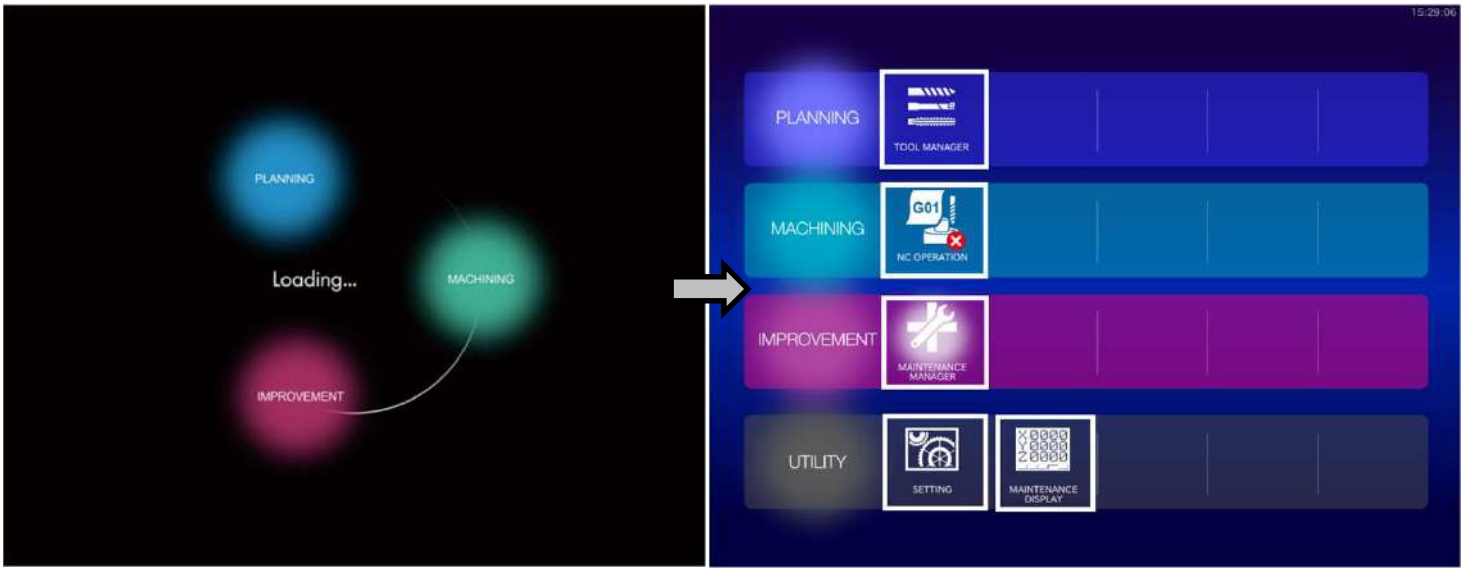
FANUC Series 0*i*-MF Plus

Operator's Handbook For Adv Mill

Table of Contents

1.1 Adv Mill Home Screen & iHMI Screen Details :-

Power-on → Splash screen → Home screen



Allows you to centralize tool information required at production sites.

Allows you to edit, change setups, and execute machining programs in the CNC

Collects CNC and tool machine information, allowing you to manage maintenance information, such as consumable life management and abnormality detection.

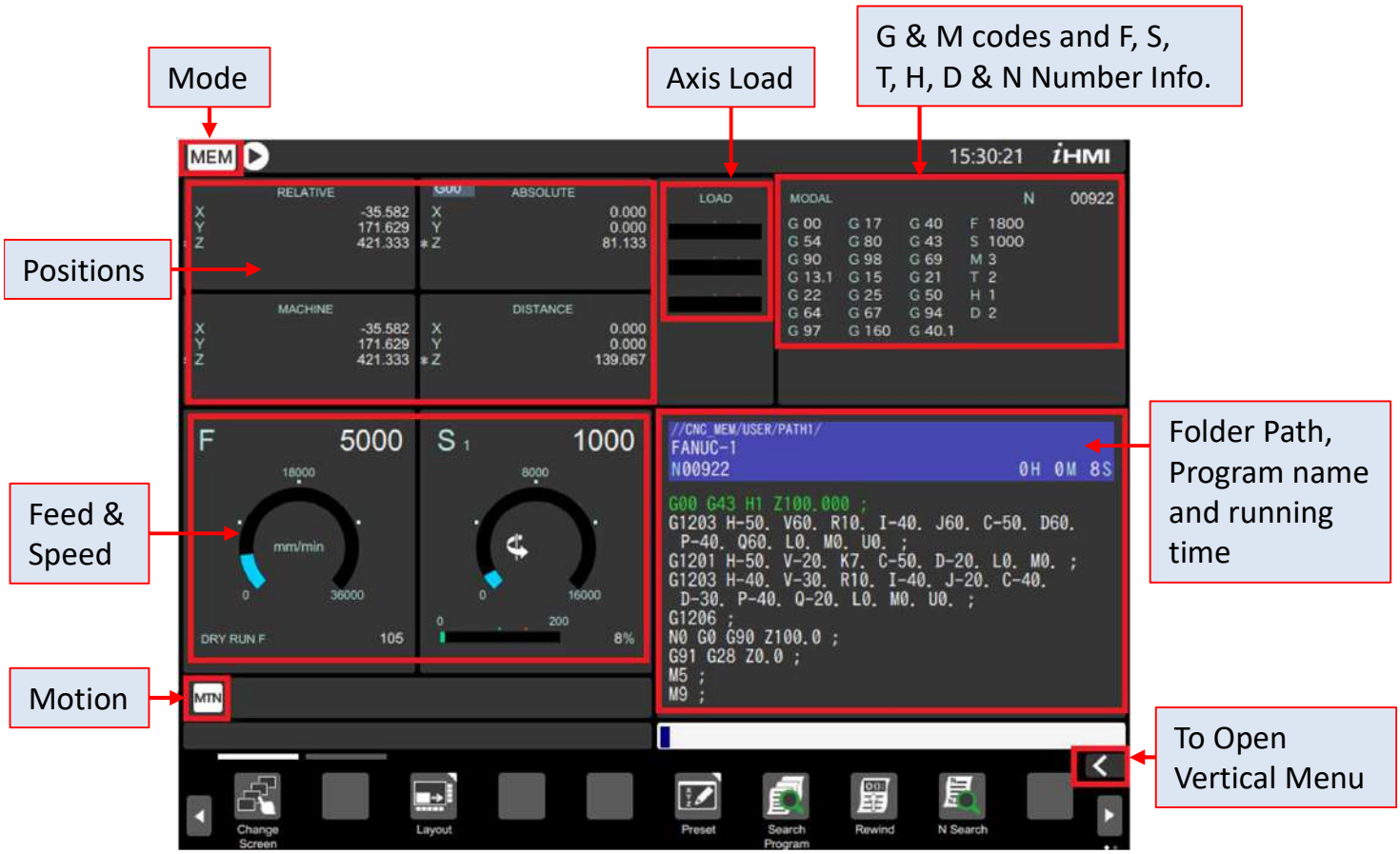
Allows you to configure the overall iHMI settings.

Display the conventional CNC screen

1. Press **MAIN MENU** Key on MDI Panel to change screen from standard Screen to Home Screen

➢ iHMI Screen Details (Auto Mode) :-

1. At Home Screen Touch **NC OPERATION** to open iHMI Screen.



➢ iHMI Screen Vertical Soft Key Details :-

1. Touch **Lower Right Arrow**  to open Vertical Soft Keys. (iHMI Options/Features)

Information Center, Alarms

Setting Slide

Workpiece Coordinates

Tool Offset

Program Manager

Custom Macro Variable

Utility

SCAN TO WATCH

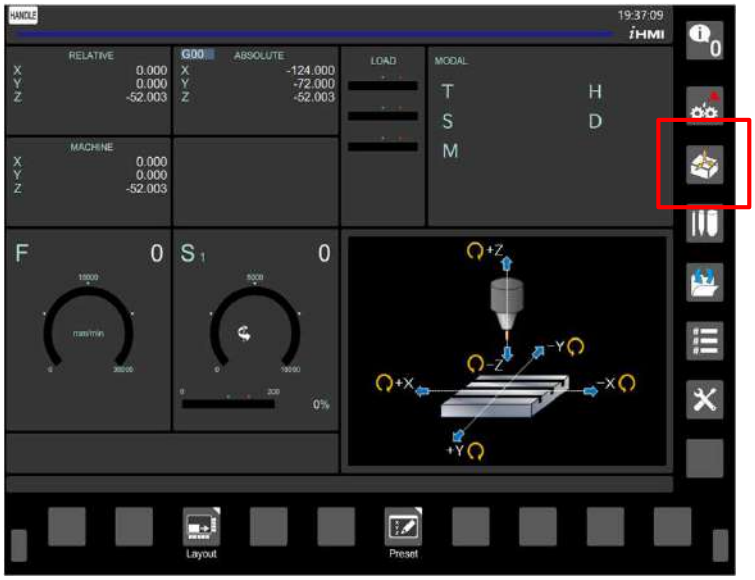
1.2 Work and Tool Offset :-

➢ Z Work offset Procedure without tool length Offset :-

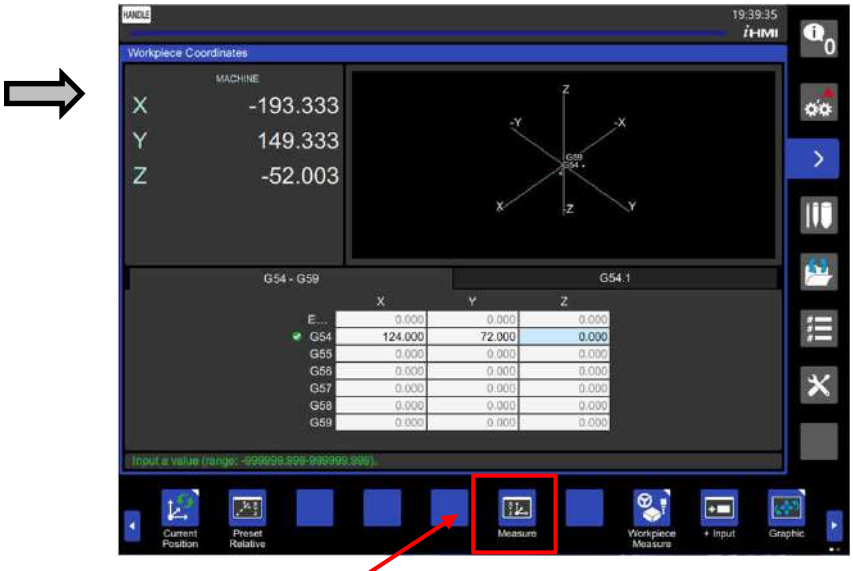
1. Select **HANDLE** mode ⇨

2. Touch Tool on Top Surface of Job

3. Select **WORKPIECE COORDINATES** vertical soft key ⇨



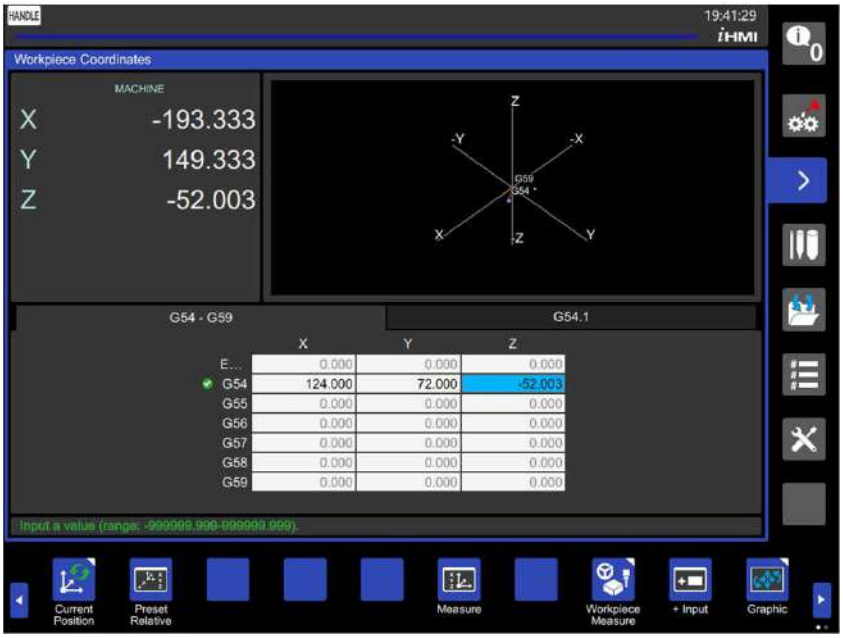
4. Place the cursor on **Z axis** at the required work coordinate number (e.g.G54)



5. Press **Measure** horizontal soft key

6. Type **0** in **Target** and check Z-axis machine position and **Result** values are same.

7. Press **OK** soft key.
8. Make sure that Z-axis machine position has registered in G54 Z-value are same

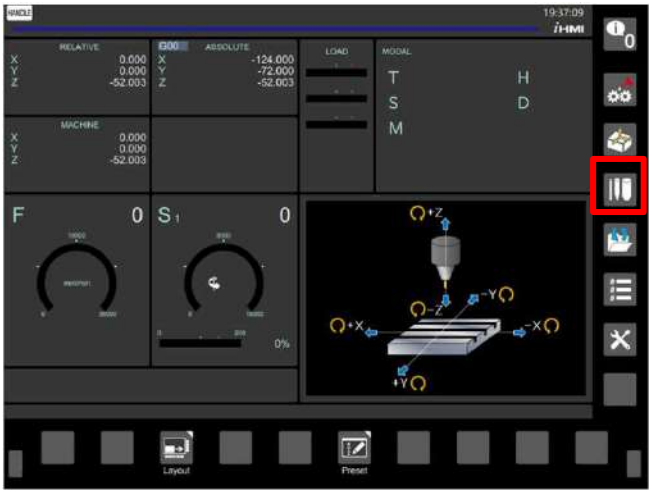


SCAN TO WATCH

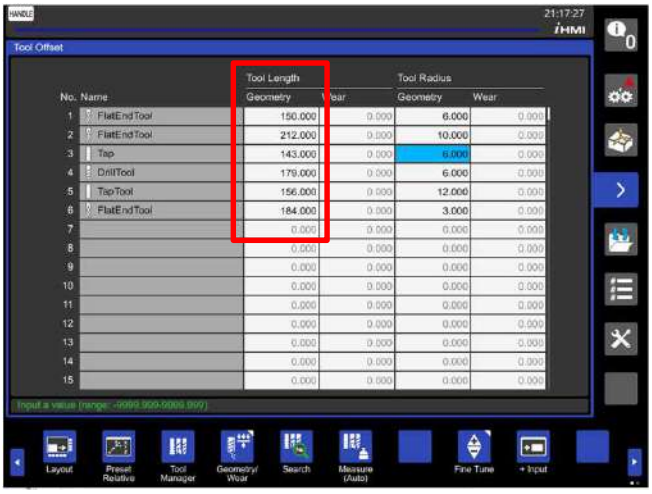
➢ Z Work offset Procedure using tool length Offset :-

1. Measure required Tool height by using Pre-setter.

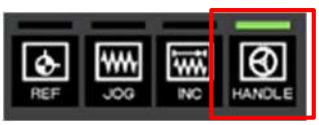
2. Select **TOOL OFFSET**  vertical soft key.



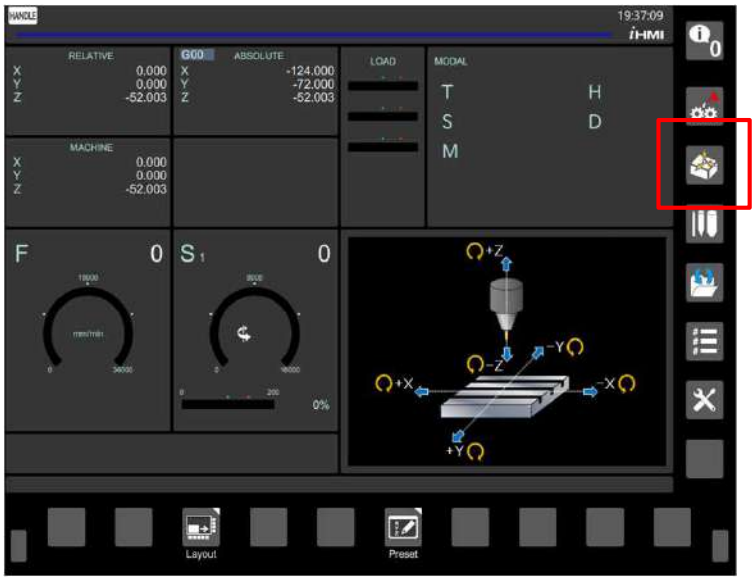
3. Enter **measured tool length** value in respective offset number in tool length geometry column.



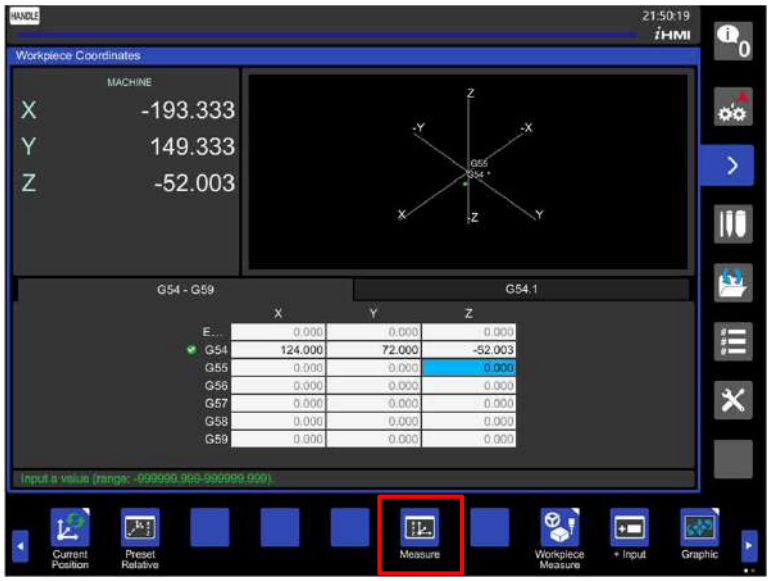
4. Select Handle mode & Touch the surface of the component where you want to make work zero (e.g. Top surface of workpiece) in Z – axis by any of the tool which we measured. (e.g. T3 =143.0 mm)



5. Select **WORKPIECE COORDINATES** vertical soft key

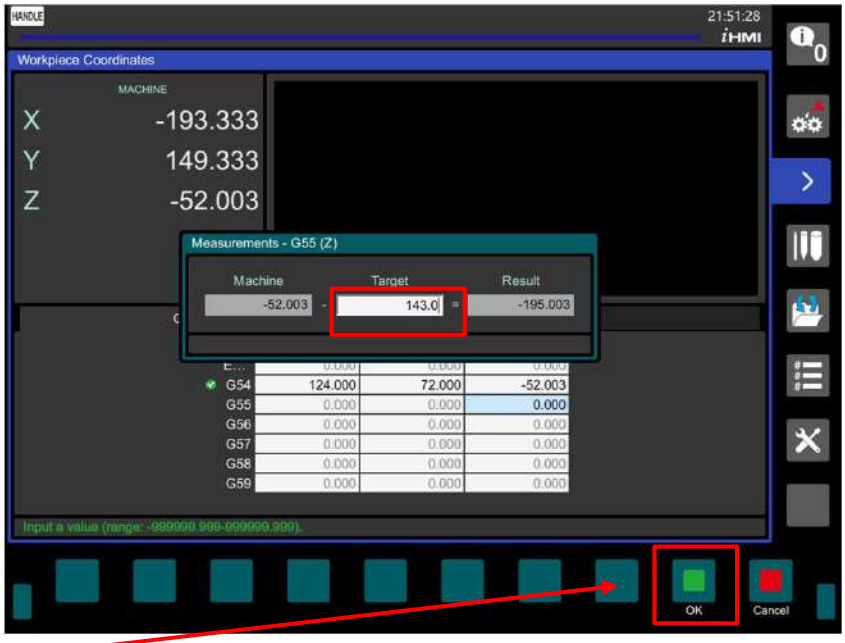


6. Place the cursor on **Z axis** at the required work coordinate number (e.g.G55)



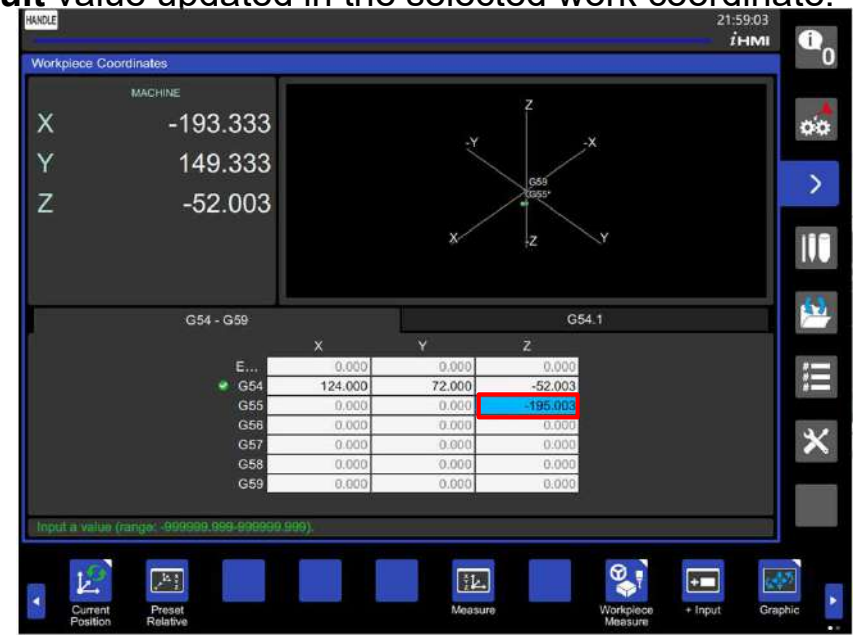
7. Press **Measure** horizontal soft key

8. Enter Tool Length value mentioned in tool offset table (e.g. here for tool no 3 , **T3=143.0)** in **Target.**



9. Press **OK** soft key.

10. Make sure that **Result** value updated in the selected work coordinate.



SCAN TO WATCH

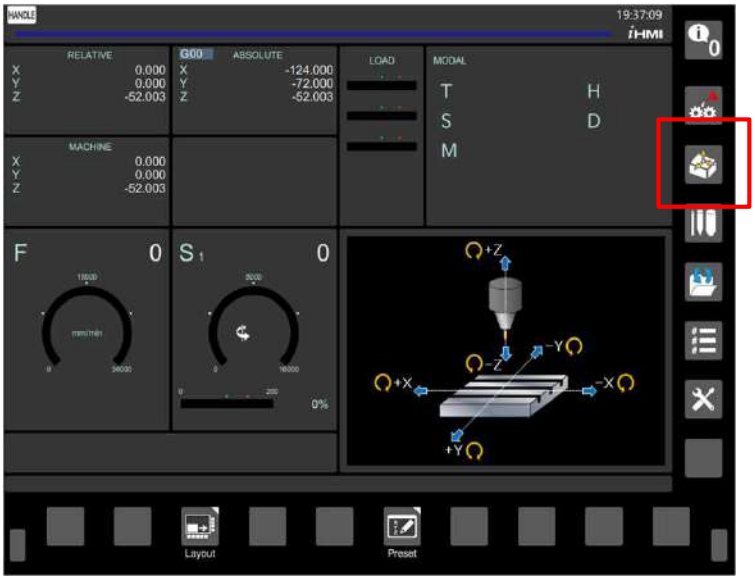
➢ Corner Work Offset (X and Y) Procedure :-

+X Face Offset:

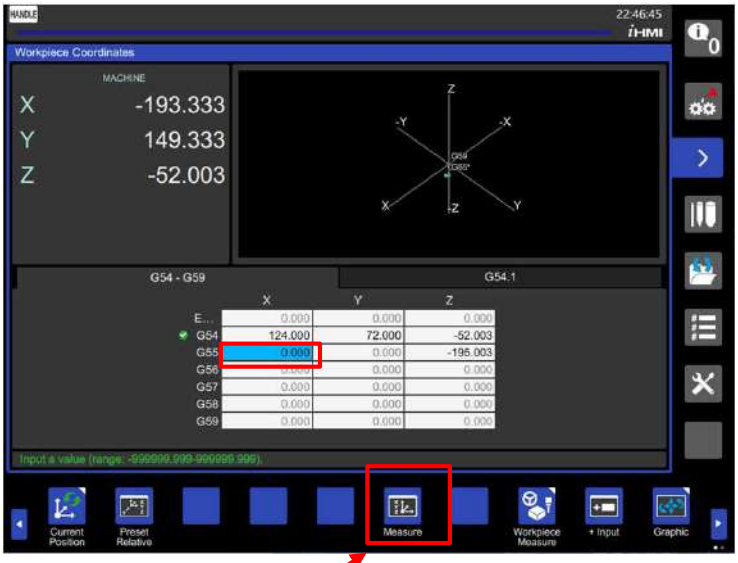
1. Select **HANDLE or JOG** mode

2. Touch Tool on +X axis surface at the workpiece

3. Select **WORKPIECE COORDINATES** vertical soft key

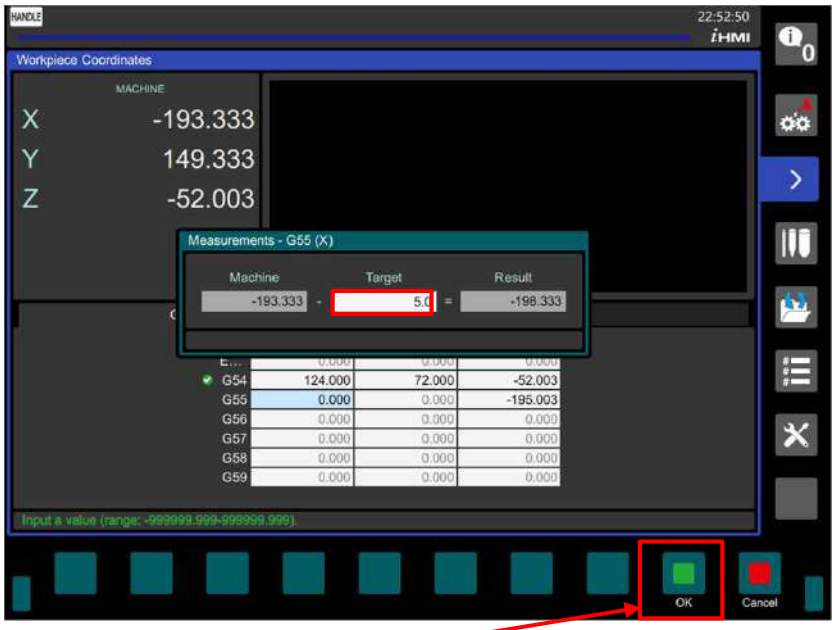


4. Place the cursor on **X axis** at the required work coordinate number (e.g.G55)



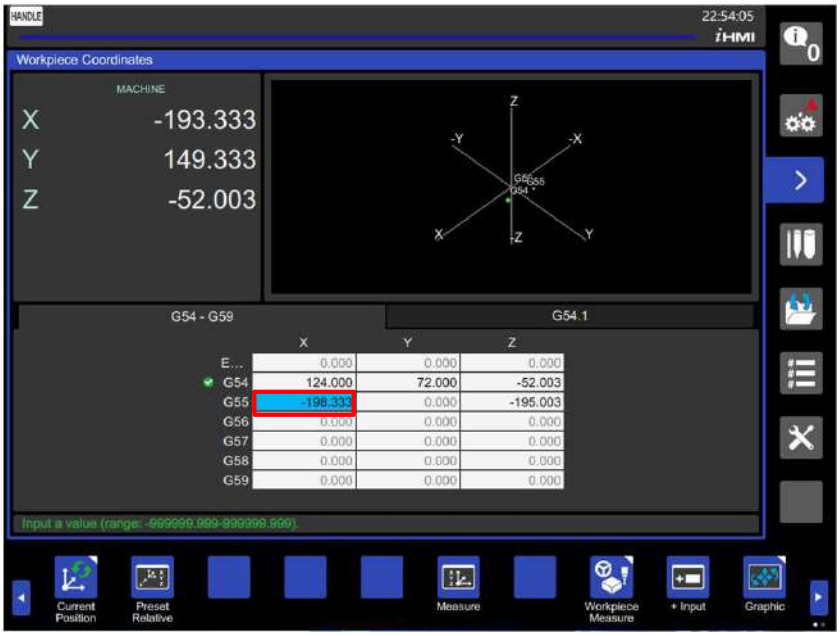
5. Press **Measure** horizontal soft key

6. Type Tool radius value (e.g.Tool Radius = 5.0 mm) in **Target** - Tool radius should be negative (-) while touching –X direction and positive touching in +X direction.



