Fadal 64MP

USER MANUAL

V1.0

Table of Contents

1 Introduction

This manual contains the basic operating information for the **Fadal 64 MP** control. This first chapter contains a brief description of the operator station and its switches and buttons. The second chapter contains descriptions for the various displays and their associated soft keys. The descriptions follow the order of the soft keys, not necessarily a typical order of usage. The third chapter contains descriptions for the typical operator sequences for common activities, such as running a program, setup a tool or work offset. The fourth chapter is the introduction of NC programming. The fifth chapter is for miscellaneous operator information and commands.

1.1 Physical Layout of Fadal 64 MP Operator's Station

As seen in Figure 1-1, the Fadal 64 MP operator's station includes the 15'' display, the standard Fadal Keyboard (with function keys), and the Fadal 64 MP Panel. The function keys are located below the screen and correspond to the soft keys of the screen accordingly.

Figure 1-1 Physical Layout of Fadal 64 MP

2 Basic Features and Operations of the CNC

This chapter describes the basic features and operations of the CNC according to each mode (or display). The modes include:

1. Auto
2. MDI
3. Jog
4. Edit
5. Tables
6. Utilities
7. Set Parameters
8. Diagnostics

As seen in Figure 2-1, the screen consists of status bar, main screen, sub-status bar, and soft-key bar. Only the main screen shall be changed when switching from modes (displays), those bar shall be always showing some important information.

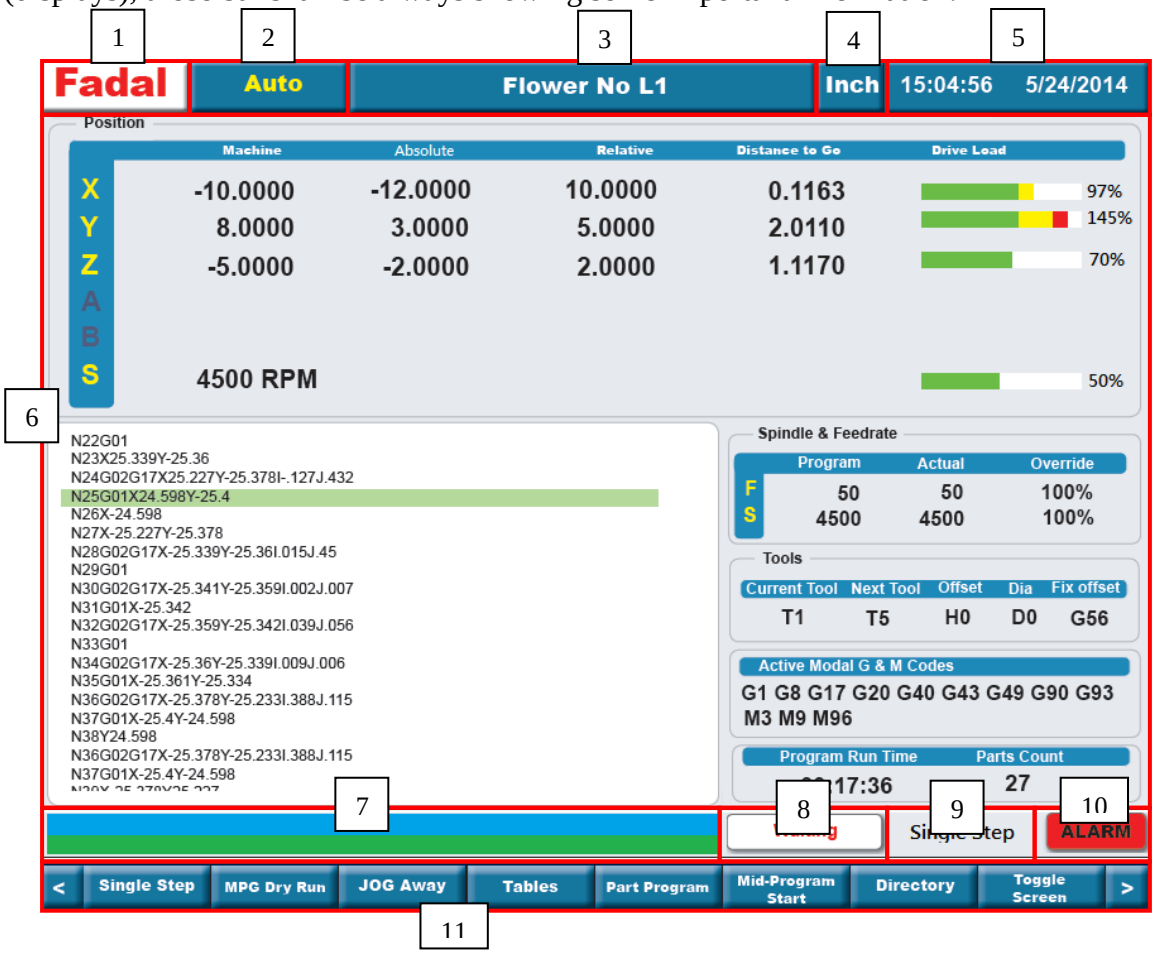


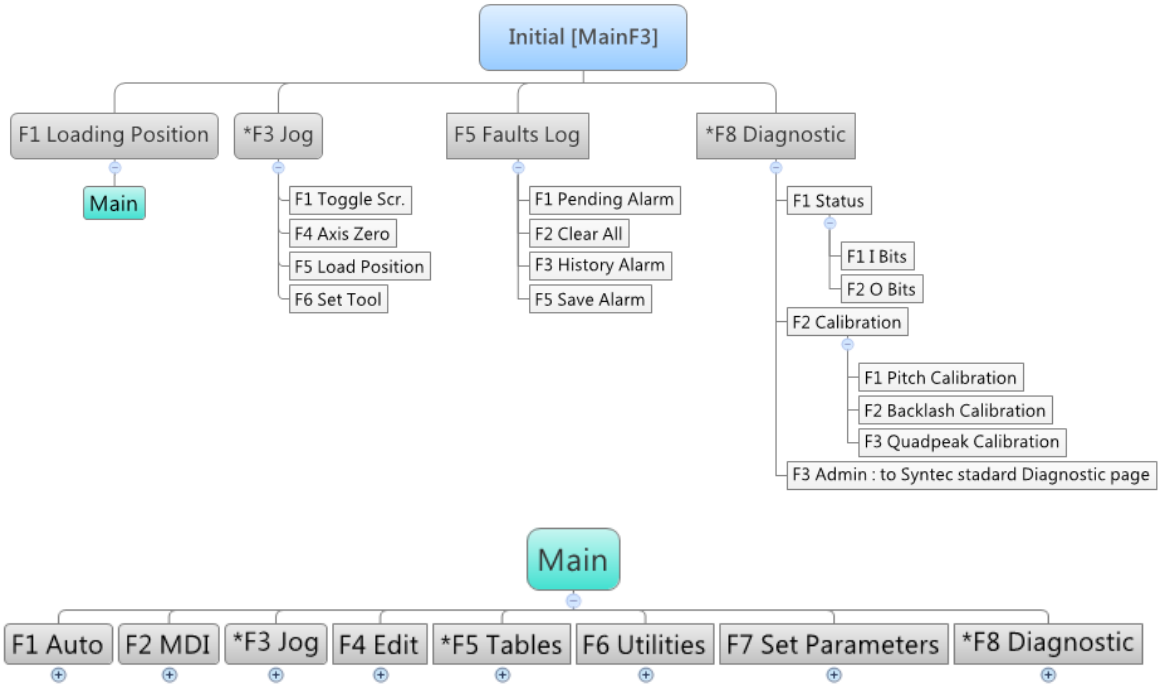
Figure 2-1 Introduction of the Screen

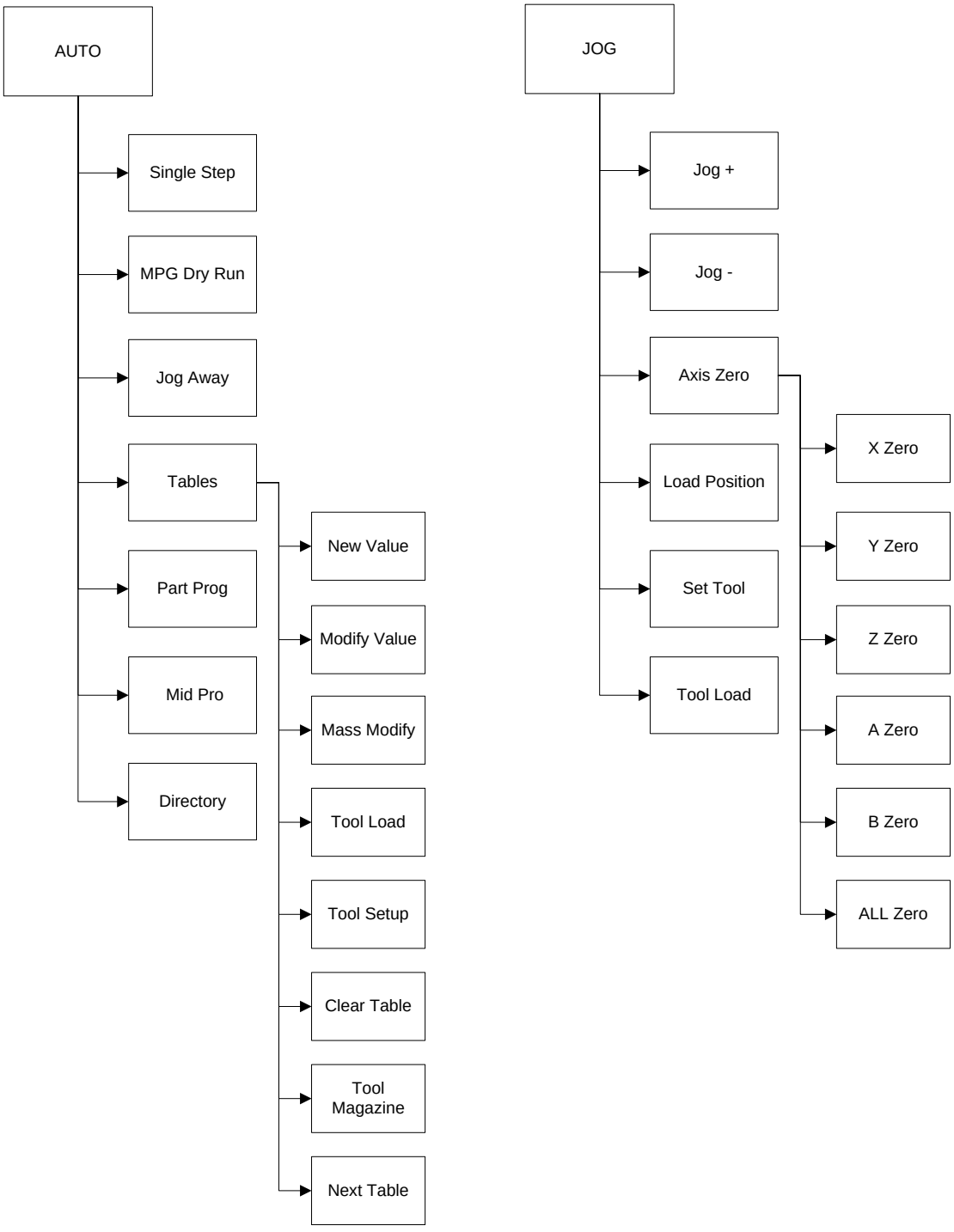
1. Status Bar : Fadal logo
2. Status Bar : Current Mode
3. Status Bar : Current Active Program ） ， Sequence No.(N) ， Line

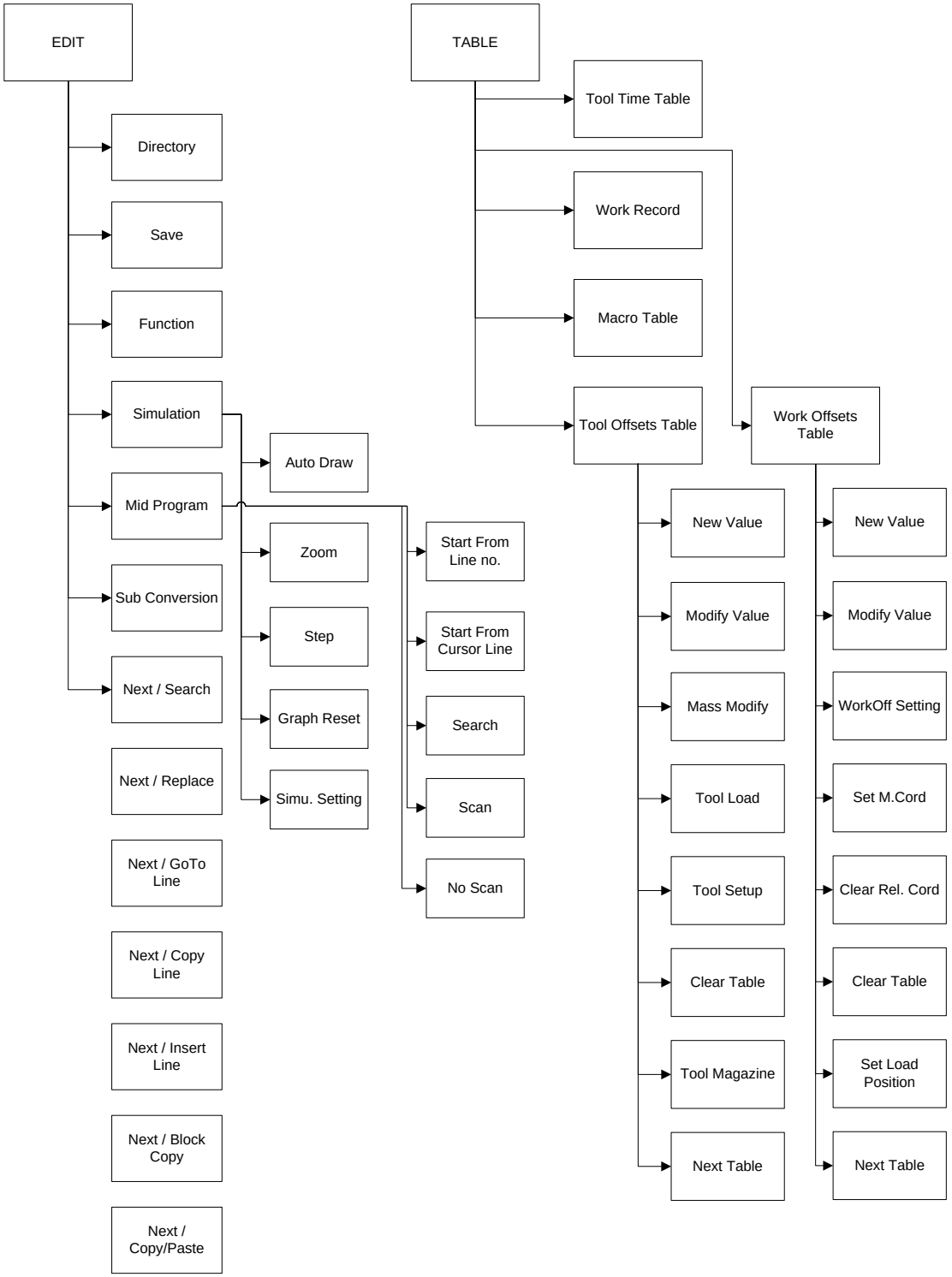
No.(L)

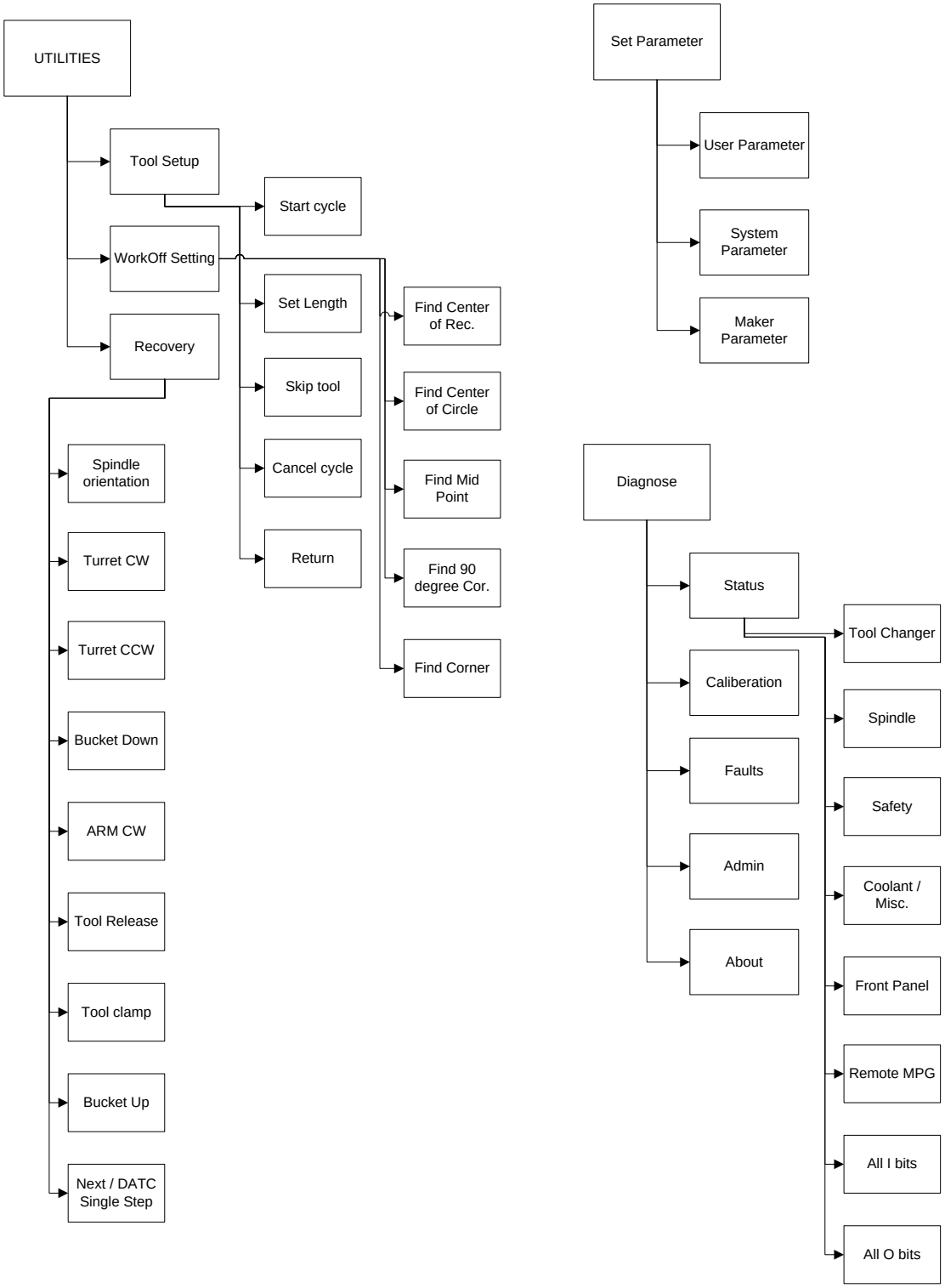
4. Status Bar : Inch/Metric
5. Status Bar : Time and Date
6. Main Screen
7. Sub-status Bar : Input Line & Hint Box
8. Sub-status Bar : Machine Status (Waiting/Running/SlideHold)
   - ⇨ Waiting : there is no faults occur currently and the machine is ready to execute the programs
   - ⇨ Running : the machine is currently executing a program
   - ⇨ SlideHold : the machine is holding at the middle of program execution
9. Sub-status Bar : Single Step/MPG Dry Run/None(blank)
   - ⇨ Single Step : Single Step status is enabled, i.e. the executing program will be execute one block at a time and stop at the end of the block
   - ⇨ MPG Dry Run : MPG Dry Run status is enabled, i.e. the machine shall execute the program according to the rotating speed of the Handwheel. CW rotation will run the program forward while CCW rotation will run the program backward.
   - ⇨ None(blank) : Neither Single Step nor MPG Dry Run is enabled
10. Sub-status Bar : Alarm Box (Flashing Red when alarm occurs)
11. Soft-key bar

Soft-key Bar tree:




AUTO
Single Step
MPG Dry Run
Jog Away
Tables
Part Prog
Mid Pro
Directory
New Value
Modify Value
Mass Modify
Tool Load
Tool Setup
Clear Table
Tool Magazine
Next Table
JOG
Jog +
Jog -
Axis Zero
Load Position
Set Tool
Tool Load
X Zero
Y Zero
Z Zero
A Zero
B Zero
ALL Zero


EDIT
Directory
Save
Function
Simulation
Mid Program
Sub Conversion
Next / Search
Next / Replace
Next / GoTo Line
Next / Copy Line
Next / Insert Line
Next / Block Copy
Next / Copy/Paste
Auto Draw
Zoom
Step
Graph Reset
Simu. Setting
Start From Line no.
Start From Cursor Line
Search
Scan
No Scan
TABLE
Tool Time Table
Work Record
Macro Table
Tool Offsets Table
Work Offsets Table
New Value
Modify Value
Mass Modify
Tool Load
Tool Setup
Clear Table
Tool Magazine
Next Table
New Value
Modify Value
WorkOff Setting
Set M.Cord
Clear Rel. Cord
Clear Table
Set Load Position
Next Table


