FANUC Series 0*i*-TF Plus

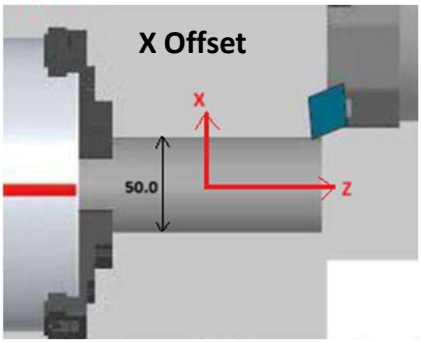
Operator's Handbook
For Lathe System

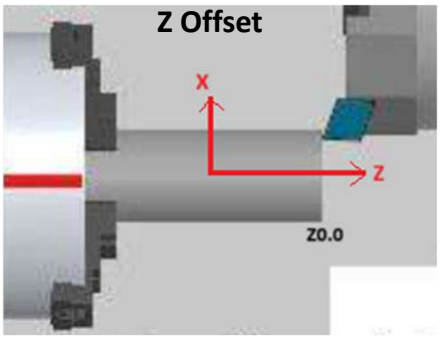
Table of Contents

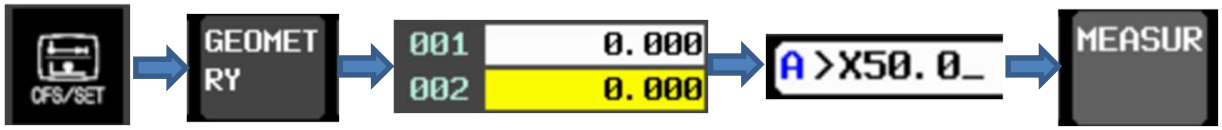
1.1 Tool Offset and Tool Type:-

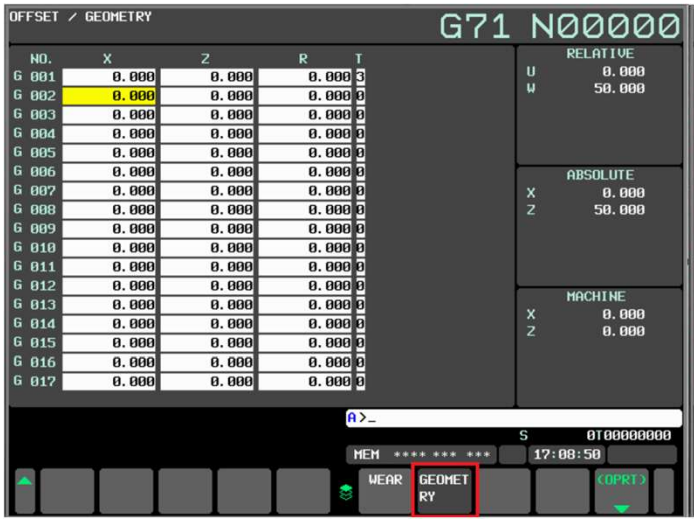
➢ X and Z offset Taking Procedure :-

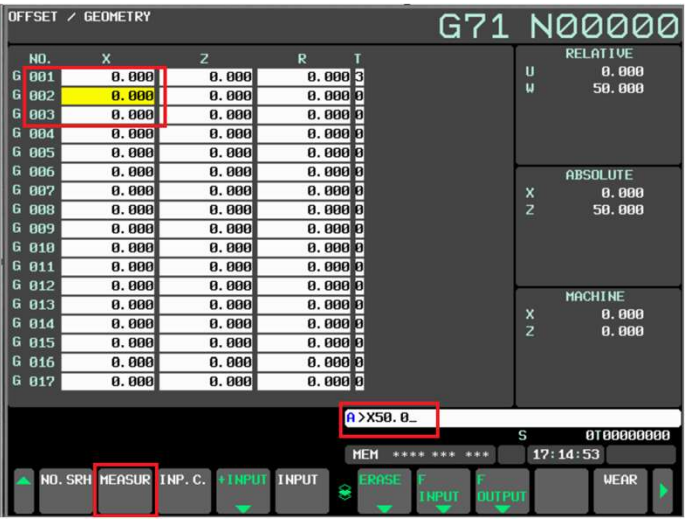
1. Select **HANDLE** mode
2. Touch Tool on **Outer Diameter for X offset**
3. Press **OFF/SET** MDI Key
4. Press **GEOMETRY** soft key
5. Select Tool offset Number
6. Type **X50.0** (Component Diameter)
7. Press **MEASURE** soft key
8. Touch Tool on **Job Face for Z offset .**
9. Type **Z0.0** (Component Face)
10. Press **MEASURE** soft key







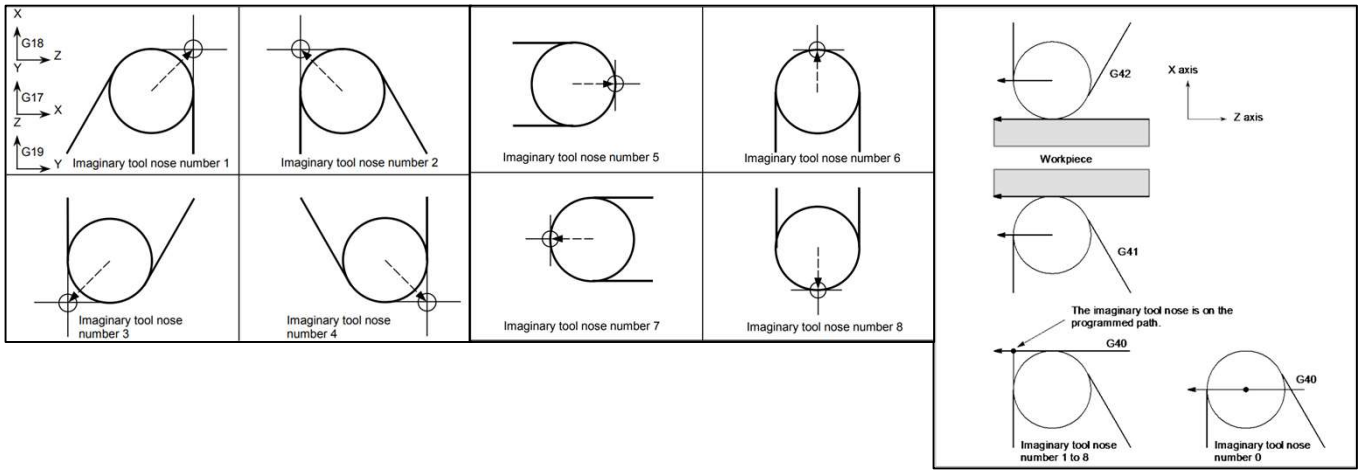


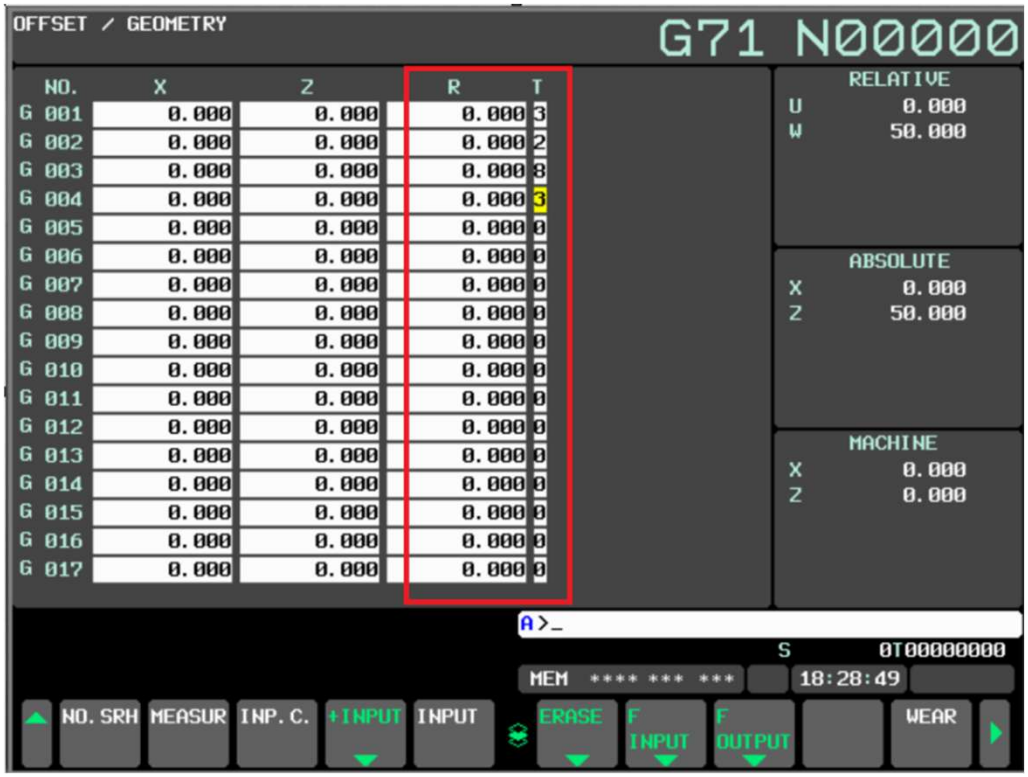


SCAN TO WATCH

➢ Tool Type Selection for various Tools in G41/G42 :-

1. Use Tool type as per below image and put Tool nose number (From 1 to 8) in **T Column** of **OFF/SET** Page.
2. Use G41(Tool Nose Radius Left) or G42(Tool Nose Radius right) in Program depending on Tool movement Direction and use G40 to cancel Tool Nose Radius
3. Put Tool insert Radius in **R Column** of **OFF/SET** Page.



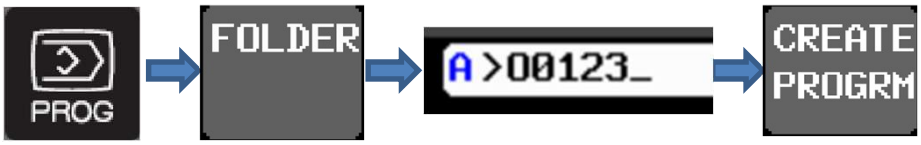


SCAN TO WATCH

1.2 Program Editing Functions :-

➢ New Program Create in CNC MEMORY :-

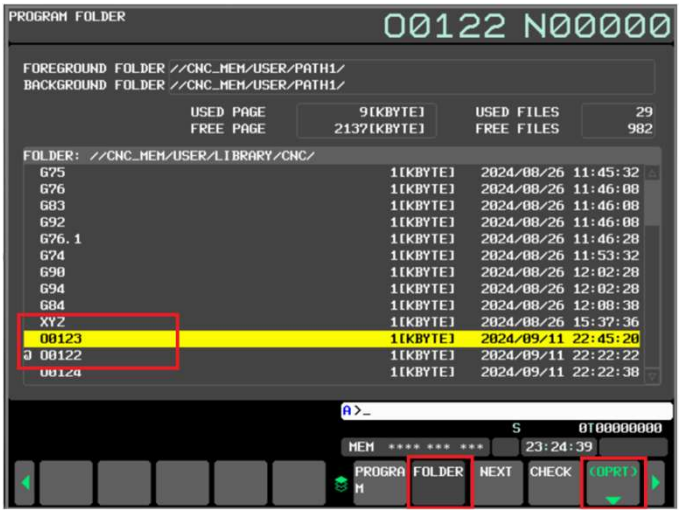
1. Select **EDIT** mode
2. Press **PROG** MDI Key
3. Press **FOLDER** soft key
4. Type Number of Program (E.g. O0123) or Program Name (E.g. PART1)
5. Press **CREATE PROGRAM** soft key



SCAN TO WATCH

➢ Program Rename :-

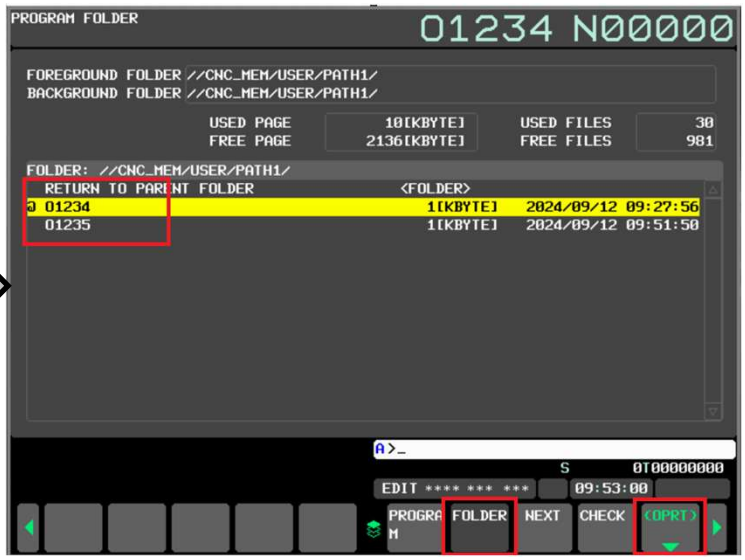
1. Select **EDIT** mode
2. Press **PROG** MDI Key
3. Press **FOLDER** soft key
4. Press **OPRT** soft key
5. Keep Cursor on Program to Rename
6. Type new name or Number
7. Press **RENAME** soft key



SCAN TO WATCH

➢ Program Lines Copy and Paste to Other Program:-

1. Select **EDIT** mode
2. Press **PROG** MDI key
3. Press **FOLDER** soft key
4. Select Program which need to copy and Press **INPUT** hard key
5. Press **OPRT** soft key



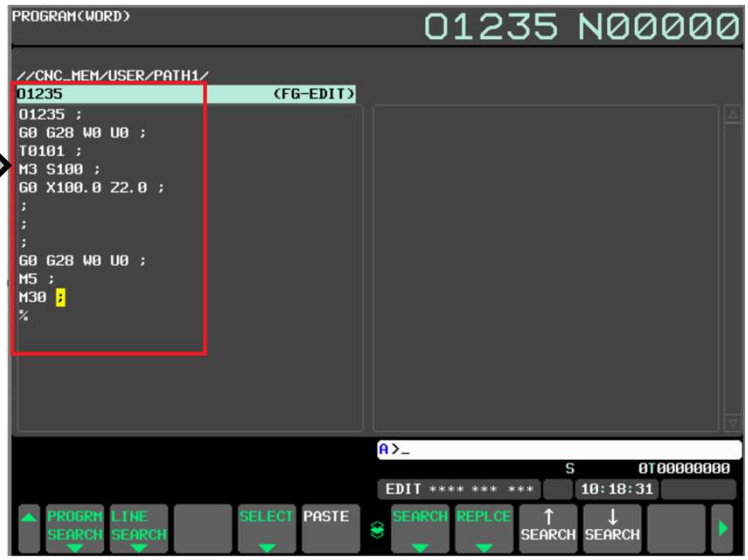
6. Keep cursor on line from where to copy and Press **SELECT** soft key

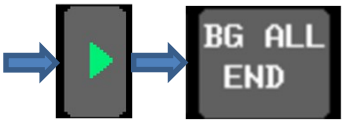
7. Press Continue **Down arrow** MDI key to select Multiple lines
8. Press **COPY** soft key