7. Press **OK** soft key.

8. Make sure that **Result** value updated in the selected work coordinate.

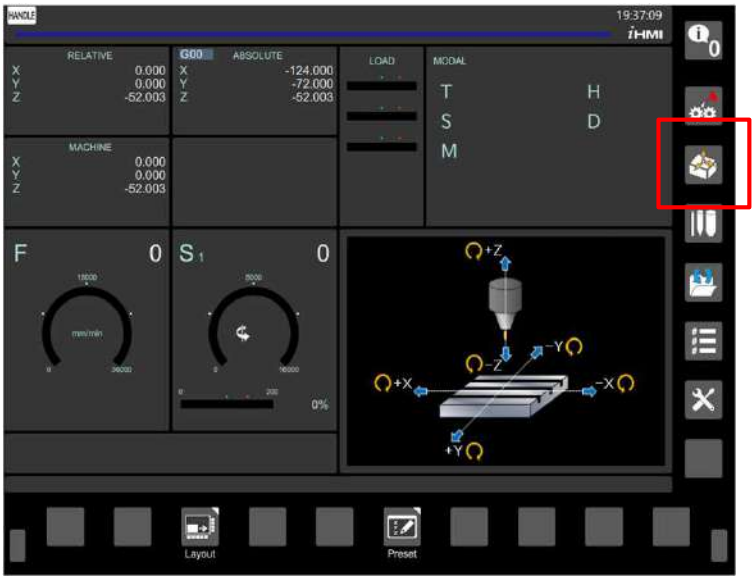


+Y Face Offset:

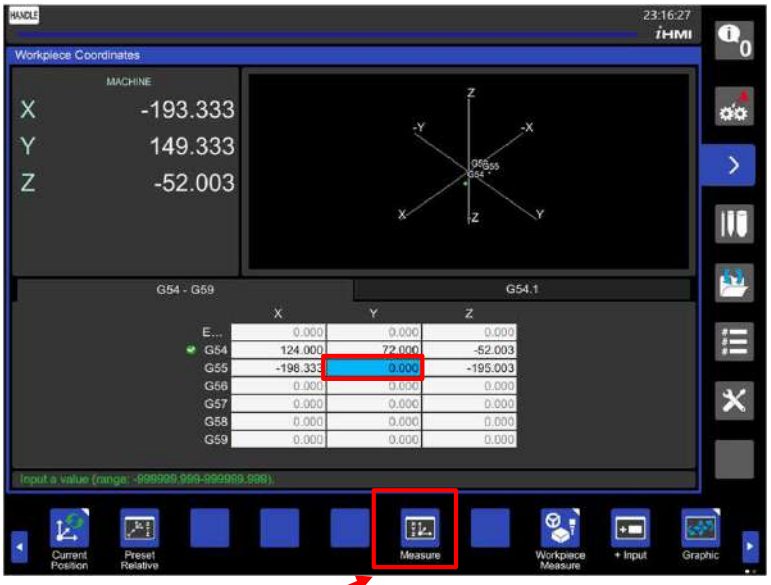
1. Select **HANDLE or JOG** mode

2. Touch Tool on +Y axis surface at the workpiece

3. Select **WORKPIECE COORDINATES** vertical soft key

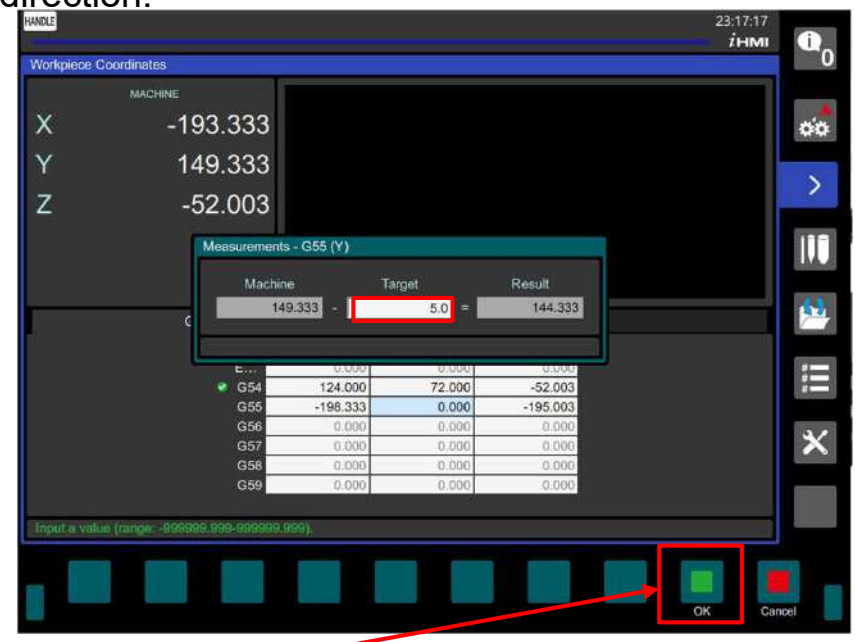


4. Place the cursor on **Y axis** at the required work coordinate number (e.g.G55)



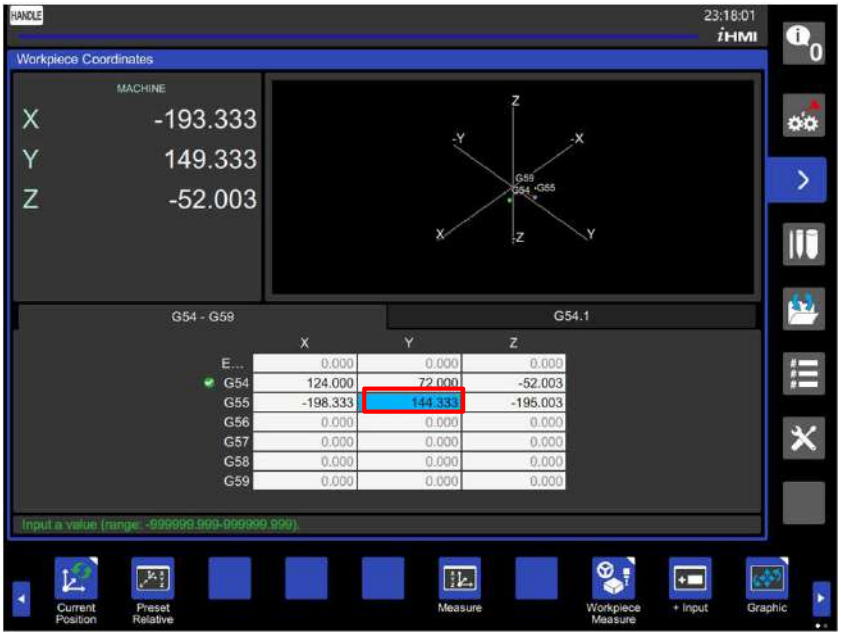
5. Press **Measure** horizontal soft key

6. Type Tool radius value (e.g Tool Radius = 5.0 mm) in **Target** - Tool radius should be negative (-) while touching –Y direction and should be positive value touching in +Y direction.



7. Press **OK** soft key.

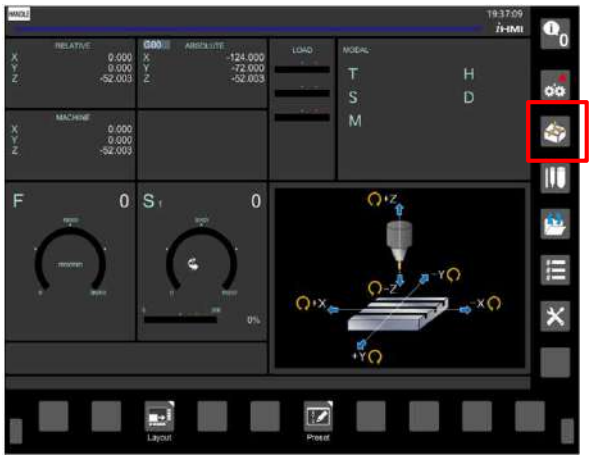
8. Make sure that **Result** value updated in the selected work coordinate.



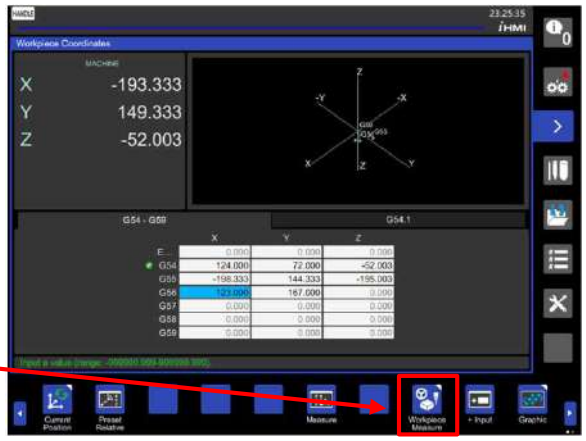
SCAN TO WATCH

➢ Four Point Centre of Square Job by using Setup Guidance

1. Select the **WORKPIECE COORDINATES** vertical soft key ⇨

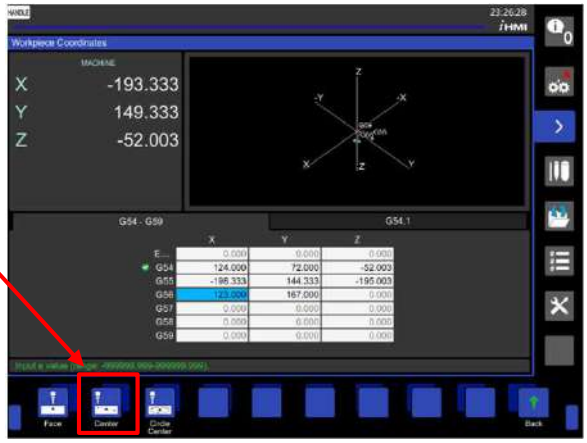


2. Place the cursor on required work coordinate and Press **WORKPIECE MEASURE** horizontal softkey ⇨

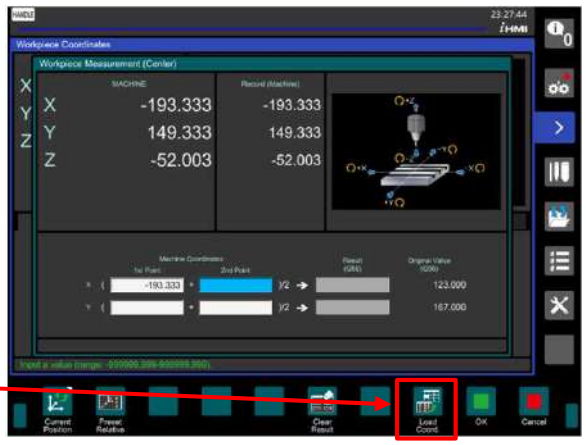


3. Select **CENTER** horizontal soft key

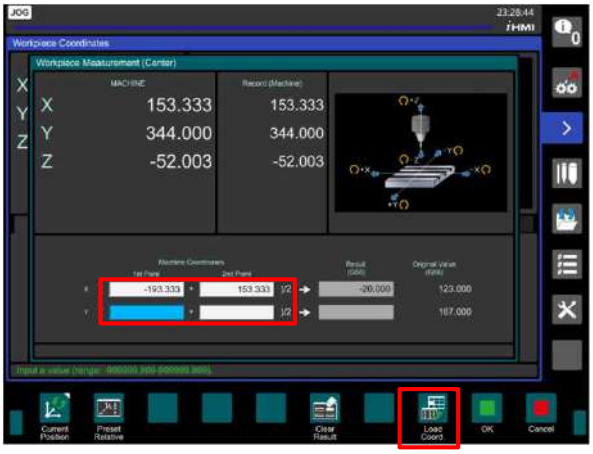
4. Select **HANDLE or JOG** mode



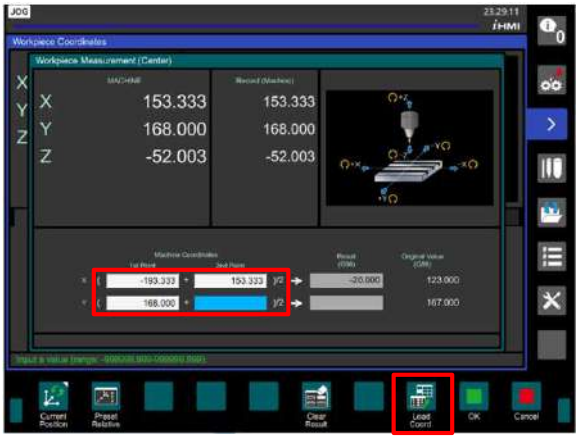
5. Place the cursor on 1st point in X axis. ⇨
6. Touch 1st point of the component.
7. Then press **Load Coordinates** soft key.



8. Place the cursor on 2nd point in X axis.
9. Touch 2nd point of the component.
10. Then press **Load Coordinates** soft key.
11. Now the center point of the X axis is measured and output in **Result** tab.



1. Then move to Y axis 1st point tab.
2. Touch 1st point of the component in Y axis in handle mode.
3. Then press **Load Coordinates** soft key.



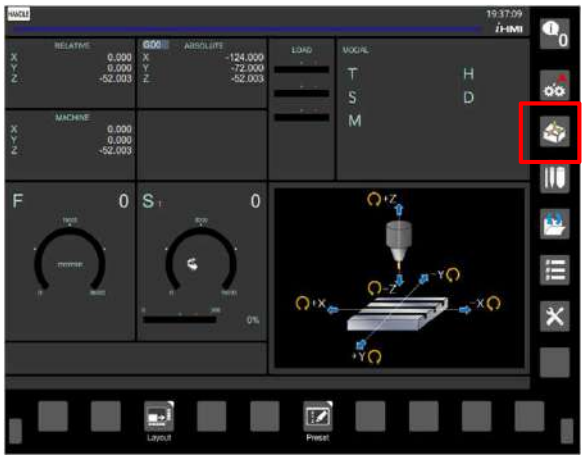
8. Place the cursor on 2nd point in Y axis.
9. Touch 2nd point of the component.
10. Then press **Load Coordinates** soft key.
11. Now the center point of the Y axis is measured and output in **Result** tab.

12. Press **OK** soft key to set the Result value in Work Coordinate.

SCAN TO WATCH

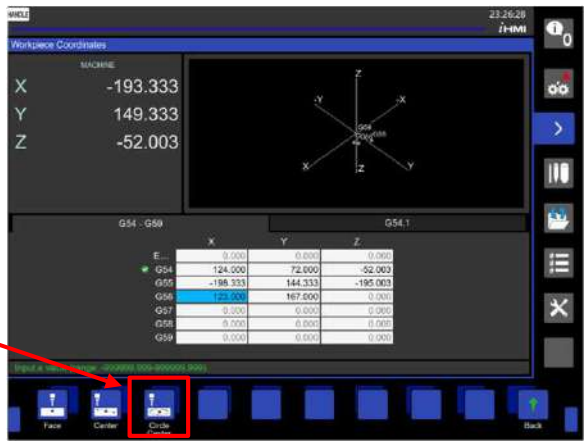
➢ Three Point Centre of Round Job by using Setup Guidance:

1. Select the **WORKPIECE COORDINATES** vertical soft key



2. Place the cursor on required work coordinate and Press **WORKPIECE MEASURE** horizontal softkey

3. Select **CIRCLE CENTER** horizontal soft key.



4. Press **Measure Number** soft key to change number of touch points (3 points or 4 points) based on the requirement.

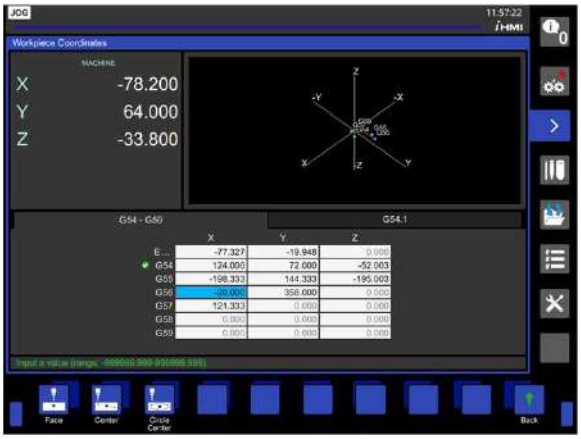
5. Select **HANDLE or JOG** mode

6. Place the cursor on 1st point.
7. Touch 1st point at the Round component.
8. Then press **Load Coordinates** soft key.
9. Now the current machine coordinate values has been recorded for Both X and Y axis.

10. Then move cursor to 2nd point.
11. Touch 2nd point at the component and press **load coordinates** soft key.

12. Like wise touch all points (Four or Three points) and press **Load Coordinates** soft key after touch.
13. After touching the final point, the **Result** point will be calculated.

14. Then press **OK** soft key to update result value in selected work coordinate.



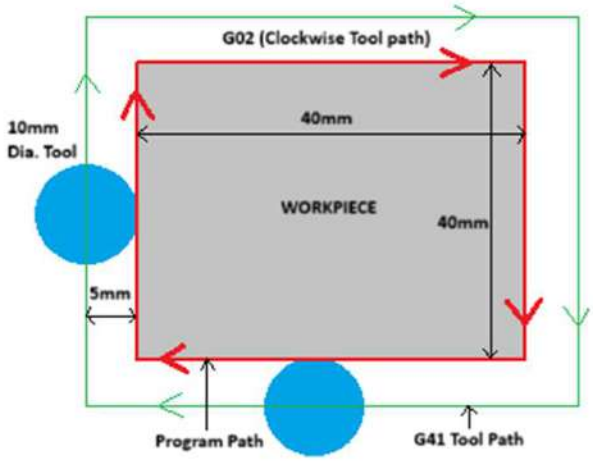
SCAN TO WATCH

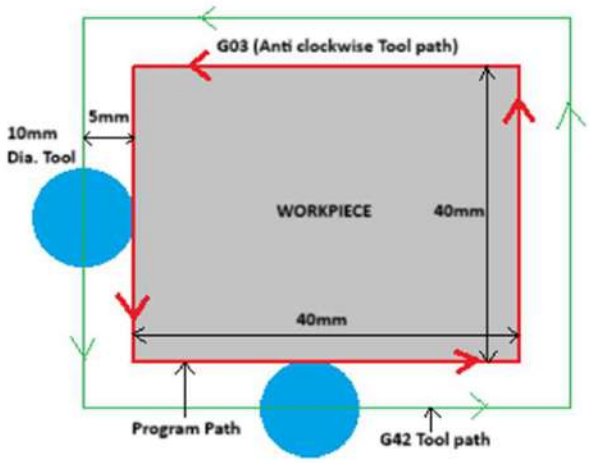
➢ Selection of G41/G42 (Cutter Radius Compensation) :-

1. Use G41 or G42 (Cutter Radius Compensation Left or Right) in Program depending on Tool movement Direction along with ‘D’ number and use G40 to cancel Cutter Radius compensation.
2. Put Cutter Radius in **GEOM (RADIUS)** Column of **OFF/SET** Page.

❖ For Outer Machining (Contour) –

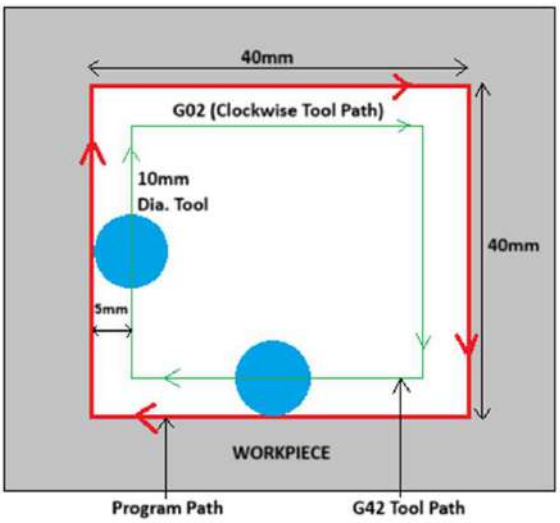
- If Cutter movement is Clockwise then use G41 in program
- If Cutter Movement is Counter clockwise then use G42 in program

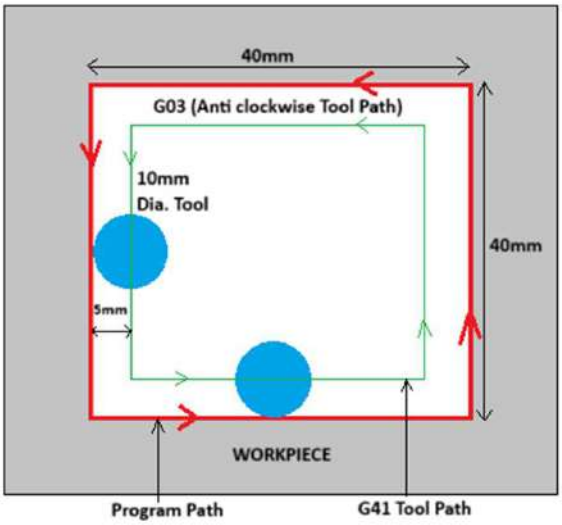




❖ For Inside Machining (Pocket) –

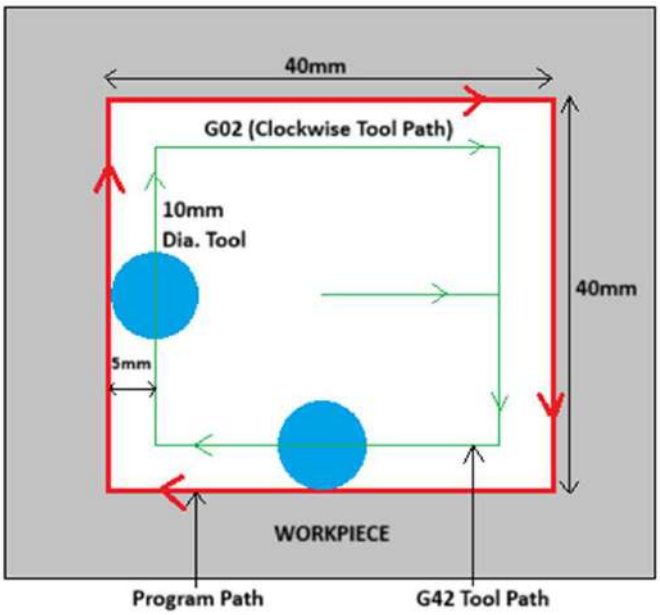
- If Cutter movement is Clockwise then use G42 in program
- If Cutter Movement is Counter clockwise then use G41 in program





❖ Inside Machining (Pocket) Drawing with Sample Program using G42 :–

```
O0001
G40 G80 G17
G91 G28 Z0.0
T01 M06
G0 G90 G54 X0 Y0
M3 S1500
G0 G43 H1 Z100.0
G01 Z-5.0 F2000
G01 G91 G42 D1 X20.0
G01 Y-20.0
G01 X-20.0
G01 Y20.0
G01 X20.0
G01 Y0.0
G01 G40 X0.0 Y0.0
G0 G90 Z100.0
G0 G91 G28 Z0.0
M5
M30
%
```



❖ **For Inside Machining (Pocket) –**
Cutter movement is Clockwise in Drawing so use G42 in program with D Number.
(D1 = Tool Radius Offset)

Tool Radius Wear

Tool Radius Offset (G42 D1)

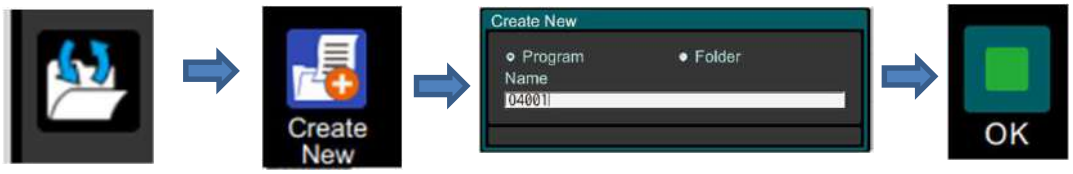
Tool Length Wear

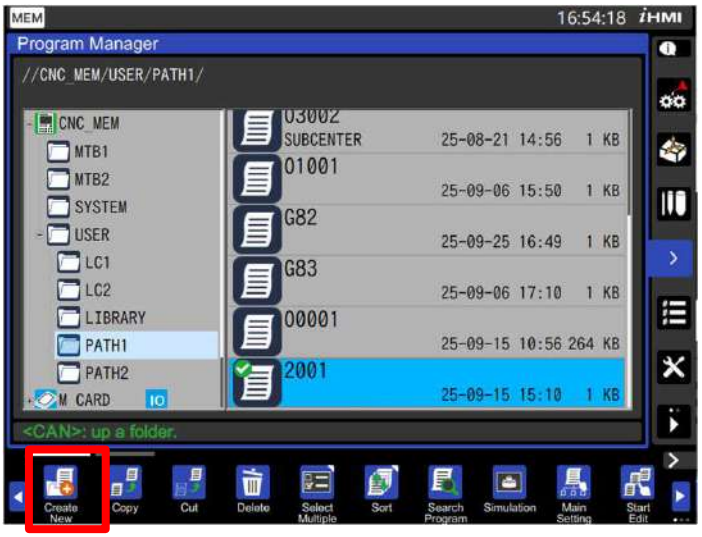
Tool Length Offset (G43 H1)

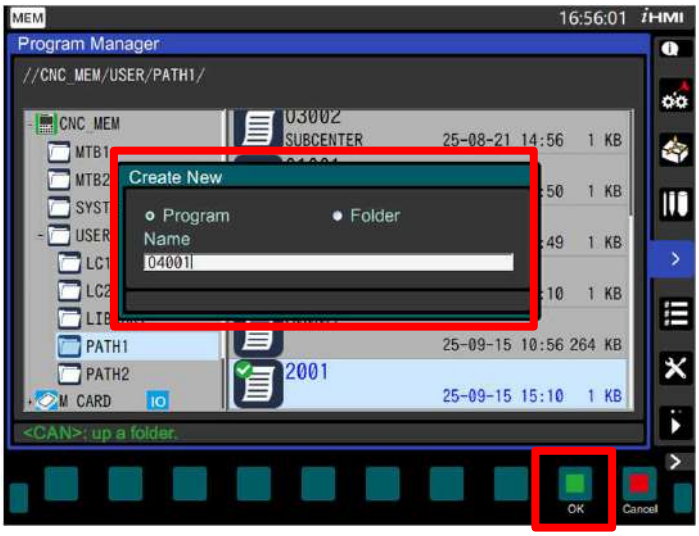
1.3 Program Editing Functions :-

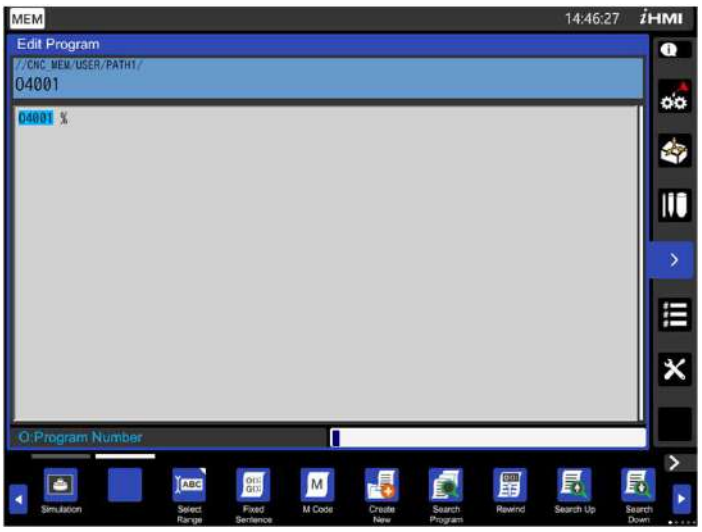
➢ New Program Create in CNC MEMORY :-

1. Press **Program manager** Vertical soft key
2. Select the folder where the program will be created, Press **Create new** soft key.
3. Select **Program** option
4. Type Number of Program (E.g. O4001) or Program Name (E.g. PART1)
5. Press **OK** soft key
6. After program creation Press **INPUT** key to open program for edit.





