SIEMENS SINUMERIK 840D CONTROL

OPERATOR MANUAL

FADAL MACHINING CENTERS, LLC

Corporate Office.....phone (818) 407-1400..... fax (818) 407-0020
Service / Parts.....phone (818) 727-2100..... fax (818) 407-1004
Programming Support.....phone (818) 727-2100..... fax (818) 407-0061

support@fadal.com
20701 Plummer Street, Chatsworth, California 91311 USA

AUGUST 2005

TABLE OF CONTENTS

1.0 POWER ON /OFF

1.1 PRE-START CHECKING STEPS

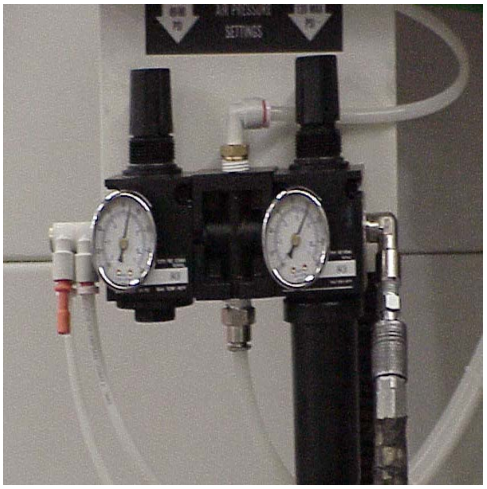
1.1.1 OIL RESERVOIR

Examine the oil levels. Both should be filled before the levels are one inch from the bottom of the reservoir. The spindle oil reservoir may have oil in it for up to six months. The way lube oil reservoir may run out of oil in one week.

NOTE
VMCs with linear way systems require grease. See the Maintenance Manual for specifications on the way lube and the spindle oil.

1.1.2 AIR PRESSURE

Visually inspect the air pressure gauge to verify that it is set to at least 80-100 PSI. Air is used to change belt ranges in the spindle, orient the spindle, activate the tool in-out cylinder, and for the air blast during a tool change. The tool changer gauge should not exceed 120 PSI.



1.1.3 WATER RESERVOIR

Most new VMC models release water collected in the water reservoir automatically. It is advisable to place an additional water trap in the air line going to the machine.

1.1.4 FLOOD COOLANT

Replenish the flood coolant level to avoid running out of coolant during execution of the program.

1.1.5 SPINDLE COOLER RESERVOIR

Examine the spindle cooler reservoir once a month.

1.2 POWER ON/OFF

1.2.1 POWER ON

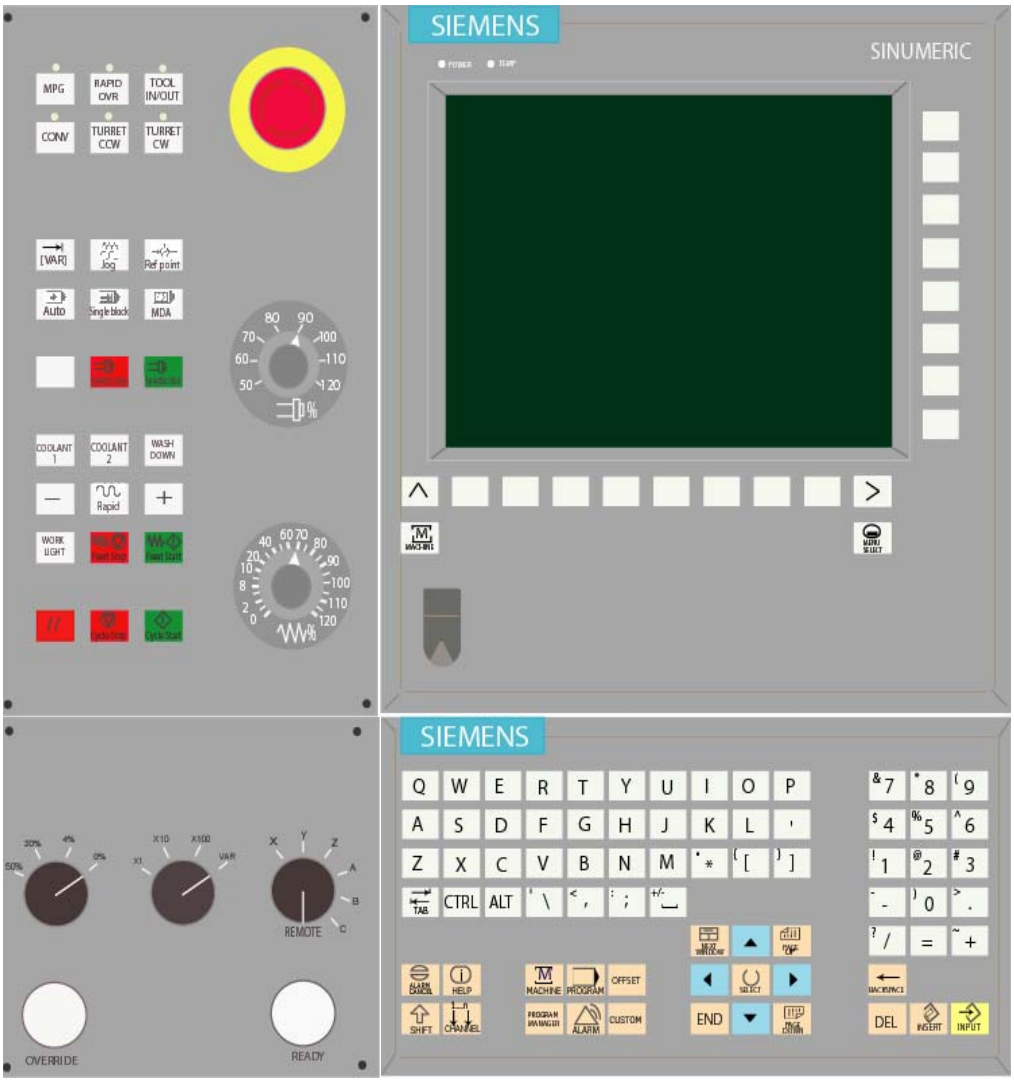
To power on the machine, press the safety lock and turn the power switch in the clockwise direction. CNC will boot up and enter operating status.

1.2.2 POWER OFF

To power off machine follow the next procedure:

1. From the MDA mode, type SETCS.
2. Press the CYCLE START hard key.
3. Leave the machine at this display.
4. Press the E-stop button.
5. To power off the machine, turn the power switch counter clockwise.

2.0 PENDANT LAYOUT / HHU



The table on the following pages shows the various buttons on the pendant. A brief description of the function of each button is given to assist the operator in becoming familiar with the control. Refer to the Siemens operator manuals for detailed instructions on how to use these buttons.

2.1. PENDANT HARD KEYS FUNCTION GUIDE

2.1.1 PENDANT KEYBOARD

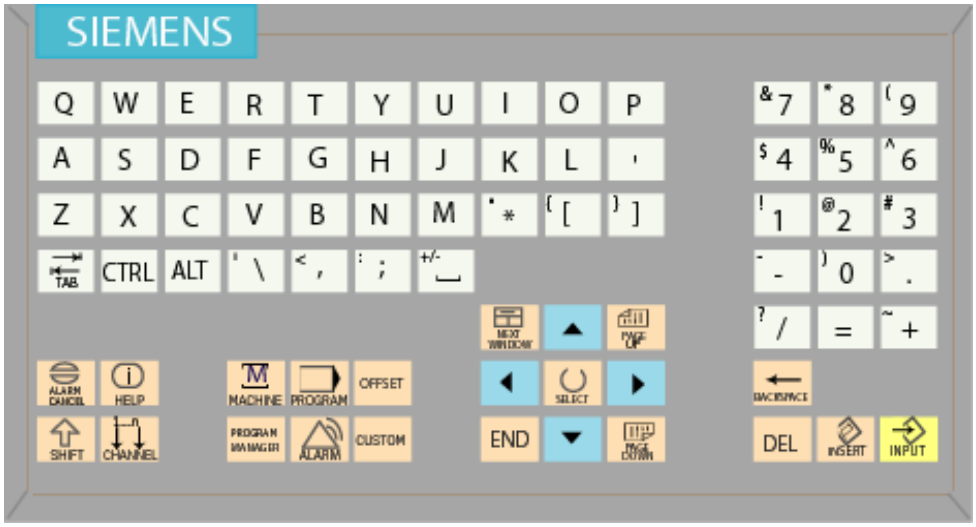


Table 2-1: Pendant Keyboard Keys

KEY	NAME	FUNCTION
	*Alarm*	Displays alarms/messages screen.
	*Alarm Acknowledge*	Resets CNC soft alarms.
	*Cursor (left, right, up, down)*	Moves the cursor around the display screen.
	*End*	Moves the cursor to the end of the program.
	*Help / Information*	Toggles between test and graphic displays in Shop Mill. Active when displayed on lower line of display.

Table 2-1: (Continued) Pendant Keyboard Keys

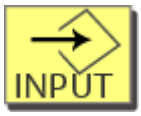
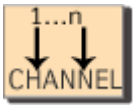
KEY	NAME	FUNCTION
	*Input*	Used to accept entry of data.
	*Insert*	Used to edit the existing data entry.
	*Next Window*	Goes to the top of the next active display.
	*Offset/Parameters*	Displays the offsets screen and offset soft keys menu.
	*Page Down*	Page down the screen display.
	*Page Up*	Page up the screen display.
	*Position*	Displays the position screen and the main soft key menu of the active mode (Manual / Auto).
	*Program Manager*	Displays the program manager screen.
	*Program*	Displays the program edit screen.
	*Select*	Allows to toggle between values.

Table 2-1: (Continued) Pendant Keyboard Keys

KEY	NAME	FUNCTION
	*Shift*	Allows to use second character.
	*Backspace*	Allows to move the cursor back on space.
	*Delete*	Deletes character.
	*Channel*	This button is not functional.
	*Custom*	This button is not functional.

NOTE
CHANNEL AND CUSTOM keys are not usable.

2.1.2 UPPER MACHINE CONTROL PANEL (MCP)

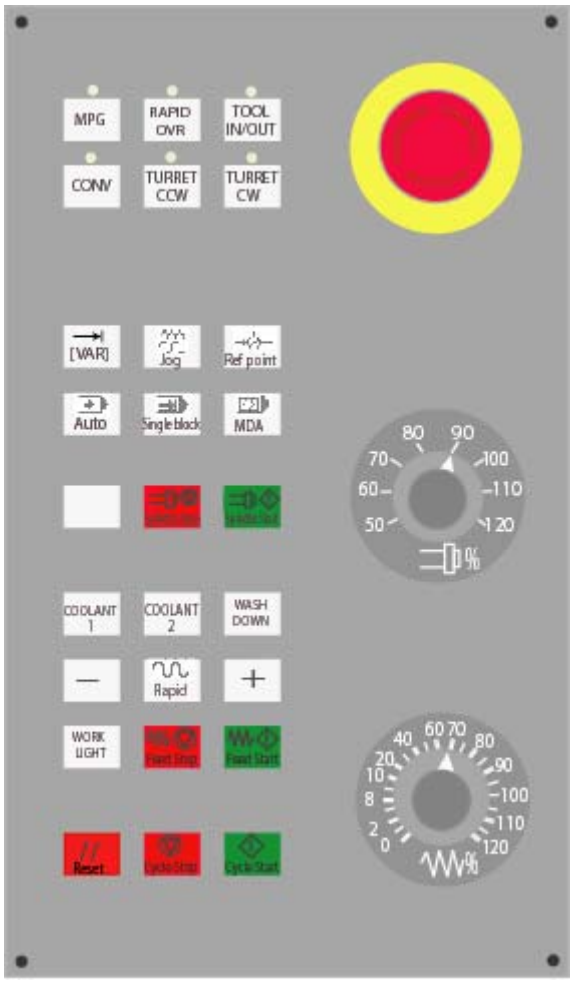


Table 2-2: Machine Control Panel Keys

KEY	NAME	FUNCTION
	*Auto*	Initiates the automatic mode.
	*Coolant 1*	Toggles the flood coolant feature on and off. Works in combination with M7 (to turn it on) and M9 (to turn it off) codes.
	*Coolant 2*	Toggles the mist coolant feature on and off. Works in combination with M8(to turn it on) and M9 (to turn it off) codes.

Table 2-2: (Continued) Machine Control Panel Keys

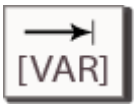
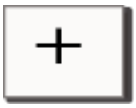
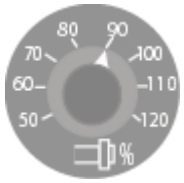
KEY	NAME	FUNCTION
	*Increment Jog*	Activates Incremental sub-mode of Manual Mode. Allows incremental JOG by JOG keys or MPG (hand wheel).
	*Jog*	Initiates Manual Mode. Note: Reference, Incremental Jog and MDA are all sub-modes of the Manual Mode.
	*MDA*	Use to access Manual Data Automatic mode in ISO. Not active in Shop Mill. In Shop Mill MDA is accessed via a soft key in the Manual Mode.
	*Minus*	Use for jogging in the negative direction.
	*Manual Pulse Generator*	Activates the MPG mode of the manual screen. The LED above the button will be lit when the MPG mode is active.
	*Plus*	Use for jogging in the positive direction.
	*Rapid Jog*	Use in conjunction with the JOG “+” and “-” buttons. Activates the rapid JOG feed rate.
	*Reference*	Activates the Reference submode of the Manual Mode.
	*Tool In/Out*	Activates the drawbar to manually load or unload a tool from the spindle. This button is active only in the Manual Mode.
	*Turret CCW*	Rotates the turret in a counterclockwise direction. This button is active only in the Manual Mode.

Table 2-2: (Continued) Machine Control Panel Keys

KEY	NAME	FUNCTION
	*Turret CW*	Rotates the turret in the clockwise direction. This button is active only in the Manual Mode.
	*Work Light*	Toggles the machine work lights ON and OFF.
	*Conveyor*	Allows to toggle the conveyor ON and OFF.
	*Reset*	Resets the currently active program and some alarms. Note: Using the reset button will reset the program to its beginning.
	*Spindle On*	Turns the spindle on in the last programmed direction and RPM, (S-word). Note that on early production models the text on the key reads “SPINDLE RIGHT”.
	*Spindle Stop*	Stops the spindle.
	*NC Cycle Start*	Starts the execution of the CNC in Auto Mode or operation in Manual Mode.
	*NC Cycle Stop*	Stops the execution of CNC program, control waits for NC Cycle Start signal or Reset signal
	*Feed Start*	Feed start (Slide Start).
	*Feed Stop*	Stop feed (Slide Hold)

Table 2-2: (Continued) Machine Control Panel Keys

KEY	NAME	FUNCTION
	*Emergency Stop*	Emergency Push button will cut the power to all axis motors, spindle drives and the tool changer. To cancel Emergency Stop first press READY pushbutton, then Reset hard key. The program will start from the beginning.
	*Spindle Override (RPM)*	Allows to override programmed spindle speed.
	*Feed rate override*	Allows to override programmed feed rate speed. When feed rate override switch is pointing 0 motion will stop.

2.1.3 LOWER MACHINE CONTROL PANEL (MCP)

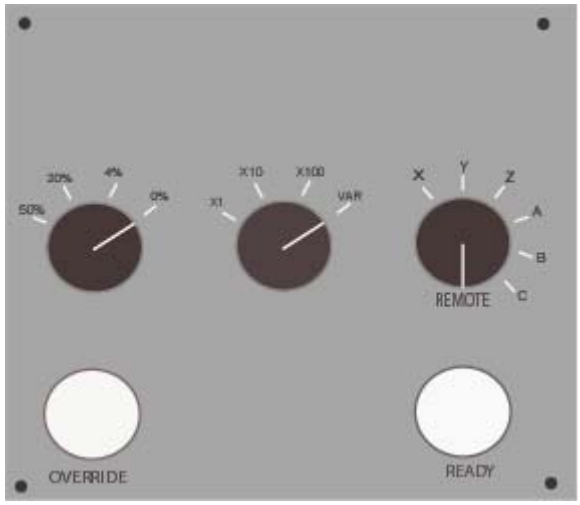
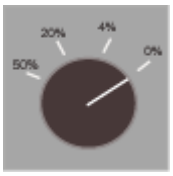
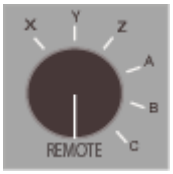


Table 2-3: Machine Control Panel Keys

KEY	NAME	FUNCTION
	*Rapid Rate Override*	Allows to override programmed rapid rate. (G0)
	*Resolution*	Allows to select increment.
	*Axis Selector*	To select the axis, use Axis Selector switch.
	*Override*	Override push button is used to override the door interlock when using MPG.
	*Machine Safety Circuits.*	Resets external safety circuits for door and an emergency stop.
	*Door Interlock*	Unlocks front door when lit. (CE machines only)

2.2. HAND HELD UNIT (HHU)

Front View

Back View

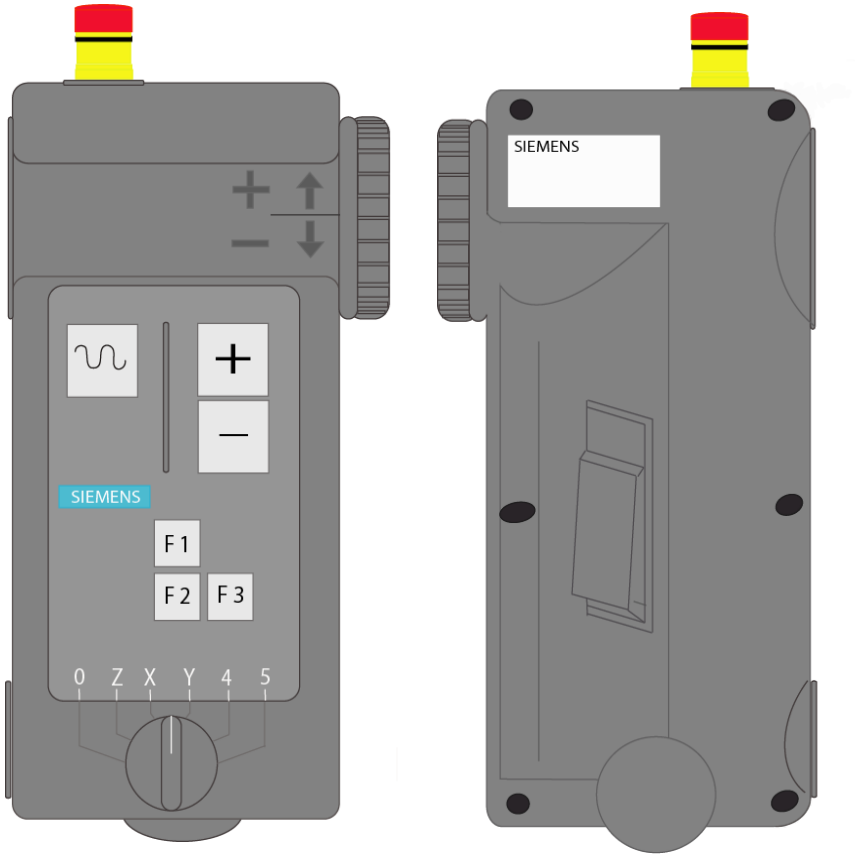


Table 2-4: Hand Held Unit hard keys

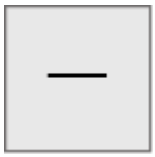
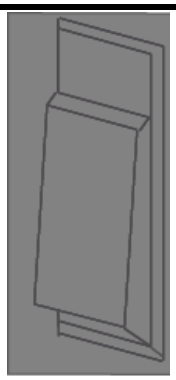
KEY	NAME	FUNCTION
	*Rapid*	Allows to select rapid jog axis when pressed together with + or -.
	*Position direction*	Allows to jog in the positive direction.
	*Negative direction*	Allows to jog axis in the positive direction.

