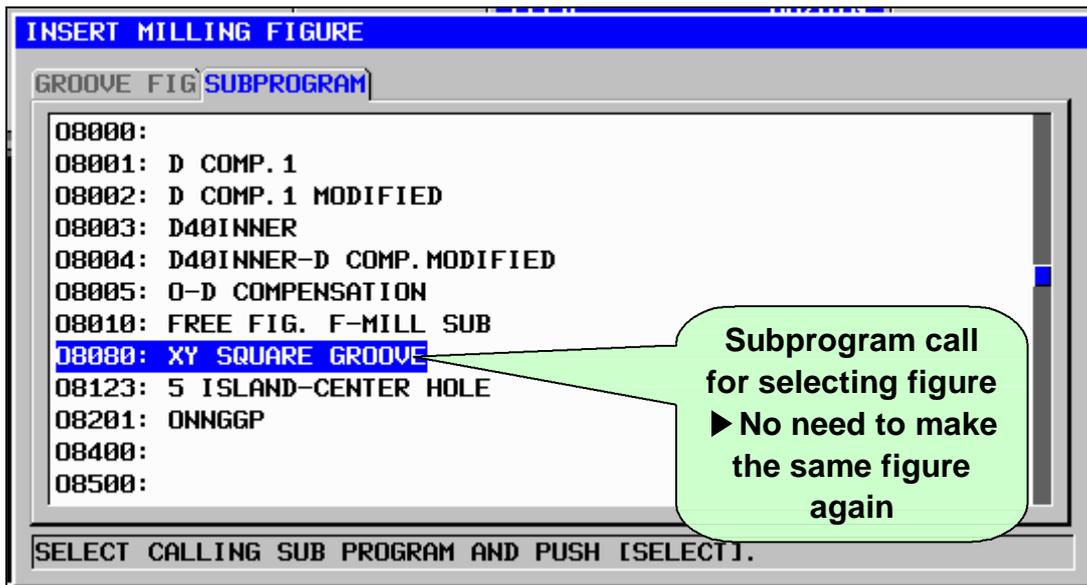
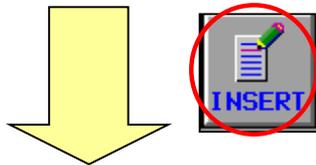
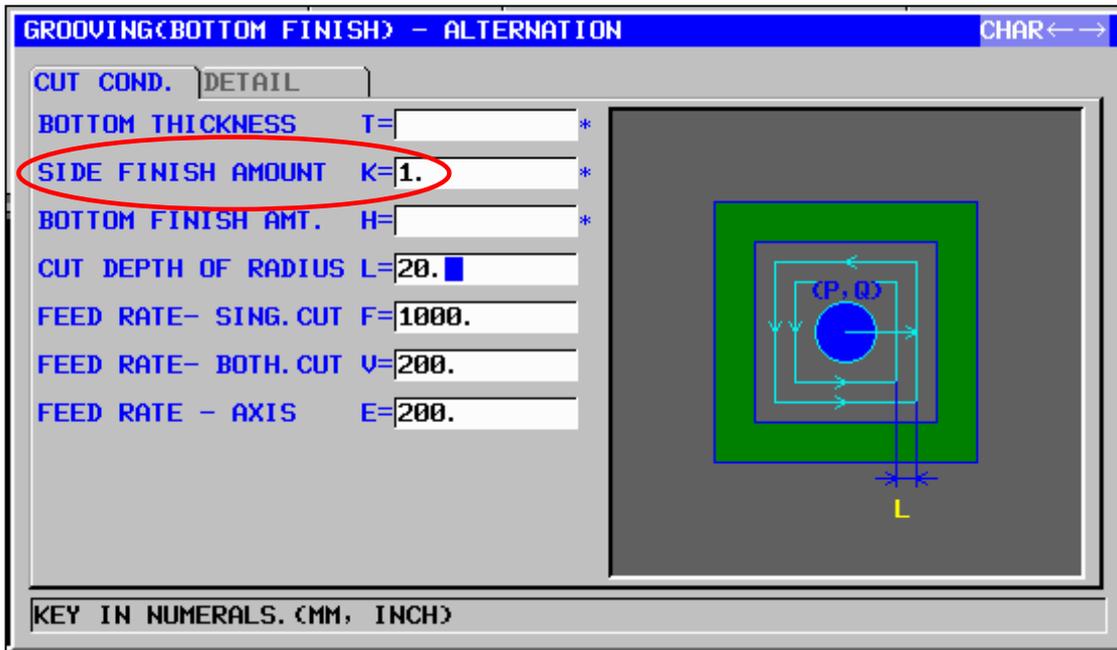
SUBPRO NO. : 8080

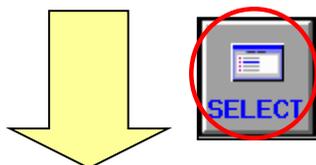
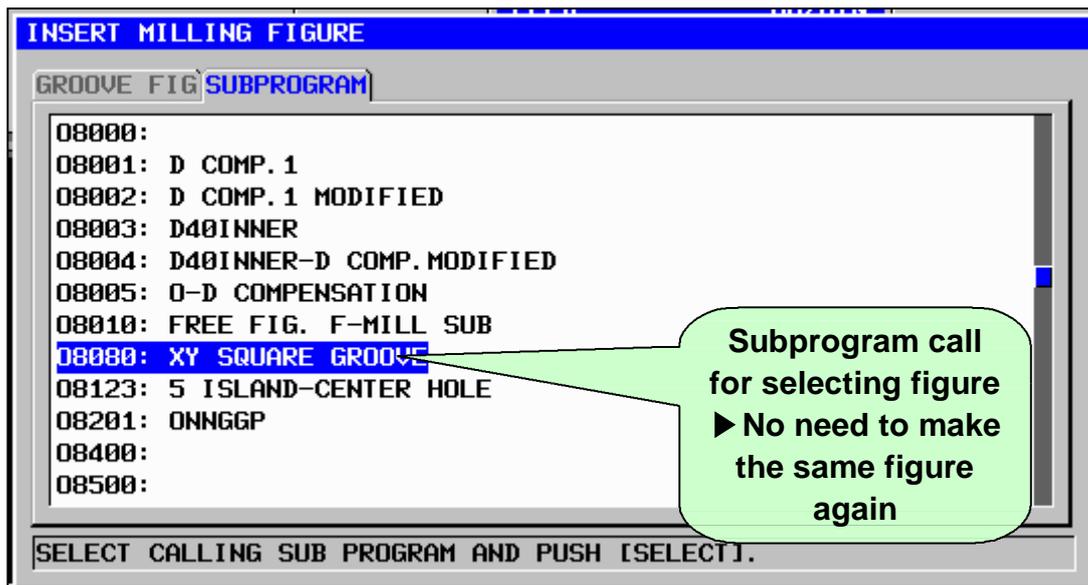
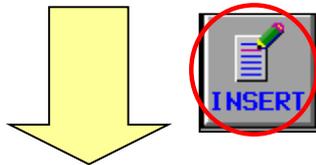
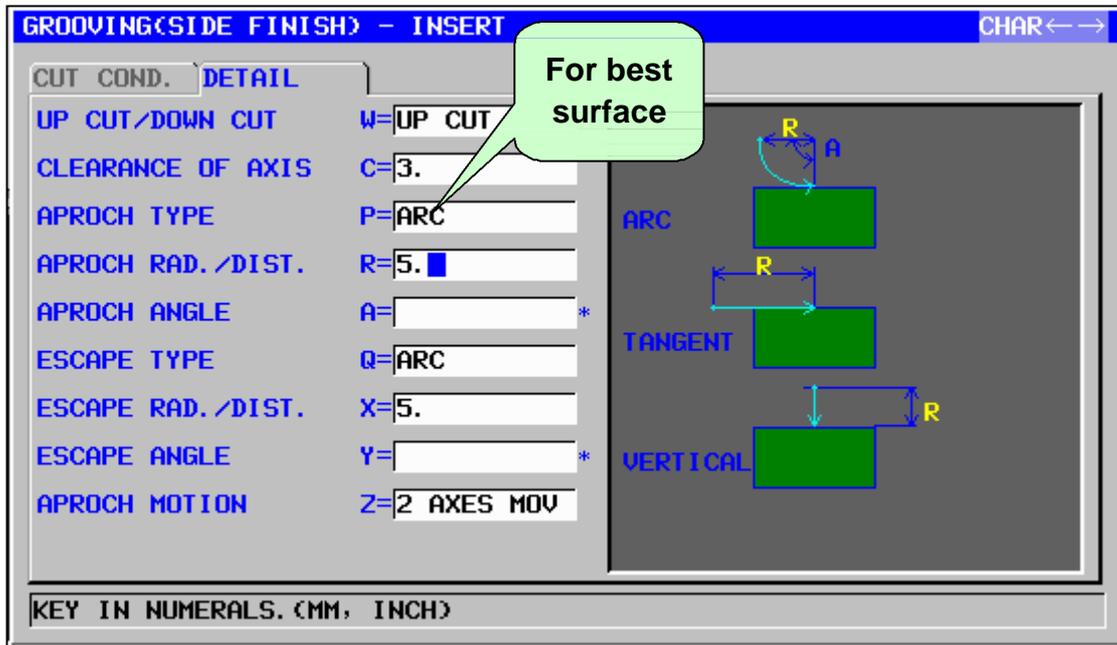
COMMENT : XY SQUARE GROOVE^

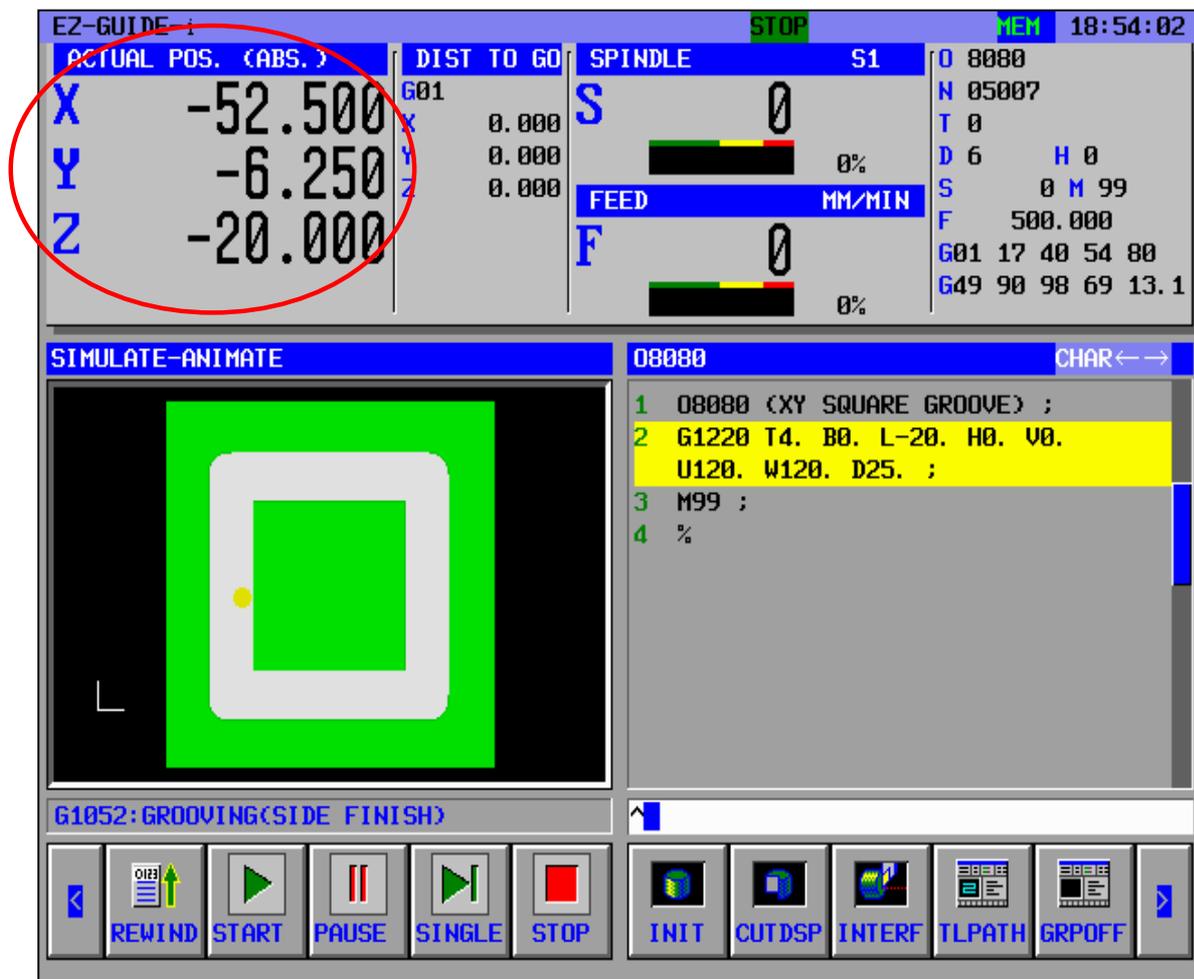
INPUT SUBPRO NO. AND COMMENT.

**Storage of figure as subprogram**









<Fig.41 Operating process for Groove cycle - Rough/Bottom Fin./Side Fin.>

The final step before actual machining is to check by simulation.

The above example shows side finish tool(D10 flat endmill) movement in XY plane.

In pre-check process, please make sure if the actual position varies correctly from the point of tool path's view.

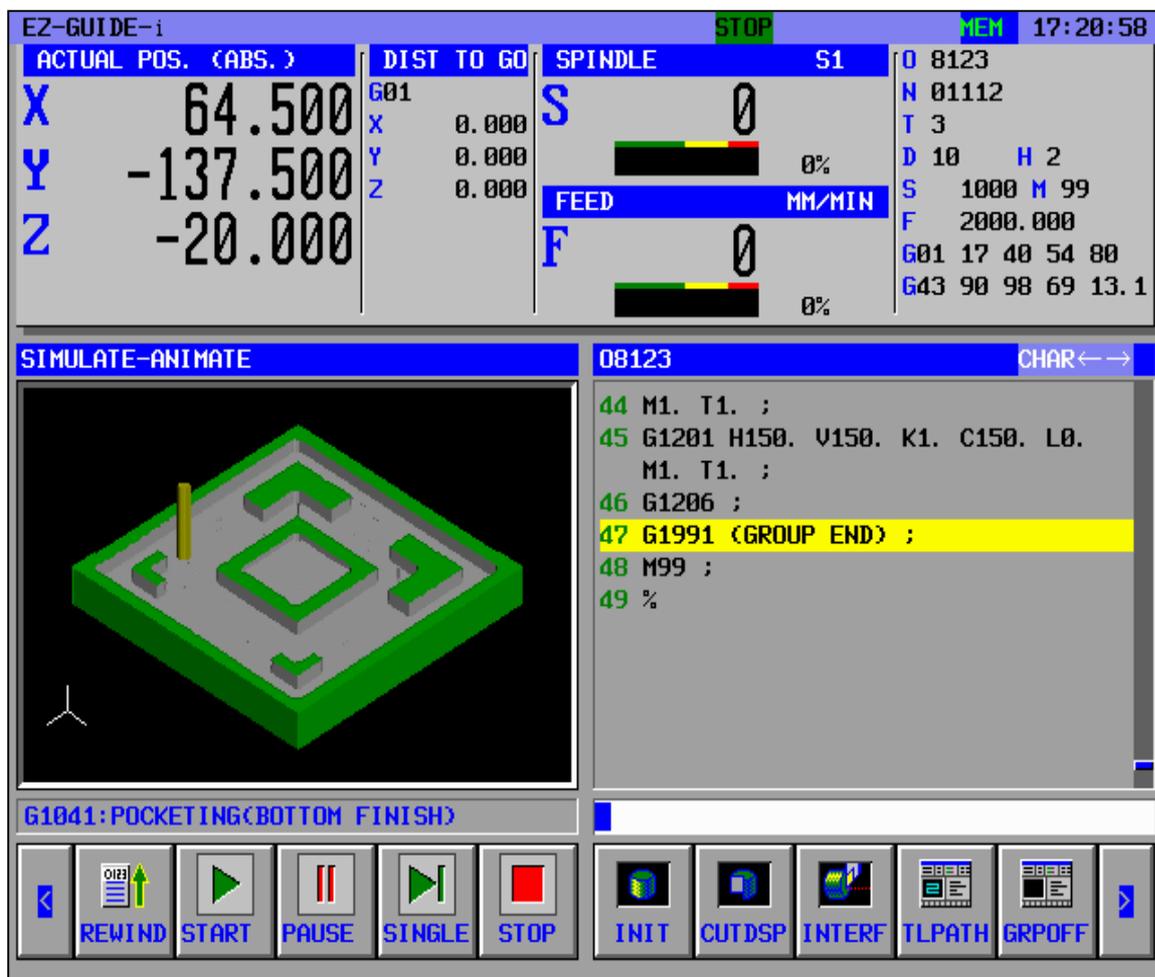
# 4. Pocketing

## 4.1 Main feature of *Pocketing* cycle

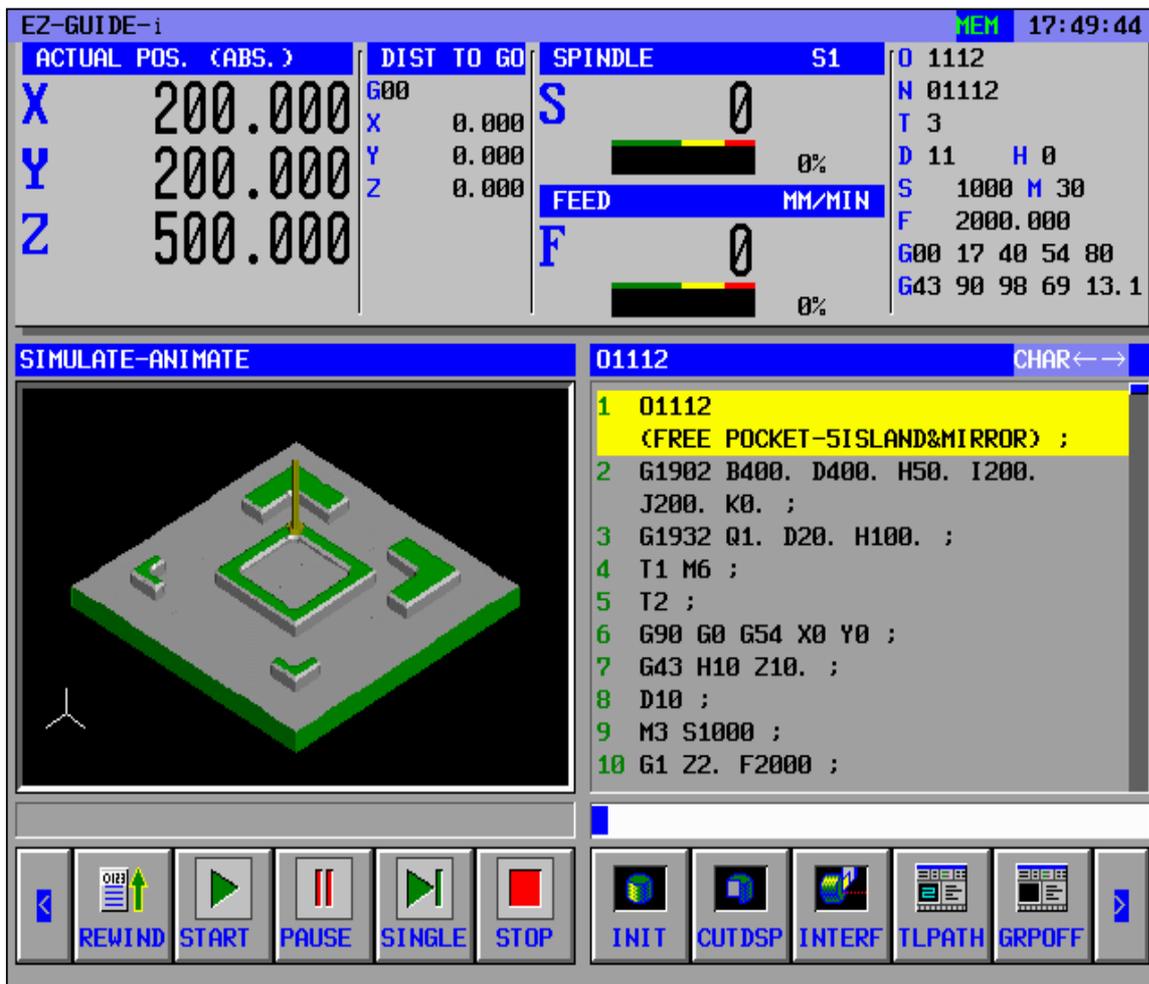
In case a pocket is programmed on any work, normally lots of tool path blocks are necessary for removing whole material to a designed depth and width .