UTILITIES
Tool Setup
Start cycle
WorkOff Setting
Set Length
Find Center of Rec.
Recovery
Skip tool
Find Center of Circle
Cancel cycle
Find Mid Point
Return
Find 90 degree Cor.
Find Corner
Spindle orientation
Turret CW
Turret CCW
Bucket Down
ARM CW
Tool Release
Tool clamp
Bucket Up
Next / DATC Single Step
Set Parameter
User Parameter
System Parameter
Maker Parameter
Diagnose
Status
Tool Changer
Caliberation
Spindle
Faults
Safety
Admin
Coolant / Misc.
About
Front Panel
Remote MPG
All I bits
All O bits

2.1 Initial Screen

Once the machine is powered up, the control shall check if the system is ready. It shall switch to the **main screen** if the system is ready. It shall remain on the **initial screen** if the system is NOT ready. In initial screen, the user can check on the machine status, change parameters, and jog the machine. The **initial screen** is as Figure 2-2 below:

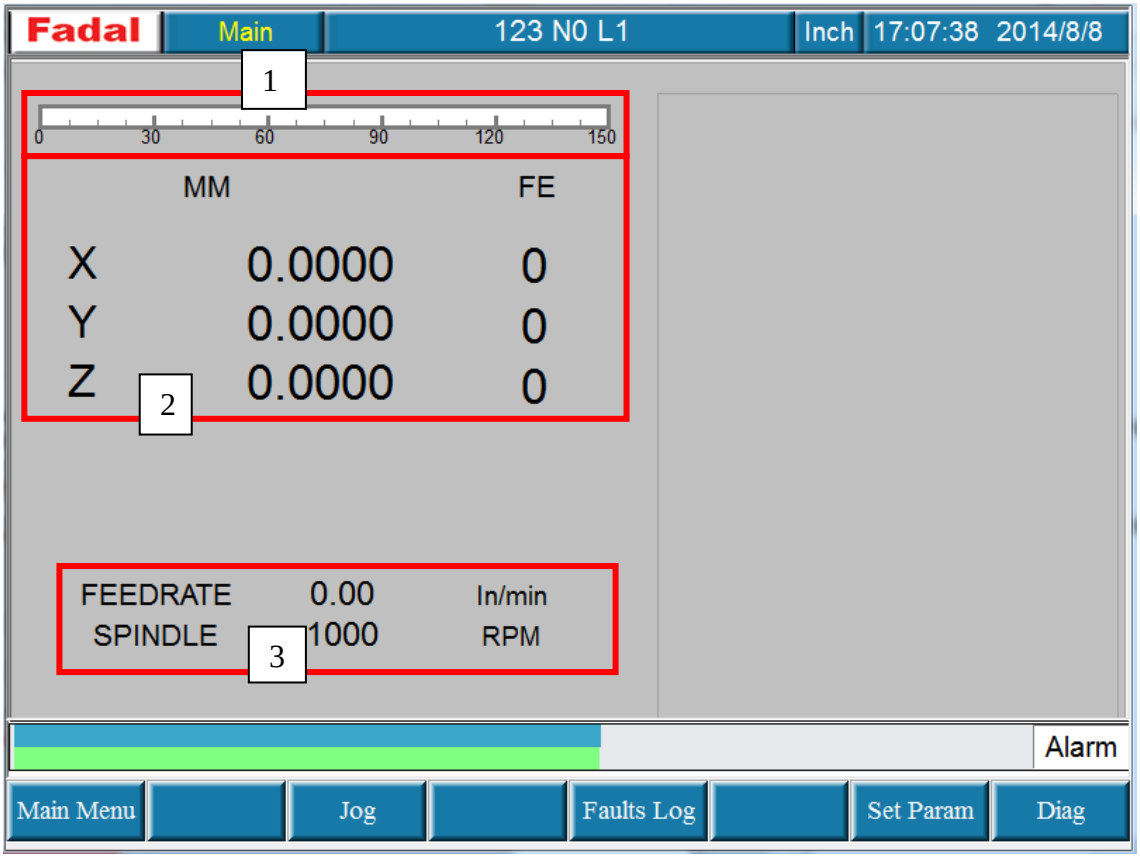


Figure 2-2 Initial Screen

1. Spindle Loading Bar
2. Machine Coordinate & Program Coordinate
3. Feedrate and Spindle RPM

2.1.1 Main Menu Softkey

Pressing or selecting the **Main Menu** softkey will switch to the main screen of the control.

2.1.2 Jog Softkey

Pressing or selecting the **Jog** softkey will switch to the JOG mode. For more description please refer to section 2.4

2.1.3 Faults Log Softkey

Pressing or selecting the **Faults Log** softkey will switch to the Pending Alarm screen. There are three softkey in this screen which are **Pending Alarm**, **History Alarm**, and **Save Alarm**.

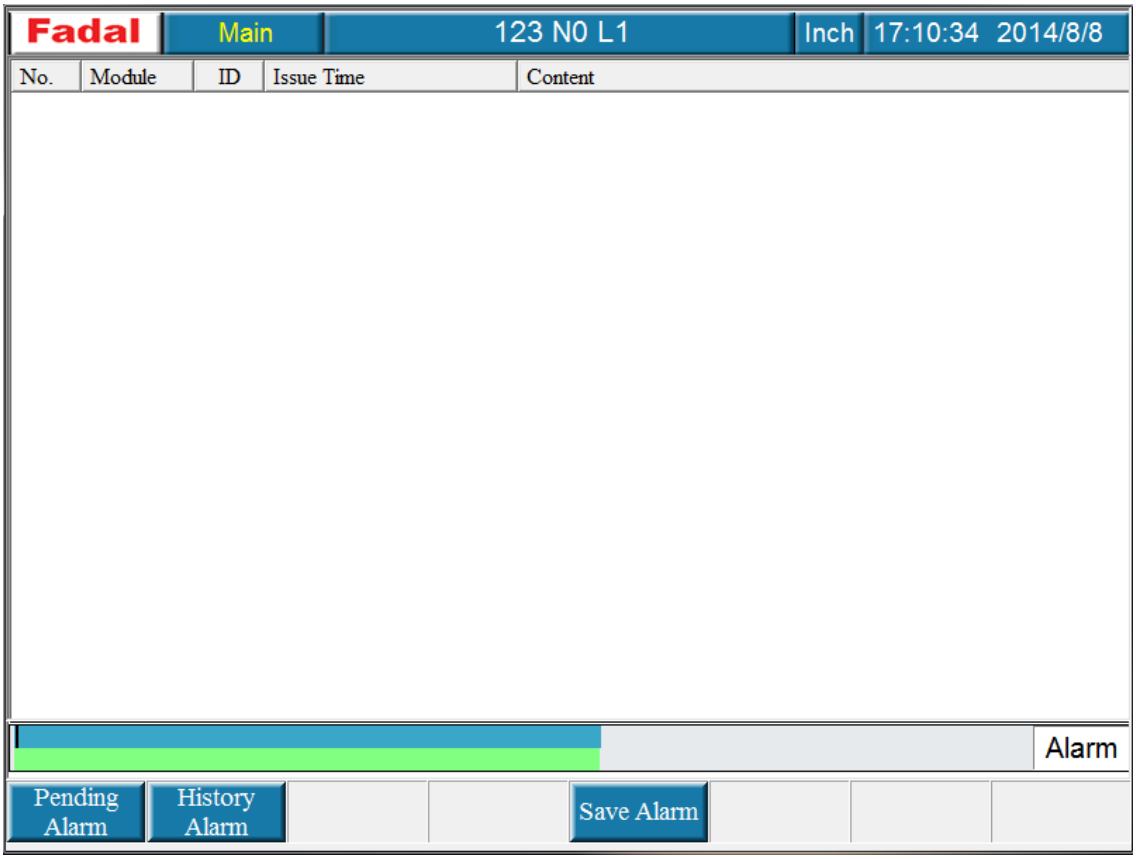


Figure 2-3 Pending Alarm screen

Pressing or selecting the **Pending Alarm** and **History Alarm** softkey will switch between Pending Alarm screen and History Alarm screen. In this alarm screen, there will be five column information: No, Module, Alarm ID, Issue Time, and Content.

Pressing or selecting the **Save Alarm** softkey will bring up a window allowing user to save the alarms to external devices (i.e. DiskA, USB drive, and Network). Un detected device shall have a red “cross” icon infront of it (as shown in Figure 2-4).

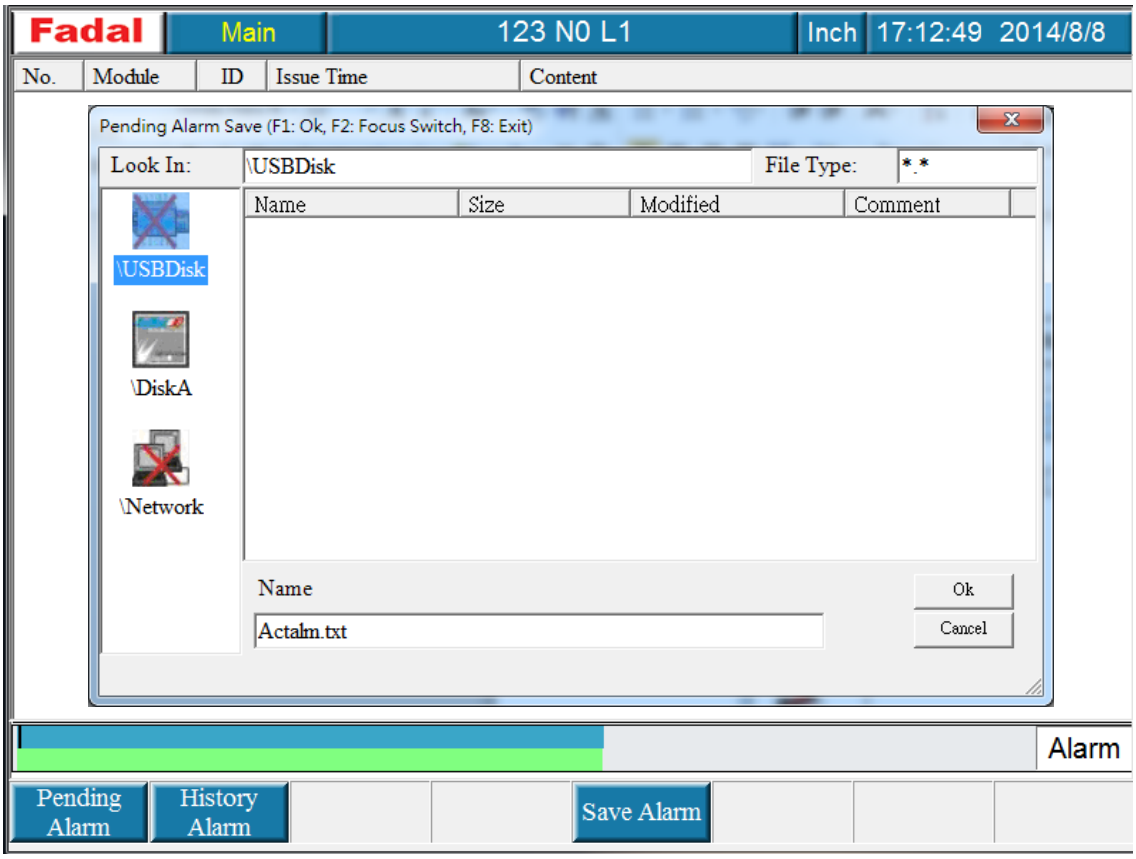


Figure 2-4 Save Alarm screen

2.1.4 Diagnostic Softkey

Pressing or selecting the **Diagnostic** softkey will switch to the diagnostic mode. For more description please refer to section 2.9.

2.2 Auto Screen

Auto screen allow user to choose and run a program. Pressing or selecting the **Auto** softkey on main function bar or *AUTO* button on the Fadal Keyboard will switch to the Auto mode. The auto screen is as shown below:

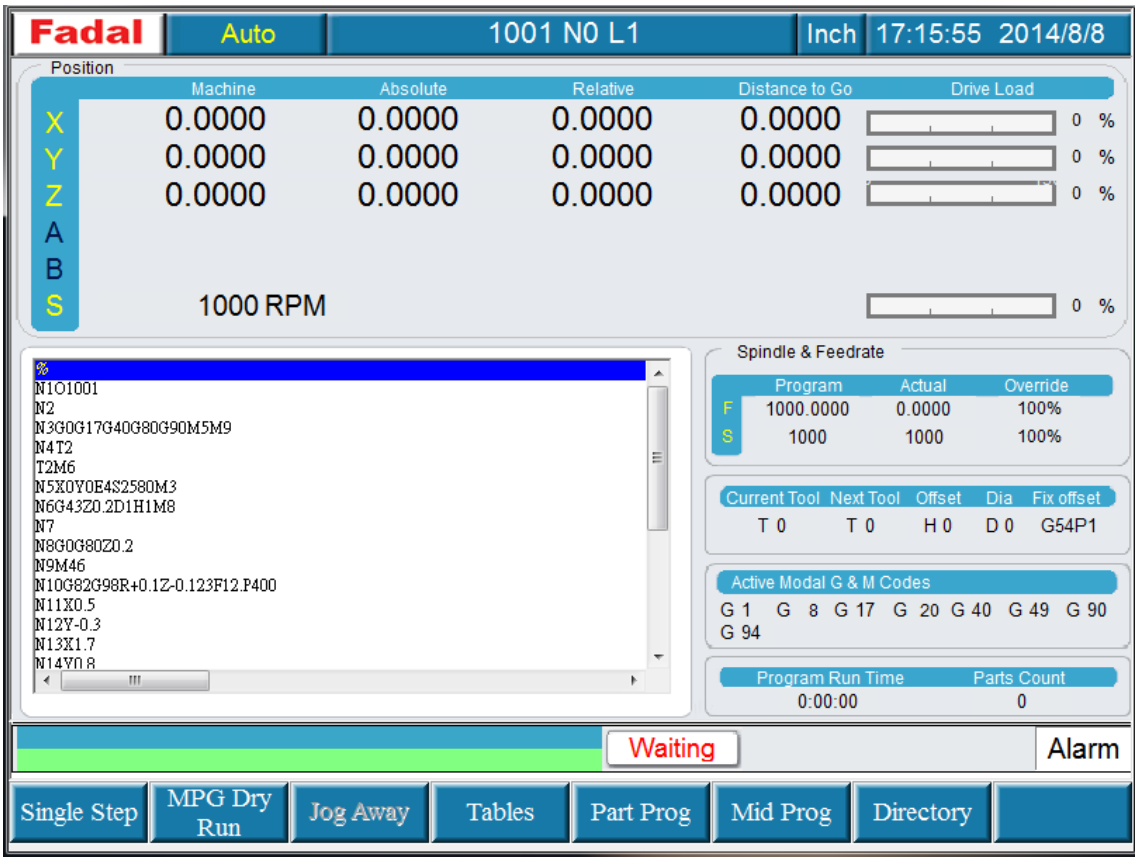


Figure 2-5 Auto screen (Waiting)

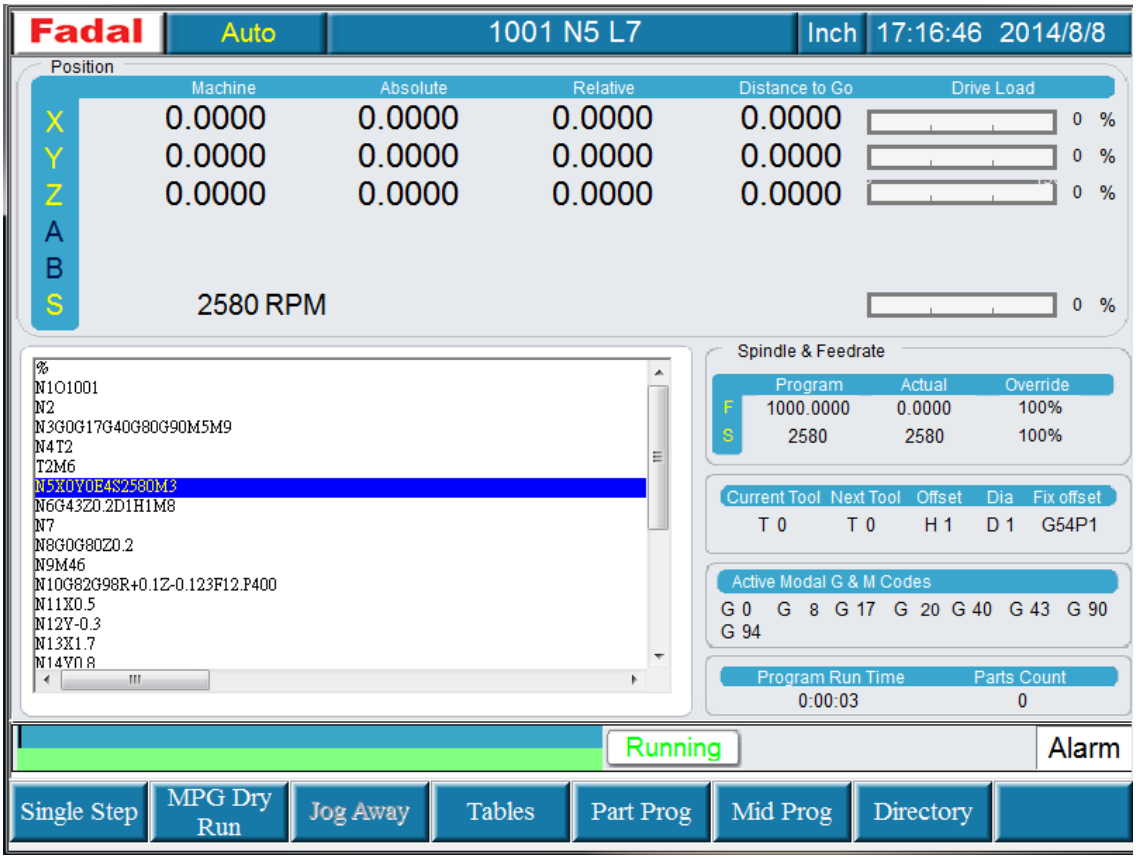


Figure 2-6 Auto screen (Running)

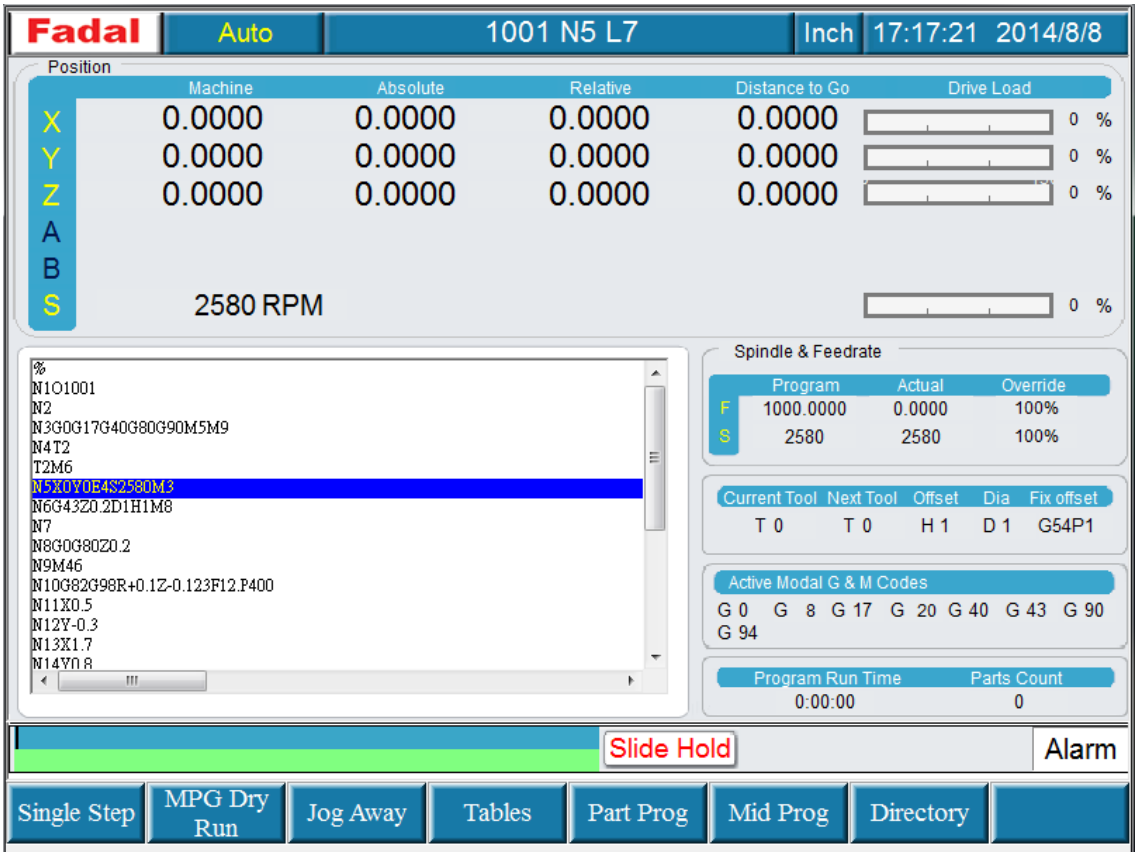


Figure 2-7 Auto screen (Slide Hold)

2.2.1 Single Step Softkey

Pressing or selecting **Single Step** softkey will disable all the other softkeys and activate “Single Step”. When “Single Step” is activated, the control shall execute one block at a time and stop at the end of the block. Pressing “Cycle Start” button will then execute the next block.

