SAMSUNG
Machine Tools

The name you know, the technology you trust.

SAMSUNG MCV SERIES

[FANUC 0i-MF CONTROL]

OPERATION AND PROGRAM TRAINING MANUAL

Index

Section	Pages
Control Panel	4
Modes & Functions	5-6
MPG Use	7
Manual Rapid / Jog Mode	8
Spindle Speed / Feed Override	9
Zero Return	9
Spindle Control	10
ATC Magazine Control	11
Auxiliary Functions	12-13
Operation Lock / Door Interlock	14
NC Screens & Hard Keys	15
Positions Screen	16
Program Screen	17-18
Tool Offsets Screen	19
Settings Screen	20
Work Offsets Screen	21
Macro Variables Screen	22
Guard Screens	23
Tool Offsets Guard Screen	24
Work Offset Guard Screen	25
Parameters Screen	26
Alarm Screen	27
Graph Screen	28
Manual Guide I Screen	29
Negative Tool Length Setting	30
Positive Tool Length Setting	31
Work Offset Setting	32
Setting ATC Tool Numbers	33
MST Commands	34
Programming Addresses	34
M Codes	35-36
G Codes	37-38
Program / Folder Management	39-41

Incremental vs Absolute Programming	42
High Speed Machining	43
Rigid Tapping	44
Example Programs	45-46
Oversize Tooling	47
Probing	48-50
SMT Custom Software	51
ATC Recovery Mode	52
ATC Maintenance Mode	53
G-Code List	54
M-Code List	55
Alarm Detail List	56
ATC Management	57-58
Spindle Warm Up	59
Inputting & Outputting Programs	60-63
DNC Use & Drip Feeding	63-64
Ethernet Settings	64
Outputting Parameters & PWE	65

Control Panel

Powering ON

1. Turn on the Main Breaker at the rear of the machine
2. On the main control panel, release the large RED Emergency Stop Button
3. Press the GREEN Power ON Button. Wait for the NC to boot completely
4. Press the GREEN M/C Ready Button and wait until the green light stops flashing

Modes & Functions

- **Memory:** for running in full automatic mode.
- **MDI**: for executing small lines of code at a time, for example S500 M3; to turn on the spindle.
- **Remote:** for drip feeding with RS232.
- **Edit:** for creating and editing programs.
- **Handle:** for using the mpg and axis selection knob for moving the machines axis.
- **Jog**: for using the axis direction buttons to manually feed an axis.
- **Rapid:** for manually moving each axis in rapid mode.
- **ZRN**: zero return function. For zero returning a single axis.

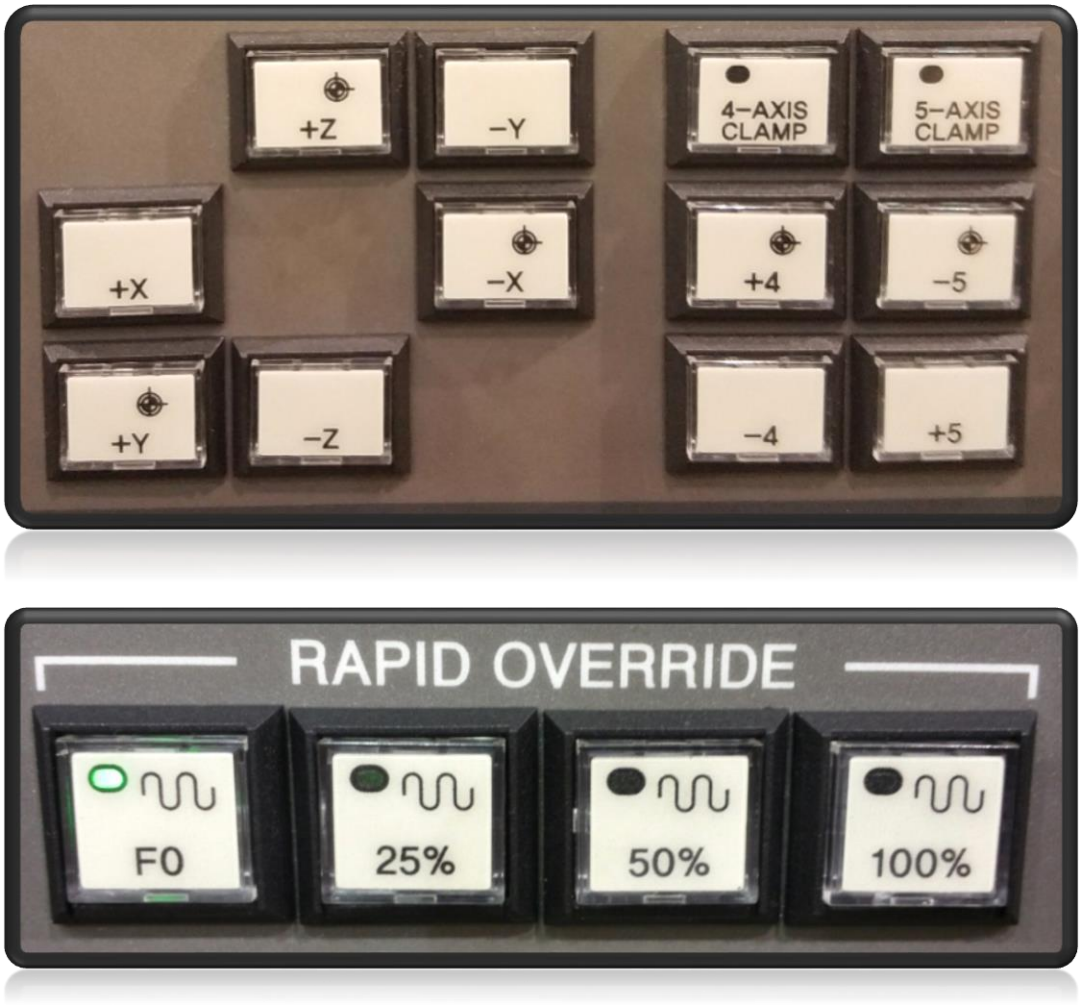
- **Single Block:** with pressing cycle start executes one line of program code at a time.
- **Optional Stop:** works with m-code **M01** in program to execute a program stop at this code.
- **Block Delete:** / in program to skip any command on same line, after /
- **Dry Run:** will multiply feed rate in program, used in conjunction with the feed override dial to select rate. Used for dry cycling, proving out program, or returning to a previously left point in program quickly.
- **Auto Power Off:** turns the machine off at M30 command. (option)
- **Z Axis Cancel:** allows running a program with the z axis locked from moving.
- **Machine Lock:** this will stop all axis of the machine from moving.
- **MST Lock:** locks out using or changing any M, S, OR T command.

MPG Use

Manually Moving the Machine Via Handle Mode:

- To manually move the machine, select **HANDLE** mode, select the **X1**, **X10**, OR **X100** increment and an **axis** selection such as **X, Y OR Z**. Every 'click' of the handle rotary dial will give movement of, **X1 = .0001", X10 = .001", AND X100 = .01"**

Manual Rapid / Jog



Manually Moving the Axis via Rapid Mode:

- To manually move an Axis, select **Rapid** mode, select a rapid increment **F0**, **25%, 50% or 100%** and press the **Axis** direction you wish to move the machine.
- ****TECH TIP**** **F0** comes set to a slow feed rate. Parameter 1401.4=0 will make F0 a Feed Rate of 0. Parameter 1421 is setting value for F0 rate if 1401.4=1.

Spindle / Feed Override

Manually Moving the Axis via Jog Mode

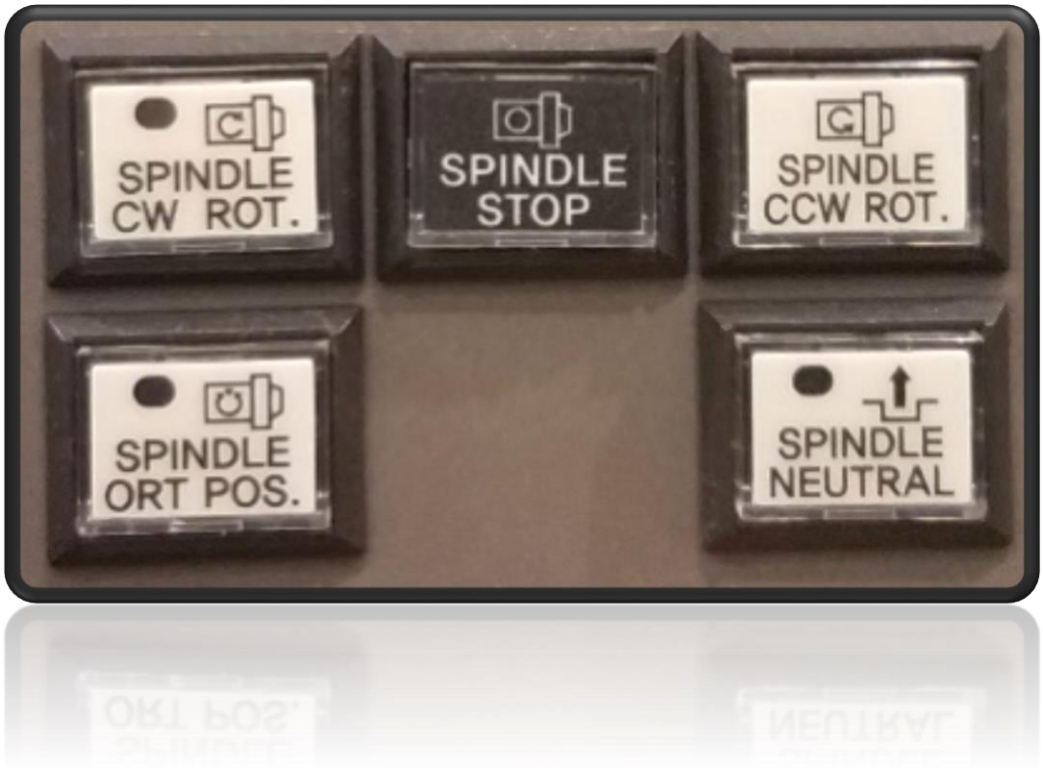
- To manually move an Axis, select **jog** mode, select a jog rate via the **feed override** knob, then press and hold the **axis** direction you wish to move the turret. During jog mode the feed override knob will adjust the rate of axis movement from 0 inches per minute, to a max of 200 inches per minute.

Zero Return

Zero Returning the Machine

- Press the **ZRN** button, then press the +Z button, -X button, and +Y button. This will zero return all axis.

Spindle Control



Turning on the Spindle Manually

- Select a manual machine mode. Type a speed command via **MDI** mode (example S500 ;) press **CYCLE START**. Then press the **CW** or **CCW** button to start the spindle. If restarting from a memory operation, select a manual mode, press **CW** or **CCW**.

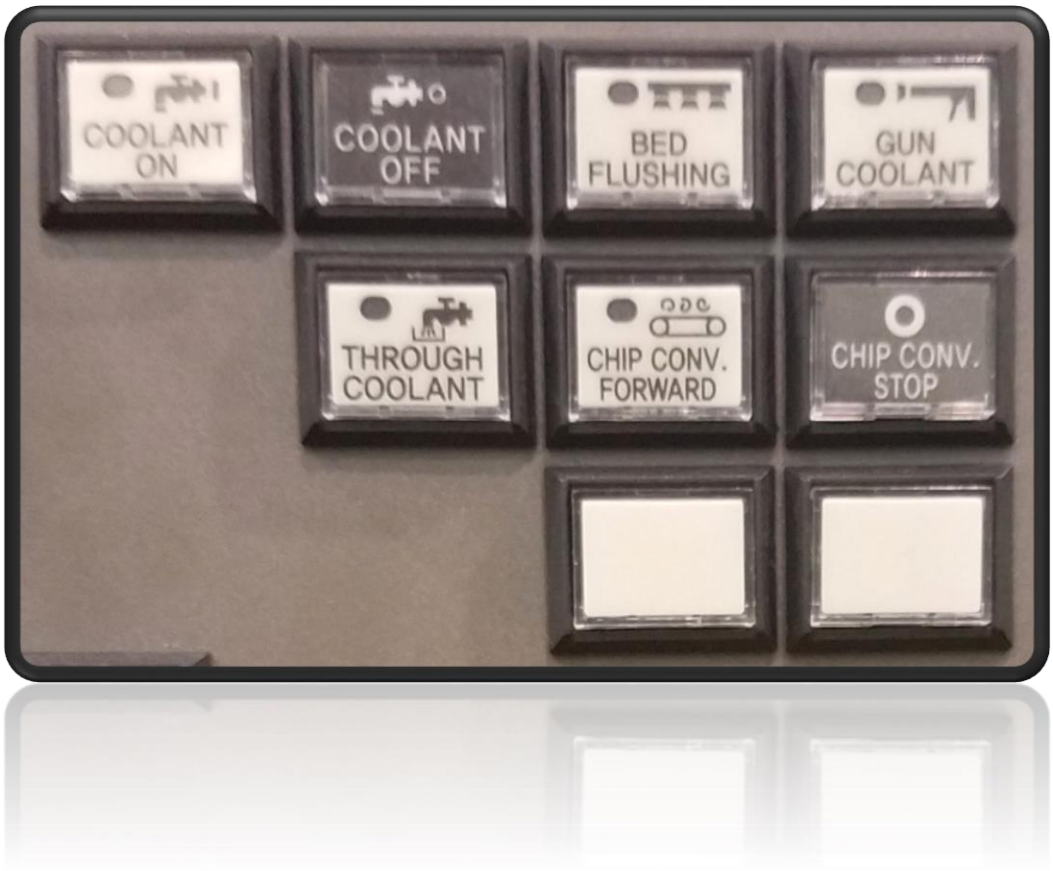
****Tech tip**** Keep Relay 9.7 = 1 will make the spindle stop when reset is pressed.

ATC Magazine Control

Using the ATC Magazine Control

- **SP. Tool NO** designates the tool currently in the spindle. **MG. TOOL NO** designates the tool in the waiting pot ready to go into the spindle. To rotate the ATC magazine press up or down to the desired tool selection in the **MG. TOOL NO** field, then press the center Rotate Button. The magazine will index to this tool. To call the new tool into the spindle press the ATC Cycle Button on the control panel.

Auxiliary Functions



Auxiliary Functions use, description, and information

- **Coolant ON**: Turns Coolant **ON**. *Machine doors must be closed and locked for this to function.
- **Coolant Off**: Turns Coolant **OFF**. *If Coolant **OFF** is pressed and held for 3 seconds until its **RED** light flashes, subsequent Coolant **ON** (M08) commands will be ignored. Pressing **OFF** again will cancel this feature.
- **Through Coolant**: Turns on the Through Spindle Coolant. Pressing a second time will turn off.
- **Bed Flushing**: Turns on the Bed Flushing for machines equipped with this option.
- **Gun Coolant**: Turns on the Coolant Gun for machines equipped with this option.

- **Chip Conv Forward**: **IF** the chip conveyor is set to **AUTO** mode, this will turn on the chip conveyor. If the chip conveyor is set to manual mode, it must be controlled from the chip conveyor.
- **Chip Conv Stop**: **IF** the chip conveyor is set to **AUTO** mode, the chip conveyor will stop. If the chip conveyor is set to manual mode, it must be controlled from the chip conveyor.