Table 2-4: (Continued) Hand Held Unit hard keys

KEY	NAME	FUNCTION
	*Tool IN/OUT*	To manually load or unload a tool into the spindle press F1 key.
	*Jog/MPG*	To switch from Jog mode into MPG and vice verse, press F2 key on the HHU. To check if MPG is active, check LED above MPG hard key on the MCP panel.
	*Resolution*	To change resolution (X1, X10, X100 or VAR) press F3 key as many times as needed.
	*Axis Selector*	To select 0, X, Z, Y, 4, or 5 axis, switch axis selector to the right position.
	*Emergency Stop*	Emergency Push button will cut the power to all axis motors, spindle drives and the tool changer. To cancel Emergency Stop first press READY pushbutton then Reset hard key. The program will start from the beginning.

Table 2-4: (Continued) Hand Held Unit hard keys

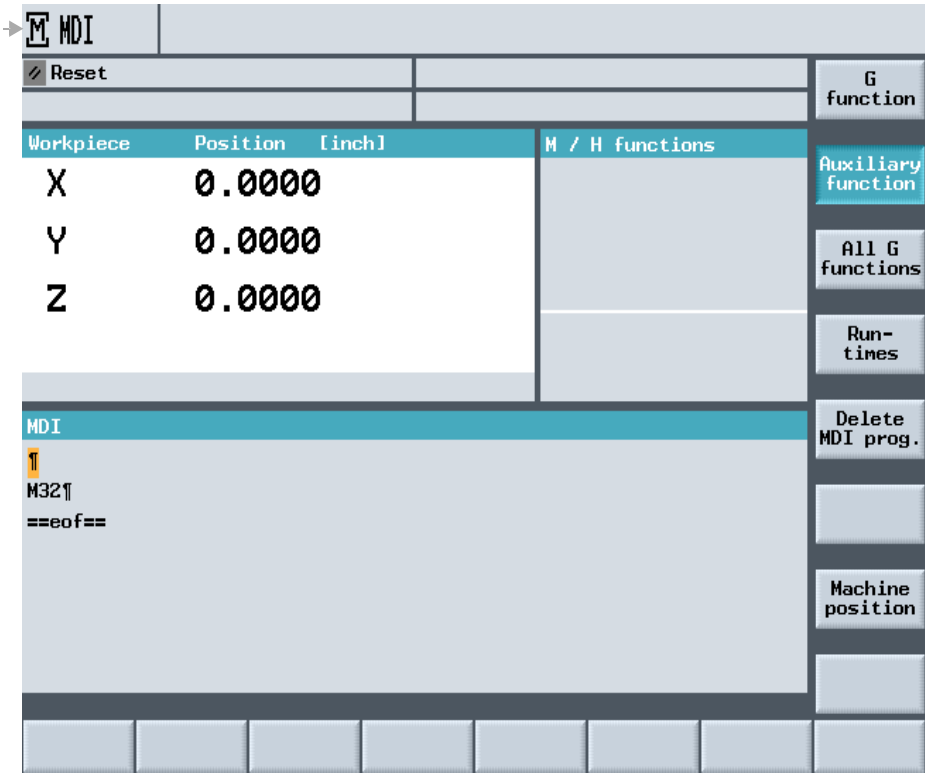
KEY	NAME	FUNCTION
	*Override*	Override push button is used to override the door interlock when using the MPG. (Exception, if the feedrate is at 0% then MPG will stop the motion) It allows to jog machine with open doors. Override push button can be either in active or inactive position: active (when push button is gently pressed), inactive (when push button is pressed harder or not pressed at all). Every time, when Override push button is in active position, operator can jog machine. Once Override push buttom is in inactive position, operator will not be able to jog machine. Machine will stop movement.
	*MPG (Manual Pulse Generator)*	To activate MPG, press MPG hard key on the MCP panel. The LED of MPG hard key will turn on. To move the tool in a plus direction, turn the manual pulse generator in the clockwise direction. To move the tool in a minus direction, turn the manual pulse generator in the counter-clockwise direction.

3.0 MANUAL OPERATION

3.1 MANUAL DATA AUTOMATIC (MDA)

To display MDI screen follow next steps:

- Press the MDA hard key.

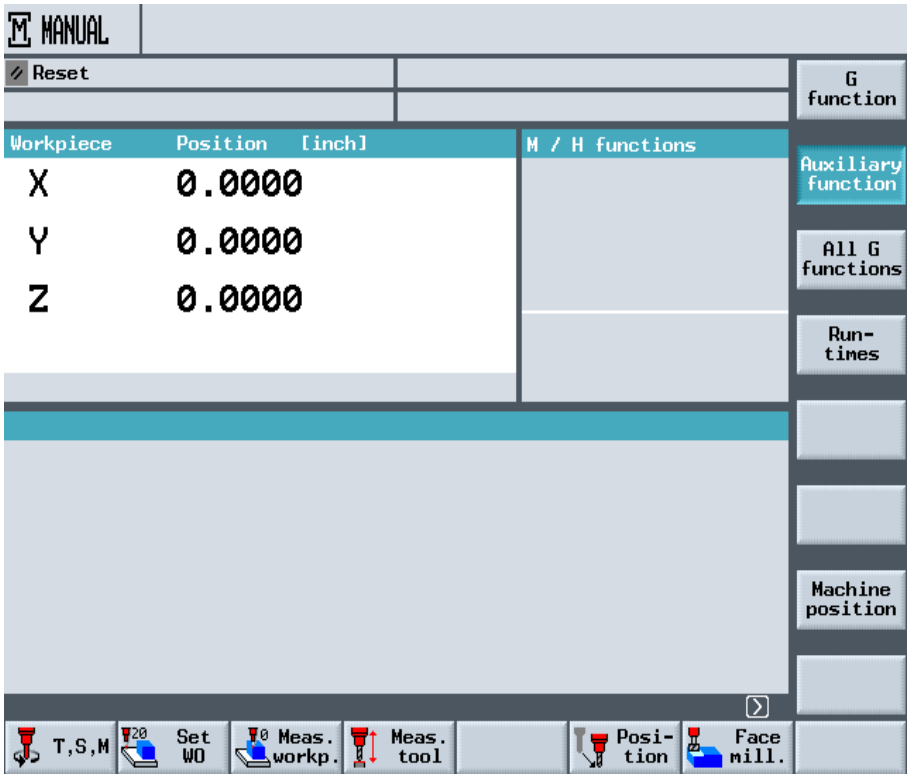


MDI display

3.2 JOG MODE

To display Jog mode position readout display follow the next step:

- Press the JOG hard key.



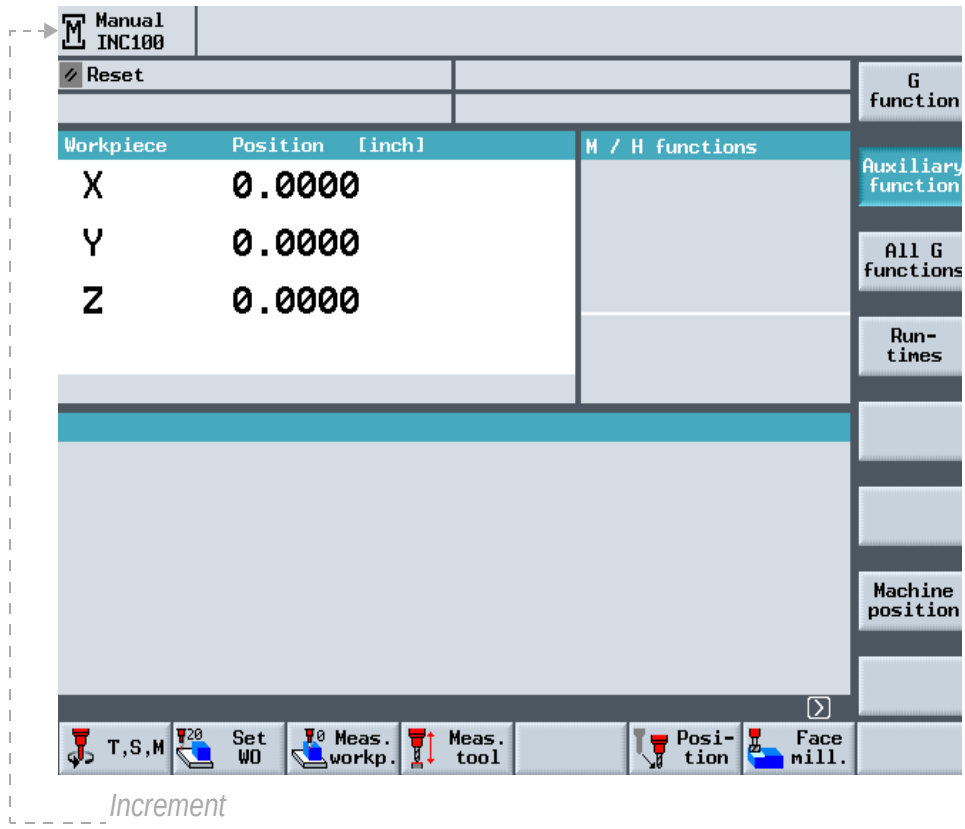
3.2.1 SETTING INCREMENT

Incremental jog using JOG hard key

1. Press JOG hard key.
2. Press VAR hard key.

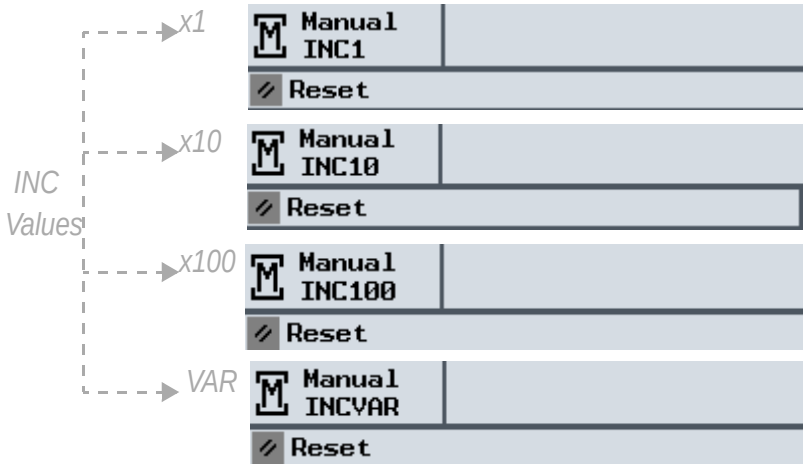
(On the upper left corner of the display we get INC...)

3. Select increment using increment selector switch to toggle between x1, x10, x100 and VAR.
4. Indicate direction pressing +/- hard key



Incremental jog using HHU

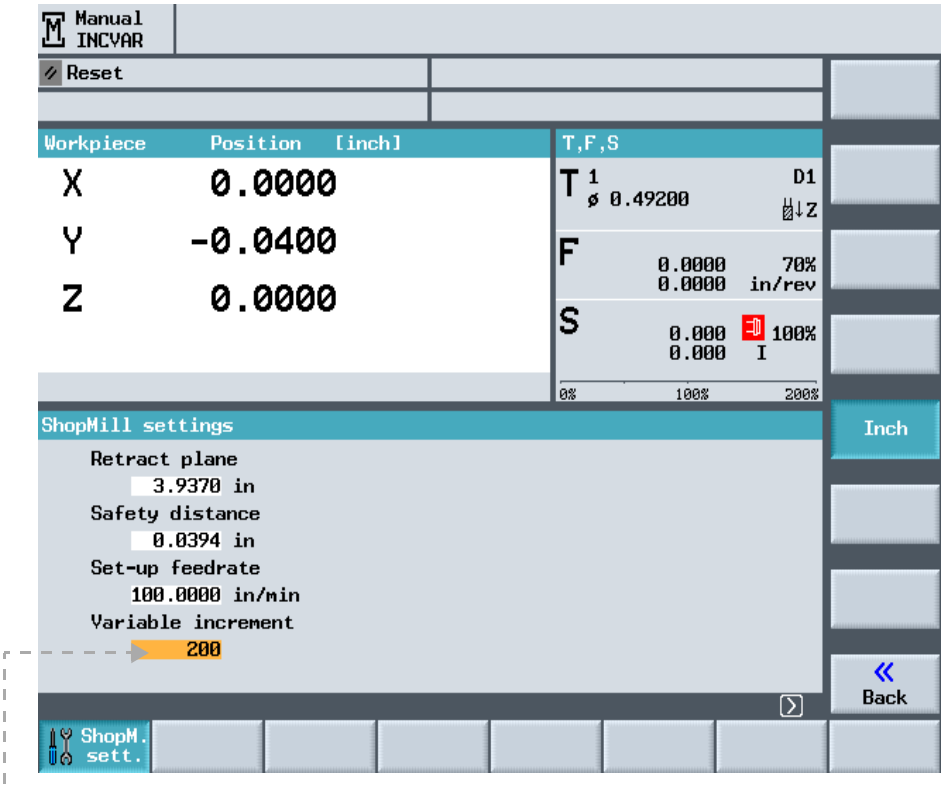
1. Select REMOTE on axis selector switch that is located on the lower MCP panel.
2. Press JOG hard key
3. Press F2 on HHU (LED above MPG should turn on)
4. Press F3 to toggle increment between x1, x10, x100 and VAR.
5. Indicate direction pressing +/- hard key.



If axis selector is set on VAR, set jog increment value manually following the next steps:

1. Press menu extention key “>”.
2. Press ShopM. Sett. soft key
3. With the blue cursor keypad arrow, cursor down to the variable increment box.
4. Type in the new value (between 0 - 200INC) and press INPUT hard key.

NOTE
1INC = 0.0001inch
1INC = 0.001mm



3.2.2 RAPID JOG

1. Press the JOG hard key and select the axis to be jogged from the axis selector switch located on the lower MCP panel.

2. Select the direction of the tool by pressing the “+” or “–” hard key.

3. Pressing the RAPID hard key together with the “+” or “-” buttons activates the Rapid Jog feed rate.

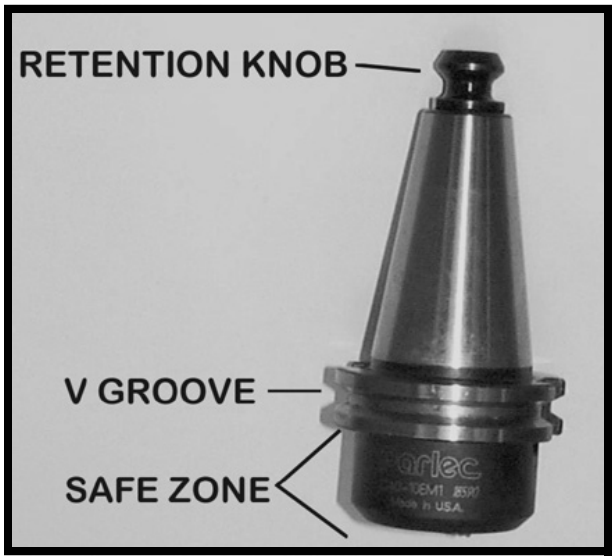
4. With both JOG and RAPID hard keys user has the option to vary the feed rate override to control the speed.

3.3 TOOL OPERATION

3.3.1 MANUAL TOOL LOADING AND UNLOADING

A tool can be manually loaded or unloaded into the spindle by using the TOOL IN/OUT hard key.

1. The tool holder must be held in the left hand with the thumb and the first finger grasping the holder below the “V” groove. No other fingers should have contact with the holder or the tool in the holder. The area below the “V” groove is called the safe zone. The safe zone is the only place where the tool holder should be held.



2. When unloading a tool from the spindle, grasp the tool in the safe zone and press the TOOL IN/OUT hard key. Keep the TOOL IN/OUT hard key pressed until the tool is completely out of the spindle.

When loading a tool into the spindle, grasp the tool in the safe zone and press the TOOL IN/OUT hard key. Place the holder into the spindle after pressing the TOOL IN/OUT hard key, not before. The keys on the nose of the spindle must fit into the key-ways on the tool holder flange. Release the TOOL IN/OUT hard key to lock the tool into the spindle.

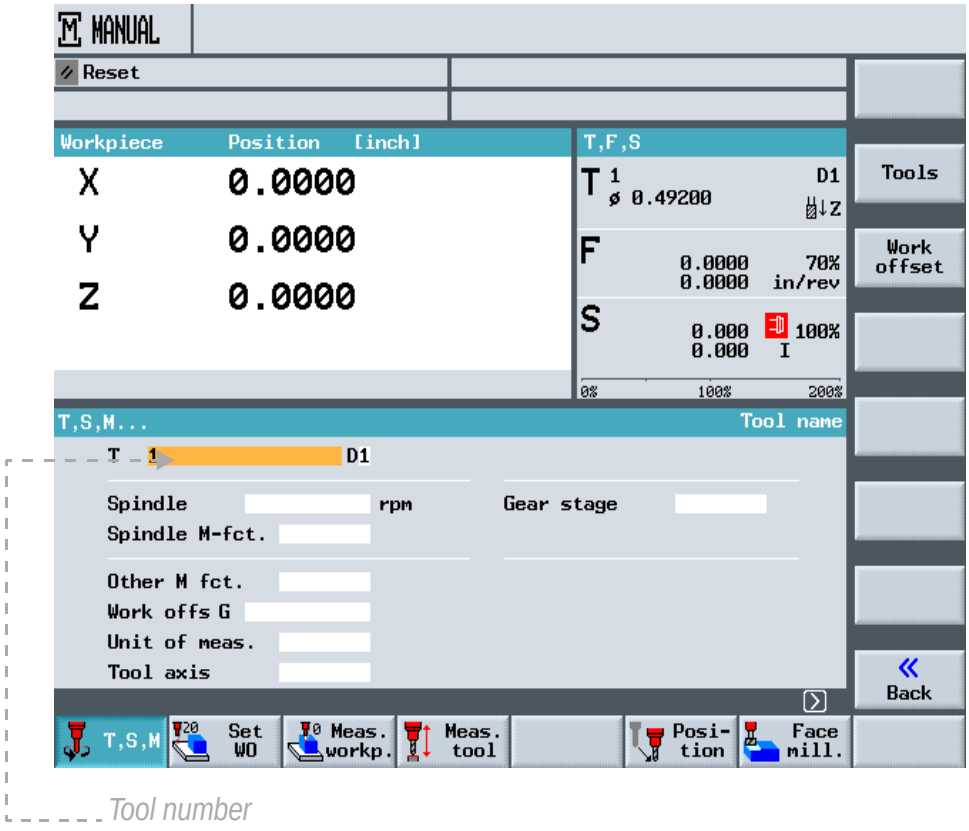
> *NOTE*
> When loading a holder into the spindle, inspect the taper for chips and dents. Remove any chips or dents from the taper with a flat stone.