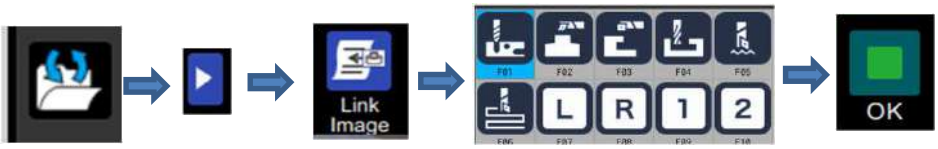


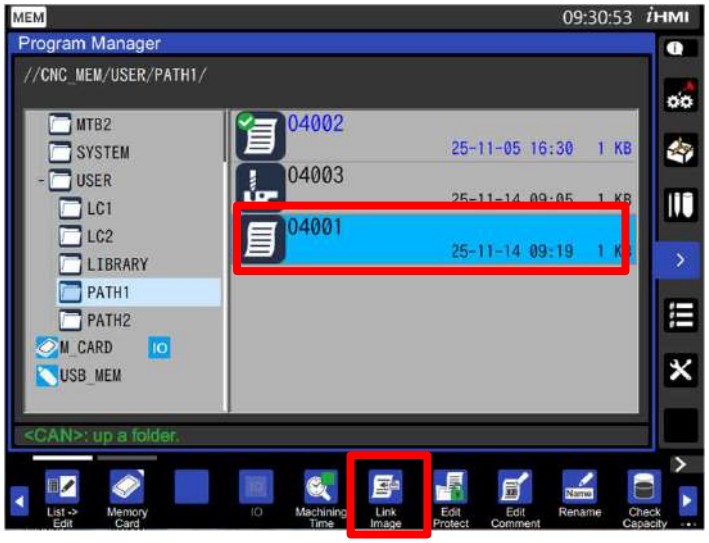


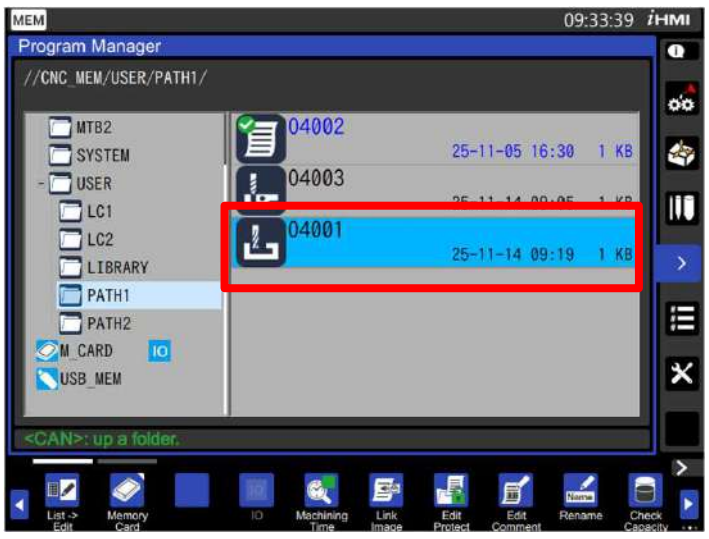
SCAN TO WATCH

➢ Link Image to Program :-

1. Press **Program Manager** Vertical soft key
2. Select the program in which image to be link.
3. Press right arrow soft key to screen change up to “**Link Image**” option.
4. Press “**Link Image**” soft key.
5. Select the option from where image to be link (internal memory, USB or memory card)
6. Select the image and press **OK.**
7. Image will be linked
8. For removing the existing linked image, No image logo to be linked with same procedure from internal memory.



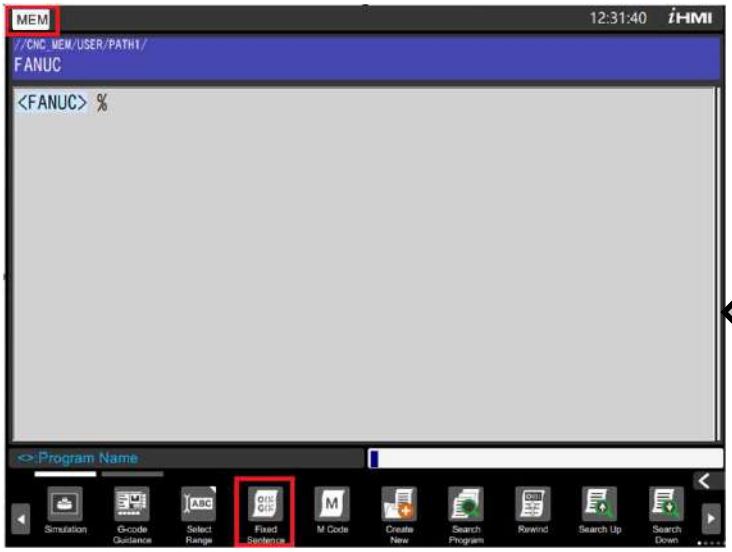




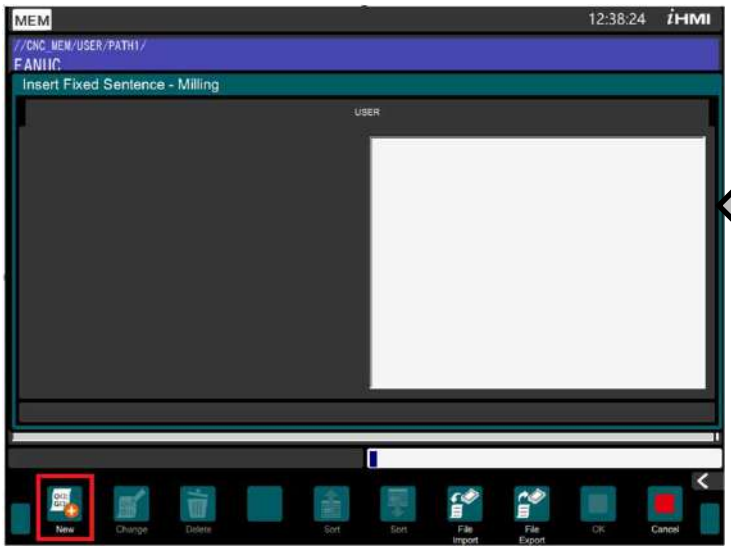
SCAN TO WATCH

➢ Use Fixed Sentence to Create/type Program Faster :-

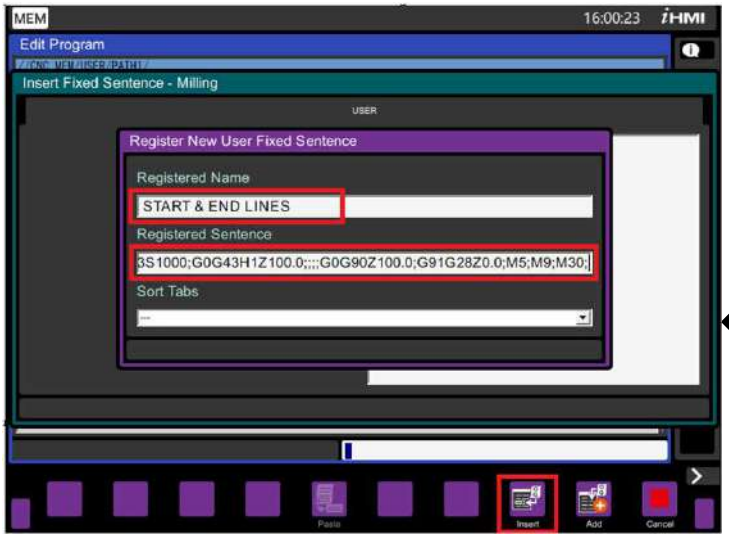
Note:- Programs lines can be typed manually or start and end lines (common used lines) can inserted directly using Fixed Sentence (Saved).



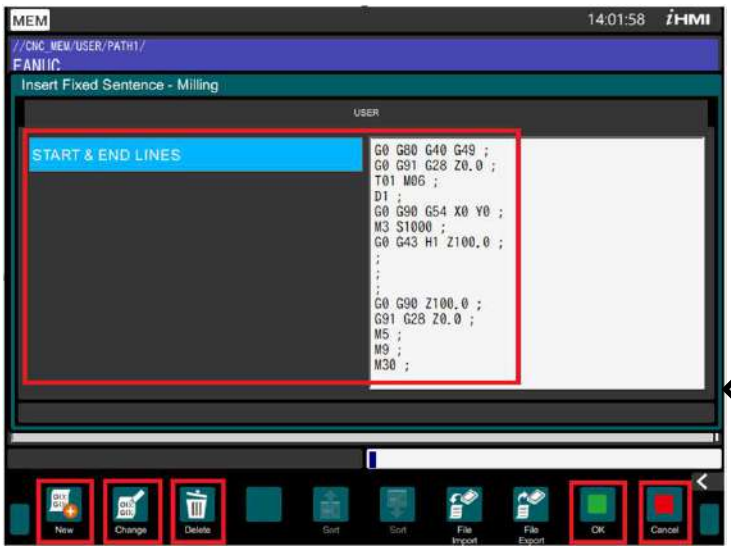
1. In Edit screen of any program (Select any Mode)
2. Touch **Fixed Sentence** soft key.



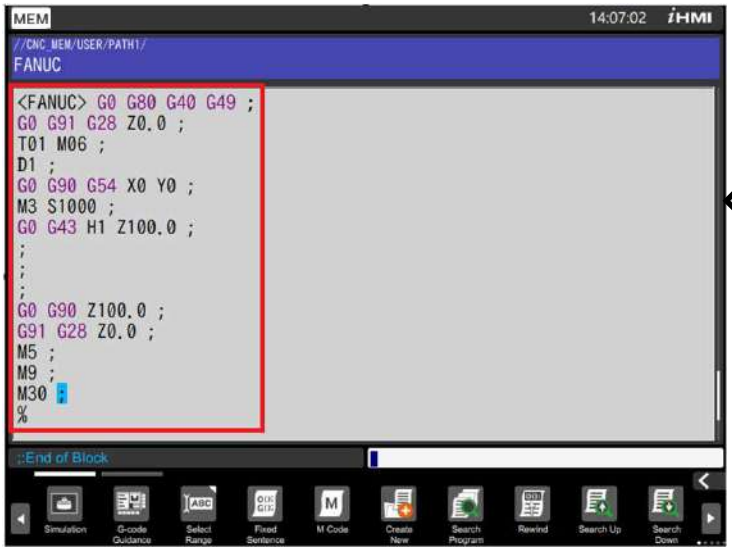
3. Touch **New** soft key.



4. Enter **Registered Name** (E.g. START & END LINES) And **Registered Sentence** (E.g. G0G40G80G49 etc.)
5. Then Touch **Insert** Soft Key



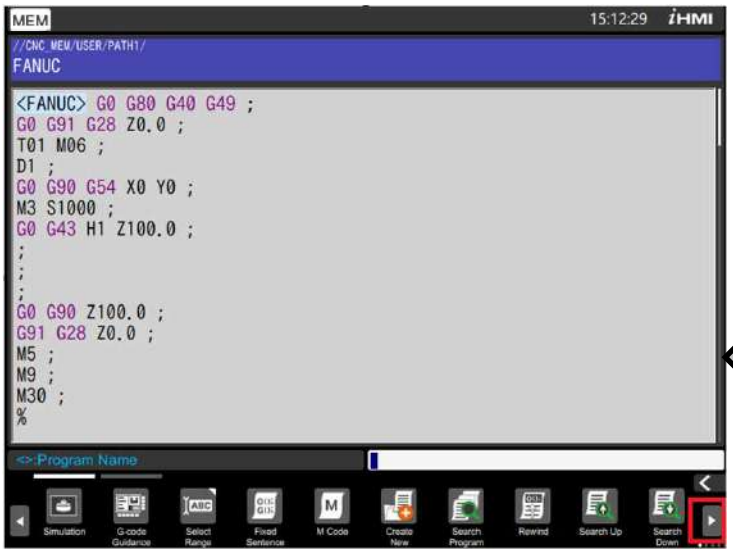
6. Fixed Sentence will Saved.
Use other Soft Keys like,
New to add another Fixed Sentence,
Change to edit existing Sentence
Delete to Remove Sentence
Cancel to Close Window
OK to insert Fixed Sentence in program



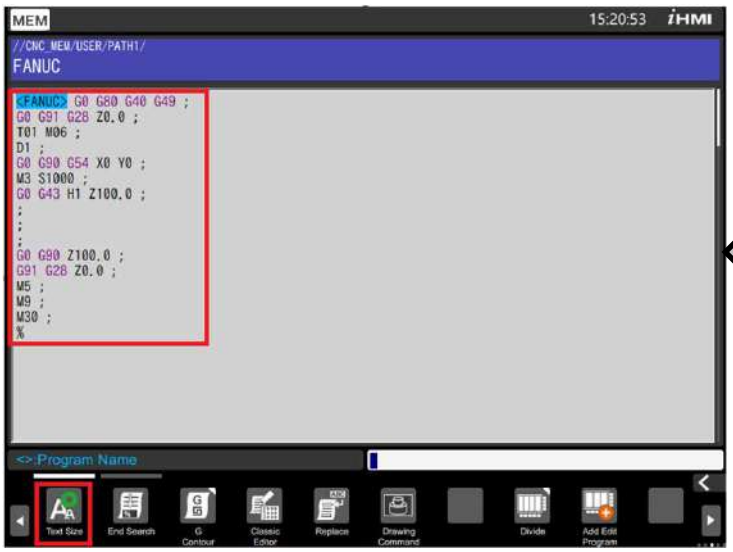
7. Touch **OK** soft key then Fixed Sentence will be inserted in Program.

SCAN TO WATCH

➢ Change Text Size of Program :-



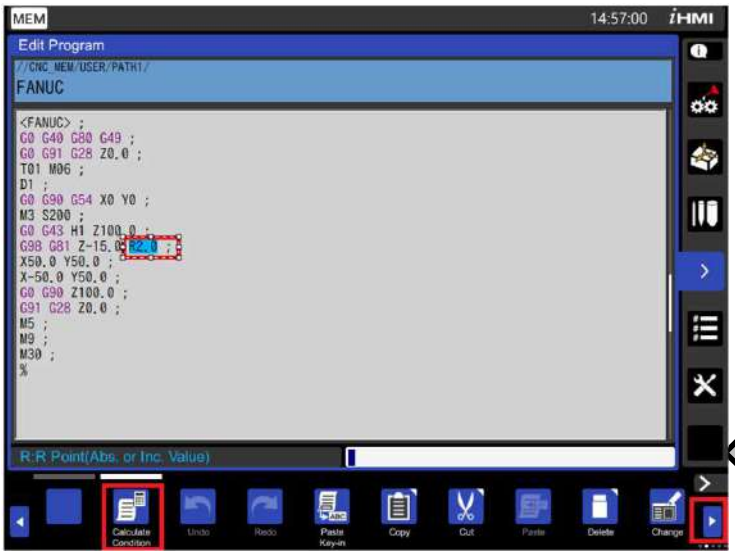
1. In Edit screen of any program (Select any Mode)
2. Touch **Right Next Page** soft key 2 times for **Text Size** Option.



3. Touch **Text Size** soft key Multiple to change Font size of program to Small, Medium or Large.

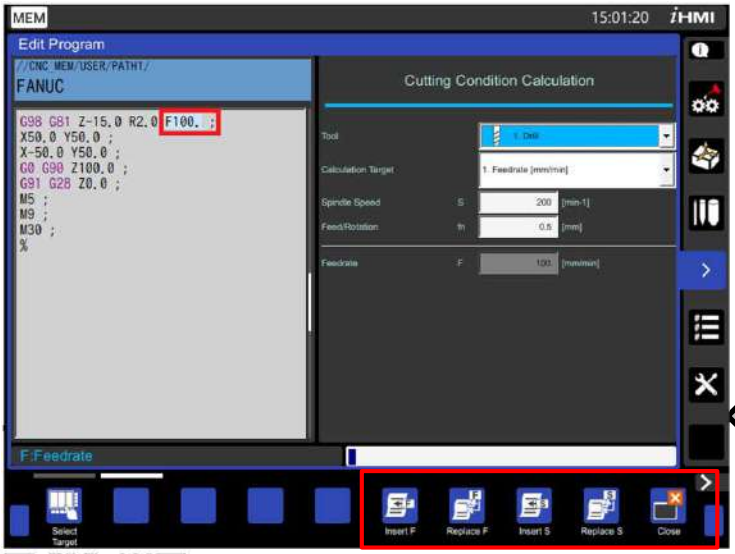
SCAN TO WATCH

➢ Use Inbuilt Cutting Condition Calculation :-



1. In Edit screen of any program (Select any Mode)
2. Keep cursor where you want insert calculated Feed.
2. Touch **Right Next Page** soft key for **Calculate Condition** Option.
3. Touch **Calculate Condition** to Open Cutting Speed/Feed Calculator

4. Select Tool (E.g. Drill) , Select Calculation Target (E.g. Feed in mm/min) Enter Spindle Speed (E.g. 200) & **INPUT** Feed/Revolution (E.g. 0.2) & **INPUT** It will Calculate Feedrate in mm/min
5. Calculation of Cutting Speed , RPM & Feed per Tooth also possible for various tools Like Face Mill , End Mill & Tap.



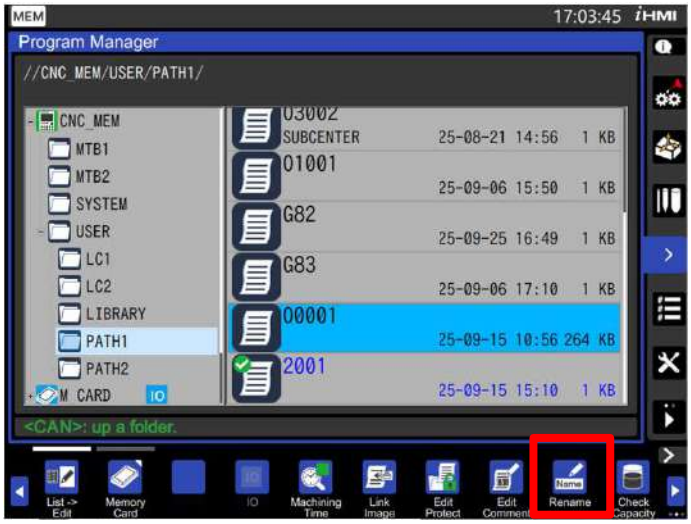
6. Touch **Insert F** soft key to insert calculated **Feedrate** in program.
7. Similarly **Replace F**, **Insert S** & **Replace S** soft keys can be used for Replace Feedrate, Insert Speed & Replace Speed respectively.
8. Touch **Cancel** to Close Window

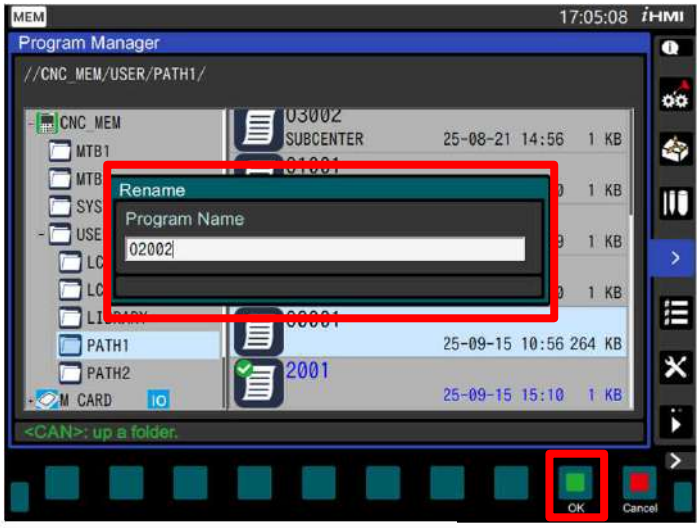
SCAN TO WATCH

➢ Program Rename :-

Note: All program can be **renamed** in all mode except **main program**. Main program can be renamed in **Edit mode** only

1. Press **Program manager** Vertical soft key
2. Keep Cursor on Program to Rename
3. Press Right arrow for next page
4. Press **RENAME** soft key
5. Type new name or Number (e.g. **O2002**)
6. Press **OK** Soft key



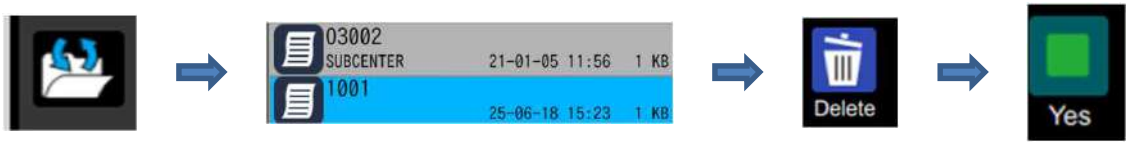


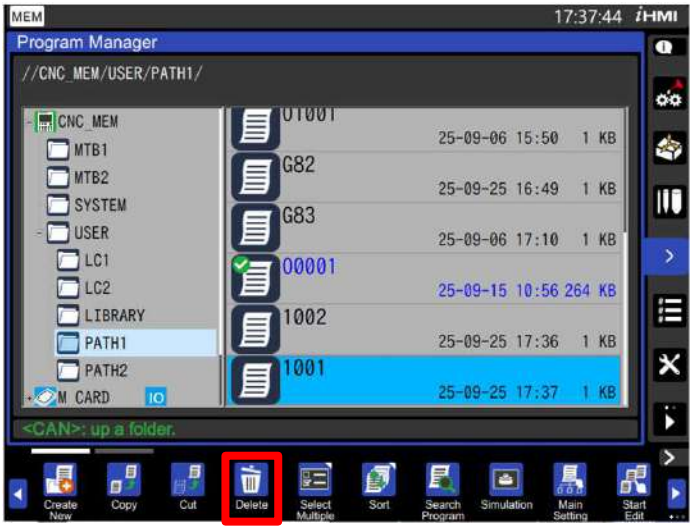
SCAN TO WATCH

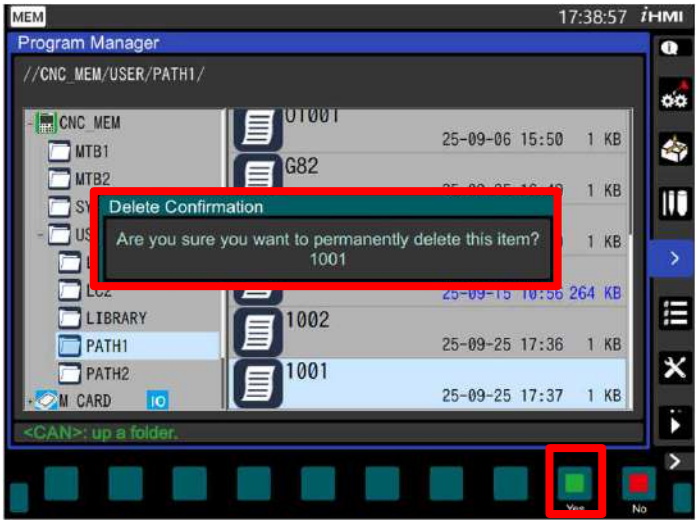
➢ Program Delete :-

Note: All program can be **delete** in all mode except **main program**. Main program can be **delete** in **Edit mode** only

1. Press **Program manager** Vertical soft key
2. Keep Cursor on Program to Delete
3. Press **DELETE** soft key (Popup Window come for Delete confirmation)
4. Press **YES** soft key