Operation Lock / Door Interlock

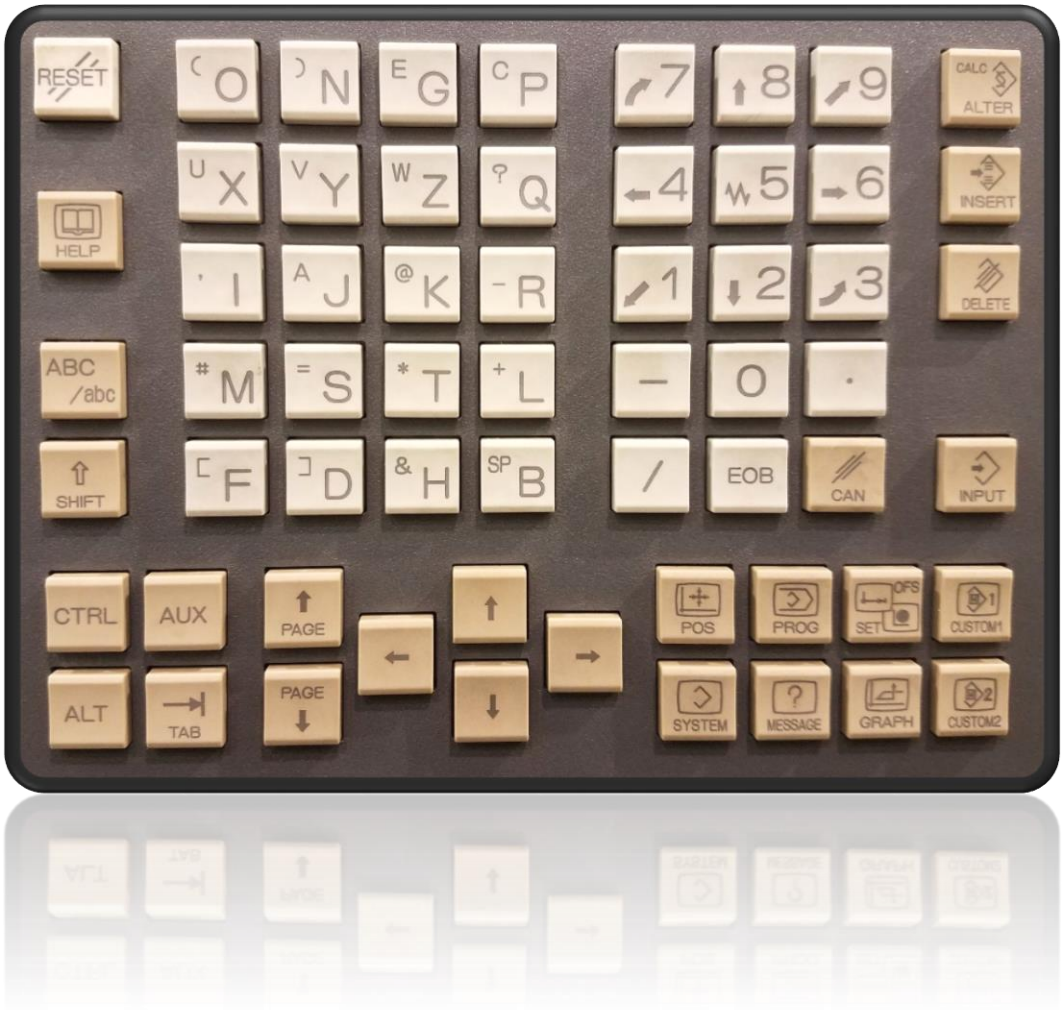
OPERATION LOCK

- **Edit** allows full use of the control, importing programs, editing programs etc. **Unlock** will allow same with the exception of locking out editing/importing programs. **Lock** completely locks out making any edits, importing programs, changing single block, optional stop, etc.

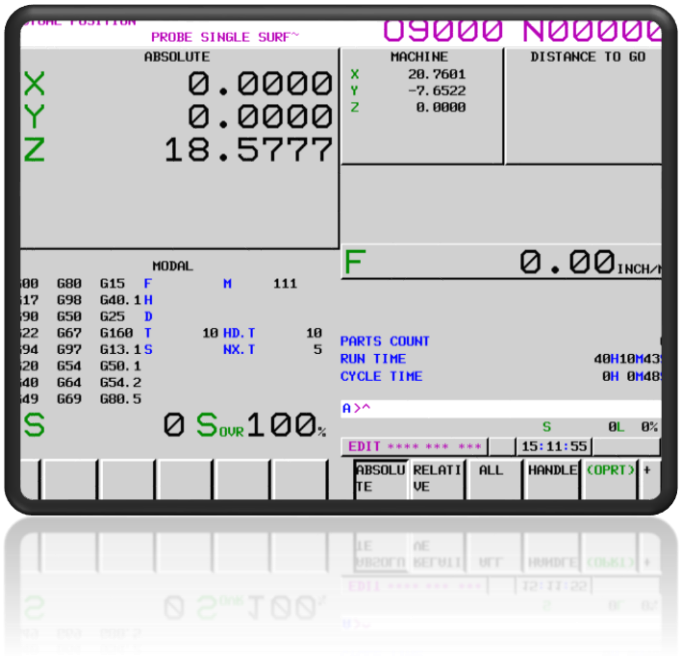
DOOR INTERLOCK:

- **Release** must be used when doors are unlocked to allow moving the axis. During release mode spindle and automatic operations are prohibited. **Lock** must be used when the doors are closed, to allow full automatic operation of the machine. *Some machines may support automatic door locking, in this case, the Door Interlock key switch should be left to lock.

NC Screens / Hard Keys



Positions Screens



POS Hard Key

- **POS** shows the positons of the axis available. **Absolute** is the position of the tool in relation to the workpiece offset. **Relative** is a user setting zero. **All**, displays all the types of positions. **Handle**, shows position during handle interrupt mode.

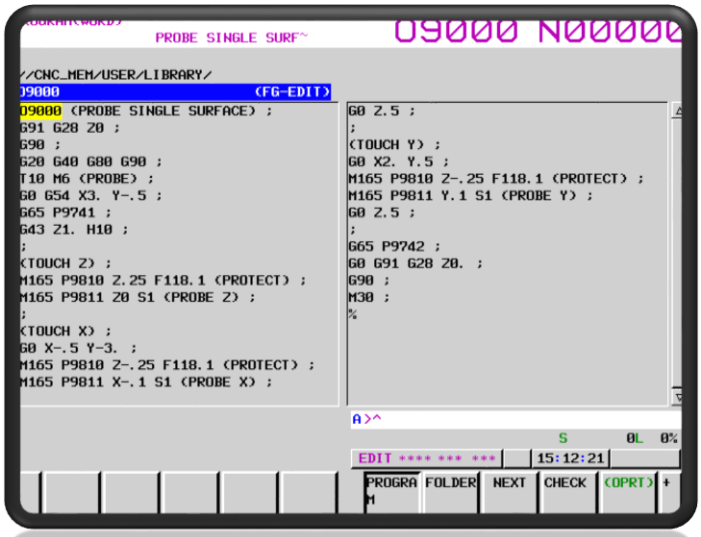
Relative position zero instruction

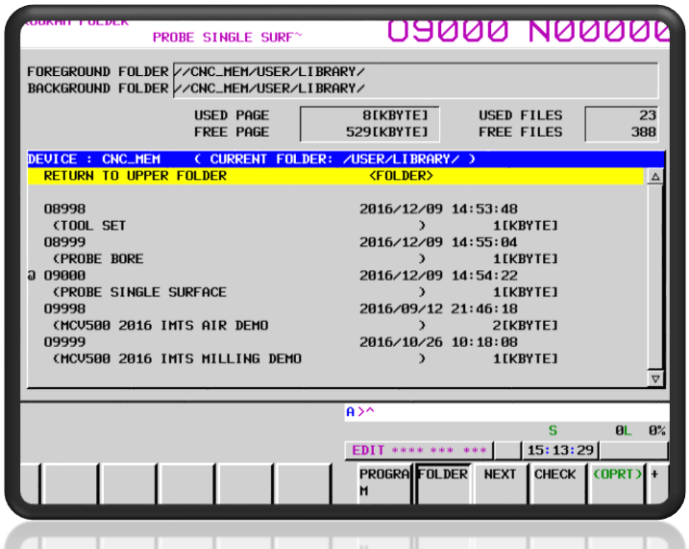
- Push **POS** hard key.
- Push **RELATIVE** soft key
- Push desired axis (i.e. x, y, z,)
- Push the **ORIGIN** soft key.
- Push the **EXEC** soft key. This will set the relative position counter to zero.

Relative position preset instruction

- Push **POS** hard key.
- Push **RELATIVE** soft key.
- Push desired axis hard key (i.e. X, Y, Z,)
- Enter a value that you want to change the relative position of that axis to. (Z26.9064) and push the **PRESET** soft key.

Program Screens





PROG Hard Key

- Toggles between the current Main program and Folder of CNC memory programs.

PROGRAM Soft Key

- Toggles between full page program, and check screen program display.

FOLDER Soft Key

- Selects the folder directory which displays all programs within the folder.

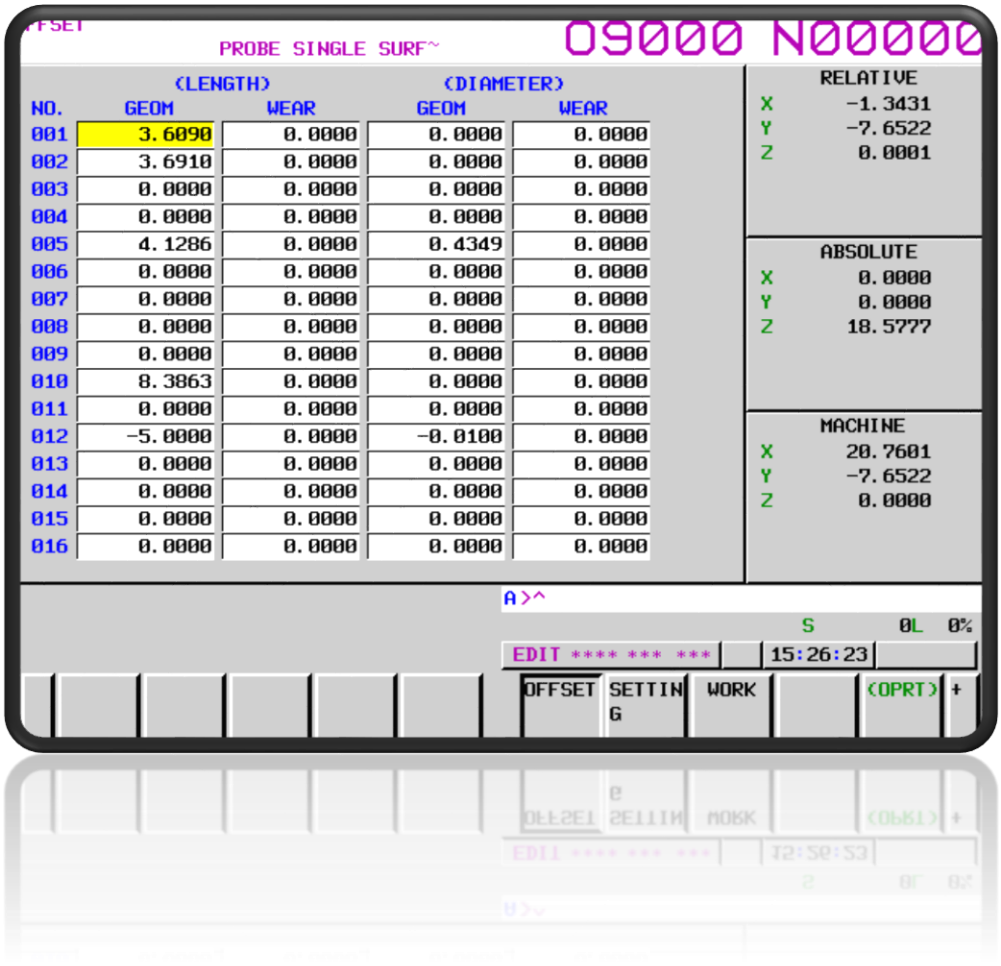
NEXT Soft Key

- Displays the next block machining information.

CHECK Soft Key

- Display the program check screen, which shows the program and all active axis position and code information.

Tool Offsets Screen

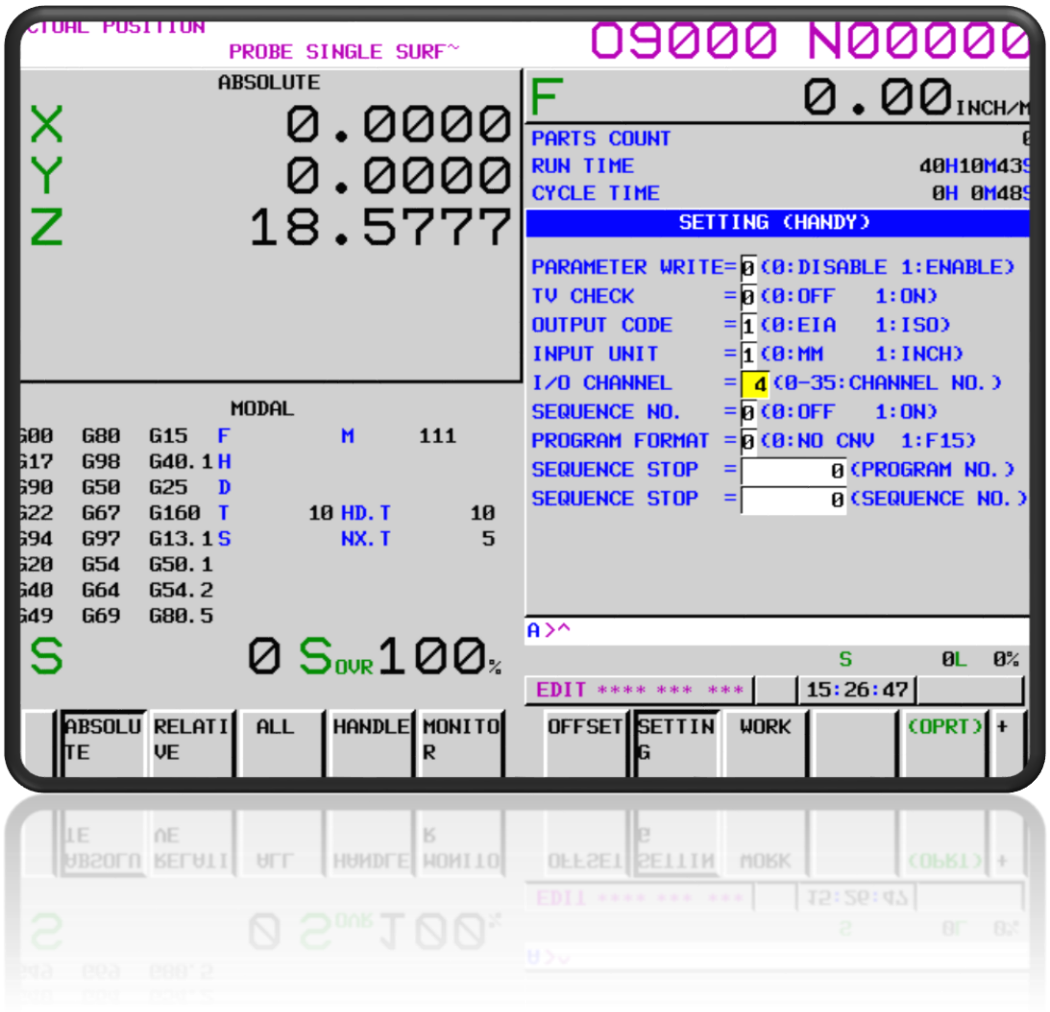


OFS/SET Hard Key

- Toggles between the OFFSET/SETTINGS page selection.

- OFFSET Soft Key: Selects the Tool Offsets screen. ***NC Parameter 5004.2** will change between radius and diameter use for the tool cutter compensation column.