9. Type other program name and Press **Down arrow** MDI key

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

11. After Program Paste Press Next Page
12. Press **BG ALL END** soft key





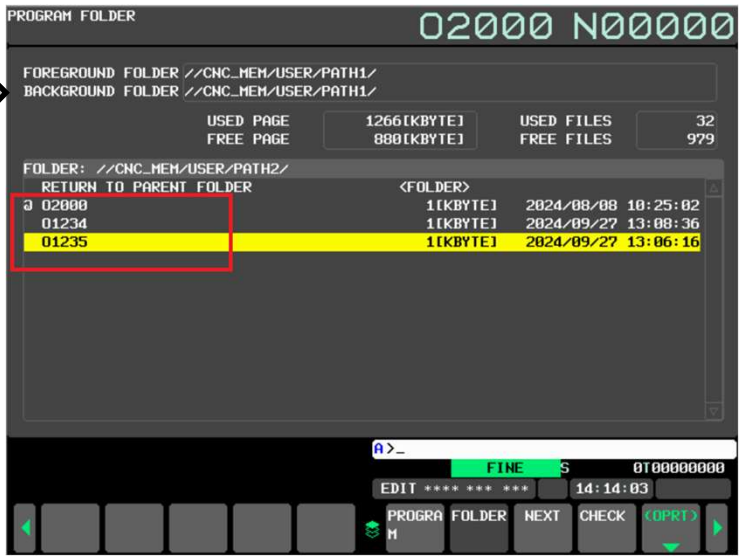
SCAN TO WATCH

➢ Program Lines Copy and Paste In BG EDIT :-

1. Select **EDIT** mode
2. Press **PROG** MDI key
3. Press **FOLDER** soft key
4. Select Program other than main program which need to copy and Press **INPUT** key

5. Press **FOLDER** soft key

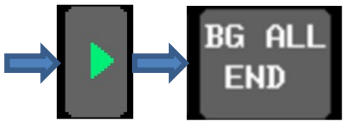
6. Select 2nd program and Press **INPUT** key



7. Press **SHIFT** MDI key then **Left Arrow** MDI key to select left side screen.
8. Press **OPRT** soft key

9. Keep cursor on line from where To copy and Press **SELECT** soft key
10. Press Continue **Down arrow** MDI key to select Multiple lines
11. Press **COPY** soft key

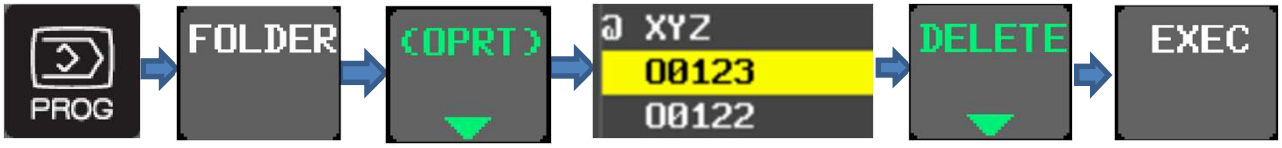
12. Press **SHIFT** MDI key then **Right Arrow** MDI key to select right side screen
13. Press **PASTE** soft key
14. After Program Paste, Press **Next Page** and Press **BG ALL END** soft key



SCAN TO WATCH

➢ Program Delete :-

1. Select **EDIT** mode
2. Press **PROG** MDI Key
3. Press **FOLDER** soft key
4. Press **OPRT** soft key
5. Keep Cursor on Program to Delete
6. Press **DELETE** soft key
7. Press **EXEC** soft key



SCAN TO WATCH

➢ Manual MST Function :-

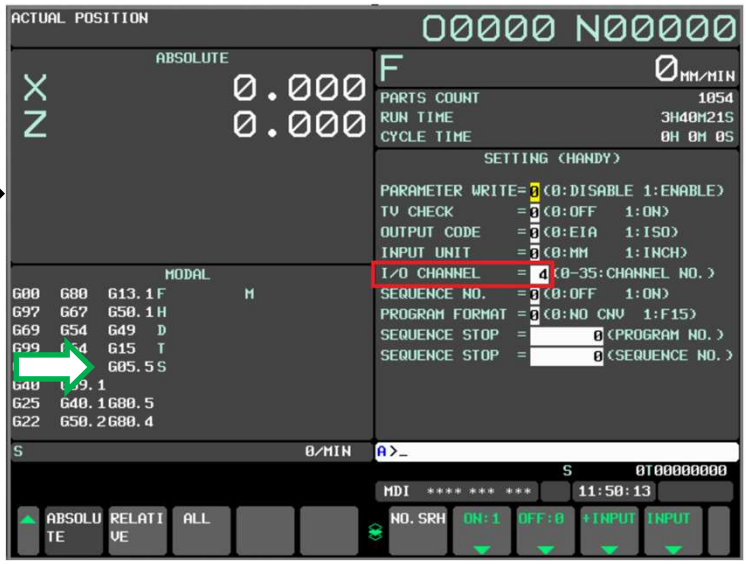
1. Manual MST Function addresses the Operator workflow complexity for executing MST Codes on Fanuc CNC.
2. Single Application to execute MST Code in any mode.
3. Faster code execution.
4. Eliminates need of mode switching and screen transition.

SCAN TO WATCH

1.3 Program Management Functions :-

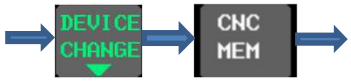
➢ Program Copy From CF/USB to CNC MEMORY (Multi List) :-

1. Select **MDI** mode.
2. Press **OFF/SET** MDI key until this screen appears
3. Make **I/O CHANNEL = 4**(CF) **/ 17**(USB)



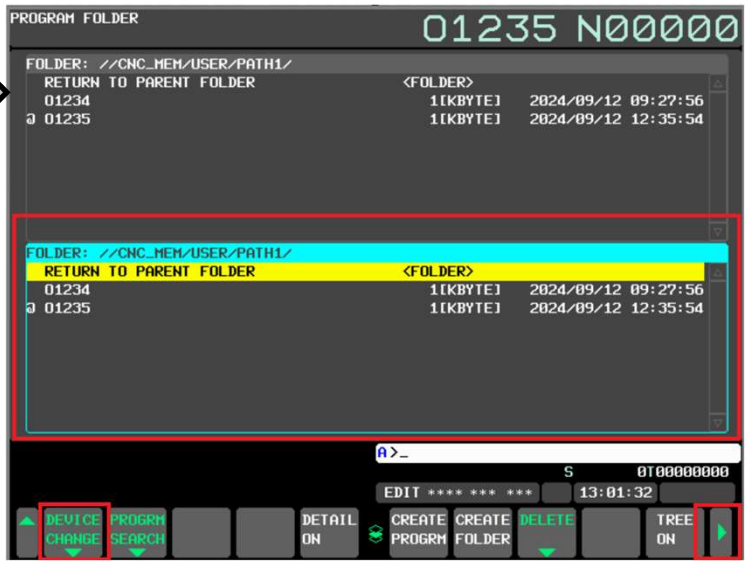
1. Select **EDIT** mode
2. Press **PROG** MDI key
3. Press **FOLDER** soft key
4. Press **OPRT** soft key

5. Press **DEVICE CHANGE** soft key
6. Press **CNC MEM soft key**

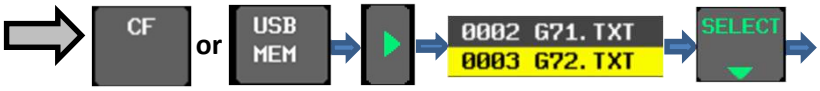


7. Press **Next Page**
8. Press **MULTI LIST** soft key
9. Press **<SHIFT> Down arrow** MDI key to Switch Screen
10. Press **Next Page** 2 Times

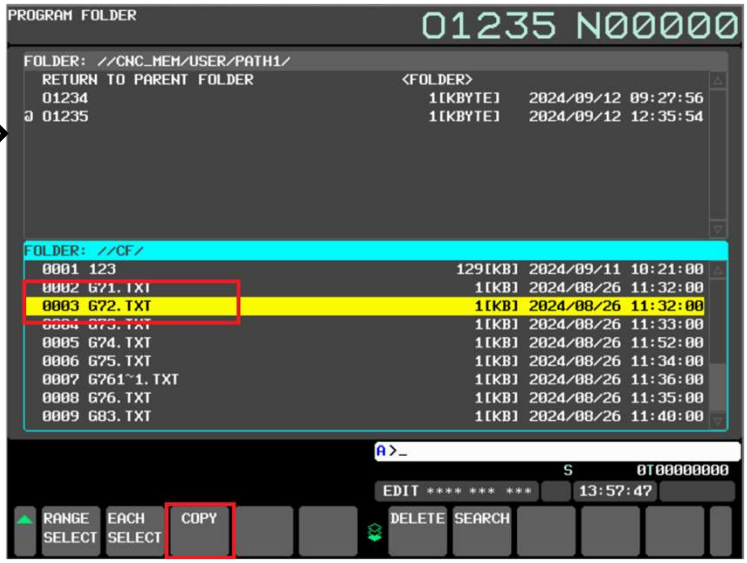
11. Press **DEVICE CHANGE** soft key



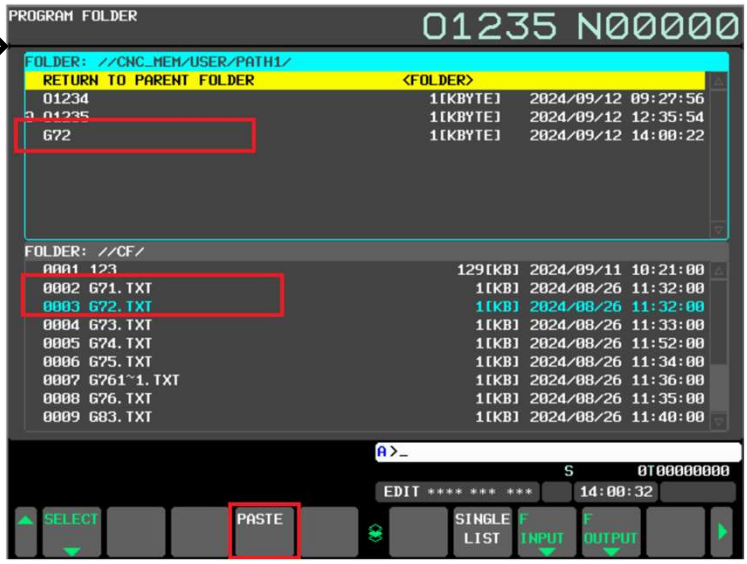
12. Press **CF/USB** soft key
13. Press **Next Page**
14. Keep cursor on Program and Press **SELECT** soft key



15. Press **COPY** soft key
16. Press **<SHIFT> Up Arrow** MDI key



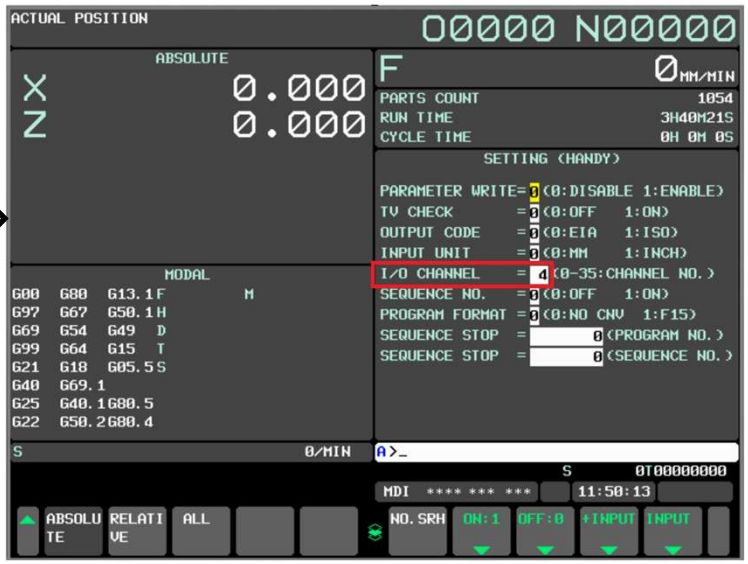
17. Press **PASTE** soft key



SCAN TO WATCH

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

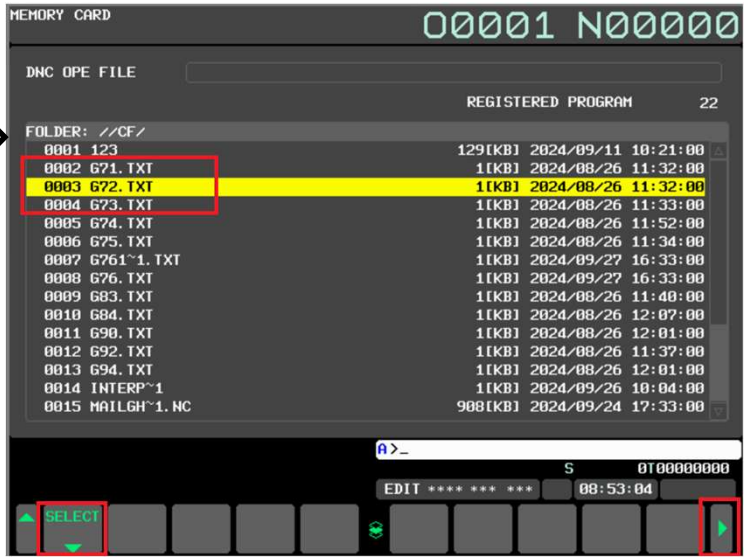
1. Select **MDI** mode.
2. Press **OFF/SET** MDI key until this screen appears
3. Make **I/O CHANNEL = 4**(CF) / **17**(USB)



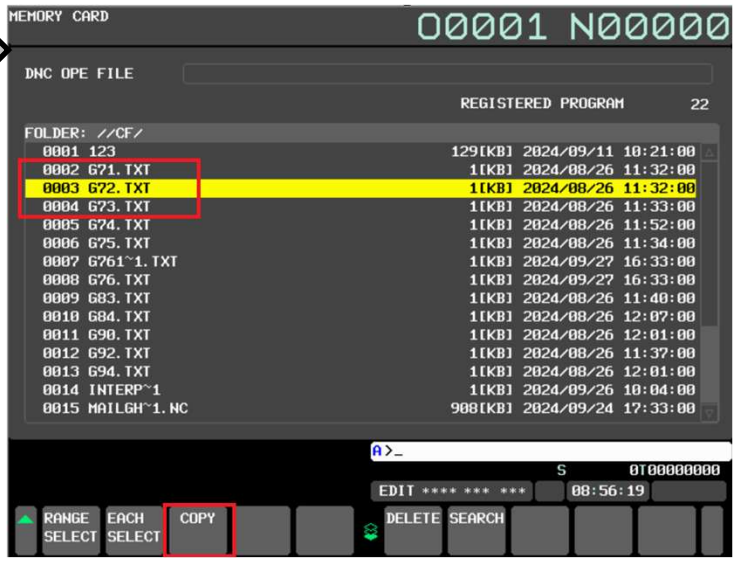
1. Select **EDIT** mode
2. Press **PROG** MDI key
3. Press **FOLDER** soft key
4. Press **OPRT** soft key

5. Press **DEVICE CHANGE** soft key
6. Press **CF/USB** soft key

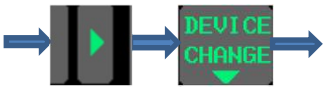
7. Press **Next Page**
8. Keep cursor on Program and Press **SELECT** soft key