PS: Single step softkey is also to be used for canceling Single step feature.

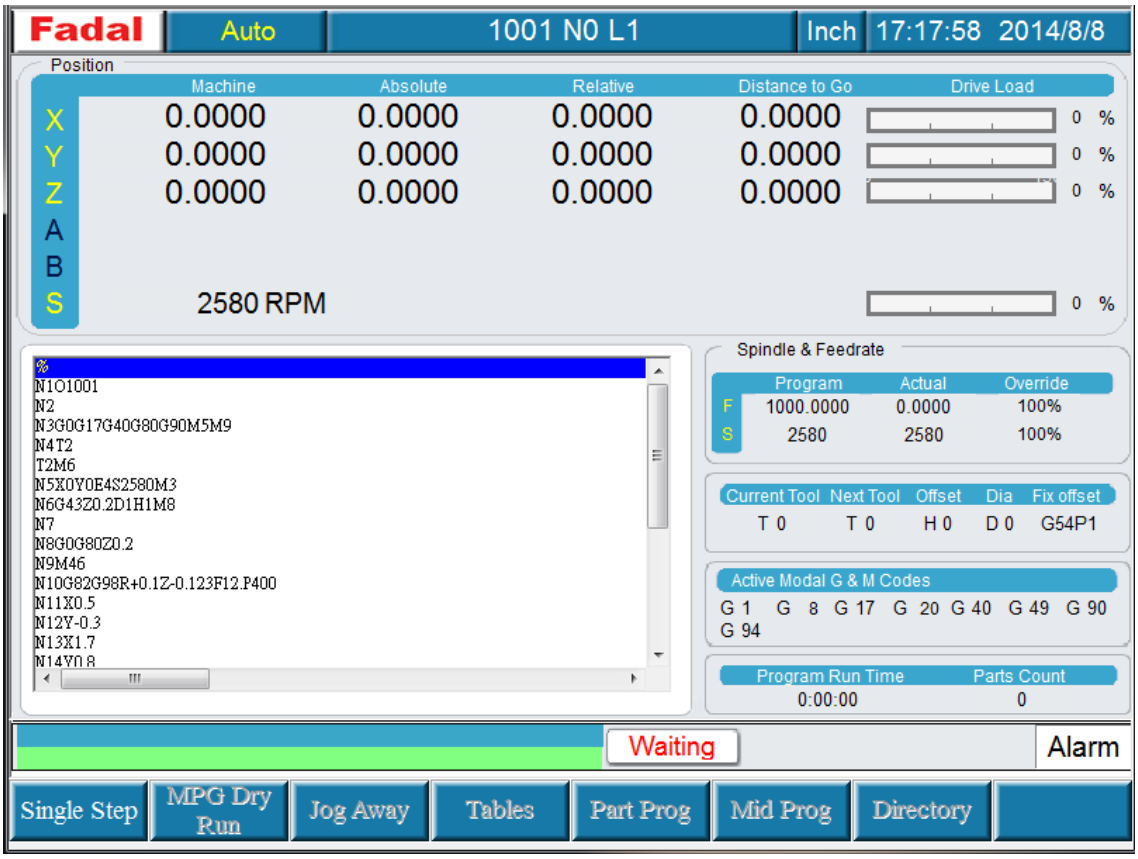


Figure 2-8 Auto screen (Single Step)

2.2.2 MPG Dry Run SoftKey

Pressing or selecting the **MPD Dry Run** softkey will allow the operator to execute the remainder of the program using the Handwheel. This feature can be used for part program check out and allow the operator to slowly move to an end point. The direction of the Handwheel will decide the direction of program executing, i.e. moving the Handwheel in the CW direction shall execute to the remainder of the program, moving the Handwheel in the CCW direction shall execute the program reversely. The speed that the axis (or axes) move is dependent on the speed of rotation of the Handwheel. The complete program can be executed under this feature. When an end of program (M2 or M30) is read the MPD mode is canceled. The operator can cancel the MPD mode by pushing or selecting the **Exit MPD** softkey. Upon exiting the MPD mode, the CNC shall return to the Auto mode and execute the remainder of the program automatically WITHOUT pressing “Cycle Start” button.

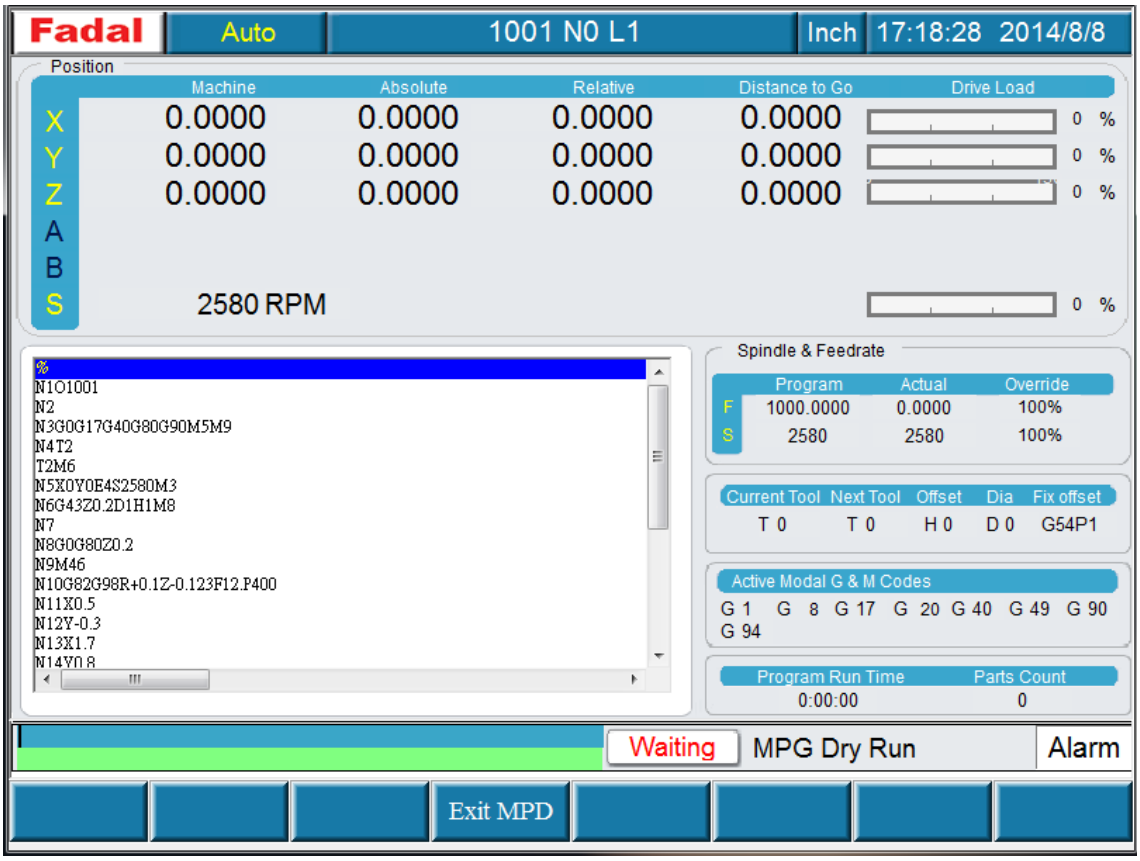


Figure 2-9 Auto screen (MPG Dry Run)

2.2.3 JogAway Softkey

This softkey will be enabled only when the machine is in Slide Hold status. Pressing or selecting the **Jog Away** softkey will allow the operator to move the tool tip away from the part for inspecting the tool, etc. When **Jog Away** is pushed, the axes will decelerate to a stand still (Slide hold Condition). Using the Handwheel, the operator can move any and all axes in any direction. Two softkeys are active when Jog Away is active:

Return Softkey

Pressing or selecting the **Return** softkey, will prompt the operator to push "Cycle Start" button to return the axes to the original coordinates in the part program where the axes were when **Jog Away** was selected. The move is a G1 speed move of X,Y and Z axes and G0 rapid move of Z axis. X and Y axes will move first if they were jogged away followed by the Z axis if it was moved. When the "Cycle Start" button is pushed, the axes will return to the original coordinates and continue running the remainder of the program automatically.

Remain and Shift Softkey

Pressing or selecting the **Remain and Shift** softkey, will return the system to the Auto mode. No motion will be executed. The amount or increment of moves performed by the Handwheel in the Jog Away mode will act as an offset for the remainder of the program until G28.1 is executed. This feature can be useful if additional or less stock is to be removed during a milling operation. Pressing the "Cycle Start" button after the

system has returned to the Auto mode will continue the part program execution.

2.2.4 Tables Softkey

Pressing or selecting the **Tables** softkey will switch to the Tables mode. For more description please refer to section 2.6.

2.2.5 Part Program Softkey

Pressing or selecting the **Part Program** softkey will enlarge the running program window and display a graphic simulation of the contour. As seen in Figure 2-10 below. Pressing the Part Program softkey again shall toggle to the previous screen.

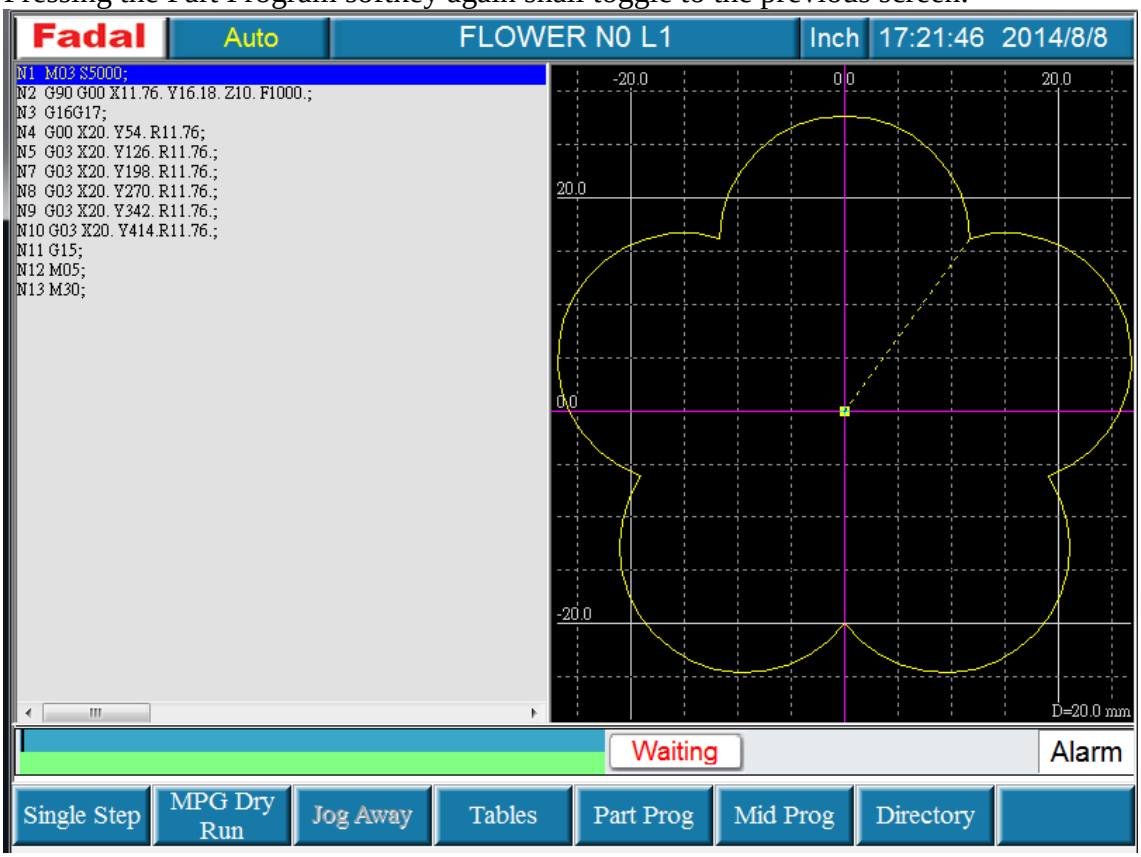


Figure 2-10 Auto screen (Part Program)

2.2.6 Mid Program Softkey

Pressing or selecting the **Mid Program** softkey will provide the operator with a method to start part program execution in the middle of a program. There are several options for starting a program in the middle. When **Mid Program** is selected the function bar shall switch to the other as shown below. There will be **Start From Line No.**, **Start From Cursor Line**, **Search**, **Scan** and **No Scan** softkeys.

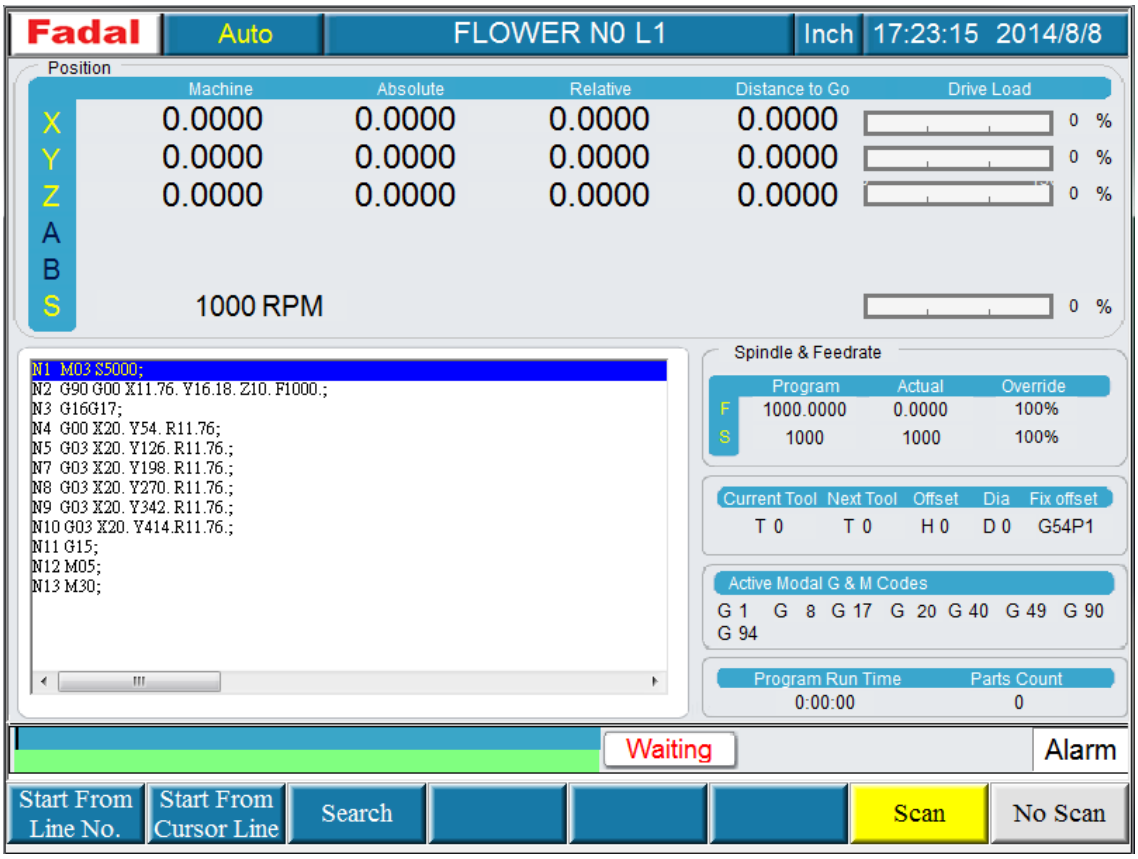
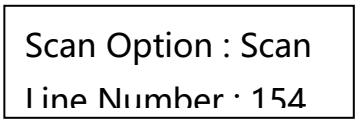


Figure 2-11 Auto Screen (Mid Program Start)

Start From Line No. Softkey

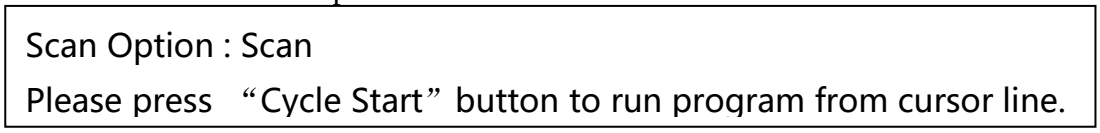
Pressing or selecting the **Start From Line No.** softkey will bring up a window which contains two information, i.e. Scan Option and Line Number. The Scan Option declare the mid program start status (Scan/No Scan), the line number defines the starting line of the mid program, its default is the cursor line. Below is an example:



Pressing *Enter* on the Fadal Keyboard shall close the pop-up window and start the scanning (if the status is Scan), push the "Cycle Start" button after that shall execute the program from that specific line.

Start From Cursor Line Softkey

Pressing or selecting the **Start From Cursor Line** softkey will bring up a window which contains the scan option status. As shown below.



Pressing *Enter* on the Fadal Keyboard shall close the pop-up window and start the scanning (if the status is Scan), push the "Cycle Start" button after that shall execute the program from that specific line.

Search Softkey

Pressing or selecting the **Search** softkey will bring up a window which contains inputbox for user to type the desired string to search. Once *Enter* on the Fadal Keyboard is pressed the control shall move the cursor to the first identified matching string. If this is not the line that user wish to start execute the program, "↑", "↓", "PageUp", and "PageDown" can be used to move the cursor.

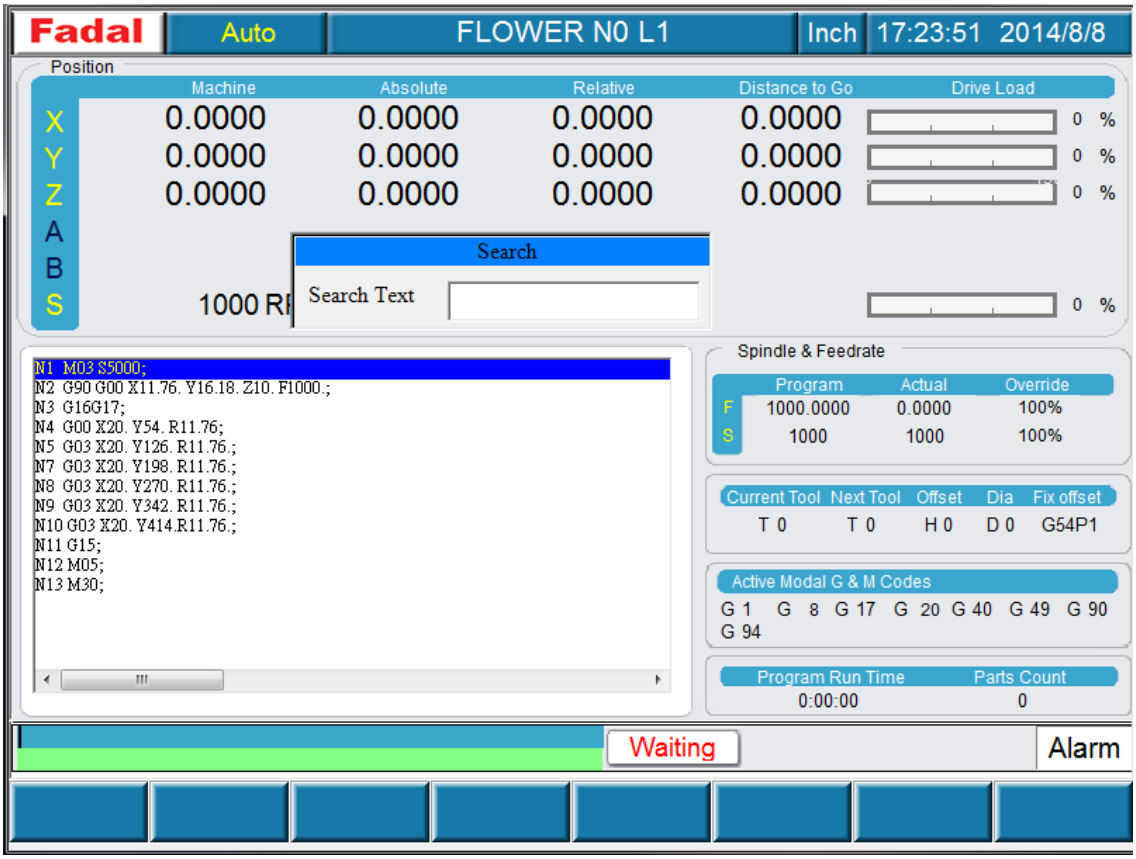


Figure 2-12 Mid Program Start - Search

Scan/No Scan Softkey

Pressing or selecting the **Scan/No Scan** softkey will change the current status to Scan/No Scan. In the Scan status, the control shall scan through the modal codes and pick up any model code to the start point. When the program is started, those previously read modal codes will be active. In contrast, No Scan status will start running the program without any previous actived modal code information.