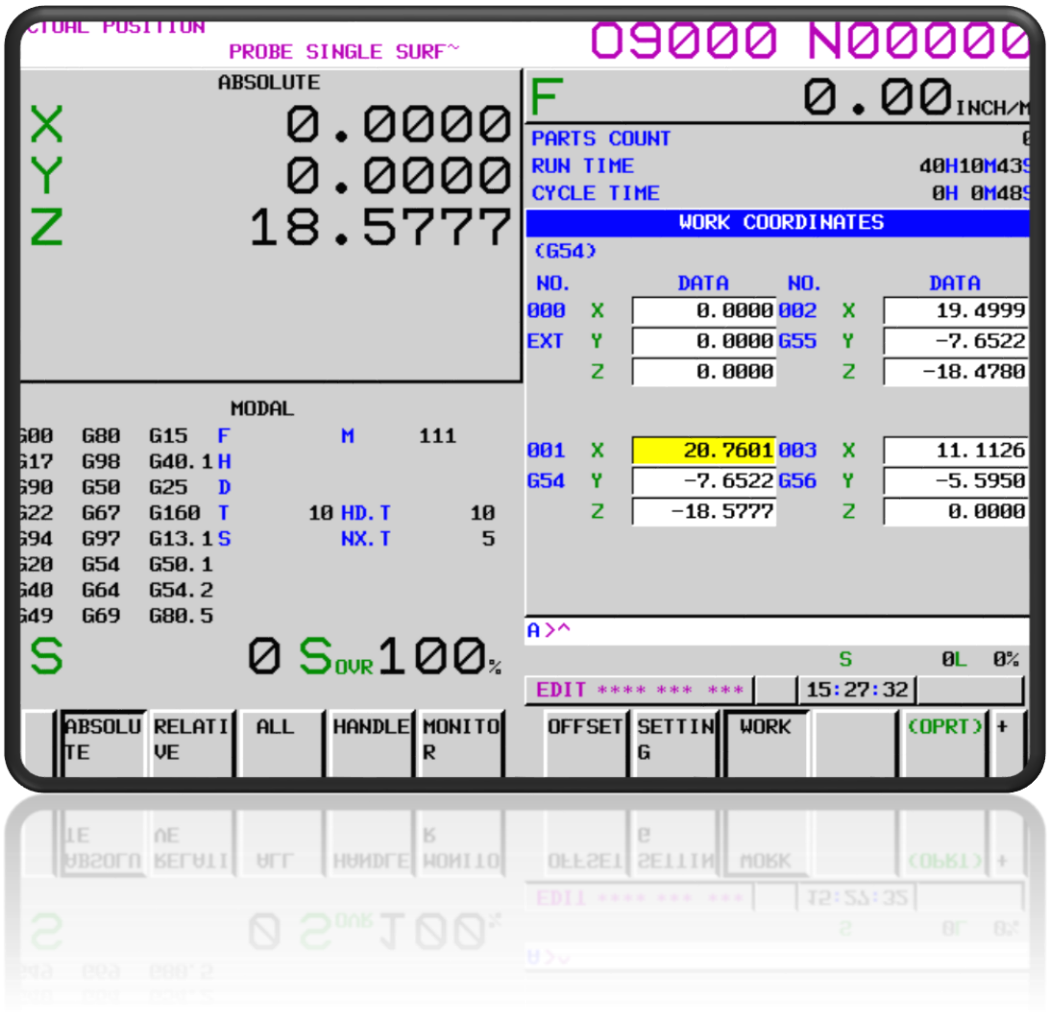
Settings Screen



SETTINGS Soft Key

- Selects screen for functions **Parameter Write** for changing parameters, setting the **I/O Channel** for using **USB**, **Memory Card**, **Ethernet** use, and when page down is pressed features like **Parts counter** and **Run Timers**.

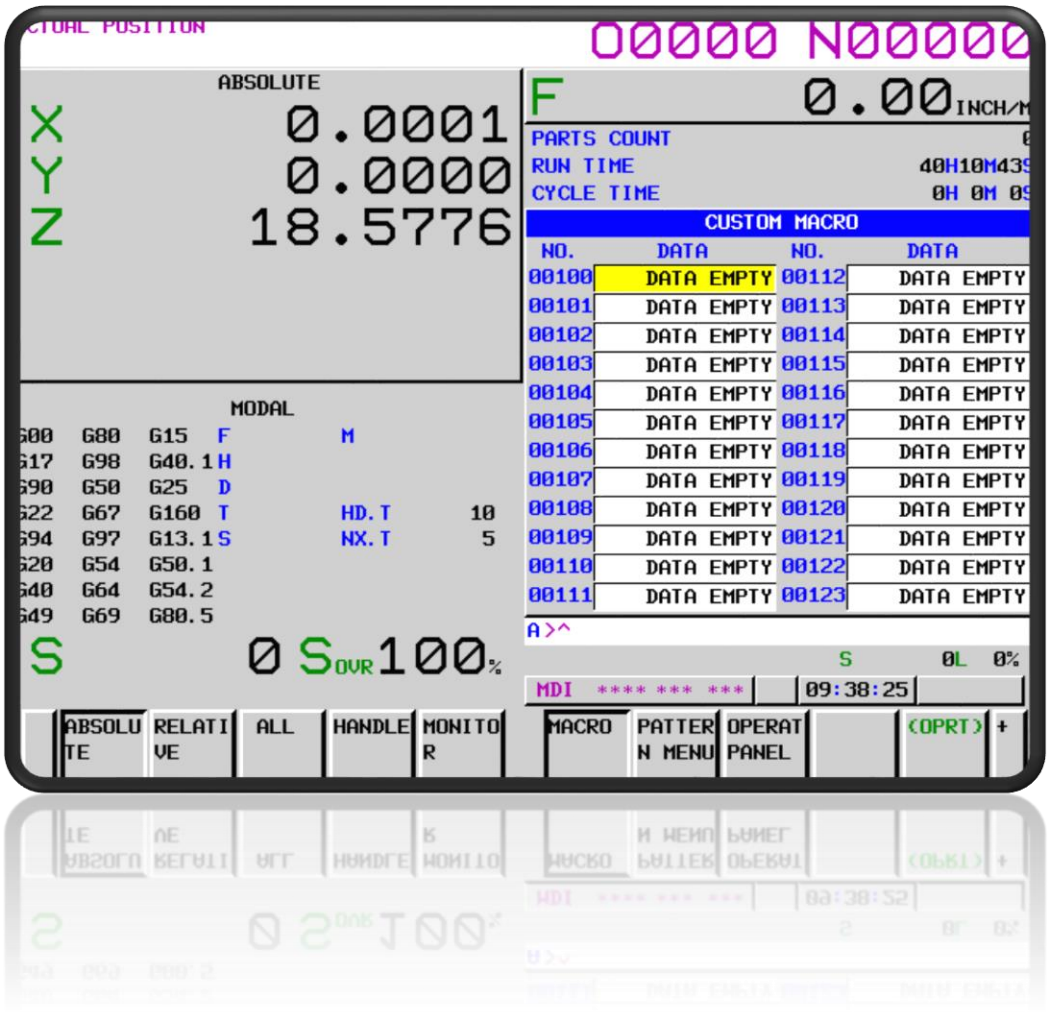
Work Offsets Screen



WORK Soft Key

- **WORK** Soft Key is available after pressing the **OFS/SET** Hard Key. Selects the pages of **WORK** Coordinates. **EXT, G54, G55, G56, G57, G58, G59**, as well as **G54.1 P1** through **G54.1 P48** are available by paging up/down.

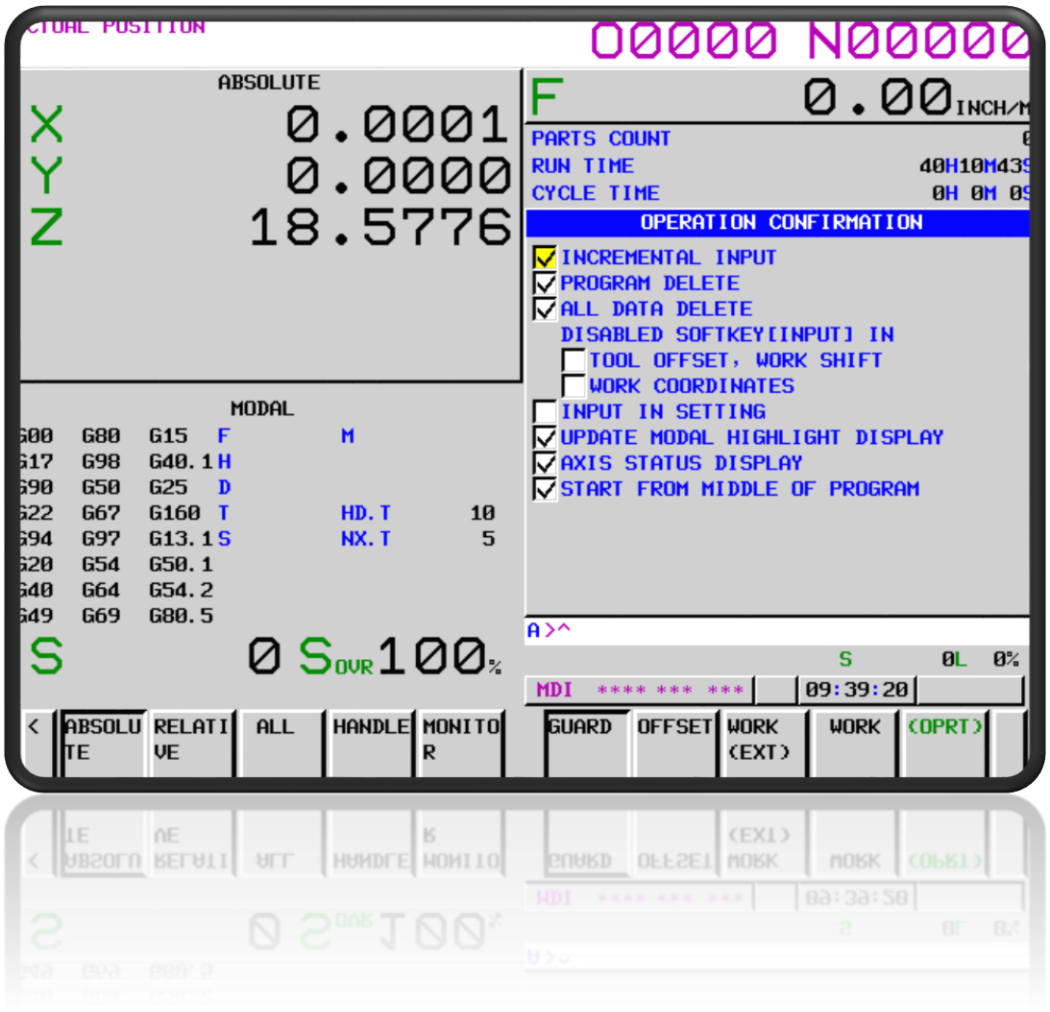
Macro Variables Screen



MACROS Soft Key

- **MACRO** Soft Key is available after pressing **OFS/SET** hard key and then pressing the right or left most soft key to display the next pages selections.
- Paging **up/down** will cursor through the user macro B variables.

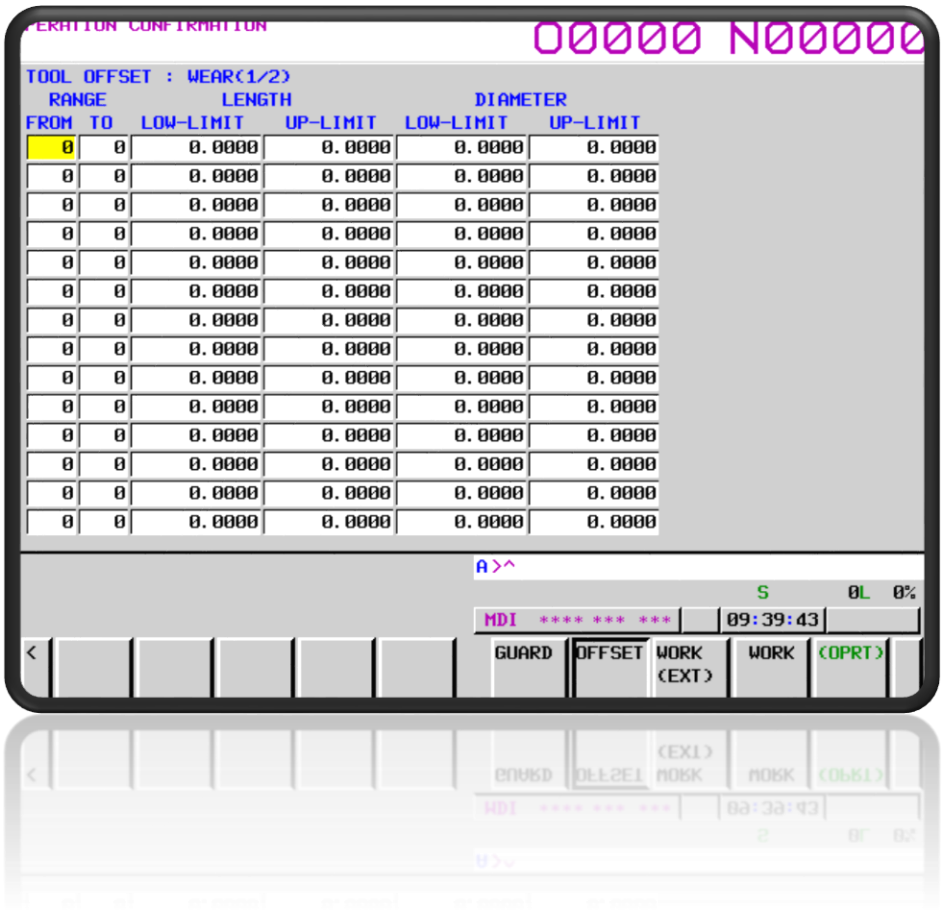
Guard Screens



GUARD Soft Key

- **GUARD** Soft Key is available after pressing **OFS/SET** hard key and then pressing the right or left most soft key to display the next pages selections.
- Page Shown sets confirmations for the fields displayed. If box is checked, and Incremental Input for example, will require a confirmation before executing. If the box is not checked, no confirmation will be required.

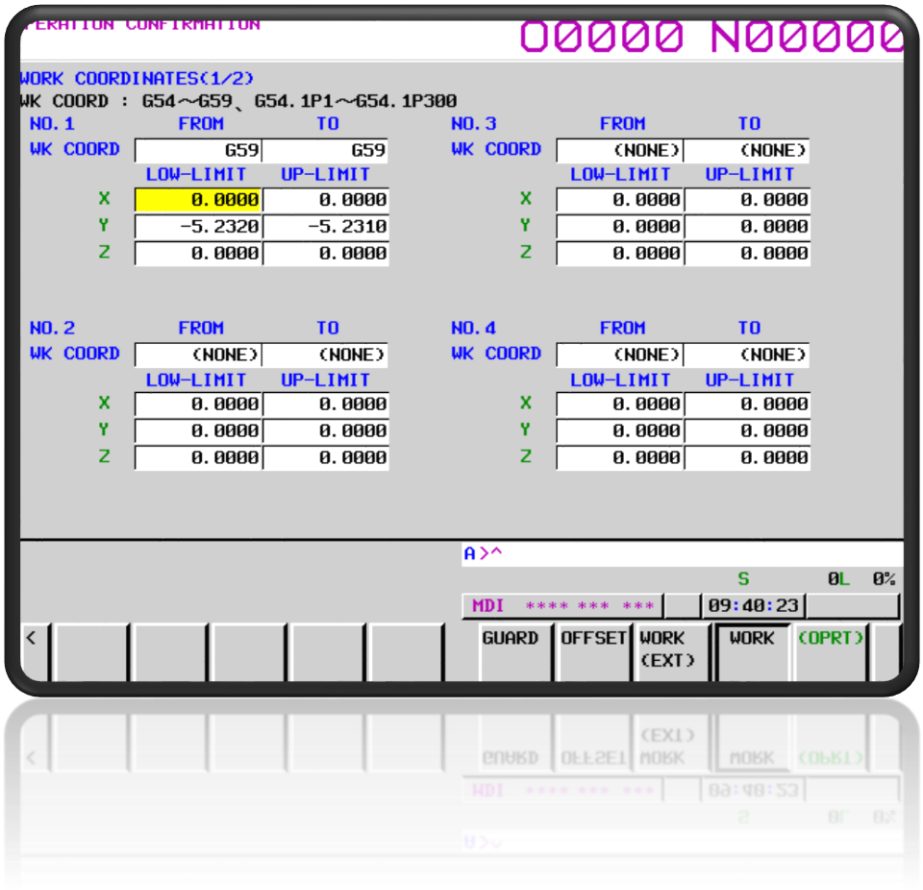
Tool Offsets Guard Screen



OFFSET Soft Key

- **OFFSET** Soft Key is available after pressing the **GUARD** soft key.
- Allows setting a range of tools, and lower and upper limits to the value allowed to be entered into the **Tool Length Geometry, Wear, Tool Diameter Geometry, and Wear Offset fields**.