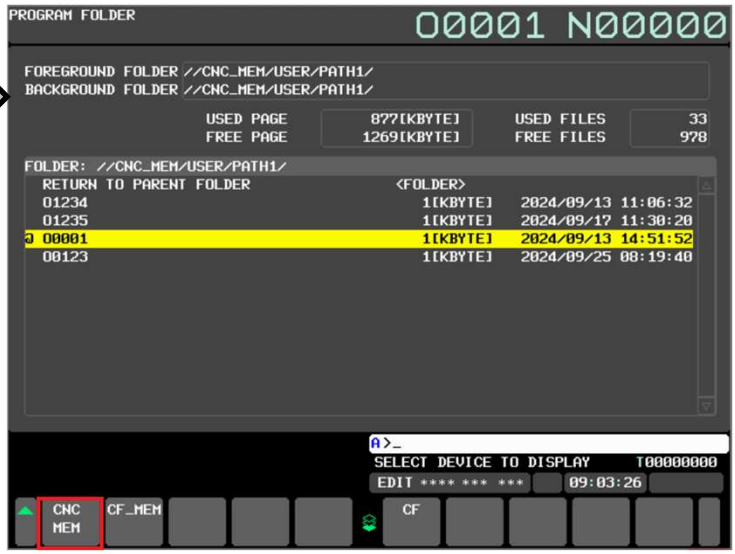
9. Press **COPY** soft key



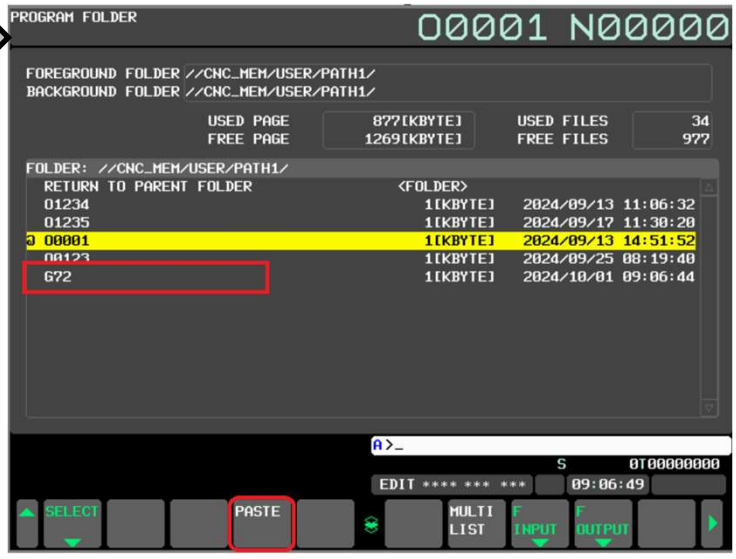
10. Press **Next Page** soft key 2 times
11. Press **DEVICE CHANGE**



12. Press **CNC MEM** soft key
13. Press Next Page



14. Press **PASTE** soft key



SCAN TO WATCH

1.4 Program Execution Functions :-

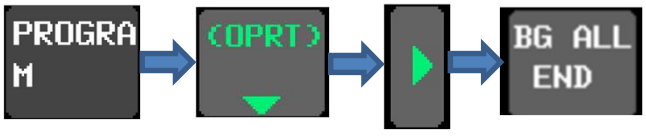
➢ Main Program Select from List:-

1. Select **EDIT** mode
2. Press **PROG** MDI key
3. Press **FOLDER** soft key
4. Press **OPRT** soft key
5. Keep Cursor on Program
6. Press **MAIN PROGRAM** soft key

7. Make sure that "∂" symbol came in front of program name ∂ 00123

NOTE:-
While selecting main program if SETTING A MAIN PROGRAM FAILED this alarm Blinks then Press BG ALL END (soft key)

➢ Procedure for BG ALL END



SCAN TO WATCH

➢ Sequence (N) Number Search :-

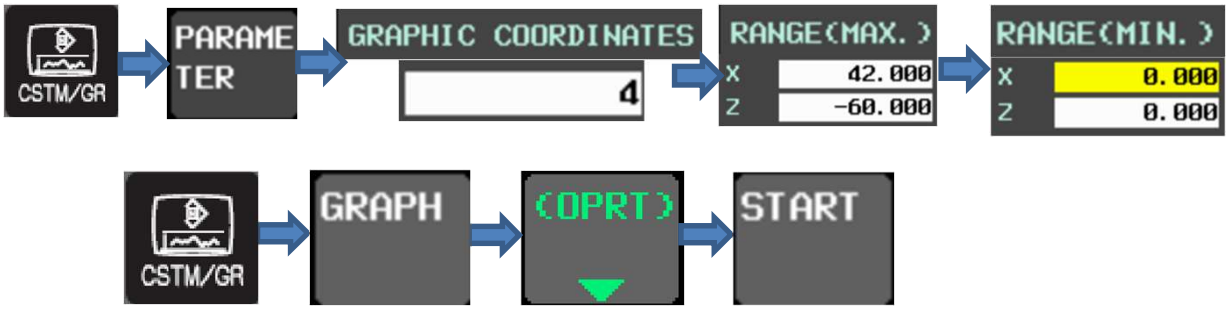
1. Select **AUTO OR MEMORY** mode
2. Press **PROG** MDI key
3. Press **PROGRAM** soft key
4. Type Sequence Number (N1 or N2 or N3 etc.)
5. Press **SEQ NO SEARCH** soft key

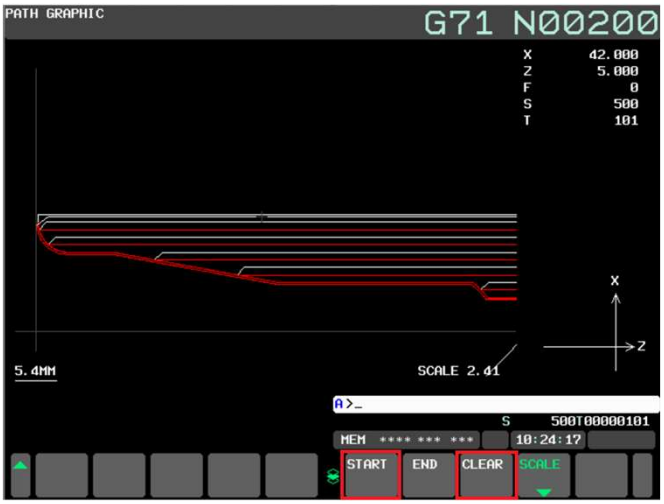
SCAN TO WATCH

➢ 2D Simulation of Running Program:-

- Following Parameter setting should done before Cycle start

1. Select **AUTO or MEMORY** mode.
2. Press **CSTM/GR** MDI key
3. Press **PARAMETER** soft key
4. Enter **GRAPHIC COORDINATES = 4**
5. Enter **RANGE(MAX.) X = 42.0** (Component Diameter)
6. Enter **RANGE(MAX.) Z = -60.0** (Turning Length in Negative)
7. Enter **RANGE(MIN.) X = 0.0**
8. Enter **RANGE(MIN.) Z = 0.0**
9. After entering above values **SCALE and GRAPHIC CENTER** Automatically Adjusted
10. To check graph while running program Press **CSTM/GR** MDI key
11. Press **GRAPH** soft key
12. Press **OPRT** soft key
13. Press **START** soft key.
13. Press **CLEAR** soft key to Erase Graph





SCAN TO WATCH

1.5 Functions to Simplify Programming (Canned Cycles):-

➢ G90/G77 (Straight or Taper Simple Turning cycle) :-

- **Cycle Format –**

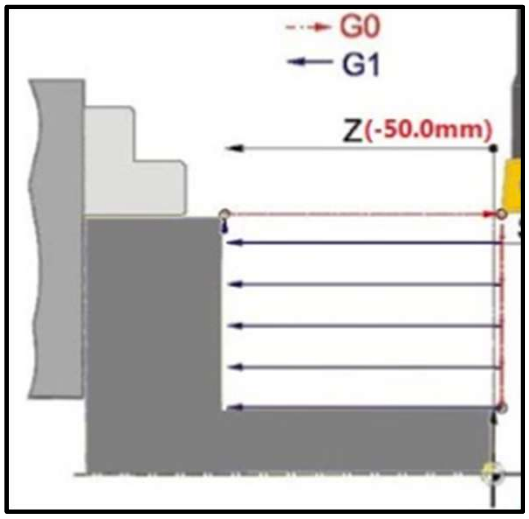
G90 X_ Z_ R_ F_

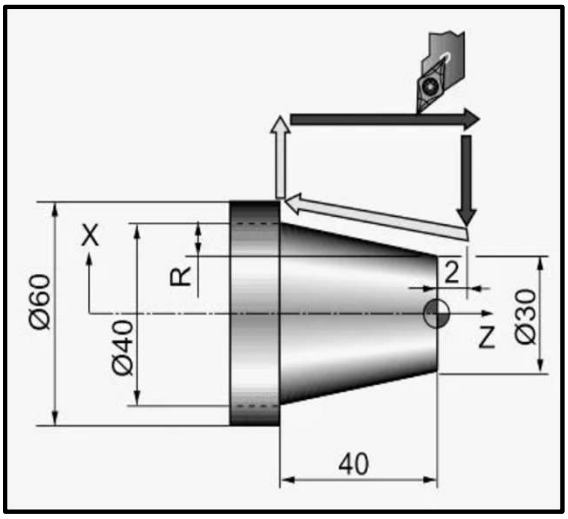
X_ = Turning X position
Z_ = Turning length
R_ = Taper amount
F_ = Feed rate (mm/rev)

- **G code Selection –**

1. G90 – For G code system A
2. G77 – For G code system B

- **Drawing with Sample Programs :-**






```
O0001
G28 W0.0 U0.0
T0101
M03 S500
G0 X60.0 Z5.0
G90 X50.0 Z-50.0 F0.3
X45.0
X40.0
X35.0
X30.0
G0 X65.0
G0 Z20.0
G28 W0.0 U0.0
M05
M30
%
```

```
O0001
G28 W0.0 U0.0
T0101
M03 S500
G0 X60.0 Z2.0
G90 X50.0 Z-40.0 R-5.0 F0.3
X45.0
X40.0
X35.0
X30.0
G0 X60.0
G0 Z20.0
G28 W0.0 U0.0
M05
M30
%
```

➢ G94/G79 (Simple or Taper Facing cycle) :-

- **Cycle Format –**

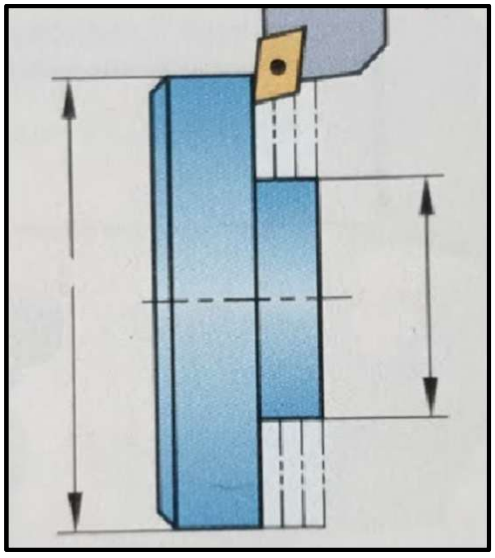
G94 X_ Z_ R_ F_

X_ = Facing end position
Z_ = Facing cut value
R_ = Taper amount
F_ = Feed rate (mm/rev)

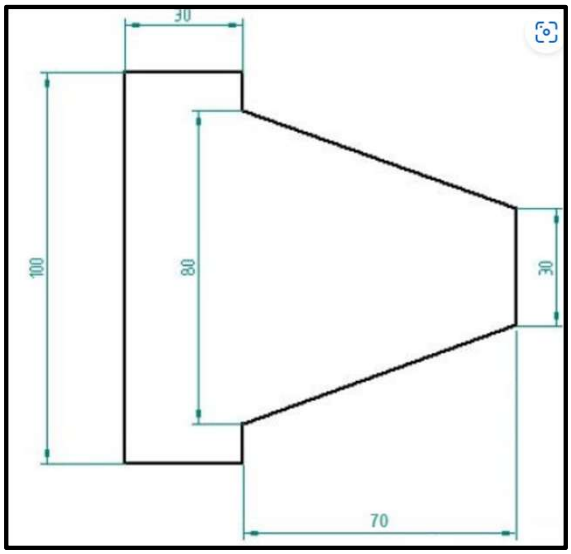
- **G code Selection –**

1. G94 – For G code system A
2. G79 – For G code system B

- **Drawing with Sample Programs :-**



```
O0002
G28 W0.0 U0.0
T0101
M03 S500
G0 X60.0 Z5.0
G94 X20.0 Z-1.0 F0.3
Z-2.0
Z-3.0
Z-4.0
Z-5.0
G0 Z20.0
G0 X60.0
G28 W0.0 U0.0
M05
M30
%
```




```
O0002
G28 W0.0 U0.0
T0101
M03 S500
G0 X100.0 Z2.0
G94 X30.0 Z-1.0 R-25.0 F0.3
Z-2.0
Z-3.0
Z-4.0
Z-5.0
G0 Z20.0
G0 X60.0
G28 W0.0 U0.0
M05
M30
%
```

➢ G92/G78 (Threading cycle) :-

- **Cycle Format –**

G92 X_ Z_ F_

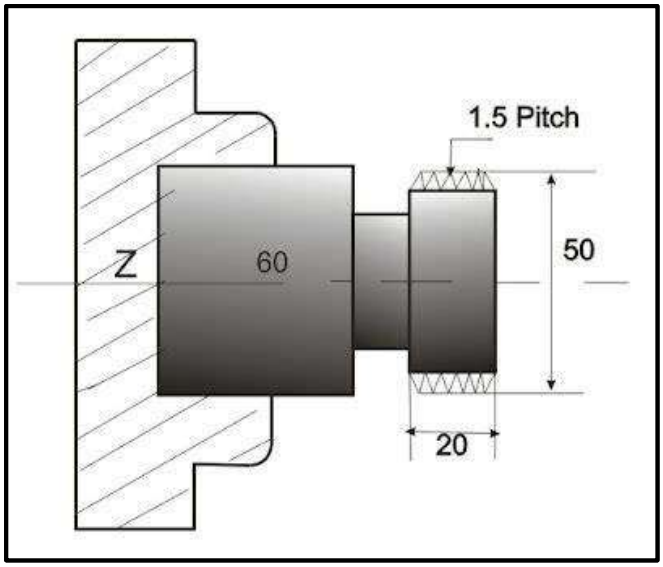
X_ = 1st cut of Thread
Z_ = Thread length
F_ = Pitch

- **G code Selection –**

1. G92 – For G code system A
2. G78– For G code system B

- **Drawing with Sample Programs :-**

```
O0003
G28 W0.0 U0.0
T0101
M03 S500
G0 X55.0 Z5.0
G92 X50.0 Z-20.0 F1.5
X49.7
X49.4
X49.1
X48.8
X48.5
X48.4
X48.16
G0 X55.0
G0 Z20.0
G28 W0.0 U0.0
M05
M30
%
```



- **Calculation for Minor Diameter :-**

Thread Height = Pitch X 0.613 = 1.5 X 0.613 = 0.9195mm
Minor Dia. = Major Dia. – 2 x Thread Height (P) = 50 – 2x0.9195 = 48.16

➢ G71 (Stock Removal – Turning Cycle)

- **Cycle Format –**

G71 U_ R_
G71 P_ Q_ U_ W_ F_

U_ = Radial Depth of cut
R_ = Tool Retract amount
P_ = First sequence number
Q_ = Last sequence number
U_ = Finishing amount in X
W_ = Finishing amount in Z
F_ = Feed rate (mm/rev)

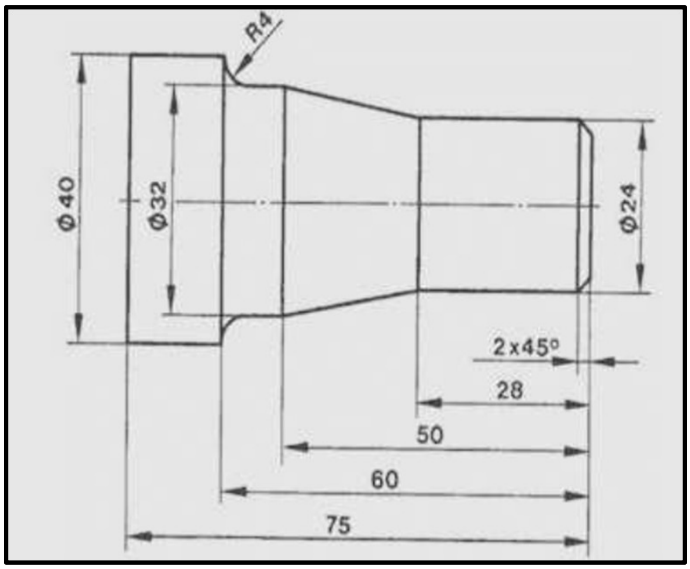
- **Finish Cycle Format –**

G70 P_ Q_ F_

P_ = First sequence number
Q_ = Last sequence number
F_ = Feed rate (mm/rev)