3.3.2 LOADING AND UNLOADING A TOOL FROM TOOL CHANGER (ATC)

NOTE
When tool holders are in the ATC (Automatic Tool Changer), they can be loaded into the spindle by using the T,S,M mode.

1. Press the T,S,M soft key.
2. Type T# (where # is the turret location of the tool to be loaded into the spindle).
3. Press the INPUT hard key.
4. Press the CYCLE START hard key to make the exchange.

NOTE
If the feed rate potentiometer is set to 0%, no motion will occur when the START button is pressed.



CAUTION
If the tool change is interrupted, the selected tool becomes a null value and the machine will try to load a tool. If this occurs, remove the tool from the spindle and repeat the operation.

M11 can be used to set the current magazine location as T1. Note that this does not change the active Tool number.

3.4 MANUALLY JOGGING THE DATC

NOTE
Machine must be in JOG mode. Multiple key combinations are required to move the dual tool arm changer.

- Press and hold the SPINDLE STOP *(d)*, FEED STOP *(e)*, and CYCLE STOP *(f)* red hard keys and the RAPID OVR *(a)* --located between the MPG and the TOOL IN/OUT hard keys.

The bucket will motion either up or down depending on its previous position.

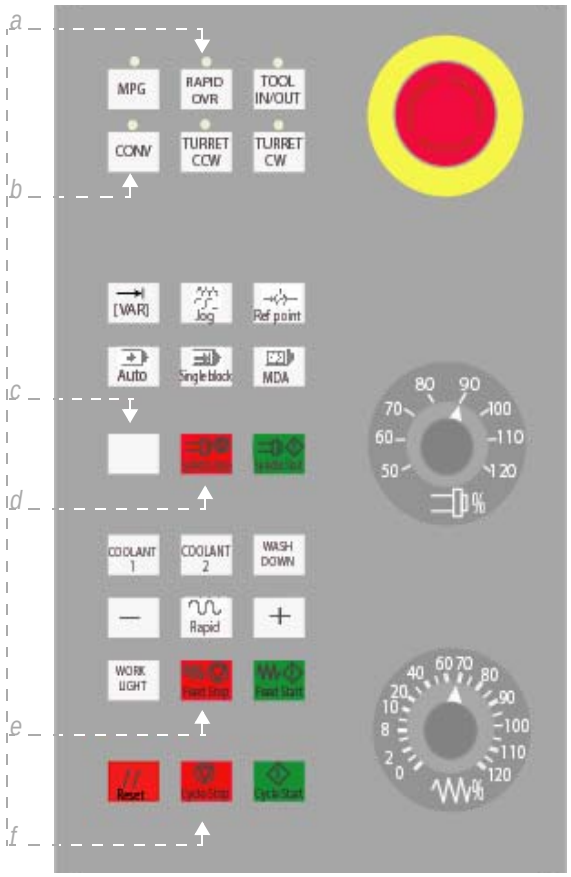
- Press and hold the SPINDLE STOP *(d)*, FEED STOP *(e)*, and CYCLE STOP *(f)* red hard keys and the CONV *(b)* --located to the left of the TURRET CCW hard key.

The arm will motion forward--depending on its previous position.

- Press and hold the SPINDLE STOP *(d)*, FEED STOP *(e)*, and CYCLE STOP *(f)* red hard keys and the blank key *(c)*--located to the left of the red SPINDLE STOP hard key.

The arm will motion backwards--depending on its previous position.

NOTE
The arm will only move if the Z-axis is at its home position.



3.5 SPINDLE OPERATION

3.5.1 ESTABLISHING SPINDLE RPM

NOTE
Use the spindle OVERRIDE switch to vary the RPM manually from 50% to 120% of the programmed value.

M3 = CW SPINDLE ON

M4 = CCW SPINDLE ON

M5 = SPINDLE OFF

M40 = AUTO BELT RANGE

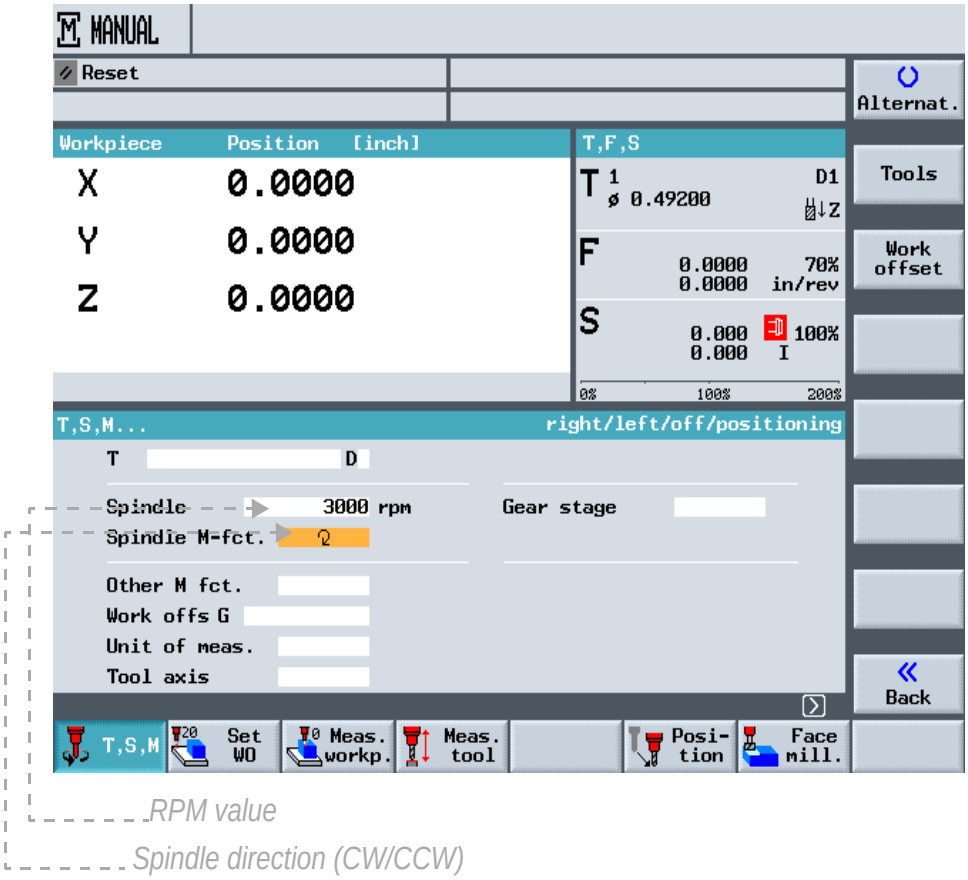
M41 = LOW BELT RANGE

M42 = HIGH BELT RANGE

CAUTION
The last RPM setting used by the control is active when the spindle is turned on manually unless it is changed in the T,S,M mode.

The spindle RPM can be started at the desired RPM and Rotational direction by using the T,S,M soft key.

1. Press the JOG hard key.
2. Press the T,S,M soft key.
3. With the blue cursor keypad arrows, cursor down to RPM box, type in the new value and press the INPUT hard key.
4. With the blue cursor keypad arrows, cursor down to the spindle M-fct box. Use the Alternat. soft key to toggle to CW and CCW direction.
5. Press the CYCLE START hard key to activate.



3.5.2 SPINDLE START

After establishing the RPM from T,S,M Mode, or at any time in any mode of operation, the spindle can be turned on manually by using the green SPINDLE START hard key button.

- Press the SPINDLE START hard key. The spindle will turn on in the last programmed direction and spindle speed (RPM).

3.5.3 SPINDLE OFF

When the spindle is on, from any mode of operation, press the red SPINDLE OFF button alone, to turn the spindle off.

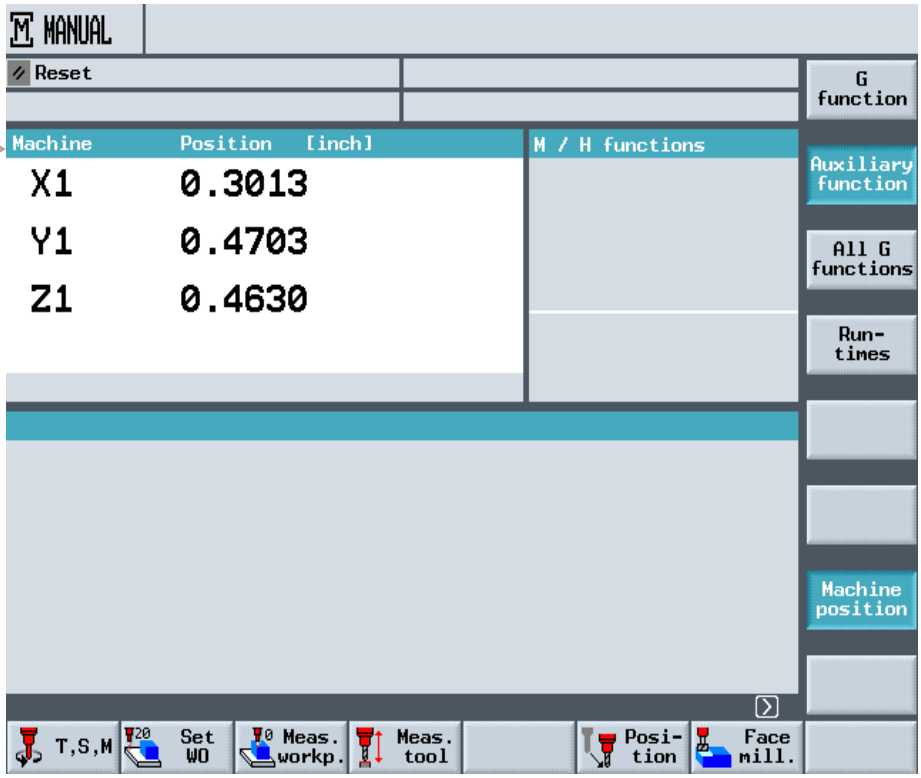
The RESET button will also turn the spindle off.

4.0 OFFSETS

4.1 COORDINATE SYSTEMS

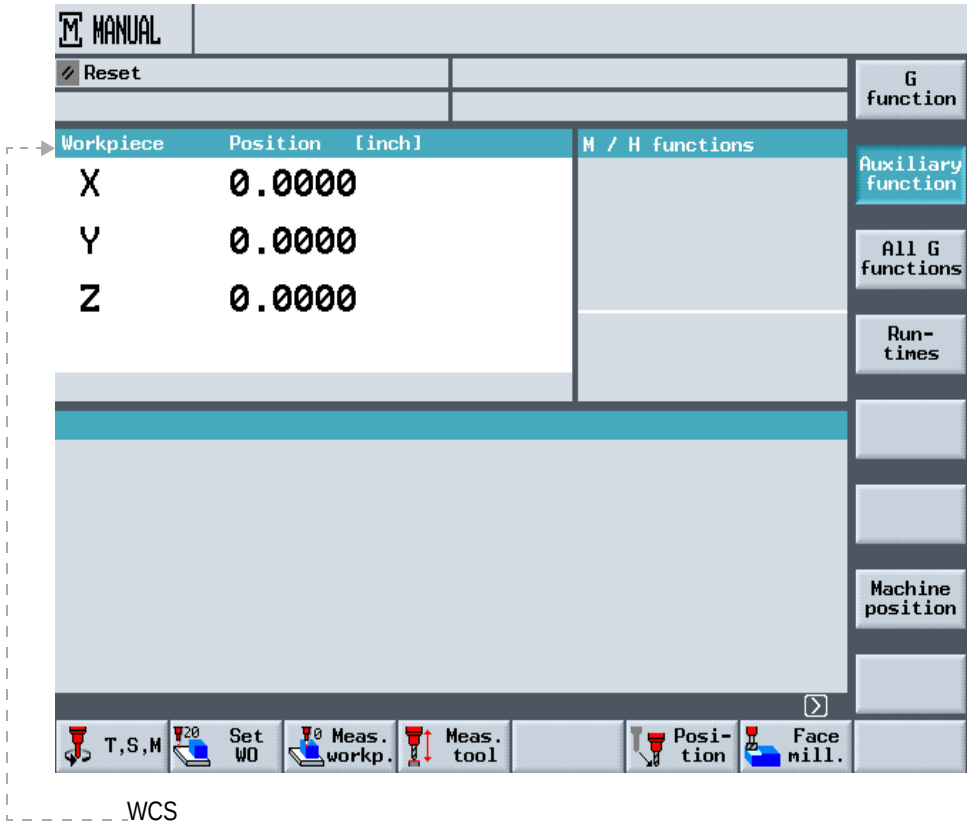
Machine coordinate system (MCS)

The machine coordinate system is based on the machine axes that are designated X1, Y1, Z1, A1 and B1 in the actual value display. At the reference point (alignment position) all machine axes = 0. Tool and pallet change positions are defined in MCS.



The Work Coordinate System (WCS)

The workpiece coordinate system is based on the MCS with offsets to shift the workpiece to zero. Workpiece programming is done in WCS using the workpiece axes designations X, Y, Z, A and B. The WCS includes all active offsets including the Base Offset, Settable Work (Zero) Offset and the Tool Offset.



In the MANUAL operating mode it is possible to change between the WCS and MCS by toggling the Act. Val. MCS soft key.

4.2 OFFSETS

The most commonly used offsets are the Base Offset, Zero (Work) Offset, and Tool Offset. These offsets are combined together to define the WCS relative to the MCS Machine Zero (Cold Start) position.

4.2.1 BASE OFFSET

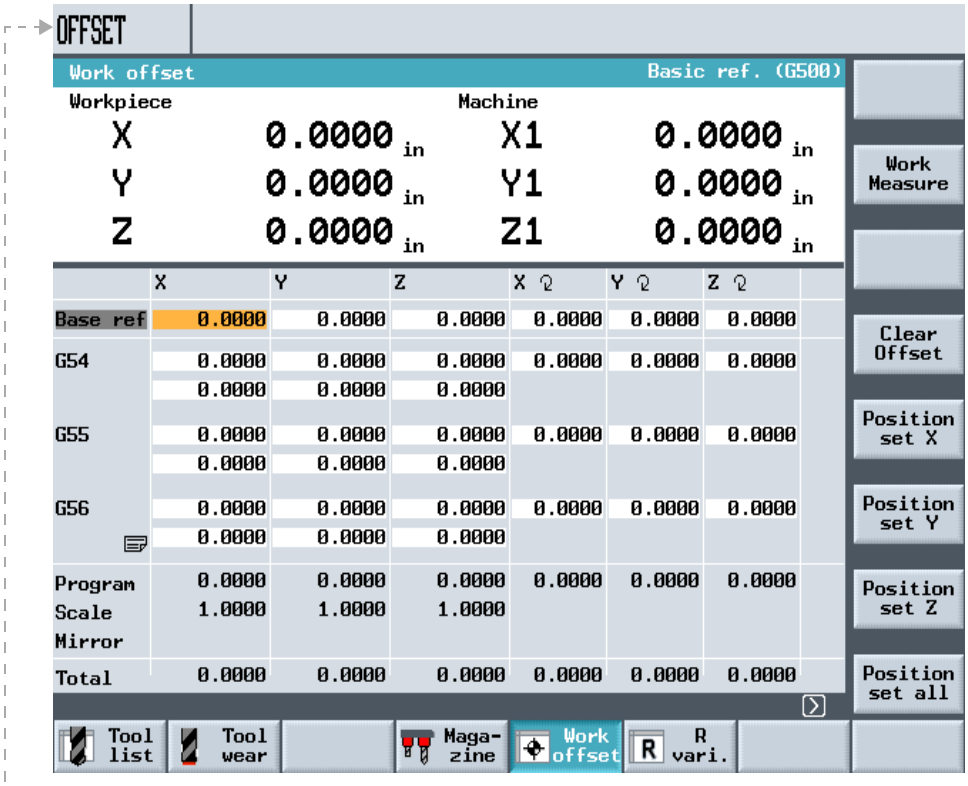
The Base Offset is always active unless G153 is programmed in the movement block (non-modal command). The Base Offset can be set to zero. The Base Offset is used to set the “Home” position relative to the Machine Zero. Based on the current axis position the Base Offset can be calculated and entered into the CNC via the SET Base or Measure Workpiece soft keys in the Manual screens (X, Y and Z only, no A or B). The Base Offset can also be manually entered for all the axes via the Zero Offset screen.

4.2.2 ZERO OFFSET

Zero Offsets (Fixture Offset) 1-8 (ZO1,ZO2,..., ZO8) are programmed via commands G54 - G57 and G505 - G508, the active ZO (example ZO3) is displayed in the light blue field below the axis actual value display. Based on the current axis position, Zero Offsets can be calculated and entered into the CNC via the Measure Workpiece soft keys in the main Manual screens (X, Y & Z only, no A and B). Zero Offsets can also be entered manually for all axes via the Zero offset screen.

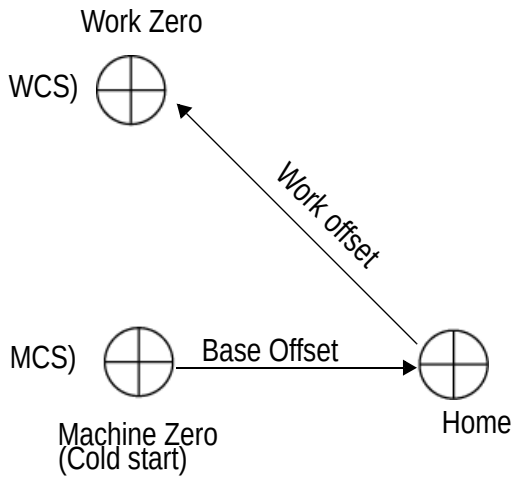
4.2.3 TOOL OFFSET

The tool changer allows each tool with its own offset for tool length and diameter, and tool wear settings for length and diameter. Tool Offsets for the active Tool number (e.g. T3) are activated with the command D1, the offset is deactivated with the command D0. The tool offset D1 is always activated as part of a tool change. Tool offset values for both length and diameter based on measured values can be calculated and entered into the CNC via the measure Tool screens in the main Manual screens. Tool offsets can also be entered manually via the Tool List screen.