In addition, a pocket with islands requests much more careful consideration rather than simple one.

**EZ Guide i** helps a operator save time in making pockets by supplying reasonably short and easy steps for programming.



<Fig.42 Pocketing simulation example - Part method>



<Fig.43 Pocketing simulation example - Blank method>

**Fig.42 & Fig.43** may convince you of the difference between **Part** and **Blank** method in making a pocket with several islands. It is the key point whether the outer wall remains or not.

The only way to machine a pocket which has some islands and no surrounding wall is the **Blank** method.

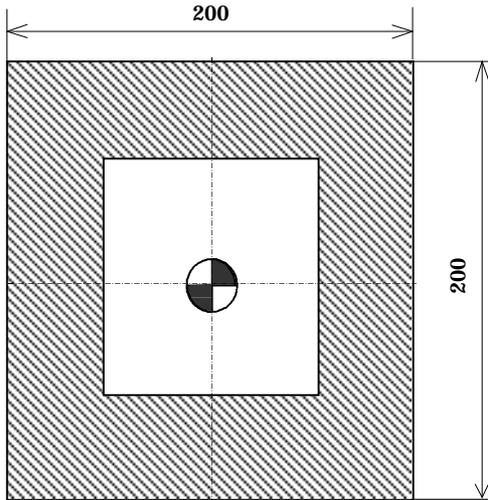
Take the final figure into consideration then choose suitable way when forming the outline of a free figure pocket.

Regarding normal shaped pockets, however, there is no need to be troubled by this step, for the default setting(**Part**) can cover them.

Refer to the 4.3 **Free Figure Pocketing** for detail information.

## 4.2 Setting parameters for *Pocketing* cycle

For operator's proper understanding, a square type pocket example on rectangular piped work (**Fig. 44**) is to be explained on **Fig.45**.



**Width of Pocket : 100\*100mm**

**Depth of Pocket : 20mm**

**Z Coordinate of Top Surface : 0**

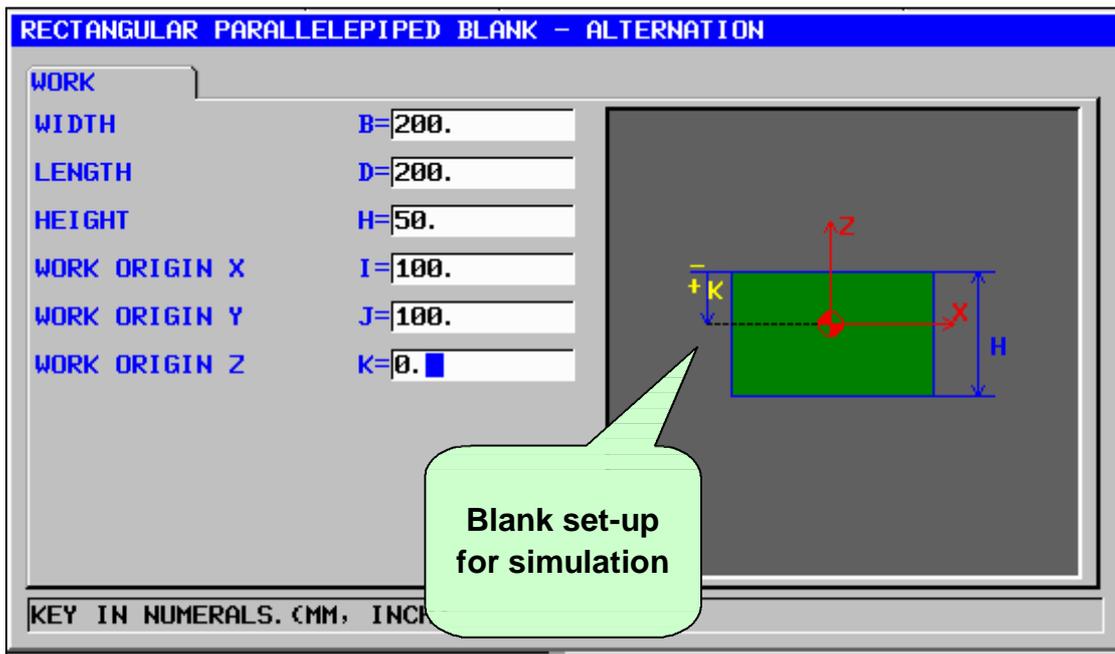
**Rough Endmill = D20 / 2flute**

**Bottom Fin. Endmill = D10 / 3flute**

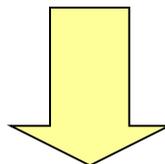
**Side Fin. Endmill = D5 / 4flute**

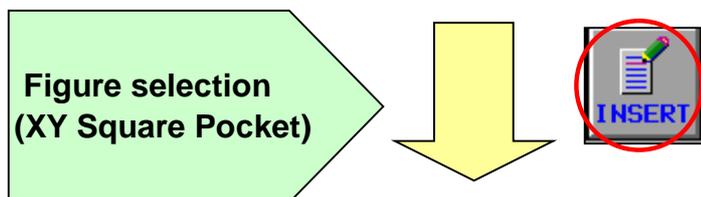
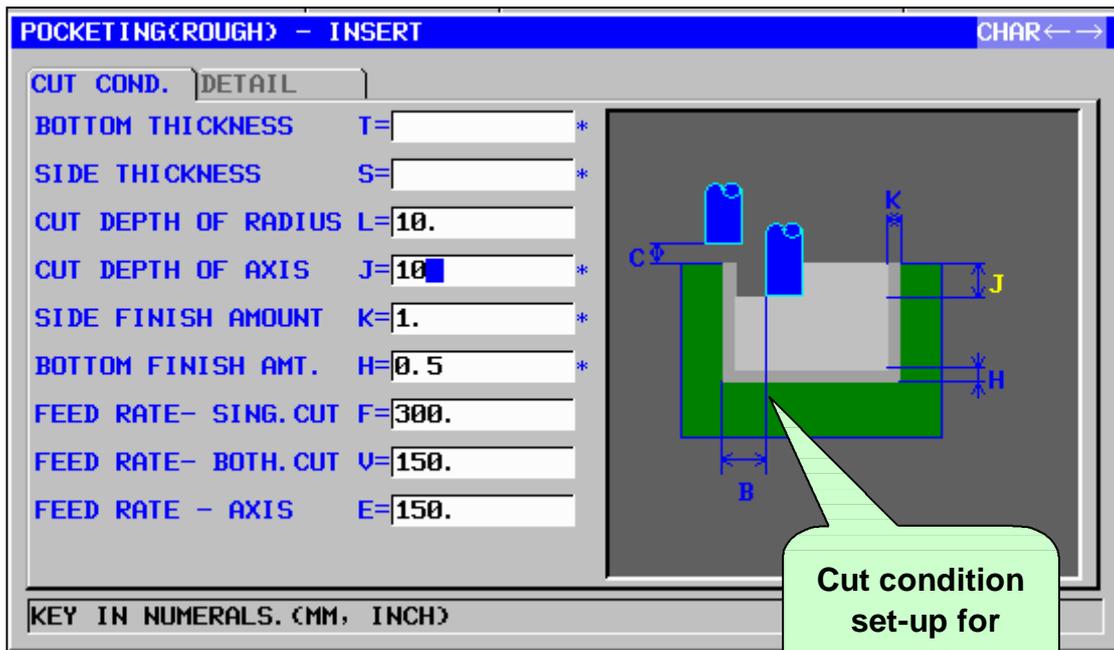
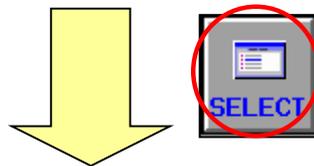
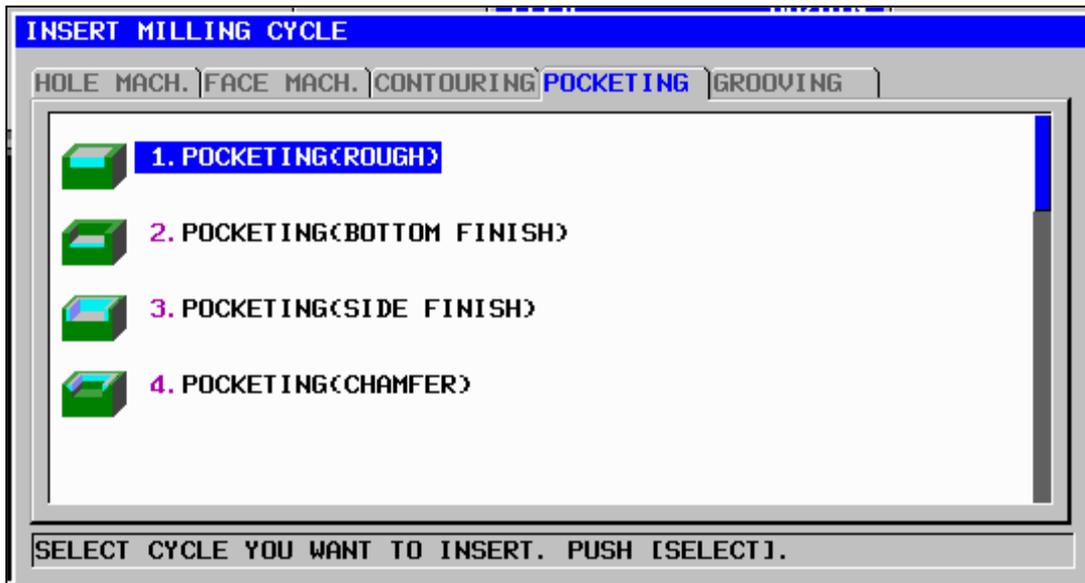
► **So final r at each corner is 2.5mm.**

<Fig.44 Square Pocket example - Dimension & cut condition>



Tool definition  
for simulation  
by D parameter  
(ex. D9;)





**XY-SQUARE - INSERT**

POS./SIZE

FIGURE TYPE T=CONCAV

BASE POSITION B=0.

HEIGHT/DEPTH L=-20.

CENTER POINT(X) H=0.

CENTER POINT(Y) V=0.

LENGTH FOR X AXIS U=100.

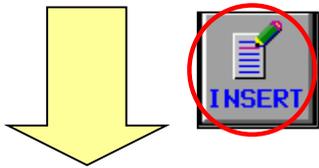
LENGTH FOR Y AXIS W=100.

CORNER RADIUS R= [ ] \*

ANGLE A= [ ] \*

UNITS: (MM, INCH)

Figure set-up for roughing



**METHOD OF FIGURE PROGRAM CREATION**

INSERT IN CURRENT PROGRAM

CREATE AS SUB PROGRAM

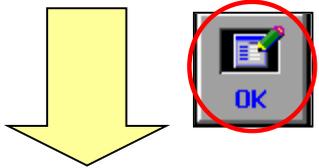
SUBPRO NO. : 8082

COMMENT : XY SQUARE POCKET^

INPUT SUBPRO NO. AND COMMENT.

Storage of figure as subprogram

Tool definition for simulation by D parameter (ex. D9;)



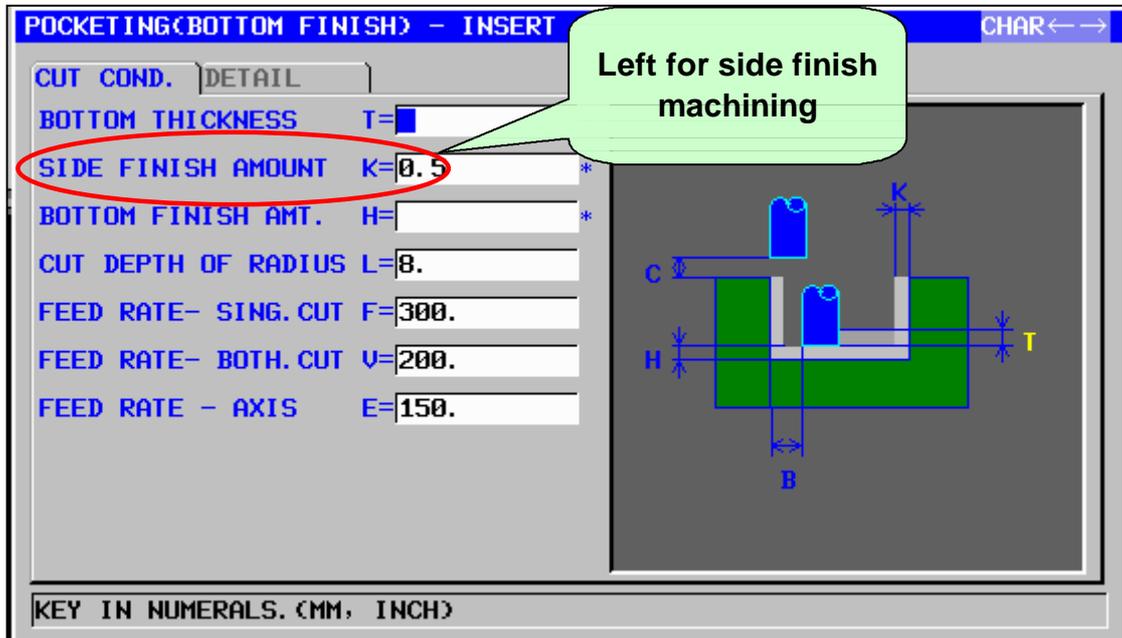
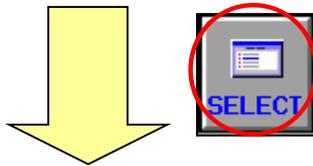
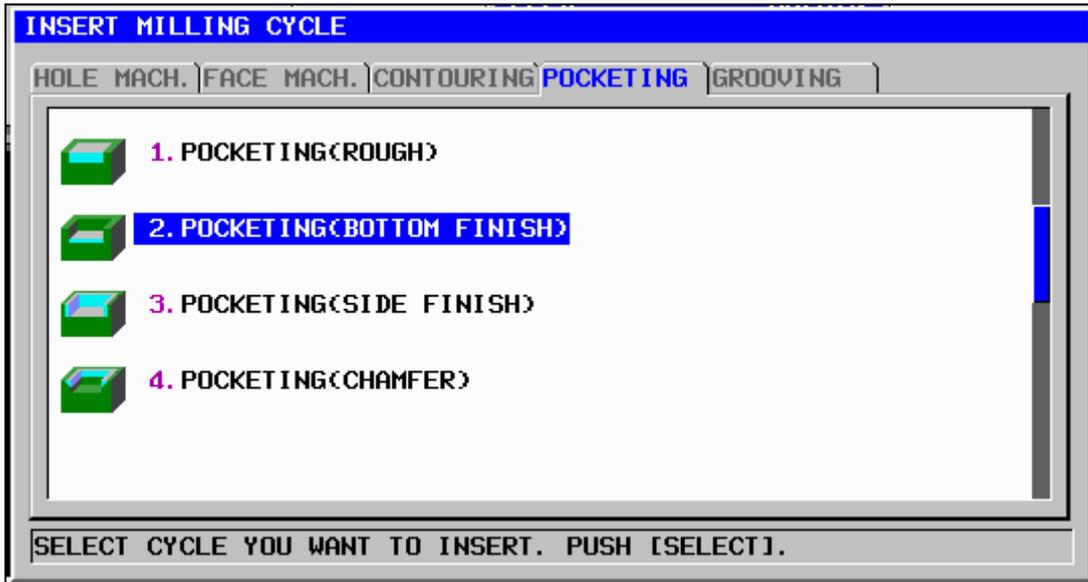
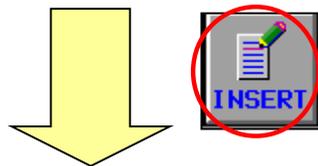


Figure selection from Subprogram directory

Tool definition for simulation by D parameter (ex. D10;)