- **Drawing with Sample Programs :-**

```
O0004
G28 W0.0 U0.0
T0101
M03 S500
G0 X42.0 Z5.0
G71 U2.0 R1.0
G71 P100 Q200 U0.5 W0.2 F0.2
N100 G01 X20.0 F0.2
G01 Z0.0
G01 X24.0 Z-2.0
G01 Z-28.0
G01 X32.0 Z-50.0
G01 Z-56.0
N200 G02 X40.0 Z-60.0 R4.0
G70 P100 Q200 F0.1
G0 X42.0 Z5.0
G28 W0.0 U0.0
M05
M30
%
```



➢ G72 (Stock Removal – Facing Cycle)

- **Cycle Format –**

G72 W_ R_
G72 P_ Q_ U_ W_ F_

W_ = Facial Depth of cut
R_ = Tool Retract amount
P_ = First sequence number
Q_ = Last sequence number
U_ = Finishing amount in X
W_ = Finishing amount in Z
F_ = Feed rate (mm/rev)

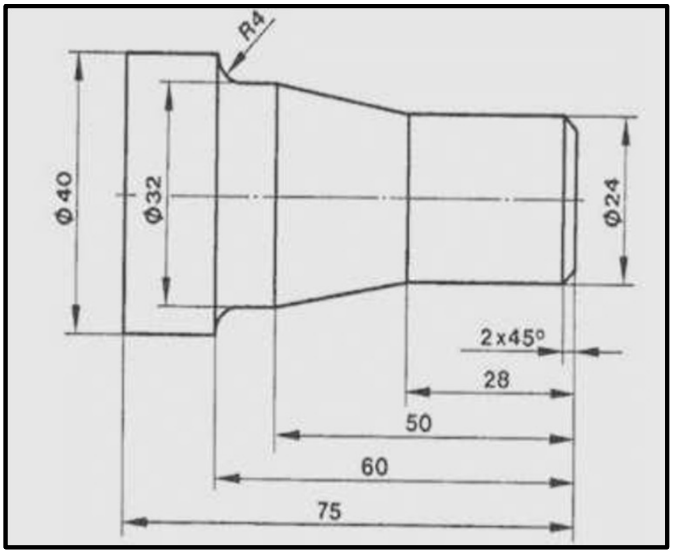
- **Finish Cycle Format –**

G70 P_ Q_ F_

P_ = First sequence number
Q_ = Last sequence number
F_ = Feed rate (mm/rev)

- **Drawing with Sample Programs :-**

```
O0004
G28 W0.0 U0.0
T0101
M03 S500
G0 X42.0 Z5.0
G72 W2.0 R1.0
G72 P100 Q200 U0.5 W0.2 F0.2
N100 G01 X40.0 Z-60.0 F0.2
G03 X32.0 Z-56.0 R4.0
G01 Z-50.0
G01 X24.0 Z-28.0
G01 Z-2.0
N200 G01 X20.0 Z0.0
G70 P100 Q200 F0.1
G0 X42.0 Z5.0
G28 W0.0 U0.0
M05
M30
%
```



➢ G73 (Pattern Repeating cycle) :-

- **Cycle Format –**

G73 U_ W_ R_
G73 P_ Q_ U_ W_ F_

U_ = X axis total stock
W_ = Z axis total stock
R_ = Number of Repeat Cuts
P_ = First sequence number
Q_ = Last sequence number
U_ = Finishing amount in X
W_ = Finishing amount in Z
F_ = Feed rate (mm/rev)

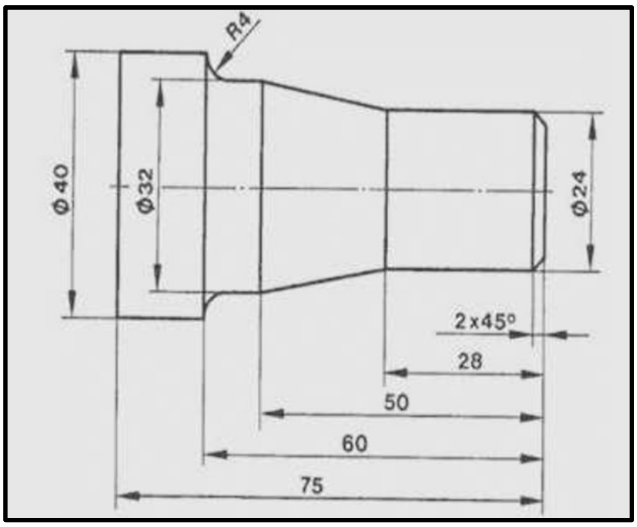
- **Finish Cycle Format –**

G70 P_ Q_ F_

P_ = First sequence number
Q_ = Last sequence number
F_ = Feed rate (mm/rev)

- **Drawing with Sample Programs :-**

```
O0005
G28 W0.0 U0.0
T0101
M03 S500
G0 X42.0 Z5.0
G73 U4.0 W1.0 R3.0
G73 P100 Q200 U0.5 W0.2 F0.2
N100 G01 X20.0 Z0.0 F0.2
G01 X24.0 Z-2.0
G01 Z-28.0
G01 X32.0 Z-50.0
G01 Z-56.0
N200 G02 X40.0 Z-60.0 R4.0
G70 P100 Q200 F0.1
G0 X42.0 Z5.0
G28 W0.0 U0.0
M05
M30
%
```



➢ G74 (Face Grooving cycle) :-

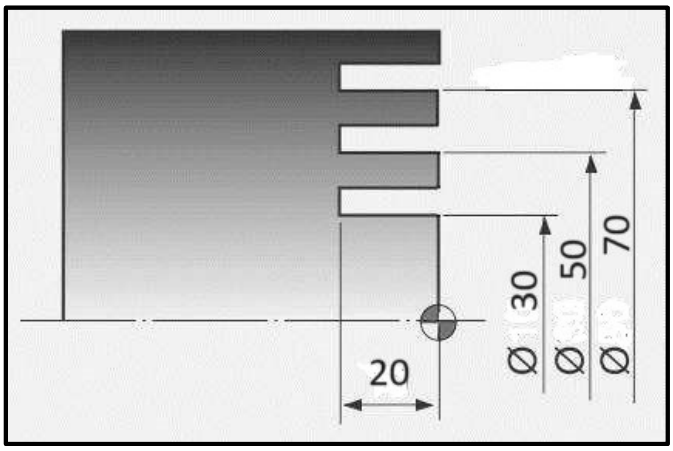
- **Cycle Format –**

G74 R_
G74 X_ Z_ P_ Q_ F_

R_ = Tool Retract amount
X_ = Last Groove diameter
Z_ = Total Groove Depth
P_ = Radial distance between 2 Grooves in Micron
Q_ = Pecking amount in Micron
F_ = Feed rate (mm/rev)

- **Drawing with Sample Programs :-**

```
O0006
G28 W0.0 U0.0
T0101
M03 S500
G0 X30.0 Z5.0
G74 R1.0
G74 X70.0 Z-20.0 P10000 Q3000 F0.2
G0 Z20.0
G28 W0.0 U0.0
M05
M30
%
```



➢ G75 (OD Grooving cycle) :-

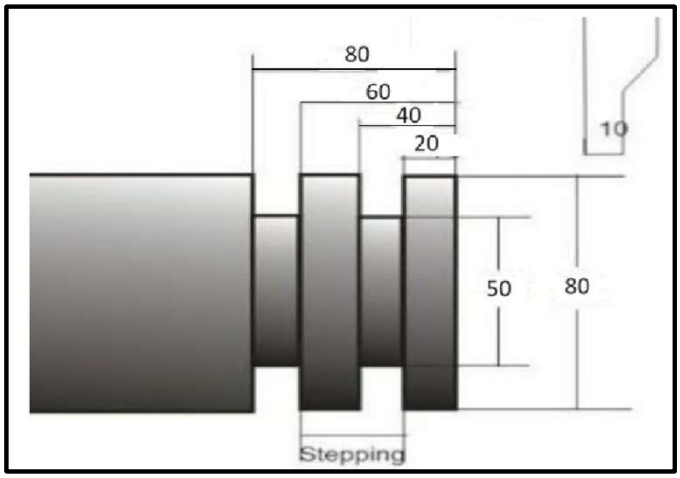
- **Cycle Format –**

G75 R_
G75 X_ Z_ P_ Q_ F_

R_ = Tool Retract amount
X_ = Groove diameter
Z_ = Last Groove Z value
P_ = Radial pecking depth in Micron
Q_ = distance between 2 grooves in Micron
F_ = Feed rate (mm/rev)

- **Drawing with Sample Programs :-**

```
O0007
G28 W0.0 U0.0
T0101
M03 S500
G0 X85.0 Z5.0
G01 Z-40.0
G75 R1.0
G75 X50.0 Z-80.0 P1000 Q40000 F0.2
G0 X85.0
G0 Z10.0
G28 W0.0 U0.0
M05
M30
%
```



➢ G76 (Threading cycle) :-

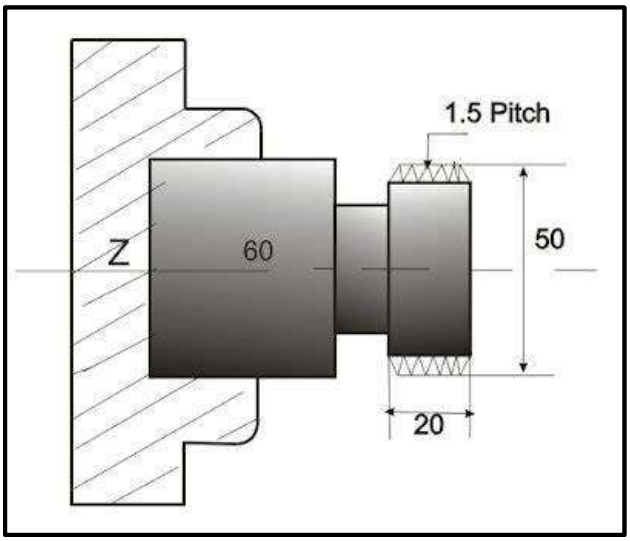
- **Cycle Format –**

G76 P(a_b_c_) Q_ R_
G76 X_ Z_ P_ Q_ F_

a_ = Number of finish pass
b_ = End chamfer amount
c_ = Tool Tip angle
Q_ = Depth of Threading cut in Micron
R_ = Finishing amount
X_ = Minor Diameter
Z_ = Thread length
P_ = Thread height in Micron
Q_ = 1st depth of cut in micron
F_ = Pitch

- **Drawing with Sample Programs :-**

```
O0008
G28 W0.0 U0.0
T0101
M03 S500
G0 X55.0 Z5.0
G76 P020000 Q200 R0.05
G76 X48.16 Z-20.0 P919 Q100 F1.5
G0 X55.0
G0 Z20.0
G28 W0.0 U0.0
M05
M30
%
```



- **Calculation for Minor Diameter :-**

Thread Height (P) = Pitch X 0.613 = 1.5 X 0.613 = 0.9195mm
Minor Dia. = Major Dia. – 2 x Thread Height (P) = 50 – 2x0.9195 = 48.16

➢ G76 (Double start Threading cycle) :-

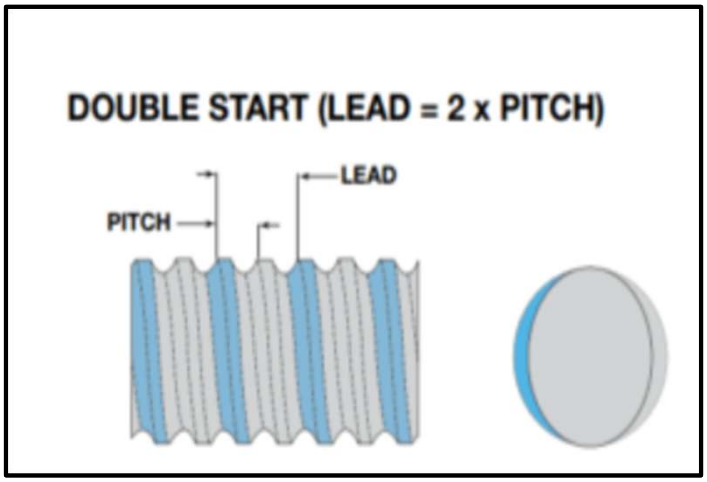
- **Cycle Format –**

G76 P(a_b_c_) Q_ R_
G76 X_ Z_ P_ Q_ F_

a_ = Number of finish pass
b_ = End chamfer amount
c_ = Tool Tip angle
Q_ = Depth of Threading cut in Micron
R_ = Finishing amount
X_ = Minor Diameter
Z_ = Thread length
P_ = Thread height in Micron
Q_ = 1st depth of cut in Micron
F_ = Lead (Pitch x No of start)

- **Drawing with Sample Programs :-**

```
O0009
G28 W0.0 U0.0
T0101
M03 S500
G0 X55.0 Z5.0
G76 P020000 Q200 R0.05
G76 X48.16 Z-20.0 P919 Q100 F3.0
G0 W1.5
G76 P020000 Q200 R0.05
G76 X48.16 Z-20.0 P919 Q100 F3.0
G0 X55.0
G0 Z20.0
G28 W0.0 U0.0
M05
M30
%
```



- **Calculation for Minor Diameter :-**

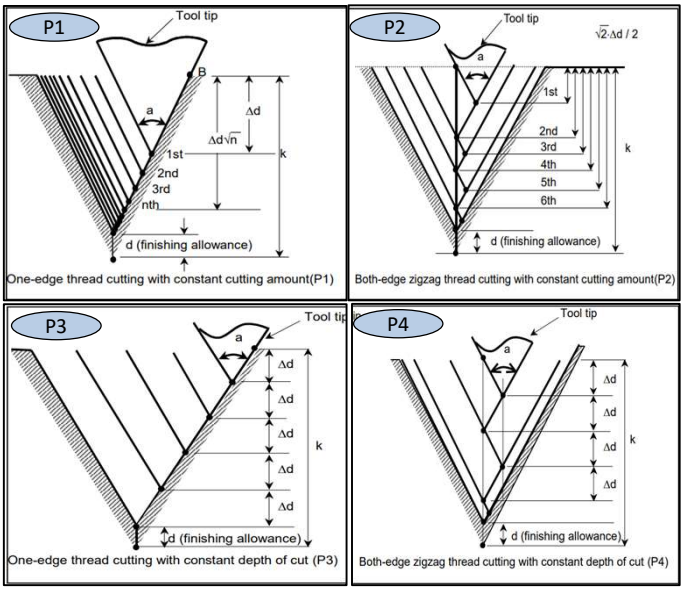
Thread Height (P) = Pitch X 0.613 = 1.5 X 0.613 = 0.9195mm
Minor Dia. = Major Dia. – 2 x Thread Height (P) = 50 – 2x0.9195 = 48.16

➢ G76.1 (Multi Threading Cycle, Both-edge zigzag threading):-

- **Cycle Format –**

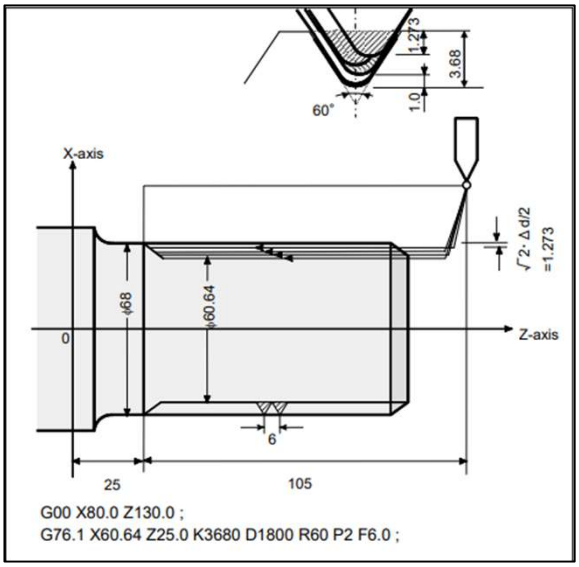
G76.1 X_ Z_ I_ K_ D_ R_ F_ P_ Q_

X_ = Minor Diameter
Z_ = Thread length
I_ = Taper amount
K_ = Thread height
D_ = Depth of 1st cut in Micron
R_ = Angle of tool nose
F_ = Lead of Thread
P_ = Thread Cutting Method (1/2/3/4)
Q_ = Thread start angle shift