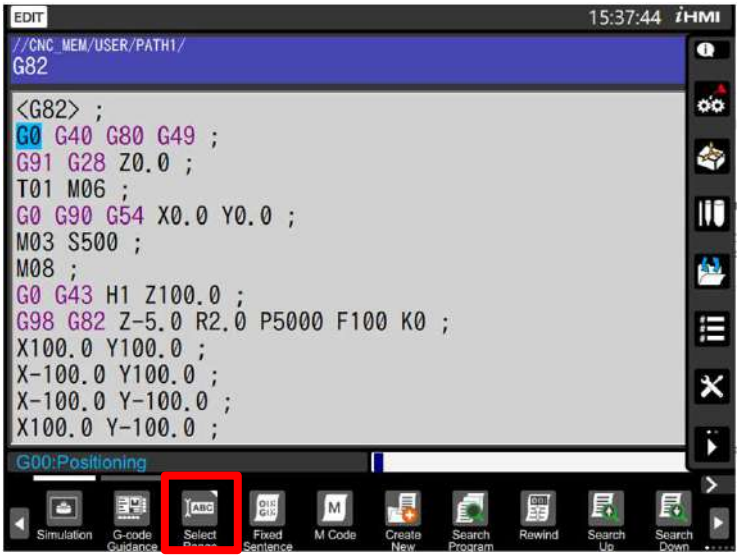




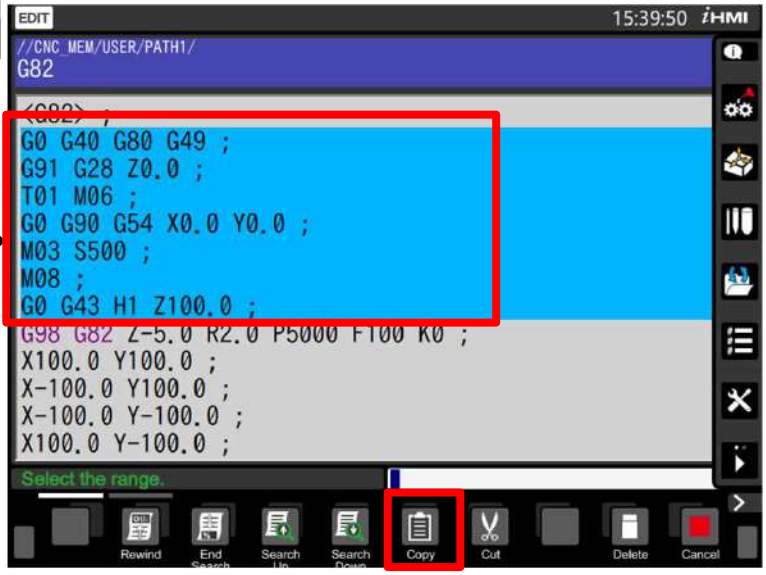
SCAN TO WATCH

➢ Program Lines Copy and Paste from Main program :-

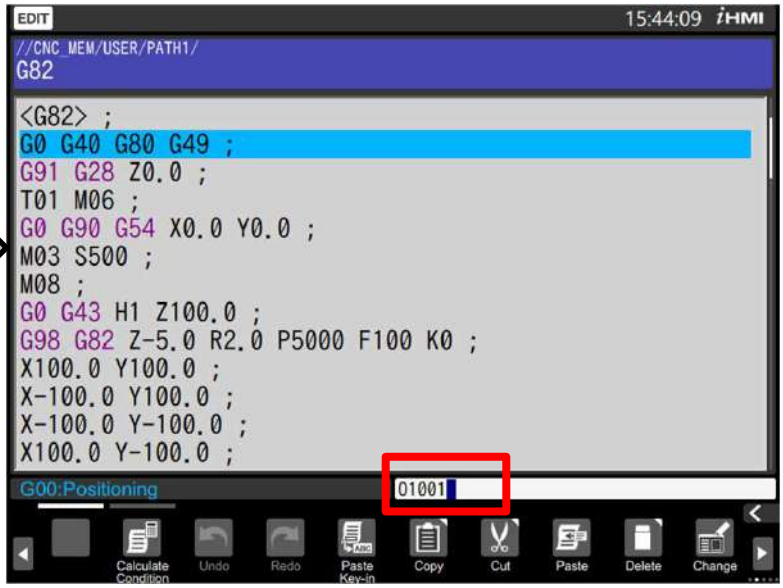
1. Select **EDIT** mode
2. Keep cursor on line from where to copy and Press **Select Range** soft key



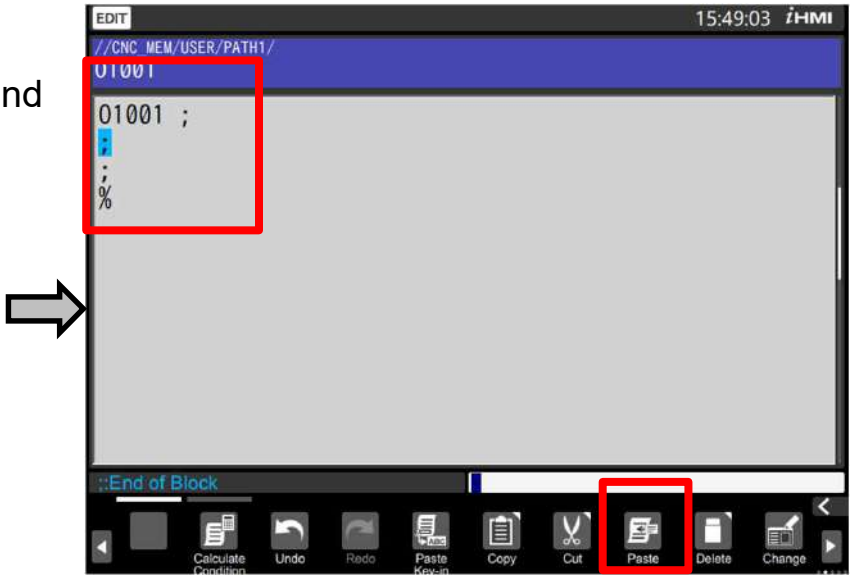
3. Press Continue **Down arrow** MDI key to select Multiple lines
4. Press **COPY** soft key

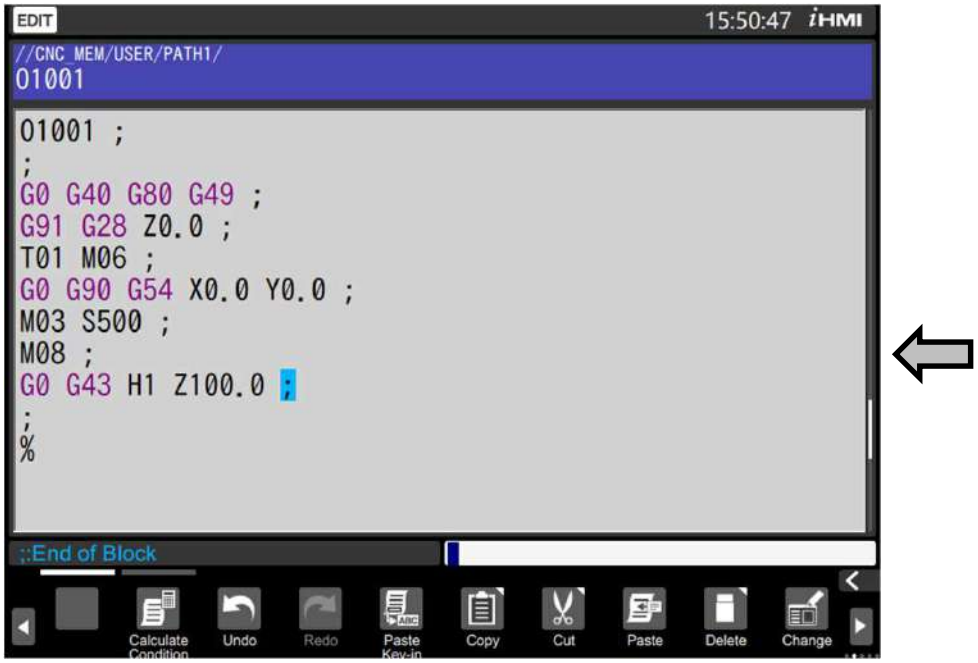


4. Type other program name and press **Down arrow** MDI key.
It will ask that this will change the main program, press **Yes** soft key



5. Keep cursor anywhere and Press **PASTE soft key**

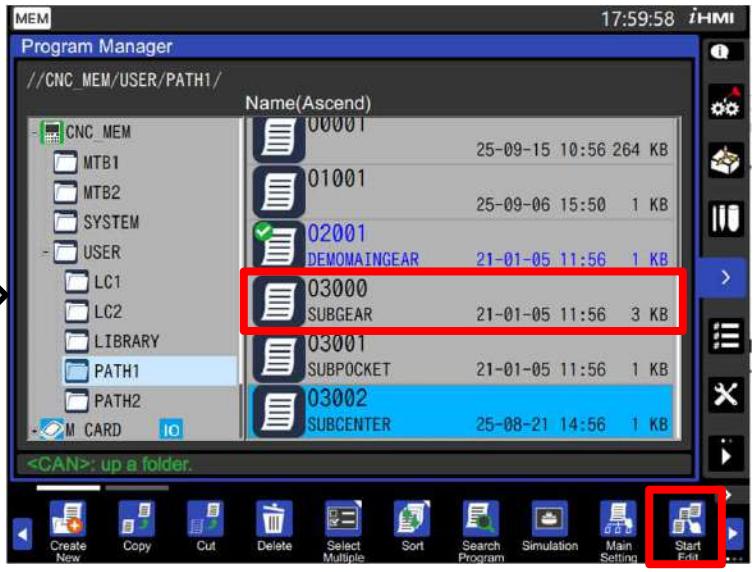




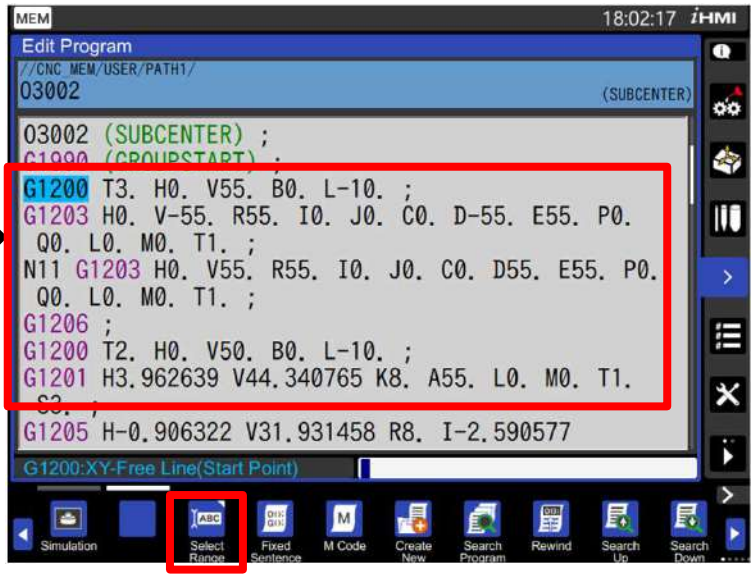
SCAN TO WATCH

➢ Program Lines Copy and Paste:-

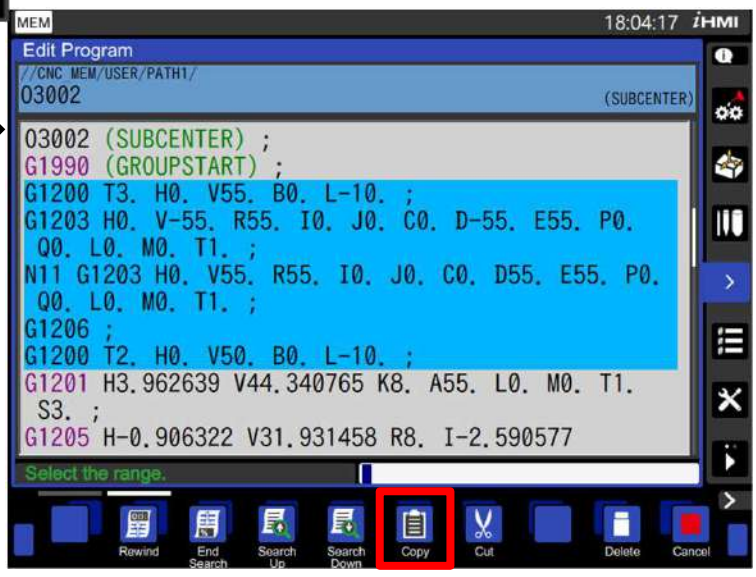
1. Press **PROG manager vertical** soft key.
2. Select Program which need to copy and Press **Start Edit** soft key or **INPUT** hard key



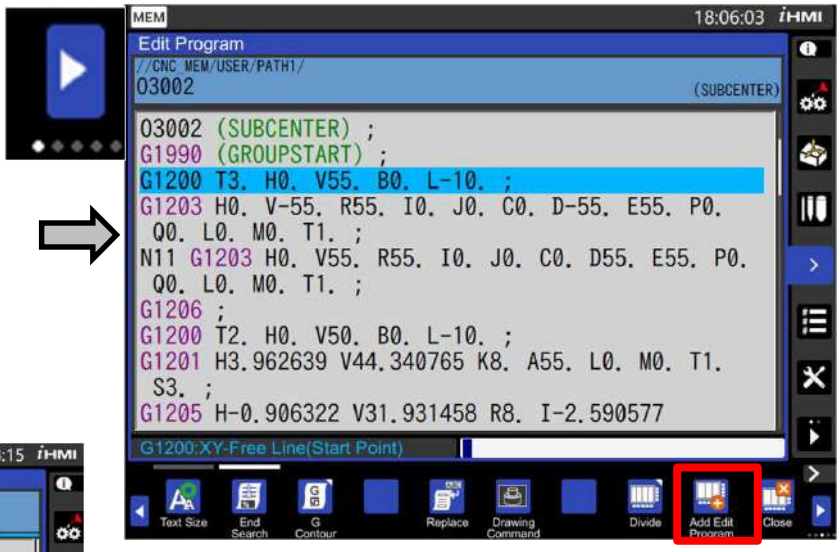
3. Press right arrow soft key for screen change up to **Select Range** option, Keep cursor on line from where to copy the program and Press **SELECT Range** soft key

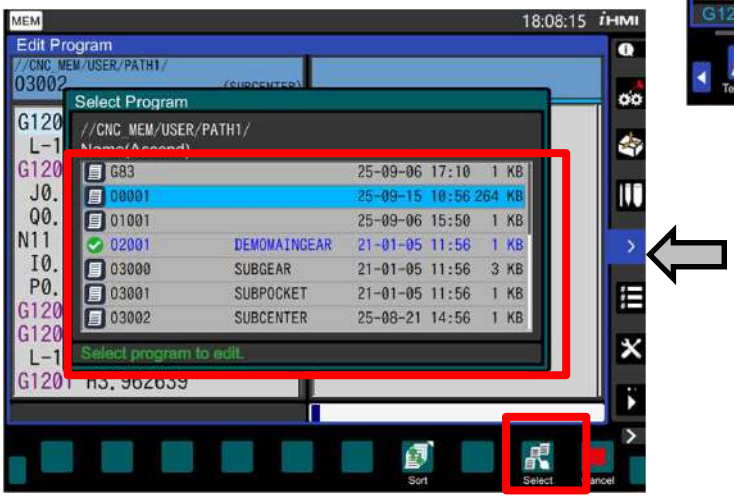


4. Press Continue **Down arrow** MDI key to select Multiple lines
5. Press **COPY** soft key



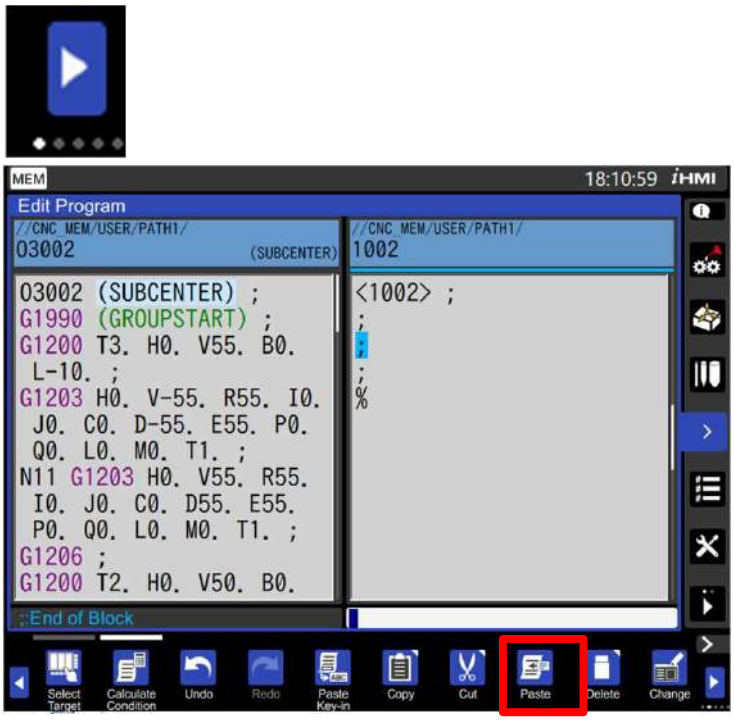
6. Press right arrow soft for screen change up to **add edit program** function.
Press **Add edit program soft key** and select the program where you want to paste and press **select**

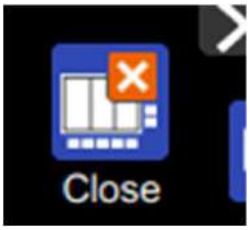


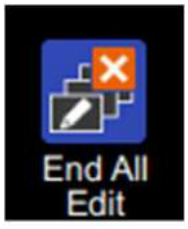


7. After Program open, Press right arrow soft key for screen change up to **Paste** option. Press **Paste** soft key to paste the copied program.

8. Press **Close** soft key to remove the one opened program and **End all Edit** soft key to end all opened program







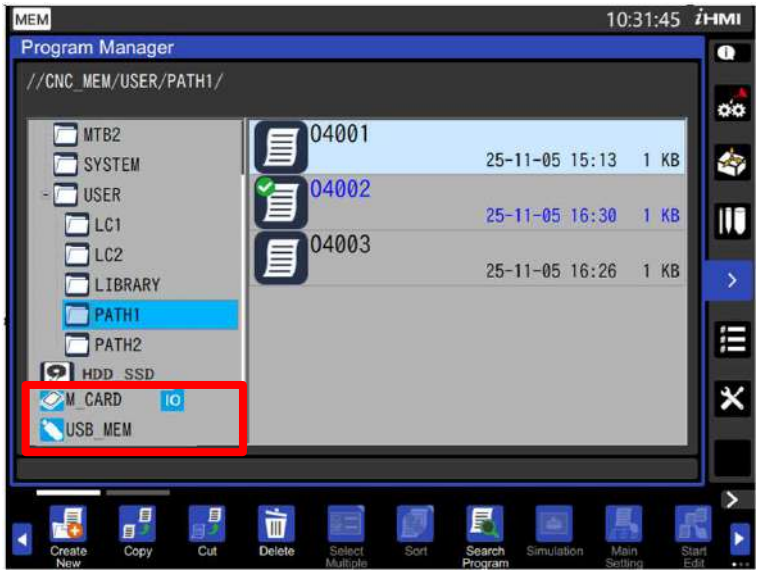
SCAN TO WATCH

1.4 Program Transfer Procedures :-

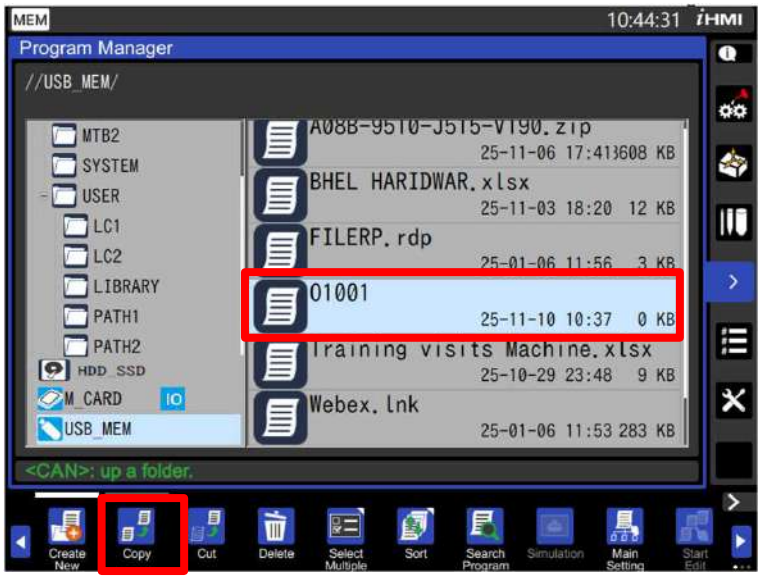
➢ Program Copy From CF/USB to CNC MEMORY :-

Note: Large size programs can copy on HDD_SSD folder (2GB memory space)

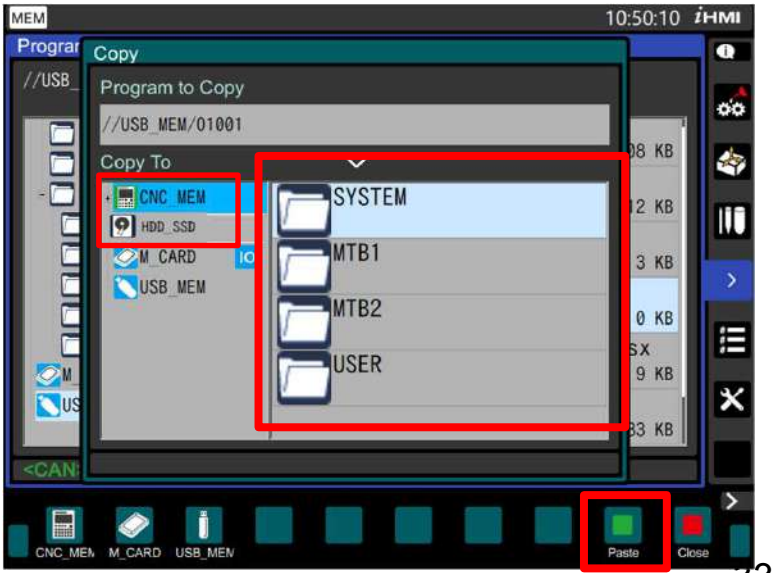
1. Press **PROG manager**  vertical soft key
2. Press **Memory card or USB** to copy the program



3. Keep cursor on Program and Press **CTRL+C** key or **Copy** soft key. (Popup window will appear for paste)



4. Select the folder (**HDD_SSD or CNC_MEM**) where you want to paste and then press **Paste** soft key or **CTRL+V** key.



SCAN TO WATCH

➢ Program Transfer by using Shared Folder Function

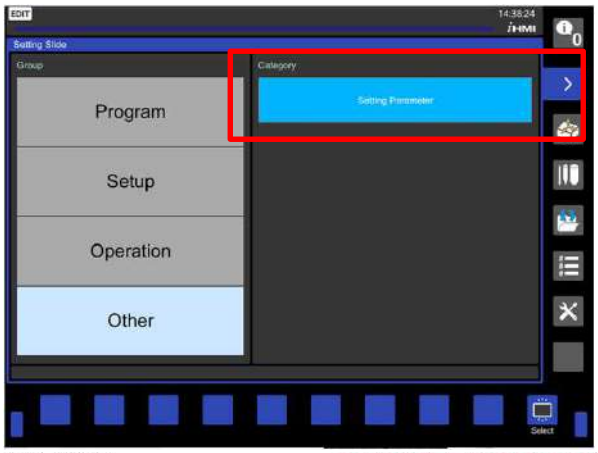
1. Firstly, connect the PC / Laptop to the CNC by Ethernet .

Setting in CNC:

2. In CNC, Press **System** function key.
3. Click **NC operation** option.

4. Press **Setting** vertical soft key then press **Other** in the iHMI screen.

5. Next select **Setting Parameter** option**.**

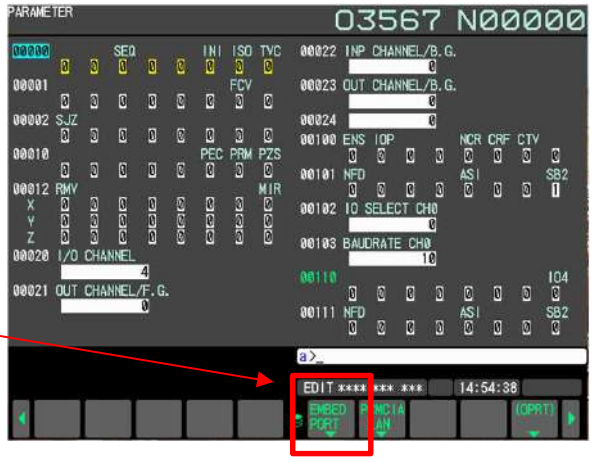


6. Maintenance screen will open.

7. Then press **S2 (system)** function key in MDI panel 

to get this screen.

8. Press **Right Arrow**  (next page) in Horizontal soft key at display until get **EMBED PORT**.



9. Select **EMBED PORT** then select **common**. Enter the *IP address, Subnet Mask which match with your PC* in the respective field.

10. Then Press **FOCAS 2** Horizontal soft key. **Enter PORT NUMBER** (TCP) and **TIME INTERVAL** values as shown in figure.

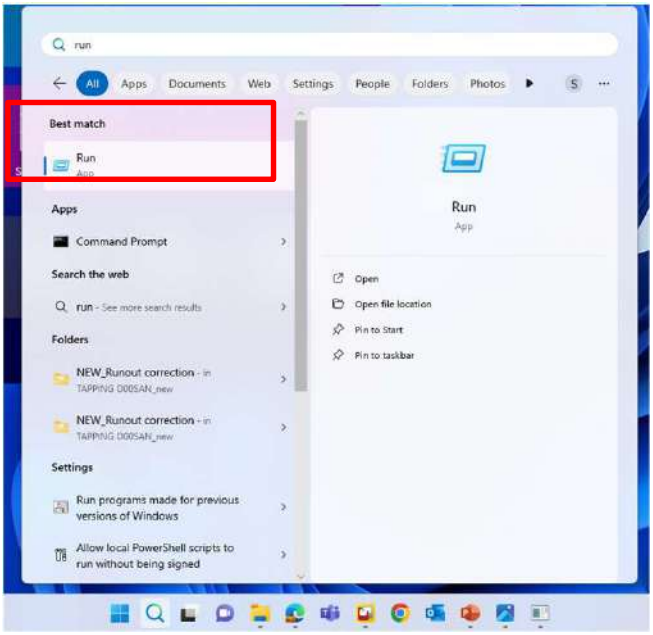
11. After changing of all values Press **RSTART** horizontal soft key to update the changes.

Setting in PC :

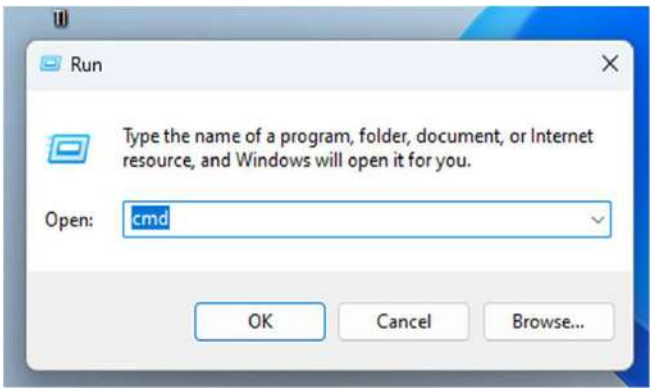
Ensure the connection by pinging to respective IP address.