2.2.7 Directory Softkey

Pressing or selecting the **Directory** softkey will bring up a window and show the programs in internal storage. User can move to the desired program and choose it as an active program by pressing *Enter*. To search for a desired program, user can use the "↑", "↓", "PageUp", and "PageDown" key, or type the name of the program directly. As shown in the Figure 2-13 below, user type "BORING_G88" shall move the cursor to that program.

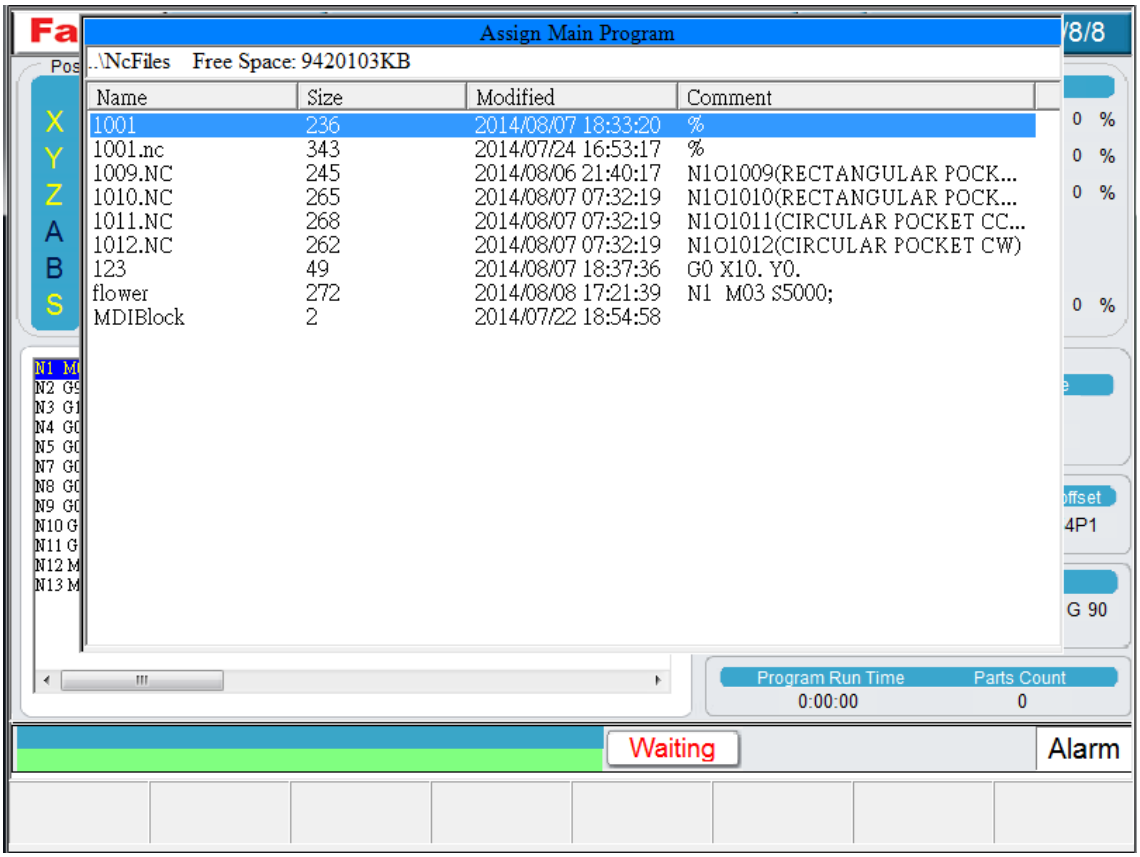


Figure 2-13 Auto Screen (Directory)

2.3 MDI Screen

MDI mode allow user to perform some tests or moving of the machine through a few manually typed program. Pressing or selecting the **MDI** softkey on main function bar or *MDI* button on the Fadal Keyboard, there a windows will pop up which is asking “Press Yes to MDI input”, after pressing Yes the screen will switch to the MDI mode. A page similar to the one shown in Figure 2-14 will be seen when the control is in MDI Mode.

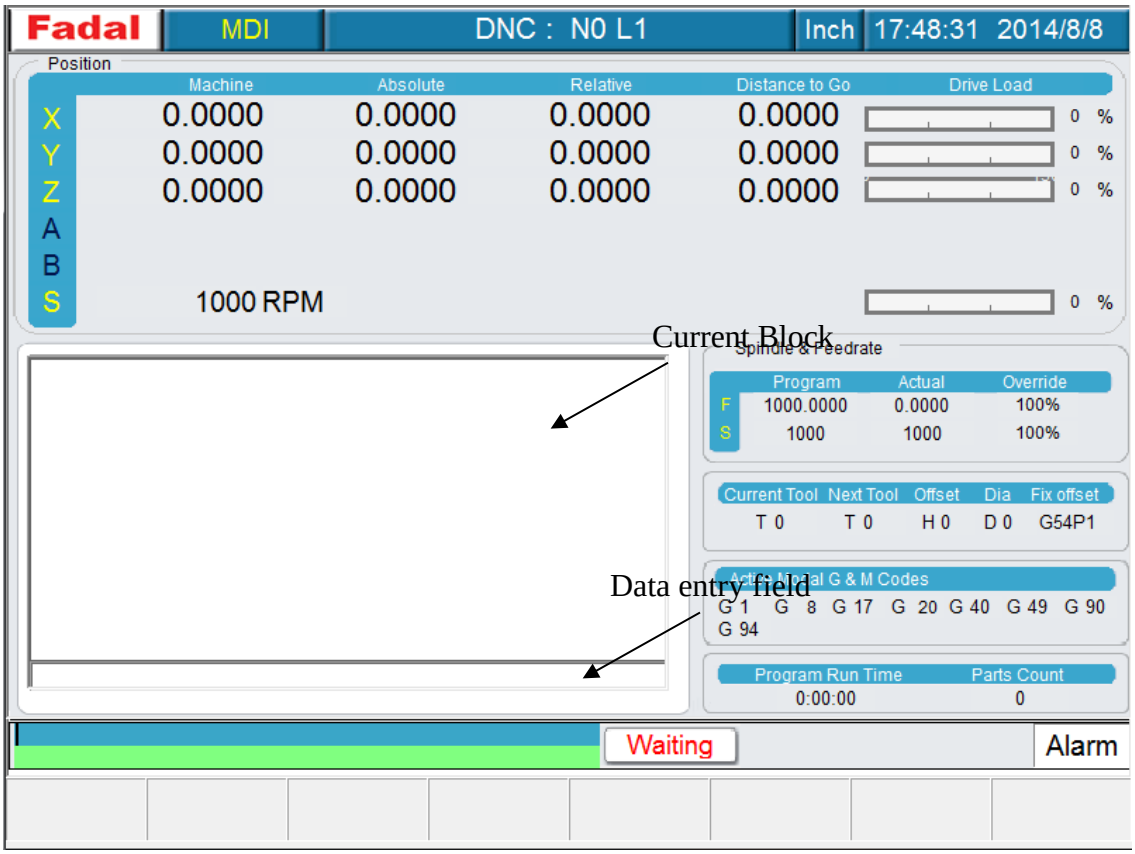


Figure 2-14 MDI Screen (1)

The mode display will change to “MDI” once MDI mode is activated, with the status display “Waiting” initially. Pressing **MDI Input** will move the focus to the data entry field. The cursor will be blinking in the data entry field (Figure 2-15). The operator enters the required syntax for the desired action. When *Enter* is pressed the block moves to the bottom of the list. The CNC will execute the blocks in the order they are entered. It is possible to enter several blocks ahead and if the CNC is in continuous mode, the block will be executed sequentially without stopping.
After the block data is entered and the *Enter* is pushed, the block moves into the queue and is ready to be executed. Pressing “Cycle Start” button will move the status of the system to “Running” and the block will execute. The system will remain in “Running” unless interrupted by “Feedhold”, Reset button, or a machine fault or Emergency Stop. If the system status is “Running” each additional block “Entered” will be executed without a “Cycle Start”. If another main operation mode is selected with data in a buffer block, the data will be lost. Figure 2-15 is an example of MDI Screen.

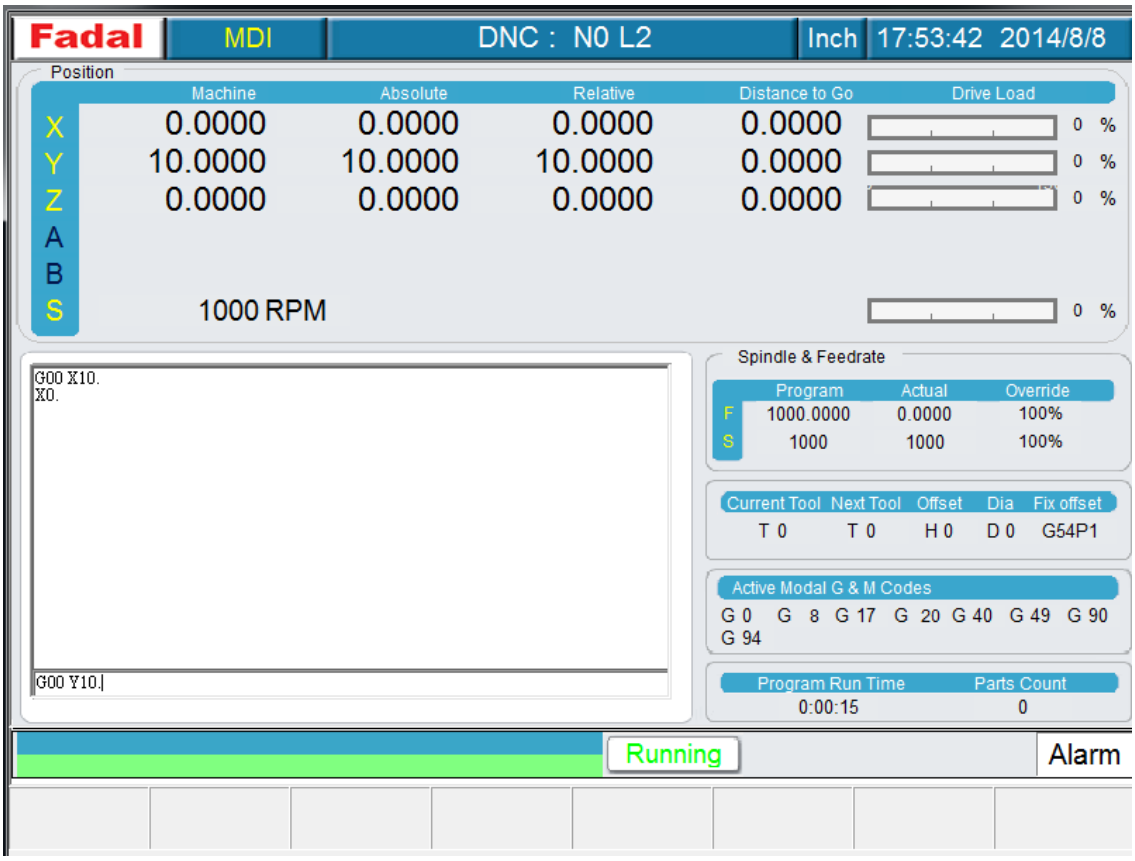


Figure 2-15 MDI Screen (2)

In this example, “G00 X10.” and “X0.” have been entered to the field and executed by the control, without a program ending code (i.e. M30 M2 etc.) nor a *Reset* command the system will still under “Running” status. Yet the user entered “G00 Y10.” at the data entry field, pressing *Enter* will move “G00 Y00” to the execute list block and will be executed directly without any further action.

2.4 Jog Screen

Jog mode allow user to manually move axes with Handwheel or the *JOG* key on keyboard. Pressing or selecting the **Jog** softkey on main function bar will switch to the Jog mode. A page similar to the one shown in Figure 2-16 will be seen when the control is in Jog Mode.

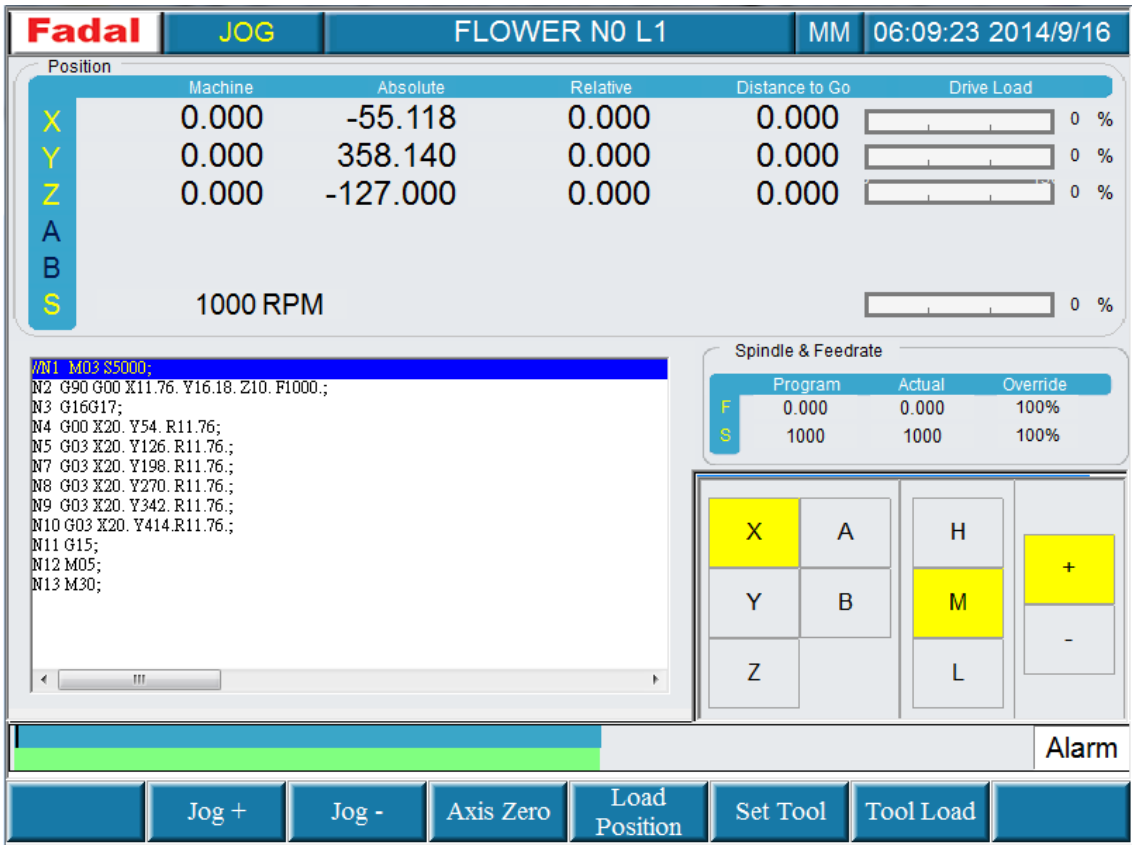


Figure 2-16 Jog Screen

There is an axis selector switch and a step size selector switch. The active axis, active step size, and active direction will be highlighted on the screen. The step size imply the moving speed of the axes : H/M/L -> 100%/10%/1% (note: 100% moving speed can be define under user parameters)

2.4.1 Jog + Softkey

Pressing or selecting the **Jog+** softkey will set the current active direction to positive direction.

2.4.2 Jog - Softkey

Pressing or selecting the **Jog-** softkey will set the current active direction to negative direction.

2.4.3 Axis Zero Softkey

Pressing or selecting the **Axis Zero** softkey will switch to another function bar as shown below:

Selecting the softkey will bring the corresponding axis to machine zero position, after pressing START button.

2.4.4 Load Position Softkey

Pressing or selecting the **Load Position** softkey will bring the axes to a pre-set position. The position can be define under user parameters or work offset table.

2.4.5 Set Tool Softkey

Pressing or selecting the **Set Tool** softkey will bring a window as shown in Figure 2-17. Insert a value then pushing *Enter* will input that value directly to the corresponding tool length, i.e. the tool length of the tool in spindle. Default value of the box will be current Z axis machine position.

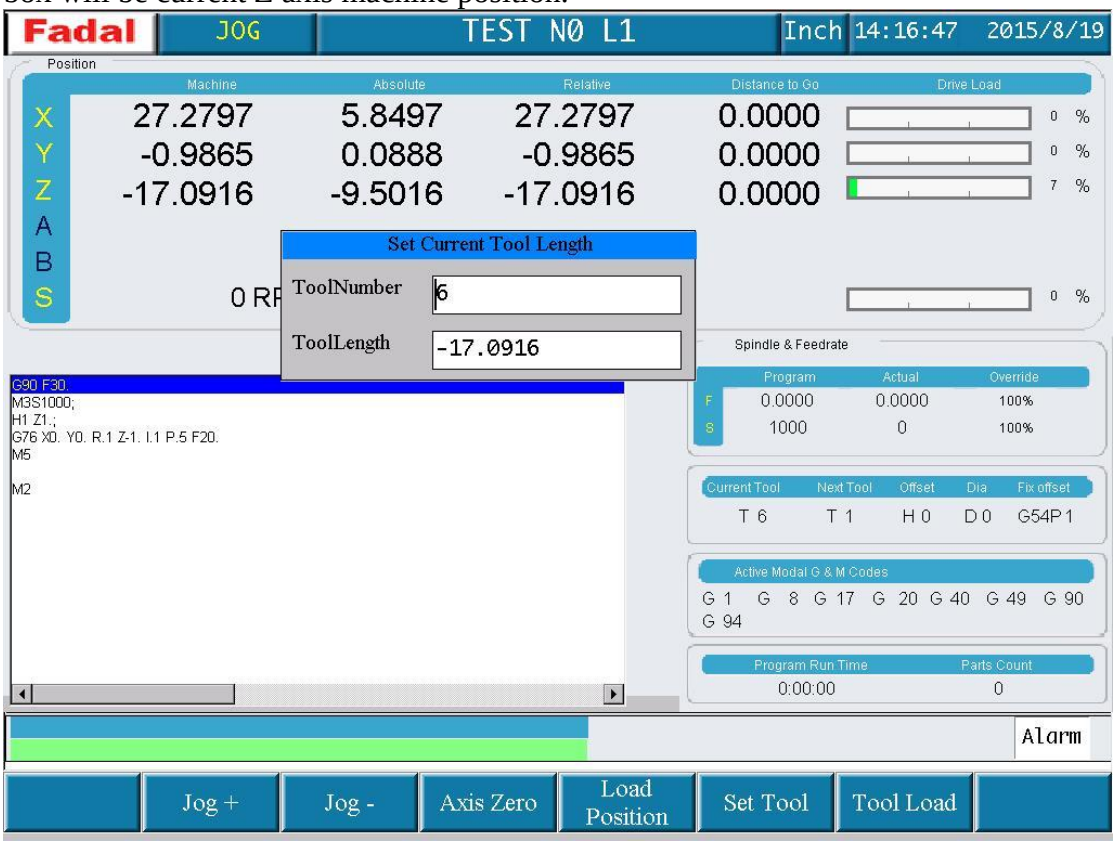
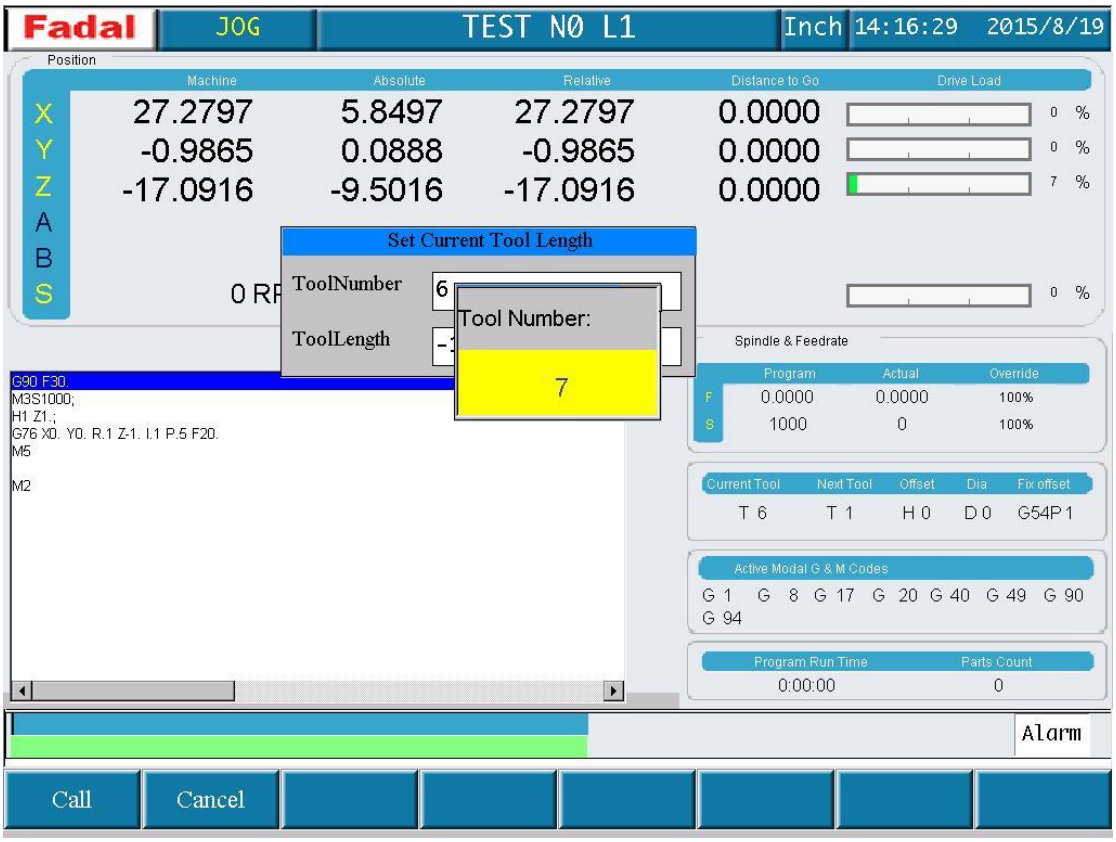


Figure 2-17 Jog Screen – Set Tool

2.4.6 Tool Load Softkey

Pressing or selecting the **Tool Load** softkey will bring a window as shown in below. Insert designated Tool number in Tool number windows and pushing *Enter* to input that value and Press Call softkey to run tool change procedure directly WITHOUT pressing any confirmation button.