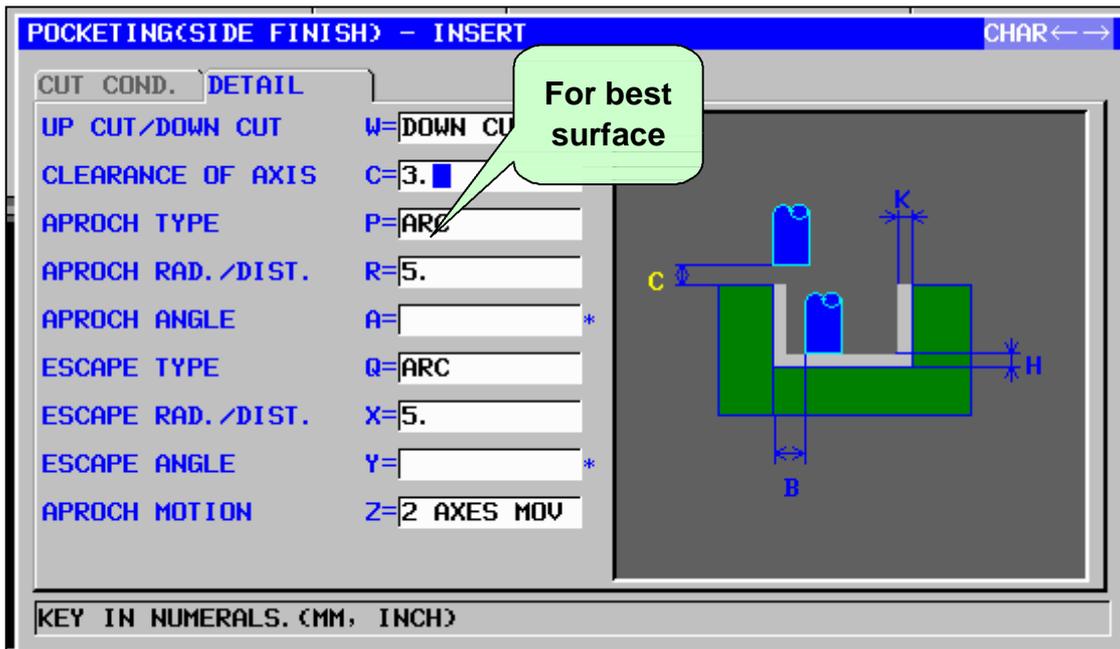
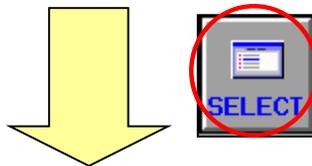
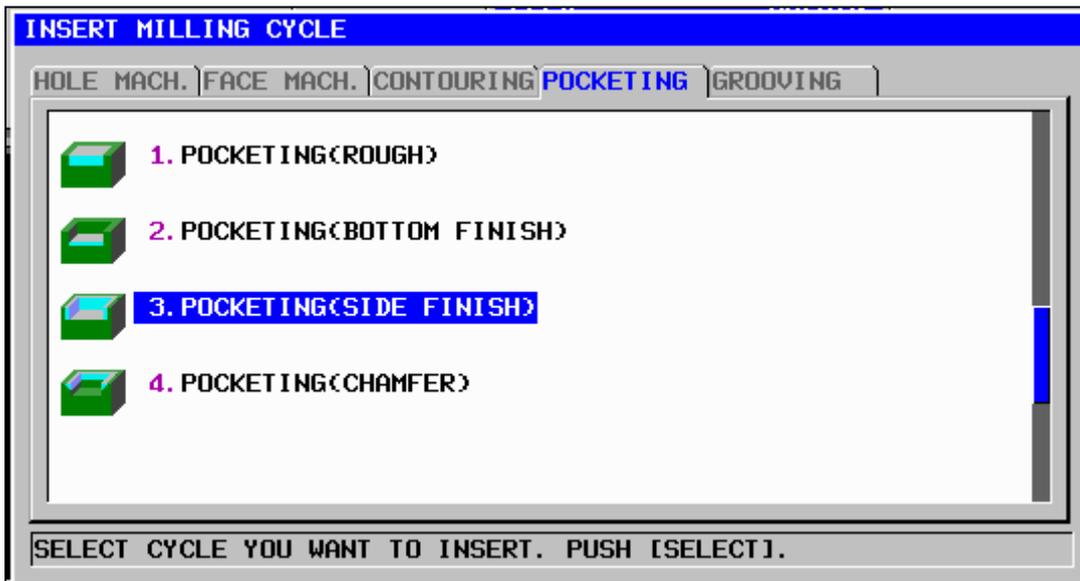
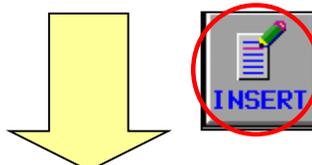
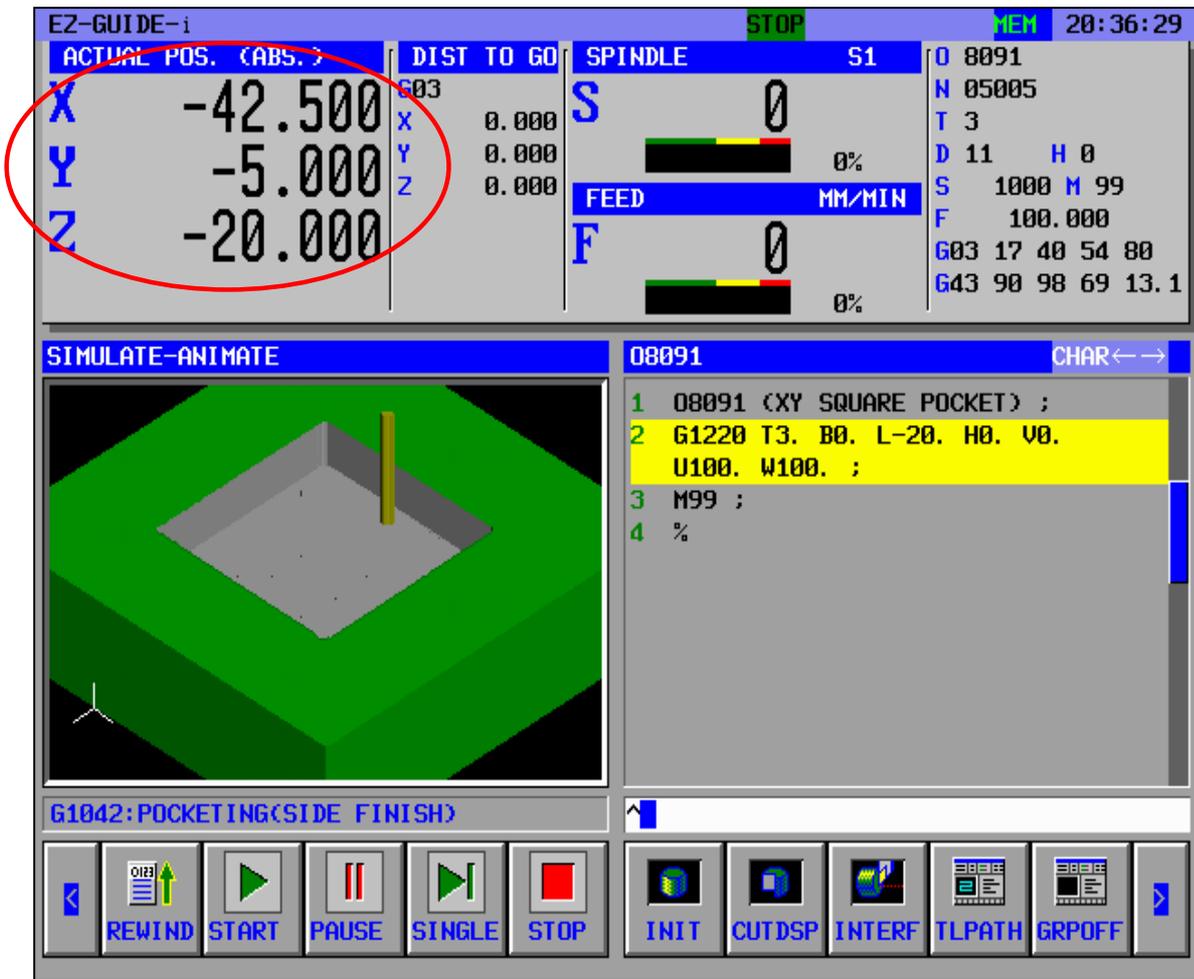


Figure selection from Subprogram directory

Tool definition for simulation by D parameter (ex. D11;)





<Fig.45 Continuous flow of Square Pocketing>

The final step before actual machining is to check by simulation.

The above example shows side fin. tool(D5 flat endmill) movement in XY plane.

In pre-check process, please make sure if the actual position varies correctly from the point of tool path's view.

**NOTE**

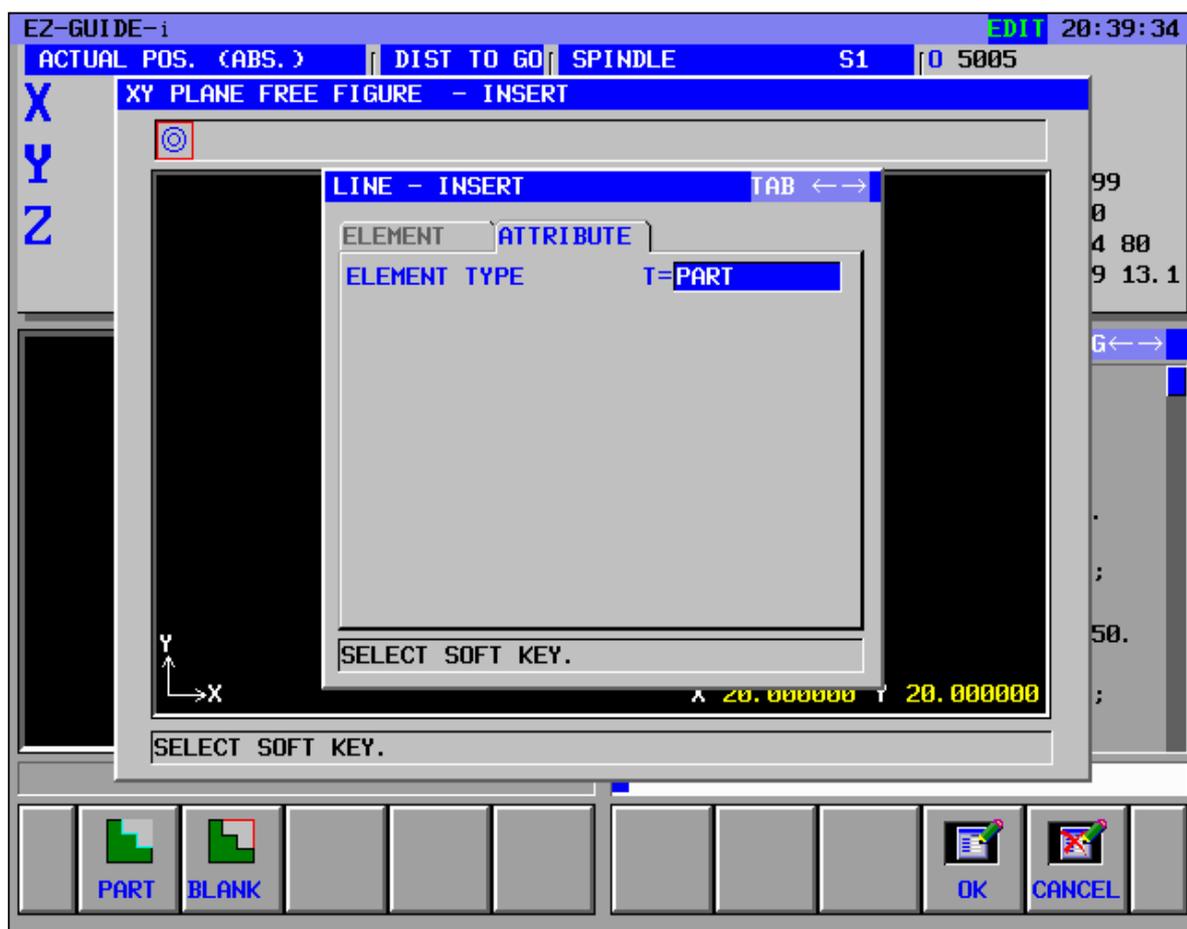
1. Bottom/Side thickness(T/S parameter) items in cut condition setup menu are just for the raw cast work which has concave shape from the beginning, to save cut time by reducing approach distance to a aimed surface.

### 4.3 Free Figure *Pocketing*

For *Pocketing* cycles, a free figure can be specified by inputting related data such as start point and linear / circular movement vector just like the other milling cycles.

As mentioned before, all cycles using milling cutter have similar concept to make arbitrary shape of machining, only except for pocket with Islands. **ONLY POCKET with ISLANDS!**

When specifying the outline of such a pocket, it should be determined whether the frame of pocket composing wall is necessary or not, for an operator should select the property of the shape first in case of arbitrary figure set-up. The default setting is *Part* method.



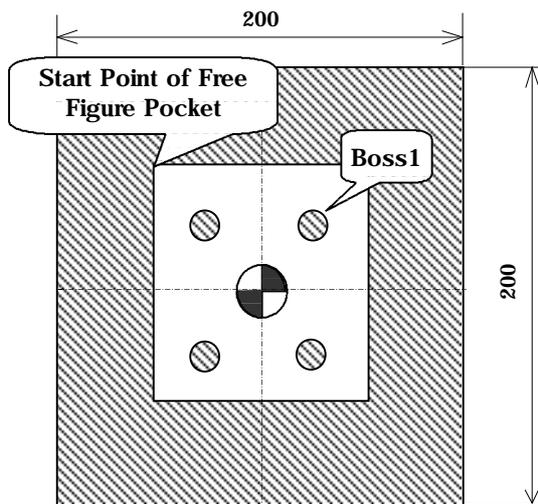
<Fig.46 Part / Blank method selection for arbitrary Pocket with islands composition>



<Fig.47 Machining example of Blank method for Pocket with one island>

Regarding simulation, **Animation** functions only in case of rectangular and cylinder type work for all machining cycles.

And figure embodiment on rectangular and cylinder work, even though it is free style, can be pre-checked by **Animation** method.



**Width of Pocket : 100\*100mm**

**Depth of Pocket : 20mm**

**Z Coordinate of Top Surface : 0**

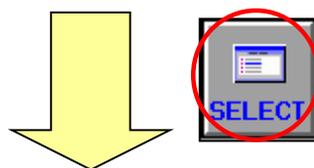
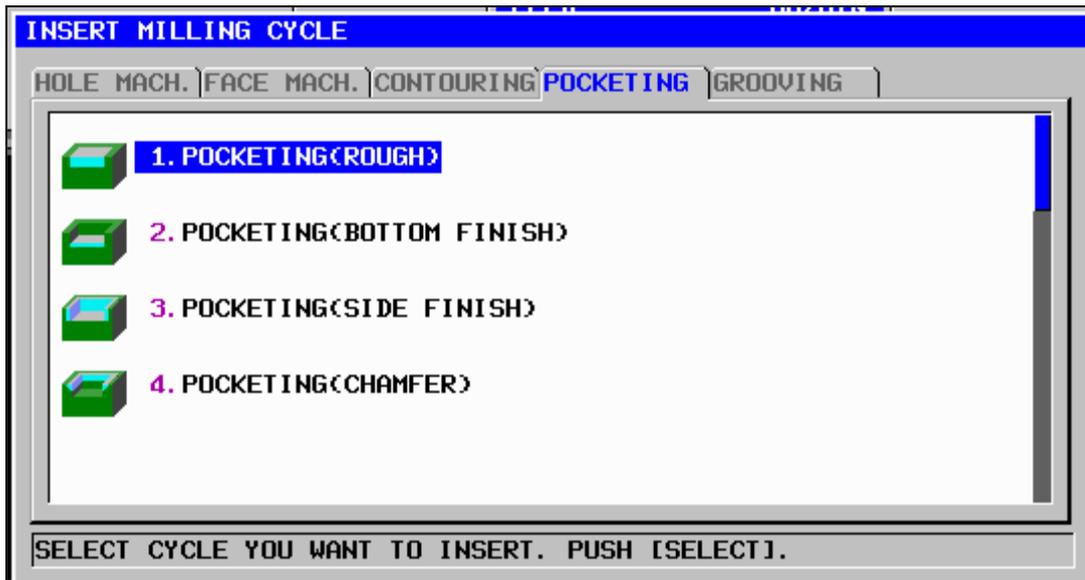
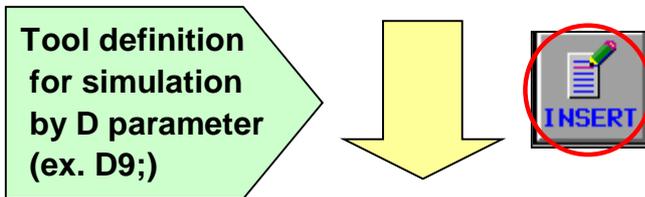
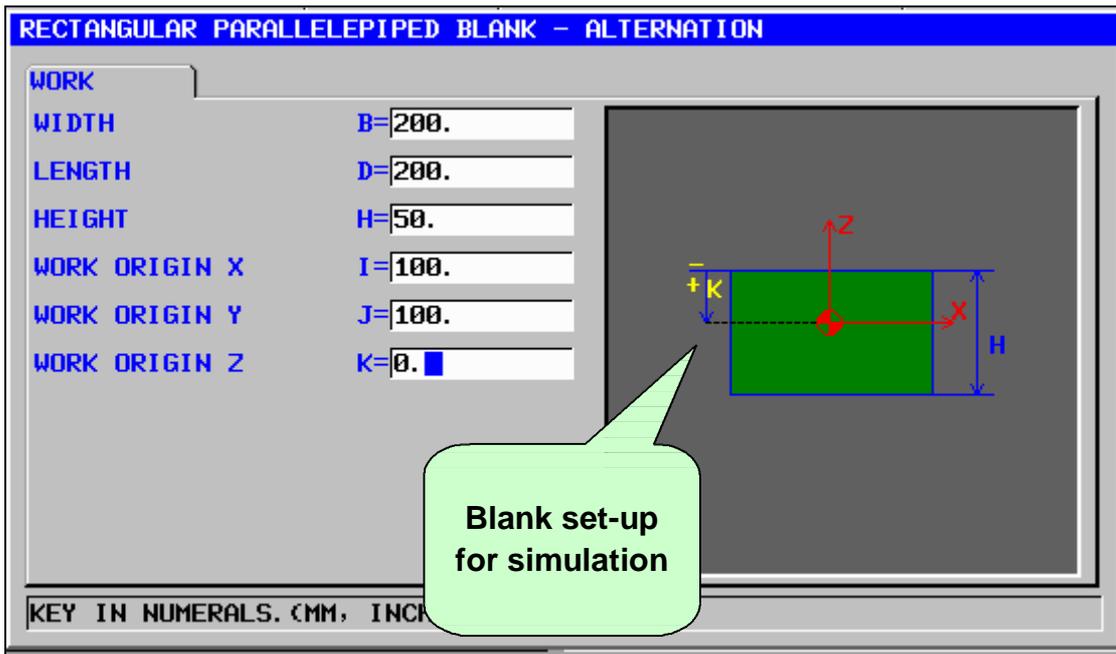
**Center of boss 1 : (25,25)**

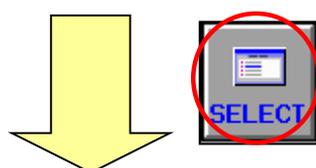
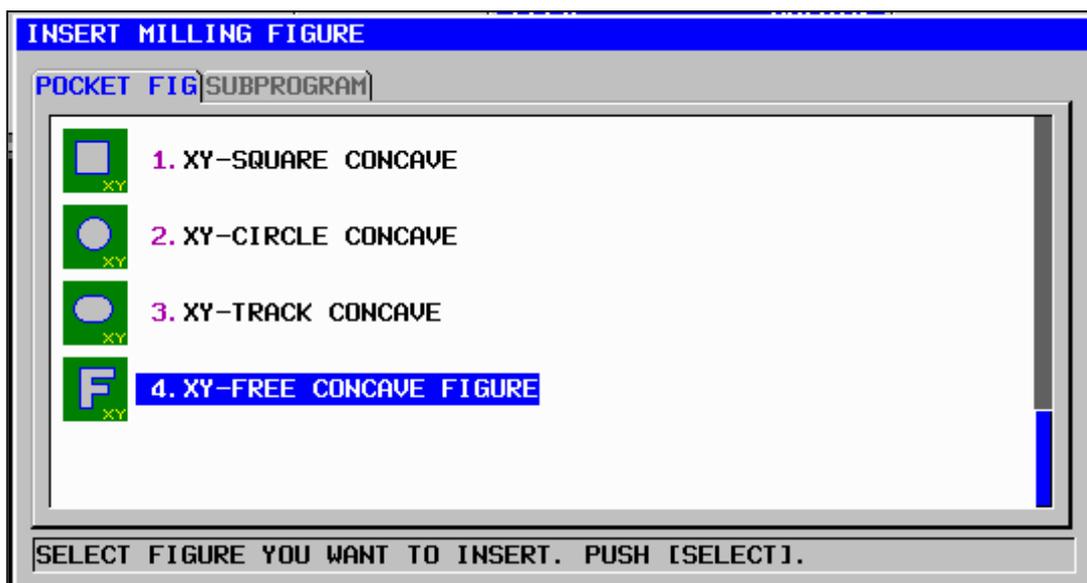
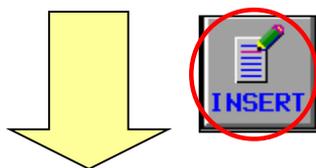
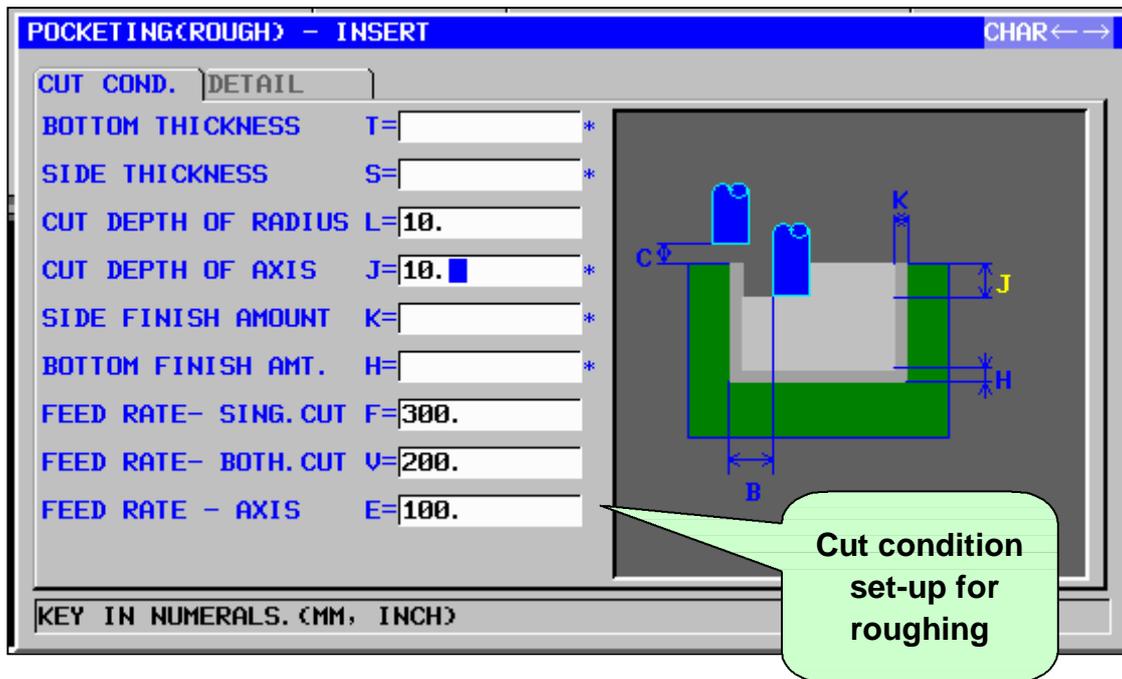
**Boss Diameter : 10mm**