1. Go to search.

2. Search **RUN** and open

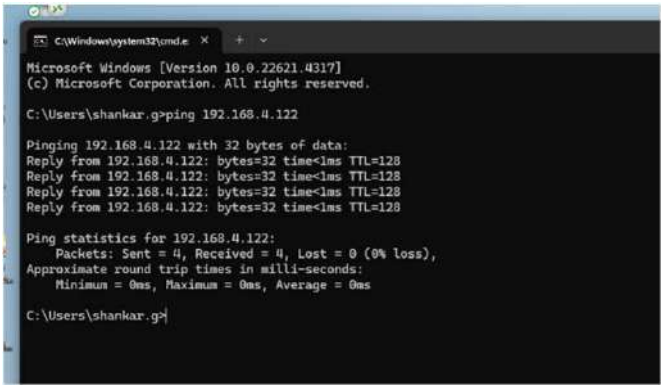


3. Type **CMD** and click **OK**

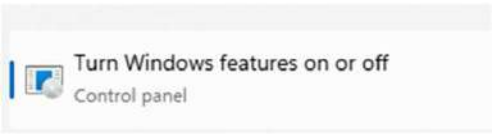


4. Type "**PING 192.168.1.1**" *(type the IP address which you entered in machine)* and press Enter button on keyboard.

5. Check if any error is coming or getting ok. If it is ok then proceed further or need to fix the error.

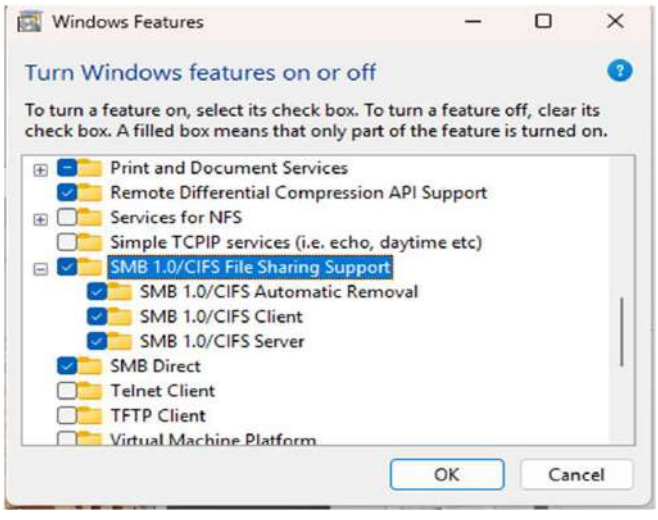


6. Search and Open "Turn windows features on or off".



7. Select "SMB 1.0/CIFS File Sharing Support" as shown in figure.

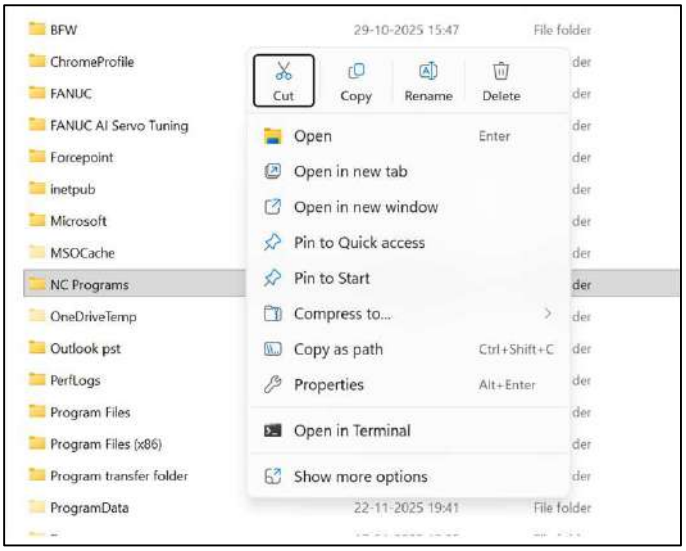
8. Then press **OK** and press **restart now** to update the changes in PC



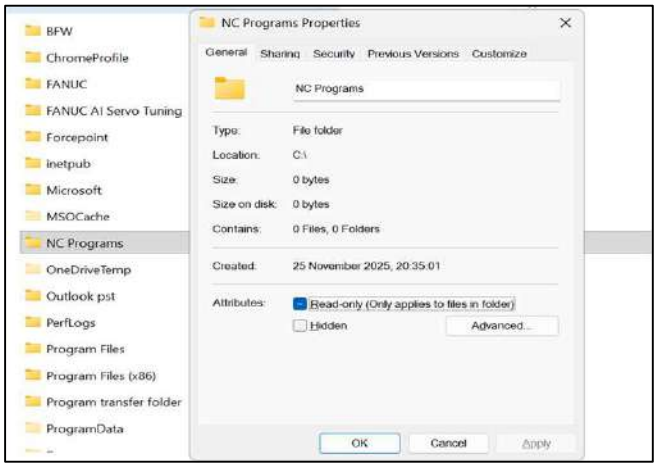
Folder sharing in PC / Laptop:

1. Create the Folder then add programs and other files what you want to share with CNC.

2. Right click on folder and click Properties.

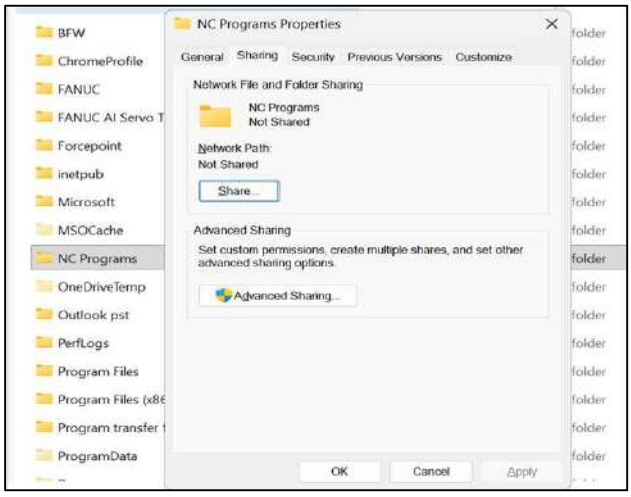


3. Click **Sharing** Tab.

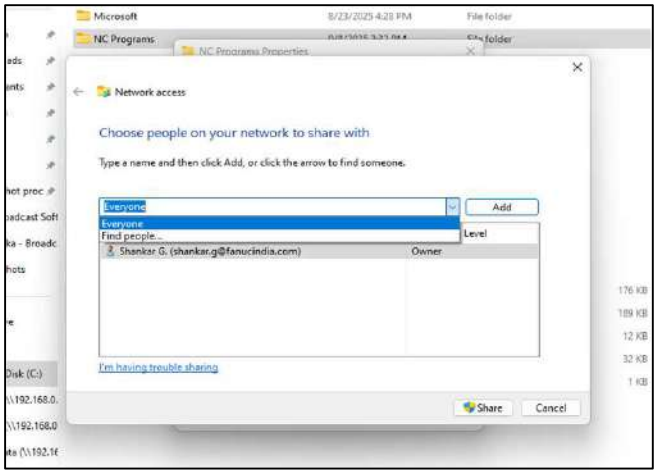


4. Following screen will open

5. Then press **share**

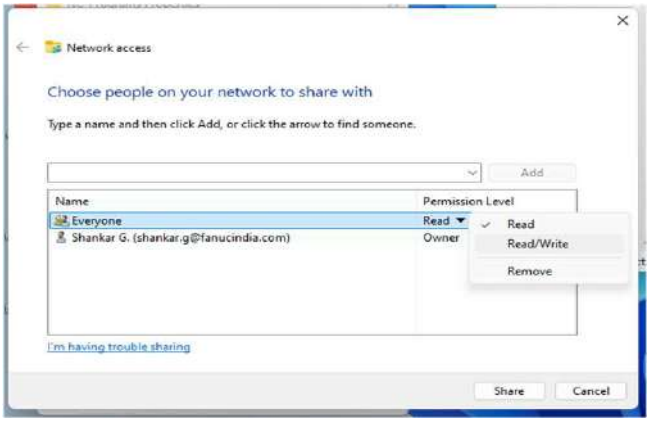


6. Select **everyone** and click **add** button

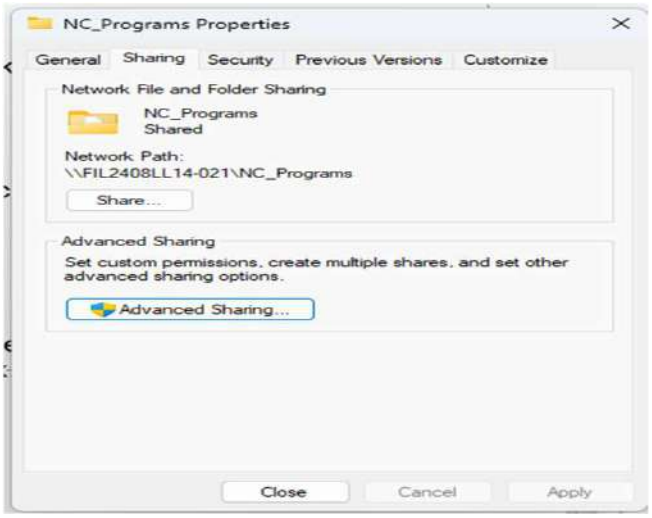


7. Give **Read/write** access for **Everyone.**

8. Click **share** then **Done**

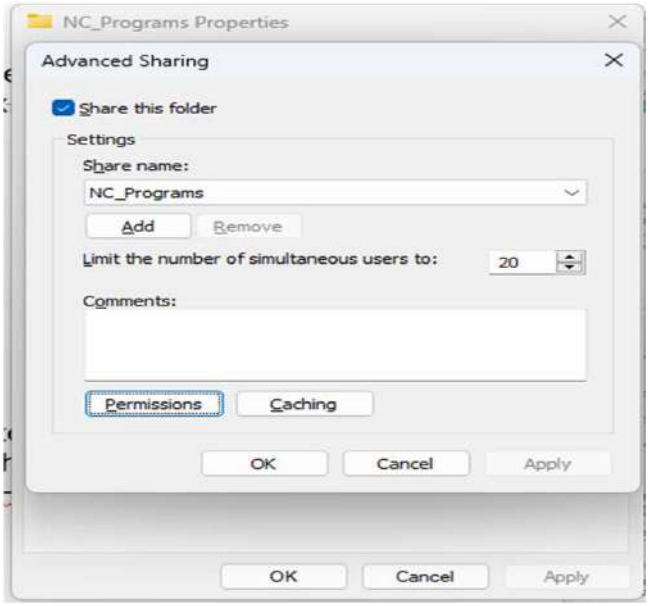


9. Then select **Advanced Sharing**.

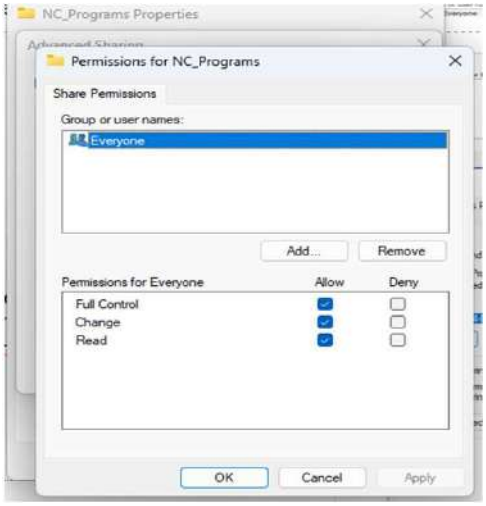


10. Tick **share this folder**

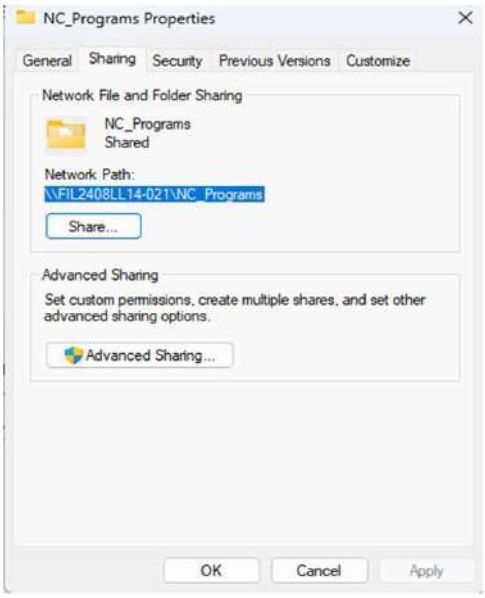
11. Click **permissions**.



12. Select the **Full Control** check box. Then press **ok** button.



13. Note down the folder path. This path only we are going to enter in iHmI screen

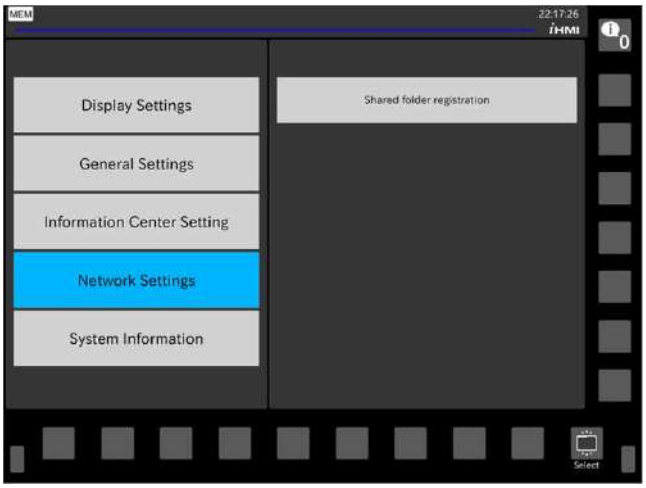


Setting in CNC:

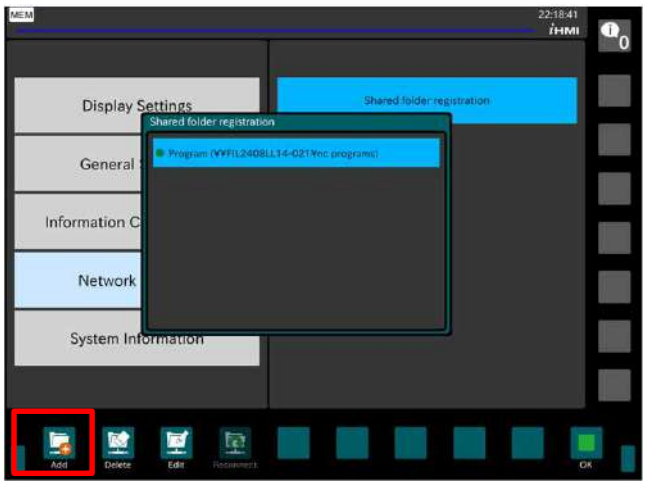
1. Select **settings** in iHMI home screen.

2. Click **Network settings**

3. Click "**Shared folder registration**"



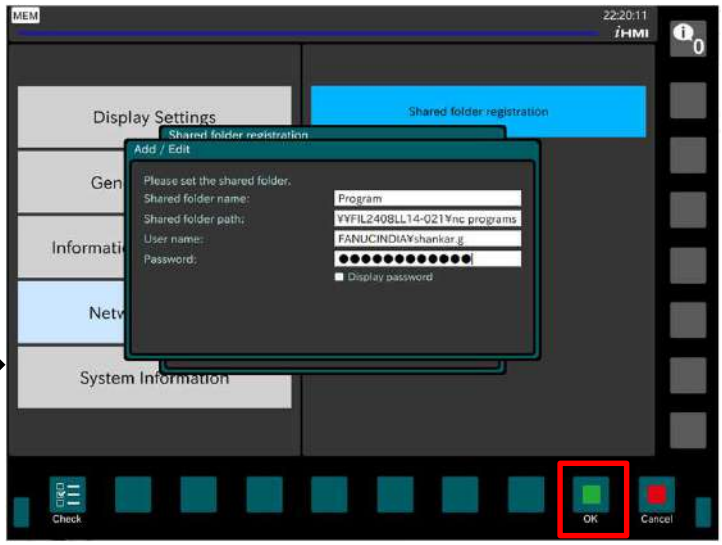
4. Press **Add** horizontal soft key to add the folder.



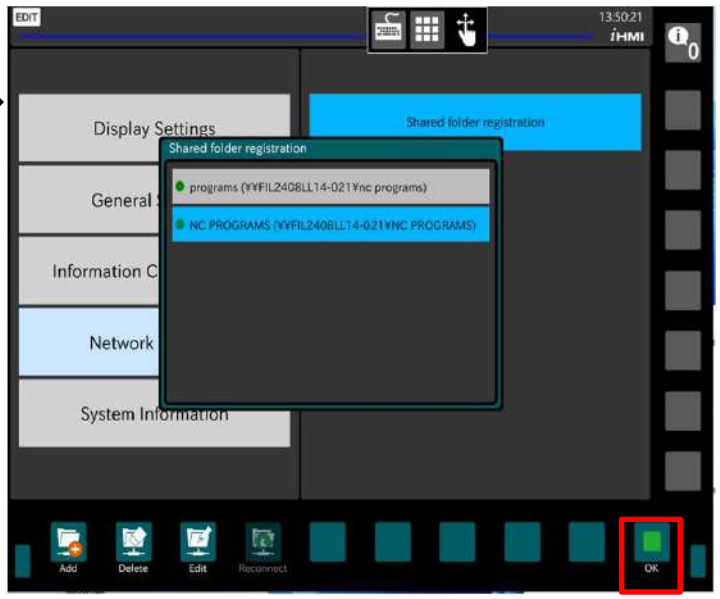
5. Enter the shared folder name (e.g. Program) what you want to keep in cnc.

6. Next enter shared folder path as it is in pc / laptop. Use “ **¥** ” (Yen sign) instead of “ **** “(slash) here.

7. Then enter **username** and **password** of your PC / Laptop.



8. Click **Ok** and **ok** again.



9. Now, the setting has completed.

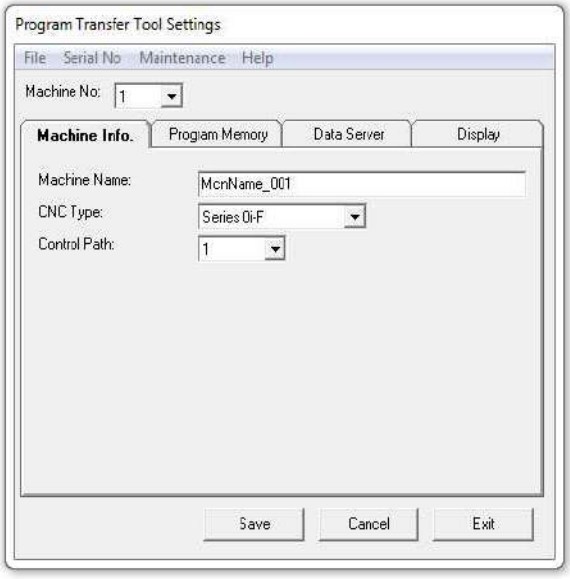
10. Then go to **program Manager** and open your shared folder.

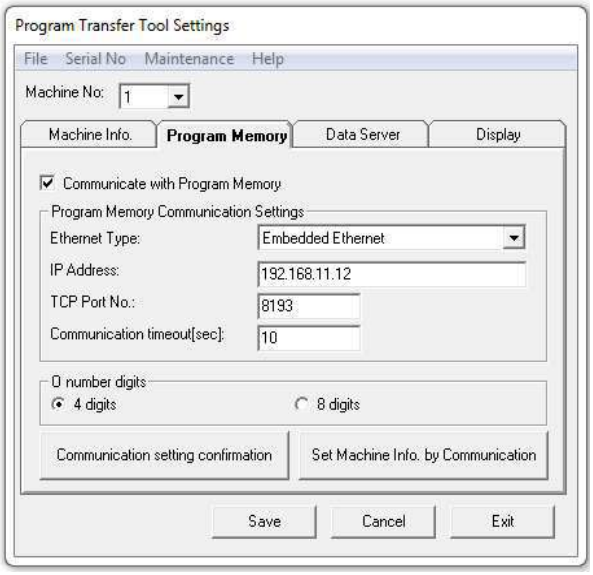
11. Now we can access the files in that shared folder.

SCAN TO WATCH

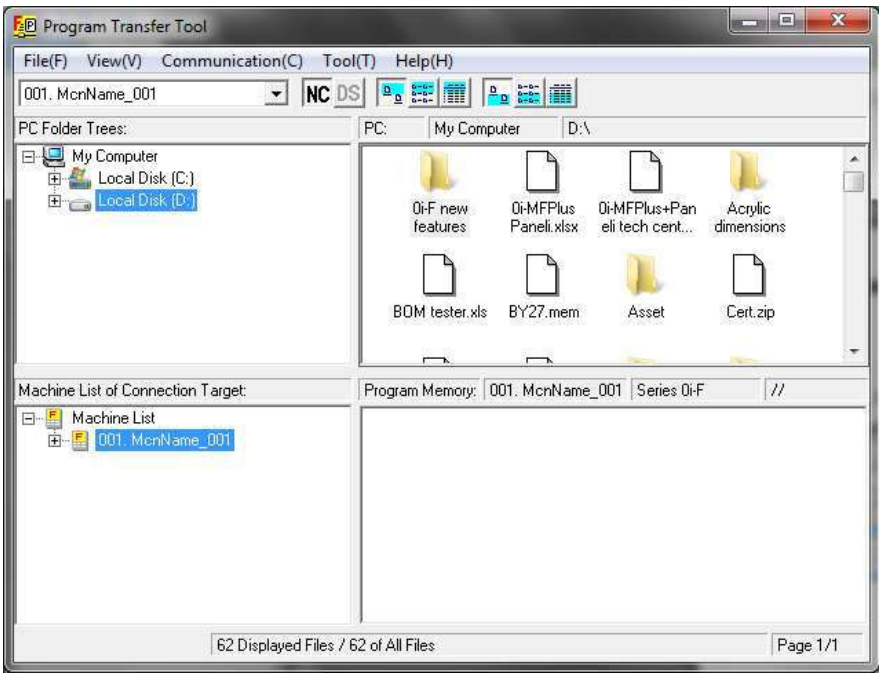
➢ Program transfer from PC to NC using Program transfer tool :-

1. Install Program transfer tool software in PC.
2. After installing, select Start → Program Transfer tool →**Setting dialog**.
3. In **Machine Info** tab, set Machine Name, CNC Type and Control path.
4. In **Program Memory tab**, Enter IP Address, TCP Port no. as per CNC.

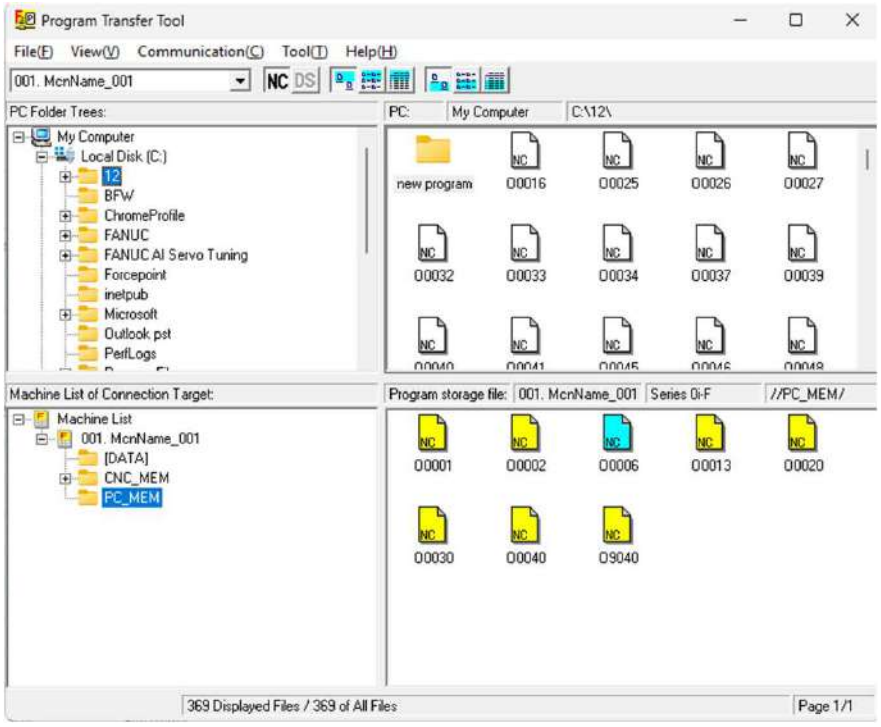




5. Select **SAVE** and **EXIT**.
6. Select Start →Program Transfer tool → **Program Transfer tool**.
7. Select **Machine List of Connection target** → Select Machine.



8. After selecting Machine → Select Following Memory
   For Internal Memory (2MB) Select **CNC_MEM**
   For 2GB Internal Memory select **PC_MEM**



9. To **Upload/Download** the Program. Just drag and drop the program from PC to CNC_MEM or PC_MEM (2GB)

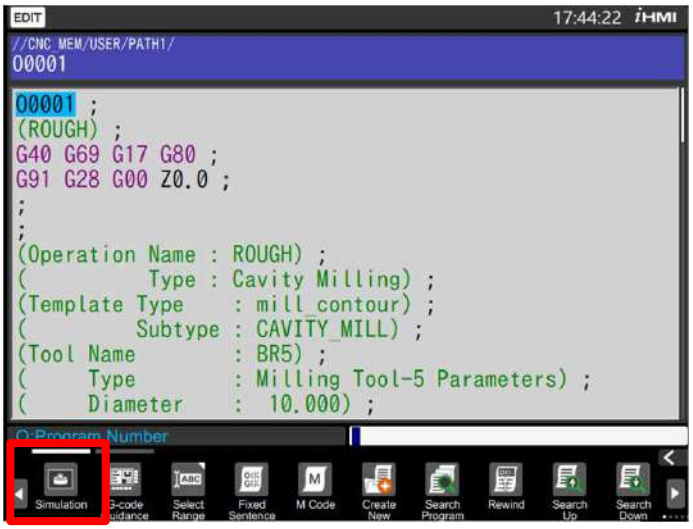
SCAN TO WATCH

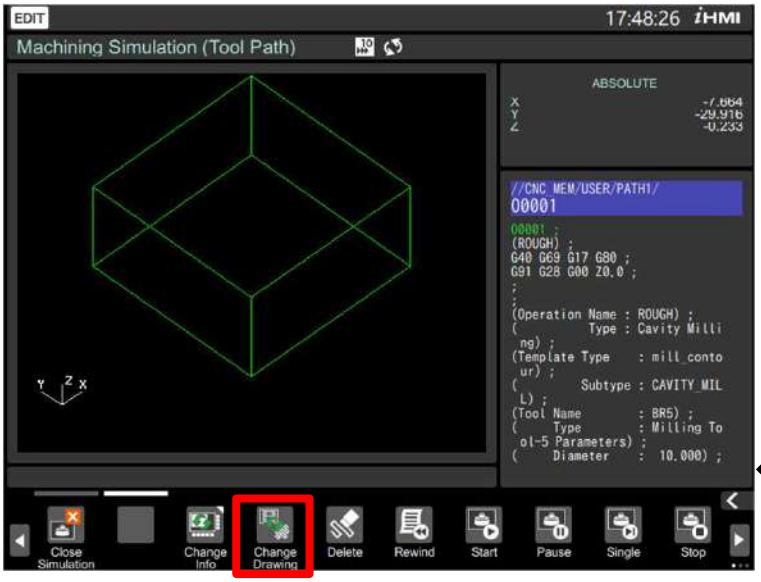
1.5 3D Toolpath and 3D Simulation of Program :-

Note:- Simulation can be performed in Edit mode

➢ 3D Toolpath of Program:-

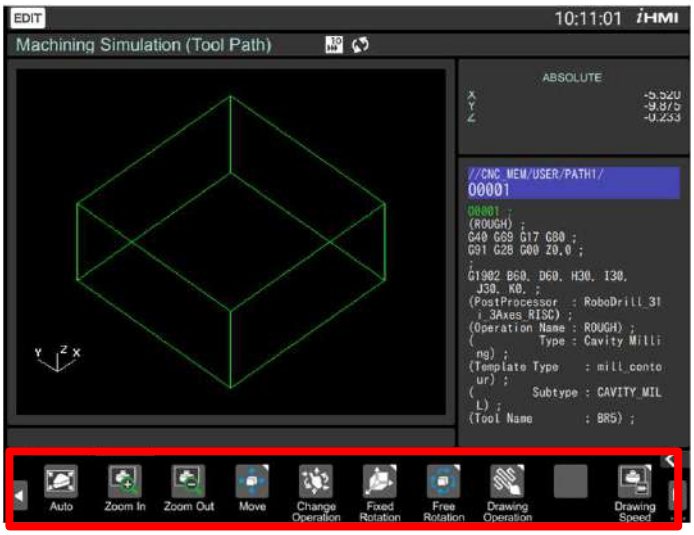
1. Select **EDIT** mode
2. Select the program for simulation and Press **Simulation** soft key

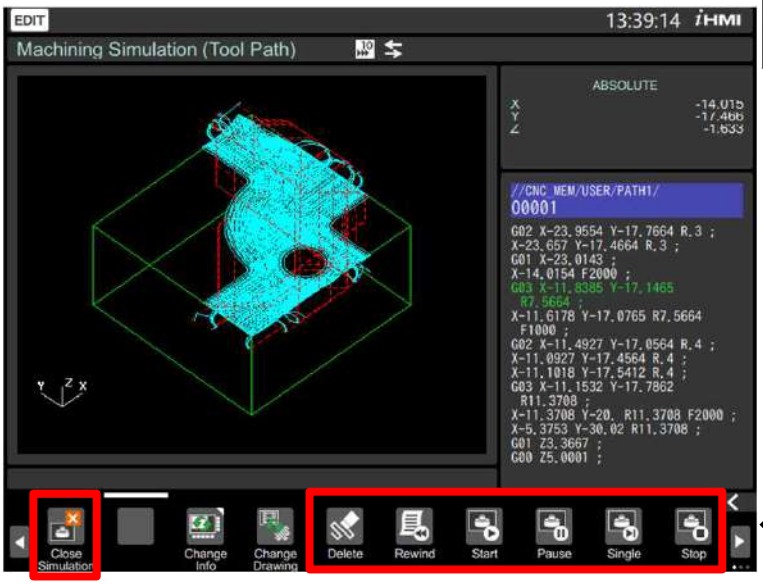




3. Press **Change drawing** soft key for 3D Toolpath

4. Press next arrow soft key for next page for Select Multiple options like Zoom in and out, rotation, simulation speed etc and adjust it as per requirement before start simulation.