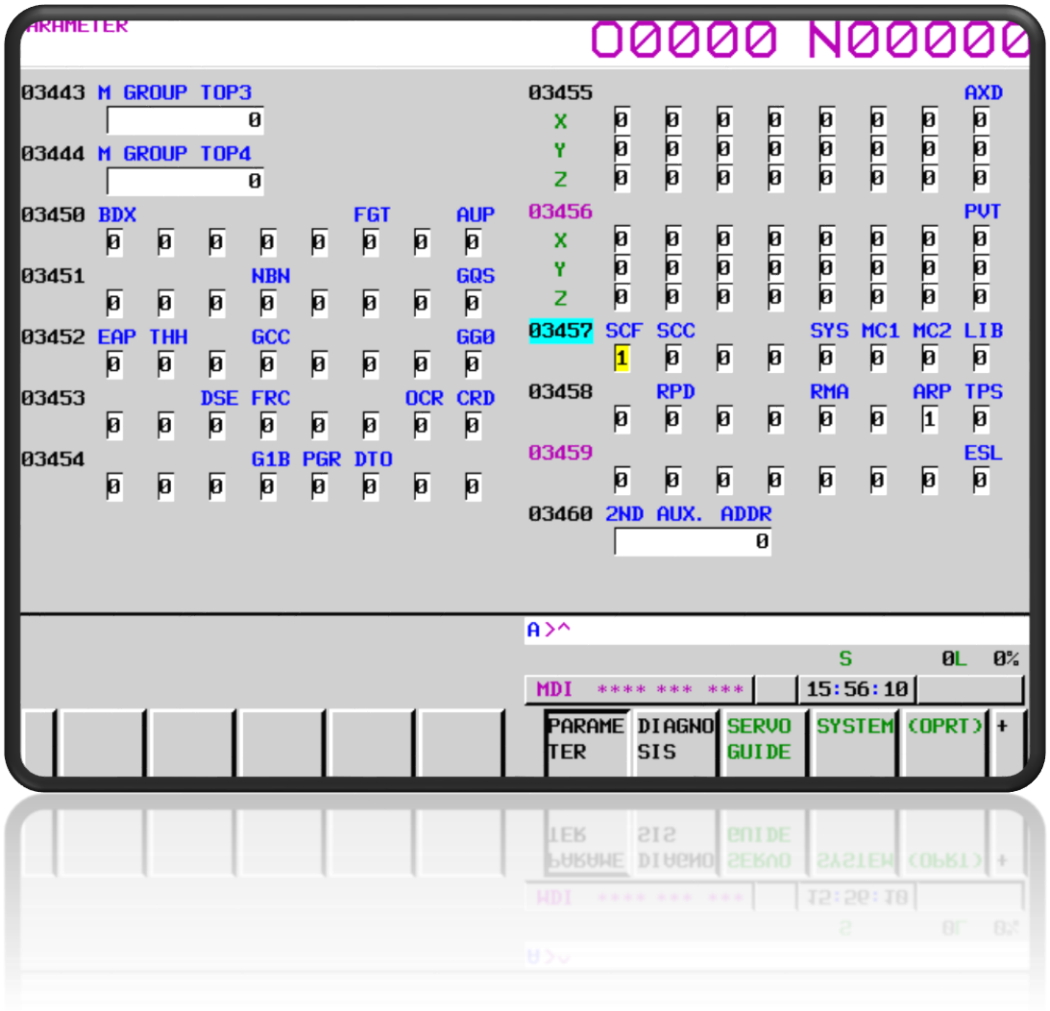
Work Offsets Guard Screen



WORK Soft Key

- **WORK** Soft Key is available after pressing the **GUARD** soft key.
- Allows setting a range of Work Coordinates, and setting a lower and upper limit to the value allowed to be entered into that range of coordinates.

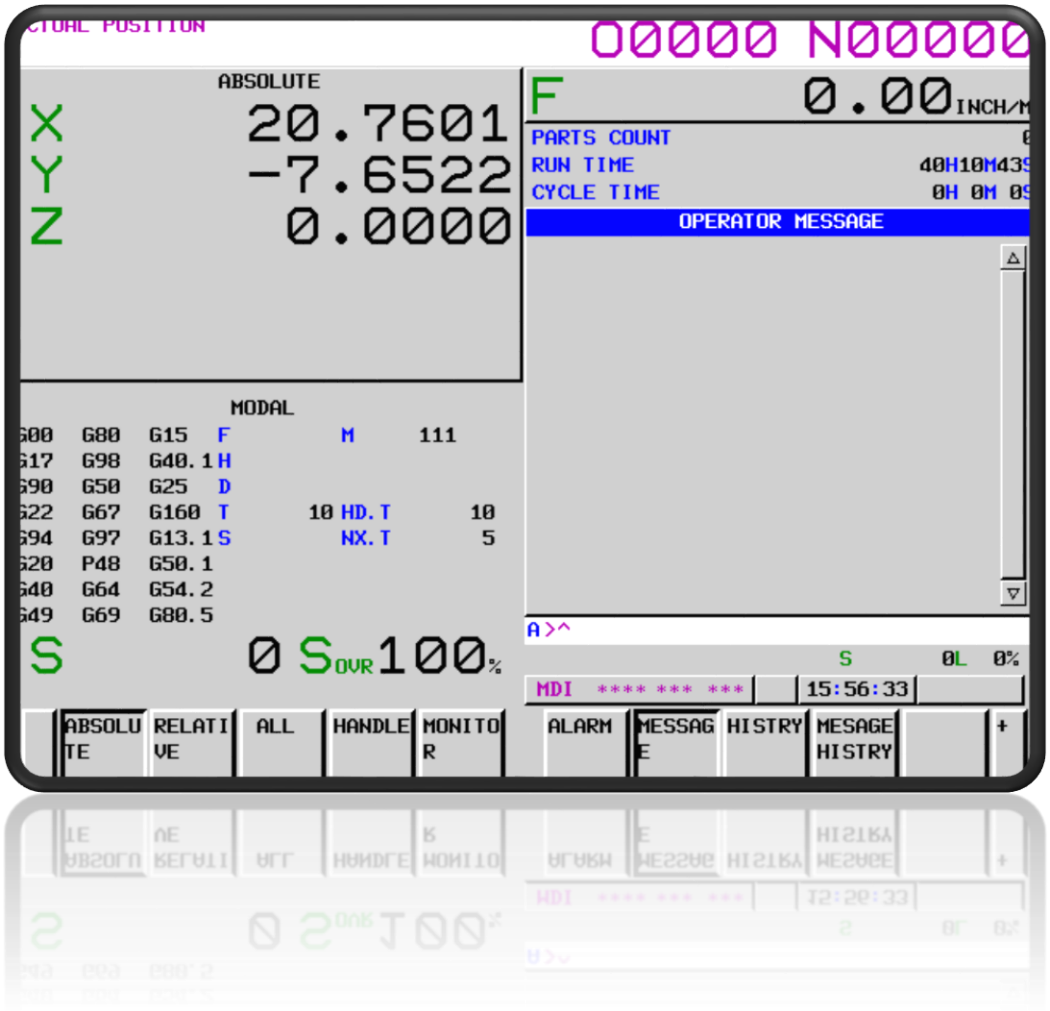
Parameters Screen



PARAMETERS Soft Key

- Available from the **SYSTEM** hard key, then **PARAMETER** soft key. To find a specific parameter in question, type in the parameter number, then press the **NO.SCH** soft key at the lower left. To change a parameter **PWE** from the **SETTINGS** page must be set to 1.

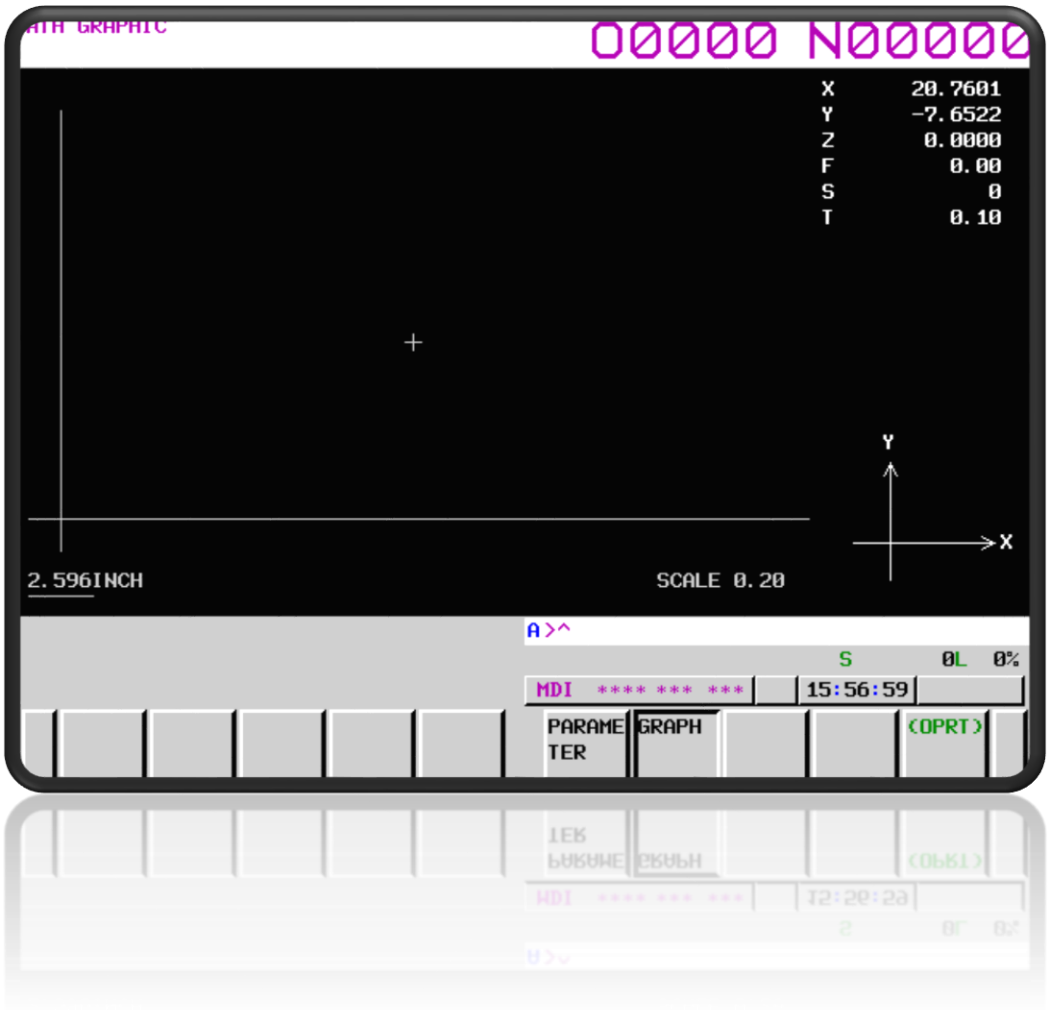
Alarms Screen



ALARMS

- Available from the **MESSAGE** hard key, then **ALARM**, **MESSAGE**, **HISTRY**, or **MESSAGE HISTRY** soft keys. This will display **SAMSUNG** alarms and **FANUC** alarms, as well as a history of each.

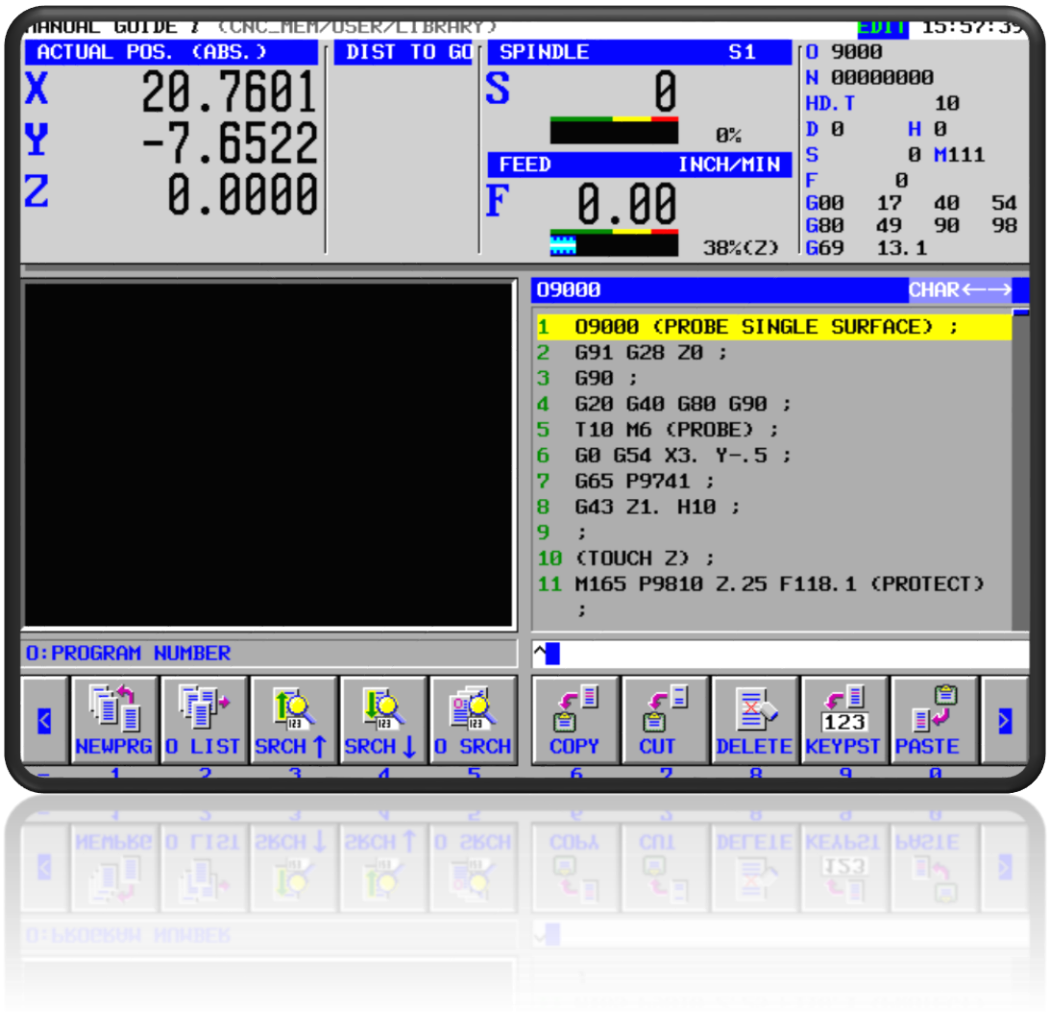
Graph Screen



GRAPH

- Used during automatic operation, shows tool path as being executed.

Manual Guide I Screen



MANUAL GUIDE I

- Available from the **Custom 2** hard key. Conversational Programming Software.

Negative Tool Length Setting

1. Push the O**ffset Setting** hard key
2. Push **Offset** soft key
3. Zero return **Z-Axis**
4. Make sure **Relative Z Position** is zero. *see note below.
5. Push **Handle** mode hard key.
6. Move **Z**-**Axis** negative to the measuring point.
7. Cursor to the desired offset number.
8. Make the cursor is in the length column.
9. Push the **Z** letter key, then push the **INP**.**C** soft key. The offset is now registered.
10. If the tool needs to have a radius compensation value entered, you may do so now, by moving the cursor to the radius column and entering a value.
11. Push the **ZRN** button.
12. Push and hold the **Z+** button until Z Axis is in the home position.
13. If needed go the **MDI** page and cycle the next tool into the spindle.
14. Repeat steps **5** through **13** for the remaining tools.
15. *Press **POS** hard key, Press **REL** soft key. Press **Z**, then **ORIGIN** soft key. Press **EXEC** soft key.

Tech Tip**, set NC Parameter 3104.3 so relative display will match machine position at machine power up

Positive Tool Length Setting

1. Remove any tool from spindle.
2. Using a gage block, bring the spindle face to the surface that you are going to set your tools to.
3. Push **POS** hard key.
4. Push **REL** soft key.
5. Push the **Z** hard key enter a **0**
6. Push the **Preset** soft key. While in this position go to offset settings h/k goto desired offset page g54-g55-ect… arrow key to the z field, type in z (size of gauge block) *example:* ***(Z2.)*** Press **Measure** s/k
7. Go to the **MDI** page and get a tool that you want to set the tool length to, into the spindle.
8. Push **Offset Setting** hard key.
9. Push **Offset** soft key.
10. Push **Handle** mode hard key.
11. Move Z-Axis negative to the measuring point (on top of gauge block used to set relative.)
12. Cursor to the desired offset number.
13. Make sure it is in the length column.
14. Push the **z** letter key, then push the **INP.C** soft key. Offset is now registered as the true tool length. This can be double checked with a scale or tape measure.
15. Push the **ZRN** button and select another tool if needed.
16. Repeat steps 10 through 15 for the all remaining tools.