2.5 EDIT Screen

Edit mode allow user to call/create a program to modify/edit. Pressing or selecting the **Edit** softkey on main function bar or the *EDIT* on the Fadal Keyboard will switch to the Edit mode. A page similar to the one shown in Figure 2-18 will be seen when the control is in Edit Mode.

Figure 2-18 Edit Screen

There will be two function bars under Edit screen, pressing “next” and “back” softkey can switch between them. As seen in Figure 2-18, the content of program is divided to different color as:

i. M/S/T code – Green
ii. G code – Blue
iii. Axes (X/Y/Z/A/B) – Black
iv. N word – Brown
v. F word – Purple
vi. Others – Black

The ground color will become yellow once modified, as N4 of Figure 2-18. This yellow ground color will be keep until saved.

2.5.1 Directory Softkey

Pressing or selecting the **Directory** softkey will bring a window as shown in Figure 2-19.

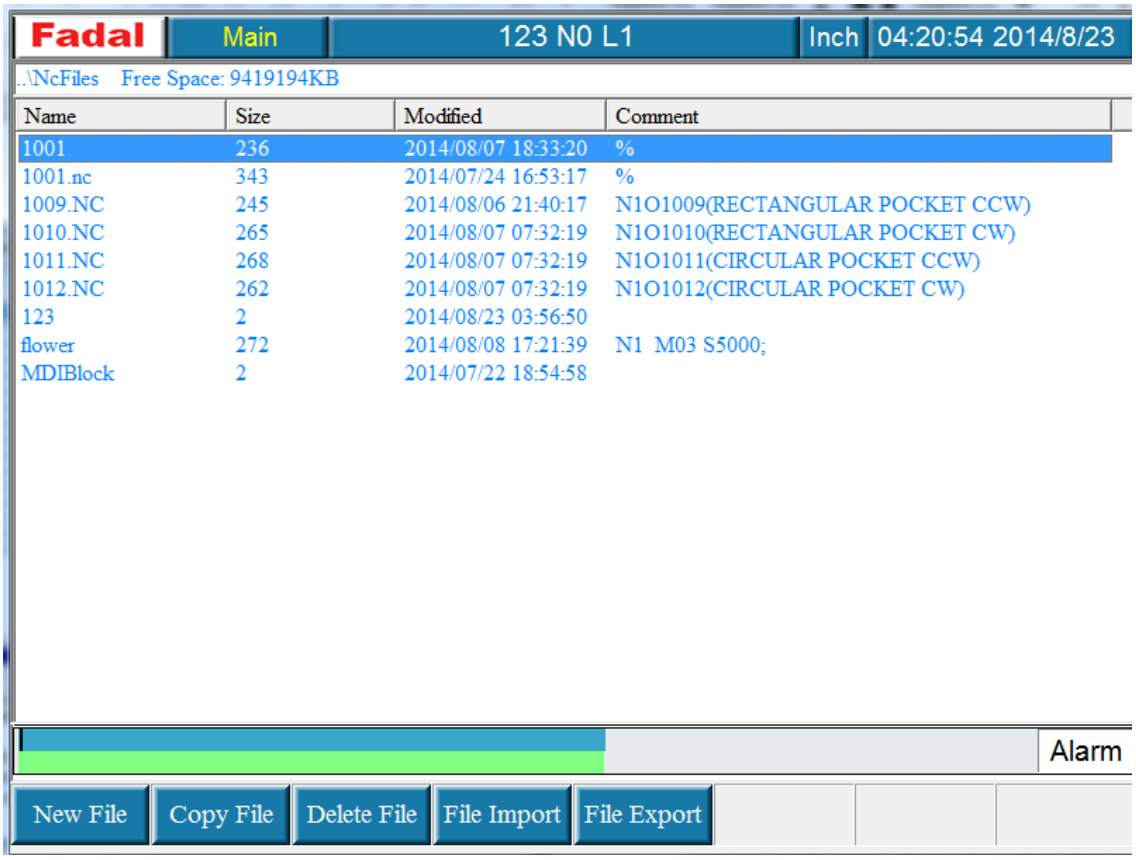


Figure 2-19 Edit Screen – Directory

User can moves the cursor with "↑""↓""Func+↑(Page UP)"" Func+↓(Page Down)" to disired program. Or type the name of the program directly will jump to the specific file. Pressing *Enter* will define the program as active program to be edit.

2.5.2 Save Softkey

Pressing or selecting the **Save** softkey will perform a "save" action, i.e. to write the modified program to the memory of the control.

2.5.3 Simulation Softkey

Pressing or selecting the **Simulation** softkey will bring up a window to show the graphic simulation of the program. As seen in Figure 2-20 below.

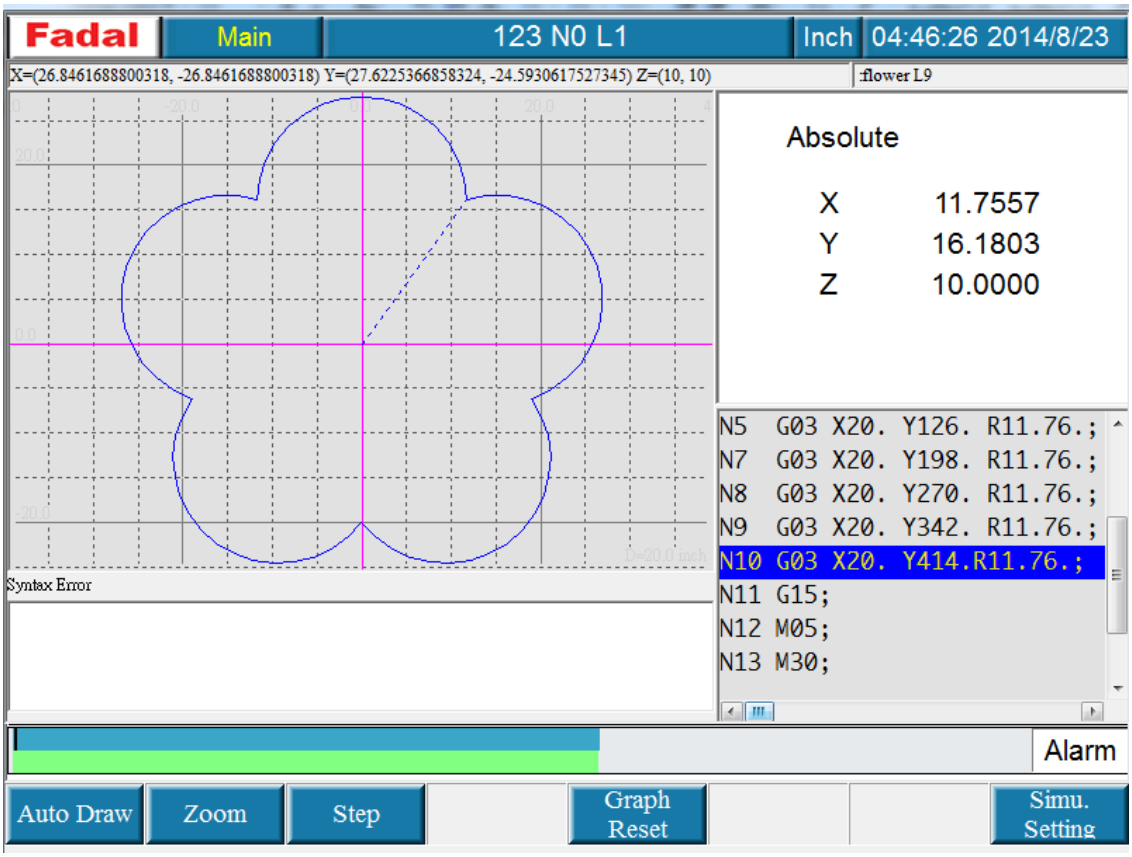


Figure 2-20 Edit Screen – Graphic Simulation

This graphic simulation screen is divided into four section: Graphic simulation section, Program coordinate section, Syntax error checking section, and Active program content section. If there is a syntax error, graphic simulation will not be perform and a syntax error message will be displayed on the syntax error checking section. Pressing **Auto Draw** softkey will re-simulate the graphic. Pressing **Zoom** softkey allow user to zoom in to specific window. Pressing **Step** softkey allow user to simulate the program step by step. Pressing **Graph Reset** softkey will auto adjust the range of the graphic simulation window. Pressing **Simu. Setting** softkey will bring up a window to perform some graphic simulation parameters setting.

2.5.4 Mid Program Softkey

Pressing or selecting the **Mid Program** softkey will switch to Auto mode and enter the Mid Program screen. For more detail please refer to section 2.2.6.

2.5.5 Subroutines Conversion

Pressing or selecting the **Sub. Conversion** softkey will bring a window which allow user to convert the position of subroutines in the files between format 1 and format2.

2.5.6 Search Softkey

Pressing or selecting the **Search** softkey will perform a general search function.

2.5.7 Replace Softkey

Pressing or selecting the **Replace** softkey will perform a replace function.

2.5.8 Goto Line Softkey

Pressing or selecting the **Goto Line** softkey will bring up a window to allow user to input a line number. Pressing *Enter* will move the cursor to that specific line.

2.5.9 Copy Line Softkey

Pressing or selecting the **Copy Line** softkey will copy the current line and paste it to a newly inserted line.

2.5.10 Insert Line Softkey

Pressing or selecting the **Insert Line** softkey will insert a new line to the below of cursor postion.

2.5.11 Block Copy Softkey

Pressing or selecting the **Block Copy** softkey allow user to copy a block program and paste it on desired position.

2.5.12 Copy/Paste Softkey

Pressing or selecting the **Copy/Paste** softkey will perform a copy/paste function.

2.6 Tables Screen

Tables mode allow user to check or set some cutting information such as tool length, work offset, and etc. This mode can be reach by pressing the **Tables** on the function bar or *Tables* on the Fadal Keyboard. There will be four tables in the control, which as:

i. Tool Offset Table
ii. Work Offset Table
iii. Macro Table
iv. Tool Time Table
v. Work Record

2.6.1 Tool Offset Table

Figure 2-21 shows the Tool offset. There are 96 entries in the Tool table. For every tool (1 to 96) there is the corresponding data:

Tool Information	Description
Length	Used with length offset – called with H word
Radius	Used for Cutter Radius Compensation CRC – called with D word
Wear of Length	Compensation of tool length
Wear of radius	Compensation of tool radius

Description	Description of tool, can be any string

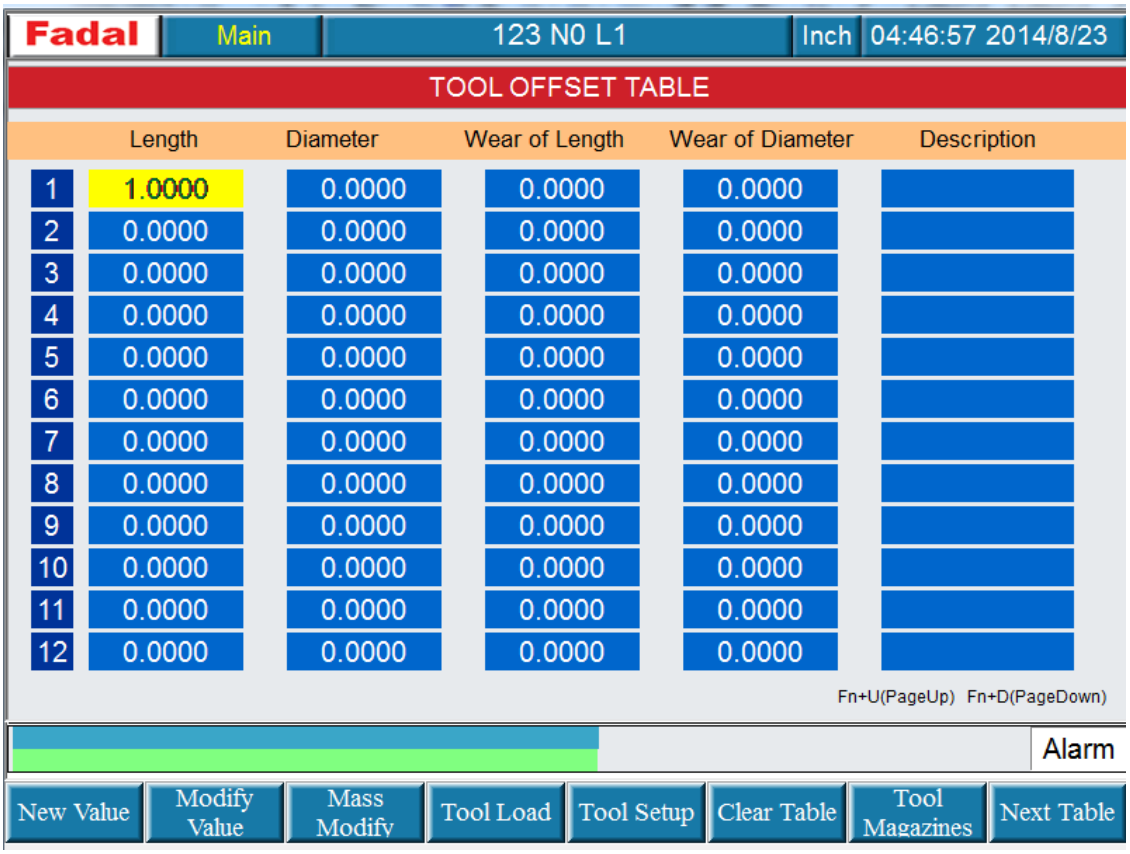


Figure 2-21 Tables – Tool Offset Table

2.6.1.1 New Value Softkey

Pressing of selecting **New Value** softkey will bring up a dialog box that will allow the user to input data or values for the selected field (cursor position). Pressing *Enter* will accept the entered data/values into the selected field. As seen in Figure 2-22, the selected field is tool length of tool no.1, and the user has entered "-5.23" into the dialog box, pressing *Enter* will insert that value to the tool length of tool no.1.

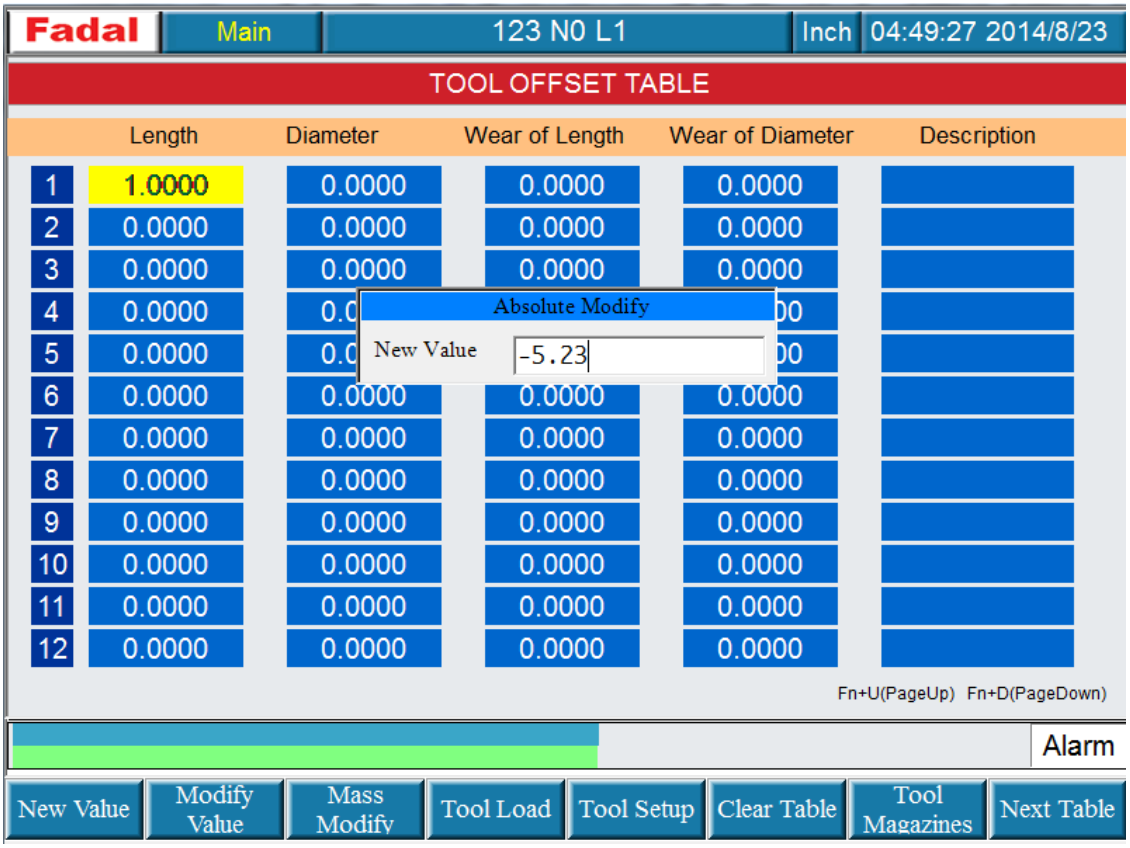


Figure 2-22 Tool Offset Table – New Value

2.6.1.2 Modify Value Softkey

Pressing of selecting **Modify Value** softkey will bring up a dialog box that will allow the user to modify data or values for the selected field (cursor position). Pressing *Enter* will accept the entered data/values into the selected field. As seen in Figure 2-23, the selected field is tool length of tool no.1 which current value is "-5.23", and the user has entered "1." into the dialog box, pressing *Enter* will modify that value to the tool length of tool no.1 to "-4.23".

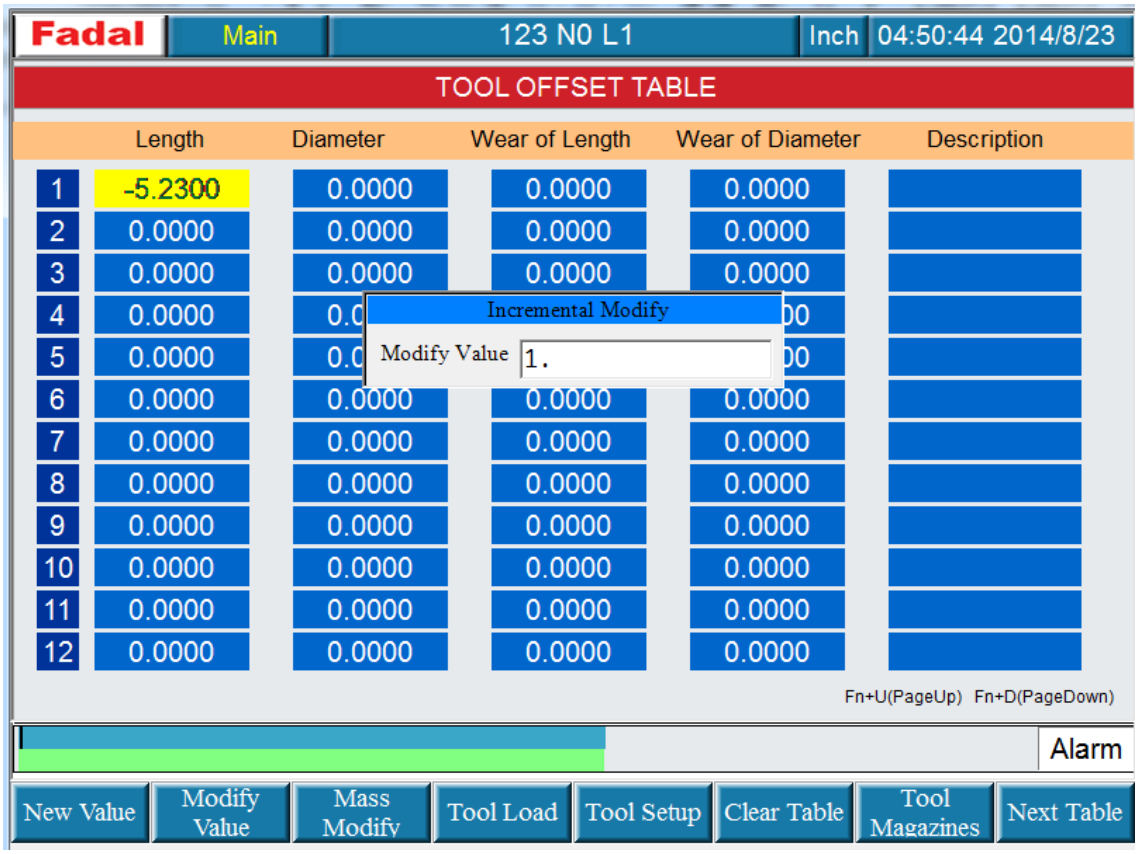


Figure 2-23 Tool Offset Table – Modify Value

2.6.1.3 Mass Modify Softkey

Pressing of selecting **Mass Modify** softkey will bring up a dialog box that allow the user to modify data or values for the desire range of tool. Pressing *Enter* will accept the entered data/values into the entered range of field. As seen in Figure 2-24, the selected field is tool length, and the user has entered from "2" to "5" with a data/value of "-3.", pressing *Enter* will modify that value to the tool length of tool no.2 to no.5.