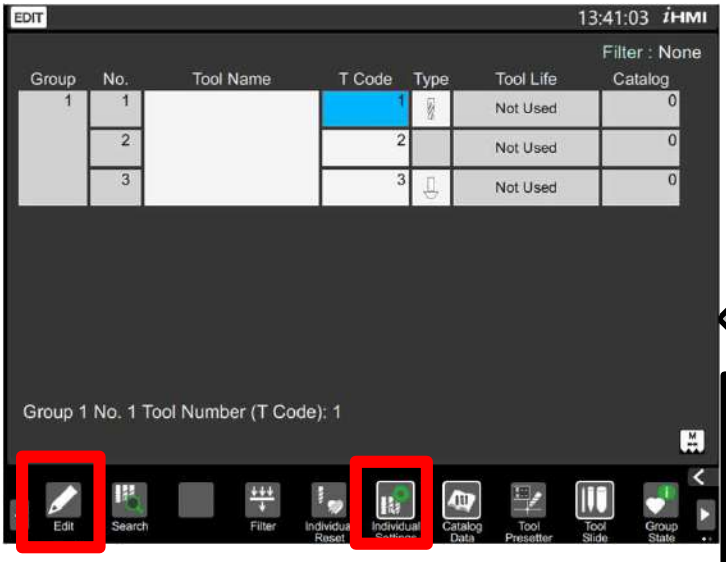




5. Press left arrow soft key for screen change and Press **START** soft key to start 3D Toolpath.
6. Press **Stop, REWIND** and **START** soft key to restart 3D Toolpath.
7. Press **Delete** to erase simulation.
8. Press **Close simulation** soft key to close

➢ 3D Simulation of Program :-

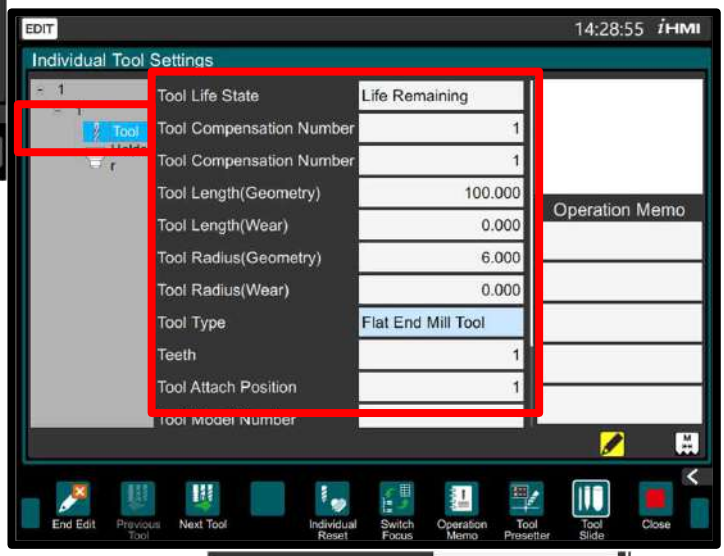
1. Select **EDIT** mode
2. Press **tool slide** vertical soft key
3. Press **Tool Manager** soft key

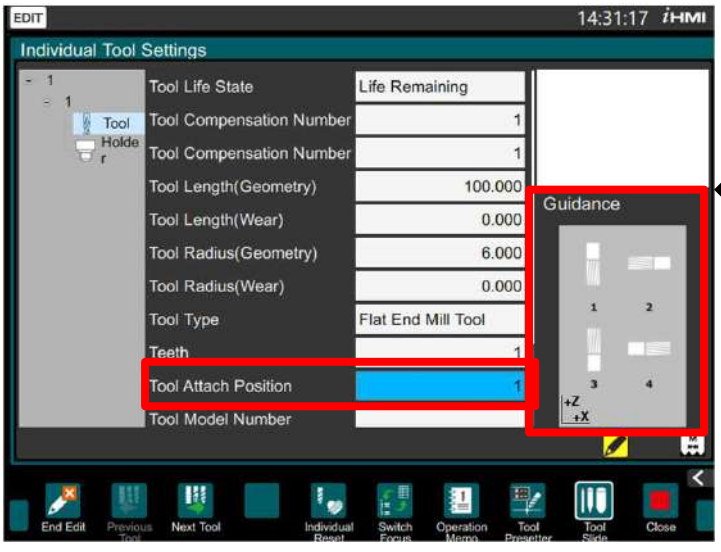


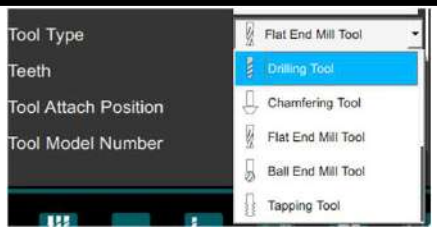
4. Press **Edit** soft key.
5. Press **Individual setting** soft key.

6. Go to tool Setting and press **switch focus** and add all tool data like Tool no., tool length, diameter, teeth etc.
7. In tool type select which tool your are using for machining.

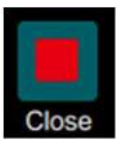
Note: Must mention the radius of tool

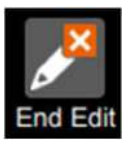


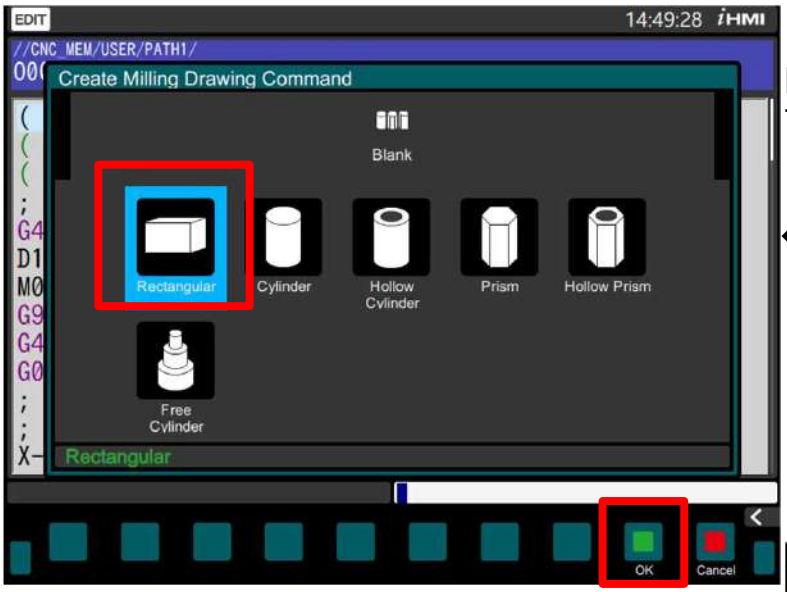




8. Select the tool attach position as per shown diagram.
9. Then press **Close**, **End edit** **& Tool Slide** soft key to come back at main screen

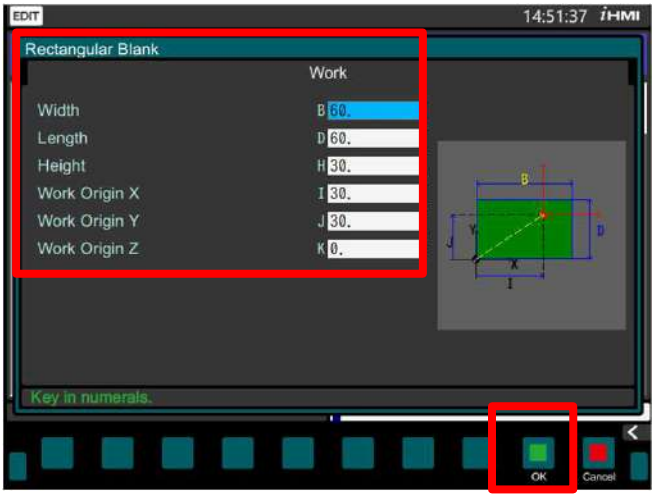






10. Keep curser in starting of the program and Press **Drawing command** to add blank for simulation

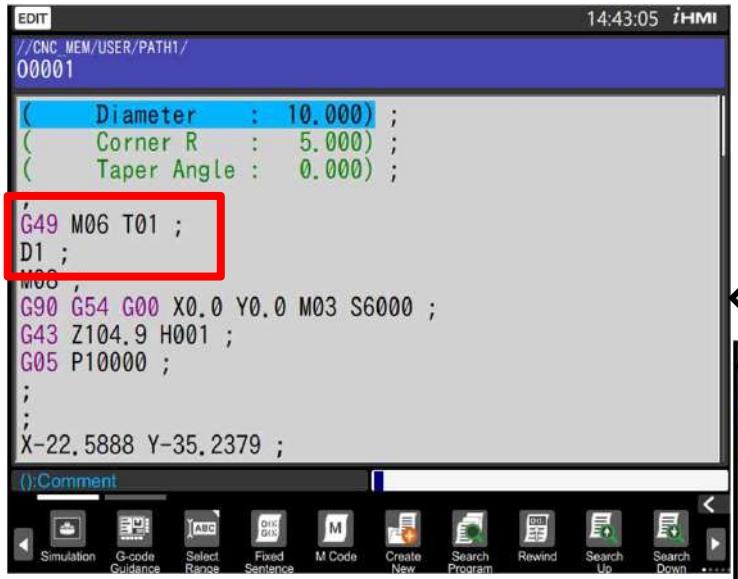
11. Select the type of raw material and press **OK**



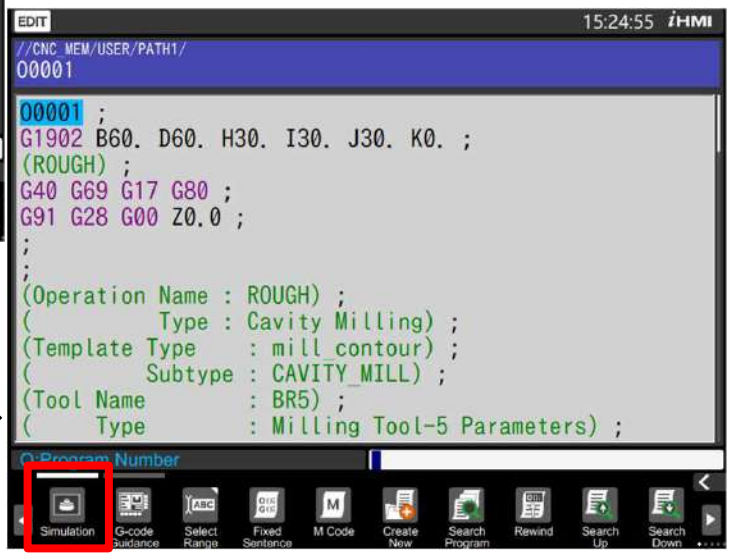
12. Insert the dimensions of block like width, length and height.
13. Mention the work offset info in I, J, K value.
14. Press **Ok,** then one line G code will insert in the part program.

Note: Press input key in this block to Modify the existing value

```
G1902 B60. D60. H30. I30. J30. K0. ;
```

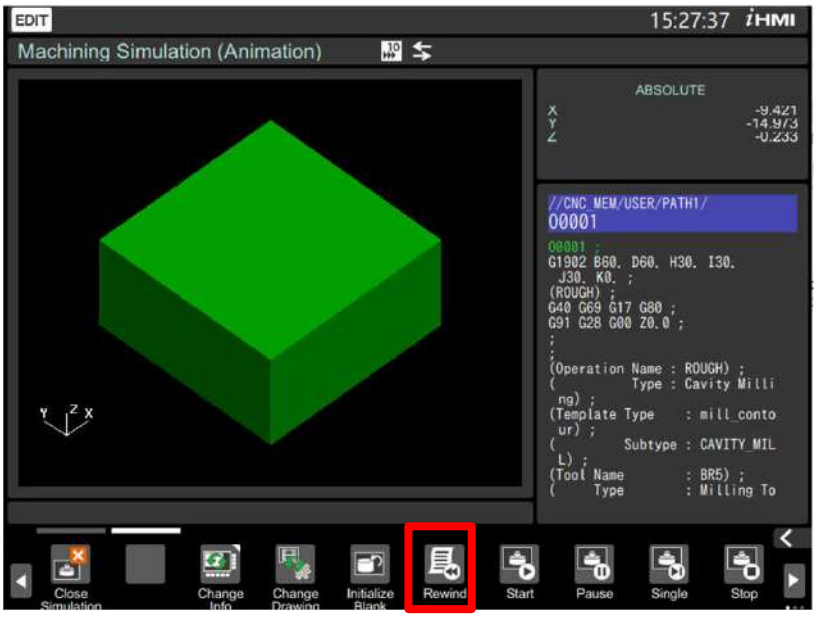


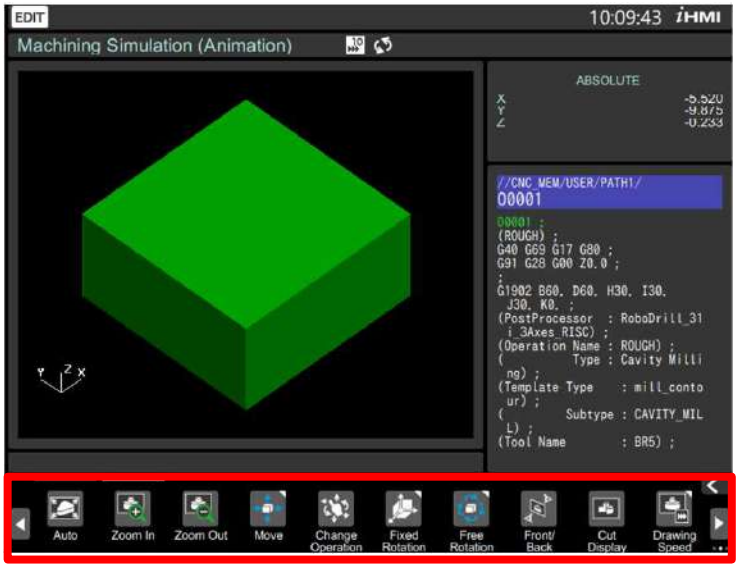
15. Make sure that same **D Number** entered below tool change command like **D1** for **T1** and **D2** for **T2**.
(If D number not entered it will not Show Tool movement in 3D simulation)



16. Press Simulation soft key for simulation screen

17. Press **Rewind** to reset the program.





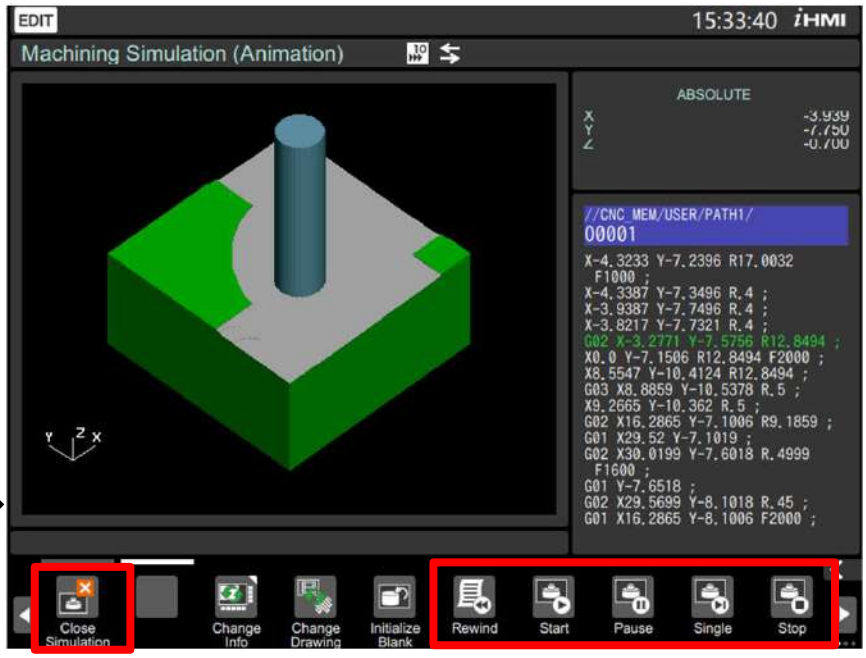
18. Press right arrow soft key to change screen for Select Multiple options like Zoom in and out, rotation, simulation speed etc and adjust it before start simulation.

19) Press **left arrow** soft key to screen change and press **Start soft key** to start the Simulation and **Start** to start the Simulation

19. Press **Stop, REWIND** and **START** soft key to restart 3D simulation

20. Press **Delete** to erase simulation.

21. Press **Close simulation** soft key to close

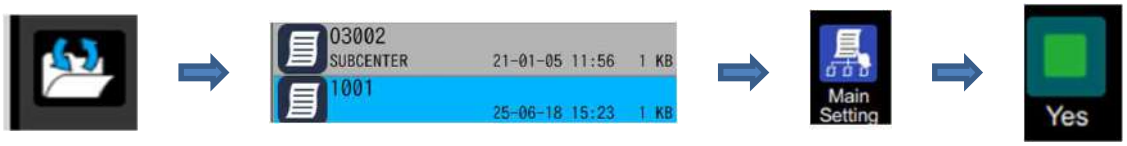


SCAN TO WATCH

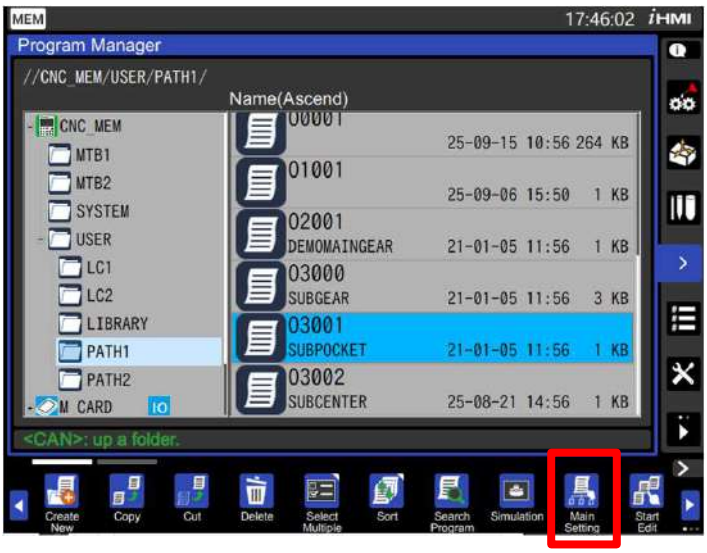
1.6 Program Execution Functions :-

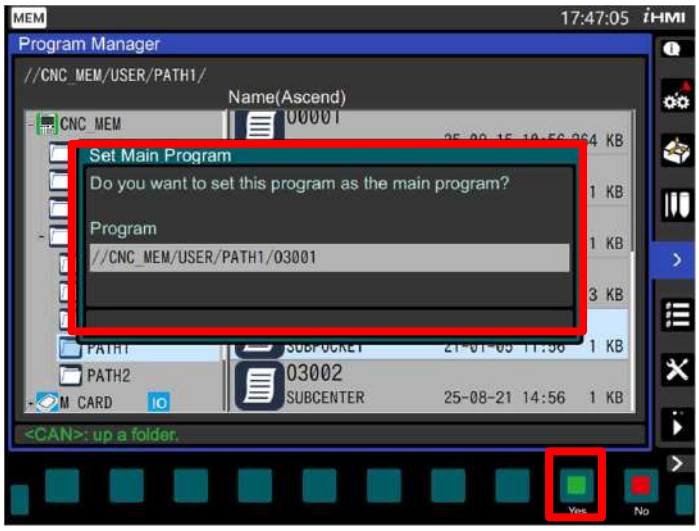
➢ Main Program Select from List:-

1. Press **PROG MANAGER** vertical soft key
2. Keep Cursor on Program
3. Press **MAIN setting** soft key
4. Press **YES** to set this program as main program



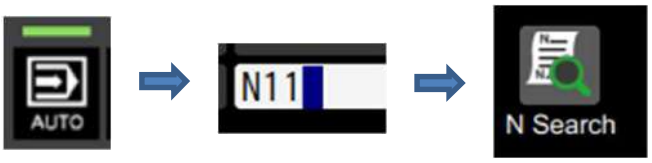
6. Make sure that "✓" symbol came in top of program name 





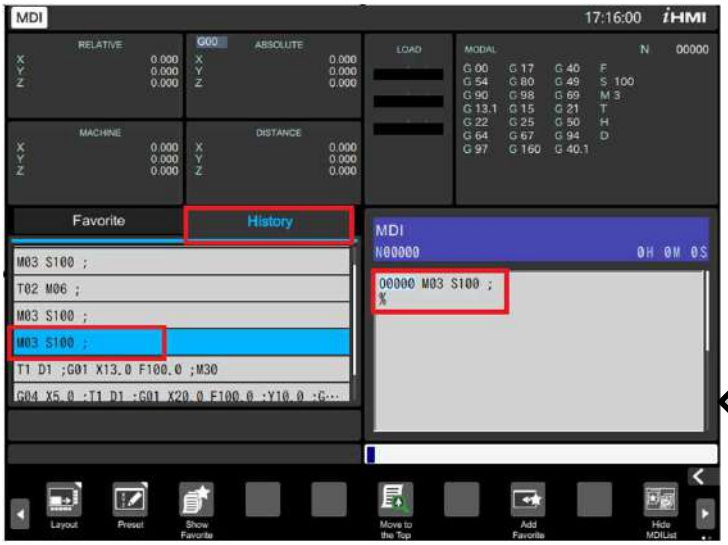
➢ Sequence (N) Number Search :-

1. Select **AUTO OR MEMORY** mode
2. Type Sequence Number (N1 or N2 or N3 etc.)
3. Press “**N SEARCH”** soft key

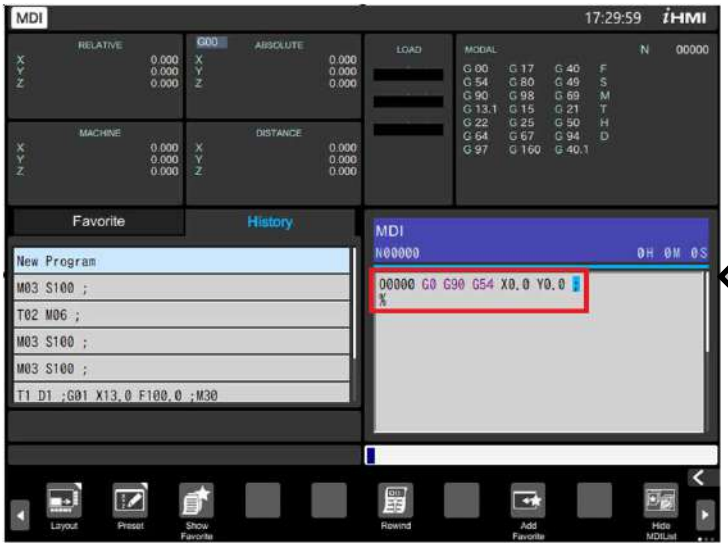


SCAN TO WATCH

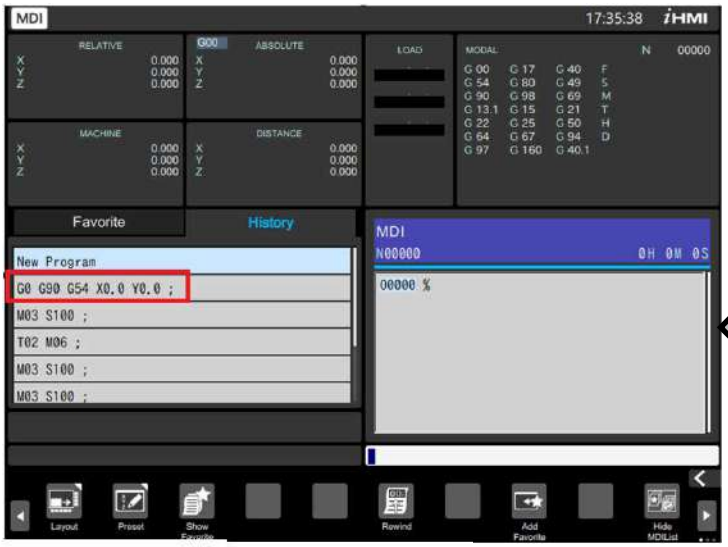
➢ Use Saved MDI Programs for Fast execution :-



1. Select **MDI** mode
2. Right side will show **MDI screen** and Left side will Show **History** of MDI Programs
3. Touch and select Program from History and cycle start to execute.



4. To Execute new **MDI** program, Touch & Select Right Side **MDI Screen** and Enter Program (E.g. **G0G90G54X0Y0**) and cycle start to execute.