- **Drawing with Sample Programs :-**

```
O0008
G0 G28 W0.0 U0.0
T0101
M03 S500
G0 X80.0 Z130.0
G76.1 X60.64 Z25.0 K3.680 D1800 R60 P2 F6.0
G0 X80.0
G0 Z150.0
G0 G28 W0.0 U0.0
M05
M30
%
```



- **Calculation for Minor Diameter :-**

Thread Height (K) = Lead X 0.613 = 6.0 X 0.613 = 3.680mm
Minor Dia. = Major Dia. – 2 x Thread Height (K) = 68 – 2x3.68 = 60.64

- **Parameters for G76.1 cycle :- 5130=1.0 (chamfer amount) , 5141 = 0.05 (Finishing Allowance)**

➢ G74 (High speed Peck Drilling) and G83 (Peck Drilling):-

- **Cycle Format –**

G74 Z_ Q_ F_

Z_ = Total Depth of Drill
Q_ = Pecking amount in micron
F_ = Feed rate (mm/rev)

- **Cycle Format –**

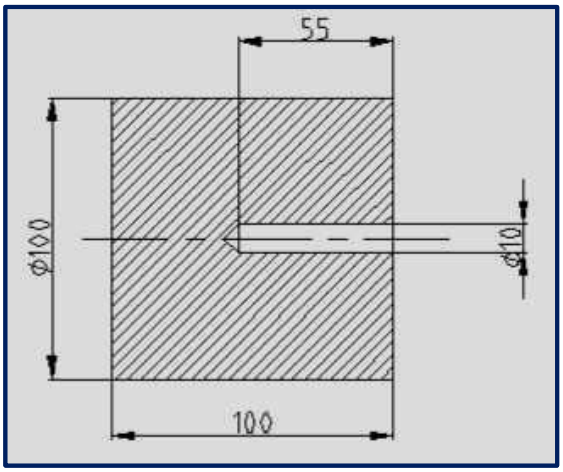
G83 Z_ Q_ F_

Z_ = Total Depth of Drill
Q_ = Pecking amount in micron
F_ = Feed rate (mm/rev)

- **Drawing with Sample Programs :-**

```
O0010
G28 W0.0 U0.0
T0101
G97 M03 S500
G0 X100.0 Z5.0
G0 X0.0
G74 Z-55.0 Q5000 F0.2
G0 Z10.0
G28 W0.0 U0.0
M05
M30
%
```

```
O0011
G28 W0.0 U0.0
T0101
G97 M03 S500
G0 X100.0 Z5.0
G0 X0.0
G83 Z-55.0 Q5000 F0.2
G0 Z10.0
G28 W0.0 U0.0
M05
M30
%
```



- **Parameters for G83 cycle :- 5101#2=1 , 5115 = 0.5**

➢ G84 (Tapping Cycle) :-

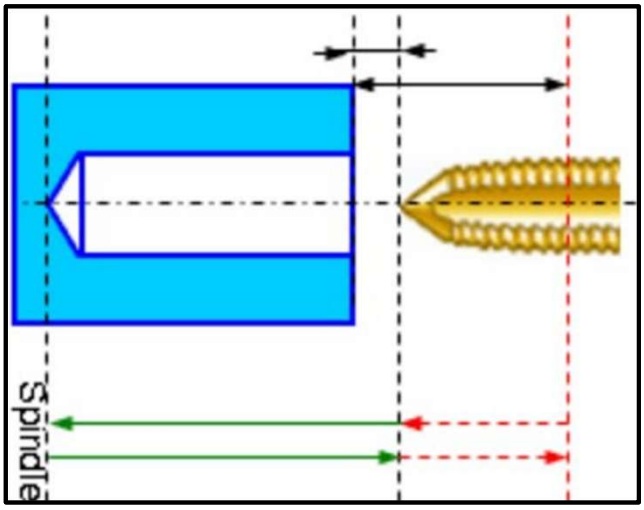
- **Cycle Format –**

G84 Z_ F_

Z_ = Thread Length
F_ = Pitch

- **Drawing with Sample Programs :-**

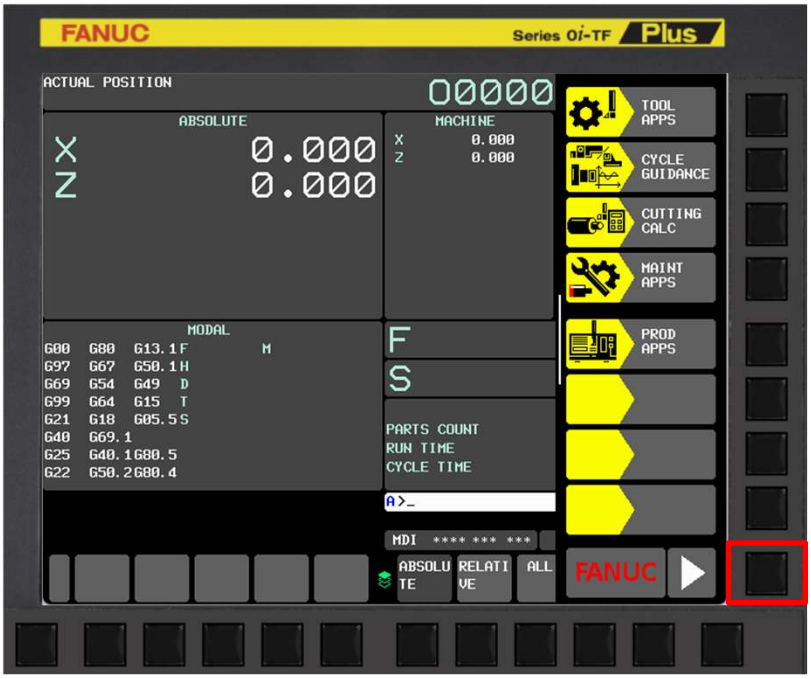
```
O0012
G28 W0.0 U0.0
T0101
G97 M03 S200
G0 X100.0 Z5.0
G0 X0.0
M29 S200
G84 Z-20.0 F1.5
M28
G0 Z10.0
G28 W0.0 U0.0
M05
M30
%
```



FANUC Series 0*i*-TF Plus

Turning Suite 2.0

1.6 Turning Suite 2.0 Main Menu :-



(Quick Access Menu)

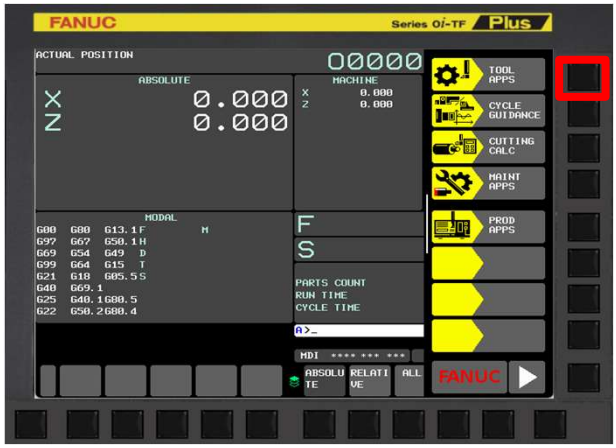
➢ Functions available in Turning Suite 2.0 :

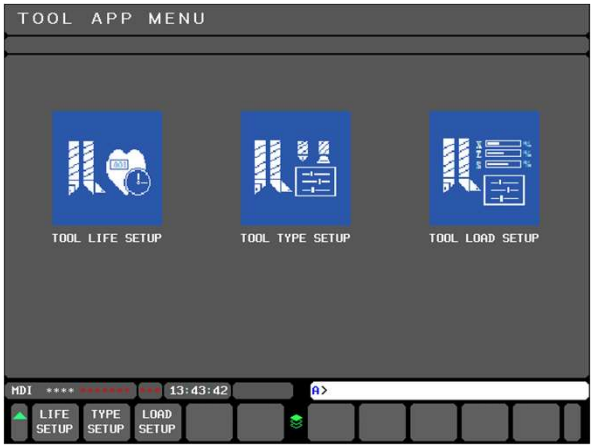
S. No.	Category	Function Name
1	Tool Apps	Tool life setup
2		Tool type setup
3		Tool load setup
4	Cycle Guidance	Threading Guidance
5		SSV
6		Part cut off
7		G Code Guidance
8	Cutting Calculator	Cutting Calculator
9	Maintenance Apps	Motor status display
10		Periodic maintenance
11		Operation level selection
12	Production Apps	Machine utilization
13		Cycle time split up
14		Power Consumption Dashboard

⚠ The Quick Access Menu can be opened only from NC screens, providing convenient access to common functions without leaving the current NC environment.

1.61 Turning Suite 2.0 Categories :-

❖ Tool Apps :





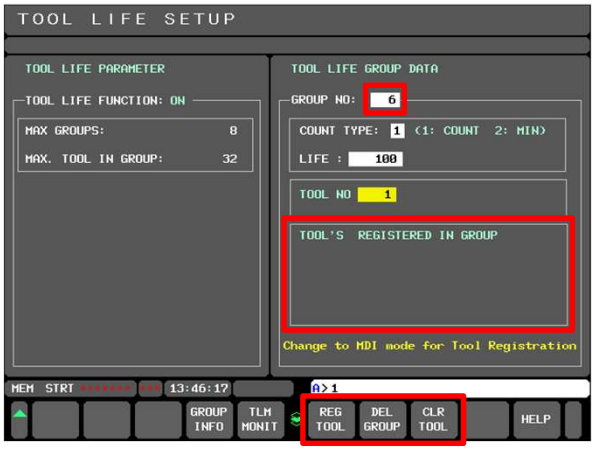
➢ Procedure to setup Tool Life :

1. Set CNC to “MDI” mode to perform Tool Life setup operations such as
   1. Register Single Tool.
   2. Delete Single Tool.
   3. Delete Group.

2. To Register a tool, enter “**Group No.**”, “**Count Type**”, “**Life**” and “**Tool No.**”. Press “**Reg Tool**” softkey.

3. Registered tools in a group is displayed on screen.

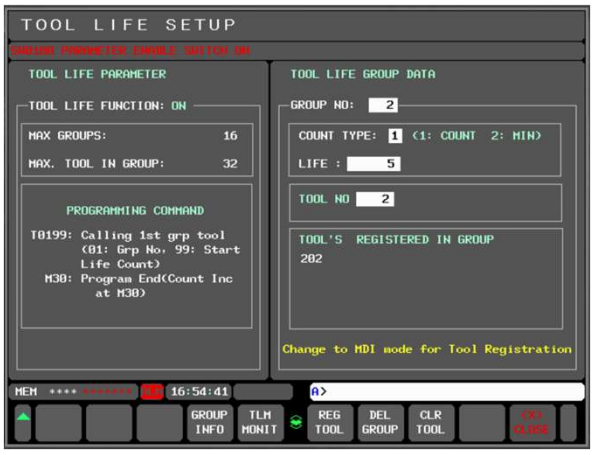
4. To delete tool, enter “**Group No.**” and “**Tool No.**”. Press “**CLR Tool**” softkey. Updated tools are displayed on screen.



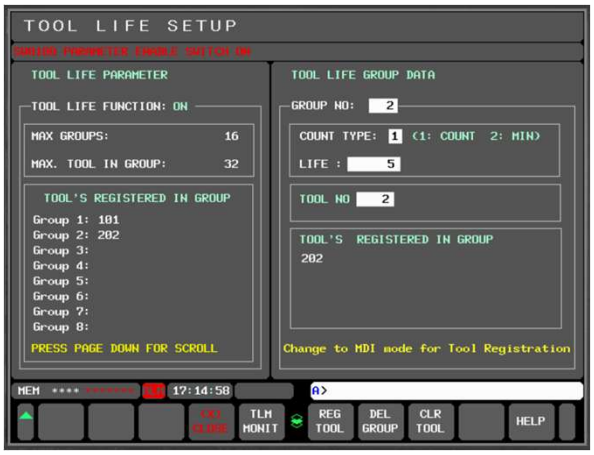
5. To delete a group, enter “**Group No.**” and press “**Del Group**” softkey.

⚠ Enable Tool life settings on CNC before using Tool Apps.

6. Select “**Help**” softkey for guidance related to programming command related to tool life counting.
7. Press “**Close (X)**” softkey to close information.

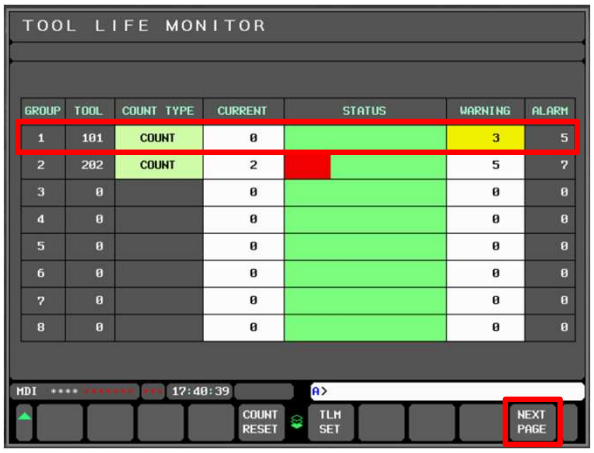


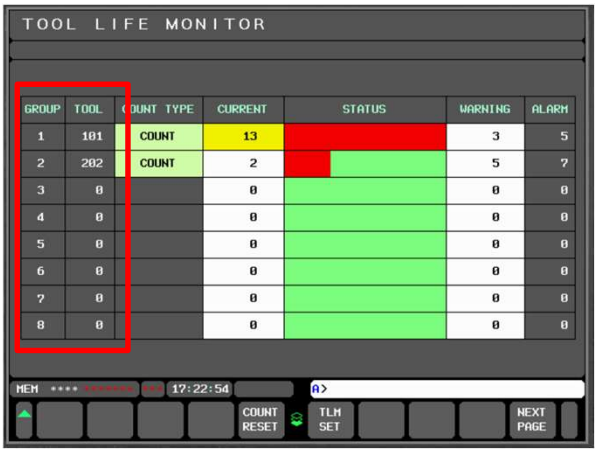
8. Select “**Group Info**” softkey to get information related to registered tools in all the groups.
9. Information is displayed at bottom left portion of screen.
10. Press “**Close (X)**” softkey to close information.



➢ Procedure to visualize Tool Life :

1. Select “**TLM MONIT**” softkey to display Tool Life Monitor screen.
2. To register “**Warning**” message for individual tool, move cursor to “**Warning**” column and input appropriate warning value less than Alarm Value displayed on screen.
3. To reset count for a tool, move cursor to “**Current**” column and press “**Count Reset**” softkey.
4. Tool life data can be visualized in bar graph format.
5. Total up to 16 tools data can be visualized on this screen.
6. Also, this screen displays active sister tool against each group no.