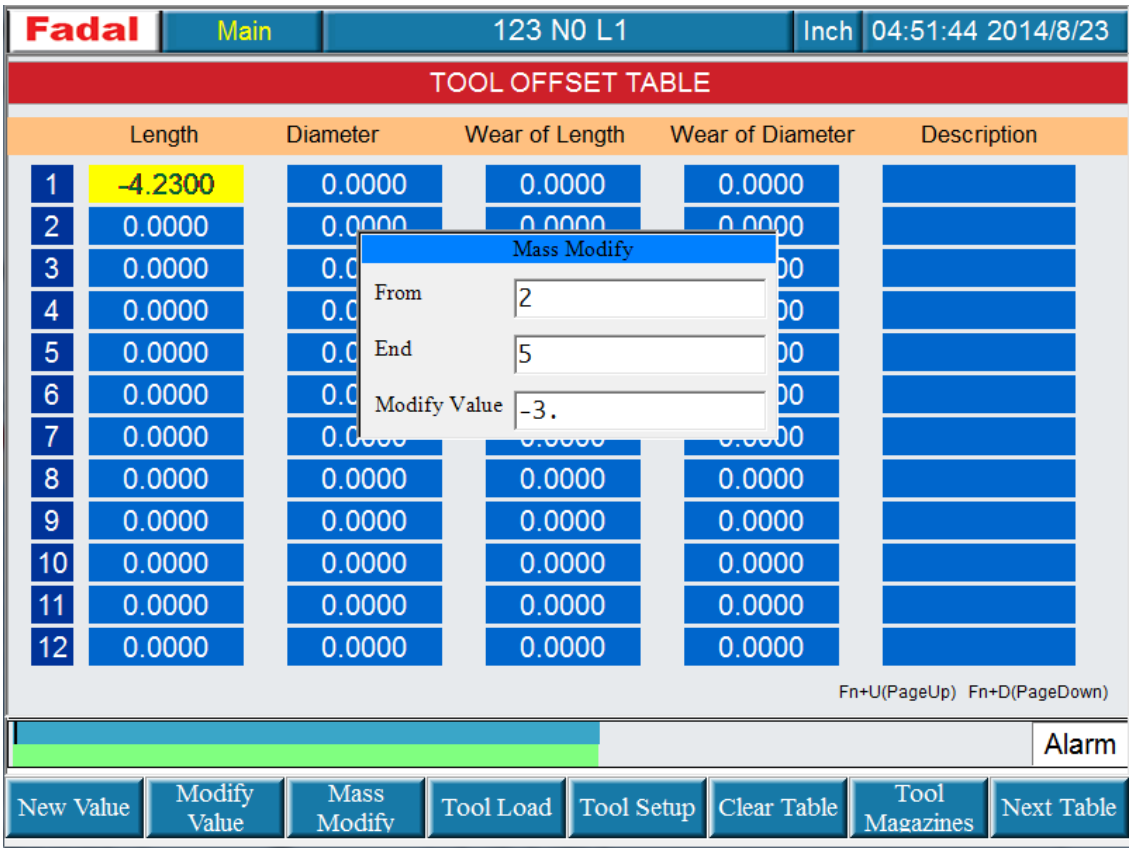


Figure 2-24 Tool Offset Table – Mass Modify

2.6.1.4 Tool Load Softkey

Pressing of selecting **Tool Load** softkey will bring up a dialog box that allow the user to enter the desired tool to be loaded. Pressing **Call** will accept the entered tool number and load the tool directly. The screen is as seen in Figure 2-25.

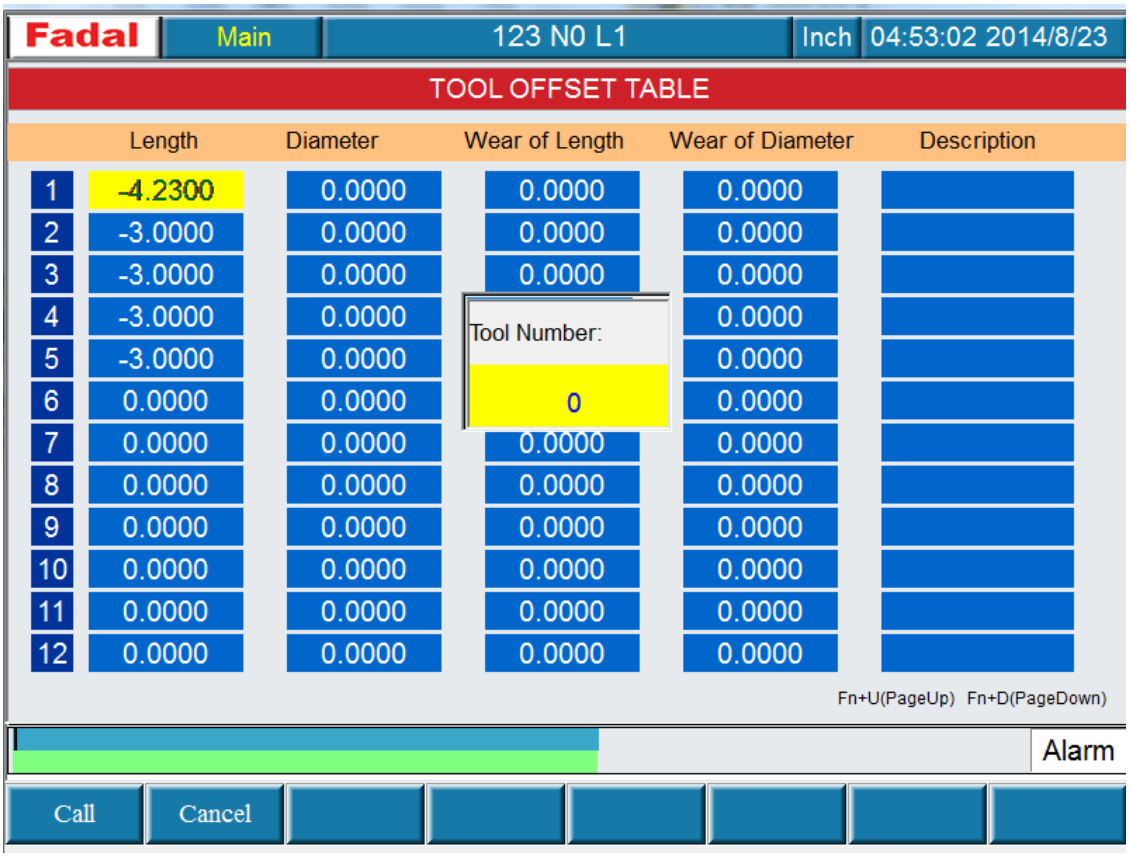


Figure 2-25 Tool Offset Table – Tool Load

2.6.1.5 Tool Setup Softkey

Setting up various tool lengths is a typical procedure of using CNC. Pressing of selecting the **Tool Setup** softkey will provide the operator with a utility for setting tool lengths. Once **Tool Setup** is pressed, a similar screen as Figure 2-26 will be seen.

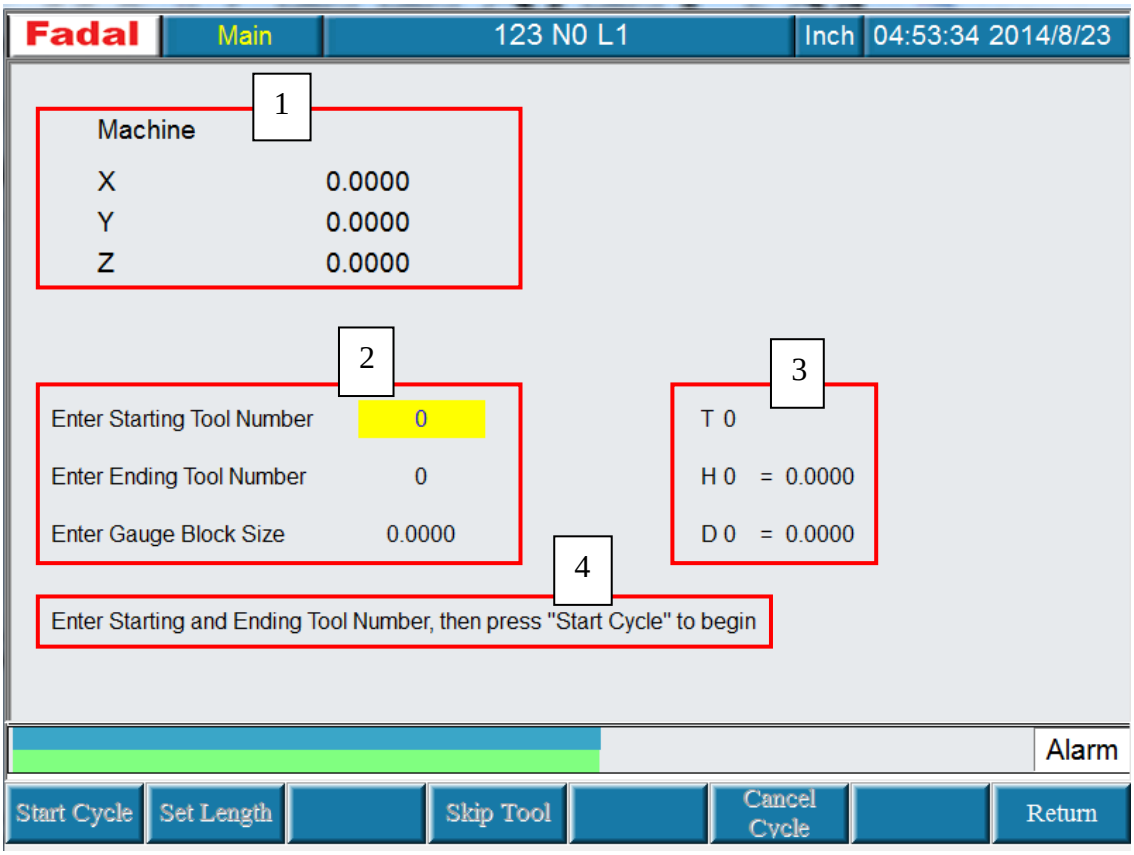


Figure 2-26 Tool Offset Table – Tool Setup

The following data will be seen in this screen：

1. Machine coordinate
2. Starting and ending tool number, gauge block size
3. Tool ready in spindle, and its H and D value
4. Operator message

User can follow the operator message to complete the tool setup procedure. To begin, user shall enter the starting and ending tool number, gauge block size (optional), press **Start Cycle** after that. The **Start Cycle** softkey will be enabled after the information above is entered (Figure 2-27).

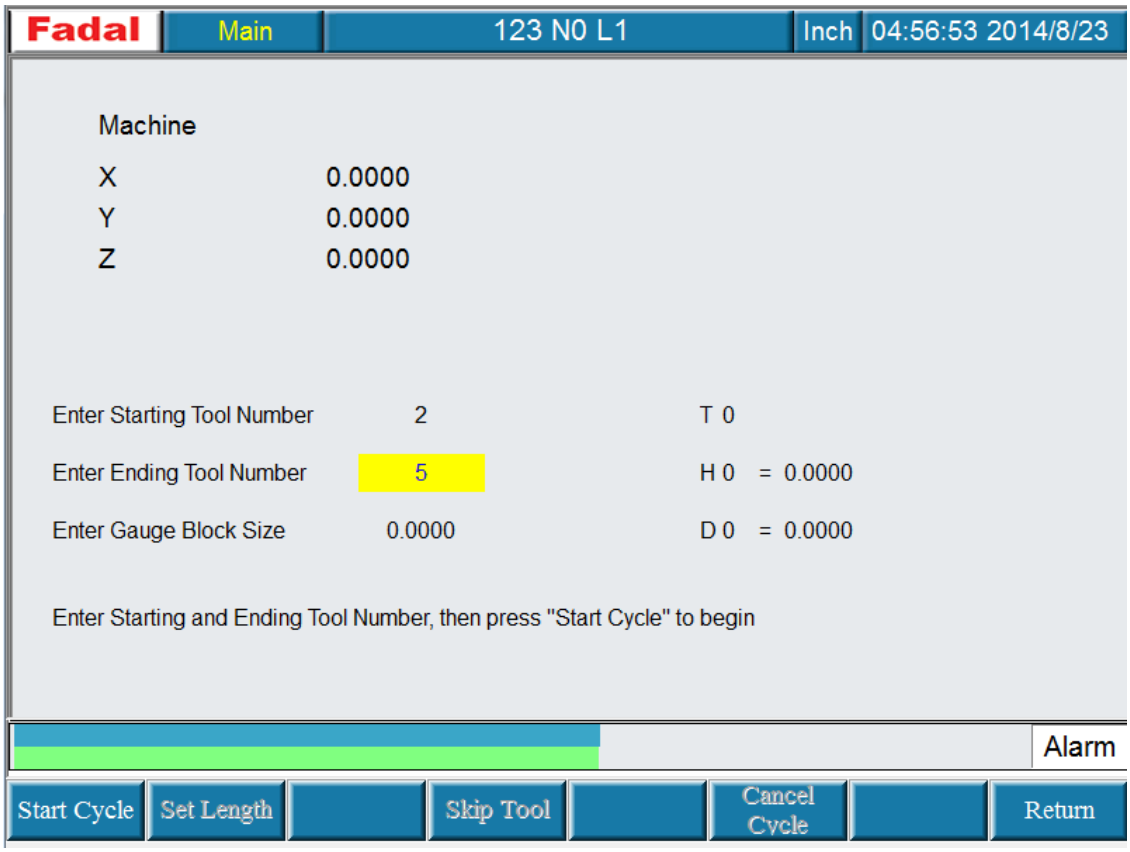


Figure 2-27 Tool Setup (1)

Once the **Start Cycle** is pressed, the operator message will change to guide the user to complete the procedure (Figure 2-28).

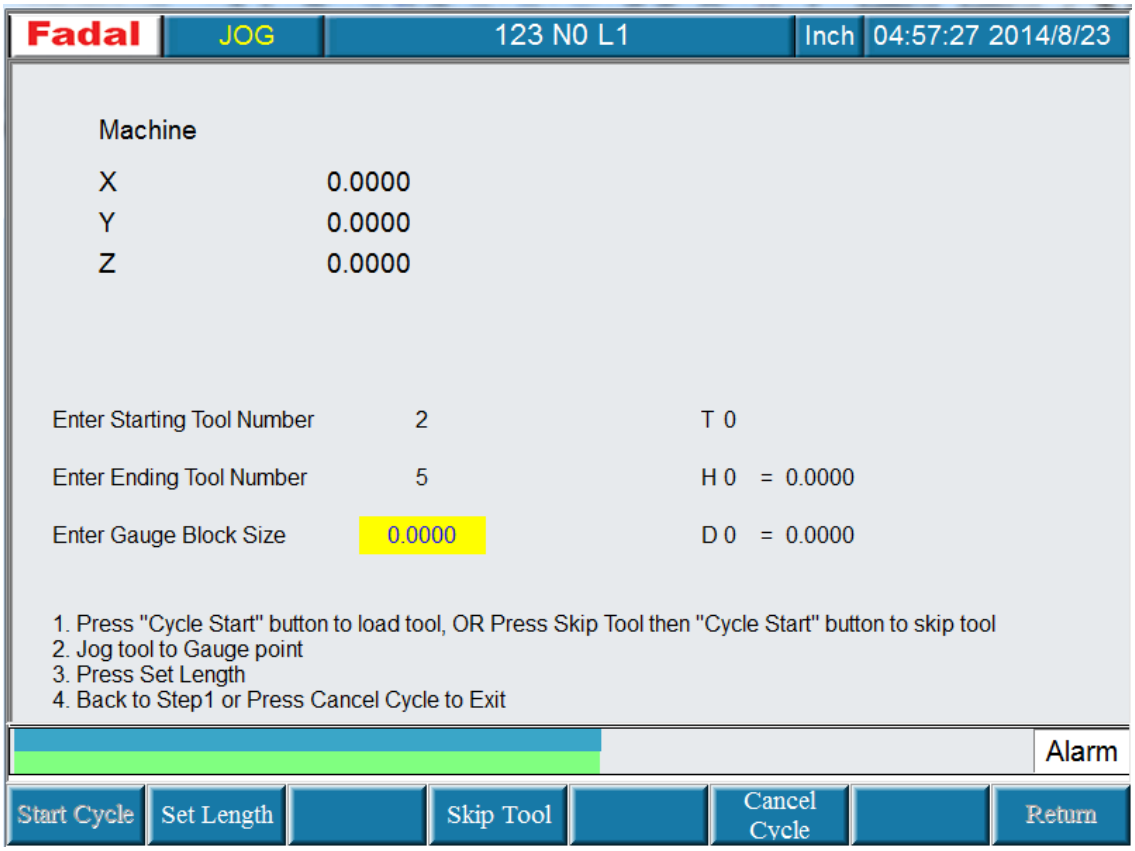


Figure 2-28 Tool Setup (2)

2.6.1.6 Clear Table Softkey

Pressing **Clear Table** softkey will bring up a dialog box to confirm with the user before clearing all the data/value in the tool offset table. Pressing **Yes** will directly clear all the data/value in the tool offfset table.

2.6.1.7 Tool Magazines Softkey

Pressing **Tool Magezines** softkey will bring up a screen (Figure 2-29) that contains the information of tool no. and pot number. Pressing **Quick Sort** under this screen will rearrange the number of tools only, i.e. set the tool in pot 1 as tool 1, the tool in pot 2 as tool 2 and so on. Pressing **Fadal Sort** under this screen will rearrange the position of the tools, i.e. put tool 1 into pot 1, put tool 2 into pot 2, and so on.

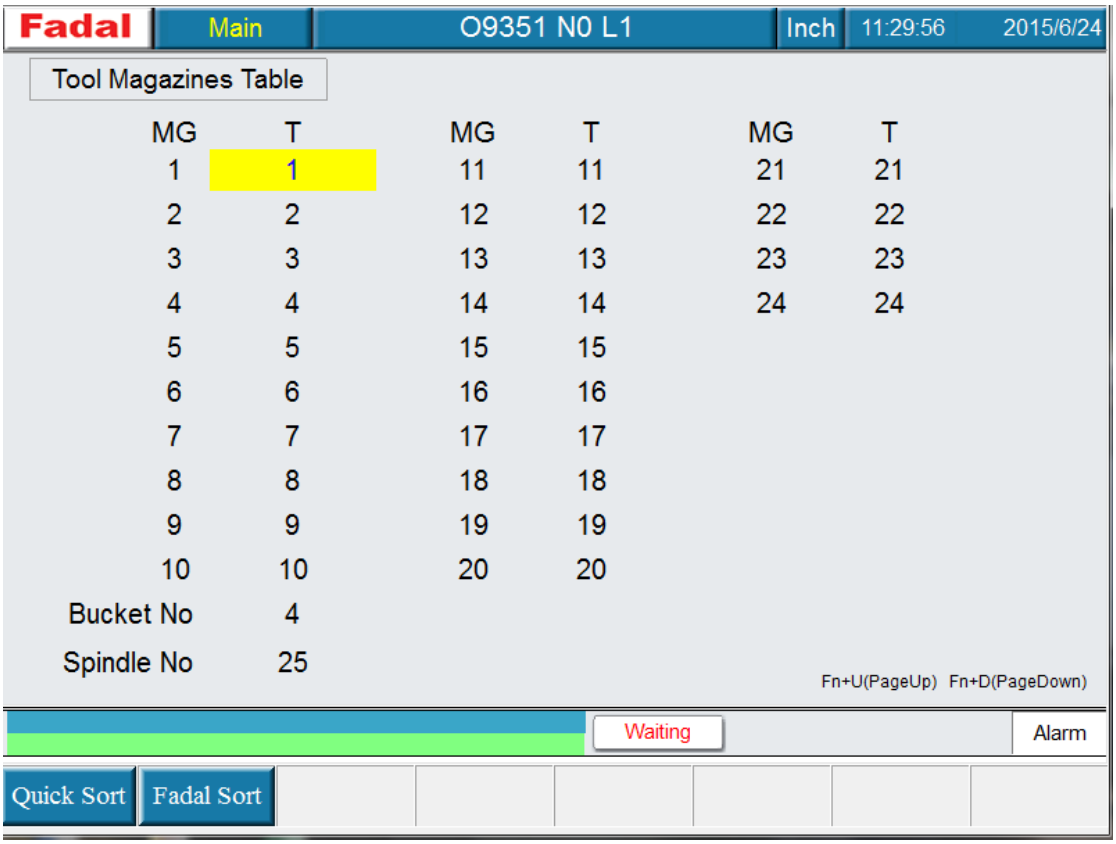


Figure 2-29 Tool Magazines Table

2.6.1.8 Next Table Softkey

Pressing **Next Table** softkey will bring up the next table which is Work Offset table screen.