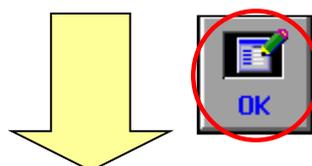
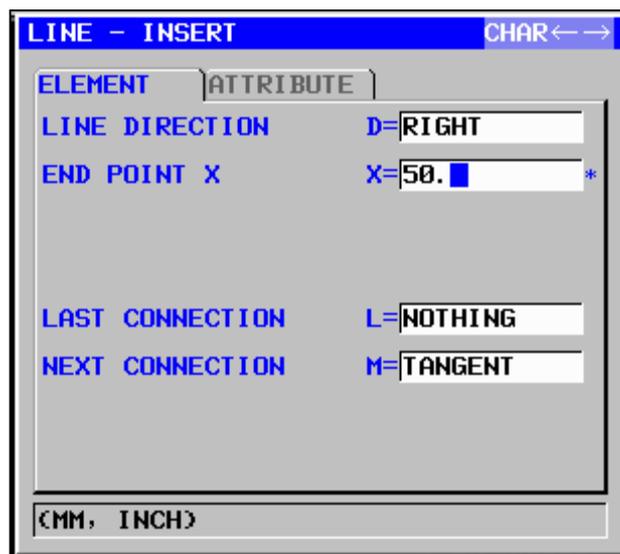
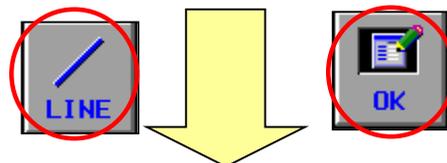
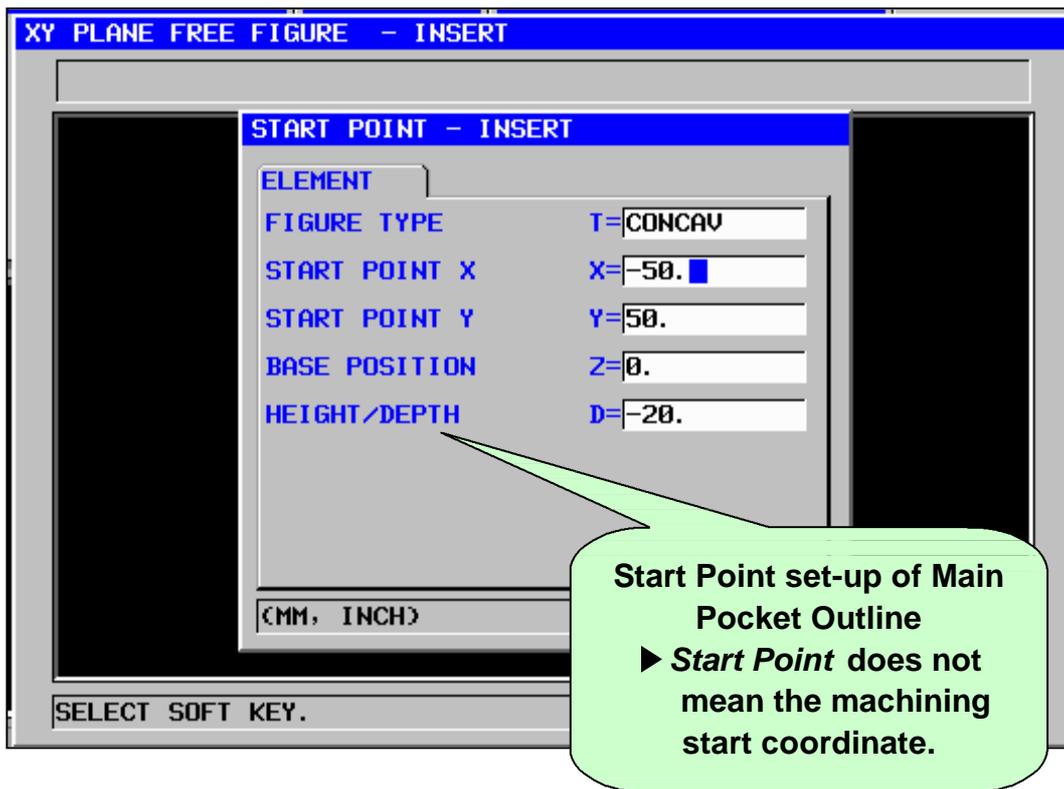
**Attribute of work : Part**

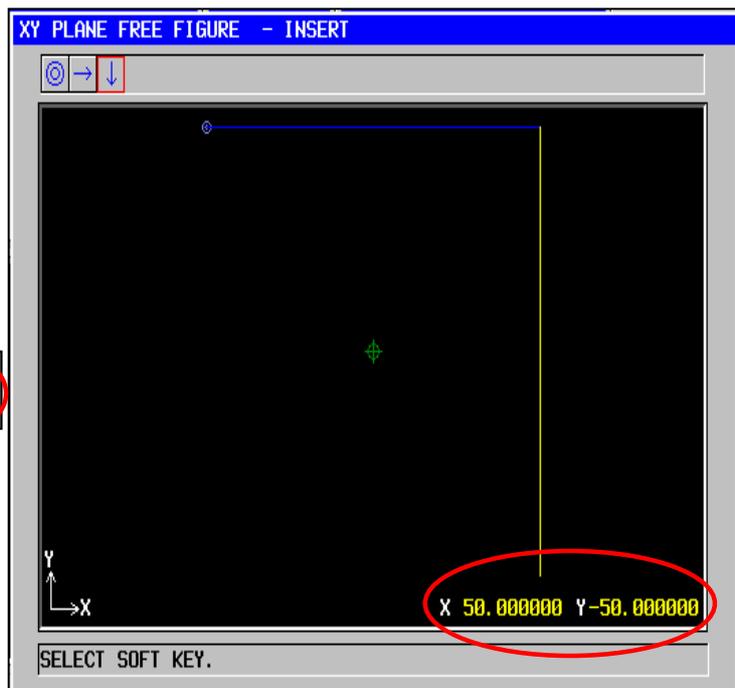
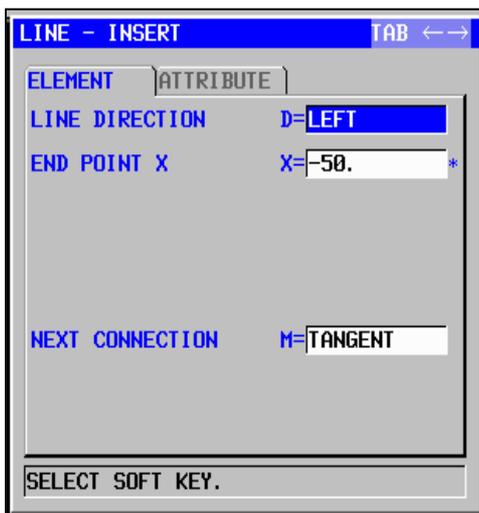
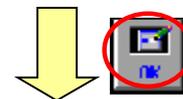
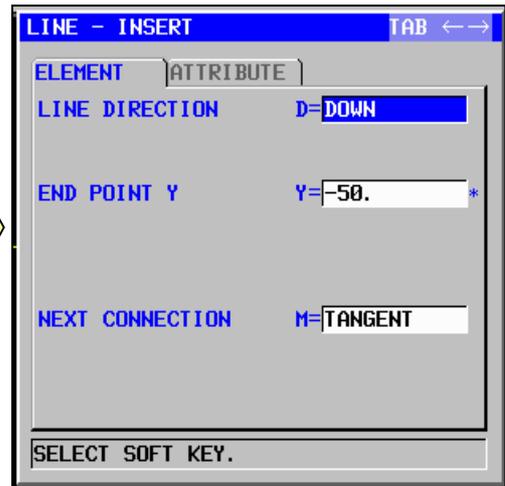
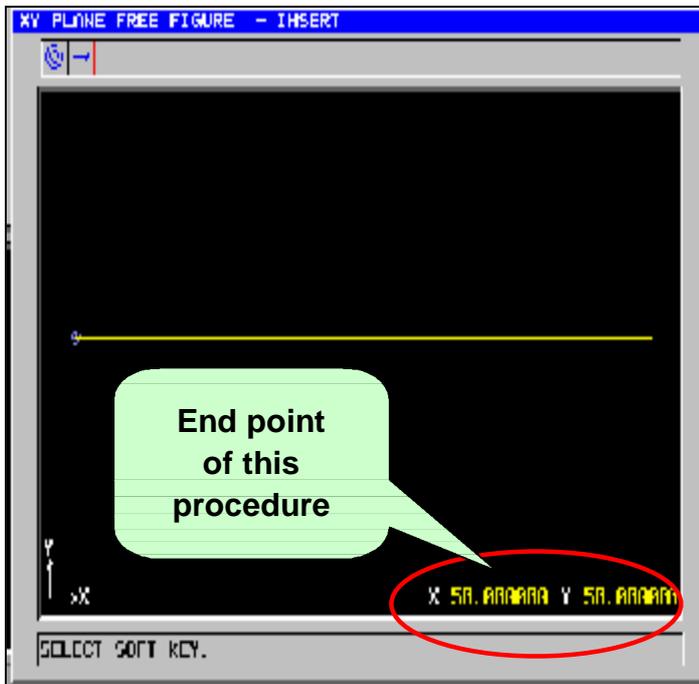
<Fig.48 Square Pocket example - with 4 bosses inside>

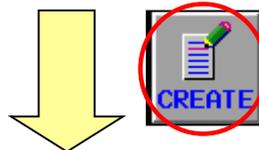
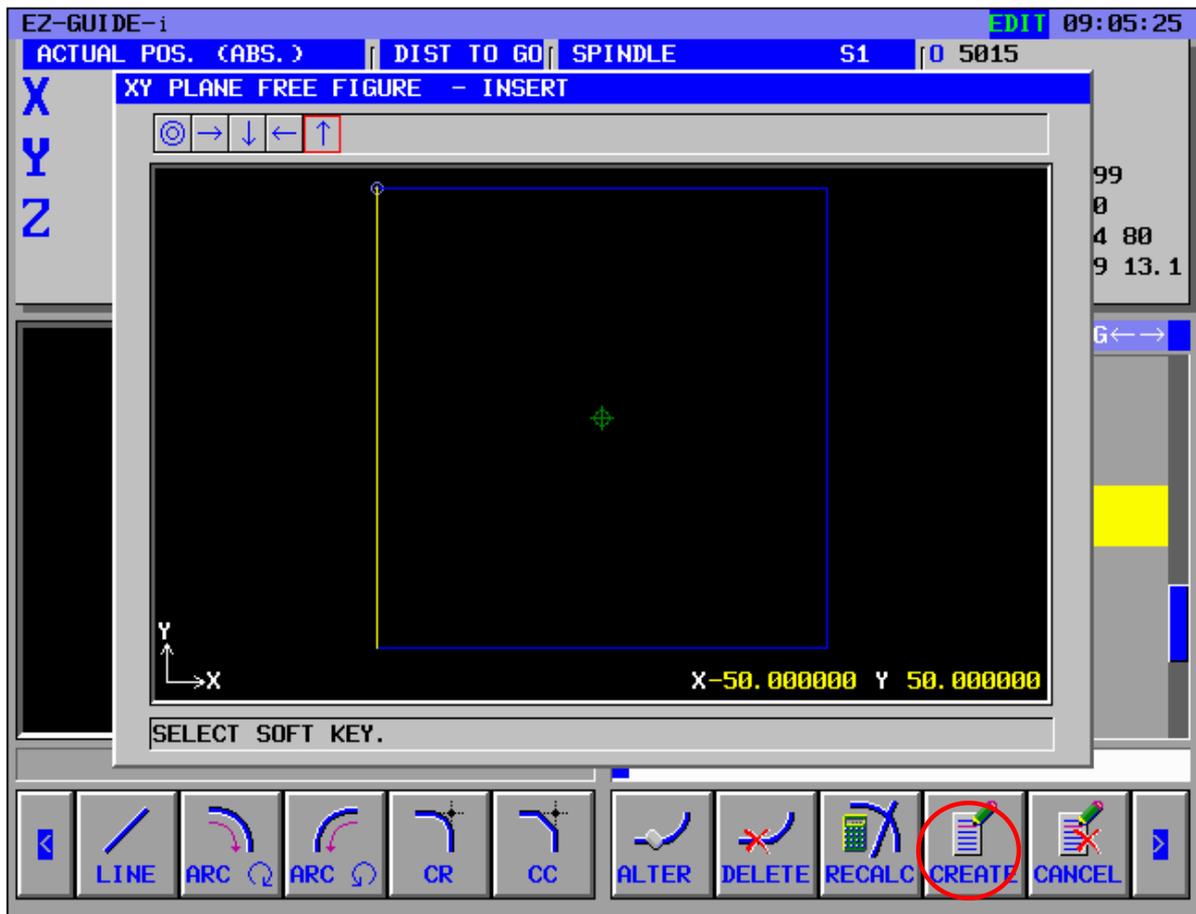
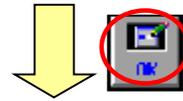
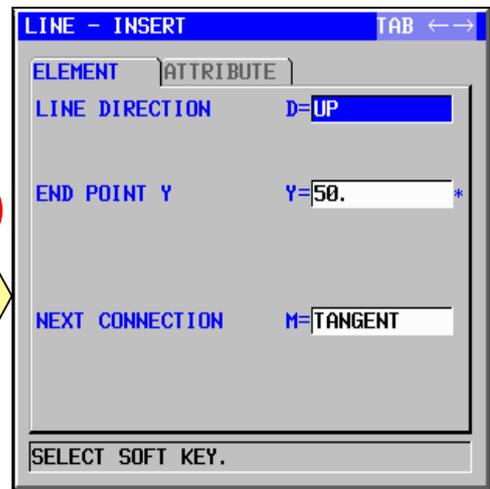
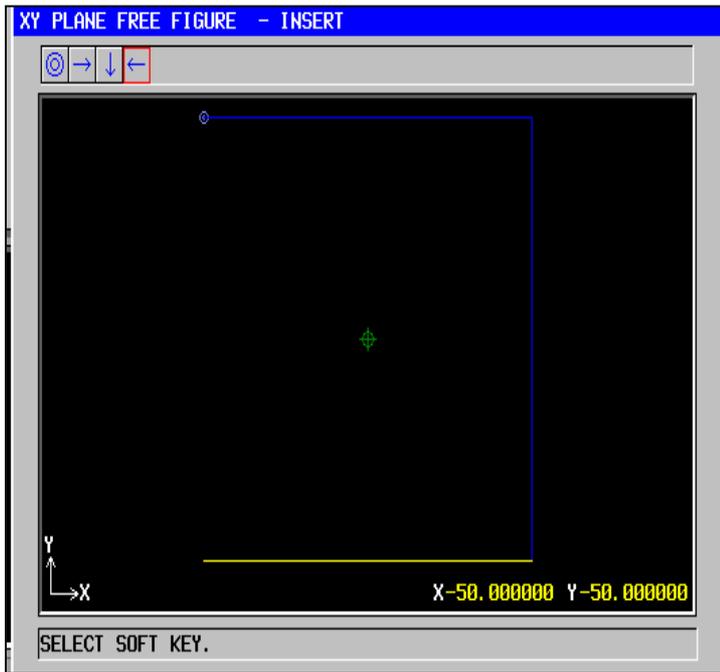
For operator's proper understanding of island making, a square type pocket example with boss(**Fig. 48**) is to be explained on **Fig.49**.