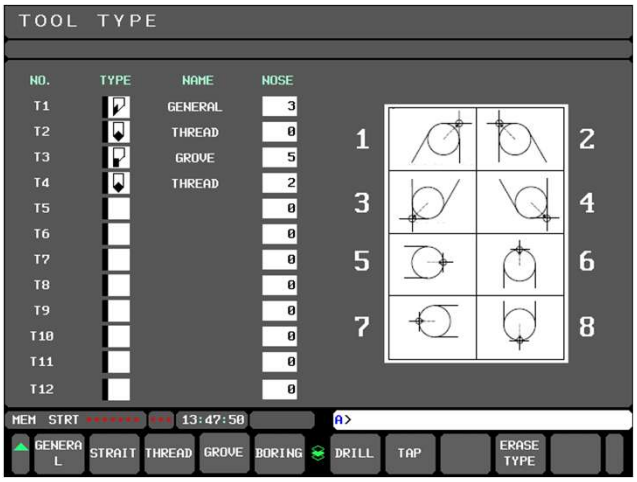




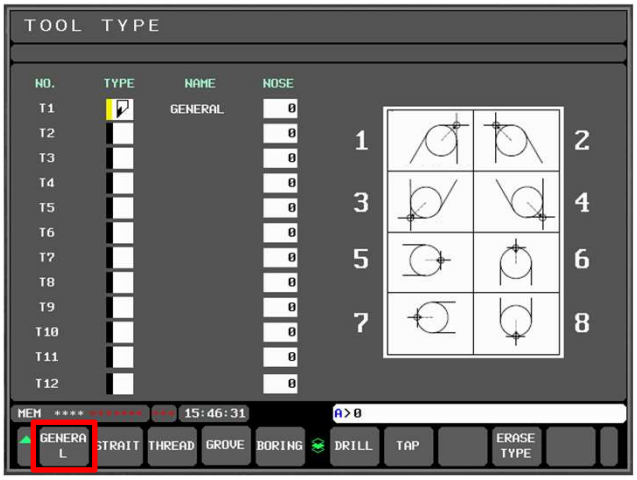
⚠ Enable Tool life settings on CNC before using Tool Apps.

➢ Procedure to setup Tool Type :

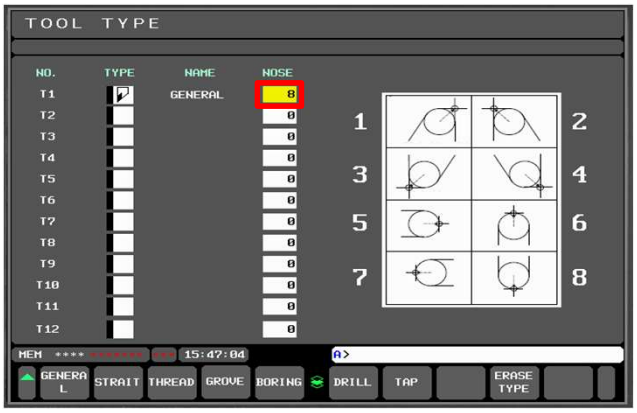
1. Open “**Tool Type**” from “**Tool Apps**” menu.
2. Move cursor to desired tool number using arrow keys on MDI.



3. Select tool type from the bottom soft keys. Ex : General, Straight, thread etc.
4. Corresponding tool image and name is displayed on screen.

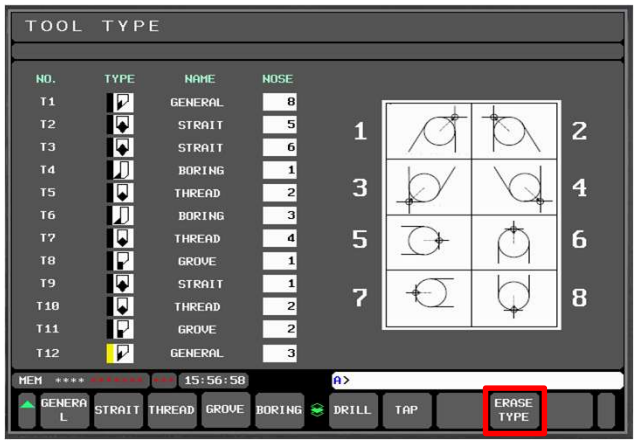


5. Move cursor to “**Nose**” column and enter nose type by referring to “**Tool Nose**” image displayed on screen.



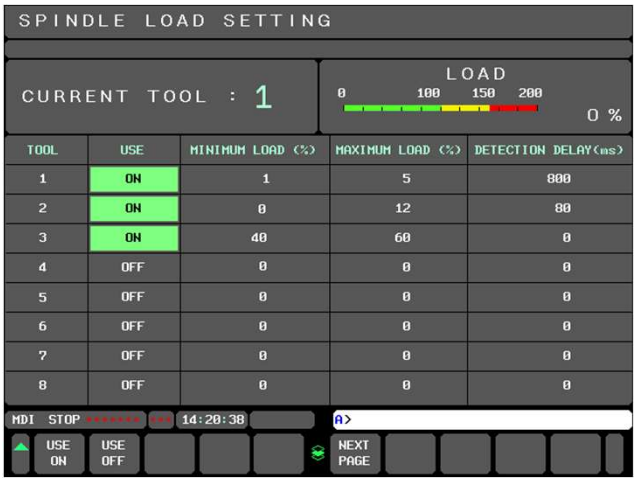
➢ Procedure to erase / clear tool type :

1. Move cursor to desired tool number.
2. Press “**Erase Type**” softkey to clear tool type.

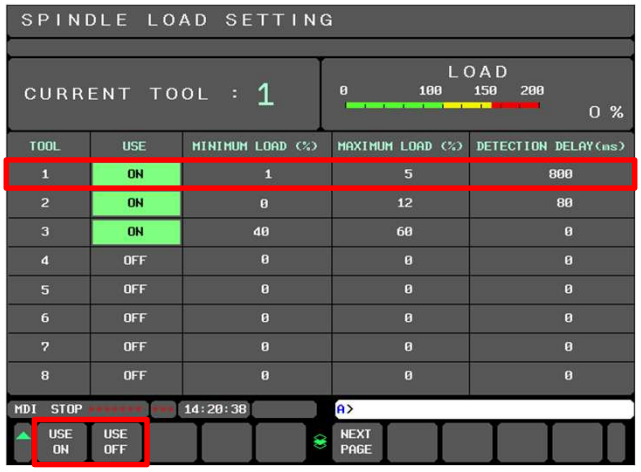


➢ Procedure to setup Tool Load :

1. Open “**Tool Load**” from “**Tool Apps**” menu.
2. Move cursor to desired tool number using arrow keys on MDI.

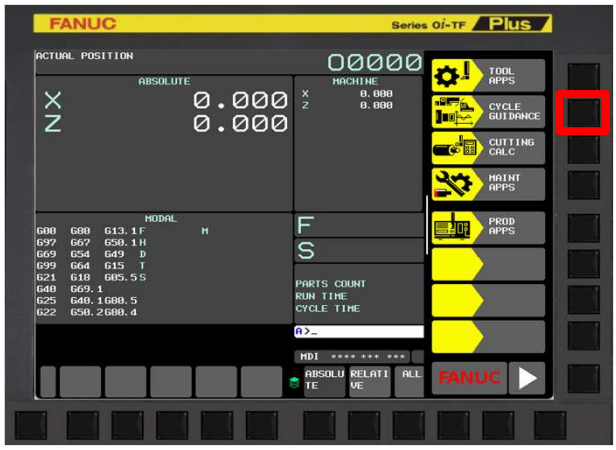


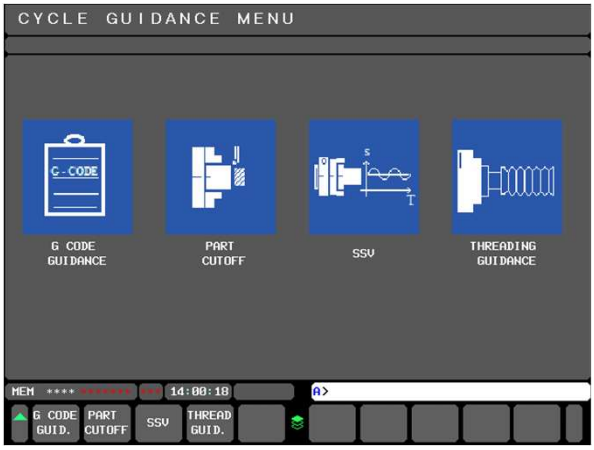
3. Press “Use On” / “Use Off” softkey to enable / disable load monitoring.
4. Enter the Minimum Load / Maximum in percentage load for the respective tool.
5. Enter the detection delay value in milliseconds.



1. Active tool and spindle load is displayed on top screen for load monitoring setup.
2. Detection delay is set and monitored in multiple of 8 (millisecond) based on PMC timer accuracy.

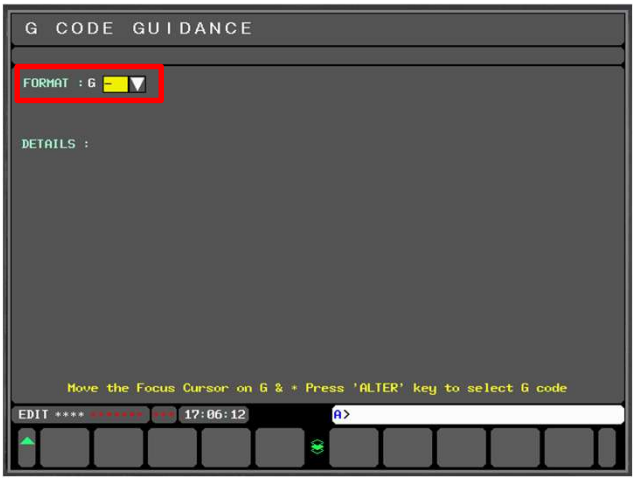
❖ Cycle Guidance Apps :



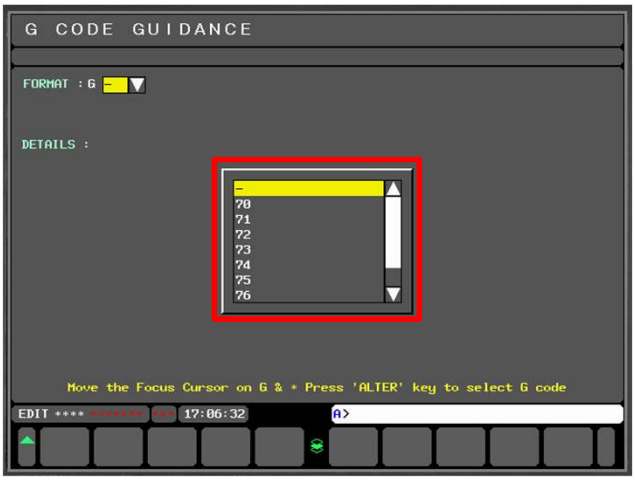


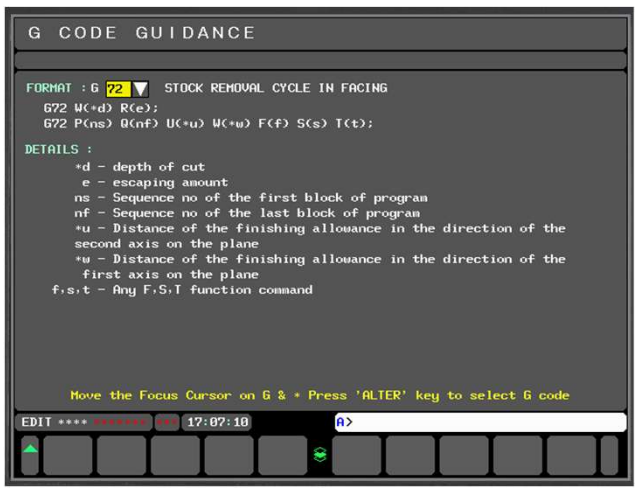
➢ Procedure to use G Code Guidance :

1. Open “**G Code Guidance**” from “**Cycle Guidance Apps**” menu.
2. Move cursor to checkbox using arrow keys on MDI.



3. Press “**ALTER**” button on MDI to open G Code list.
4. Select the desired G Code and press “**INPUT**” button on MDI.



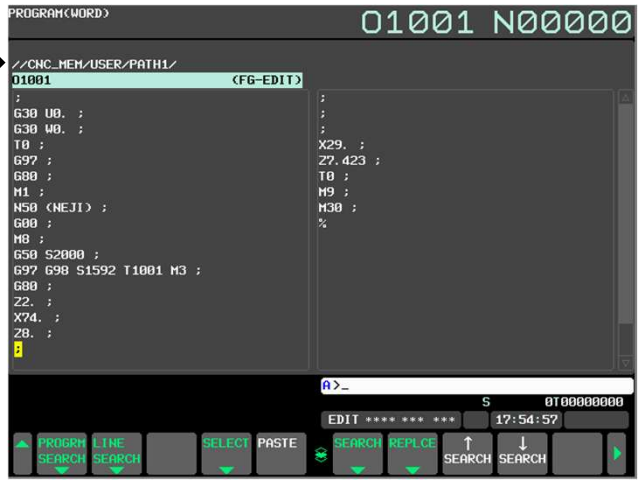


5. Guidance related to specified G Code is displayed on screen.

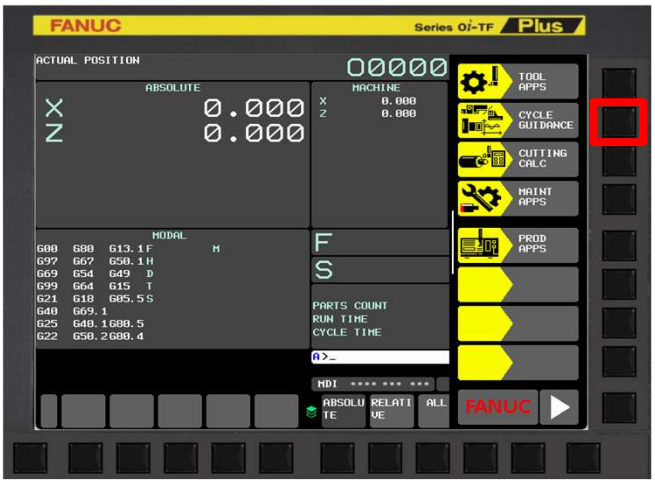
➢ Procedure to use Part Cut Off Cycle :

1. Set CNC to “**EDIT**” mode.

2. Open program in “**FG-EDIT**” mode and move the cursor to line no. where cycle is to be inserted.

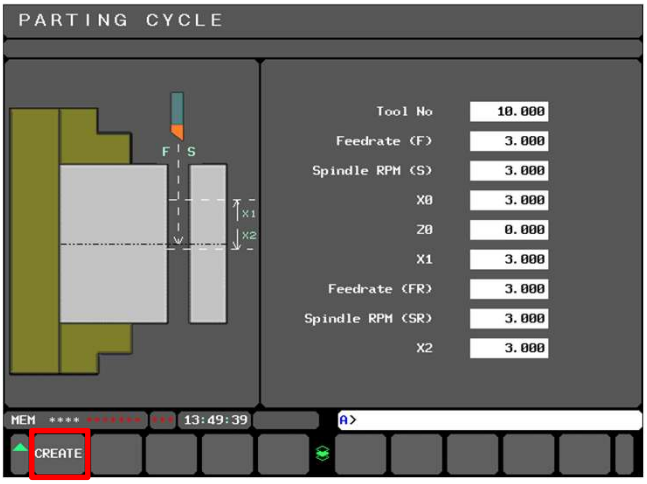


3. Open “**Parting Cycle App**” from “**Cycle Guidance Menu**” using **Quick Access Menu**.



4. Enter the data on screen required to create parting cycle.

5. Press “**Create**” softkey to insert cycle inside main program .

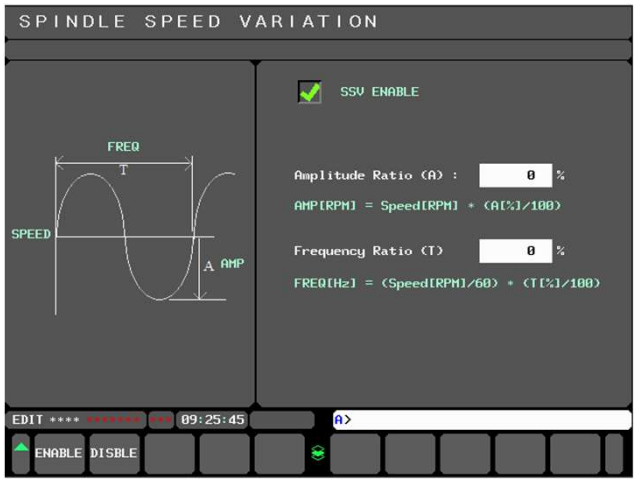


5. Cycle is inserted in Main Program.

> ⚠ Cycle cannot be inserted in background programs or BG-EDIT mode

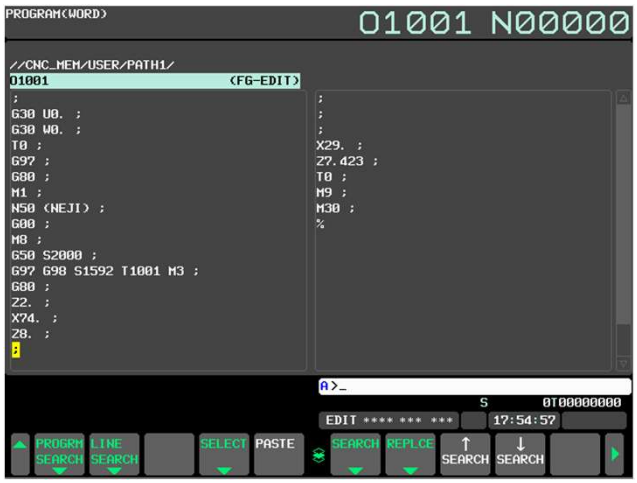
➢ Procedure to use Spindle Speed Variation :

1. Select “**Spindle Speed Variation**” screen from “**Cycle Guidance Menu**”.
2. Move cursor to “**SSV Enable**” checkbox to enable / disable the spindle speed variation feature.
3. Enter **Amplitude Ratio** and **Frequency ratio**.