2.6.2 Work Offset Table Screen

Work Offset Table allow user to check/modify the work offset data. Pressing or selecting the **Next** on the function bar of tool offset table will reach work offset table screen, the one similar to Figure 2-30.

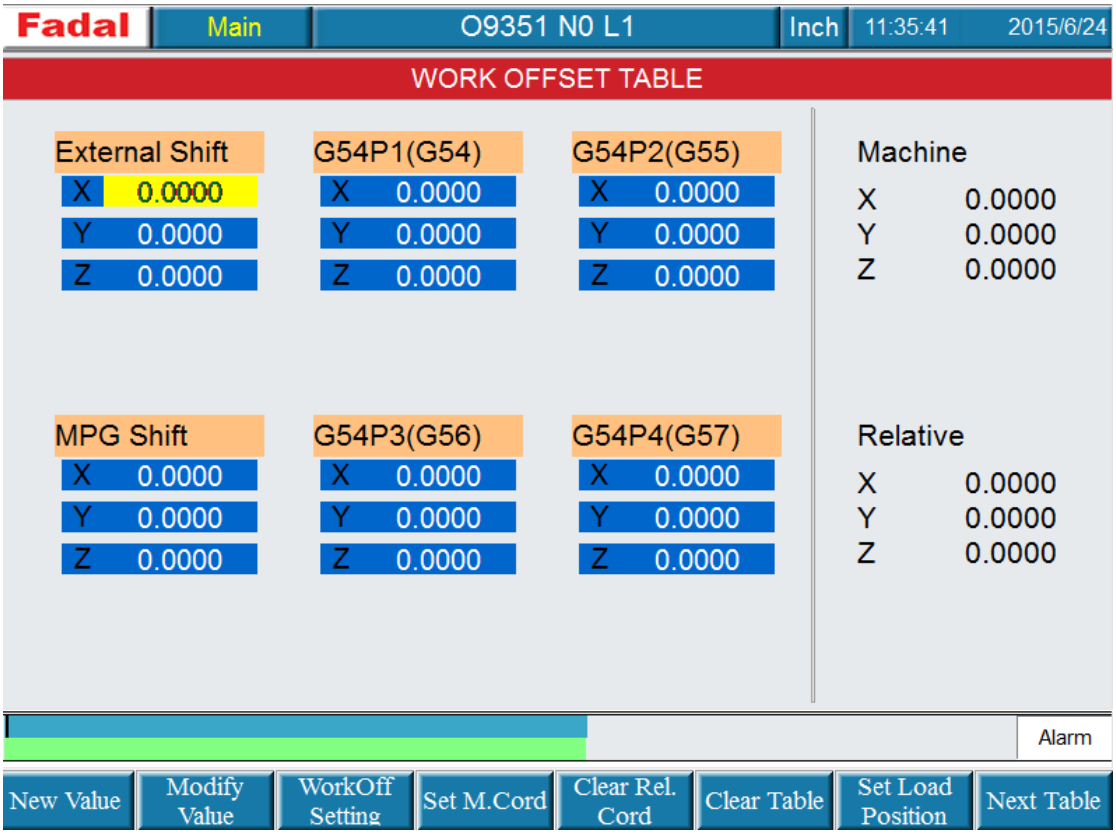


Figure 2-30 Work Offset Table

There are 100 entries for work offset in the control, i.e. E1~E100 (or G54 P1~P100), plus an external shift coordinate and a MPG shift coordinate. 6 entries will be shown in each pages, user can switch between pages through “Func+Up” “Func+Down” command.

2.6.2.1 New Value Softkey

Pressing of selecting **New Value** softkey will bring up a dialog box that will allow the user to input data or values for the selected field (cursor position). Pressing *Enter* will accept the entered data/values into the selected field. As seen in Figure 2-31, the selected field is X axis of external shift, and the user has entered “5.” into the dialog box, pressing *Enter* will insert that value to selected field.

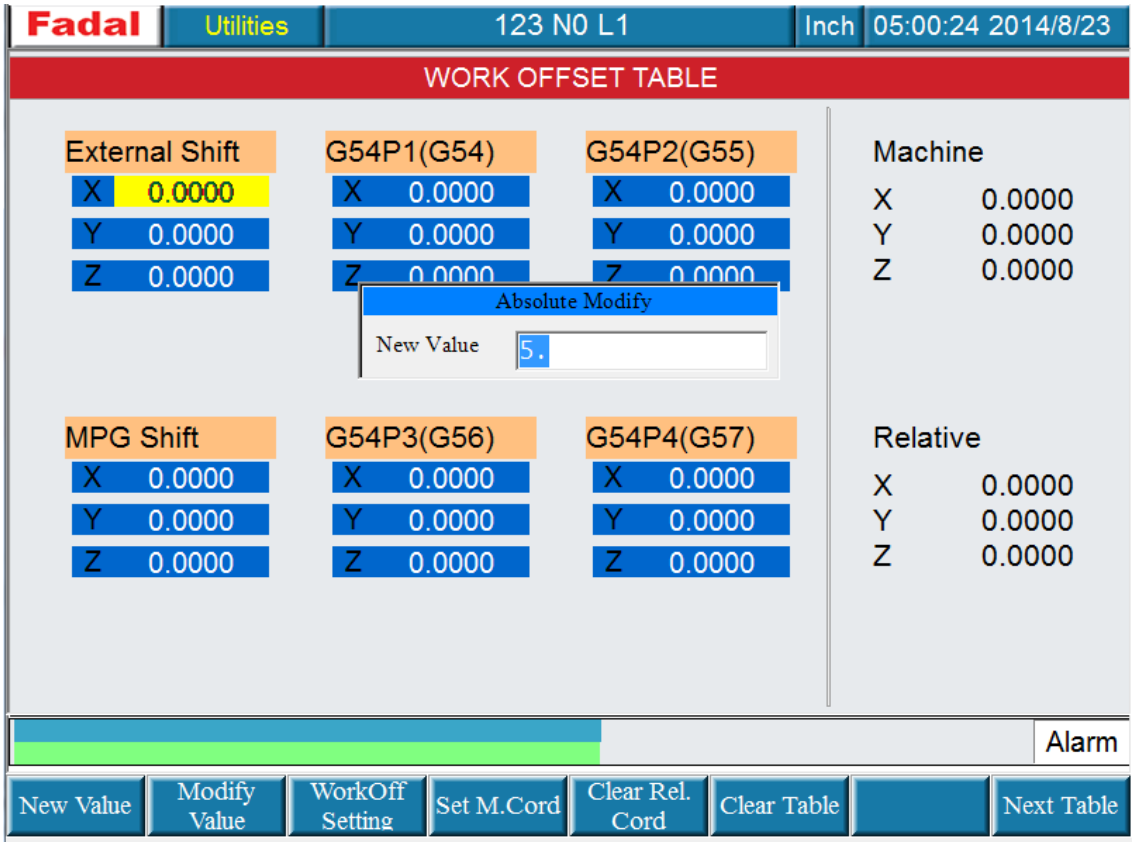


Figure 2-31 Work Offset Table – New Value

2.6.2.2 Modify Value Softkey

Pressing of selecting **Modify Value** softkey will bring up a dialog box that will allow the user to modify data or values for the selected field (cursor position). Pressing *Enter* will accept the entered data/values into the selected field. As seen in Figure 2-32, the selected field is Y axis of E1 offset which current value is “20.5”, and the user has entered “3.” into the dialog box, pressing *Enter* will modify that value to “23.5”.

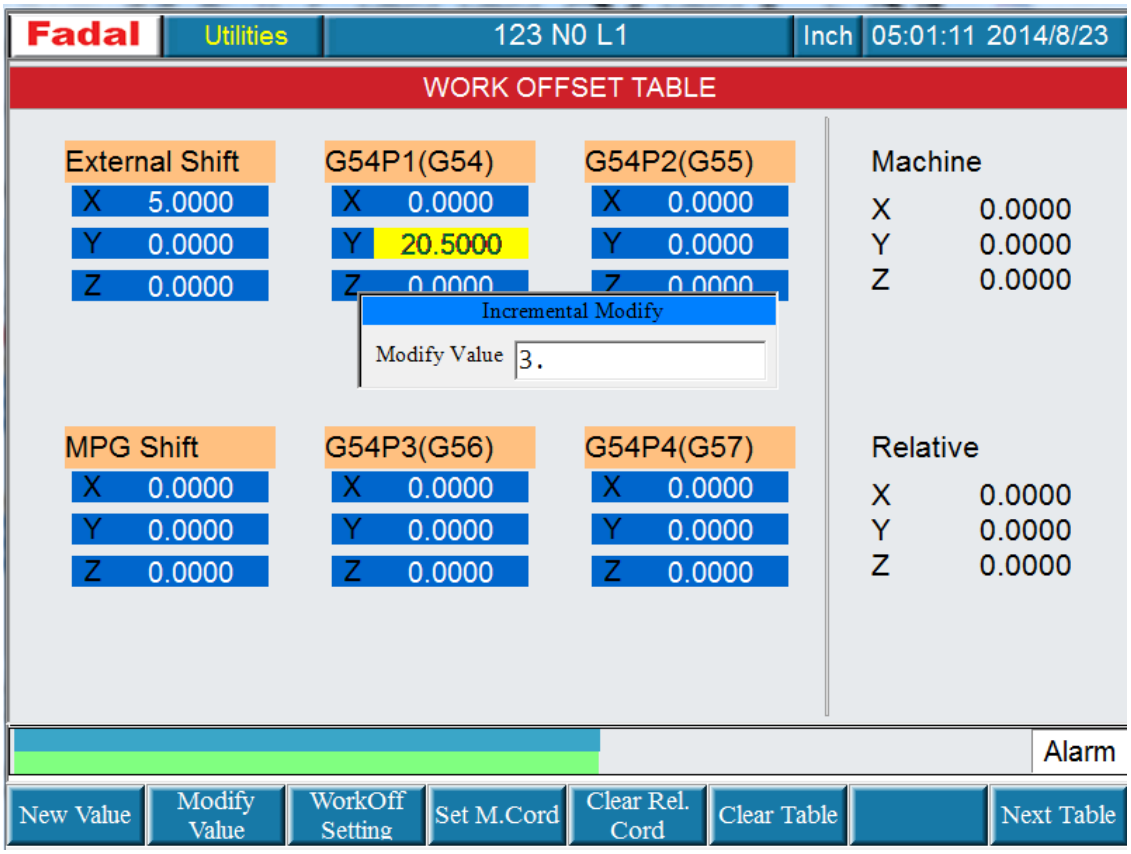


Figure 2-32 Work Offset Table – Modify Value

2.6.2.3 Work Offset Setting Softkey

Pressing of selecting **Work Offset Setting** softkey will bring up a window that consists five different cycles for setting up work offset. For more details, please refer to section 2.7.2.

2.6.2.4 Set M. Cord

This button represnet "Set Machine Coordonate" Pressing of selecting **Set M. cord** softkey will copy the machince coordinate position to where the foucus is, which makes it easy for user to apply machine coordinate to work offset.

2.6.2.5 Clear Table Softkey

Pressing **Clear Table** softkey will bring up a dialog box to confirm with the user before clearing all the data/values of current work offset. Pressing **Yes** will directly clear the data/value in current work offset.

2.6.2.6 Set Load Position Softkey

Pressing **Set Load Position** softkey will bring up a dialog box for user to input the load position. You can bring axies to this postion by pressing **Load Position** softkey. Please check 2.4.4 for more information.

2.6.2.7 Next Table Softkey

Pressing **Next Table** softkey will bring up the next table which is macro table

screen.

2.6.3 Macro Table Screen

Macro Table allow user to check/modify the macro data. Pressing or selecting the **Next** on the function bar of work offset table will reach macro table screen, the one similar to Figure 2-33.

Figure 2-33 Macro Table

There are 100 entries of macro variables (V1-V100). 50 variables are display in each pages, user can switch between pages through “Func+Up” “Func+Down” command.

2.6.3.1 Next Table Softkey

Pressing **Next Table** softkey will bring up the next table which is tool time table screen.

2.6.4 Tool Timer Table Screen

Tool Time Table allow user to check/modify the parameter regarding to tool timer. Pressing or selecting the **Next** on the function bar of macro table will reach tool timer table screen, the one similar to Figure 2-34.

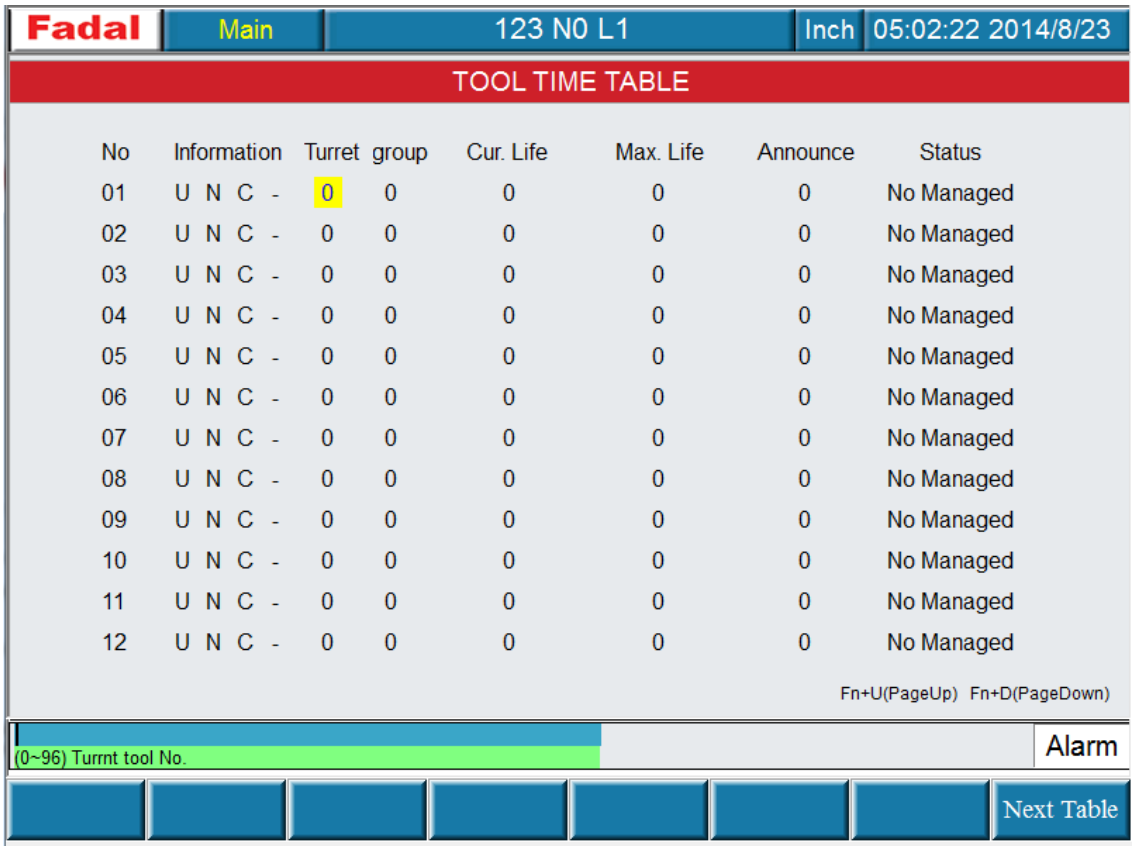


Figure 2-34 Tool Timer Table

For more details regarding this screen, please refer to section 3.3.

2.6.4.1 Next Table Softkey

Pressing **Next Table** softkey will bring up the next table which is tool offset table screen.

2.6.5 Work Record Table

Work Record Table allow user to check and save the work record, following information will be displayed on the screen: Program name, Start/end time, total time, total part counter and comment .Work record table's figure shown in below.

2.6.5.1 Save Work Softkey

Pressing Save work record softkey will bring up the new windows with save options to designated external drive.

2.6.5.2 Clear Work Softkey

Pressing **Clear work** softkey will pop up the new windows confirmation to delete work record.

2.6.5.3 Next Table Softkey

Pressing **Next Table** softkey will bring up the next table which is tool offset table screen.

2.7 Utilities Screen

Utilities mode can be reach by pressing the **Utilities** on the function bar or *Utilities* on the Fadal Keyboard. This main screen of utilities is as:

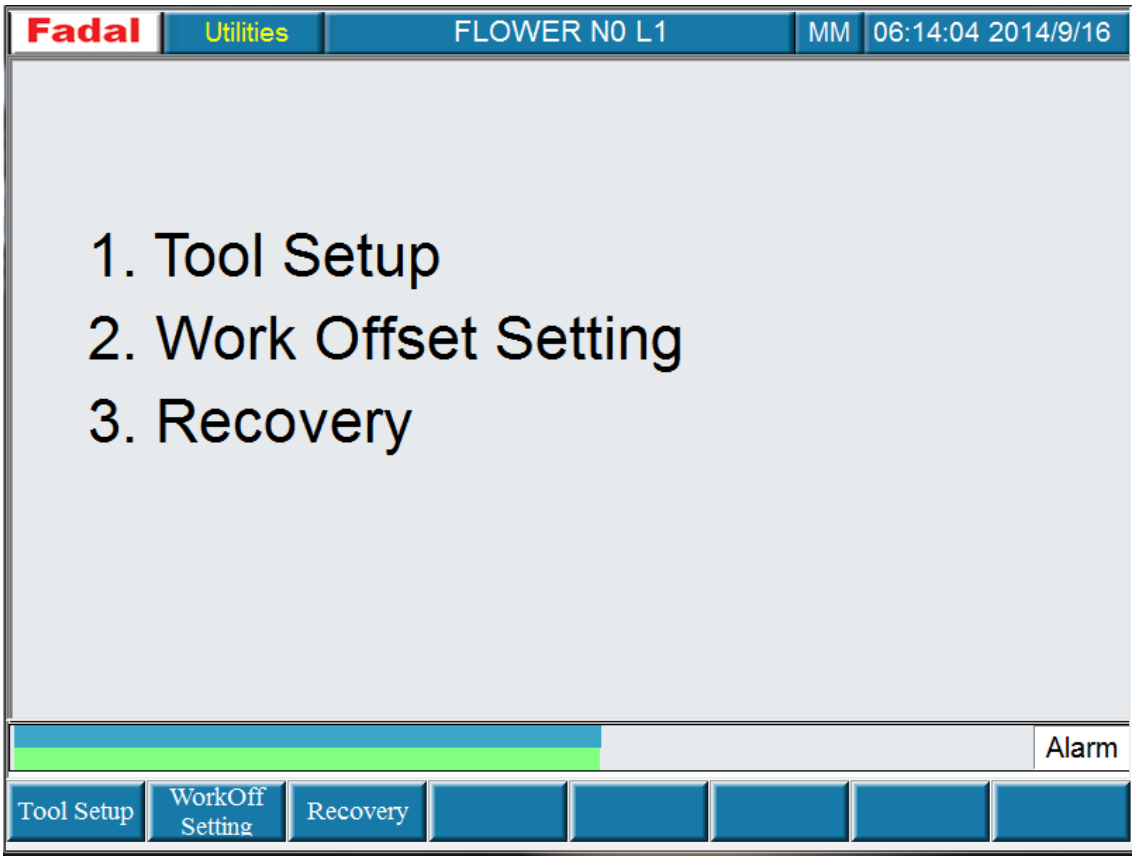


Figure 2-35 Utilities

2.7.1 Tool Setup Softkey

Pressing of selecting the **Tool Setup** softkey will provide the operator with a utility for setting tool lengths. For details of tool setup please refer to section 2.6.1.5.