Tech Tip**, Set NC parameter 3104.3=1 so relative display will match machine position at power up

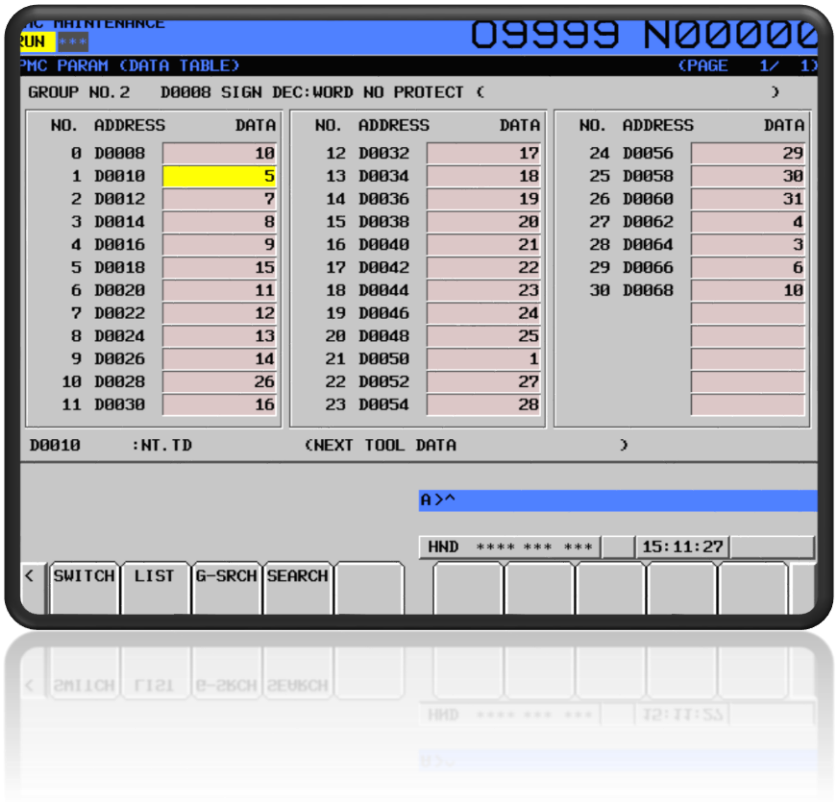
Setting Work Offsets

1. Using an indicator or edge finder pick-up piece part zero.
2. Press the **Offset/Settings** hard key, then push the **Work** soft key.
3. Cursor to the desired work offset you wish to set. Push the letter key of the axis in which you have highlighted **(i.e. X, Y, Z…)**. Then push **0**, then the **measure** soft key. If you are using a gage tool or a tool with a known length to set your work offset, enter that length in the command line after entering the axis letter, then push the measure soft key. **(i.e. Z8.5000 – measure soft key)**
4. Repeat for each axis that is to be set.
5. Input soft key enters values from the command line into the area that is highlighted.
6. Input+ soft key incrementally adds the value from the command line into the area that is highlighted. EXEC is required to confirm the new value.
7. Measure soft key takes the value from the command line and incrementally adds the value to the machine position and puts the resulting value into the area that is highlighted.

Setting ATC Tool Numbers

(FOR MACHINES WITHOUT SMEC ATC SOFTWARE)

1. PRESS THE **SYSTEM** HARD KEY.
2. RIGHT ARROW SOFT KEY UNTIL **PMC MAINT**, PRESS **PMC MAINT**.
3. RIGHT ARROW SOFT KEY UNTIL **DATA**. PRESS **DATA**.
4. PRESS **OPRT**.
5. PRESS **ZOOM**.
6. PAGE DOWN TO **D0010** AS SHOWN IN PICTURE. THIS IS **POT 1**.
7. **NO**. 1-24/30/40 ETC ARE THE POTS AND CAN SET TO ANY TOOL # ***EXCEPT*** **THE TOOL NUMBER THAT CURRENTLY IS IN THE SPINDLE.**



*****DO NOT ROTATE THE MAGAZINE UNTIL YOU HAVE UPDATED THE LIBRARY*****

*****TECH TIP** SET NC PARAMETER **3032** TO **3** TO ALLOW A **3 DIGIT T CODE**, FOR USE OF TOOL NUMBERS UP TO **999*****

M.S.T. Commands

1. Push **PROG** hard key.
2. Put machine in **MDI** mode.
3. Program (MDI) will appear at the bottom of the screen.
4. Enter desired **M, S,** or **T** codes followed by a semi-colon at the end.
5. Push **INSERT** hard key.
6. Push **CYCLE START** to activate.

Programming Addresses

X Axis Name
Y Axis Name
Z Axis Name
A Axis Name
B Axis Name
G Predatory Command
M Machine Function
S Spindle Speed
T Tool Number
H Length Offset Number
D Radius Offset Number
F Feed rate
I Center Point Coordinates Incremental from Start Point in G2, G3 for X
J Center Point Coordinates Incremental from Start Point in G2, G3 for Y
K Center Point Coordinates Incremental from Start Point in G2, G3 for Z
N Sequence Number
Program Number Label (Fanuc Format)
P Number of Loop Times
Q Peck Depth in G73, G83
R Radius amount for G2, G3
Rapid Plane in Drilling Cycles

*THE FOLLOWING ADDRESSES DO NOT USE DECIMAL POINTS: **M, S, T, P, O, N, L, H**

M-Codes

M00	**Program Stop**
M01	**Optional Stop**
M02	**Program End**
M03	**Spindle CW**
M04	**Spindle CCW**
M05	**Spindle Stop**
M06	**ATC Cycle**
M07	**Oil Mist (Option)**
M08	**Flood Coolant ON**
M09	**Coolant OFF**
M10	**4th Axis Clamp**
M11	**4th Axis Un Clamp**
M19	**Spindle Orient**
M25	**Tool Life Signal**
M27	**Rigid Tapping CW**
M28	**Rigid Tapping CCW**
M29	**Rigid Tapping**
M30	**Program End & Rewind**
M48	**Feed Override Inhibit**
M49	**Feed Override Apply**
M50	**Air Blow (Option)**
M53	**Tool Setter ON (Option)**
M54	**Tool Setter Wait (Option)**
M55	**Probe ON (Option)**
M56	**Probe OFF (Option)**
M57	**Tool Setter Air Blow**
M58	**Through Spindle Coolant ON**
M59	**Through spindle Coolant OFF**
M60	**APC Rotate (MCV380TD Only)**
M61	**APC Pallet 1 Confirm (MCV380TD Only)**
M62	**APC Pallet 2 Confirm (MCV380TD Only)**
M68	**Chip Conveyor ON (Option)**
M69	**Chip Conveyor OFF (Option)**
M71	**Tailstock Forward (Option)**
M72	**Tailstock Reverse (Option)**
M73	**Tailstock Neutral (Option)**
M75	**Mirror Image ON**
M76	**Mirror Image OFF**

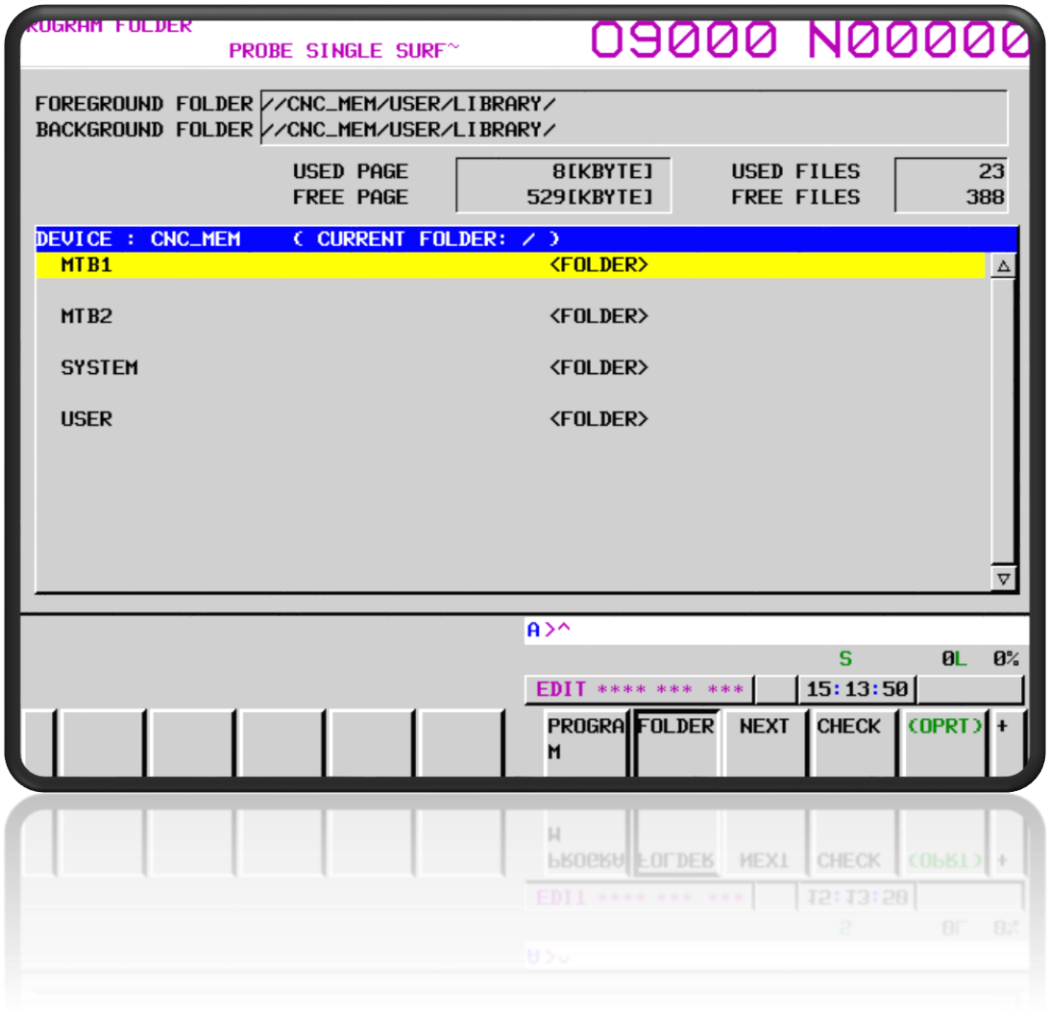
M85	Auto Door Open (Option)
M86	Auto Door Close (Option)
M89	Parts Counter UP Signal
M98	Sub Program Call
M99	Sub Program Call End
M100	Pot Down
M101	ATC Arm Cycle
M102	Pot UP
M103	ATC Mag Turn 1 Pot
M104	Spindle Tool Un Clamp
M105	Spindle Tool Clamp
M109	Work Probe Setting Mode (Option)
M110	Work Probe Selected (Option)
M111	Work Probe Not Selected (Option)
M153	Through Spindle Air Blow ON
M154	Through Spindle Air Blow OFF
M198	External Sub Program Call
M200	Renishaw Tool Measure Cycle (Option)
M300	Renishaw Tool Broken Cycle (Option)

G-CODES

G00	Rapid Traverse
G01	Linear Interpolation (Cutting Feed)
G02	CW Circular Interpolation
G03	CCW Circular Interpolation
G04	Dwell
G04.1	G Code Prevent Buffering
G05.1	APC / AI Contour Control
G09	Exact Stop
G10	Programmable Data Input
G11	Programmable Data Input Cancel
G15	Polar Coordinates Cancel
G16	Polar Coordinates Command
G17	XY Plane Selection
G18	ZX Plane Selection
G19	YZ Plane Selection
G20	Input in Inch
G21	Input in MM
G22	Stored Stroke Check ON
G23	Stored Stroke Check OFF
G27	Reference Position Return Check
G28	Return To Reference Position
G29	Movement From Ref Position
G30	2nd,3rd, and 4th Ref Positon Return
G31	Skip Function
G40	Tool Radius Comp Cancel
G41	Tool Radius Comp Left
G42	Tool Radius Comp Right
G43	Tool Length Comp +
G44	Tool Length Comp -
G49	Tool Length Comp Cancel
G50	Scaling Cancel
G51	Scaling ON
G50.1	Programmable Mirror Image Cancel
G51.1	Programmable Mirror Image ON
G52	Local Coordinate System Setting
G53	Machine Coordinate System Setting
G54-G59	Work Offset Coordinates
G54.1P1-G54.1P48	48 Extra Work Offset Coordinates
G60	Single Direction Positioning
G61	Exact Stop Mode

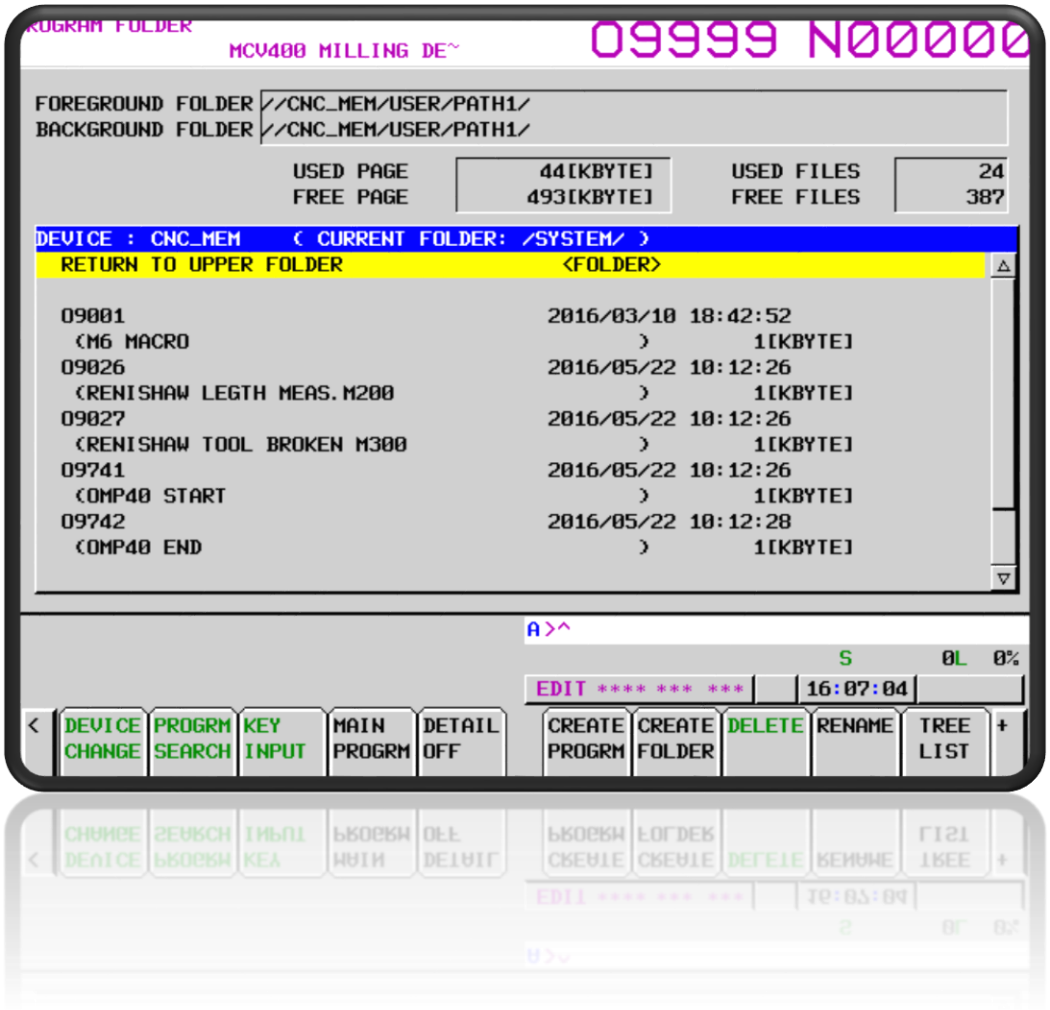
G62	Automatic Corner Override
G63	Tapping Mode
G64	Cutting Mode
G65	Macro Call
G68	Coordinate System Rotation
G69	Coordinate System Rotation cancel
G73	Peck Drilling Cycle
G74	Left hand Tapping Cycle
G76	Fine Boring Cycle
G80	Drilling Cycle Cancel
G81	Drilling Cycle
G82	Spot Drilling Cycle
G83	Peck Drilling Cycle
G84	Tapping Cycle
G85	Boring Cycle
G86	Boring Cycle
G87	Back Boring Cycle
G88	Boring Cycle
G89	Boring Cycle
G90	Absolute Programming
G91	Incremental Programming
G92	Setting For Workpiece Coordinate System
G93	Inverse Time Feed
G94	Feed Per Minute
G95	Feed Per Rotation
G98	Canned Cycle: Return to Initial Point
G99	Canned Cycle: Return to R Level

Folder / Program Management



FOLDER INFO

- Under the FOLDER Soft Key selection is the ability to Select, Create, and Choose folders to store programs. MTB1, MTB2, and System Folders contain any Samsung specific programs. Under the USER Folder, new end user Folders can be created, or the Library, Path 1, or Path 2 folders can be used to store end user programs.