Offset screen

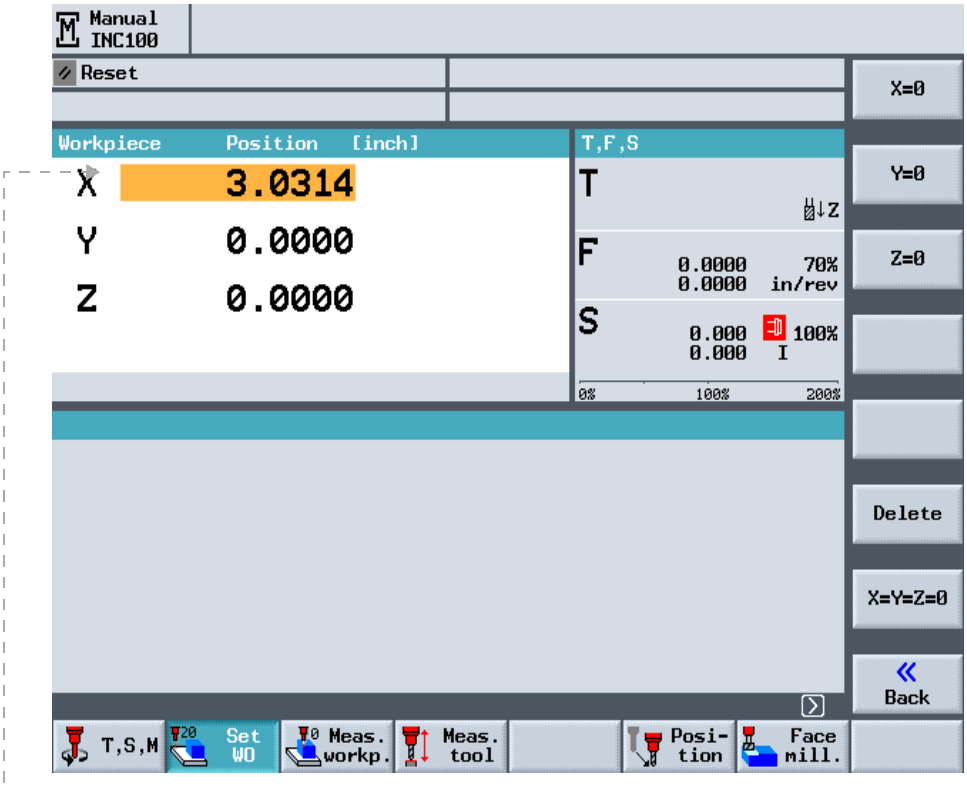
The figure below is a two dimensional diagram illustrating the various coordinate systems and their relationship.



4.2.4 USING THE SET BASE SOFT KEY TO SET THE BASE OFFSET

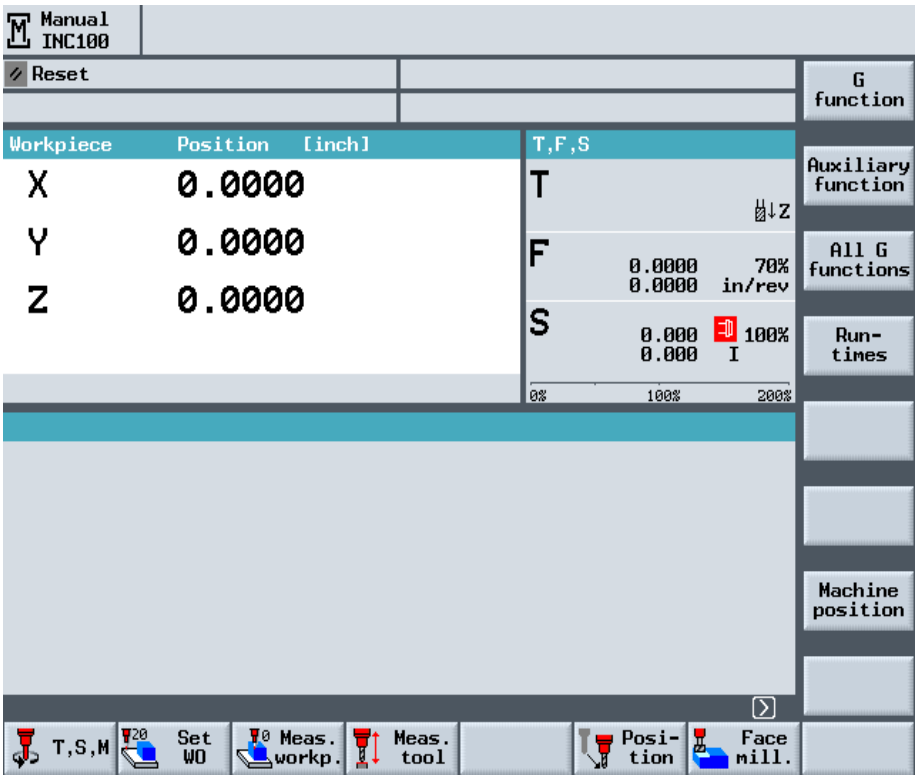
NOTE
Only the X, Y and Z Axes can be set with this method. The Base Offset for the A and B axes must be set directly in the Zero Offset screen.

1. Press JOG hard key.
2. Press Set WO soft key. The X-axis will be highlighted in orange.
3. Switch axis selector to Remote.
4. Press the MPG hard key to activate the MPG hand wheelin the HHU.
5. Select the X-Axis with the axis selector switch.
6. Jog the X-Axis to the desired Base (Home) position.



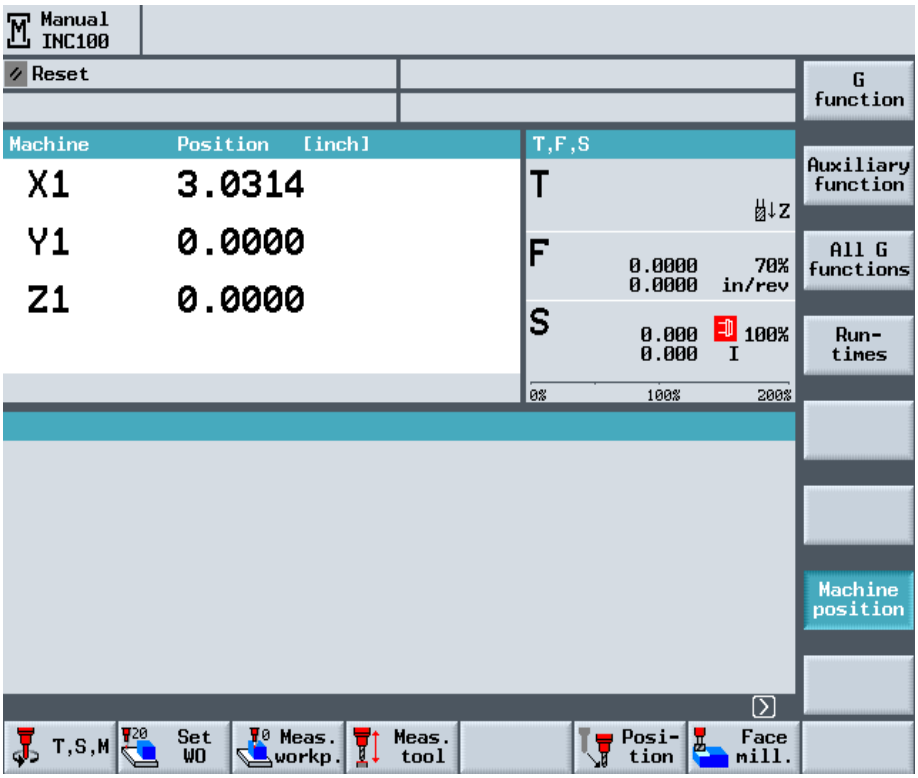
X axis Base position

7. Press the X=0 soft key to set the X-Axis base position for the WCS. The WCS position for the X-Axis will reset to Zero.



8. Press Machine Position soft key. MCS screen displays.

9. Toggle to the MCS display. The WCS base position for the machine will be displayed. In the example below the X-Axis was jogged to +3.0314". The MCS display for the X-Axis now shows the Machine Zero (Cold Start) position is -1.00 inches away from the base position set for the X-Axis in the WCS.



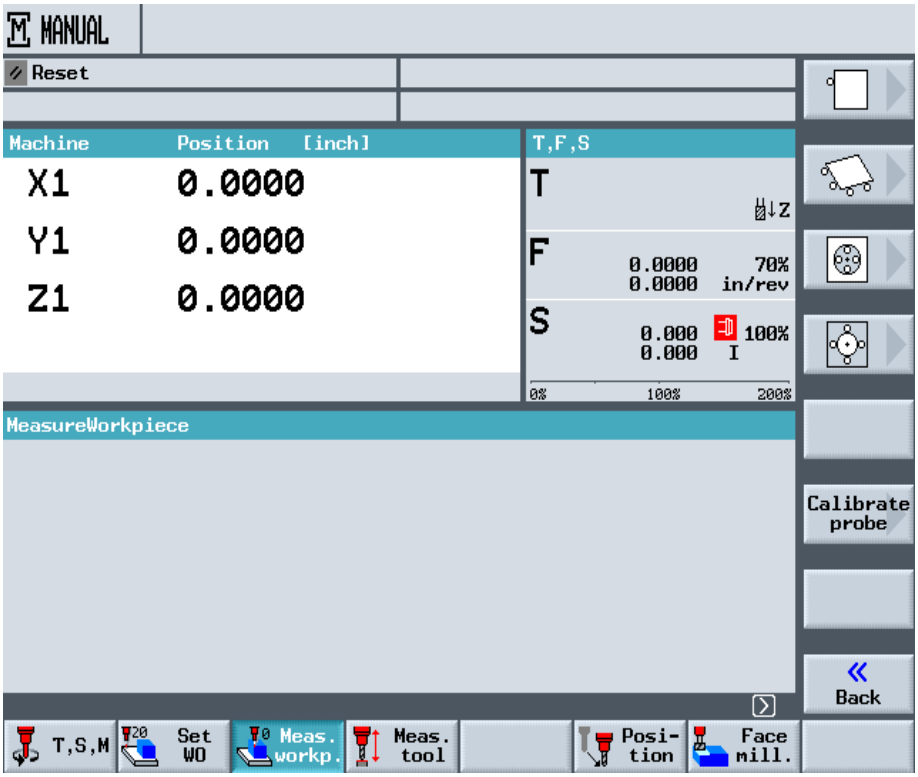
10. To set the Y and Z axes base position, perform steps 1-6 for each axis. Use the cursor key pad arrows to highlight the appropriate axis.

4.2.5 USING THE MEASURE WORKPIECE SOFT KEY TO SET THE BASE AND ZERO OFFSETS

NOTE
Only the X, Y and Z Axes can be set with this method, the Base Offset for the A and B axes must be set directly in the Zero Offset screen.

1. Press JOG hard key.

2. Press the Measure Worp. soft key. The operator has five options to choose from on the vertical soft keys: Edge, Corner, Hole, Spigot and Calibrate probe. Choose the option that will be used to set the offsets.



3. For all five options the operator must select:

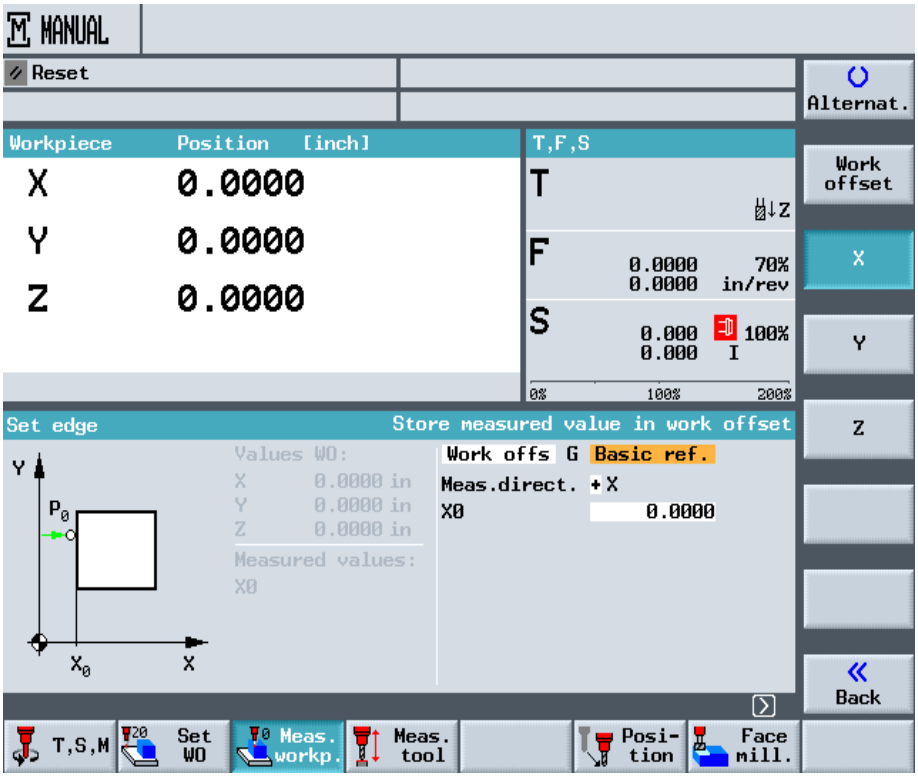
- Which Offset to Calculate, Base or Zero Offsets 1-8 (G54-G57, G505-G508).

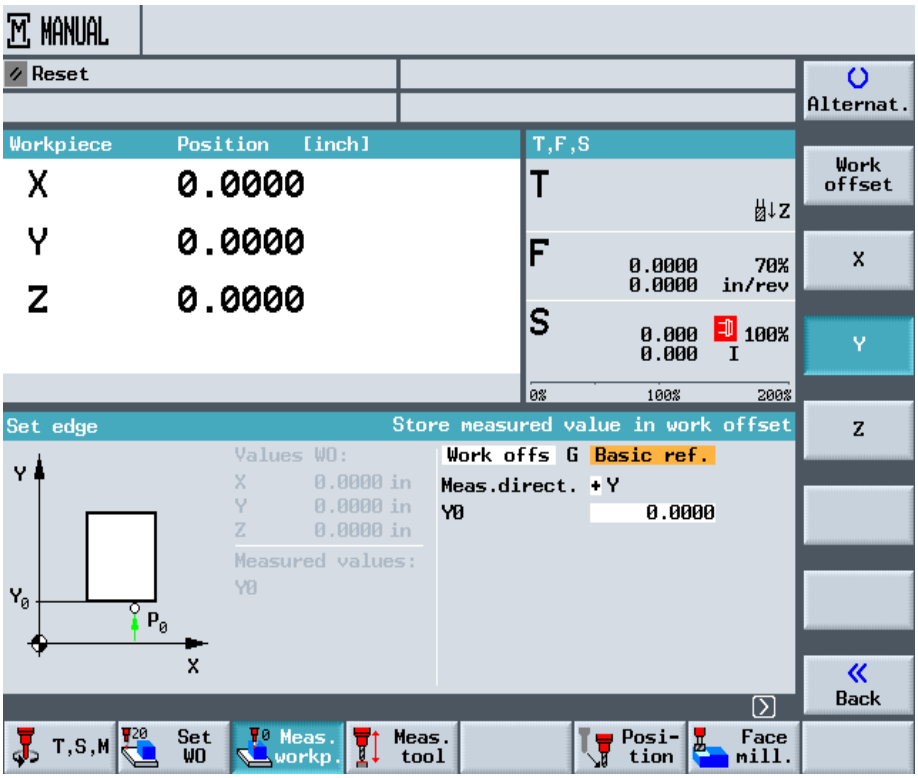
EXAMPLE To calculate the Zero Offset 4 (G57) a value of 4 would be entered in the “Zero offs” field.

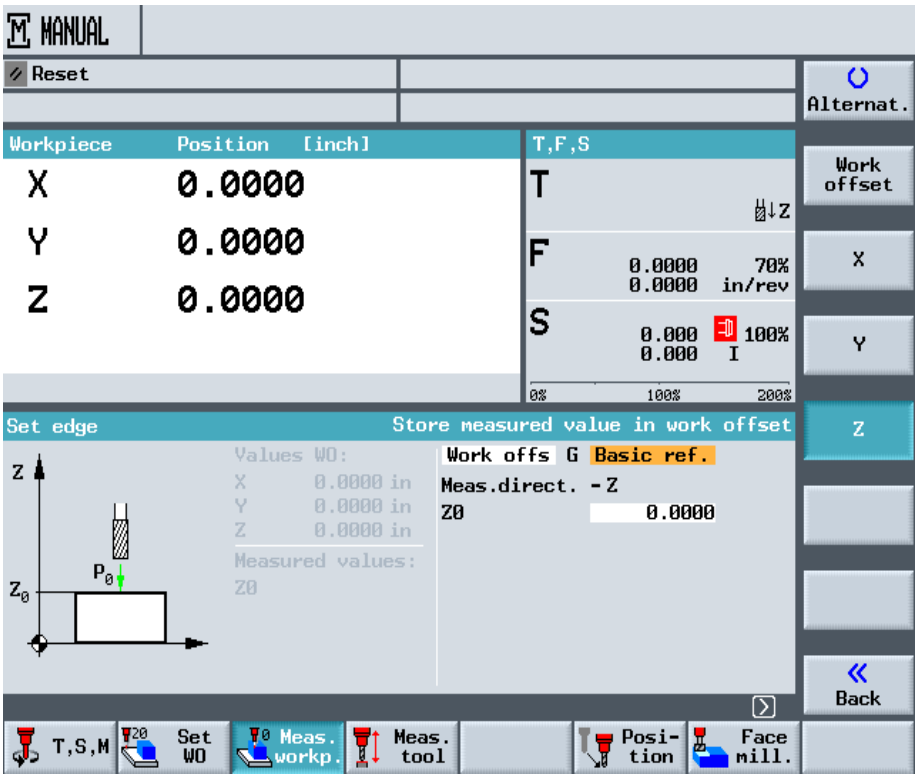
- The Known dimension of the point being measured.

EXAMPLE The “X0” field in the edge, is the X measurement.

4. Select the Edge soft key when using the X, Y and Z WCS position to determine the base setting by using an edge finder or touch probe on the part from each WCS axis.

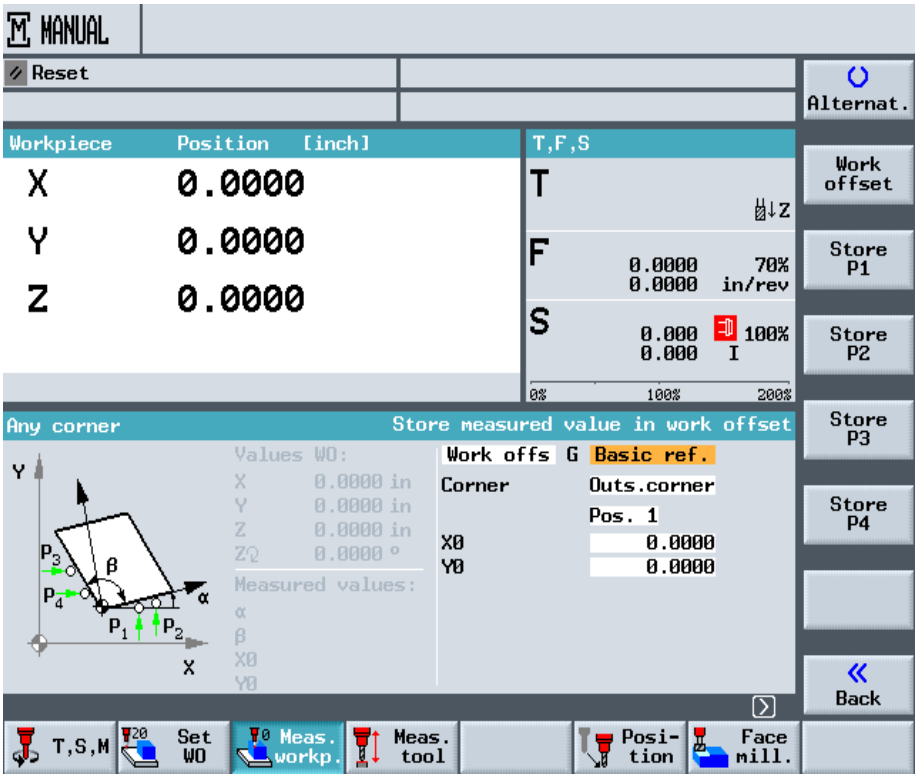



MANUAL
Reset
Alternat.
Workpiece
Position [inch]
X 0.0000
Y 0.0000
Z 0.0000
T,F,S
T
↓Z
F 0.0000 70%
0.0000 in/rev
S 0.000 100%
0.000 I
0% 100% 200%
Work offset
X
Y
Z
Back
Set edge
Store measured value in work offset
Values WO:
X 0.0000 in
Y 0.0000 in
Z 0.0000 in
Measured values:
Y0
Work offs G Basic ref.
Meas.direct. +Y
Y0 0.0000
T,S,M
Set WO
Meas. workp.
Meas. tool
Posi-tion
Face mill.