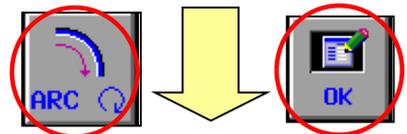
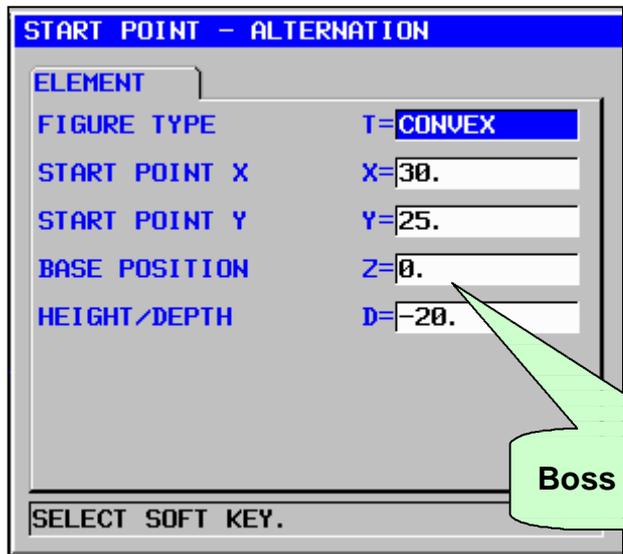
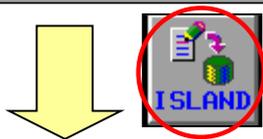
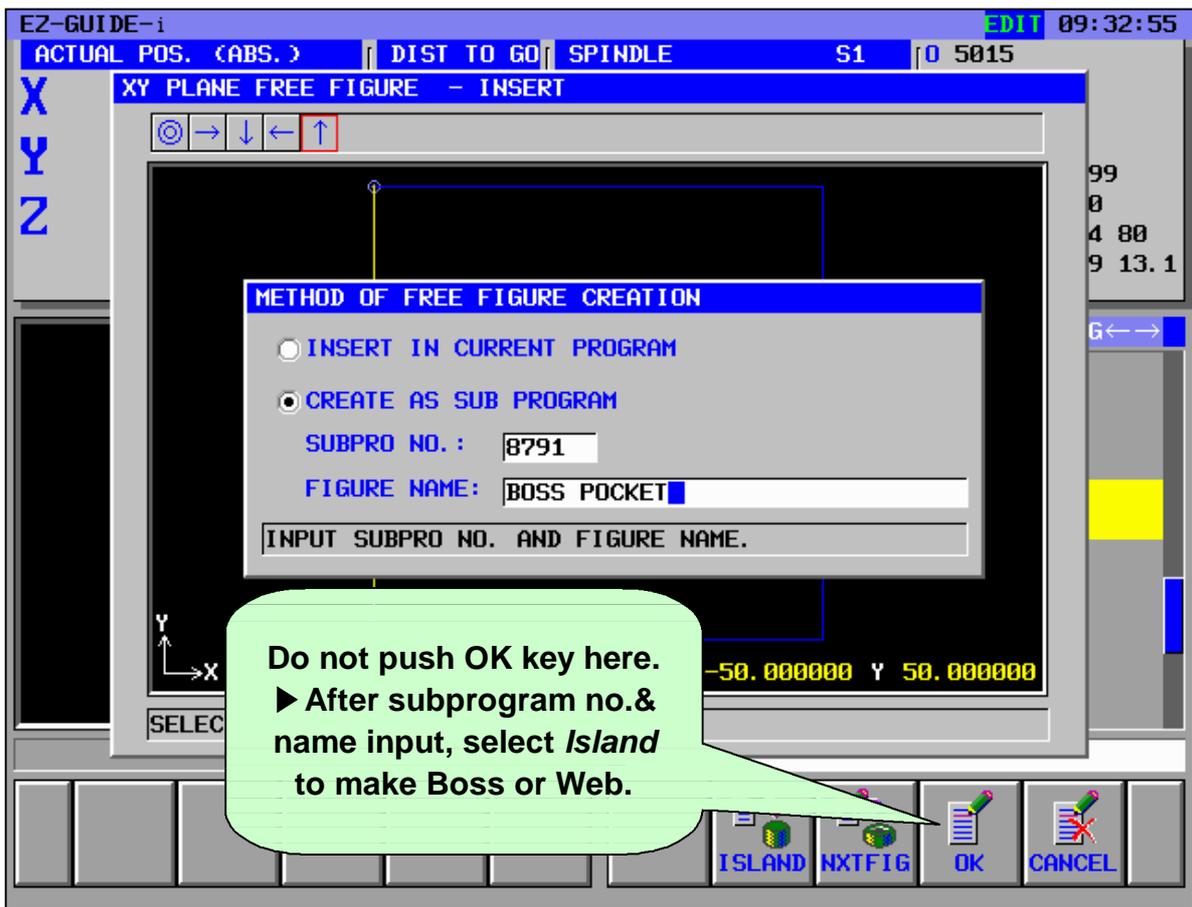








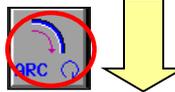
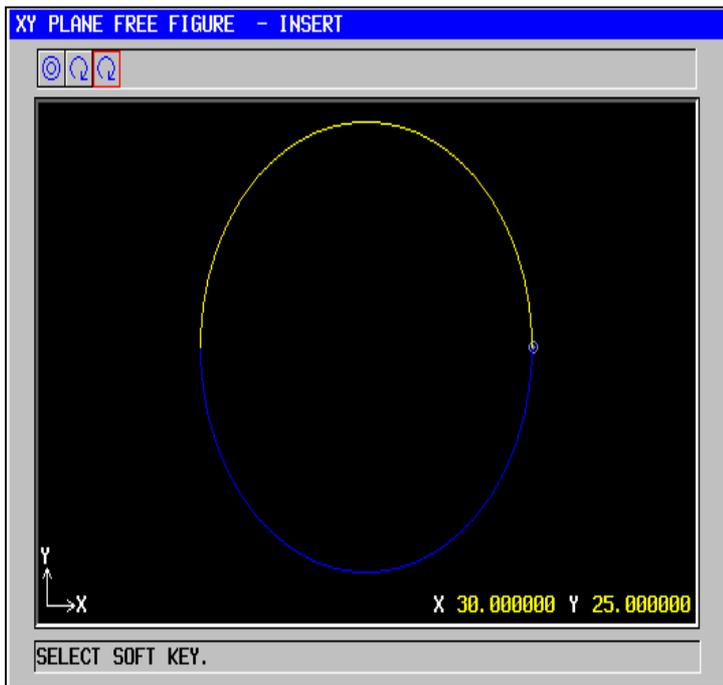
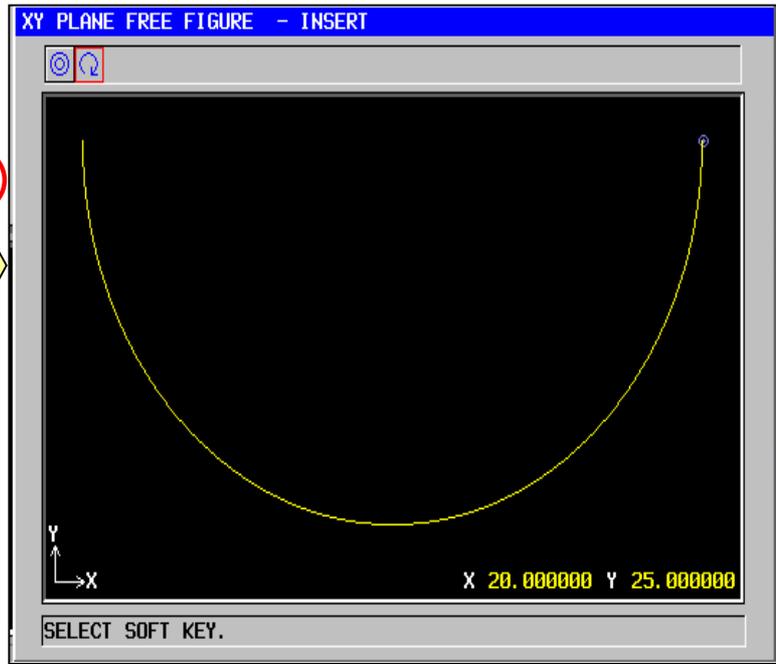




ARC (CW) - INSERT CHAR←→

ELEMENT	ATTRIBUTE
END POINT X	X=20.
END POINT Y	Y=25.
RADIUS	R=5.
CENTER POINT CX	CX=25.
CENTER POINT CY	CY=25.
LAST CONNECTION	L=NOTHING
NEXT CONNECTION	M=TANGENT

(MM, INCH)

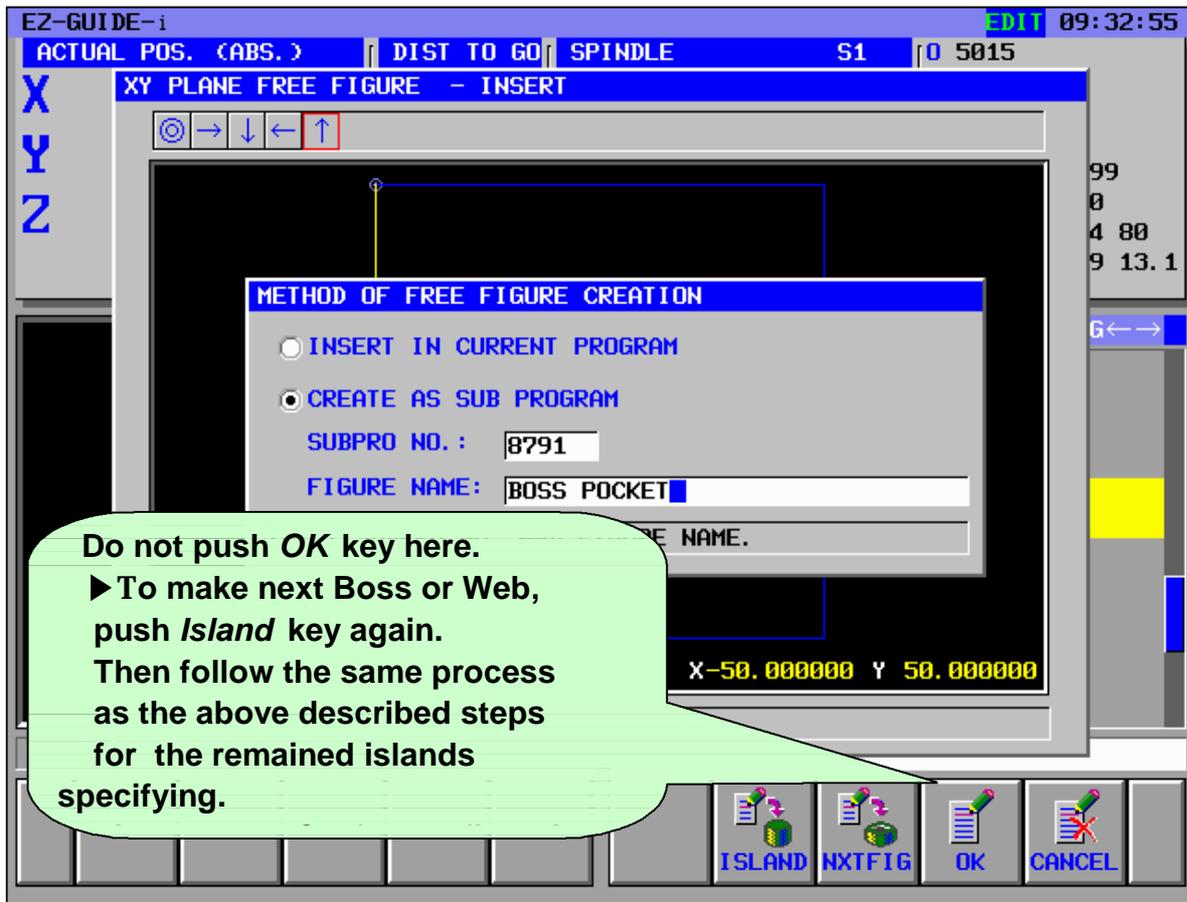


ARC (CW) - INSERT CHAR←→

ELEMENT	ATTRIBUTE
END POINT X	X=30.
END POINT Y	Y=25.
RADIUS	R=5.
CENTER POINT CX	CX=25.
CENTER POINT CY	CY=25.
NEXT CONNECTION	M=TANGENT

(MM, INCH)

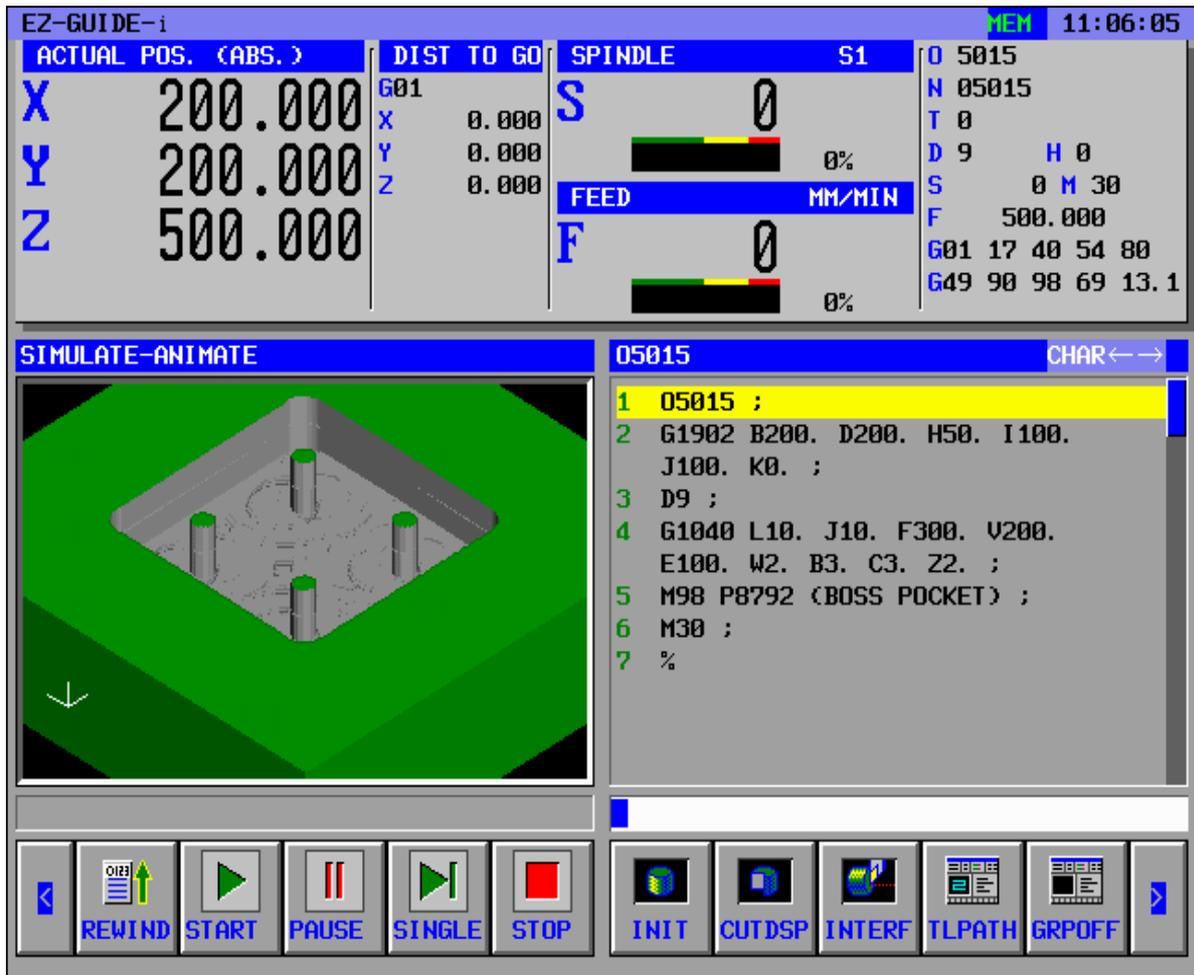




<Fig.49 Continuous flow of XY Free figure set-up in Contouring cycle>

NOTE

1. To remove inconvenience of setting the same free figure several times, mirror/copy/rotation/shift function are now under development by FANUC.



<Fig.50 Free Figure Contouring example animation with 4 islands>

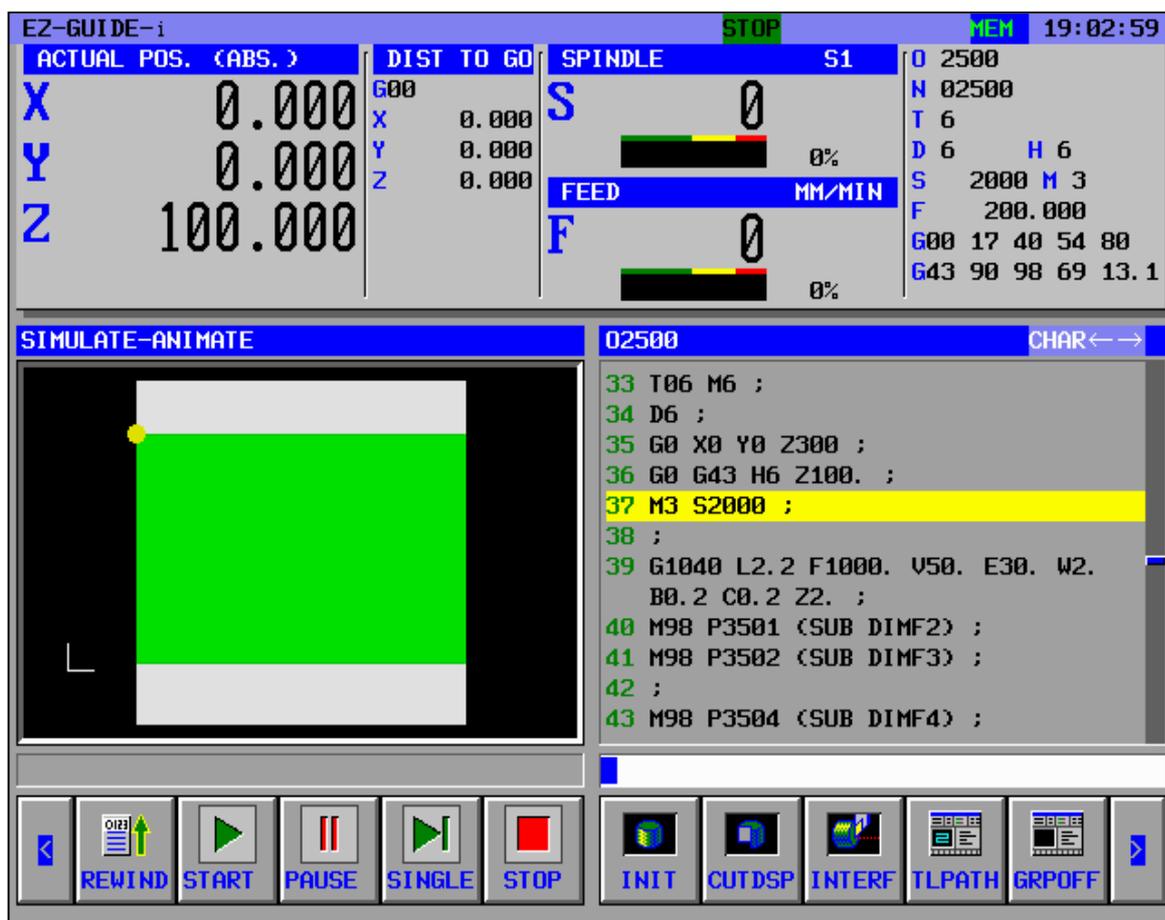
# 5. Contouring (side surface trimming)

## 5.1 Main feature of Contouring cycle

The purpose of **Contouring** cycle is to cut side surface of work to a commanded depth and width. Normally the tool path is to be closed pattern. But, in some cases operators would like to machine uni-directional step at one side. If so, please make use of **Partial Contouring** cycle even though face cutter is used.

As mentioned, **Animation** functions only in case of rectangular and cylinder type work for all machining cycles.

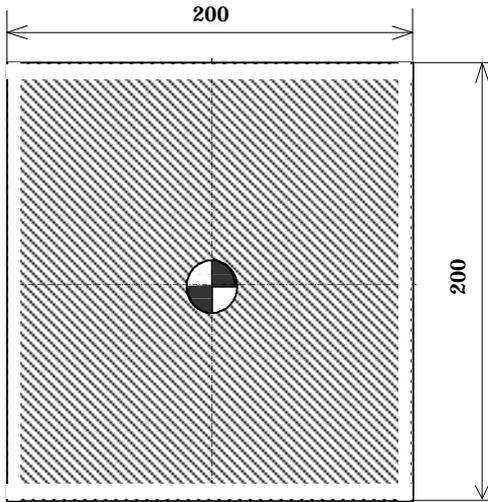
For arbitrary shape machining of **Outer Wall Contouring**, make use of **Tool Path Drawing** or **MCHDRW** menu.



<Fig.51 Partial Contouring simulation example>

## 5.2 Setting parameters of *Contouring* cycle

For operator's proper understanding, a square type Contouring example on rectangular piped work (*Fig. 52*) is to be explained on *Fig.53*.