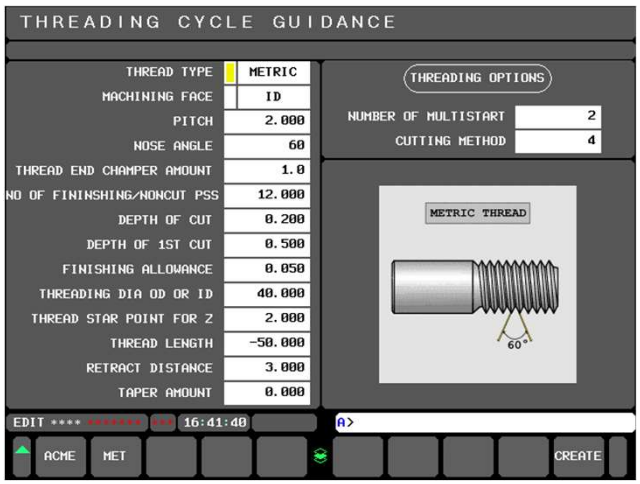


➢ Procedure to use Threading Guidance :

1. Set CNC to “**EDIT**” mode.
2. Open program in “**FG-EDIT**” mode and move the cursor to line no. where cycle is to be inserted.
3. Open “**Threading Cycle Guidance**” from “**Cycle Guidance Menu**”.

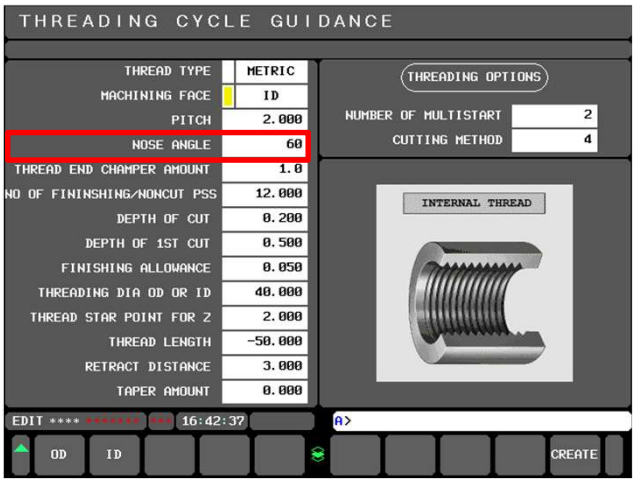


4. Select the type of thread.
   1. Acme
   2. Metric
5. Select Thread Face Type.
   1. OD
   2. ID

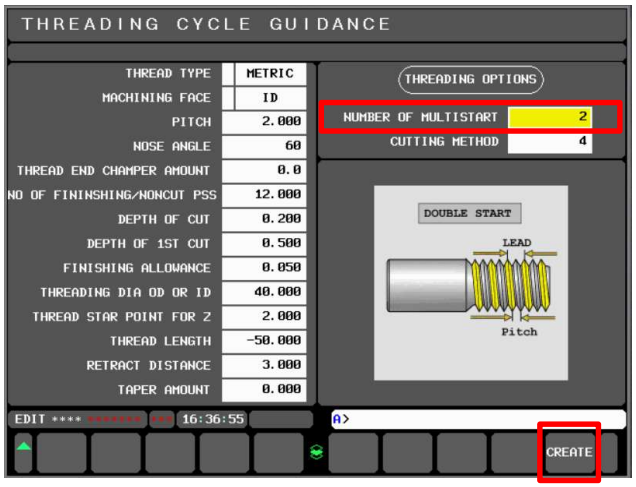


6. Nose Angle Value by default is auto assigned as **60° for Metric** and **29° for Acme**.
7.
8. Adjust the Nose angle if required.

9. Similarly, input all the required values. **All fields are mandatory to enter.**



6. Select “Multi Start” option for multi start threading.
7. Select Value as 1 to 4.
8. By default, when no multi start is required, default value is 1.

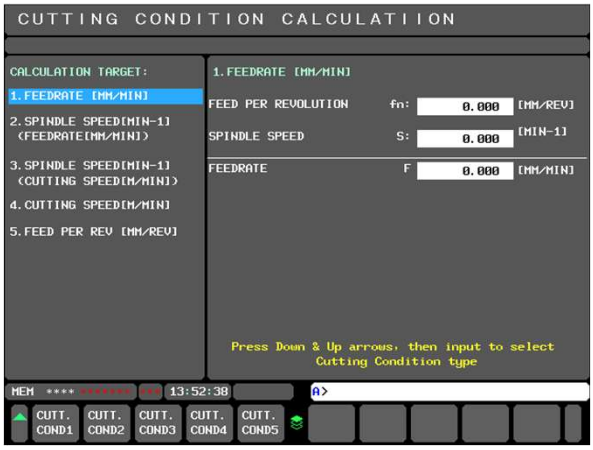


9. Select the cutting method.
10. Select value from 1 (P1) to 4 (P4).

Cutting Method	
P1	One-edge cutting, constant cutting amount
P2	Two-edge cutting, constant cutting amount
P3	One-edge cutting, constant cutting depth
P4	Two-edge cutting, constant cutting depth

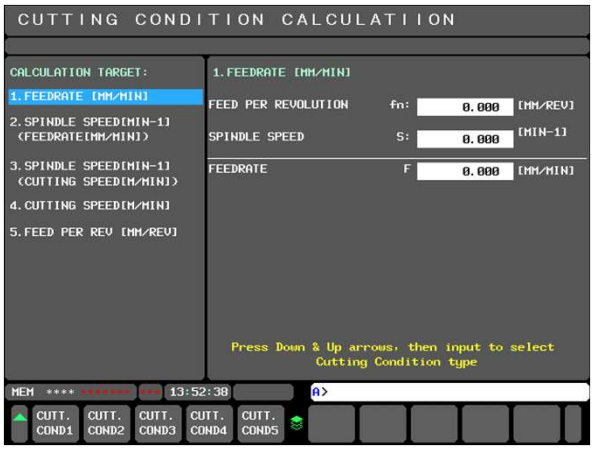
11. Press “Create” button to insert cycle inside Main Program.

❖ Cutting Condition Calculator :



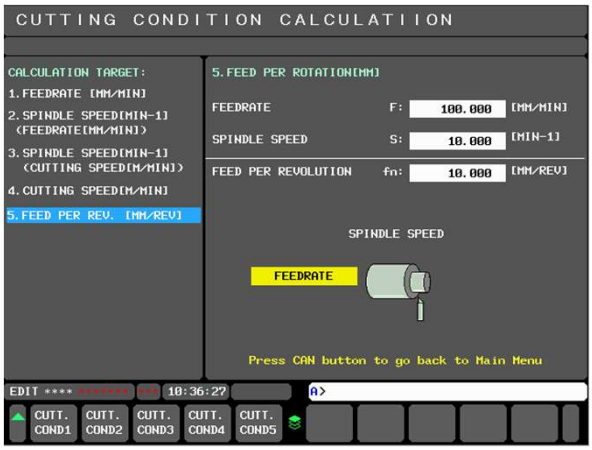
➢ Procedure to use Cutting Condition Calculator :

1. There are 5 types of cutting condition available as below:
   1. **Feedrate [mm/min]**
   2. **Spindle Speed [min-1] for feedrate**
   3. **Spindle Speed [min-1] for cutting speed**
   4. **Cutting Speed [m/min]**
   5. **Feed Per Rotation [mm]**



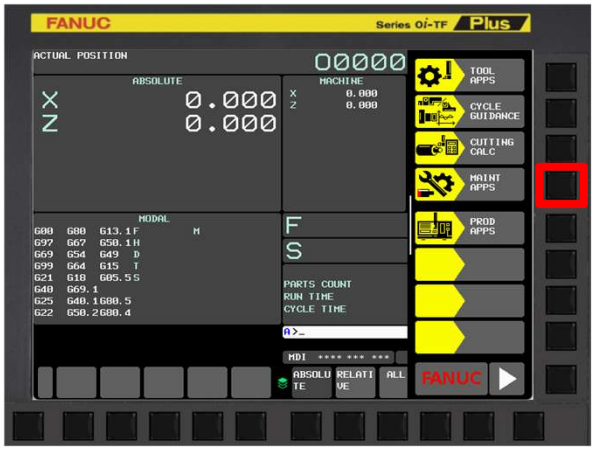
2. Select the type of cutting condition to be calculated from bottom softkey.
   1. **Cutt. Condi. 1**
   2. **Cutt. Condi. 2**
   3. **Cutt. Condi. 3**
   4. **Cutt. Condi. 4**
   5. **Cutt. Condi. 5**

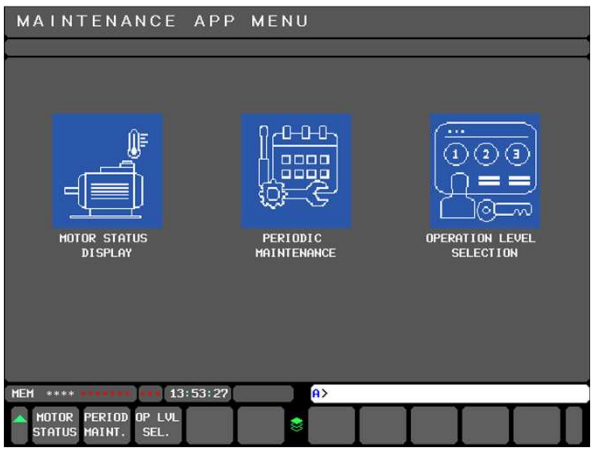
3. Enter the required data displayed on screen.
4. Calculation is done directly, based on the user input.



5. Press “**CAN**” button on MDI to exit calculation.

❖ Maintenance Apps :



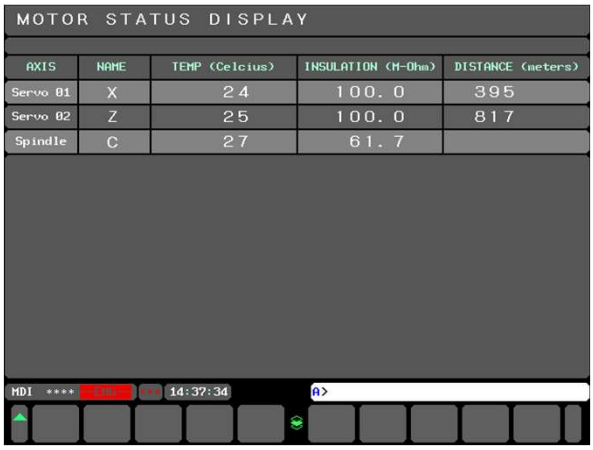


➢ Procedure to use Motor Status Display :

1. Application display maintenance data related to motor such as temperature, Insulation and Distance travelled by motor.

2. If machine is already in “Emergency” mode. Release and put the machine back to “Emergency” mode.

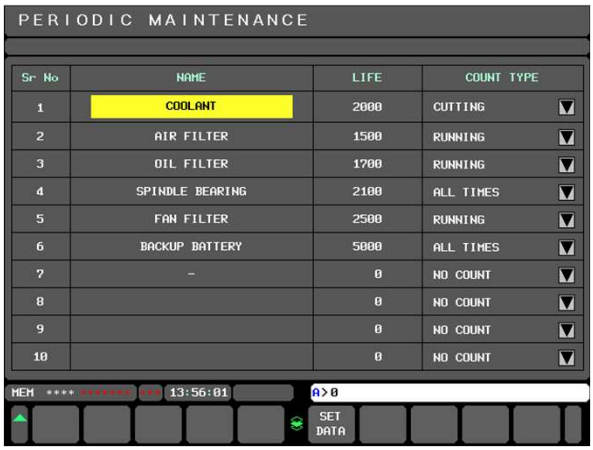
3. This updates Insulation resistance and distance travelled information.



> ⚠ CNC Parameters related to Insulation resistance and distance travelled should be enabled before using this screen.

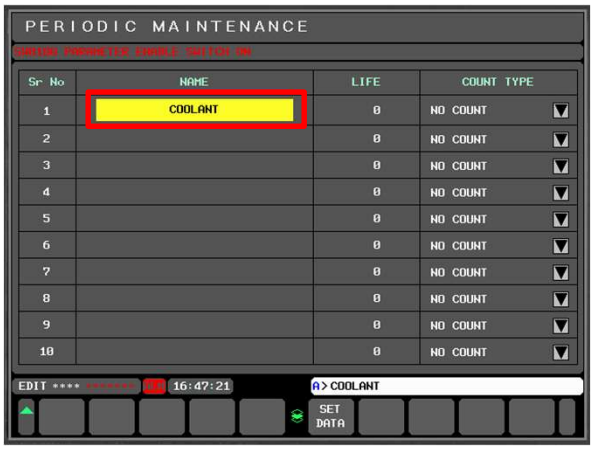
➢ Procedure to use Periodic Maintenance :

1. This application provides a user-friendly interface for setting up, registering, and automatically managing periodic maintenance tasks on machines, including continuous life tracking to ensure timely maintenance reminders and minimize machine downtime.