FOLDER INSTRUCTION

- Pressing **OPRT** reveals options for **device change, program search, key input, main program, detail, create program, create folder, delete, rename, and tree list.**
- **DEVICE CHANGE**: Allows selecting an external device to view
- **MAIN PROGRAM**: Selects the highlighted program as the main program to run. Hard key up/down to select.
- **DETAIL ON/OFF**: Turns on/off the program line comments.

- **CREATE PROGRAM**: Key in the name of a new program, and press create program to make new program. Program names can any combination of numbers and letters of up to 5 digits total. When parameter 11304.3=1 up to 8 digit program names are available.
- **CREATE** folder: Key in the name of a new folder, press create folder to make a new folder.
- **DELETE**: Deletes the highlighted program selection.
- **RENAME**: Type in a new name for a folder, press rename, the folder highlighted will be renamed.
- **TREE LIST**: Will display the folder tree.

Incremental (G91) vs Absolute (G90) Programming

- When in absolute mode (**G90**) all program numbers are in reference from the work piece zero point.
- When in incremental mode (**G91**) all program numbers are in reference from the current tool position.
- If the tool position is at **X5**. **Y5**. And in the absolute mode (**G90**) **G0 X1**. **Y1**. Is commanded, both the **X**-axis and **Y**-axis move 4.0 inches, and the new current position **X1**.0 & **Y1**.0 in absolute values.
- If the tool position is at **X5**. **Y5**. And in the incremental mode (**G91**) **G0 X1**. **Y1**. Is commanded, both the **X-**axis and **Y**-axis move 1.0 inch, and the new current position is **X6**. & **Y6** in absolute values.
- Note how the tool moves differently in the two examples above. The **G90** is in reference to the work part zero, while **G91** moves the tool an incremental amount in the direction commanded.

High Speed Machining

The Samsung MCV Series come standard with a 20 block look ahead AICC function. This can be used for better tool path control during aggressive machining using 3D cutter paths or any helical or spiral type toolpaths using a high feed rate and very small machine movements. This is activated with a G5.1Q1 command before the machining cycle begins. **It also should be turned off at the end of every tool with a G5.1Q0 command before changing to the next tool**.

PROGRAMMING EXAMPLE:

T2M6 (1" HIGH FEED MILL)
T1
G90G58G0X3.375Y-3.M3S3820
G43Z0.1H2M9
G0Z0.01
G5.1Q1 (TURN ON AICC)
G1Z0F250.00
G3Z-0.0417I-0.375J0
I-0.375J0
X3.3689Y-2.9264I-0.4479J0F300.00
X3.3507Y-2.8547I-0.4366J-0.0727
X3.3369Y-2.8197I-0.4174J-0.1441
X3.3202Y-2.786I-0.403J-0.1788
X3.3003Y-2.7533I-0.3862J-0.2125
X3.2776Y-2.7224I-0.3668J-0.2455
X3.2517Y-2.6929I-0.3446J-0.2768
…
…
G0Z.1
G5.1Q0 (TURN OFF AICC)
G0G91G28Z0
M1

*LOOK AHEAD OF 40 BLOCK, 200 BLOCK, AND 400 BLOCK ARE AVAILABLE AS OPTIONS FOR AN ADDITIONAL COST. AS WELL AS A FULL OPTION PACKAGE FOR HIGH SPEED MACHINING SUITED TO 3D DIE/MOLD TYPE WORK.

Rigid Tapping

COMMAND: G84 X, Y, R, Z, AND F
X = POSITION OF MACHINE
Y = POSITION OF MACHINE
R = RAPID PLANE
Z = FINAL DEPTH
F = FEED PER INCH

Do not start spindle with **M3**. **M29** is used to activate rigid tapping. **This must be placed in the line above the tapping cycle line.** When tapping, feed rate override and spindle override are not active.
Program example:

T1M6
G0G90G54X0Y0
G43Z.1H1
S100M29
G84R.1Z-1.F10.
G80
G91G28Z0
M1

Example Programs

```
%
O1111 (FACE MILLING / PROFILING)
G0G91G28X0Y0Z0
G0G90G17G40G49G80
T1M6 (3" 40 DEGREE FACE MILL)
T3
G5.1Q1 (ACTIVATE AICC LOOK AHEAD)
G90G58G0X7.6887Y-1.3M3S1200
G43Z0.25H1M9
G0Z0.01
G1Z0F125.
G3X5.9887Y-3.I0J-1.7F100.
G2I-2.9887J0F125.
G3X6.2387Y-3.25I0.25J0
G0Z0.25
M5
G5.1Q0 (TURN OFF AICC LOOK AHEAD) G49
M9
G91G28Z0Y0M5
M30
%

%
O1112 (DRILLING)
G0G91G28X0Y0Z0
G0G90G17G40G49G80
T2M6 (1/2” DRILL)
T3
G90G58G0X3.Y-3.M3S2000
G43Z0.25H2M8
M58
G81G98R.1Z-1.0F20.
X1.Y-1.
X5.Y-1.
X5.Y-5.
X1.Y-5.
G80M9
G91G28Z0Y0M5
M30
```

```
%


%
O1113 (PECK DRILLING)
G0G91G28X0Y0Z0
G0G90G17G40G49G80
T5M6 (.5” DRILL)
T6
G90G58G0X3.Y-3.M3S1500
G43Z0.25H5M8
G83G98R.1Z-1.0F10.Q.1
X1.Y-1.
X5.Y-1.
X5.Y-5.
X1.Y-5.
G80M9
G91G28Z0Y0M5
M30
%


%
O1114 (RIGID TAPPING)
G0G91G28X0Y0Z0
G0G90G17G40G49G80
T7M6 (1/4-20 TAP)
T4
G90G58G0X3.Y-3.
G43Z0.25H7M8
S200M29
G84G98R.1Z-.8F10.
X1.Y-1.
X5.Y-1.
X5.Y-5.
X1.Y-5.
G80M9
G91G28Z0Y0
M30
%
```

Oversize Tooling

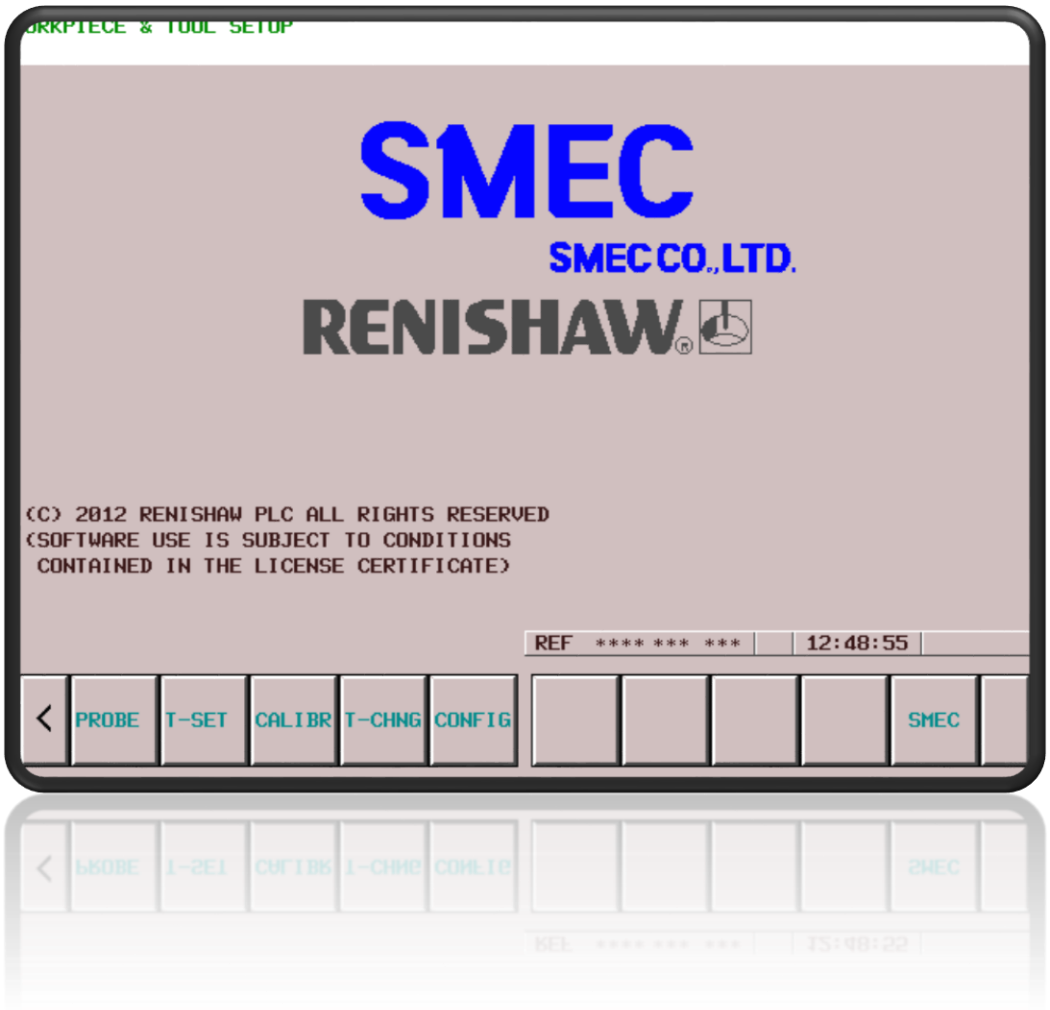
To use an oversize tool in the **ATC** magazine, the oversize tool must be left with an open tool changer pocket on either side, **and the oversize tool must always go back into the pocket it was taken out of. Special care with the NC program must be taken to assure this:**

PROGRAM EXAMPLE:

T1 M6
G90G54G0X1.2345Y5.6789S1000M3;
G43Z2.H1M8;
G81R.1Z-1.F10.:
X10.;
NOTE: NO T-CODES CAN BE COMMANDED BETWEEN M6 COMMANDS.
Y10.;
X-10.;
G80M9;
G0G91G28Z0;
M1;
M6 **(PUTS TOOL 1 BACK INTO SAME POT IT CAME FROM)**
T2 **(BRINGS TOOL 2 INTO THE WAIT POSITION)**
M6 **(PUTS TOOL 2 INTO SPINDLE)**
G90G54G0X1.2345Y5.6789S160M3
G43Z2.H2M8
G81R.1Z-.75F16.
X10.
Y10.
G80M9
G0G91G28Z0

Note: No tool will be in magazine waiting position while oversize tool is in use.

Probing (Option)



Probing with Renishaw GUI (Option)

- Available via the Custom 1 Button and Renishaw Soft Key; please see the Renishaw GUI Probing V2 Manual for complete usage instructions.

Probe G-Code Programming for machine with Renishaw GUI

- Please see the Renishaw OTS & OMP Mill Programming Manuals for complete cycle information and programming details. See below for generic programming examples.
- Note the tool setter M-Codes, M200 for Renishaw Tool Set Cycle, and M300 for Renishaw Tool Broken Cycle. For Probing Cycles note use of M165, which calls to the Renishaw GUI and replace the G65 found in the Renishaw Programming Manual. *OMP ON, OFF, OTS ON, OFF, and Tool Broken Programs must be in the System Folder of CNC memory to make use of the Probing and Tool Setter programming.

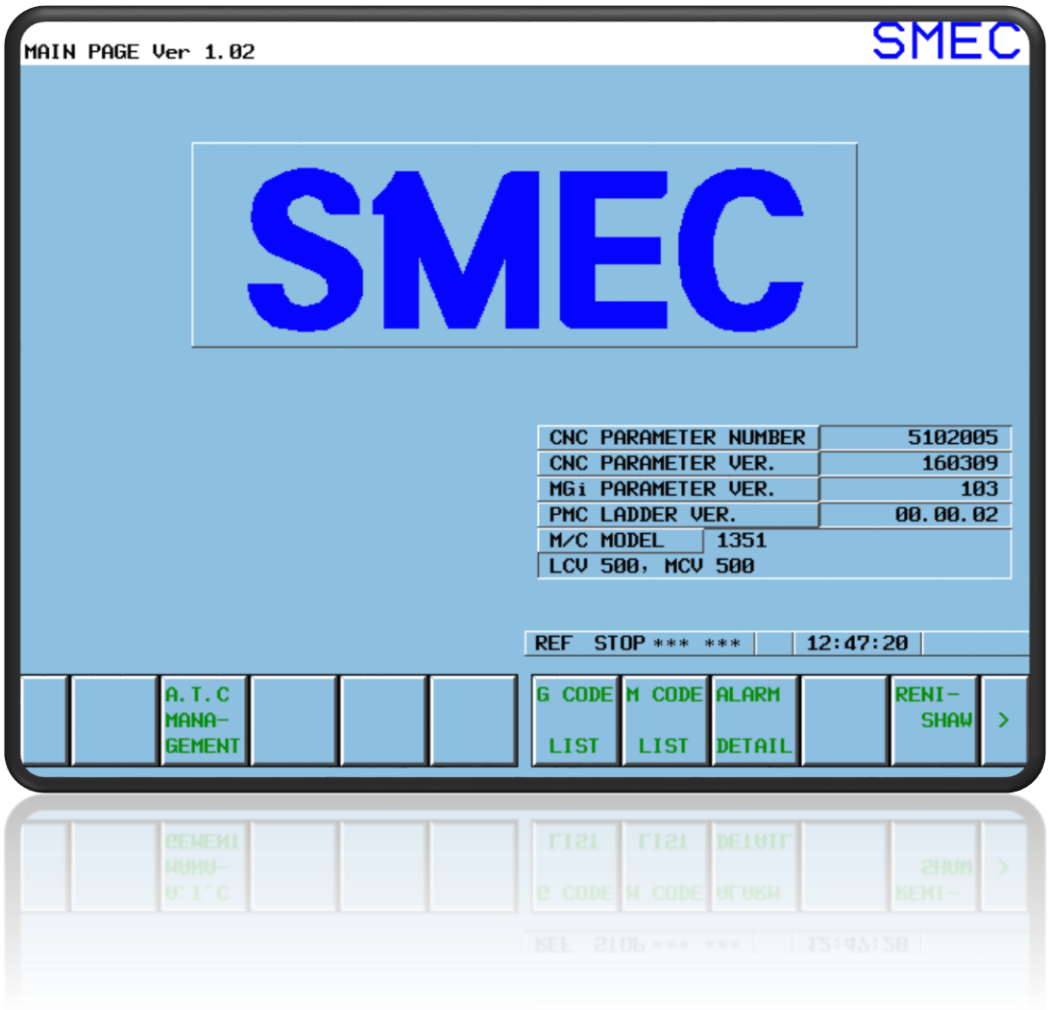
Example Tool Set Program (Calling to GUI)

```
%
O8998(M200 TOOL SET PROGRAM EXAMPLE)
G91G28Z0
G90
T5M6
M200T5D.433Y4.128B3(B3=L+D)(TOOL SET T5)(Y=APPROX LENGTH OF TOOL)(D=DIA)
T6
T6M6
M200T6D.5Y5.B3(B3=L+D)(TOOL SET T6)(Y=APPROX LENGTH OF TOOL)(D=DIA)
T5
G91G28Z0Y0
G90
M30
%
```

Example Probing Programs (Calling to GUI)

```
%
O8999(PROBE BORE EXAMPLE)
G91G28Z0
G90 G20G40G80G90
T10M6(PROBE)
G0G54X0Y0
G65P9741 (PROBE ON)
G43Z1.H10
M165P9810Z-.25F118.1(PROTECTED MOVE)
M165P9814D2.52S1(PROBE 2.52 DIAMETER HOLE TO G54 OFFSET)
G65P9742(PROBE OFF)
G0G91G28Z0.
G90
M30
%
%
O9000(PROBE SINGLE SURFACES EXAMPLE)
G91G28Z0
G90 G20G40G80G90
T10M6(PROBE)
G0G54X3.Y-.5
G65P9741(PROBE ON)
G43Z1.H10
 (TOUCH Z)
M165P9810Z.25F118.1(PROTECTED MOVE)
M165P9811Z0S1(PROBE Z SURFACE SET G54 OFFSET)
(TOUCH X)
G0X-.5Y-3.
M165P9810Z-.25F118.1(PROTECT)
M165P9811X0S1(PROBE X SURFACE SET G54 OFFSET)
G0Z.5
(TOUCH Y)
G0X2.Y.5
M165P9810Z-.25F118.1(PROTECT)
M165P9811Y0S1(PROBE Y SURFACE SET G54 OFFSET)
G0Z.5
G65P9742(PROBE OFF)
G0G91G28Z0.
N500G90
M30
%
```