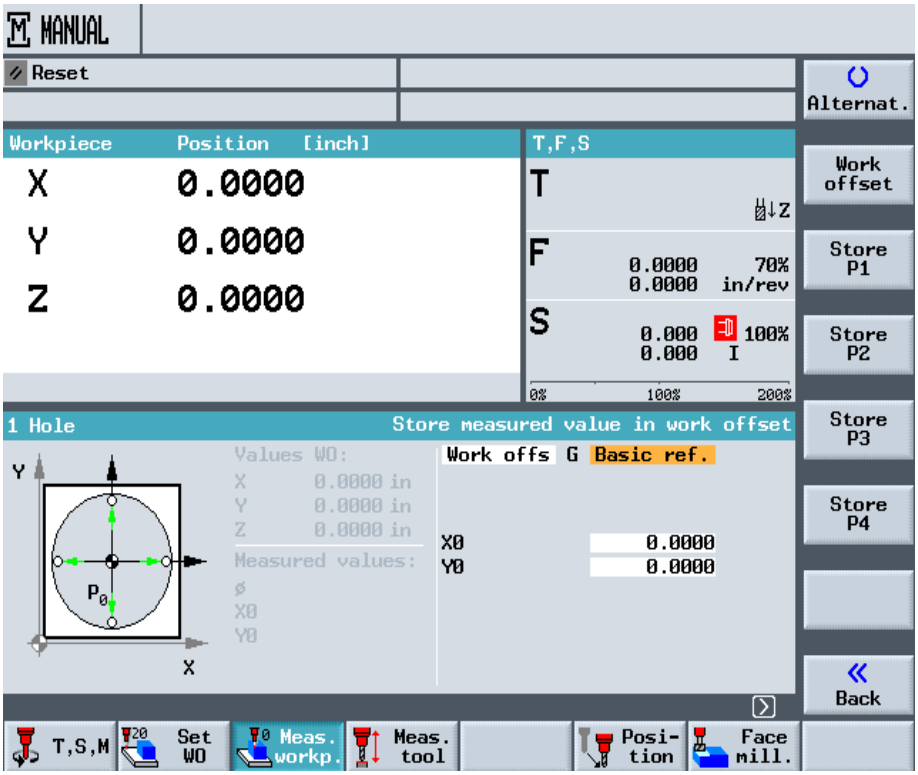


NOTE
An edge finder or probe can be used for these procedures.

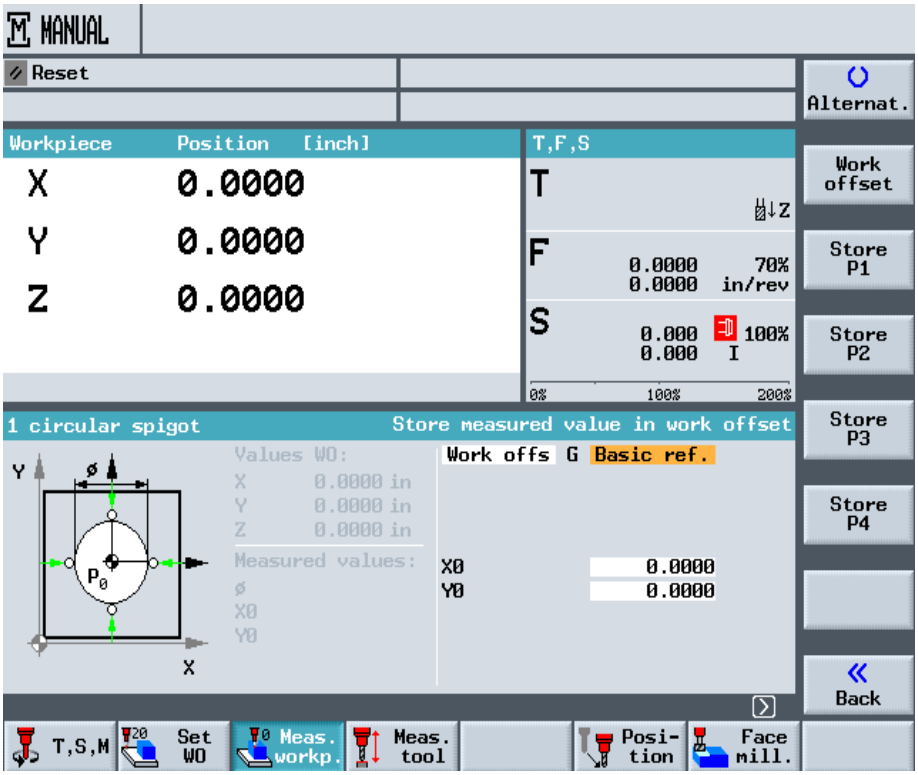
5. Select the Corner soft key when setting the WCS base position by determining the orientation of the workpiece to a selected corner.



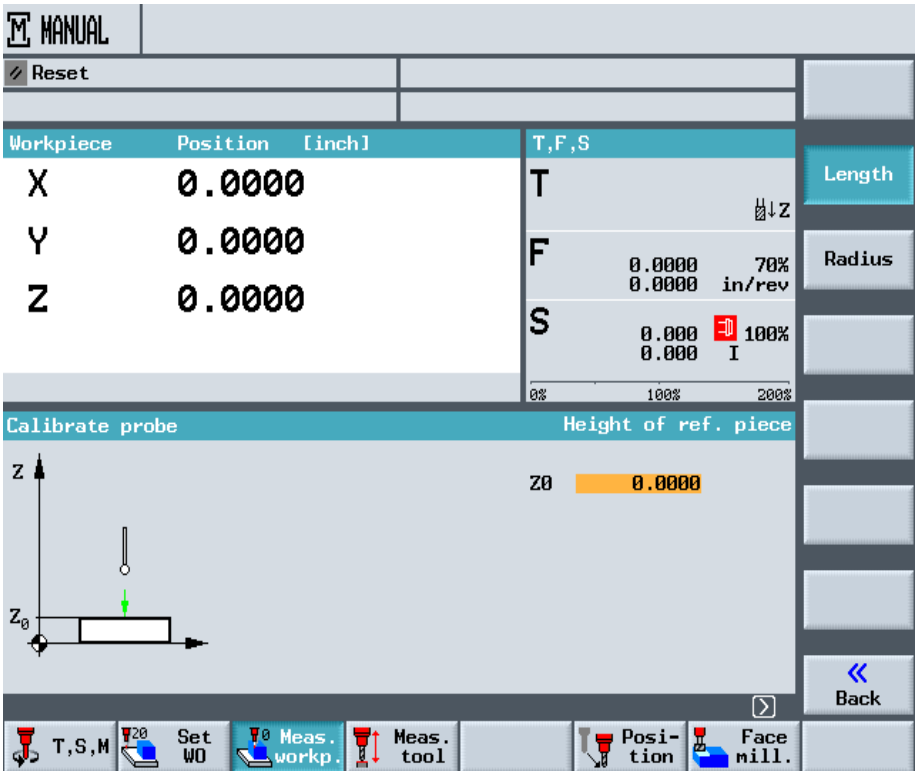
6. Select the Hole soft key when setting the WCS base position by determining the center of a hole at four positions.



7. Select the Spigot soft key when setting the WCS base position by determining the center of a cylinder at four positions.



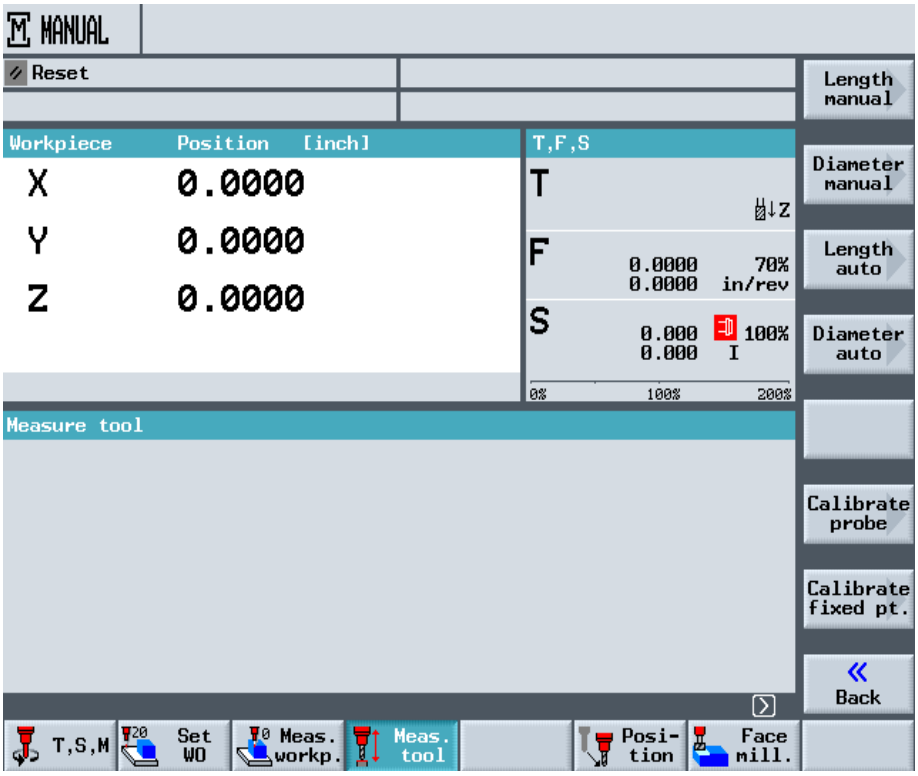
8. Select the Calibrate Probe soft key when checking the accuracy of the probe against standards for length or radius.



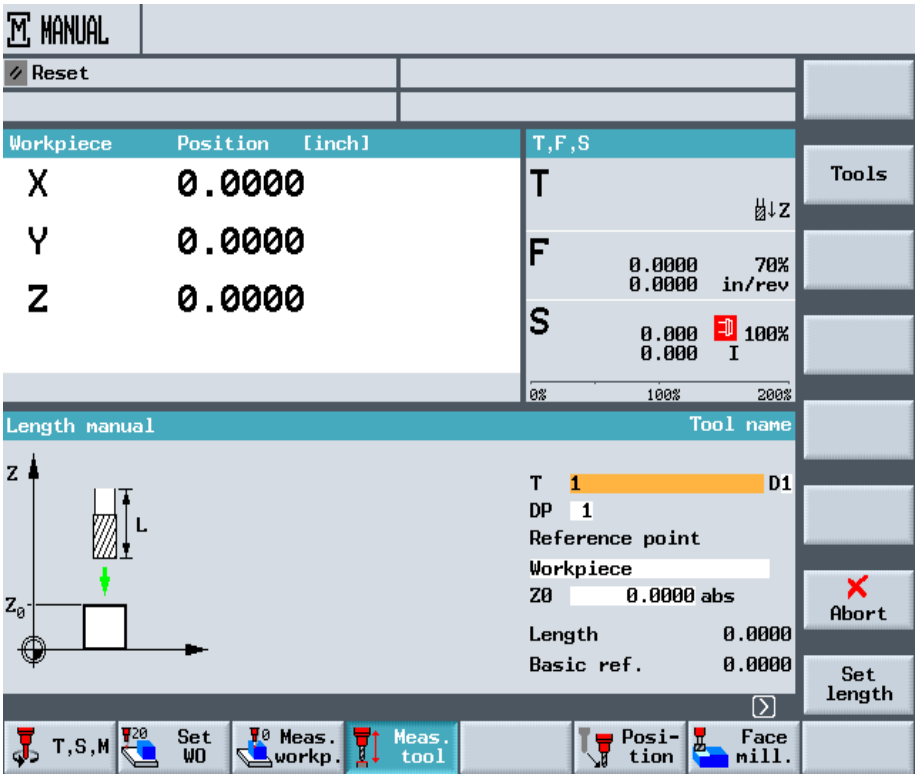
4.2.6 USING THE MEASURE TOOL SOFT KEY TO SET THE TOOL OFFSET

1. Press JOG hard key.

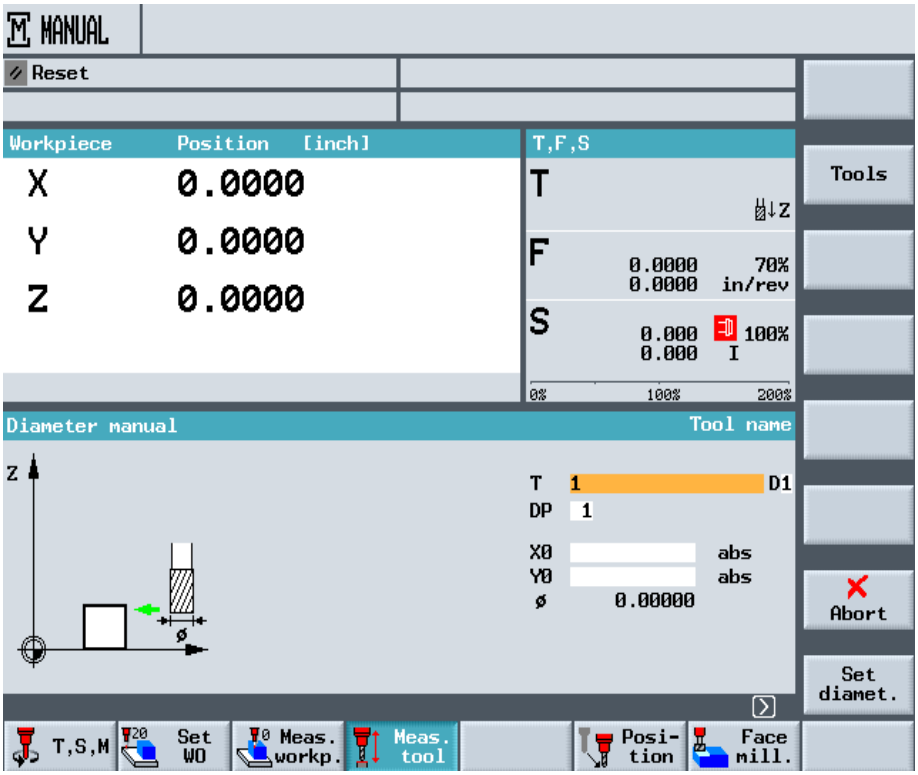
2. Press the MEASURE TOOL soft key. The operator has six options to choose from: Length Manual, Diameter Manual, Length Auto, Diameter Auto, Calibrate probe and Calibrate Fixed point.



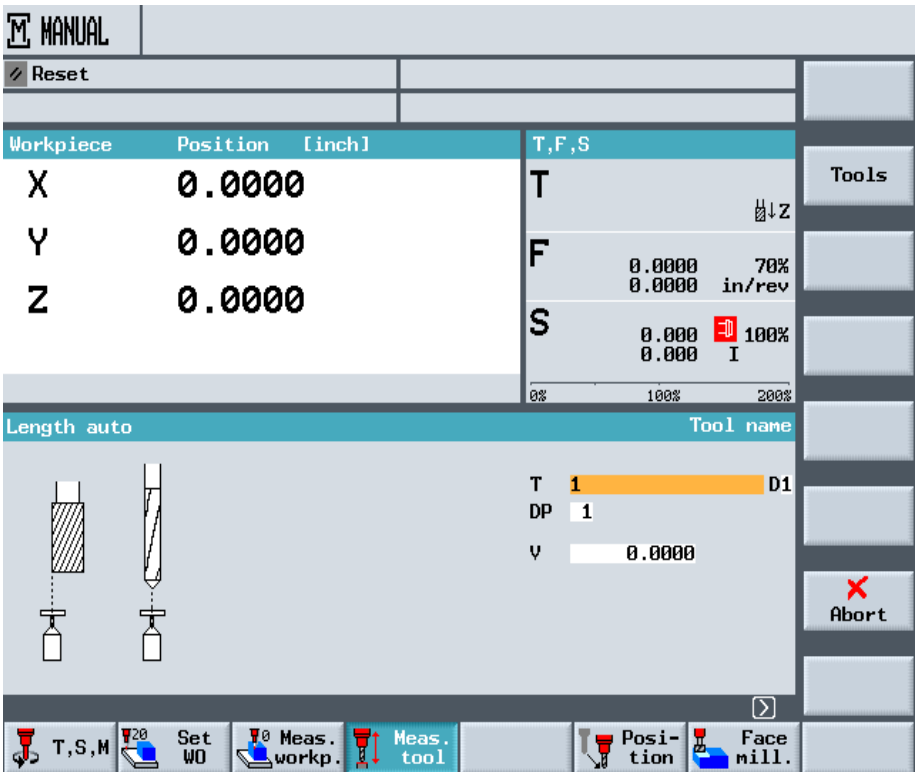
3. Select the Length Manual soft key when manually setting the tool length offset by jogging the tool to a reference block or part.



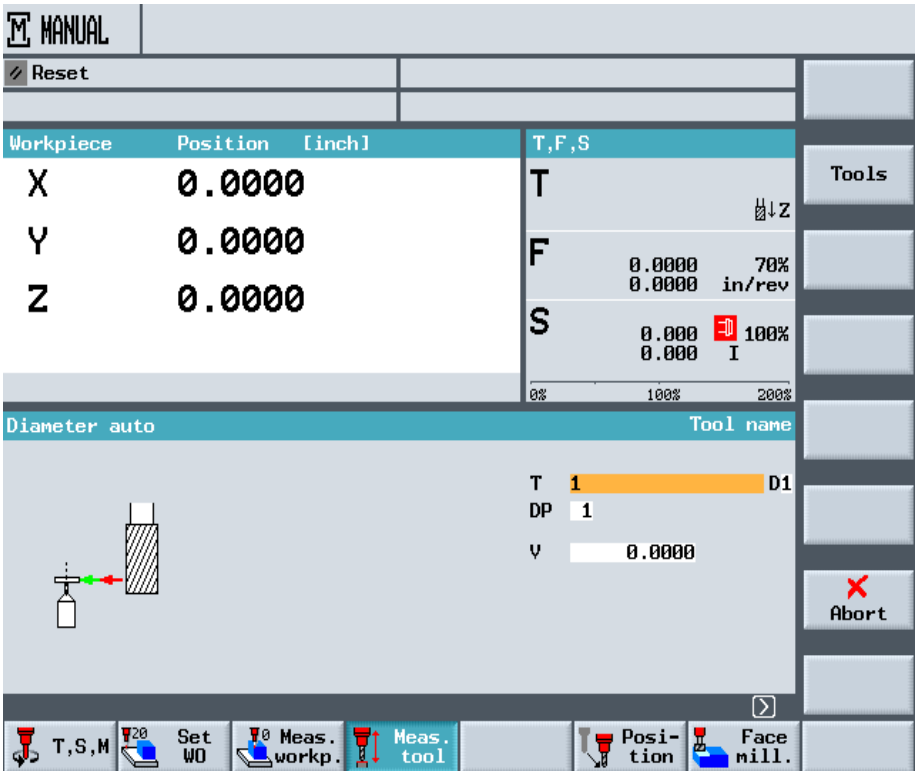
4. Select the Diameter Manual soft key when manually setting the tool diameter offset by jogging the tool to a reference block or part.