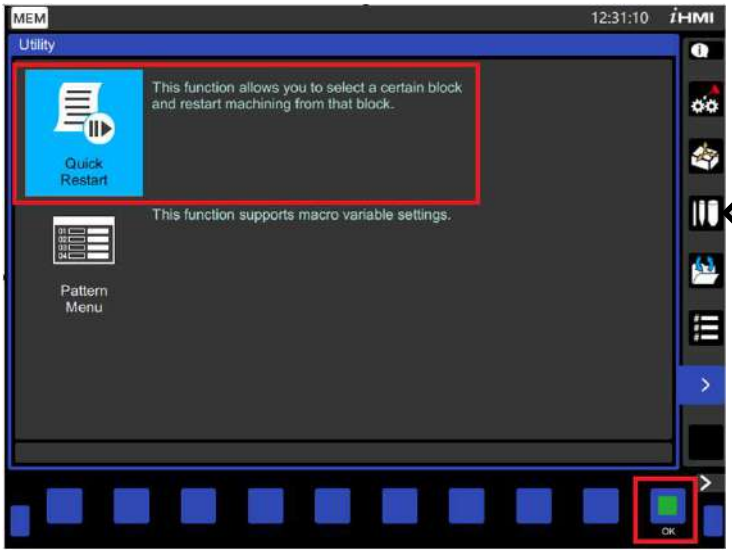
5. After Execution of any New MDI Program it will saved in **History** and to execute same MDI Program next time, Touch and select Program from **History** and cycle start

SCAN TO WATCH

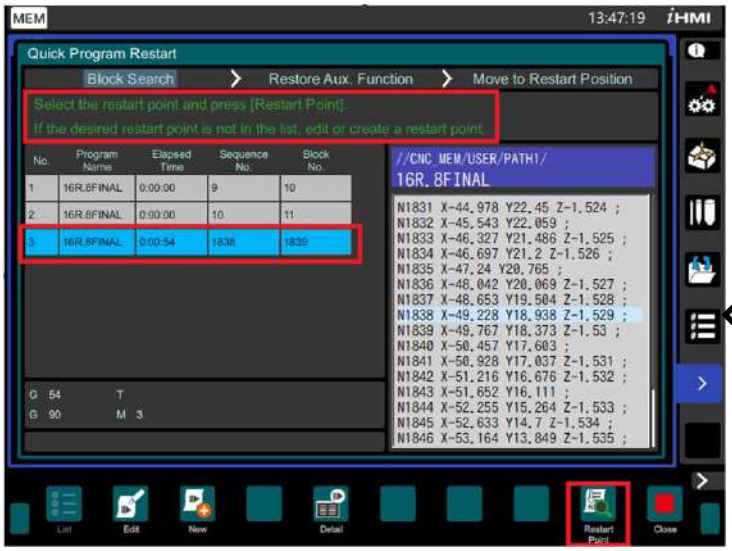
1.7 Quick Program Restart :-

➢ Quick Program Restart after Power Failure or after Reset :-

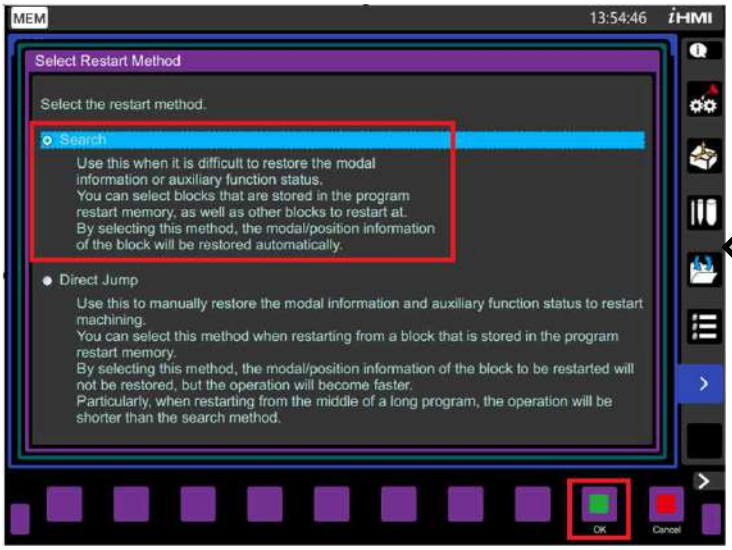
1. Take **Z-axis Plus** using **JOG** or **HANDLE** mode.
2. Select **AUTO** or **DNC** mode
3. Touch **Lower Right Arrow**  to open Vertical Soft Keys. (iHMI Options/Features)
4. Touch **Utility**  Soft Key



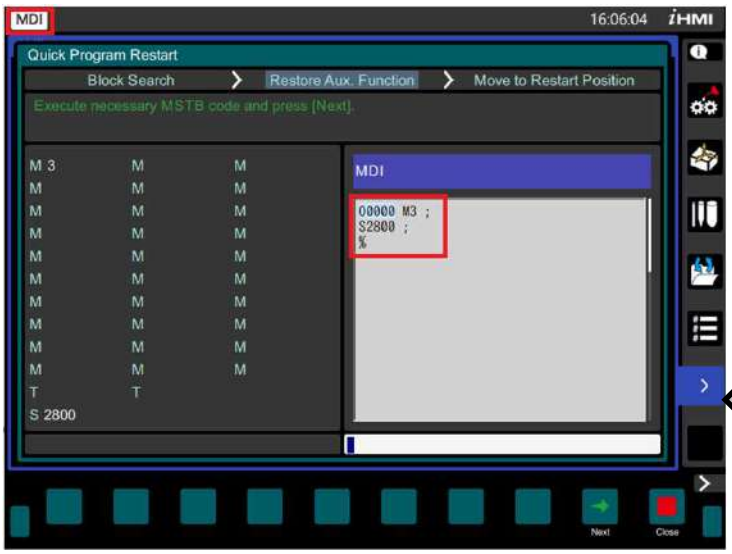
5. Select **Quick Restart** option and Touch **OK** Soft Key.



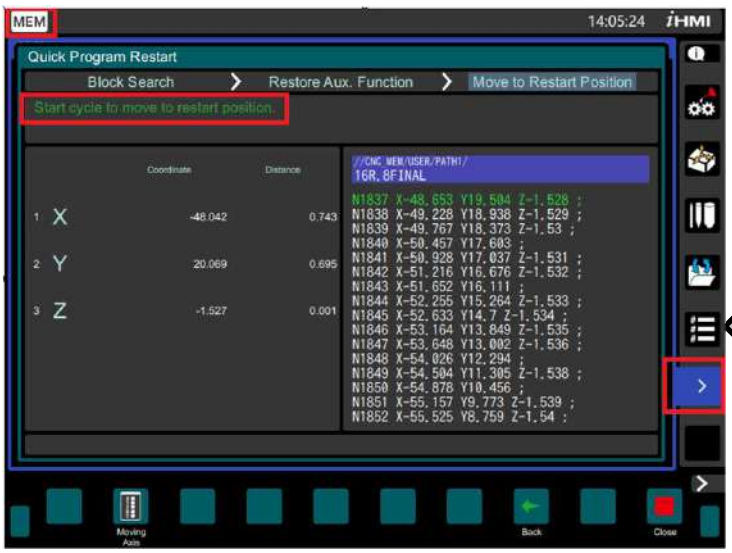
6. Select Lowest Restart Point (**e.g. No.3**) and Touch **Restart Point** Soft Key. (Follow the Guidelines Shown on Screen for easy Quick Program Restart)



7. Select **Search** as **Restart Method** and Touch **OK** Soft Key.



8. After Search Completed, Select **MDI** Mode. Verify and edit MDI Program and add other Auxiliary Functions like coolant and Press **Cycle Start,** Spindle should start rotating along with coolant.
(**Note** - Remove Unwanted M and T codes from MDI Program)

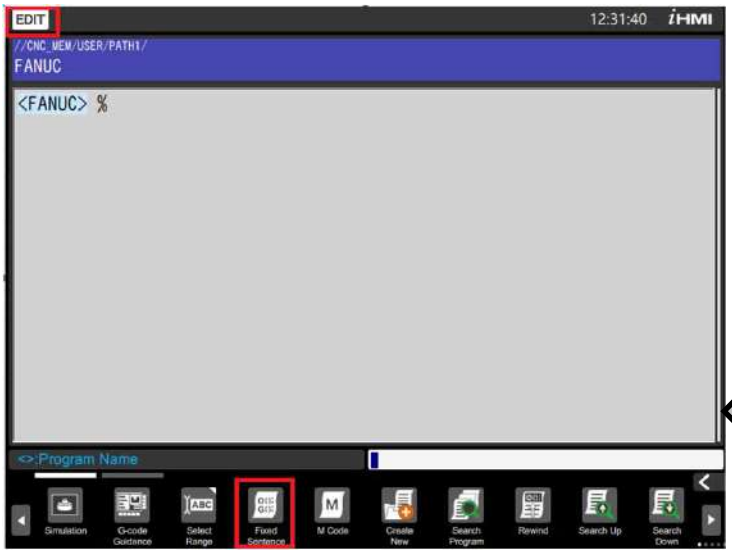


9. Select **AUTO** or **DNC** mode and Press **CYCLE START**.
10. Touch **Right Arrow** Soft Key  to close Restart Window.

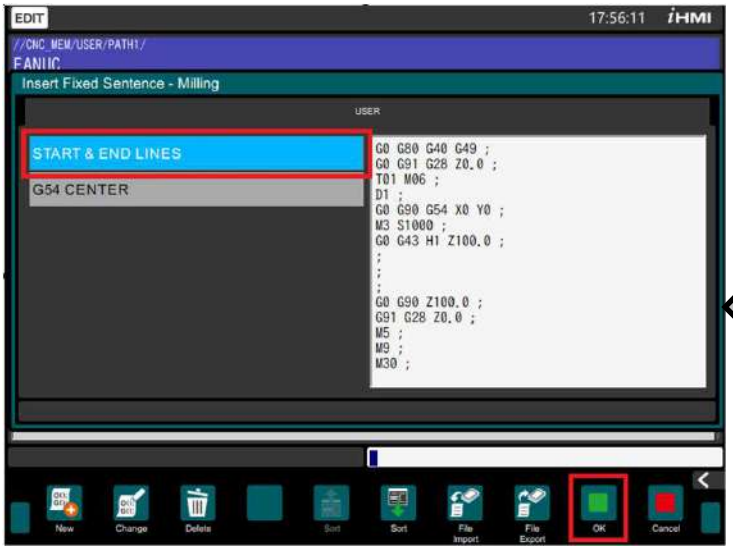
SCAN TO WATCH

1.8 G code Guidance :-

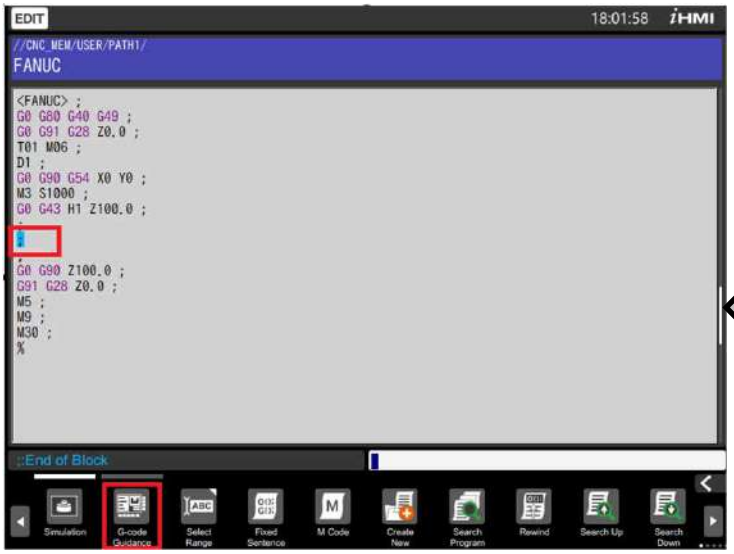
➢ PCD Peck Drilling Program :-



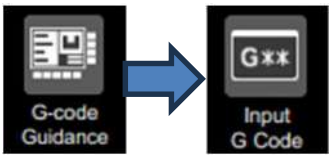
1. Select **EDIT** mode
2. It will show your Main Program that had been selected.
3. Touch **Fixed Sentence** soft key. (Please refer to **Section 1.3** for creation of fixed sentence)

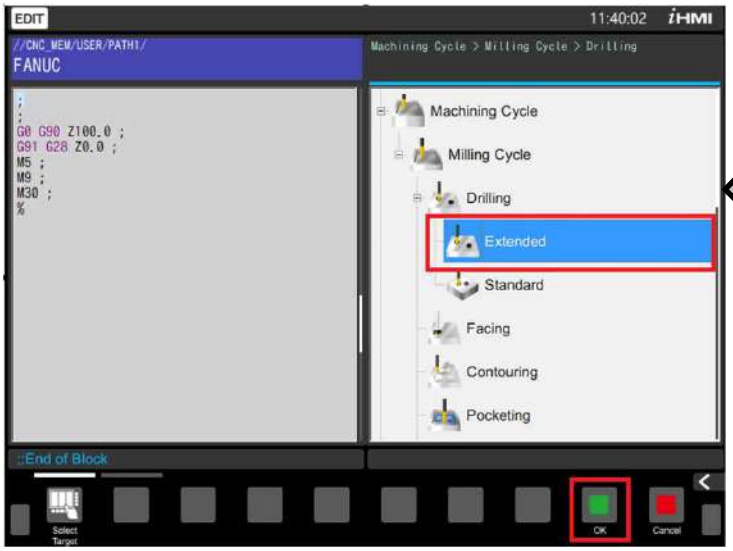


4. Select Required Fixed Sentence (E.g. **START & END LINES)** and Touch **OK** soft Key . (**NOTE –** Refer Content **1.3** for **Fixed Sentence)**

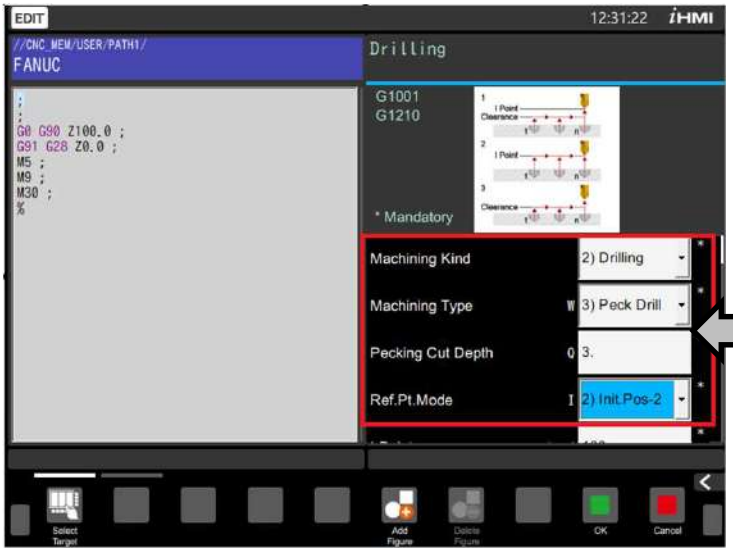


5. Keep cursor between Starting & Ending Lines and Touch **G-code Guidance** Soft Key.
6. Touch **Input G Code** Soft Key.



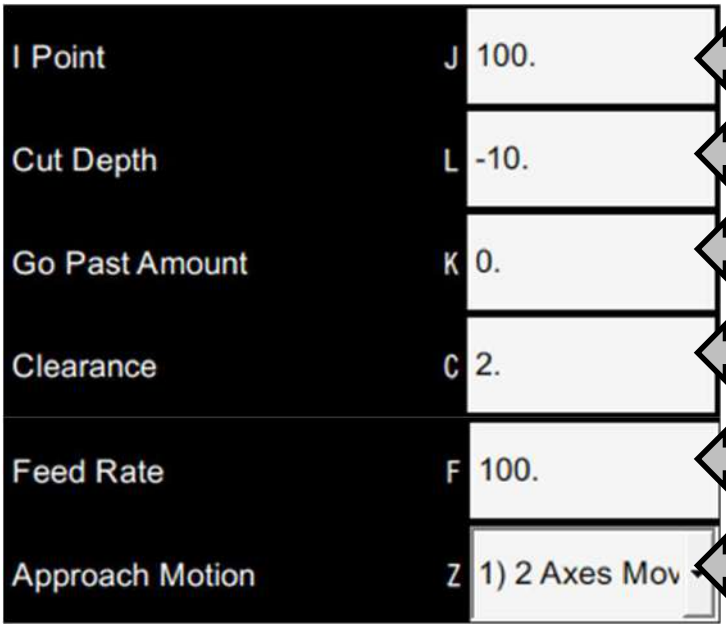


7. Select **Extended** option from **Drilling** & Touch **OK** soft key



8. Fill the Data in Tabular Format to Create Program Easily.
(**“ * ” Star Mark** Data is compulsory & Data without star mark is optional)

Select Operation- Drilling , Tapping etc.
Select Type – Peck Drill , Normal etc.
Enter Pecking Depth.
Select G98/G99 (Init.Pos-2/Init.Pos-1)



Enter Safe Z axis Value.

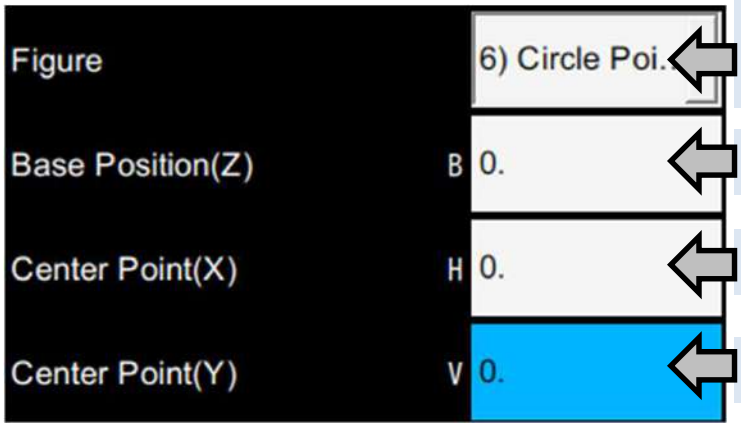
Enter Total Drilling Depth.

Enter Extra Depth (Keep it 0).

Enter Drill Start Position (R).

Enter Drilling Feed rate.

Select Hole Approach Motion as 2 Axes Movement

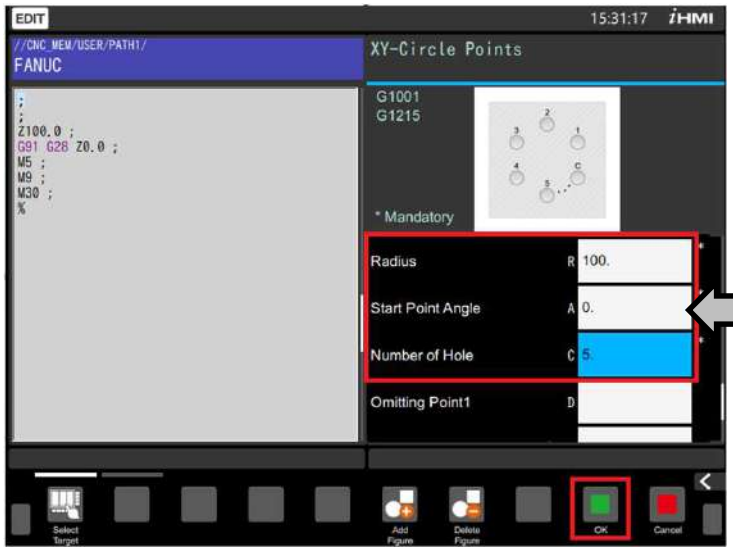


Select Drilling Figure- Random, Circle, Square , Grid etc.

Enter Base Position as Zero.

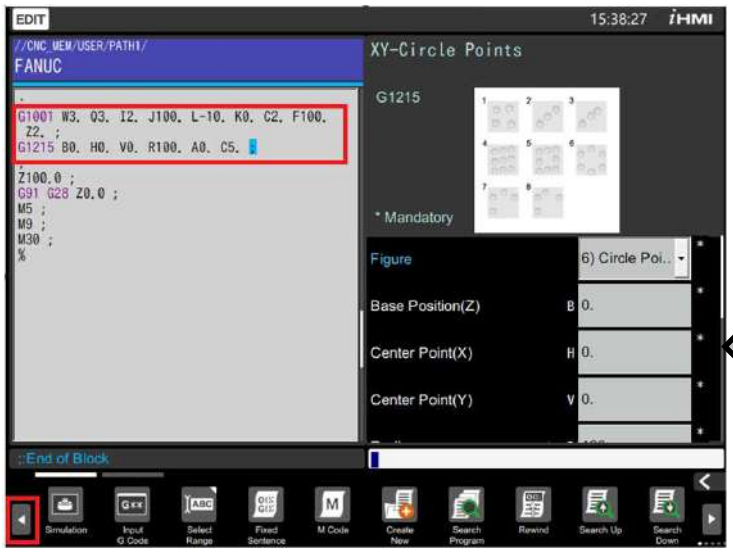
Enter X Axis Coordinate of PCD Centre.

Enter Y Axis Coordinate of PCD Centre.



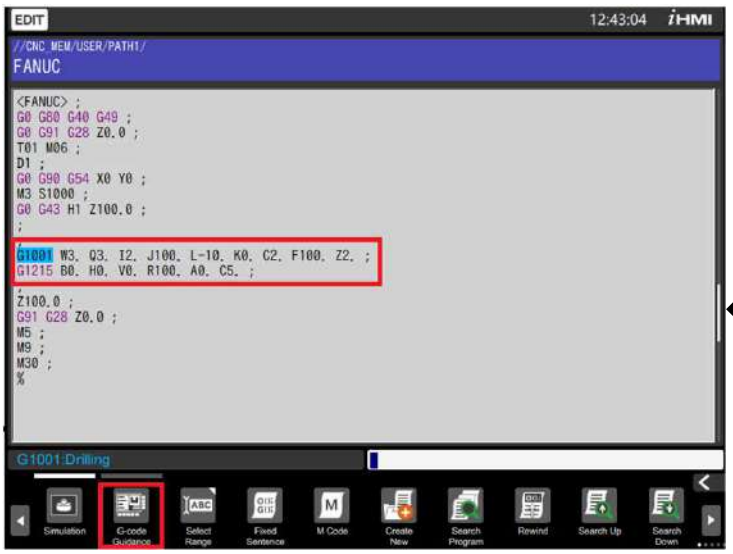
Enter Radius of PCD.
Enter 1st Hole Angle (Start Angle)
Enter No. of Holes

9. After all data filled, Touch **OK** soft Key.

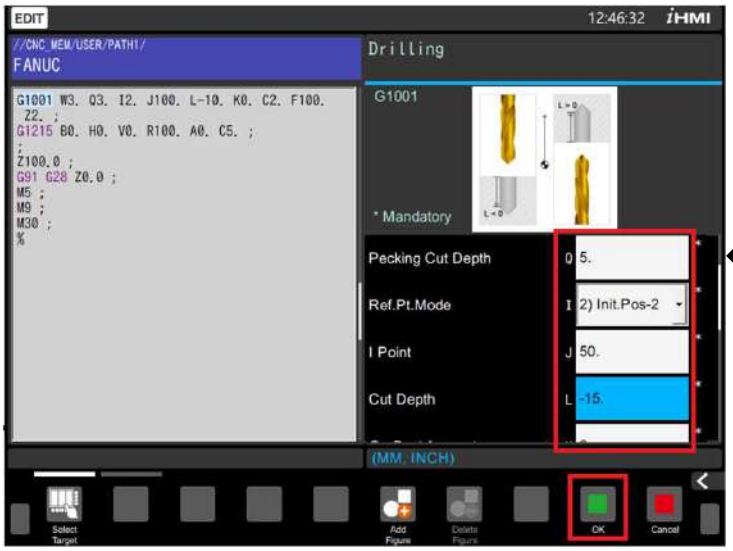


10. **Program lines** will be inserted between starting and ending lines.
11. Touch **Left Next Page** and Touch **Close Guidance** Soft Key to close G Code Guidance window.

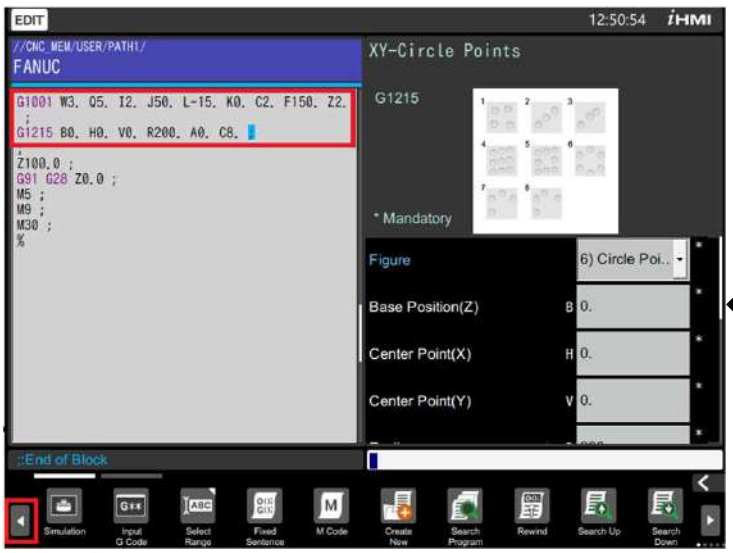
SCAN TO WATCH



12. After Cycle Program generation If want to make any changes in **Cycle** then keep cursor On Cycle Program lines (E.g. G1001,G1215) and Touch **G-code Guidance** Soft Key.



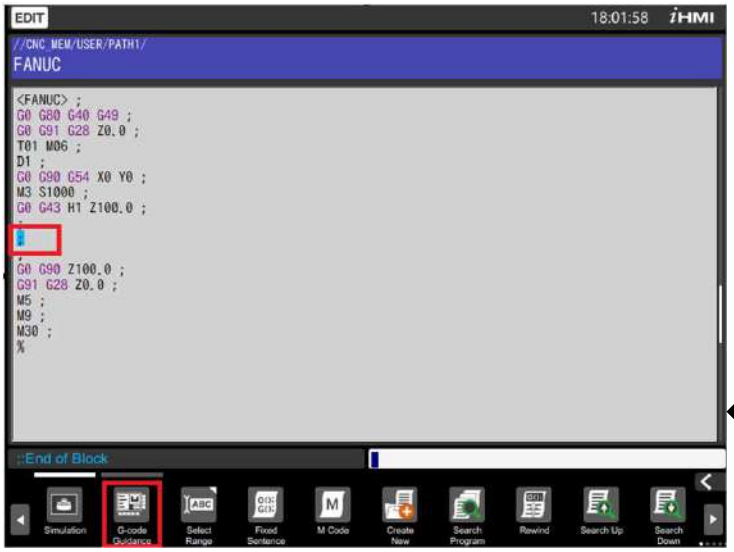
13. It will show Existing Cycle Data , Touch and **Edit or Add New Cycle data** as required.
& Touch **OK** soft key.



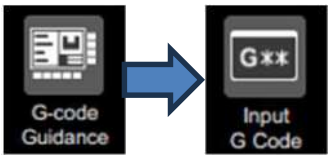
14. **Cycle Program** will be changed as per new Data.
15. Touch **Left Next Page** and Touch **Close Guidance** Soft Key to close G Code Guidance window.