**Dimension : 200\*200\*50mm**

**Cut Depth Radius : 10mm**

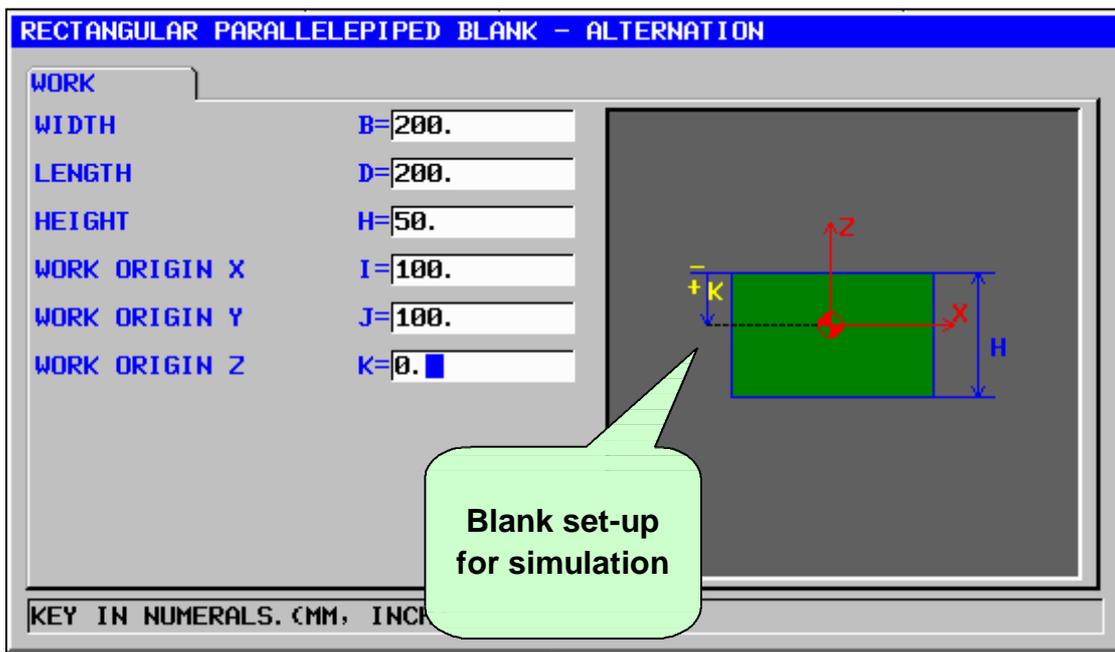
**Cut Depth Axis : 20mm**

**Z Coordinate of Top Surface : 0**

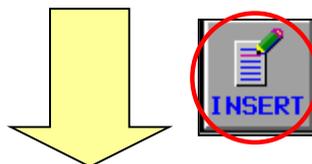
**Rough Endmill = D20 / 2flute**

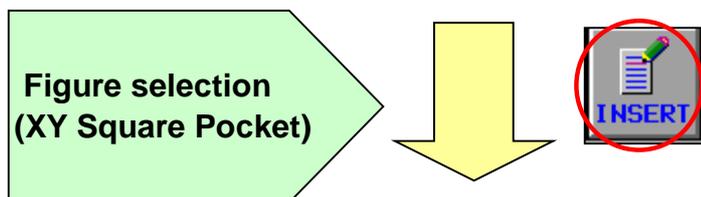
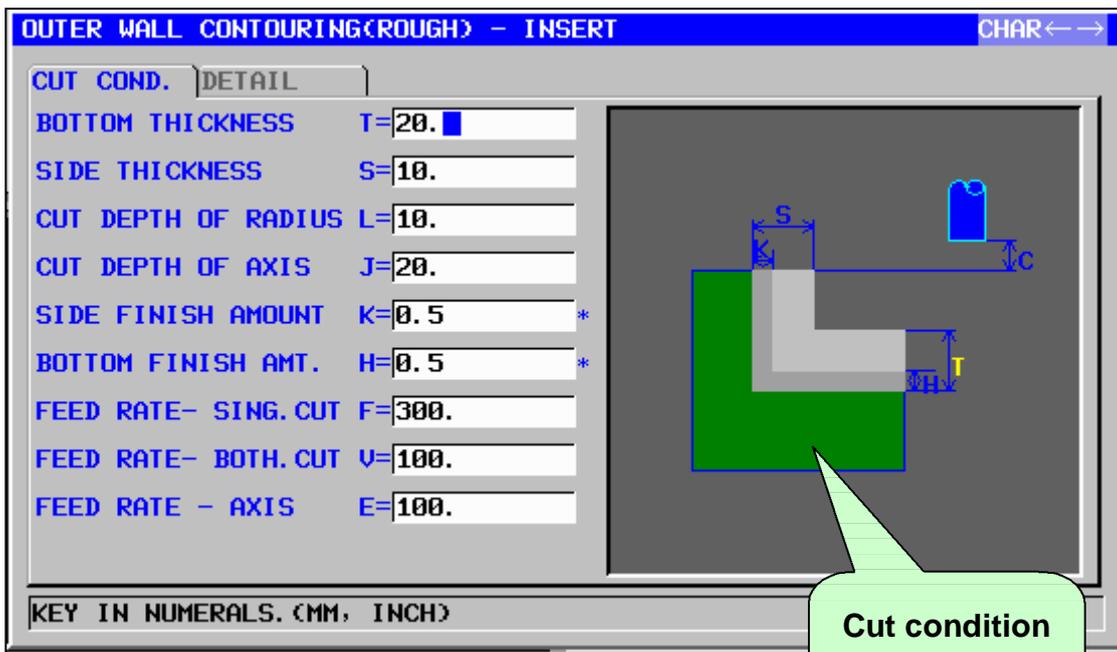
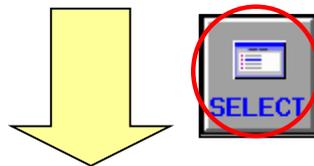
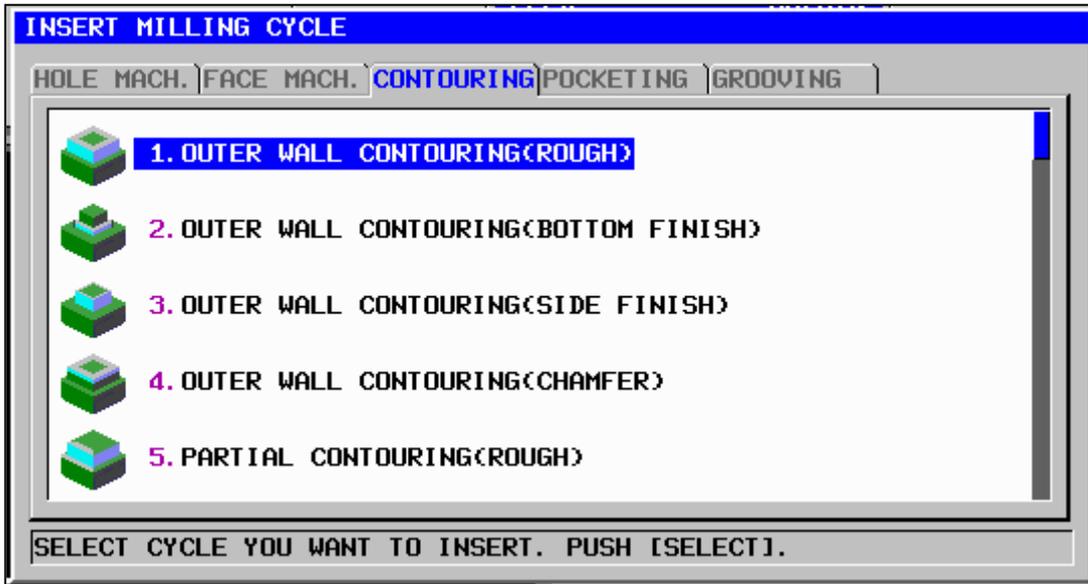
**Bottom/Side Fin. Endmill = D12 / 3flute**

<Fig.52 Square Pocket example - Dimension & cut condition>



Tool definition  
for simulation  
by D parameter  
(ex. D9;)





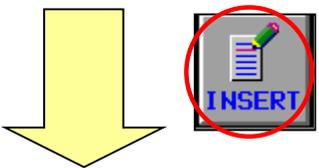
**XY-SQUARE - INSERT**

POS./SIZE

FIGURE TYPE	T=CONVEX
BASE POSITION	B=0.
HEIGHT/DEPTH	L=-20.
CENTER POINT(X)	H=0.
CENTER POINT(Y)	V=0.
LENGTH FOR X AXIS	U=180.
LENGTH FOR Y AXIS	W=180.
CORNER RADIUS	R= <input type="text"/> *
ANGLE	A= <input type="text"/> *

UNITS: (MM, INCH)

Figure set-up for roughing



**METHOD OF FIGURE PROGRAM CREATION**

INSERT IN CURRENT PROGRAM

CREATE AS SUB PROGRAM

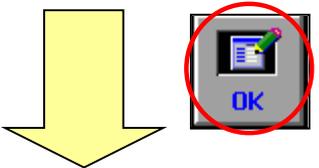
SUBPRO NO. :

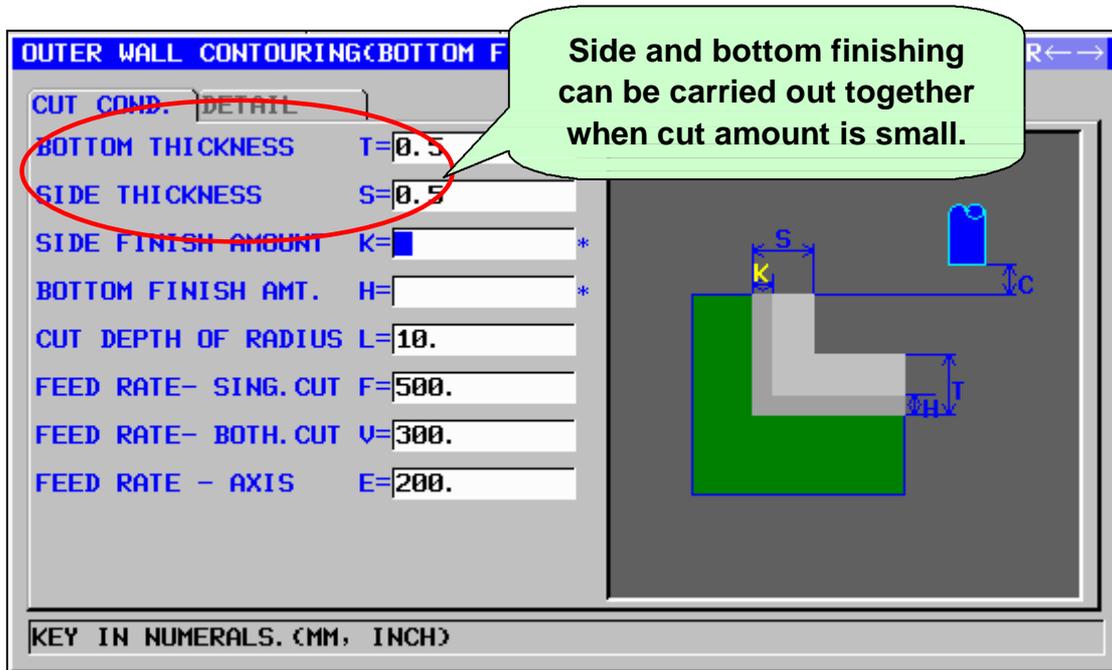
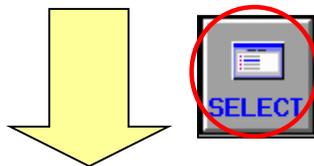
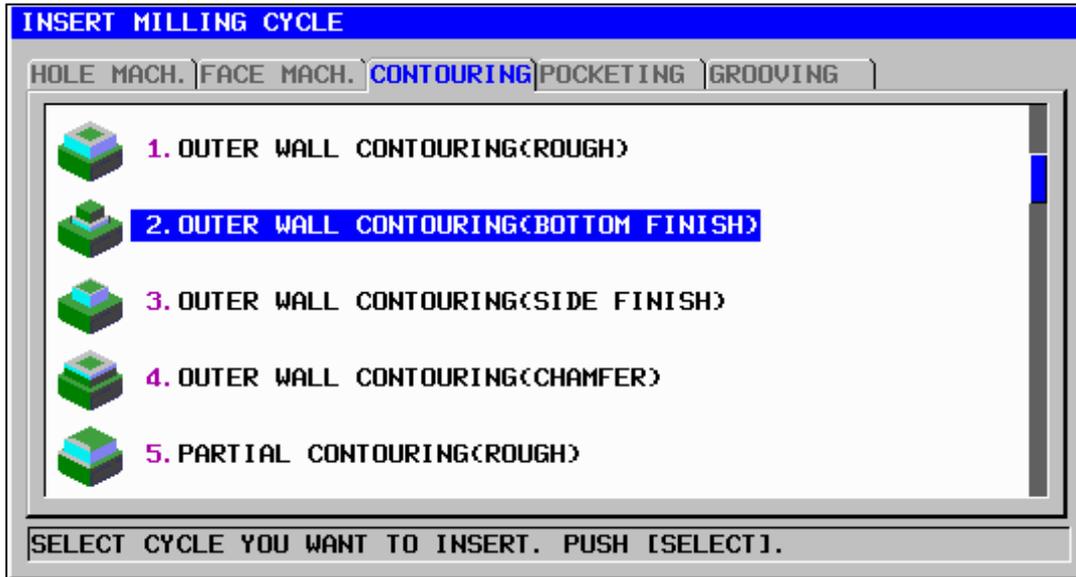
COMMENT :

INPUT SUBPRO NO. AND COMMENT.

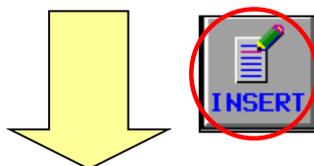
Storage of figure as subprogram

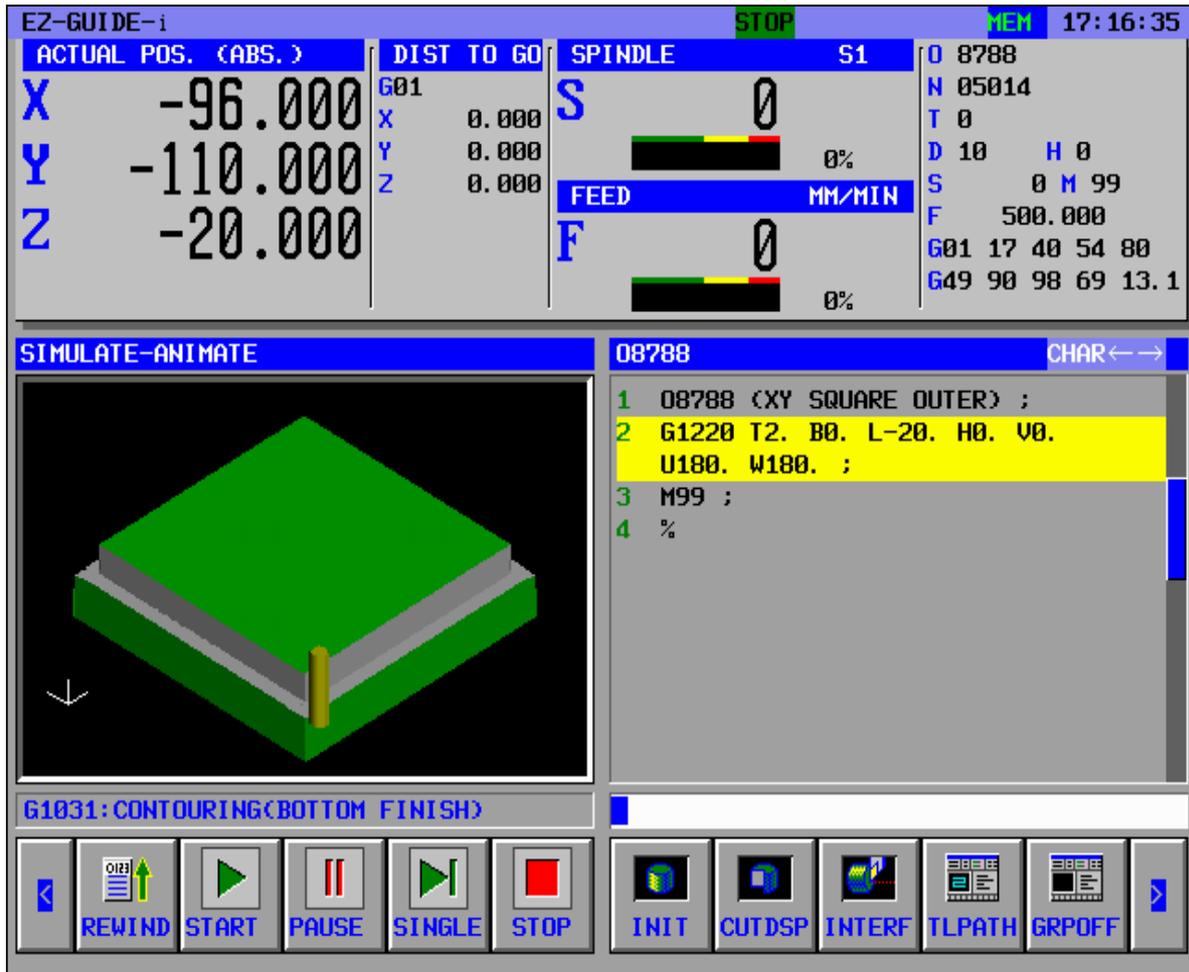
Tool definition for simulation by D parameter (ex. D10;)





The same figure calling from Subprogram directory





*<Fig.53 Continuous flow of Square Contouring>*

The final step before actual machining is to check by simulation.

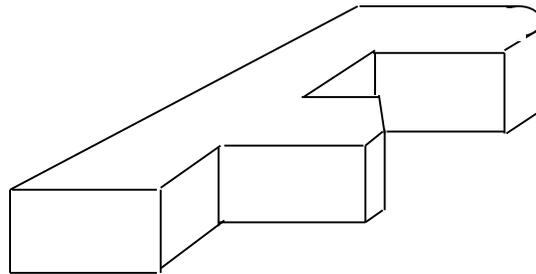
The example shows side fin. tool(D12 flat endmill) movement in XY plane.

In pre-check process, please make sure if the actual position varies correctly from the point of tool path's view.

### 5.3 Free Figure *Outer Wall Contouring*

In *EZ-i* system, you can apply the already established figure which was used for other cycles, by changing the original figure property.

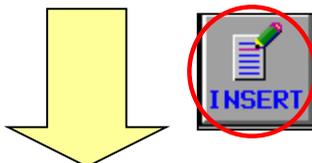
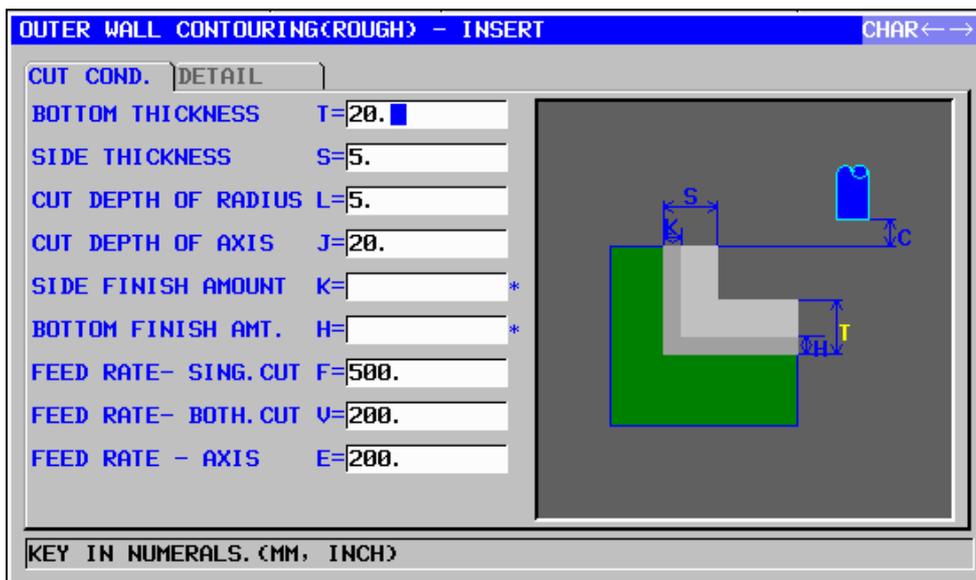
The following example shows how to modify a *Facing* free figure for *Outer Wall Contouring* arbitrary work.