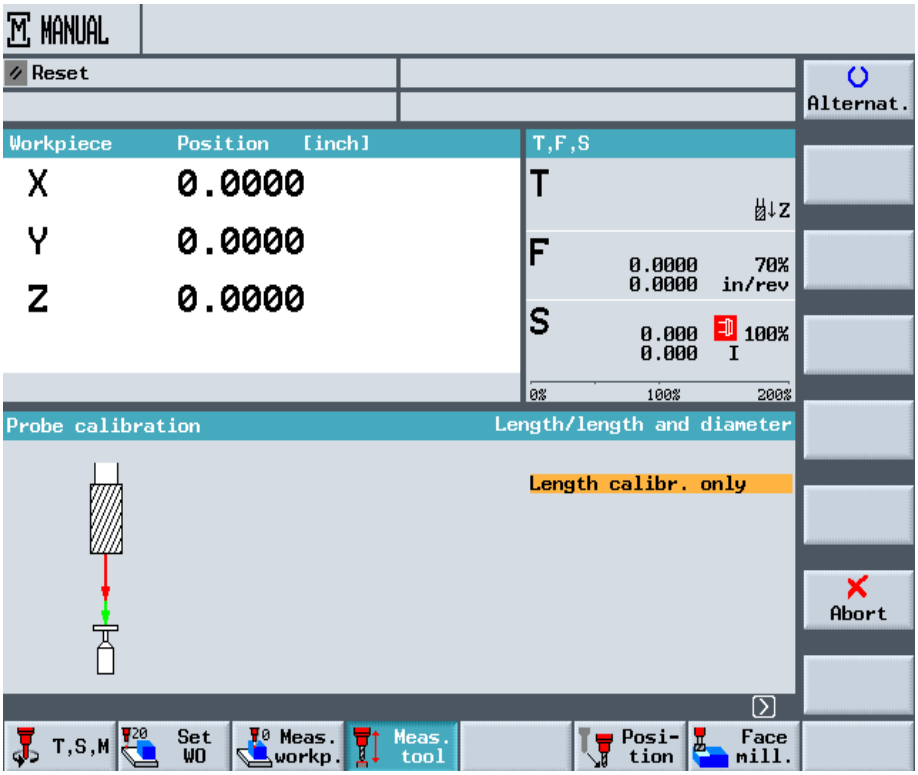
5. Select the Length Auto soft key when setting the tool length offset with a touch probe.



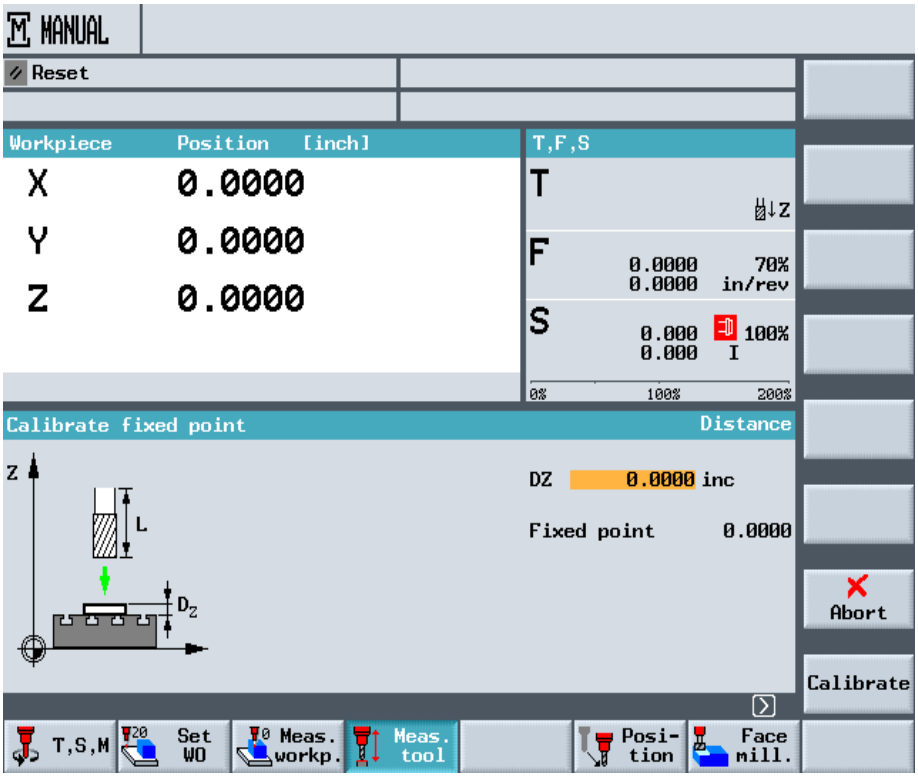
6. Select the Diameter Auto soft key when setting the tool diameter offset with a touch probe.



7. Select the Calibrate Probe soft key when checking the accuracy of the probe against standards for length or radius.



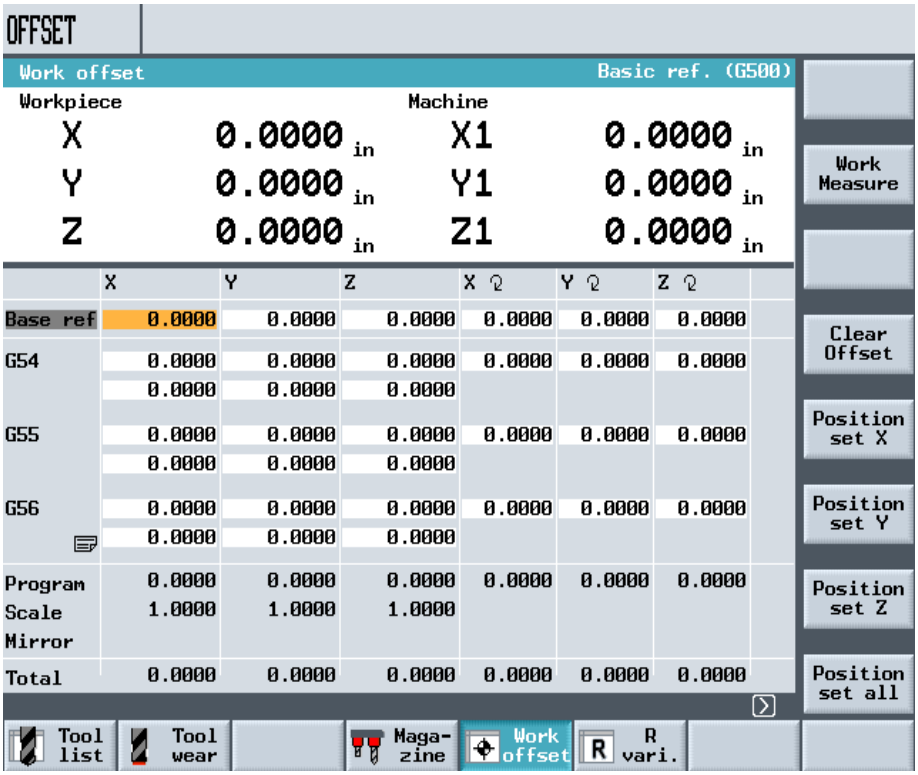
8. Select the Calibrate Fixed point soft key.



4.2.7 USING THE ZERO OFFSET SOFT KEY TO SET THE BASE AND ZERO OFFSETS

1. Press OFFSET soft key to display the main manual display.

2. Press WORK OFFSET soft key. The zero offsets table displays.



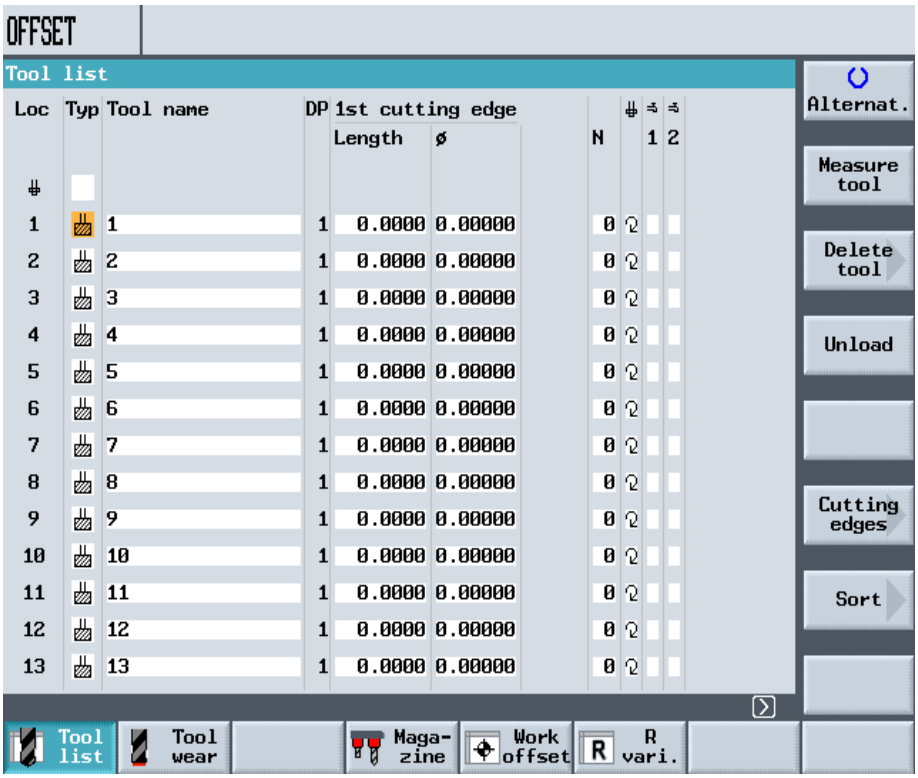
3. Use the cursor key pad arrows to move to the desired fields in the table for editing.

4. The table shows the Base Offset and 3 Zero Offsets at a time. Additional Zero Offsets can be viewed by using the PAGE UP/PAGE DOWN hard keys as indicated in the display. Zero Offsets for the 4th and 5th axes can be accessed by using the Further Axes soft key.

4.2.8 USING THE TOOL SOFT KEY TO SET THE TOOL OFFSETS

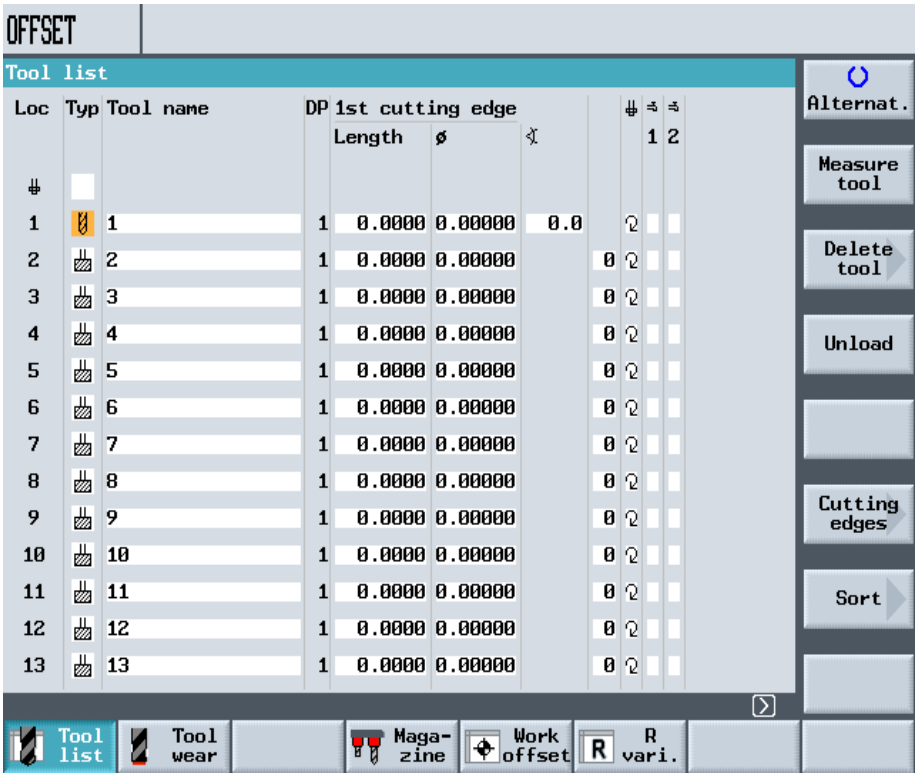
1. The TOOL soft key is accessed from the main manual display. To get to the main manual display press the OFFSETS hard key.

2. Press the Tool List soft key. The Tool List table is displayed.

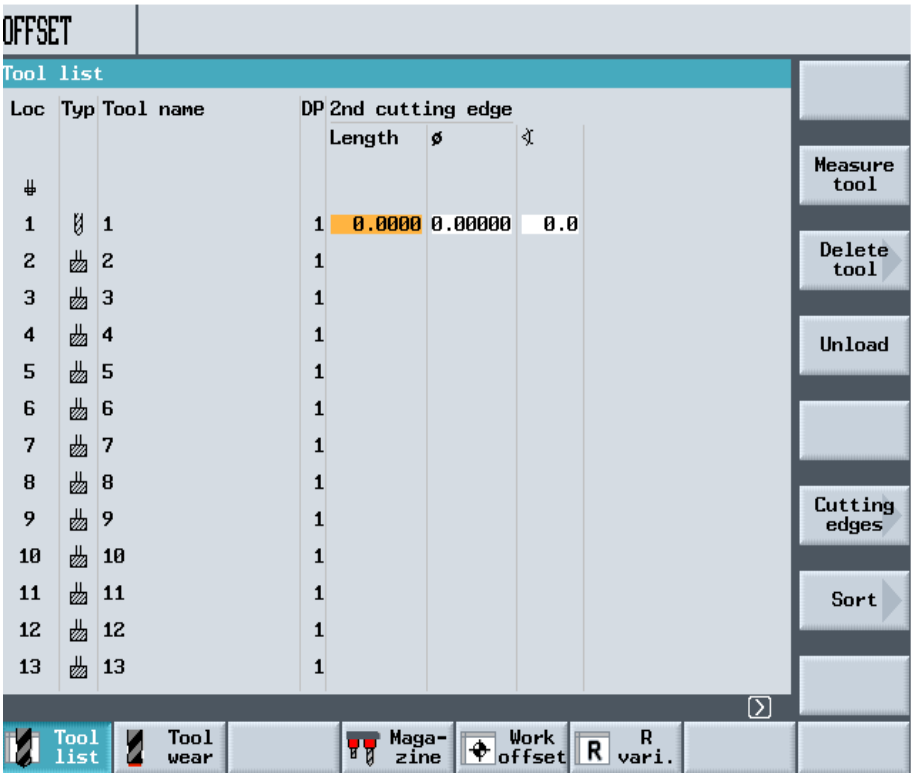


3. Use the cursor key pad arrows to move to the desired fields in the table for editing.

4. Use the ALTERNATE soft key to toggle between options available in the Tool List variable fields. In the example below the ALTERNATE soft key was used to change the tool type for tool number 1 (compare with previous screen display).



5. The default setting of the tool list shows the 1st cutting edge. To set offsets for tools with two cutting edges press the Cutting Edges soft key.



Use the Cutting Edges soft key to toggle between the displays. The CNC command D2 is used to activate offsets for the cutting edges.

4.2.9 SETTING TOOL LENGTH OFFSET

The point where the tools will be set, is called a gauge point. This is a common starting position for all the tools. This is where the programmer has established the Z axis zero position for the part program (not to be confused with Z zero at the MCS position).

To set the TLO (Tool Length Offset) manually:

1. Locate all tools specified for the program and load the tools into tool holders. Place the holders close to the machine. Place the machine in the Manual Mode.

2. Load tool #1 into the spindle using the TOOL IN/OUT button.

3. Place a gauge block, of any available size, on top of the part.

4. Press the MPG button and use the Manual Pulse Generator to jog the Z axis until the tip of the tool is just above the top of the gauge block. Select smaller increments for jog, and jog the tool down until the tip of the tool is close enough to the gauge-block for the desired tool length offset.

5. Remove the block from under the tool.

6. Press the Position soft key to return to the Main Manual screen.

7. Press the Measure Tool soft key.

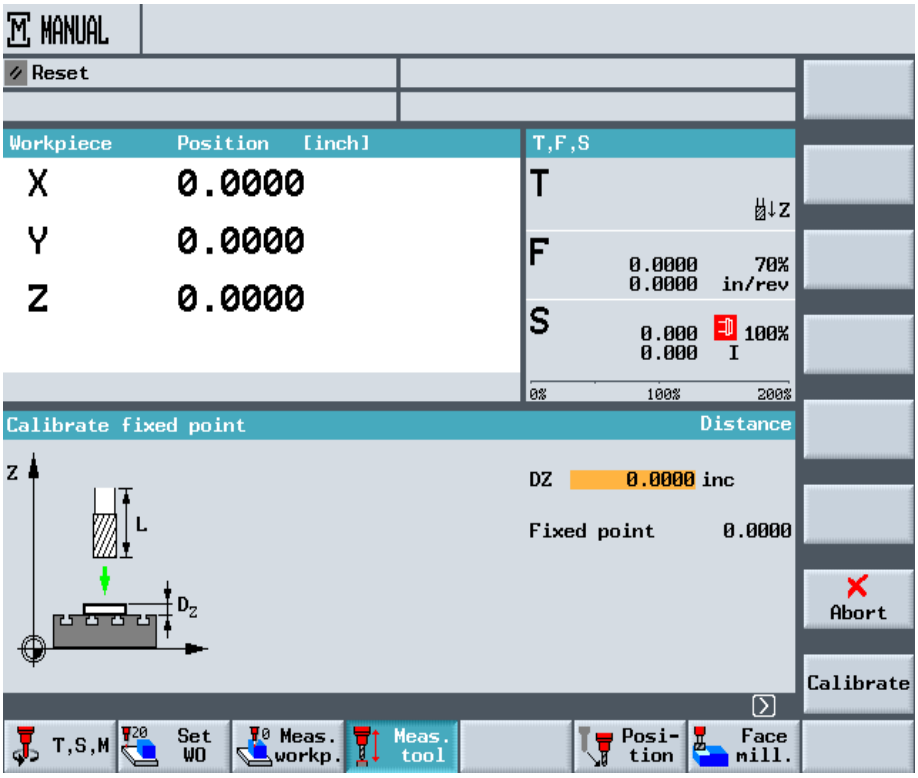
8. Press the Length Manual soft key.

9. Use the cursor keypad arrows, cursor down to highlight Z0 _____ ABS.

10. Type in the gauge amount (tool block size).

11. Press the Set Length soft key button.

This inputs the tool length based on the current Z axis location into the tool table:



5.0 GENERAL INFORMATION

5.1 FINDING MACHINE REFERENCE (COLD START)

The FADAL machine tool has software limits and does not contain position limit switches. Therefore, the machine tool must be physically located at set alignment marks. The Siemens control automatically powers on in the machine reference mode. It is recommended that the machine be shut down at its axis alignment position to simplify the Power On procedure.