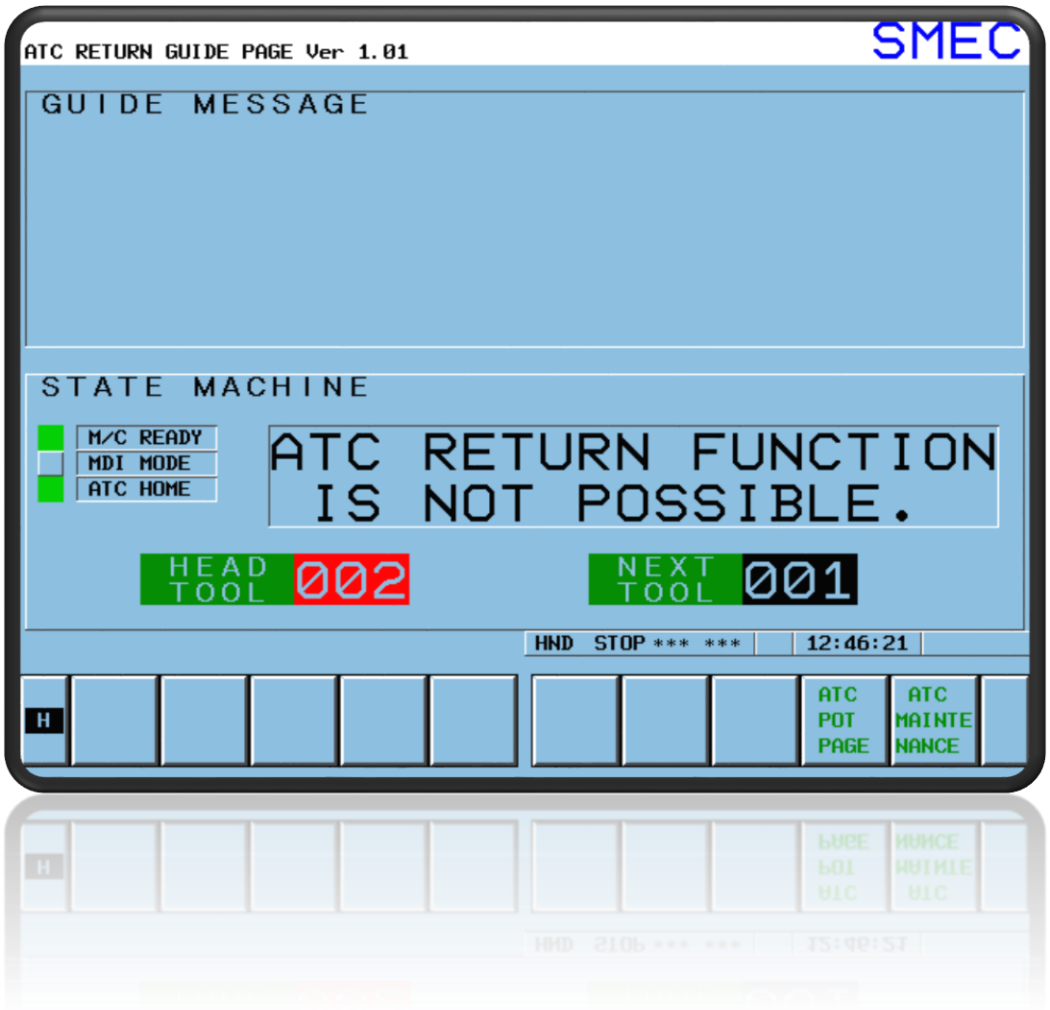
SMT Custom Software (Option)



SMEC Custom Software Contents

- ATC Management
- G-Code List
- M-Code List
- Alarm Detail
- Renishaw

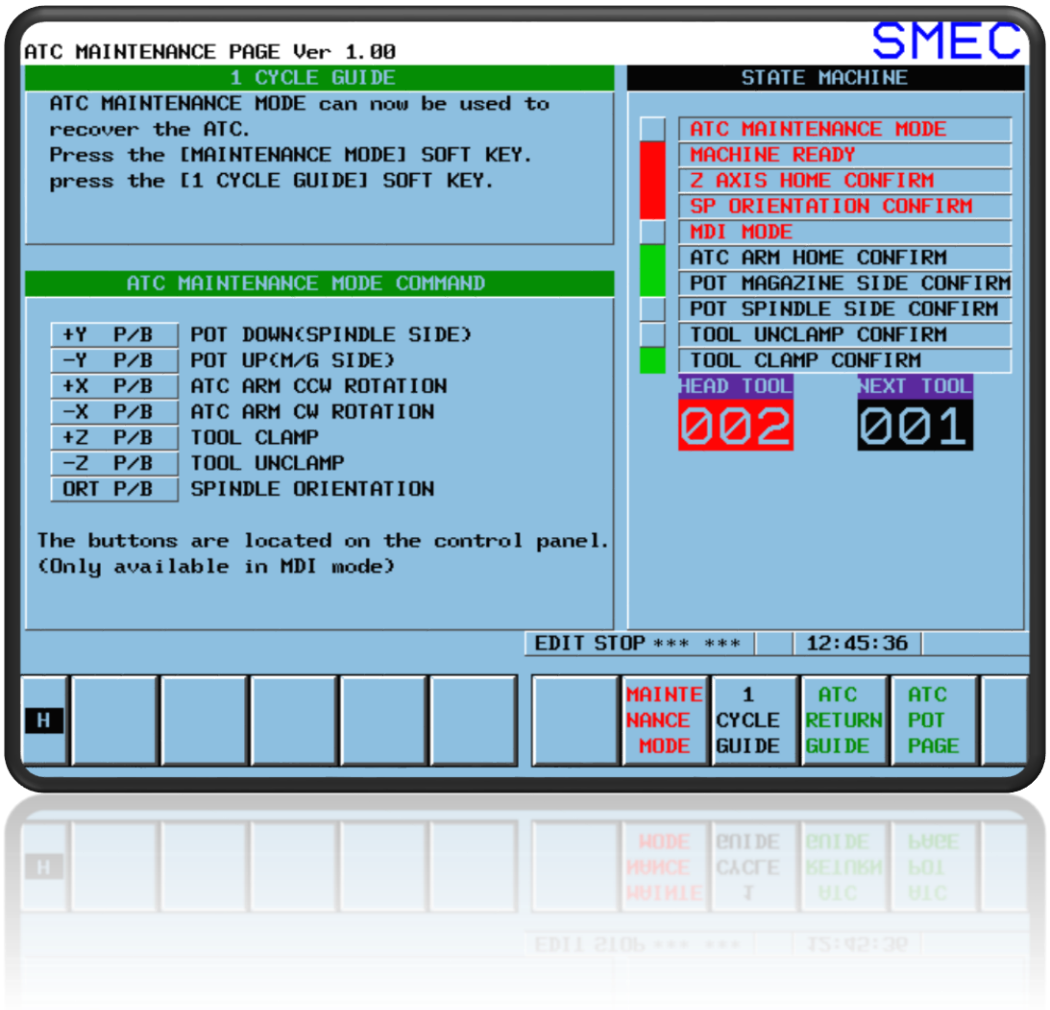
ATC Recover Function



ATC Recovery Guide Function Use

- In the event of an interrupted tool change, please select the Custom 1 Button, ATC Management Soft Key, and ATC Return Guide Soft Key. This will enable message direction to recover from a failed ATC Cycle. Follow the directions as instructed.

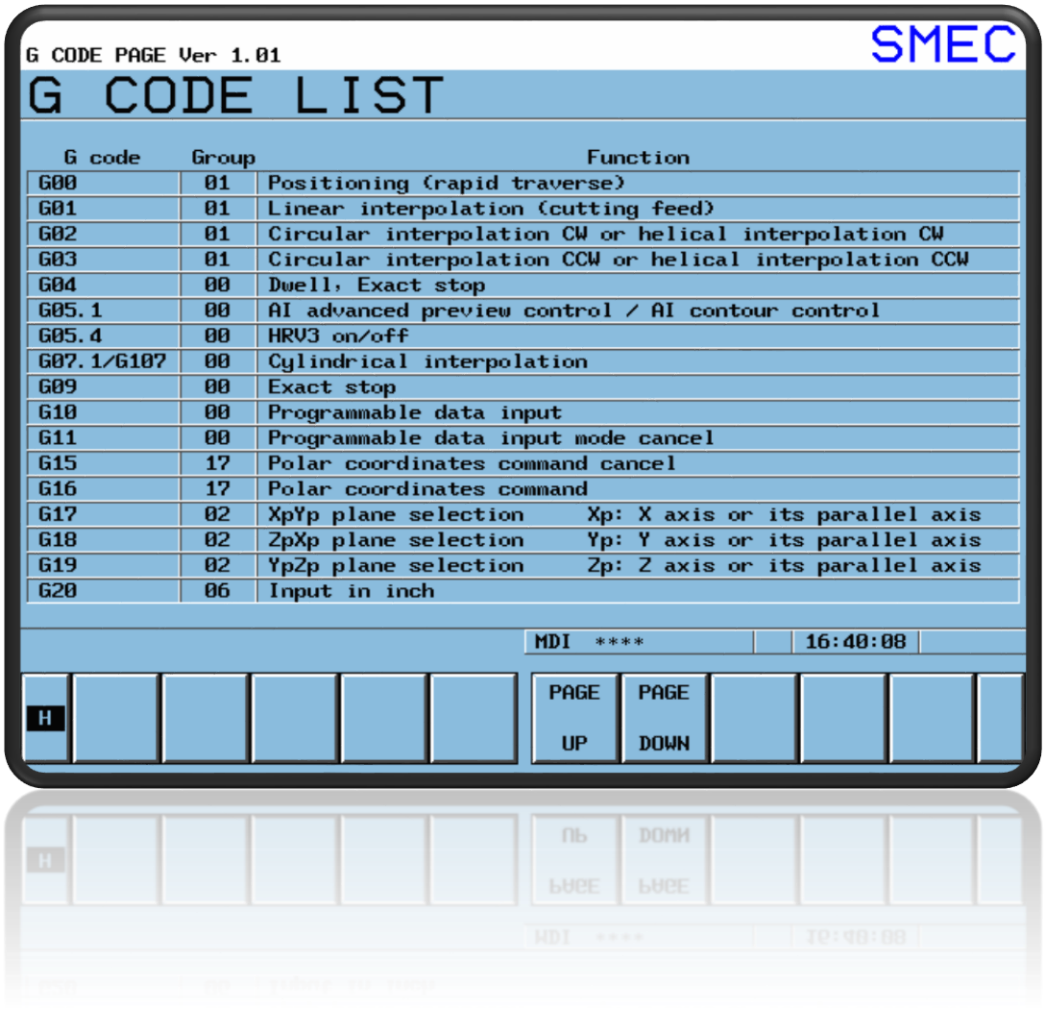
ATC Maintenance Mode



ATC Maintenance Mode Guide Use

- In the event the ATC Return Guide will not return an interrupted/jammed ATC Cycle, you may need to select the ATC Maintenance Mode to fully recover. Follow the instructions based on your conditions, and recover the ATC. If you a situation you cannot recover from, please consult your local Samsung Machine Tools Dealer.

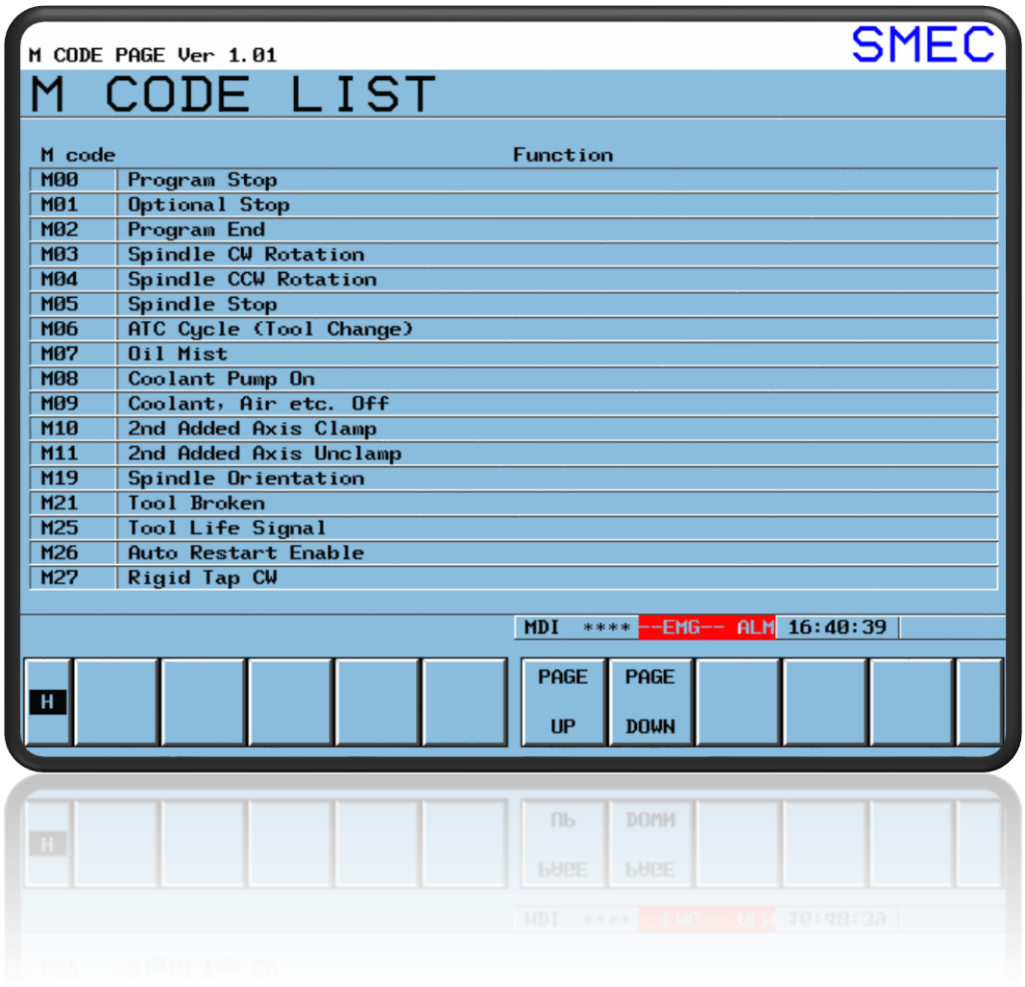
G-Code List



G-Code List

- Under the Custom 1 Button, pressing the G-Code List soft key will display a list of G-Codes for the machine.