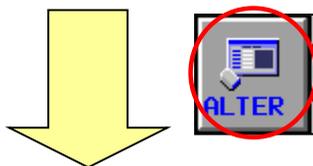
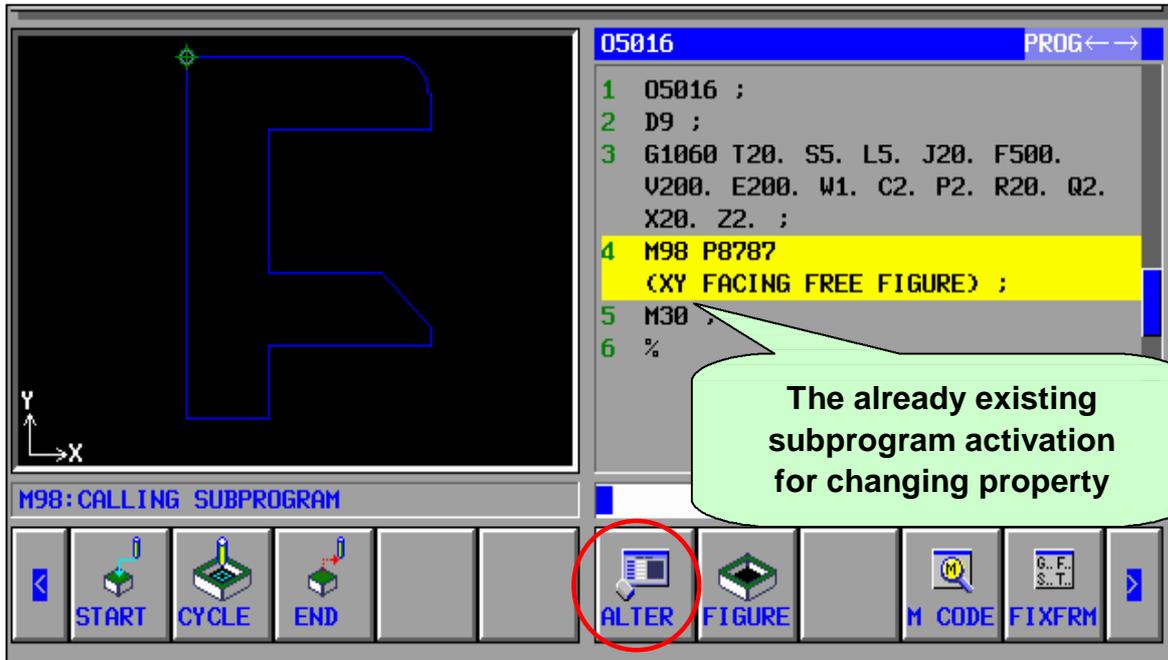
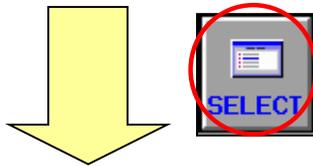
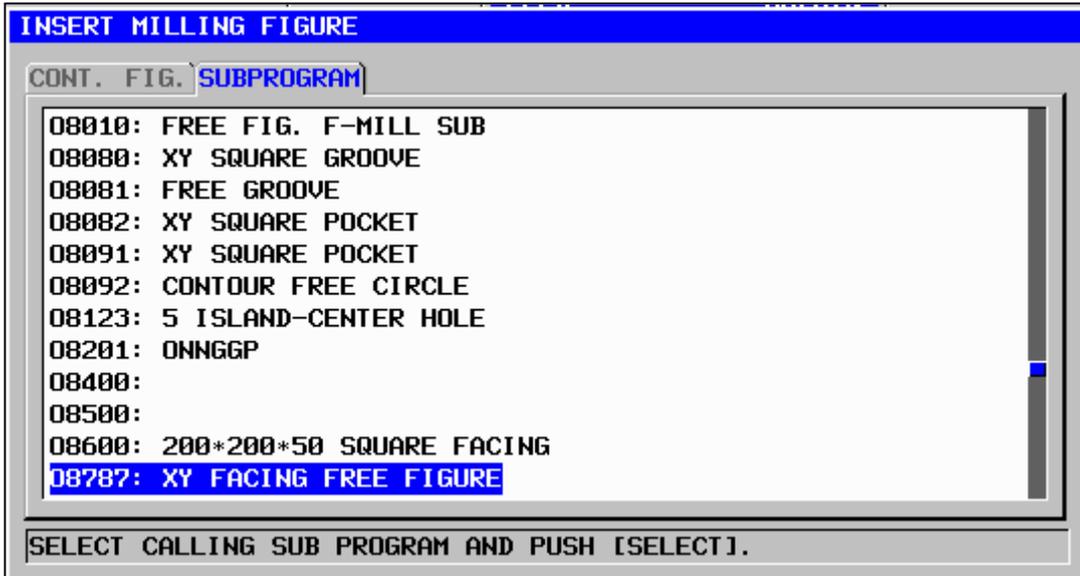


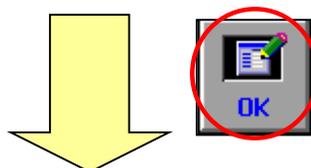
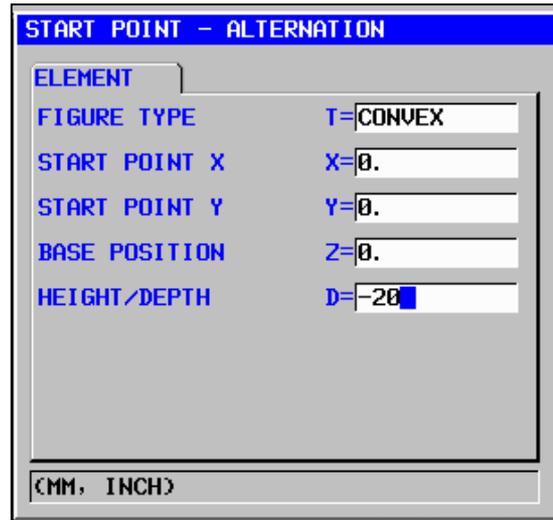
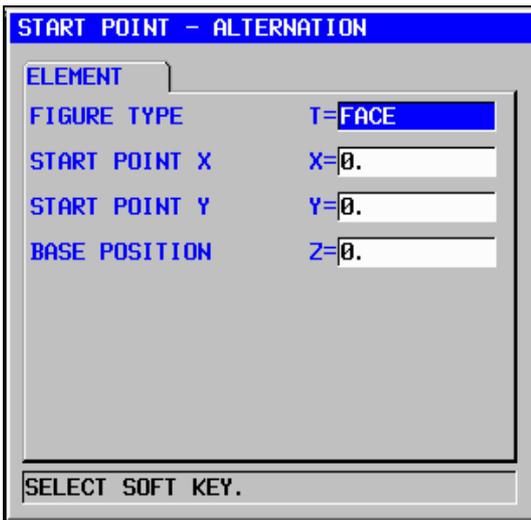
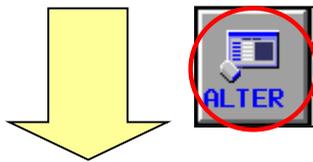
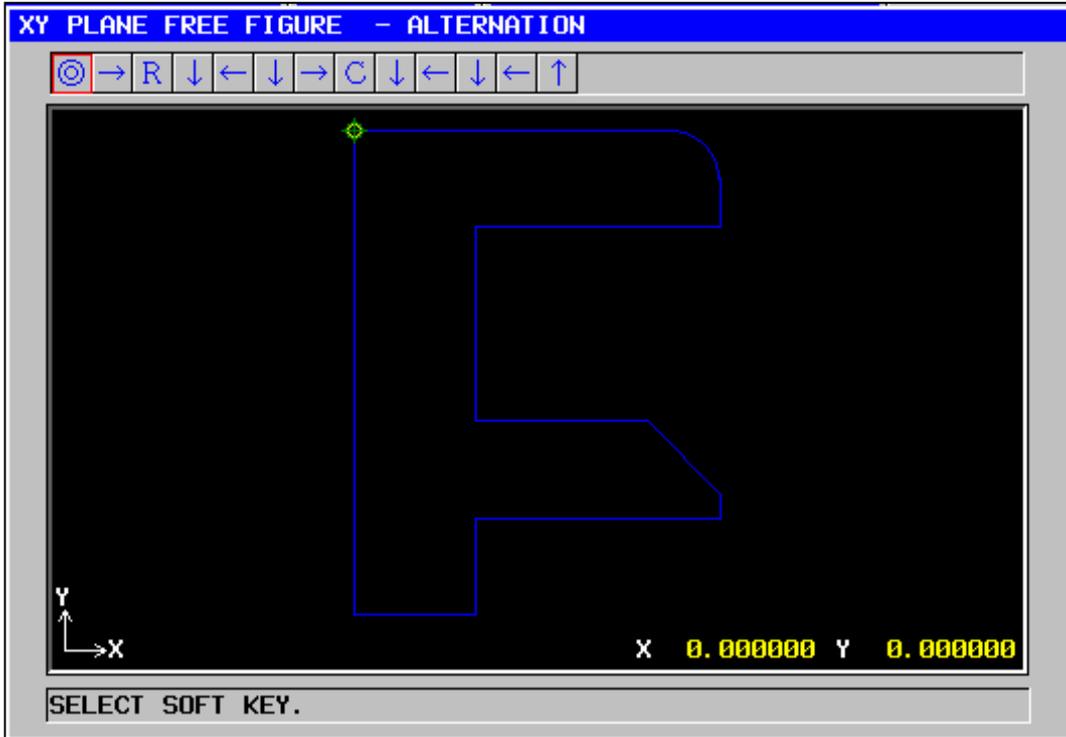
The side face of the above free style work is to be machined by D12 flat endmill to the depth of radius 5mm and Z-axis 20mm.

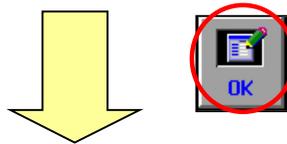
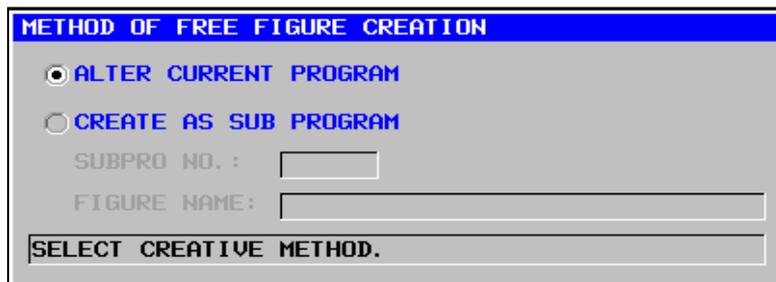
You have already made the shape form for *Facing*.

The programmer has only to modify the figure definition into Contouring machining as follows ;









EZ-GUIDE-i		MEM		15:15:34	
ACTUAL POS. (ABS.)		DIST TO GO		SPINDLE S1	
X	200.000	G01	X	0.000	S
Y	200.000	X	Y	0.000	0
Z	500.000	Z	Z	0.000	0%
				FEED MM/MIN	
				F	
				0	
				0%	
				D 9 H 0	
				S 0 M 30	
				F 500.000	
				G01 17 40 54 80	
				G49 90 98 69 13.1	

SIMULATE-TOOLPATH		05016		CHAR←→	
		1 05016 ;			
		2 D9 ;			
		3 G1060 T20. S5. L5. J20. F500. V200. E200. W1. C2. P2. R20. Q2. X20. Z2. ;			
		4 M98 P8787 (XY FACING FREE FIGURE) ;			
		5 M30 ;			
		6 %			

REWIND	START	PAUSE	SINGLE	STOP	DISP	NODISP	CLEAR	ANIME	GRPOFF
--------	-------	-------	--------	------	------	--------	-------	-------	--------

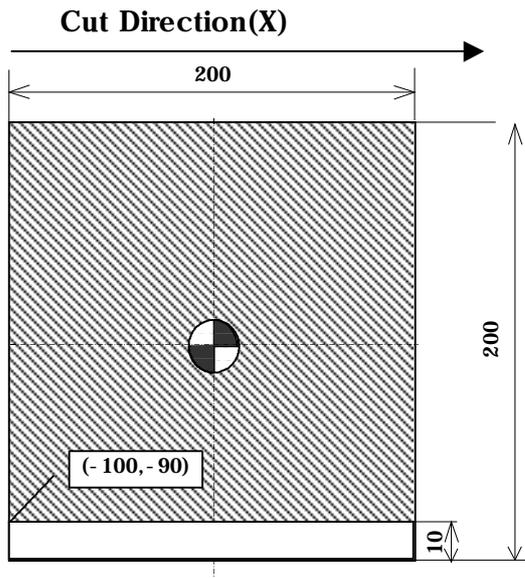
<Fig.54 Continuous flow of Free Contouring by altering other cycle figure>

**NOTE**

1. The original property of figure is wholly changed for new cycle. If you want to have the original value remained, copy the subprogram to be changed before modifying.
2. For **Free Figure Contour** program, make use of **Tool Path Drawing** function to pre-check mistakes.

## 5.4 How to set-up *Partial Contouring*?

This cycle is for any open step shape on work side-face.

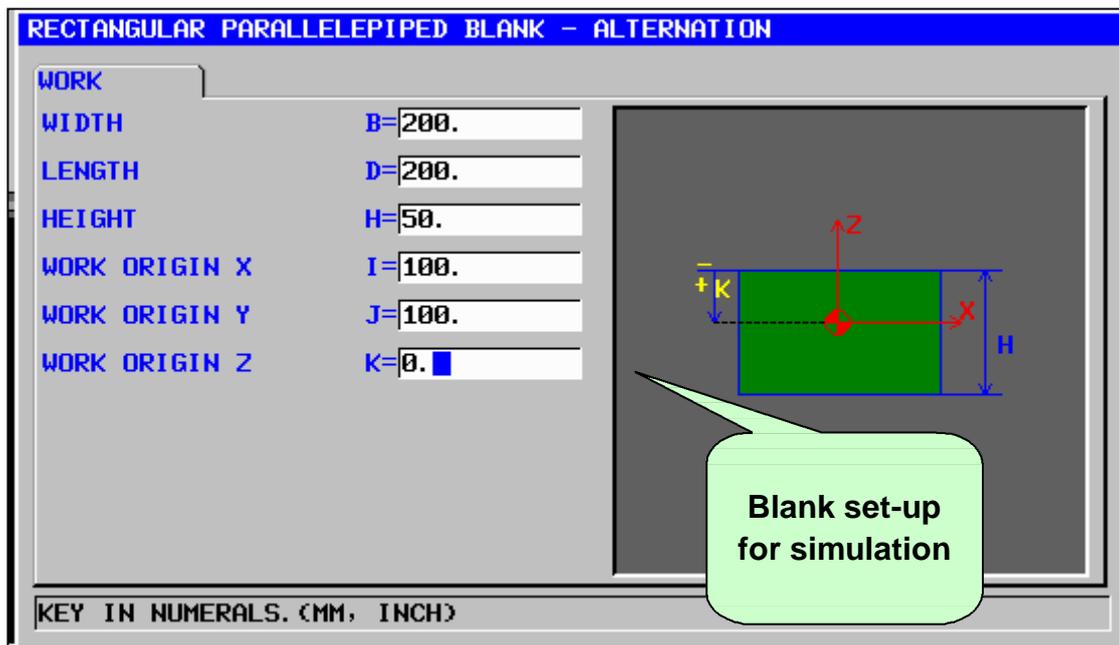


**Dimension : 200\*200\*50mm**

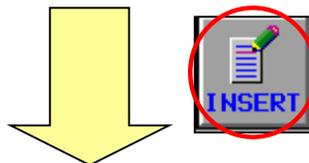
**Cut Depth Axis : 20mm**

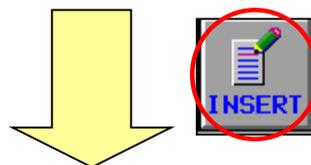
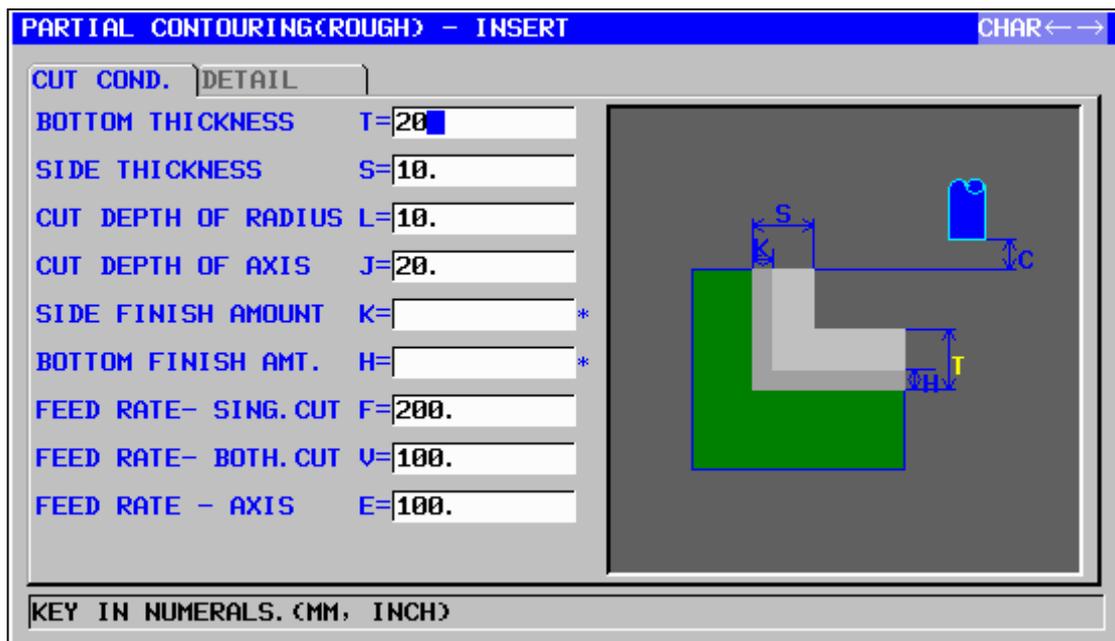
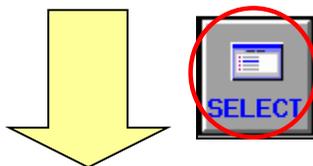
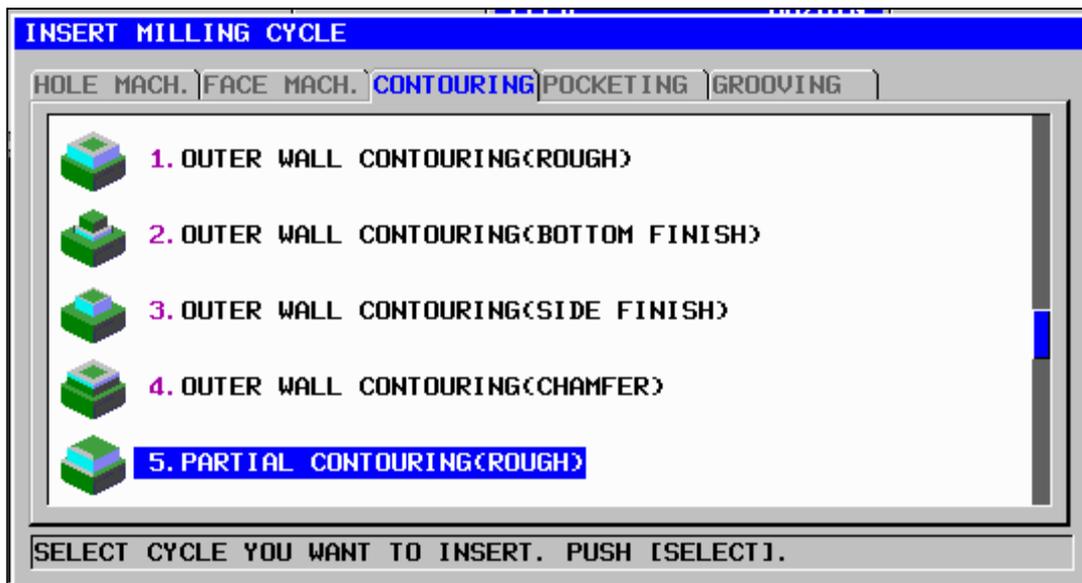
**Z Coordinate of Top Surface : 0**