To align each axis, place the machine in JOG:

1. Press the MPG hard key until the LED above the key is lit.
2. Find the alignment marks for each axis using the axis selector switch and the MPG.
3. Press the REF hard key button “reference point return”. Use the Axis selector switch to select the desired axis.

Select the X axis then press + hard key button.

Select the Y axis then press + hard key button.

Select the Z axis then press + hard key button.

Select the A axis then press + hard key button.

Select the B axis then press + hard key button.

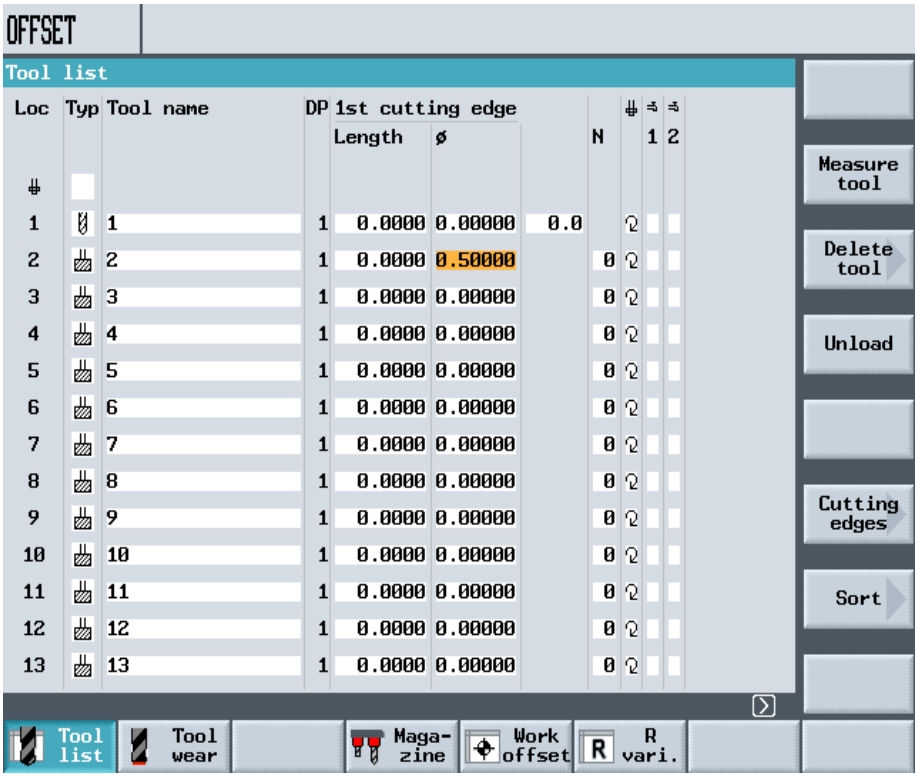
When each axis has completed finding the reference position this  symbol will appear to the left of the axis. Press the JOG, MPG or AUTO hard keys to exit the Reference submode.

NOTE
In the reference mode the position display will indicate the actual position in WCS (X, Y, Z) or MCS (X1, Y1, Z1). The ACT. VAL. MCS soft key can be used to toggle between the WCS and MCS coordinate display.

5.2 TOOL DIAMETER INPUT

To enter tool diameter offsets follow the next steps:

1. Press the OFFSET hard key.
2. Press the Tool List soft key.
3. Use the cursor keypad arrows, cursor down and over to highlight the diameter (Ø symbol) for the desired tool.
4. Type in the diameter amount, then press the INPUT hard key.

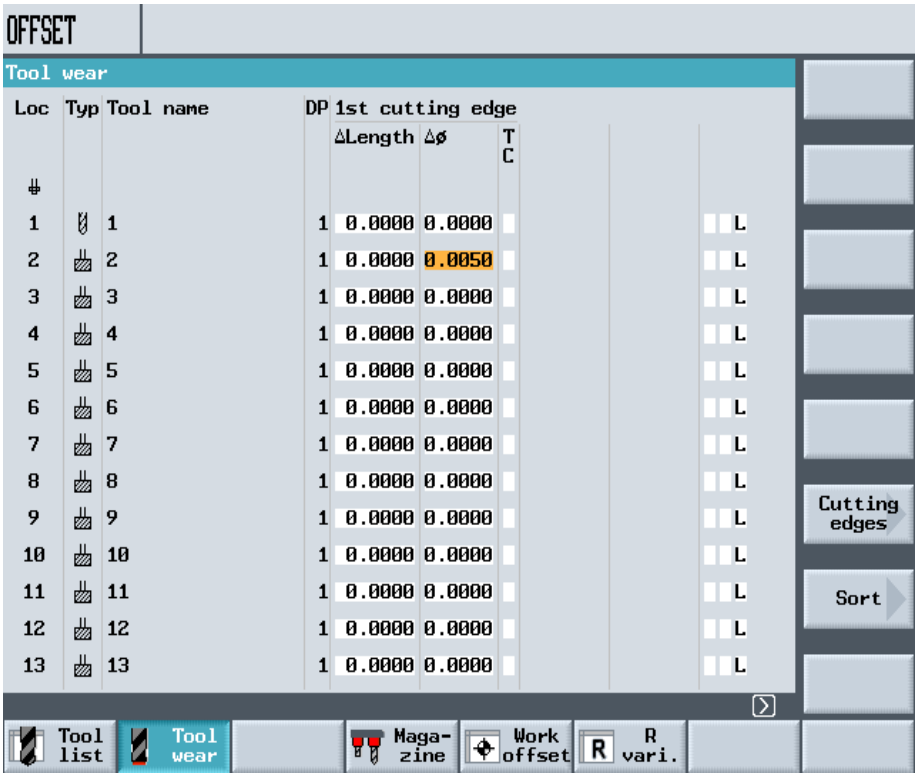


5.3 TOOL WEAR TABLE

The operator has the option to adjust the tool length or the tool diameter by an incremental value.

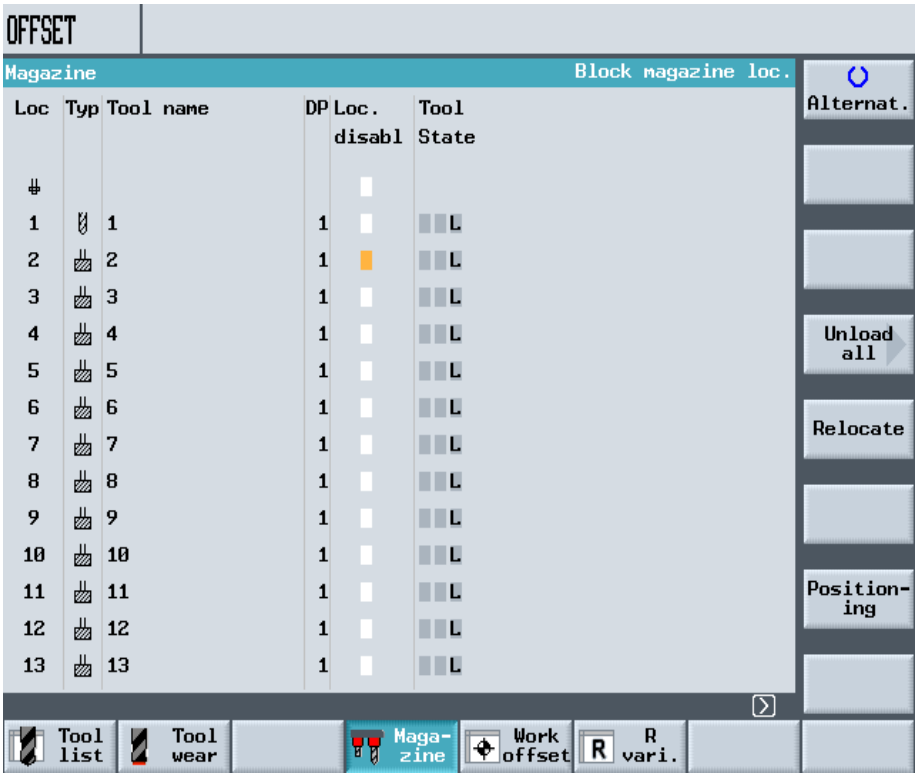
1. Press the OFFSET hard key.
2. Press the Tool wear soft key.
3. Use the cursor keypad arrows, cursor down and over to highlight tool length or diameter.
4. Type in incremental amount then press the INPUT hard key.

> *NOTE*
> When tool length offset is reset by choosing SET LENGTH from the OFFSET page, wear offsets will be zeroed.



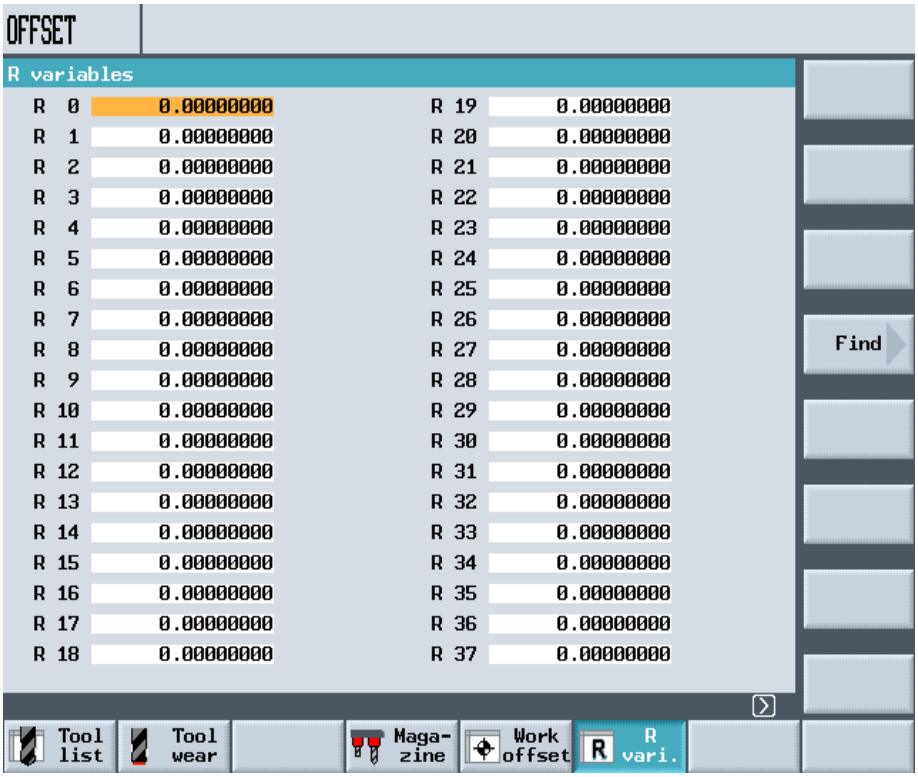
5.4 MAGAZINE TABLE

- Press OFFSET hard key. MAGAZINE soft key displays active tool and turret location.



5.5 R VARIABLE TABLE

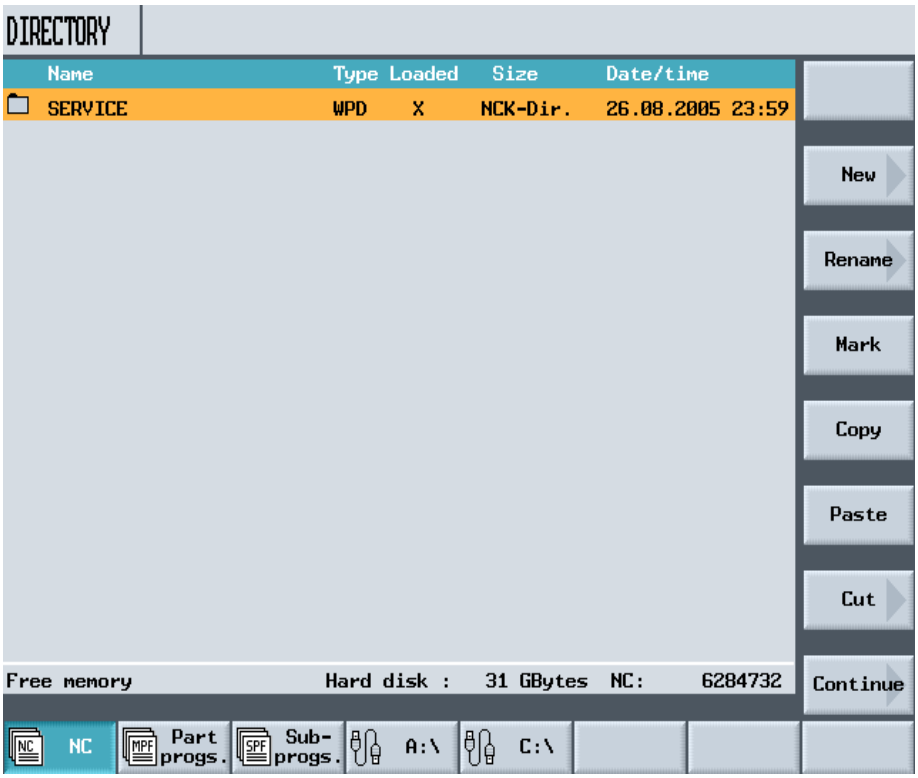
- Press OFFSET hard key. *R vari*. soft key displays the user defined R parameters that can be activated by the program.



5.6 A NEW PROGRAM FOR AUTO

To create a new program for AUTO follow the next steps:

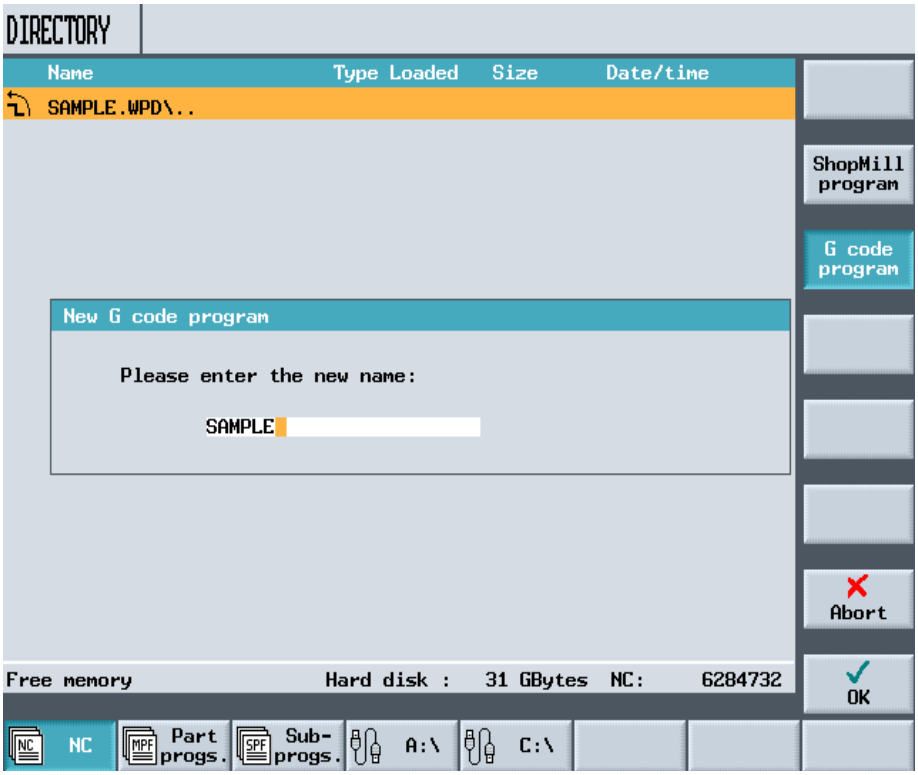
1. Press the PROGRAM MANAGER hard key.



2. Press the Program soft key.

3. Press the New soft key.

4. The operator has the option of using Shop Mill or G-code programming by selecting the appropriate soft key.



5. Type in the new program name.

6. Press the INPUT hard key.

5.7 EDITING AN EXISTING PROGRAM

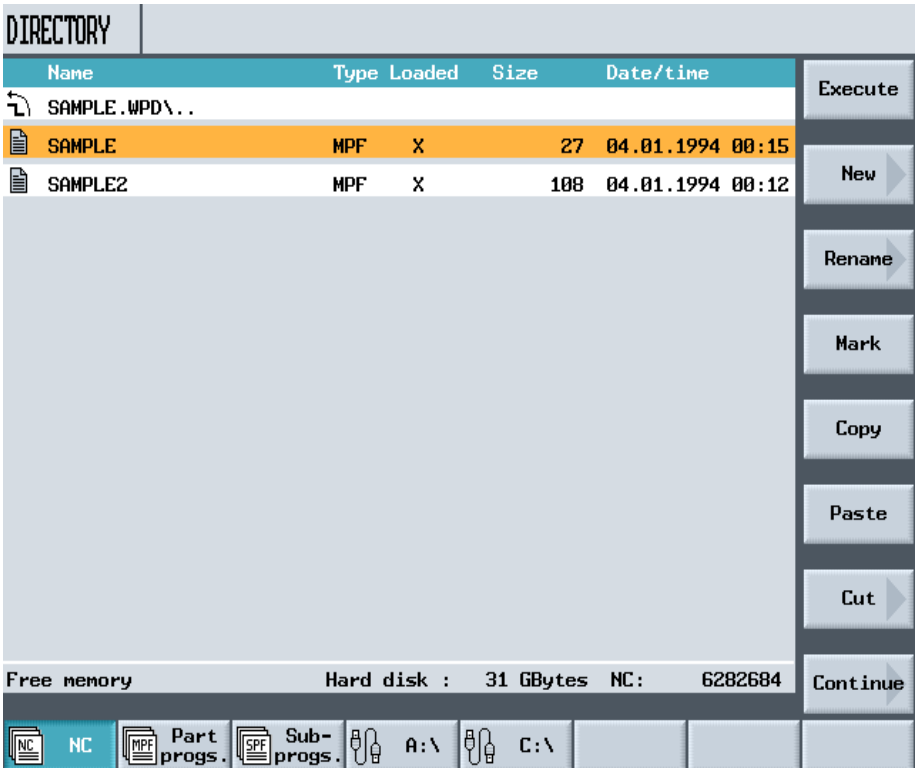
Follow the next steps:

1. Press the Program manager hard key.
2. Use the cursor keypad arrows, cursor down and over to highlight the program name.
3. Press INPUT hard key.
4. If the program is in the Shop Mill format, use the blue cursor keys to select (Arrow Up, Arrow Down) and open (Arrow Right) the desired step for editing.

5.8 CHOOSING A PROGRAM TO RUN IN AUTO

NOTE
To choose a program to run in AUTO it must first be active.

1. Press the PROGRAM MANAGER hard key.
2. Use the cursor keypad arrows, cursor down and over to highlight the program name.
3. Press the Execute soft key. The main screen in auto will be active.
4. Press the NC CYCLE START hard key to begin automatic operation.