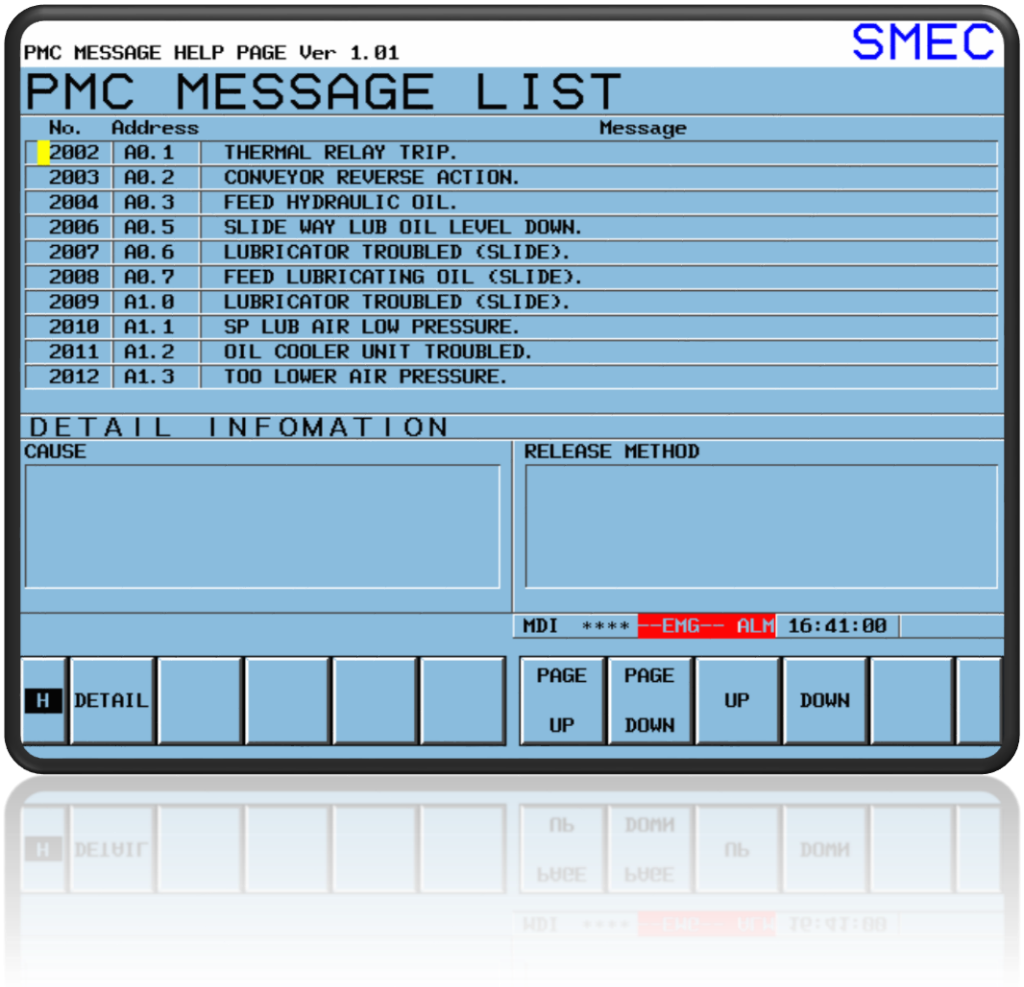
M-Code List



M-Code List

- Under the Custom 1 Button, pressing the M-Code List soft key will display a list of M-Codes for the machine.

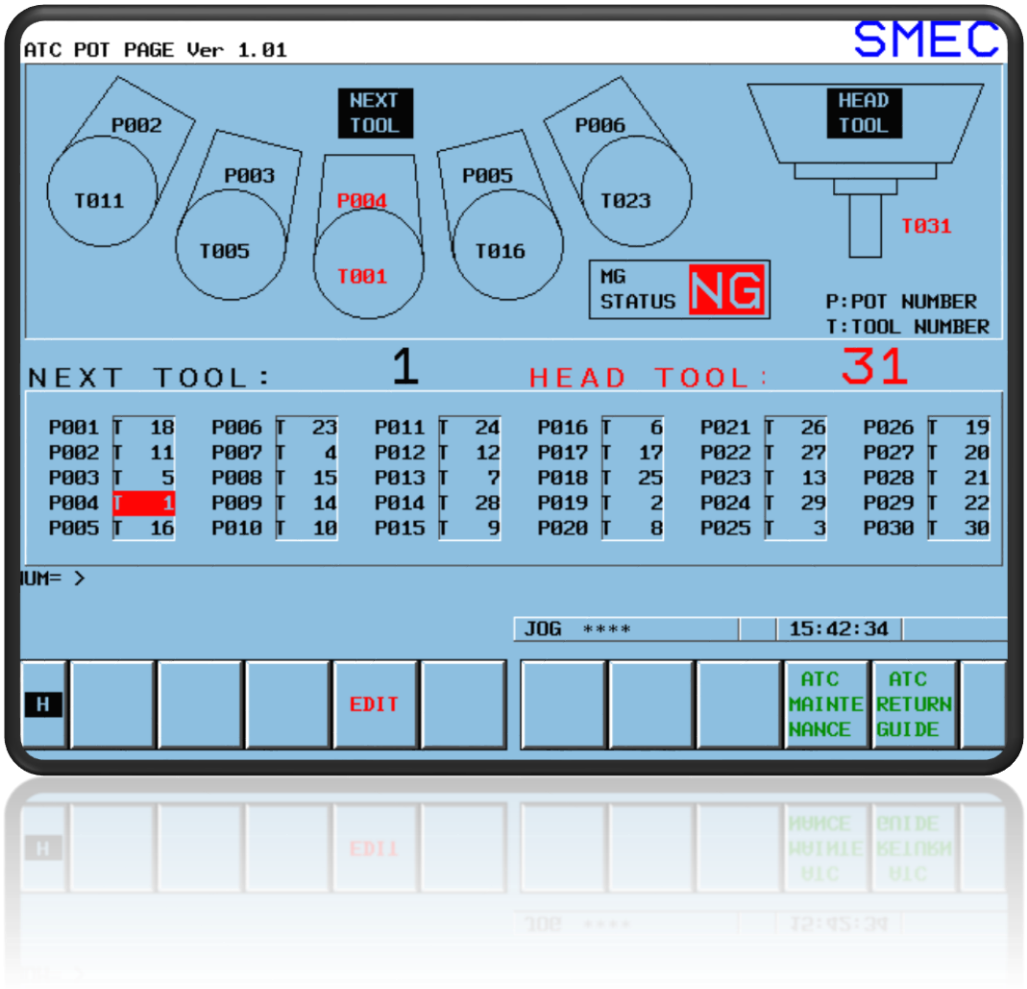
Alarm Detail



Alarm Detail List

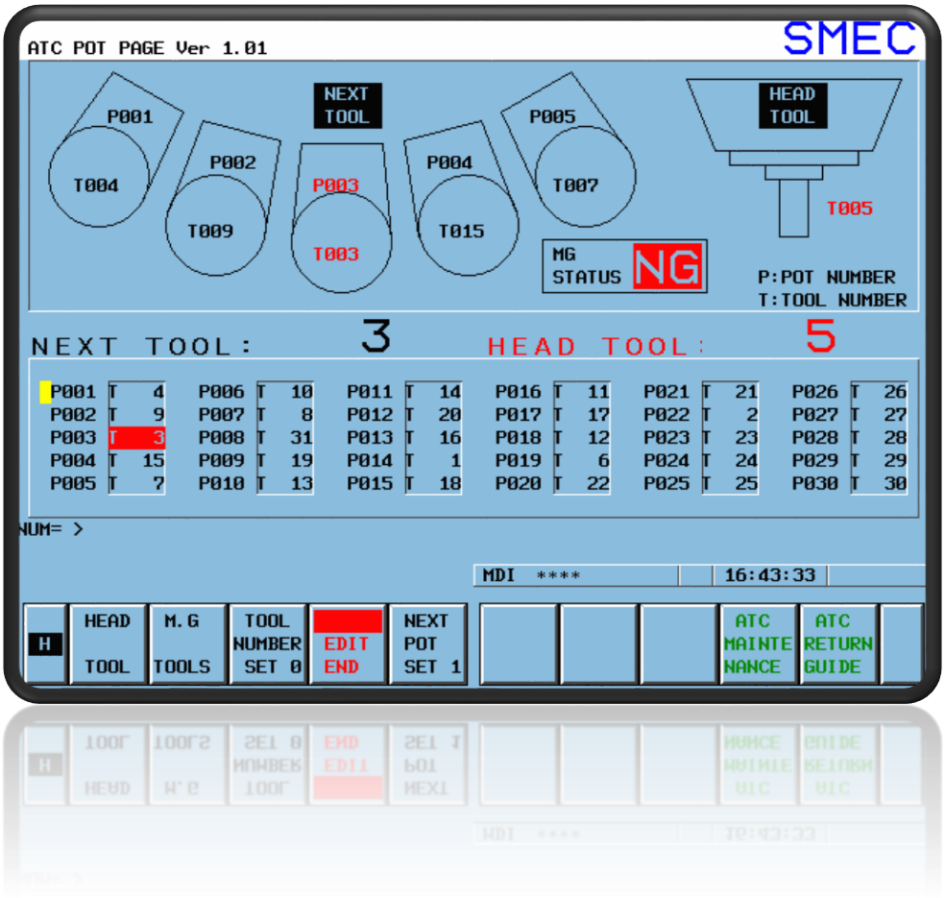
- Under the Custom 1 Button, pressing the Alarm Detail soft key will display a list of PMC addresses and messages including detail of any alarms.

ATC Managment



ATC Management Function

- Allows viewing and editing of the ATC Tools and their Pots.



ATC Management Function Use

- Head Tool: Switches the cursor to the spindle tool selection.
- MG Tool: Switches the cursor to the ATC Pot selection.
- Tool Number Set 0: Sets the highlighted POT to Tool 0.
- EDIT End: Ends the ATC Pot Editing.
- NEXT Pot Set 1: CAUTION, Sets all pots in sequential order!

SPINDLE WARM UP

For the long-standing use of the spindle use of the spindle bearing, be sure to execute a spindle warm-up operation program before high rpm operating the machine. A good rule of thumb for spindle warm up: if running under half the max rpm of the spindle, no warm up is needed. If running a considerable time over half the spindles max rpm, a warm up program should be run.

6,000 RPM SPINDLE (12 MINUTE WARM UP)

```
S1500
G4X180.
S3000
G4X180.
S4500
G4X180.
S6000
G4X180.
M30
```

12,000 RPM SPINDLE (12 MINUTE WARM UP)

```
S1500
G4X90.
S3000
G4X90.
S4500
G4X90.
S6000
G4X90.
S7500
G4X90.
S9000
G4X90.
S10500
G4X90.
S12000
G4X90.
M30
```

COMMUNICATIONS

For the following instructions be sure to set the correct I/O channel for your chosen INPUT/OUPUT device:

I/O Channel Settings;

- RS232 = 0
- Memory Card = 4
- USB = 17
- Ethernet = 9
- Data Server = 5

Changing the I/O Channel;

- Press the **OFS/SET** hard key.
- Press the **SETTING** soft key.
- Press the **MDI** mode key.
- Cursor to the **I/O** channel field (middle of the screen).
- Type in the number of your device and press **INPUT**.

To INPUT a program from the FOLDER screen;

- Put the machine into **EDIT** mode.
- Press the **PROGRAM** hard key.
- Press the **FOLDER** soft key.
- Enter the letter **O** followed by the program number. Example, **O1111**.
- Press the **F INPUT** soft key then press the **EXEC** soft key.

To OUTPUT a program from the FOLDER screen;

- Put the machine into **EDIT** mode.
- Press **PROGRAM** hard key.
- Press the **FOLDER** soft key.
- Enter the letter **O** followed by the program number you wish to output. Example, **O1111**.
- Press the **F OUTPUT** soft key followed by the **EXEC** soft key.
- The program will now be transferred to the external device.

To INPUT a program from the PROGRAM screen;

- Put the machine into **EDIT** mode.
- Press the **program** hard key.
- Press the **program** soft key.
- Press **OPRT**.
- Press the **+** soft key until you see F Input.
- Key in **O** and the program number. Example, **O1111**.
- Press the **EXEC** soft key.

To OUTPUT a program from the PROGRAM screen;

- Put the machine into **EDIT** mode.
- Press the **program** hard key.
- Press the **program** soft key.
- Press OPRT
- Press the **+** soft key until you see F Output.
- Key in **O** and the program number. Example, **O1111**.
- Press the **EXEC** soft key.

To INPUT a program from the EXTERNAL DEVICE screen;

- Put the machine into **EDIT** mode.
- Press the **PROGRAM** hard key.
- Press the **FOLDER** soft key.
- Press the **OPRT** soft key.
- Press the **DEVICE CHANGE** soft key.
- Select the device your **I/O** channel is set to (if PMCIA card, select MEMORY CARD).
- Your external device will now be displayed.
- Key in the program number you wish to input. Example **O1111**.
- Press the **F INPUT** soft key.
- Press the **EXEC** soft key.

To OUTPUT a program from the EXTERNAL DEVICE screen;

- Put the machine into **EDIT** mode.
- Press the **PROGRAM** hard key.
- Press the **FOLDER** soft key.
- Press the **OPRT** soft key.
- Press the **DEVICE CHANGE** soft key.
- Select the device your **I/O** channel is set to (if PMCIA card, select MEMORY CARD).
- Your external device will now be displayed.
- Key in the program number you wish to input. Example **O1111**.
- Press the **F OUTPUT** soft key.
- Press the **EXEC** soft key

To INPUT a program with file extension such as .NC or .TXT from the EXTERNAL DEVICE screen;

- Put the machine into **EDIT** mode.
- Press the **PROGRAM** hard key.
- Press the **FOLDER** soft key.
- Press the **OPRT** soft key.
- Press the **DEVICE CHANGE** soft key.
- Select the device your **I/O** channel is set to (if PMCIA card, select MEMORY CARD).
- Your external device will now be displayed.
- Cursor **UP/DOWN** to the program you wish to input.
- Press the **+** soft key once.
- Press the **F INPUT** soft key.
- Press the **F GET** soft key.
- Press the **F SET** soft key.
- Press the **EXEC** soft key.

To DNC (Drip Feed) a program with from a MEMORY CARD or ETHERNET connection;

- Put the machine into **EDIT** mode.
- Press the **PROGRAM** hard key.
- Press the **FOLDER** soft key.
- Press the **OPRT** soft key.
- Press the **DEVICE CHANGE** soft key.
- Select the device your **I/O** channel is set to (if PMCIA card, select MEMORY CARD).
- Your external device will now be displayed.
- Cursor **UP/DOWN** to the program you wish to DNC.
- Put the machine into **MDI** mode.
- Press the **DNC** Set soft key.
- Put the machine into **RMT** mode.
- Press the **PROGRAM** hard key.
- Press **CYCLE START**.
- Program will now be displayed as it runs.
- For Drip Feed via Ethernet be sure parameter 14885.1=1 is set for the function to work.
- During DNC editing of the program, or starting mid program will not be possible.

To Drip Feed a program from a MEMORY CARD via an EXTERNAL PROGRAM CALL;

- Make sure parameter **138.7=1**.
- Make sure parameter **3404.2=1**.
- Call the program with the code **M198 P1111** for example, to call program **O1111**.
- An **M198** can be done via MDI, or within a program. It can call an entire program, or sub programs of perhaps high speed machining code.
- During Drip Feeding Editing of the program, or starting mid program will not be possible.

Ethernet settings;

- To use the Ethernet Function on Oi-F control requires the use of an FTP Server on your PC. A free option we suggest is FileZilla.
- Be in **MDI** mode.
- Press the **OFS/SET** hard key.
- Press the **SETTING** soft key.
- Cursor to **PARAMETER WRITE ENABLE** and type in a **1**, then press the **INPUT** hard key.
- Press the **SYSTEM** hard key.
- Press the right arrow soft key until the **EMBED PORT** soft key is displayed.
- Press the **COMMON** soft key.
- In the **IP ADDRESS** field type in the ip address of the machine. Example **192.168.100.231**.
- Cursor down to the **SUBNET MASK** and enter **255.255.255.0**.
- **ROUTER IP ADDRESS** leave blank.
- Press the **FTP TRANSFER** soft key.
- Enter the ip address of the computer.
- Cursor to **PORT NUMBER** and enter **21**.
- Enter a name in the **USERNAME** field. Typical setting is the machine model.
- Enter a name in the **PASSWORD** field. Typical setting is the machine model again.
- Press the page down hard key. In the **login directory** field press the **/** key then press hard key **input**. Power off and on the control. Test the operation.

Note- It is the customer's responsibility to properly setup and connect to the FTP ServerDealer assistance can be provided, however correct IP addresses are required***

Outputting the parameters on Oi-F control:

- Make sure the I/O channel is set to the correct communication channel.
- Select the machine **EDIT** mode.
- Press the **SYSTEM** hard key.
- Press the **PARAM** soft key.
- Press the **OPRT** soft key.
- Press the **+** soft key.
- Press the **F OUTPUT** soft key.
- Press the **ALL** soft key.
- Press the **EXEC** soft key.

Parameter Write Enable Procedure (PWE):

- Press the **OFS/SET** hard key.
- Press the **SETTING** soft key.
- Press the **MDI** mode key.
- Cursor to the **PARAMETER WRITE** field (top of the screen).
- Type in the number **1** and press **INPUT**. Alarm message 100 will be active until this is turned off again.

NOTES

NOTES

NOTES

SAMSUNG Machine Tools

The name you know, the technology you trust

www.samsungmachinetools.com

N25 W23287 Paul Rd
Pewaukee, WI 53149
262-521-1100

VER. 2015.05.26