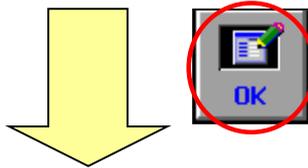
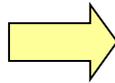
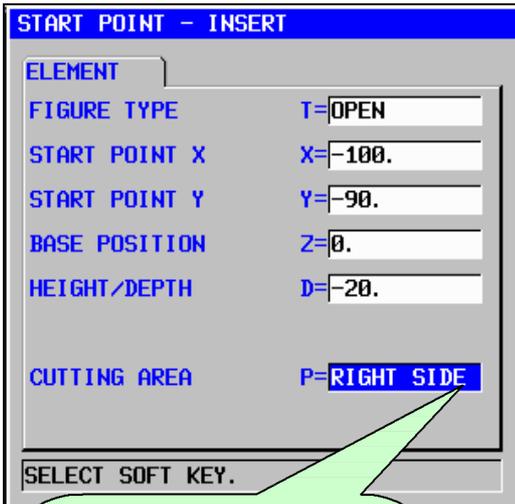
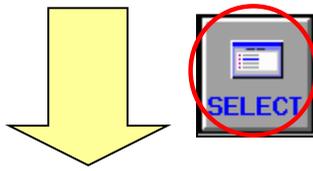
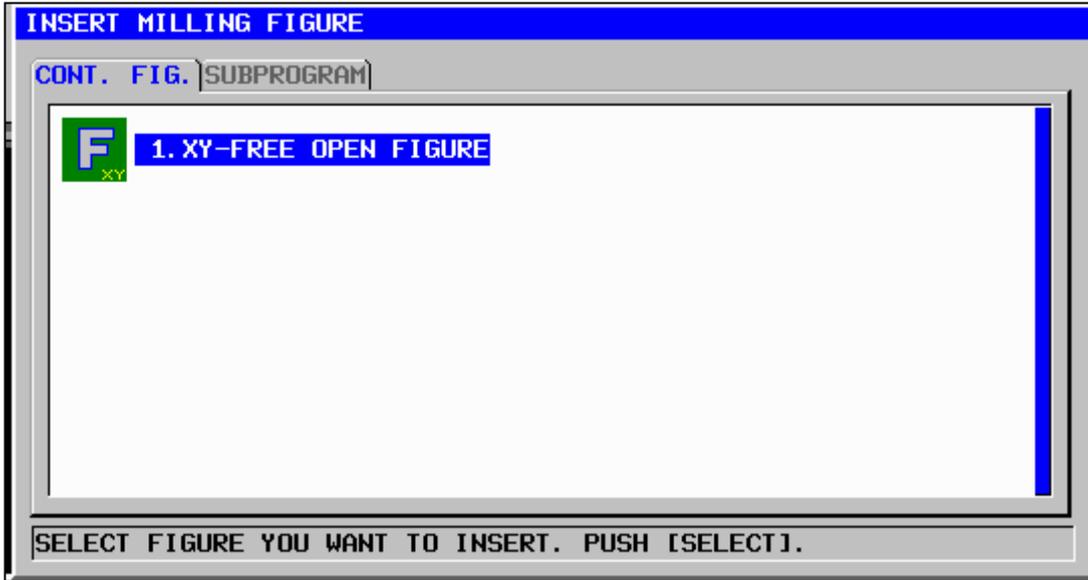
**Rough Endmill = D20 / 2flute**



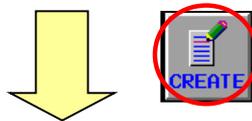
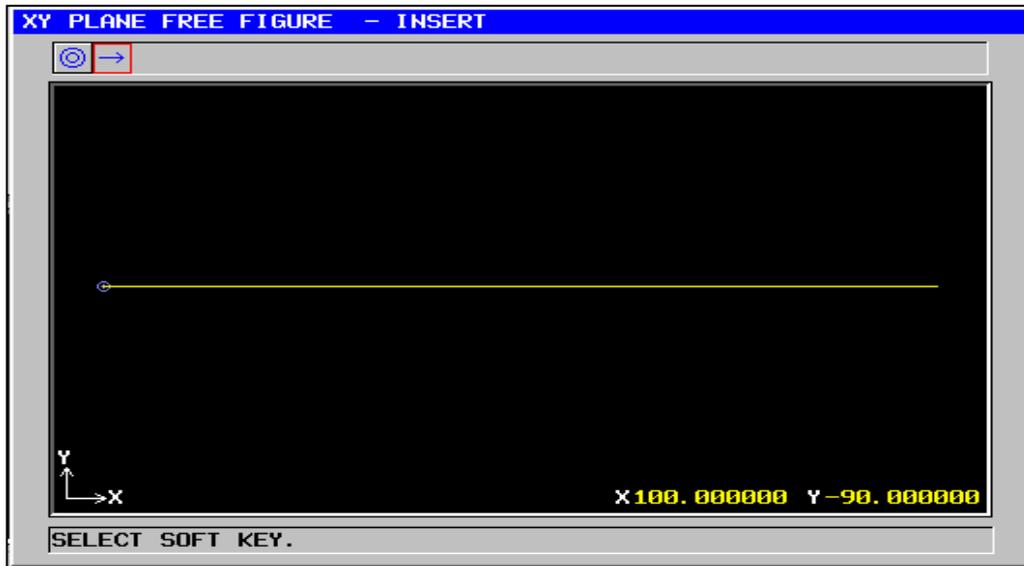
Tool definition  
for simulation  
by D parameter  
(ex. D9;)







*Right Side* means the right section of drawn figure line is to be machined, from the point of *Start Point's* view.



EZ-GUIDE-i		STOP		MEM		09:39:04	
ACTUAL POS. (ABS.)		DIST TO GO		SPINDLE		S1	
X	100.000	G01		S	0	O	5017
Y	-100.000	X	0.000			N	05016
Z	-20.000	Y	0.000			T	0
		Z	0.000			D	9 H 0
				FEED	MM/MIN	S	0 M 99
				F	0	F	200.000
						G01	17 40 54 80
						G49	90 98 69 13.1

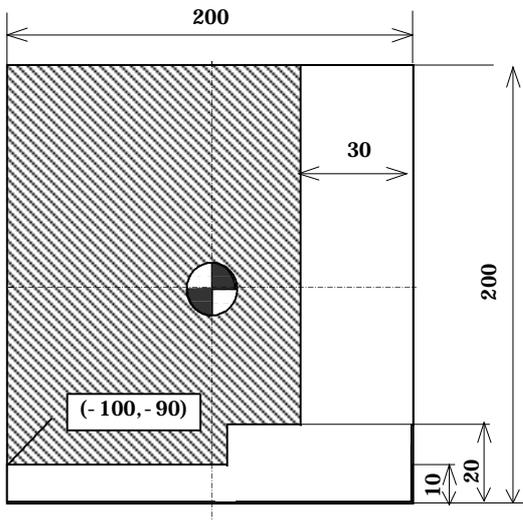
SIMULATE-ANIMATE		05017		CHAR←→	
		5 G1200 T5. H-100. V-90. B0. L-20. P1. ;			
		6 G1201 H100. V-90. K1. C100. L0. M0. T1. ;			
		7 G1206 ;			
		8 M30 ;			
		9 %			

G1030: CONTOURING(ROUGH)

REWIND	START	PAUSE	SINGLE	STOP	INIT	CUTDSP	INTERF	TLPATH	GRPOFF
--------	-------	-------	--------	------	------	--------	--------	--------	--------

<Fig.55 Continuous flow of Partial Contouring>

In case of rather complicated shape of step, additional consideration regarding **Side Thickness**, **Cutting Area** and **Start Point** parameters. Please compare the steps described in **Fig.55** with the following sequence. For this example, down cut is applied to roughing for saving power and tool life, and up-cut to finishing for better surface.



**Dimension : 200\*200\*50mm**

**Cut Depth Axis : 20mm**

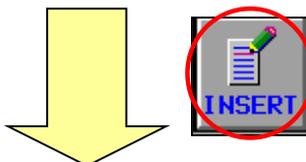
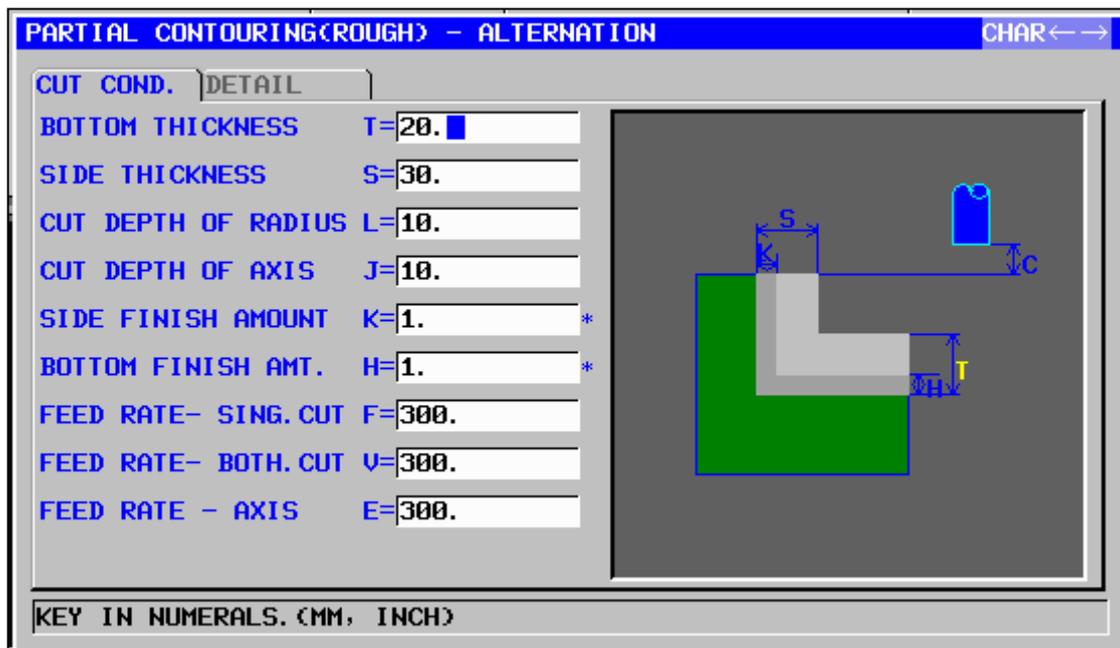
**Z Coordinate of Top Surface : 0**

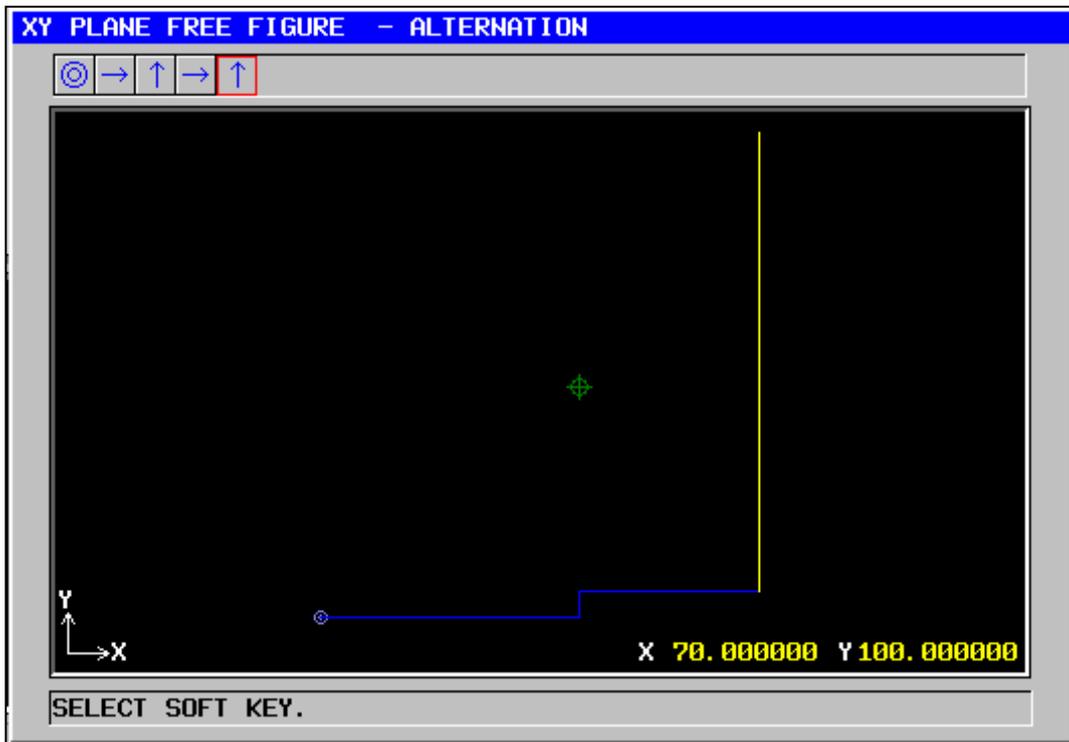
**Rough Endmill = D20 / 2flute**

**Bottom Fin. Endmill = D16 / 3flute**

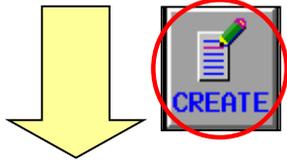
**Side Fin. Endmill = D10 / 4flute**

► **So the final R of inner corner is to be 5mm.**





Free Figure saving in the Subprogram directory



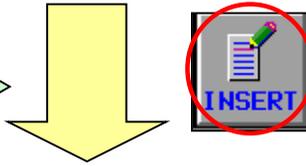
**PARTIAL CONTOURING<BOTTOM FINISH> -**

CUT COND.	DETAIL
BOTTOM THICKNESS	T=1.
SIDE THICKNESS	S=30.
<b>SIDE FINISH AMOUNT</b>	<b>K=1.</b>
BOTTOM FINISH AMT.	H=
CUT DEPTH OF RADIUS	L=10.
FEED RATE- SING. CUT	F=300.
FEED RATE- BOTH. CUT	V=300.
FEED RATE - AXIS	E=300.

KEY IN NUMERALS. (MM, INCH)

It should be set for side finishing operation.

The same figure calling from Subprogram directory

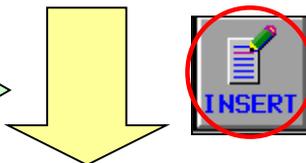


**PARTIAL CONTOURING(SIDE FINISH) - INSE**

CUT COND.	DETAIL	
SIDE THICKNESS	S=	1.
SIDE FINISH AMOUNT	K=	*
NUMBER OF FINISHING	B=	*
FEED RATE- SING. CUT	F=	300.
FEED RATE- BOTH. CUT	V=	300.
FEED RATE - AXIS	E=	300.

It should be set for the remained side finishing amount to be removed.

The same figure calling from Subprogram directory

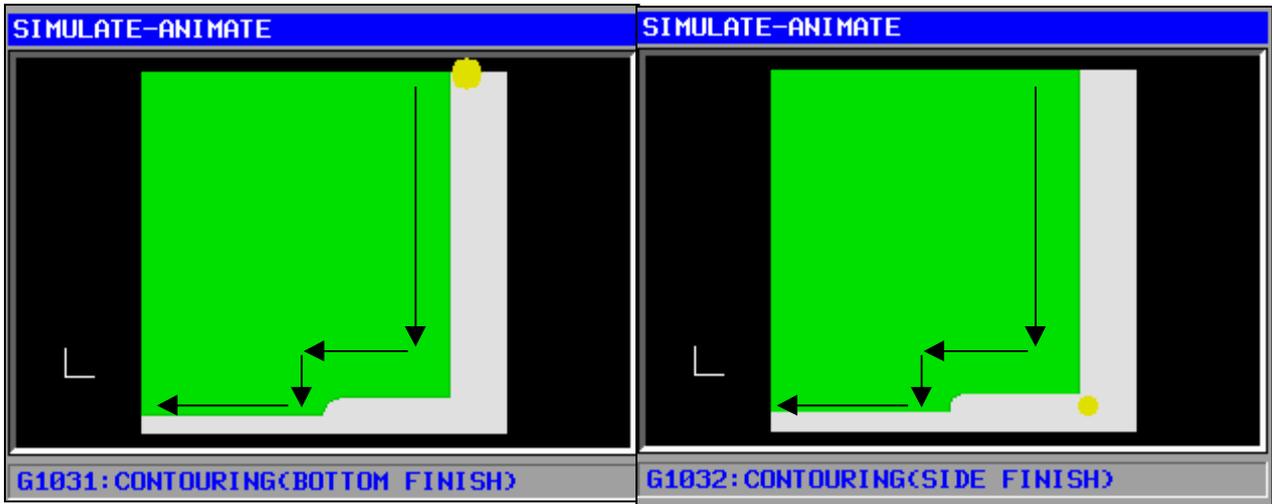


**SIMULATE-ANIMATE**

G1030: CONTOURING(ROUGH)

You can see the actual tool path. (Down Cut / Right side removal)

Operators do not have to change the property of figure but have to modify the cycle cut condition if cutting direction is in need of alternation from up/down to down/up cut.



EZ-GUIDE-i		STOP		MEM 18:34:05	
ACTUAL POS. (ABS.)		DIST TO GO		SPINDLE S1	
X	75.000	G01	X 0.000	S	0
Y	-85.000	X	Y 0.000		0%
Z	-20.000	Z	Z 0.000	F	0
				FEED	MM/MIN
					0%
					0 8020
					N 05020
					T 0
					D 22 H 0
					S 0 M 99
					F 300.000
					G01 17 40 54 80
					G49 90 98 69 13.1

SIMULATE-ANIMATE		O8020		CHAR←→	
		5 G1201 H70. V-80. K1. C70. L0. M0. T1. ;			
		6 G1201 H70. V100. K3. D100. L0. M0. T1. ;			
		7 G1206 ;			
		8 M99 ;			
		9 %			
G1032: CONTOURING<SIDE FINISH>					

REWIND	START	PAUSE	SINGLE	STOP	INIT	CUTDSP	INTERF	TLPATH	GRPOFF
--------	-------	-------	--------	------	------	--------	--------	--------	--------

<Fig.56 Continuous flow of complicated Partial Contouring>

**DOOSAN INFRACORE EZ Guide-i Revision Record**

Edition	Date	Contents