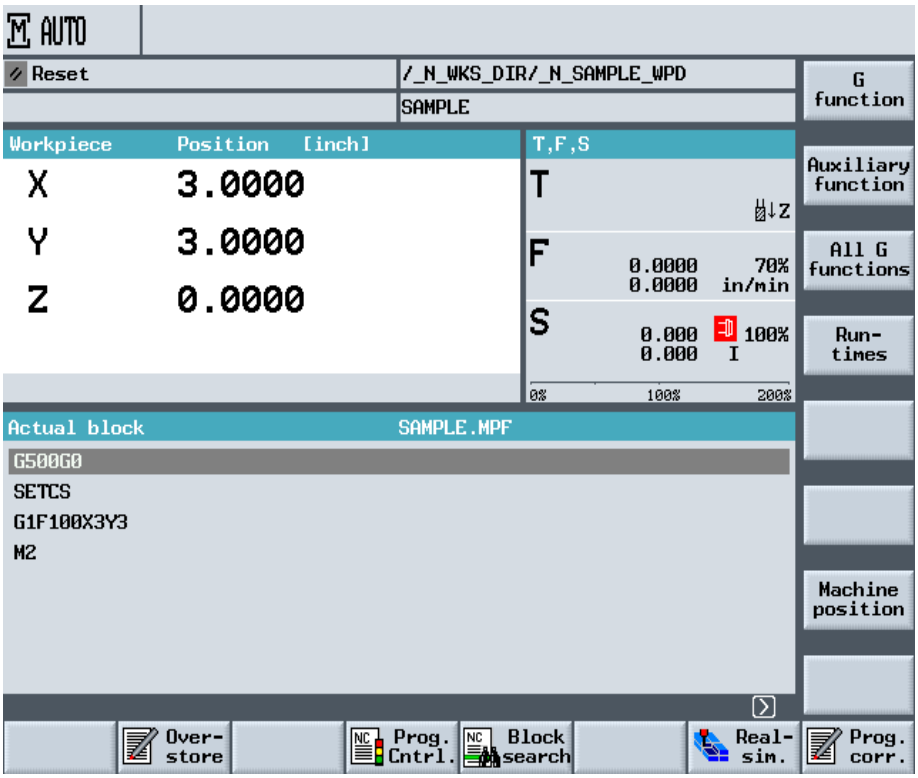
5.9 AUTO, RUNNING A PROGRAM

The AUTO button will only execute the currently active program.

To run the currently active program:

1. Press the AUTO hard key.

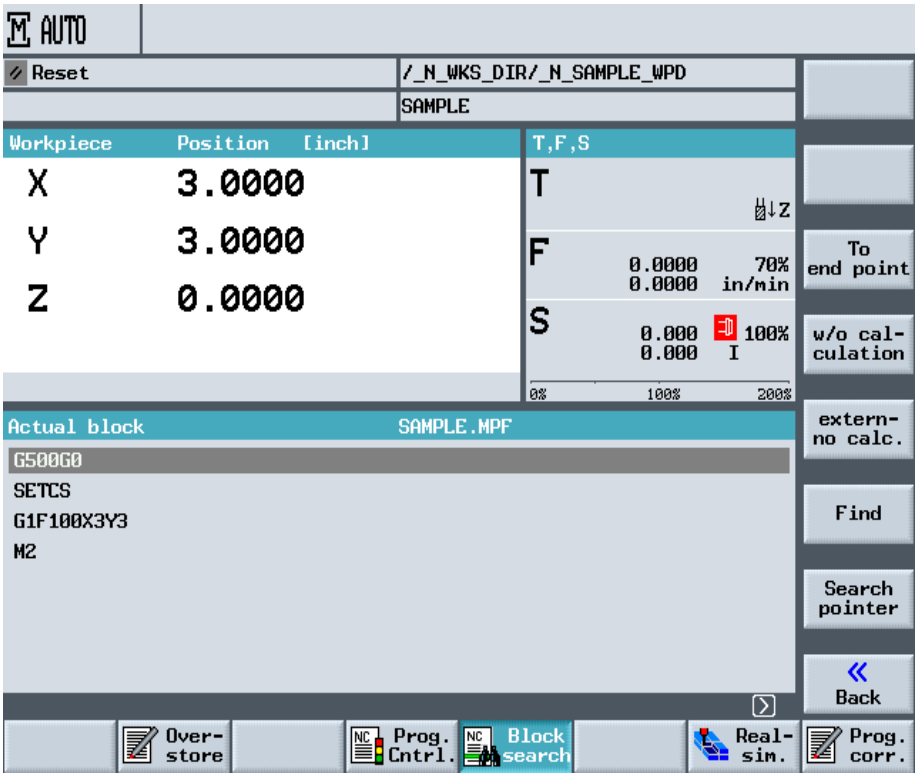
2. Press the NC Cycle Start hard key to begin automatic operation.



NOTE
NC Cycle Start is only active in the main Auto Screen.

5.10 MID-TAPE (PROGRAM) START

Using the mid-tape start option of the auto command, the program can be started from any block. The options are available from the BLOCK SEARCH soft keys on the auto display.



5.11 OEM ALARM (V050805)

Table 5-1: Alarm List

ALARM NO.	DEFINITION	REACTION	REMEDY	SOLLUTION
66500 Spindle Orientation Failed	The spindle failed to orient properly	Display alarm. Axis movement disable. NC start disable.	Check the spindle for proper positioning. Check orientation sensor.	Clear alarm with Reset key. Restart part program. Contact Service.
67000 Is The Spindle Empty? If Yes Press Cycle Start To Continue	Verify there is no tool in the spindle before going on with the tool change to avoid damaging the tool magazine. Tool zero is in the spindle.	Display alarm. Axis movement disable. NC start disable.	Verify there is not tool in the Spindle and continue as prompted.	Continue part program. Contact Service.
67001 Load Tool Active	The tool load function is active.	Display message. Axis movement disable. NC start disable.	Normal message during tool load.	Continue part program. Contact Service.
67002 Tool (Un)load Not Possible; Tool In Spindle	The spindle is the load/unload position. If a tool is already there tool loading/ unloading is not possible.	Display alarm. Axis movement disable. NC start disable.	Call T0 (tool zero) to remove the tool from the spindle.	Continue part program. Contact Service.
510007 Air/Oil system Fault	Fault in the Spindle Air/Oil system.	Display alarm. Program interrupted. Axis movement disable.	Refer to alarms 700056, 700057, 700058 for the cause.	Press the reset key and restart the program.
510008 User Feed Stop	User has pressed the Feed Stop key.	Display alarm. Program interrupted. Axis movement disable.	Remove feed stop condition.	Press the Feed Start key or Cycle Start key.
510009 Feed Stop Selected >> Spindle Not Rotating	User feed stop condition prevents the spindle from starting.	Display alarm. Program interrupted. Spindle rotation disable.	Remove feed stop condition.	Press the Feed Start key or Cycle Start key.
510010 ATC Not In Home Position	Automatic Tool changer not in home position.	Display alarm. Program interrupted. Axis movement disable. Spindle rotation disable.	Check tool changer mechanism for malfunctions/obstructions. Bring the DATC arm to home position manually.	Clear alarm with Reset key. Restart part program. Contact Service.
510011 DATC Magazine Out Of Synchronization: Re-referencing needed.	Tool magazine position count has been lost.	Display alarm. NC start disable. Program interrupted.	Bring the tool magazine pocket 1 to tool change position, then run M11.	Alarm will re-set by running M11. Restart part program. Contact Service.

Table 5-1: (Continued) Alarm List

ALARM NO.	DEFINITION	REACTION	REMEDY	SOLLUTION
510012 Dual Arm Active	The dual arm gripper in the tool changer in movement.	Display alarm. Axis movement disable. Spindle rotation disable.	Normal during tool change. Check dual-arm tool changer interface if displayed outside tool change.	Clear alarm with Reset key. Restart part program. Contact Service.
510100 DATC Arm Motor Overload	Dual arm tool changer arm motor overload tripped.	Display alarm. Axis movement disable. Program interrupted.	Check DATC arm for mechanical problems. Check DATC arm motor, brake, and electrical connections. Re-set DATC arm motor overload sensor.	Clear alarm with Reset key. Restart part program. Contact Service.
510101 DATC Magazine Motor Overload	Dual arm tool changer magazine motor overload tripped.	Display alarm. Axis movement disable. Program interrupted.	Check DATC magazine for mechanical problems. Check DATC magazine motor, brake, and electrical connections. Re-set DATC magazine motor overload sensor.	Clear alarm with Reset key. Restart part program. Contact Service.
510102 DATC Arm Time-out	Dual arm tool changer arm failed to move in time allowed.	Display alarm. Axis movement disable. Program interrupted.	Check DATC arm for mechanical problems. Check DATC motor and electrical connections. Check any conditions that will prevent the DATC from normal function (door open, lack of compressed air, etc.)	Clear alarm with Reset key. Restart part program. Contact Service.
510103 DATC Magazine Time-out	Dual arm tool changer magazine failed to move in time allowed.	Display alarm. Axis movement disable. Program interrupted.	Check DATC magazine for mechanical problems. Check DATC magazine motor and electrical connections. Check any conditions that will prevent the DATC from normal function (lack of compressed air, motor brake malfunction, etc.)	Clear alarm with Reset key. Restart part program. Contact Service.
510104 DATC Tool Pocket Time-out	Dual arm tool changer tool pocket failed to move in time allowed.	Display alarm. Axis movement disable. Program interrupted.	Check DATC tool pocket for mechanical problems. Check any conditions that will prevent the DATC from normal function (door open, lack of compressed air, etc.) Check tool pocket actuation valves, electrical connection, fuses.	Clear alarm with Reset key. Restart part program. Contact Service.

Table 5-1: (Continued) Alarm List

ALARM NO.	DEFINITION	REACTION	REMEDY	SOLLUTION
510105 Spindle Draw Bar Arm Time-out	Draw bar mechanism failed to close/open in time allowed.	Display alarm. Axis movement disable. Program interrupted.	Check draw bar for mechanical problems, lack of compressed air. Check spindle for jammed tools. Check any conditions that will prevent the DATC from normal function (door open, lack of compressed air, etc.)	Clear alarm with Reset key. Restart part program. Contact Service.
510106 Z Axis Not In Tool Change Position	DATC is prevented to move because Z axis is not in position.	Display alarm. Axis movement disable. Program interrupted.	Move the Z axis to the cold start position. Check the Z-axis-in-position sensor. Verify Cold Start position.	Clear alarm with Reset key. Restart part program. Contact Service.
510124 ATC Active	Automatic Tool Changer in movement.	Display alarm. Axis movement disable. Spindle rotation disable.	Normal during tool change. Check ATC mechanism if displayed outside tool change.	Clear alarm with Reset key. Restart part program. Contact Service.
510125 Gear Change In Progress	Spindle is undergoing a gear change.	Display alarm. Program interrupted. Spindle rotation disable.	Normal during spindle speed range change. Check belt actuators and /or sensors. Contact Service	Alarm must clear after a few seconds. Power machine off and on. Contact Service.
510126 M5 In Progress	Spindle stop command in progress	Display alarm. Spindle rotation stopped.	Normal during spindle stop.	Alarm must clear once spindle stops. Contact Service.
510127 ATC Motor Failure	Automatic Tool Changer failure. Tool changer board failure. Geneva type only: Magazine failed to rotate.	Display alarm. Program interrupted. Spindle rotation disable.	Press Reset key. Check tool changer for mechanical problems. Check air pressure to machine (air-oil board is interlocked). Check emergency stop hardware loop (110VAC circuit is also interlocked).	Press RESET. Contact Service.
510128 ATC only 21 Tools	The tool changer is a 21 tool type and an attempt has been made to use more than 21 tools.	Display alarm. Program interrupted. Spindle rotation disable.	Check the tool number being called. Check the options available for the machine.	Press Reset key. Contact Service.
510200 Option Rigid Tapping Not Available	Rigid tapping was tried in a machine not equipped with this option.	Program interrupted (read-in disabled). Display alarm.	Check the options available for the machine.	Press Reset key. Contact Service.

Table 5-1: (Continued) Alarm List

ALARM NO.	DEFINITION	REACTION	REMEDY	SOLLUTION
510216 Oiler level too low	Way-lube oiler level too low.	Display alarm. Program interrupted (read-in disabled).	Fill up way-lube oiler reservoir. Program will continue automatically.	Program will continue as soon as oil level is high again. Press Reset key. Contact Service.
510300 POWER ON Needed To Set Spindle Set Up	M83 has been run to measure the spindle belts.	Display alarm. NC-start disabled.	Power the machine off and on.	Power the machine off and on. Contact Service.
510308 Tool Loading Active	The tool loading function is active. A tool loading into the magazine is taking place.	Display alarm. NC-start disabled.	Normal during tool loading. If tool loading is interrupted and alarm lingers, power machine off and on and try tool loading/unloading again.	Press Reset key. Alarm must be cleared to run programs. Contact Service.
510309 Tool Unloading Active	The tool unloading function is active. A tool unloading from the magazine is taking place.	Display alarm. NC-start disabled.	Normal during tool unloading. If tool unloading is interrupted and alarm lingers, power machine off and on and try tool loading/unloading again.	Press Reset key. Alarm must be cleared to run programs. Contact Service.
600108 User Spindle Stop	The spindle stop key in the machine control panel has been pressed.	Display alarm. Axis movement disable. Spindle rotation disable.	Press the spindle start (green key) in the machine control panel. Press the Reset key. Press the Cycle Start key.	Press the Cycle Start key. Contact Service.
600109 Spindle Stop LOCK	The mechanical lock for spindle orientation is engaged.	Display alarm. Spindle rotation disable.	Normal during tool change. Appears when using M19 and M111.	Call an M5, M3, M4 command. Press Reset key for more than 2 seconds Contact Service.
700032 X Axis Needs Referencing	The ìXî axis has not been referenced (cold started).	Display alarm. NC-start disableD until all axes are referenced.	Reference (cold start) the axis.	Reference (cold start) ALL axes. Contact Service.
700033 Y Axis Needs Referencing	The ìYî axis has not been referenced (cold started).	Display alarm. NC-start disableD until all axes are referenced.	Reference (cold start) the axis.	Contact Service. Reference (cold start) ALL axes.
700034 Z Axis Needs Referencing	The ìZî axis has not been referenced (cold started).	Display alarm. NC-start disableD until all axes are referenced.	Reference (cold start) the axis.	Reference (cold start) ALL axes. Contact Service.

Table 5-1: (Continued) Alarm List

ALARM NO.	DEFINITION	REACTION	REMEDY	SOLLUTION
700035 A Axis Needs Referencing	The ìAî axis has not been referenced (cold started).	Display alarm. NC-start disableD until all axes are referenced.	Reference (cold start) the axis.	Reference (cold start) ALL axes. Contact Service.
700036 B Axis Needs Referencing	The ìBî axis has not been referenced (cold started).	Display alarm. NC-start disableD until all axes are referenced.	Reference (cold start) the axis.	Reference (cold start) ALL axes. Contact Service.
700039 Doors Open	Doors-closed monitoring circuit open.	Display alarm. User Feed Stop. User Spindle Stop.	Check doors to be completely closed. Check doors-closed monitoring circuit (sensors, etc.). Check air pressure, air-oil board and auger board (interlocked).	Close doors. Use the Override button in JOG (Manual) mode. Contact service.
700040 Handwheel Available Only In Jog Mode	MPG or INC keys pressed while in Automatic mode.	Display alarm.	Change over to JOG (Manual) mode before using the handwheel.	Contact service if alarm lingers.
700041 Feed Override = 0	A manual movement was attempted while the Feed Override is set to zero.	Display alarm.	Set Feed Override knob to desired setting other than zero. Check Feed Override switch.	Set Feed Override knob to desired setting other than zero. Contact Service.
700042 Please Press MDA hard Key	MDA hard key was pressed when in Shop Mill.	Display alarm. No change to MDA mode until MDA soft key (menu under the screen) is used.	Use MDA soft key to change over to MDA mode in Shop Mill.	Contact Service if alarm lingers.
700049 Function Available only in REEMOTE	This function is available only in the Hand Held Unit (Remote MPG).	Display alarm.	Use the Hand Held Units keys or make an axis selection in the machine control panel.	Contact Service if alarm lingers.
700050 Chiller Unit Fault	The chiller unit controller in the cabinet detected a fault condition.	Display alarm. Machine continues to run but risk of spindle thermal changes occur.	Check chiller temperature sensor connections. Verify power supply to LOGO! Unit. Check LOGO! Unit.	Contact Service if alarm lingers.
700051 Move to safe location and perform TEST STOP	CE machines equipped with Safety Integrated require a Test-stop every 8 hours to verify the correct works of the machine safety.	Display alarm. Machine continues to run.	Alarm will be displayed until the axes are moved to a safe location, changed to JOG mode and with the doors open the Override and CYCLE STOP keys are pressed at the same time to initiate the Test Stop check.	Once the Test Stop is performed, continue the program in a normal way. Contact Service if alarm lingers.

Table 5-1: (Continued) Alarm List

ALARM NO.	DEFINITION	REACTION	REMEDY	SOLLUTION
700056 Air Pressure Failure	The spindle air-oil air pressure is too low.	Display alarm. Machine continues to run.	Check incoming air pressure. Check spindle air-oil air pressure sensor.	Press the RESET key. Contact Service if alarm lingers.
700057 Oil Pressure Failure	The spindle air-oil oil pressure is too low.	Display alarm. Machine continues to run.	Check oil pressure when oiler is activated. Check spindle air-oil oil pressure sensor.	Press the RESET key. Contact Service if alarm lingers.
700058 Spindle Oiler Level Too Low	The spindle air-oil oiler needs to be refilled.	Display alarm. Machine continues to run.	Re-fill oiler reservoir. Check reservoir level sensor.	Press the RESET key. Contact Service if alarm lingers.

5.12 M CODES

Table 5-2: M code list

CODE	FUNCTION	NOTES
M0	Unconditional NC stop	
M1	Conditional NC stop	
M2	NC program end	
M3	Spindle CW	
M4	Spindle CCW	
M5	Spindle Stop	
M6	Execute tool change	
M7	Activate Coolant 1	
M8	Activate Coolant 2	
M9	Stop all coolant	
M11	Set current tool pocket number to 1	Used to re-synchronize tool magazine
M19	Orient tool and lock	
M31	Exchange pallet	Reserved
M32	Call pallet A	Reserved
M33	Call pallet B	Reserved
M34	Unload pallet	Reserved
M40	Spindle auto gear range change	
M41	Spindle change to low gear	
M42	Spindle change to high gear	
M52	Air/Oil oil Priming	Air/Oil oil pump cycled once
M60	Set M60/M61 output	Misc. M Function. Apply A axis brake.
M61	Reset M60/M61 output	Misc. M Function. Release A axis brake.
M62	Set M62/M63 output	Misc. M Function. Apply B axis brake.
M63	Reset M62/M63 output	Misc. M Function. Release B axis brake.
M64	Set M64/M65 output	Misc. M Function. Also probe 1
M65	Reset M64/M65 output	Misc. M Function. Also probe 2
M66	Set M66/M67 output	Misc. M Function
M67	Reset M66/M67 output	Misc. M Function

Table 5-2: (Continued) M code list

CODE	FUNCTION	NOTES
M68	Set M68/M69 output	Misc. M Function
M69	Reset M68/M69 output	Misc. M Function
M83	Spindle set-up	
M84	Spindle belt measurement	
M85	Activate A axis	
M86	Deactivate A axis	Used to remove rotary table
M87	Activate B axis	
M88	Deactivate B axis	Used to remove rotary table
M111	Lock spindle	Triggered by orientation magnet
M150	Actuate Auger/HydroSweep	

INDEX