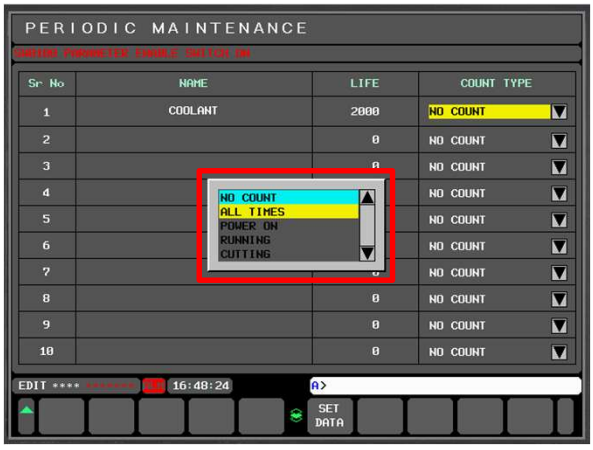


2. Set CNC to “**MDI**” mode.

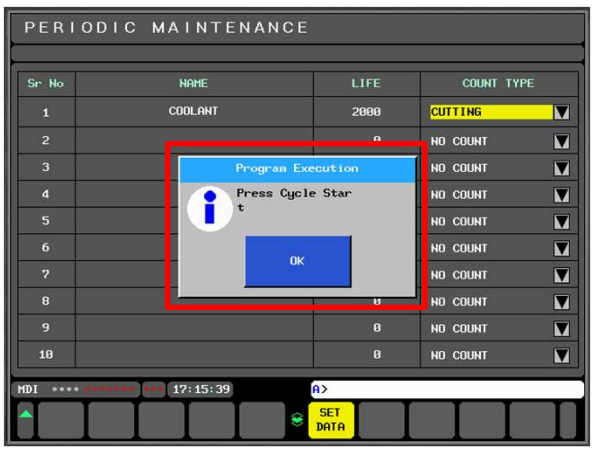
3. Move cursor to “**Name**” column to register the machine component to be monitored.
4. Move cursor to “**Life**” to setup life for registered component.



5. Move cursor to “**Count Type**”. Press “**ALTER**” on MDI to select count type.
    1. No Count
    2. All Time
    3. Power On
    4. Running
    5. Cutting



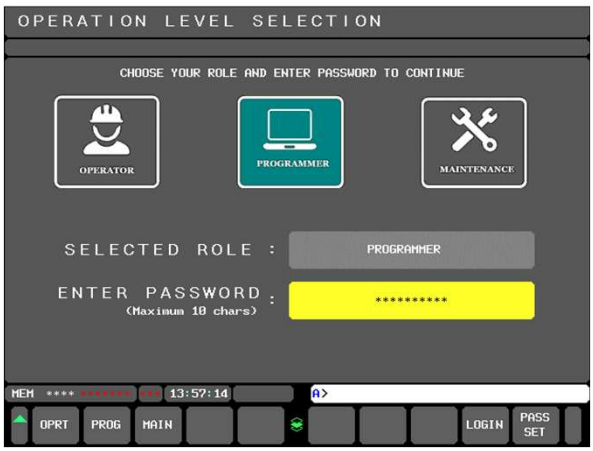
6. Press “**Set Data**” softkey. Press Cycle Start to register component to CNC.



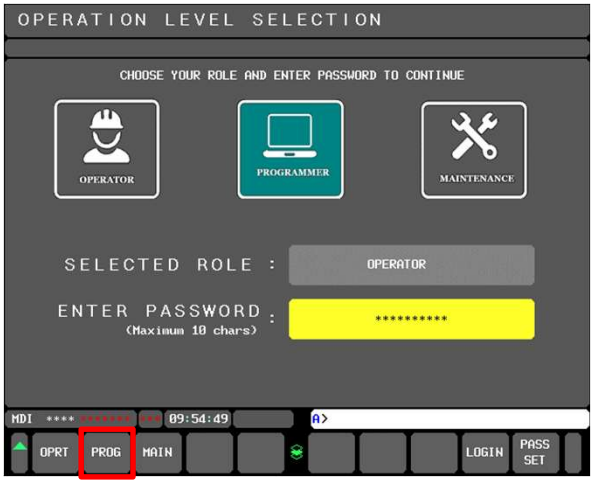
➢ Procedure to use Operation Level Selection :

1. This application implements a secure role-based access control system for CNC machines, requiring users to select a role and enter a password.

Role	Access / Operations
Operator	Wear offset modifications
Programmer	Program editing, offset setting
Maintenance	Full Access

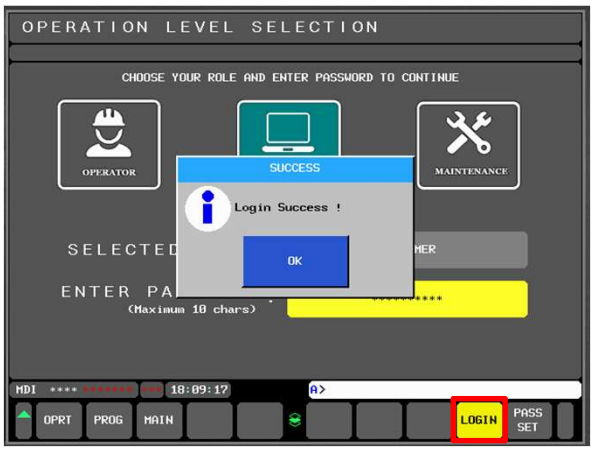


2. Press “**Prog**” (Programmer) or “**Main**” (Maintenance) softkey to selected role.



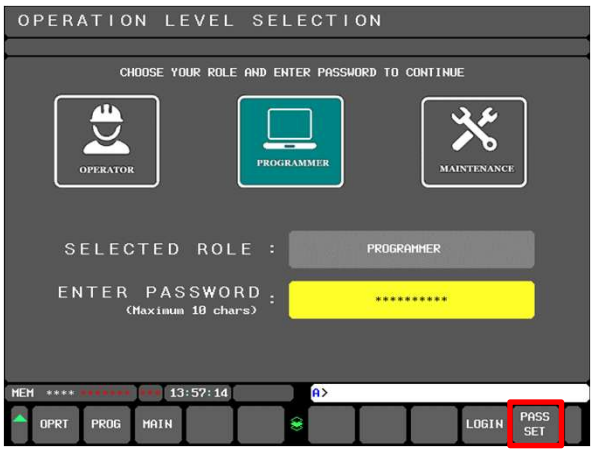
3. Move cursor to “**Password**” input and enter password. Press “**Login**” softkey to login to selected role.

4. To return to Operator mode, press “**Oprt**” and select “**Login**” softkey



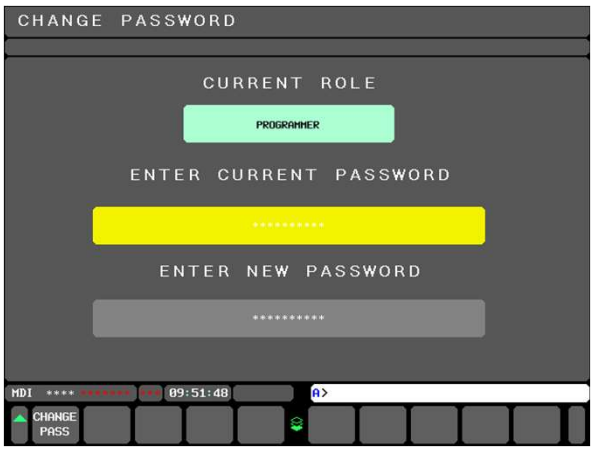
➢ Procedure to change Programmer / Maintenance Password :

1. To change the password for Programmer / Maintenance level, select the role as “Programmer” or “Maintenance”.
2. Login to selected role with current password.



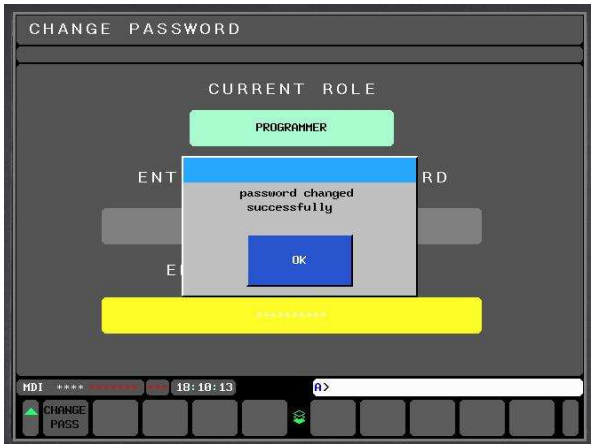
3. Once, login is successful, press “**Pass Set**” softkey.

4. Password change screen is displayed.



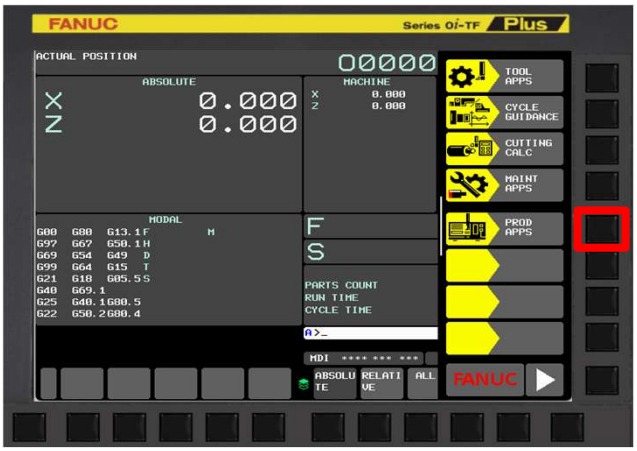
5. Enter the current password and new password to be set .

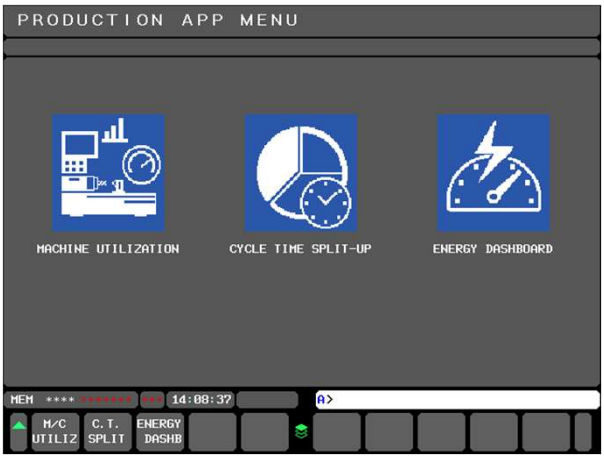
6. Press “Change Pass” softkey to change password.



7. To return to Operator mode, press “**OPRT**” and select “**Login**” softkey

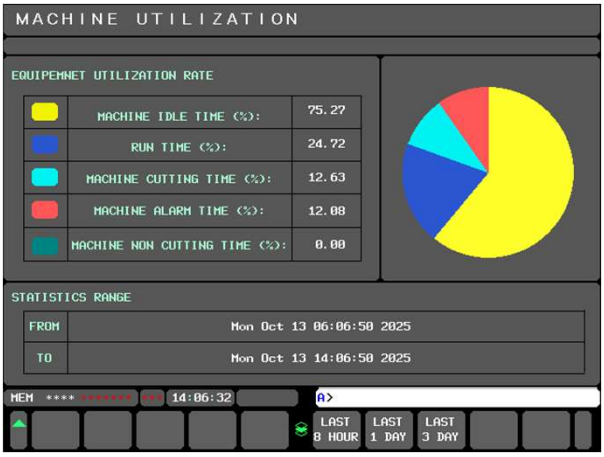
❖ Production Apps :





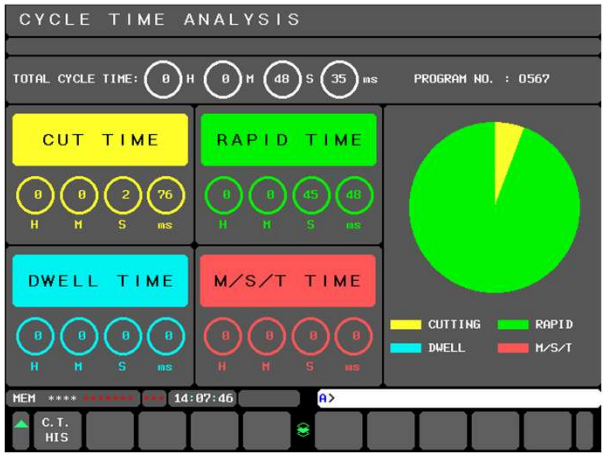
➢ Procedure to use Machine Utilization :

1. Select “Machine Utilization” screen from “Maintenance Apps” screen.
2. Machine Utilization can be visualized for the following duration :
    1. Last 8 hours.
    2. Last 1 day.
    3. Last 3 days.
3. Machine data is categorized into following :
    1. Machine Idle Time.
    2. Machine Run Time.
    3. Machine Cut Time.
    4. Machine Alarm Time.
    5. Machine Non-Cut Time.



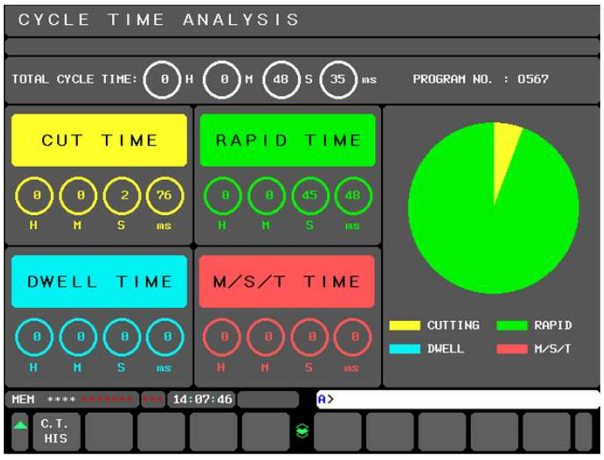
➢ Procedure to use Cycle Time Split-up :

1. This application provides real-time cycle time monitoring for CNC machines, displaying detailed breakdowns of cutting, rapid, dwell, and M/S/T times in an interactive pie chart and allows users to download this data in CSV format for further analysis.



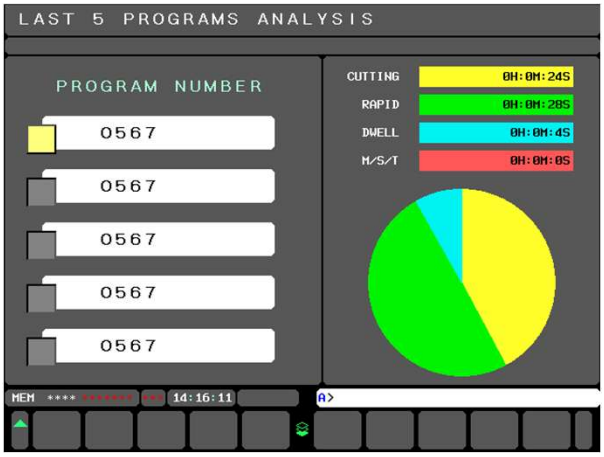
2. Cycle Time split up information is displayed in real time while the program is running on CNC.

3. Data can be visualized when program running in “**MEM**” mode.



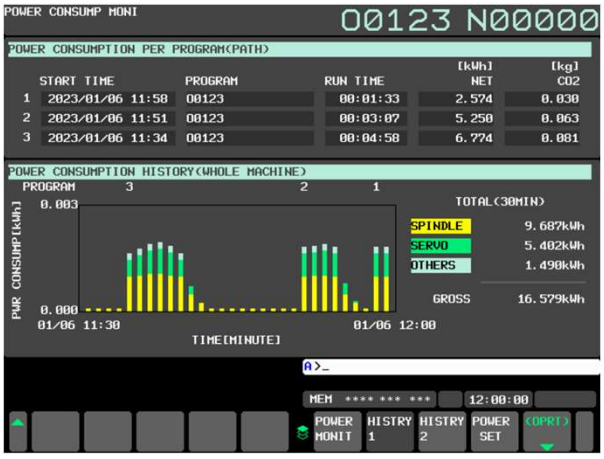
4. To visualize the historical data, press “**C.T His**” softkey.

5. Last 5 executed CNC program split up analysis can be visualized.



➢ Procedure to use Power Consumption Dashboard :

1. This application displays actual power consumption and carbon dioxide emissions for each machining program and provides a time-series graph of the machine's overall power consumption for monitoring and analysis.



> ⚠ Power Consumption should be enabled in CNC. Appropriate Coefficient parameters should be set before using this screen.

2. Revision Record

Edition	Date	Contents
01	20-Dec, 2024	First Time release
02	24-Nov, 2025	Webinar QR code added
03	28-Mar, 2026	Turning Suite 2.0 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