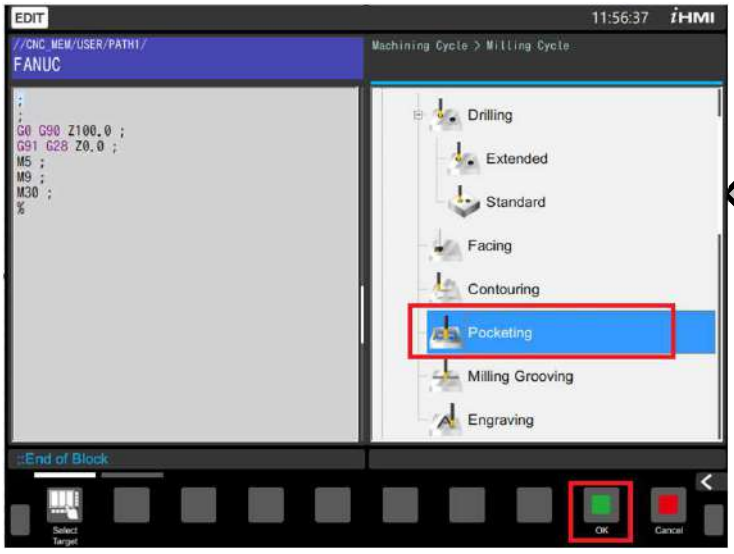
SCAN TO WATCH

➢ Square Pocketing Program :-

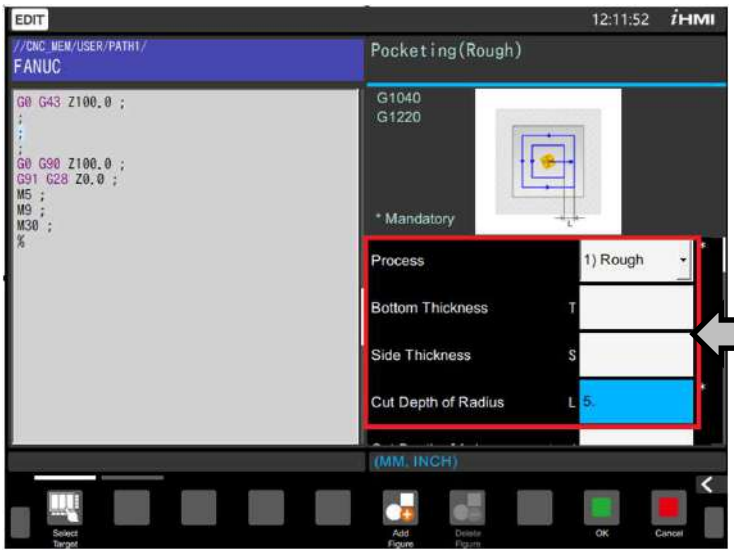


1. In **EDIT** mode , after adding Starting & Ending lines .
2. Keep cursor between Starting & Ending Lines and Touch **G-code Guidance** Soft Key.
3. Touch **Input G Code** Soft Key.



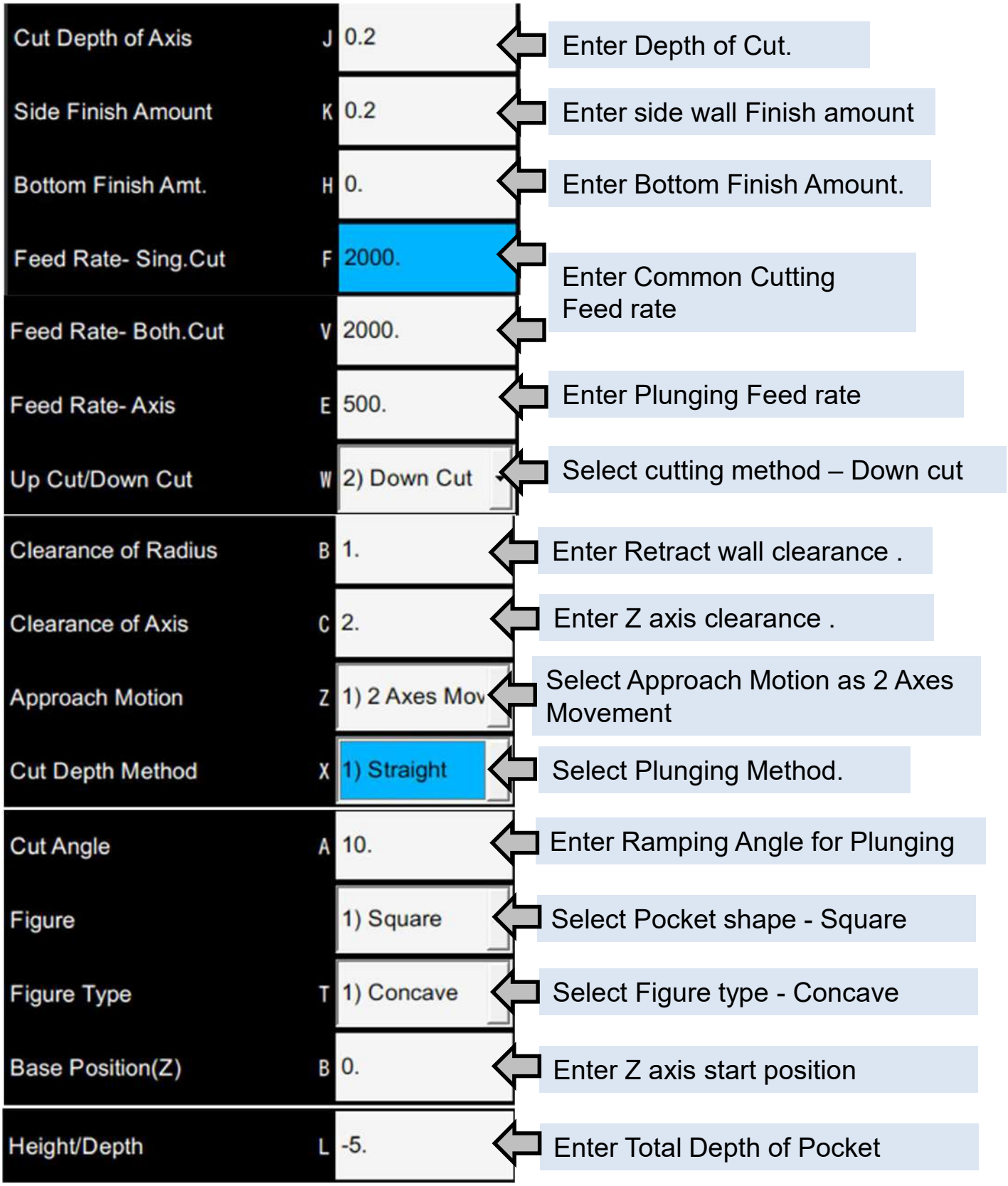


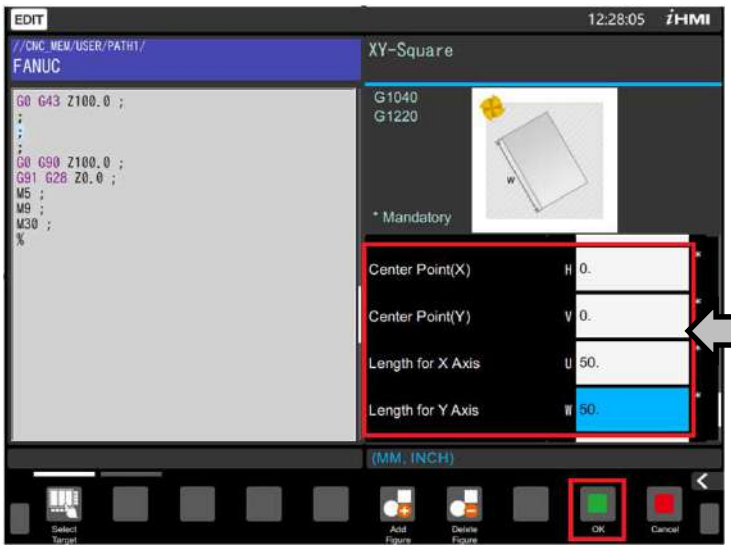
4. Select **Pocketing** & Touch **OK** soft key



5. Fill the Data in Tabular Format to Create Program Easily.
(**“ * ” Star Mark** value is compulsory & Value without star mark is optional)

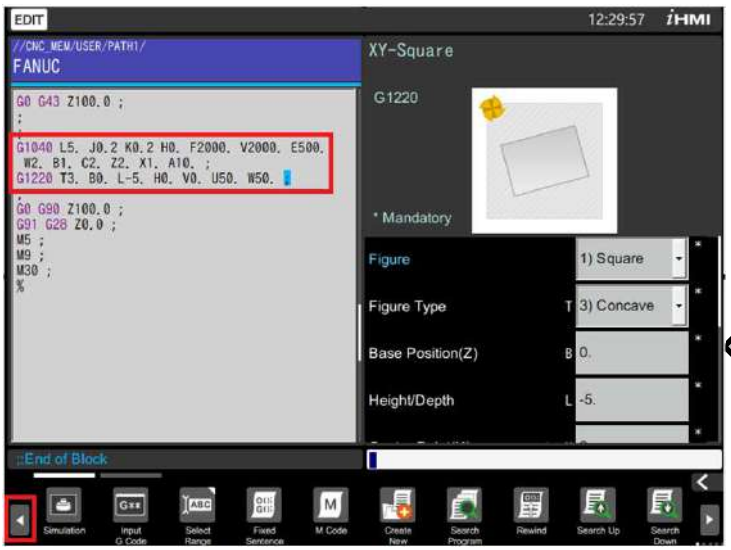
Select Process – Rough , Finish etc.
Bottom Thickness – Optional
Side Thickness - Optional
Enter Stepover (70% of cutter value)


Cut Depth of Axis
J
0.2
Enter Depth of Cut.
Side Finish Amount
K
0.2
Enter side wall Finish amount
Bottom Finish Amt.
H
0.
Enter Bottom Finish Amount.
Feed Rate- Sing.Cut
F
2000.
Enter Common Cutting Feed rate
Feed Rate- Both.Cut
V
2000.
Feed Rate- Axis
E
500.
Enter Plunging Feed rate
Up Cut/Down Cut
W
2) Down Cut
Select cutting method – Down cut
Clearance of Radius
B
1.
Enter Retract wall clearance .
Clearance of Axis
C
2.
Enter Z axis clearance .
Approach Motion
Z
1) 2 Axes Mov
Select Approach Motion as 2 Axes Movement
Cut Depth Method
X
1) Straight
Select Plunging Method.
Cut Angle
A
10.
Enter Ramping Angle for Plunging
Figure
1) Square
Select Pocket shape - Square
Figure Type
T
1) Concave
Select Figure type - Concave
Base Position(Z)
B
0.
Enter Z axis start position
Height/Depth
L
-5.
Enter Total Depth of Pocket



Enter X & Y Coordinates of Pocket
Enter Length & Width of Pocket

6. After all data filled, Touch **OK** soft Key.

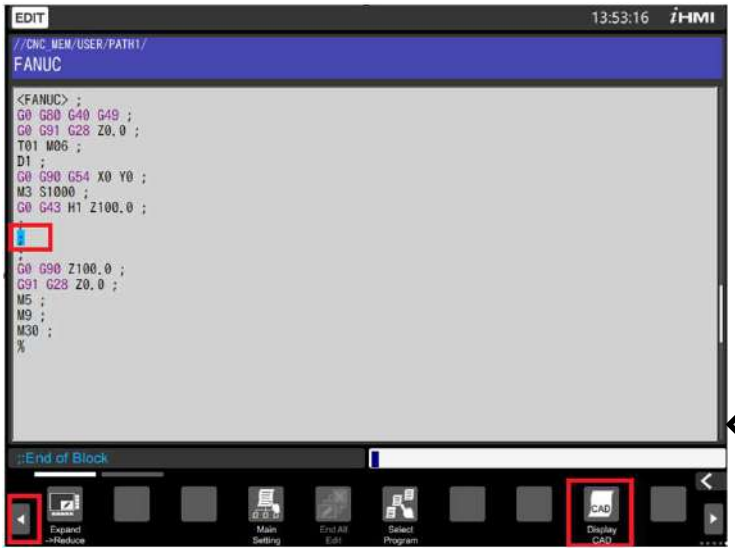


7. **Program lines** will be inserted between starting and ending lines.
8. Touch **Left Next Page** and Touch **Close Guidance** Soft Key to close G Code Guidance window.

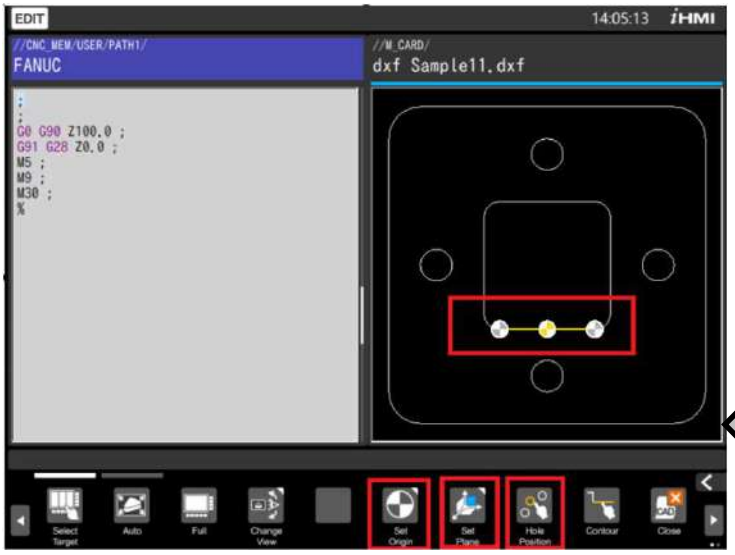
SCAN TO WATCH

1.9 CAD (DXF) File Import :-

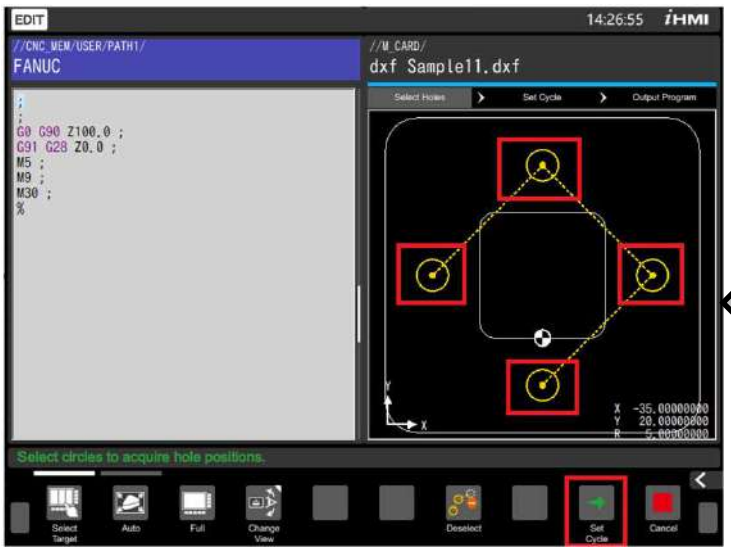
➢ Peck Drilling Program :-



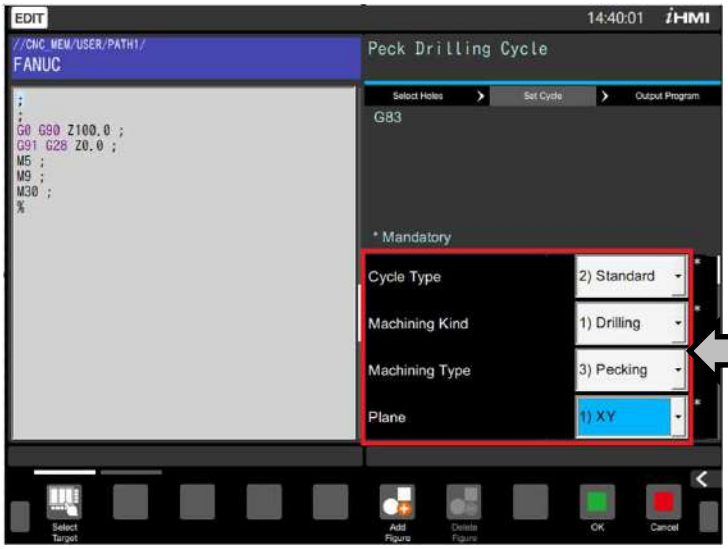
1. In **EDIT** mode , after adding Starting & Ending lines .
2. Keep cursor between Starting & Ending Lines and Touch **Left Next Page**
3. Touch **Display CAD** Soft Key.
4. Select **CAD File** from **M-Card/USB/Shared Folder** and Touch **OK** soft key to open CAD file.



5. Touch **Set Origin** Soft Key & Touch any Point on drawing to set Centre and Touch **OK** soft key
6. Touch **Set Plane** Soft Key & Touch **XY** Soft Key (Plane selection)
7. Touch **Hole Position** Soft Key

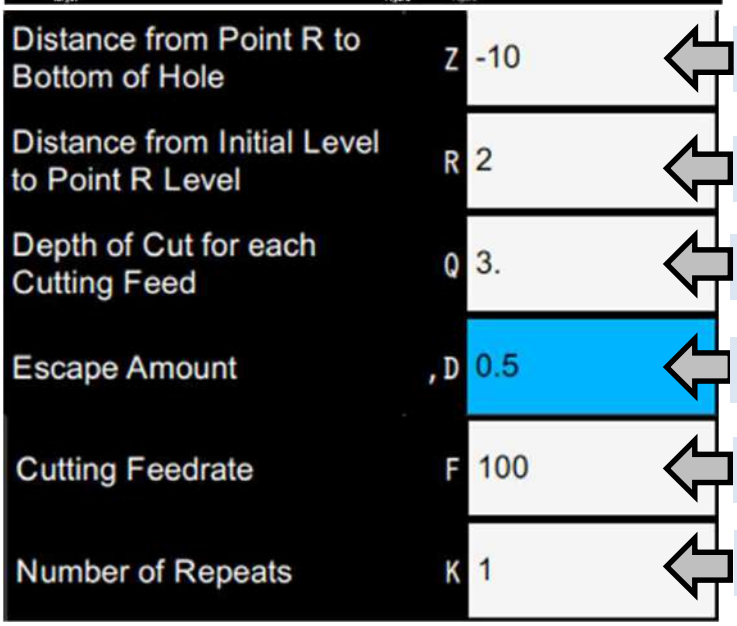


8. Touch **Each hole** to Select Drilling Position.
9. After selecting all holes Touch **Set Cycle** Soft Key



10. Fill the Data in Tabular Format to Create Program Easily.
(**“ * ” Star Mark** value is compulsory & Value without star mark is optional)

Select Cycle – Standard
Select Operation- Drilling , Tapping etc.
Select Type – Peck Drill , Normal etc.
Select Plane as XY



Enter Total Drilling Depth.

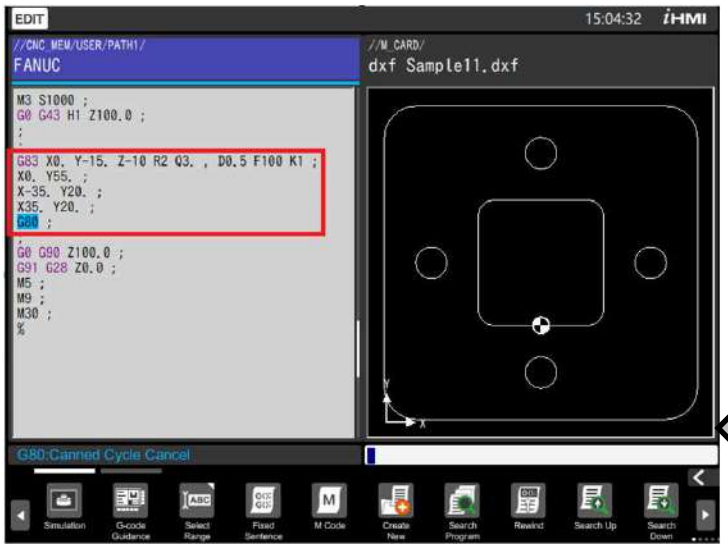
Enter Drill Start Position (R).

Enter Pecking Amount.

Enter Retract Distance.

Enter Drilling Feed rate.

Enter Number of Repetition - 1.

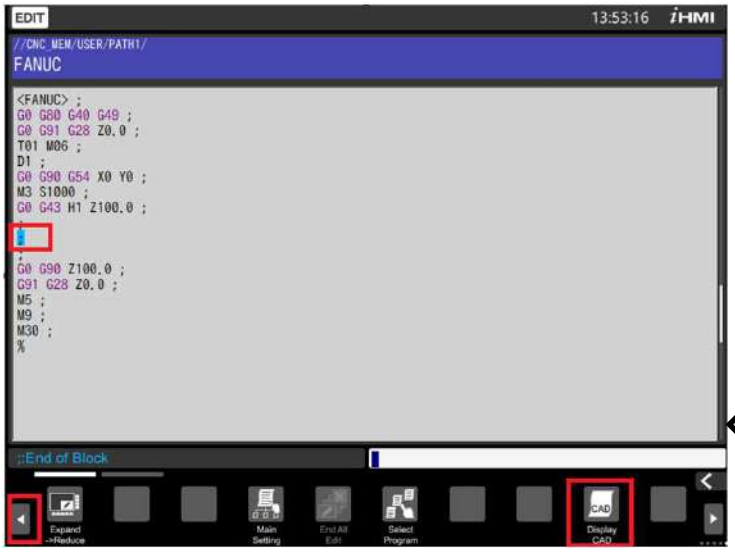


11. After all data filled, Touch **OK** soft Key.

12. **Program lines** will be inserted between starting and ending lines.
13. Touch **Left Next Page** and Touch **CAD Close** Soft Key to Close CAD window.

SCAN TO WATCH

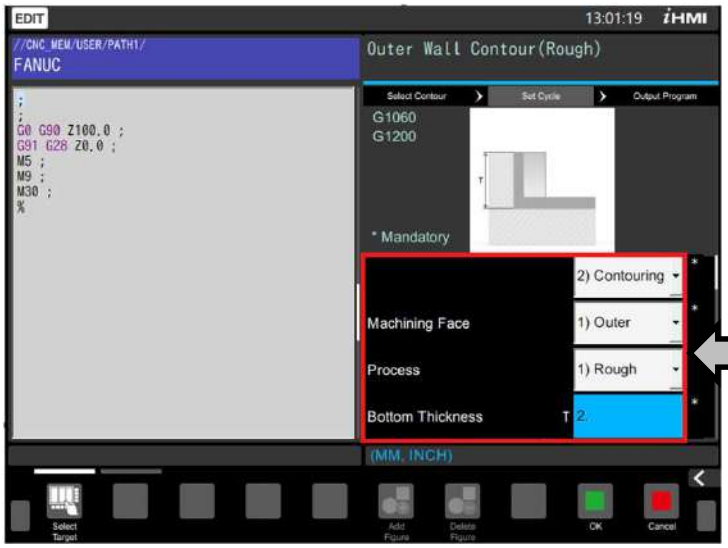
➢ Contouring Program :-



1. In **EDIT** mode , after adding Starting & Ending lines .
2. Keep cursor between Starting & Ending Lines and Touch **Left Next Page**
3. Touch **Display CAD** Soft Key.
4. Select **CAD File** from **M-Card/USB/Shared Folder** and Touch **OK** soft key to open CAD file.

5. Touch **Set Origin** Soft Key & Touch any Point on drawing to set Centre and Touch **OK** soft key
6. Touch **Set Plane** Soft Key & Touch **XY** Soft Key (Plane selection)
7. Touch **Contour** Soft Key

8. Touch **Each Side** of outer profile and make close loop in yellow colour.
9. After selecting all sides Touch **Set Cycle** Soft Key



10. Fill the Data in Tabular Format to Create Program Easily.
(" * " **Star Mark** value is compulsory & Value without star mark is optional)

Select Cycle – Contouring
Select Machining Face - Outer
Select Process – Rough
Enter Total Stock in Z – 2.0

Side Thickness	S	2.	Enter Outer Stock
Cut Depth of Radius	L	5.	Enter Stepover (70% of cutter value)
Cut Depth of Axis	J	0.2	Enter Depth of Cut.
Side Finish Amount	K	0.2	Enter side wall Finish amount
Bottom Finish Amt.	H	0.	Enter Bottom Finish Amount.
Feed Rate- Sing.Cut	F	2000.	Enter Common Cutting Feed rate
Feed Rate- Both.Cut	V	2000.	
Feed Rate- Axis	E	500.	Enter Plunging Feed rate
1st Feed Override %	M	100.	Keep 100 %
Up Cut/Down Cut	W	2) Down Cut	Select cutting method – Down cut
Clearance of Axis	C	2.	Enter Z axis clearance .
Approach Type	P	1) Arc	Select Cutter entry in Arc

Enter Cutter Entry Distance

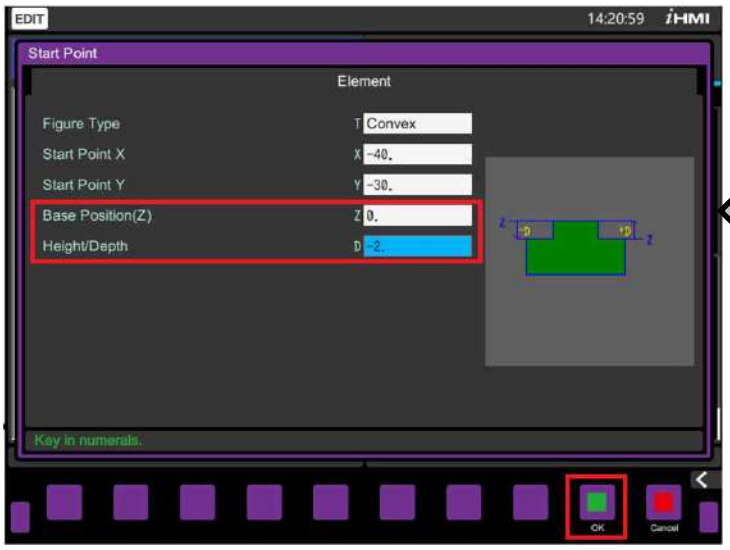
Select Cutter Exit in Arc

Enter Cutter Exit Distance

Select Approach Motion as 2 Axes Movement

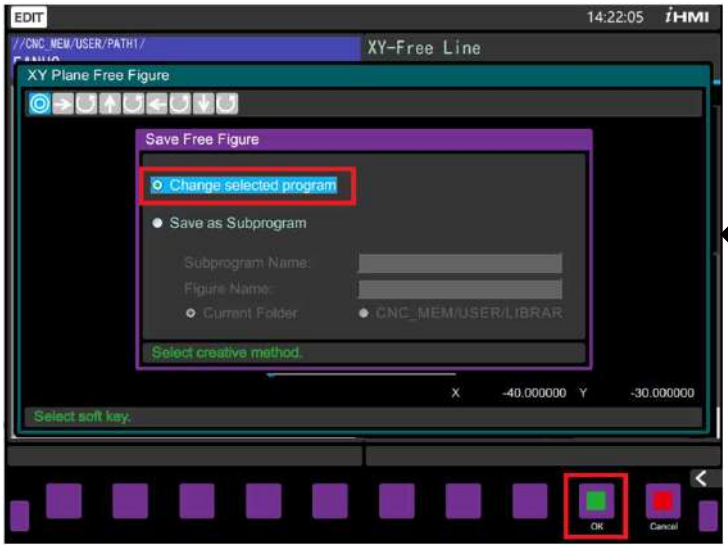
Select Figure as Free Figure

11. After Selecting **Free Figure** it will open Selected Contour Profile.
12. Touch **Alter** Soft Key.



13. Enter **Base Position** (Z start Point) as **'0'** & **Depth** (Total Depth) as **-2**.
14. Touch **OK**  Soft Key

15. Touch **Create**  Soft Key



16. Select Creative Method as **Change Selected Program**.

    & Touch **OK**  Soft Key.

17. Again Touch **OK**  Soft Key.

18. **Program Lines** will be inserted between starting and ending lines.
19. Touch **Left Next Page**  and Touch **CAD Close**  Soft Key to close CAD window.

SCAN TO WATCH

2. Revision Record

Edition	Date	Contents
01	24-Nov, 2025	First Time release
02	31-Mar, 2026	Video QR code added

Service Support Toll Free

1800-102-0050

FANUC INDIA

CNC ROBOT ROBODRILL ROBOSHOT ROBOCUT SERVICE IIoT ACADEMY

41-A, Electronic City, Hosur Road, Bangalore - 560100, Karnataka. INDIA

+91 80 2852 0057 | +91 80 2852 0109 | www.fanucindia.com