2.7.2 Work Offset Setting Softkey

Work offset setting function allow user to setup a work offset in different ways. There are 5 options for setting up work offset:

i. Find Center of Rectangular
ii. Find Center of Circle
iii. Find Midpoint
iv. Find 90 Degree Corner
v. Find Corner

2.7.2.1 Find Center of Rectangular

The screen of find center of retacgular is as below:

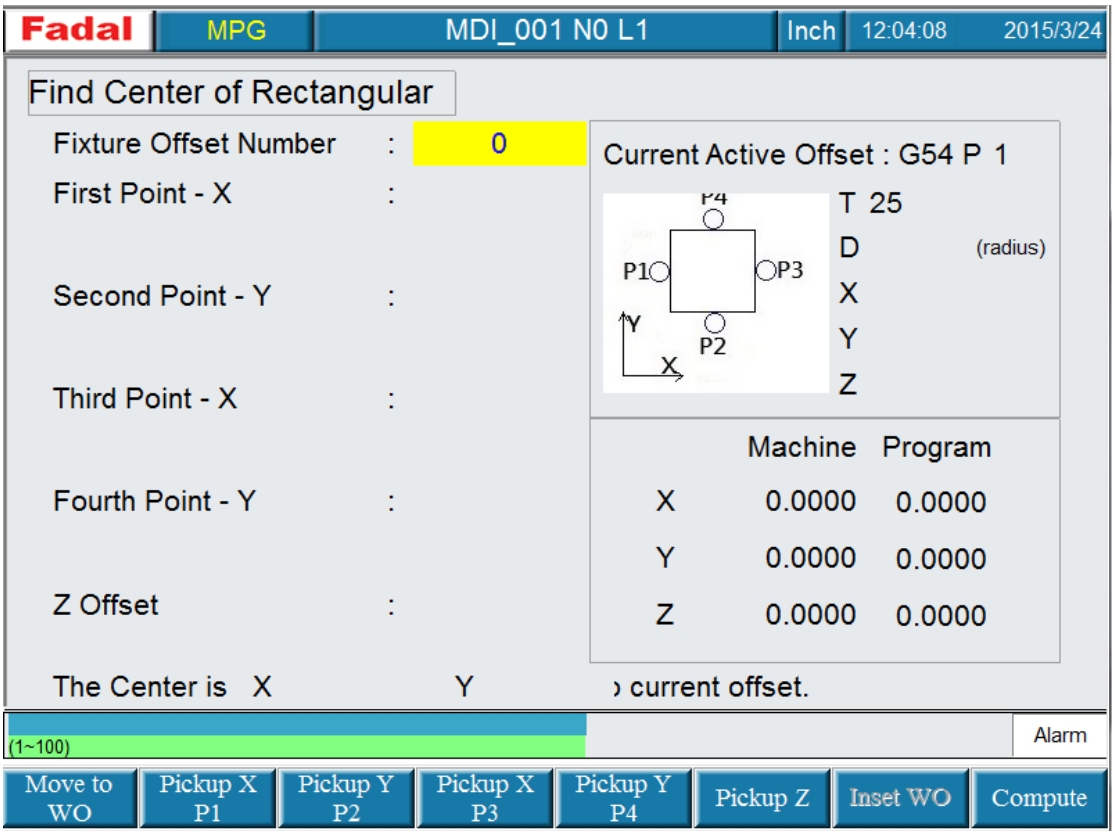


Figure 2-35 Utilities – Find Center of Rectangular

The Find Center of Rectangular utility is for setting the work offsets for a rectangular fixture or part where the part zero will be the center of the part. With this utility the operator can locate the center of a rectangular fixture/part by finding 4 points on the rectangular. These points can either be all on the outside or the inside of the part. For this utility the diameter of the locator in not required. Operator can first enter 1~100 into "Fixture Offset Number" to decide active offset. After that operator can move the axes to the edge of part and press **Pickup XY P1~P4** to pick up coordinate of each points. The operator can locate the Z position for this work offset and press or select the **Pickup Z** softkey. When all four points have been located and inputted into the proper fields, pressing or selecting the **Compute** softkey will calculate the center of the rectangular fixture or part. The rectangular center value will be displayed on the screen.

After the rectangular center has been computed, pressing or selecting the **Insert WO** softkey will insert the rectangular center values into the selected work offset number. Pressing or selecting the **Move to WorkOffset** softkey anytime during the setup will move to all the axes the current fixture offset locations. When the softkey is pressed, a confirm message will be display

"Pressing YES will cause all axes to move to current offset."

Press **YES** softkey will cause the axes to rapid to the current offset locations.

2.7.2.2 Find Center of Circle

The screen of find center of circle is as below:

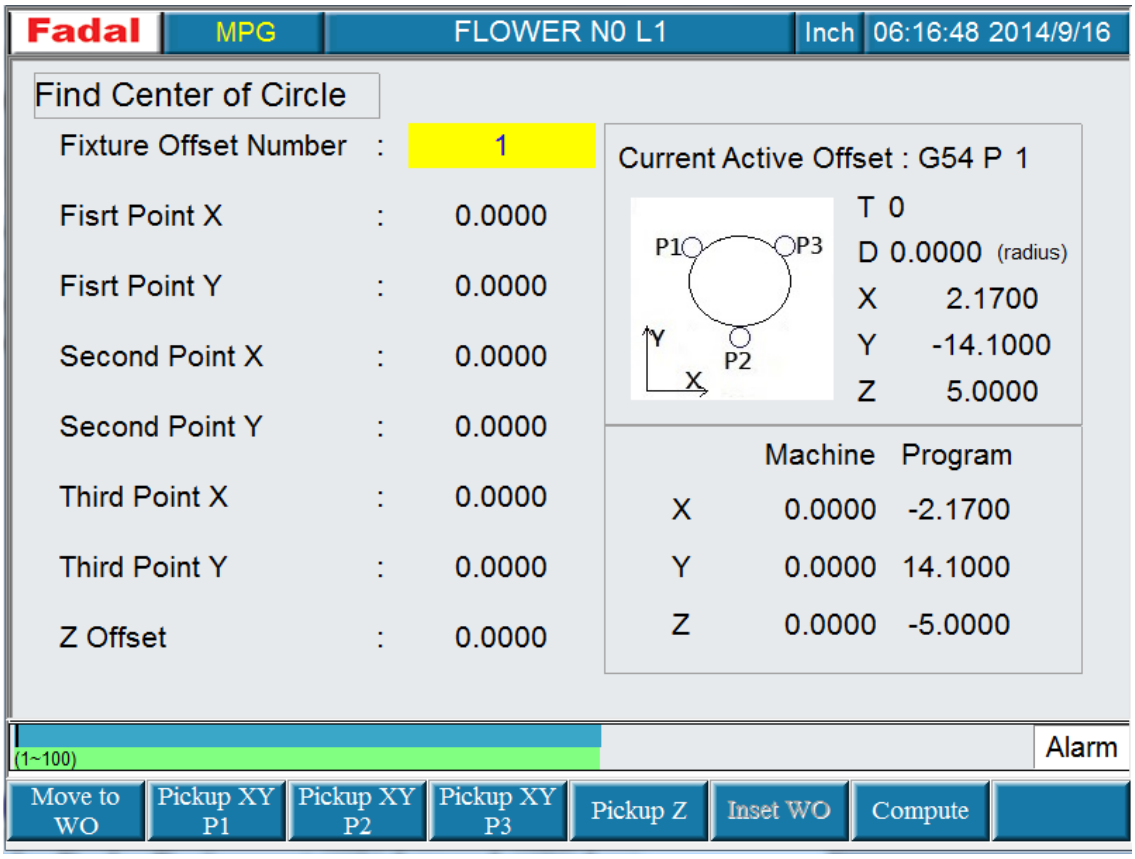


Figure 2-36 Utilities – Find Center of Circle

The Find Center of Circle utility is for setting the work offsets for a round fixture or part where the part zero will be the center of the part. With this utility the operator can locate the center of a round fixture/part by finding 3 points on the circle. These points can either be all on the outside or the inside of the part. For this utility the diameter of the locator in not required. Operator can first enter 1~100 into "Fixture Offset Number" to decide active offset. After that operator can move the axes to the edge of part and press **Pickup XY P1~P3** to pick up coordinate of each points. The operator can locate the Z position for this work offset and press or select the **Pickup Z** softkey. When all three points have been located and inputted into the proper fields, pressing or selecting the **Compute** softkey will calculate the center of the round fixture or part. The circle center value will be displayed on the screen.

After the circle center has been computed, pressing or selecting the **Insert WO** softkey will insert the circle center values into the selected work offset number. Pressing or selecting the **Move to WorkOffset** softkey anytime during the setup will move to all the axes the current fixture offset locations. When the softkey is pressed, a confirm message will be display

"Pressing YES will cause all axes to move to current offset."

Press **YES** softkey will cause the axes to rapid to the current offset locations.

2.7.2.3 Find Midpoint

The screen of find midpoint is as below:

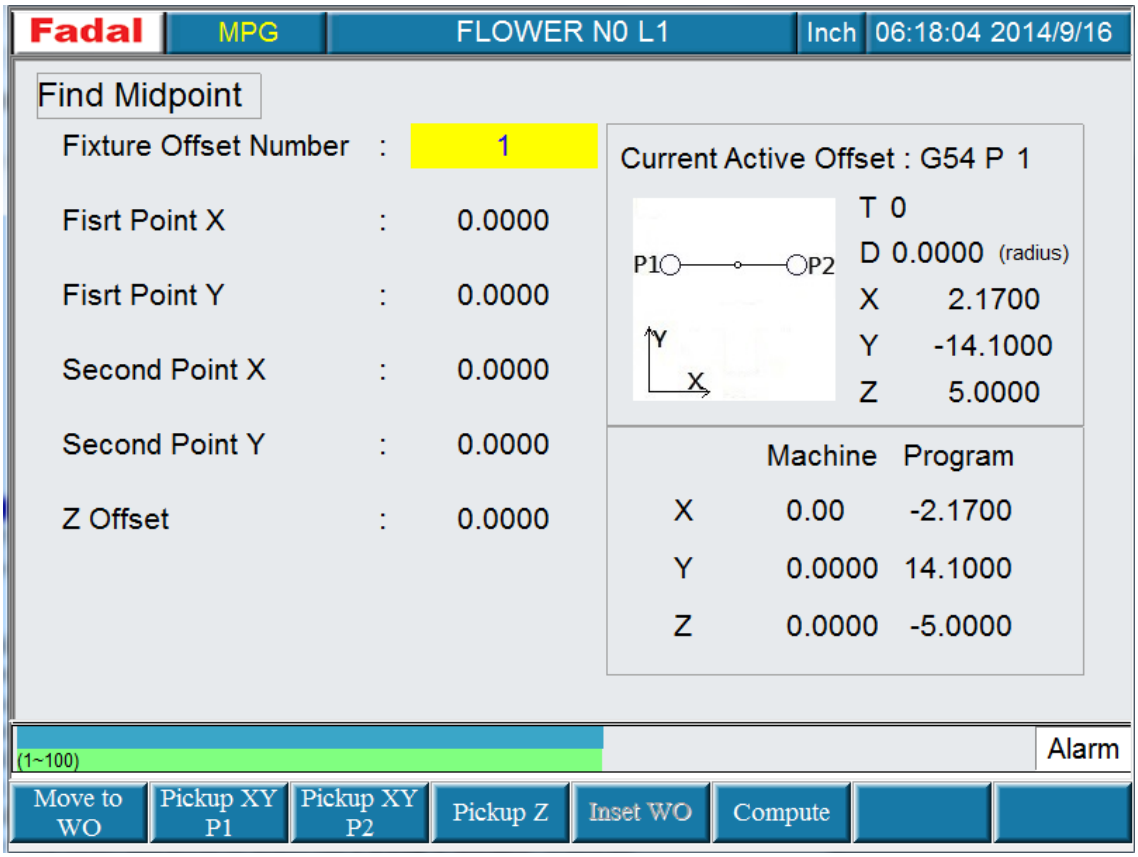


Figure 2-37 Utilities – Find Midpoint

The Find Midpoint utility is for setting the work offsets for a fixture or part where the part zero will be the center of the part. With this utility the operator can locate the center or midpoint of a fixture/part by finding 2 points on each side. These points can either be all on the outside or the inside of the part. For this utility the diameter of the locator in not required. Operator can first enter 1~100 into "Fixture Offset Number" to decide active offset. After that operator can move the axes to the edge of part and press **Pickup XY P1~P2** to pick up coordinate of each points. The operator can locate the Z position for this work offset and press or select the **Pickup Z** softkey. When all two points have been located and inputted into the proper fields, pressing or selecting the **Compute** softkey will calculate the midpoint. The midpoint value will be displayed on the screen.

After the midpoint has been computed, pressing or selecting the **Insert WO** softkey will insert the midpoint values into the selected work offset number. Pressing or selecting the **Move to WorkOffset** softkey anytime during the setup will move to all the axes the current fixture offset locations. When the softkey is pressed, a confirm message will be display

"Pressing YES will cause all axes to move to current offset."

Press **YES** softkey will cause the axes to rapid to the current offset locations.

2.7.2.4 Find 90 deg. Corner

The screen of find 90 deg. corner is as below:

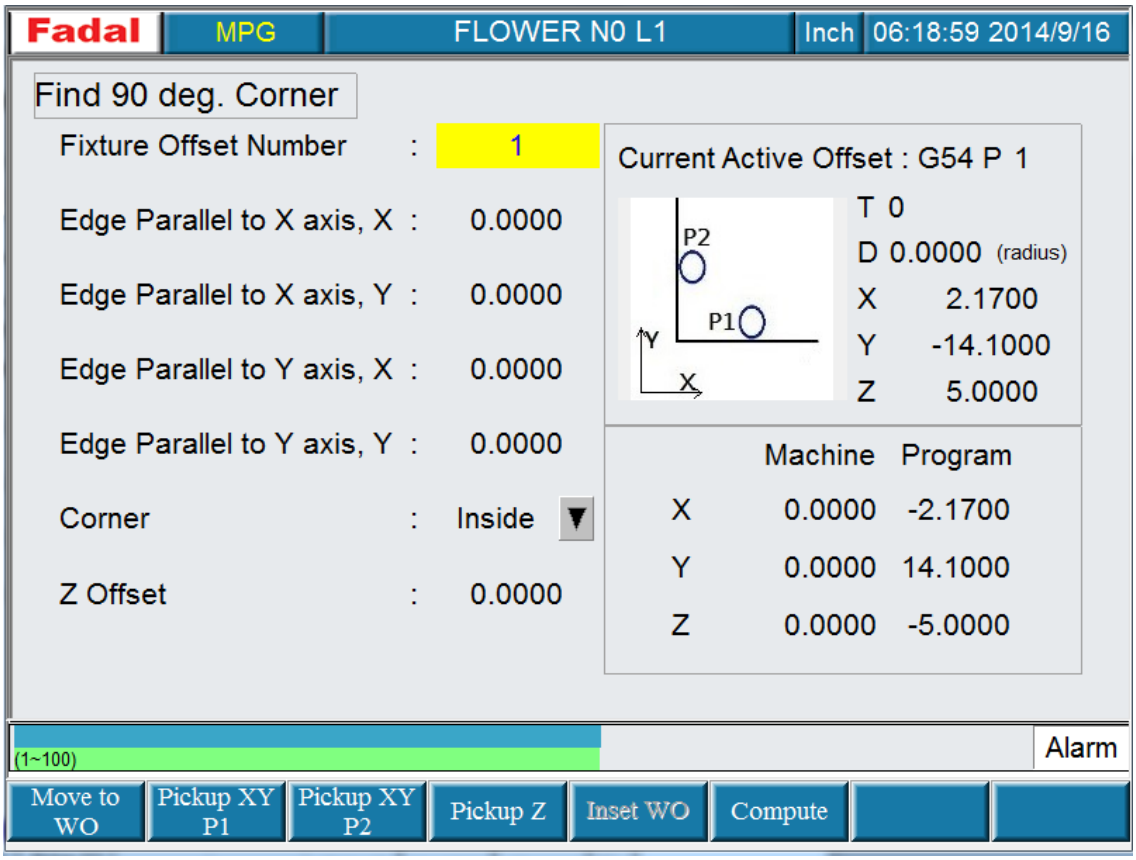


Figure 2-38 Utilities – Find 90 deg. Corner

The Find 90 deg. Corner utility is for setting the work offsets for a fixture or part where the part zero will be the corner of the part. With this utility the operator can locate the 90 deg. corner of a fixture/part by finding 2 points on each side. These points can either be all on the outside or the inside of the part. For this utility the diameter of the locator in not required. Operator can first enter 1~100 into "Fixture Offset Number" to decide active offset. After that operator can move the axes to the edge of part and press **Pickup XY P1~P2** to pick up coordinate of each points. The operator can locate the Z position for this work offset and press or select the **Pickup Z** softkey. When all two points have been located, inside/outside corner has been defined, and inputted into the proper fields, pressing or selecting the **Compute** softkey will calculate the 90 deg. corner. The corner value will be displayed on the screen.

After the 90 deg. corner has been computed, pressing or selecting the **Insert WO** softkey will insert the corner values into the selected work offset number. Pressing or selecting the **Move to WorkOffset** softkey anytime during the setup will move to all the axes the current fixture offset locations. When the softkey is pressed, a confirm message will be display

"Pressing YES will cause all axes to move to current offset."

Press **YES** softkey will cause the axes to rapid to the current offset locations.

2.7.2.5 Find Corner

The screen of find corner is as below:

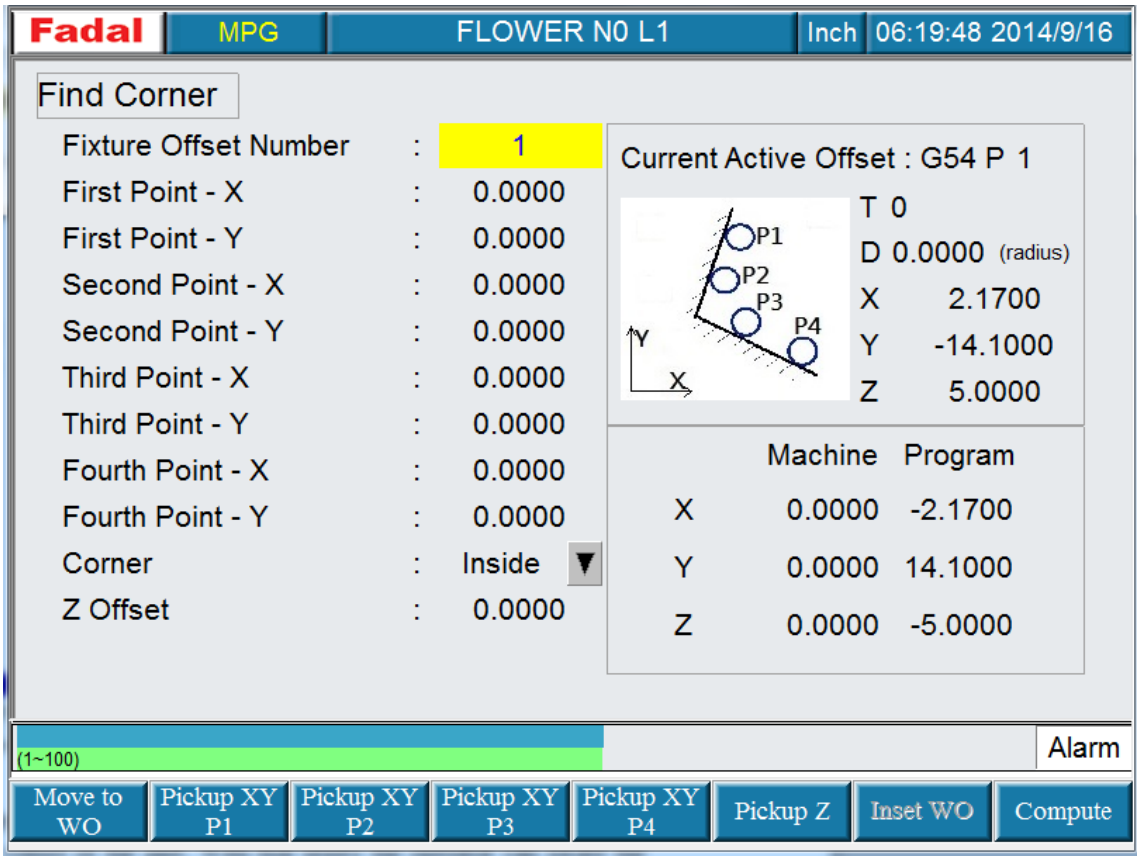


Figure 2-39 Utilities – Find Corner

The Find Corner utility is for setting the work offsets for a fixture or part where the part zero will be the corner of the part. With this utility the operator can locate the corner of a fixture/part by finding 4 points on each side. These points can either be all on the outside or the inside of the part. For this utility the diameter of the locator in not required. Operator can first enter 1~100 into "Fixture Offset Number" to decide active offset. After that operator can move the axes to the edge of part and press **Pickup XY P1~P4** to pick up coordinate of each points. The operator can locate the Z position for this work offset and press or select the **Pickup Z** softkey. When all four points have been located, inside/outside corner has been defined, and inputted into the proper fields, pressing or selecting the **Compute** softkey will calculate the corner. The corner value will be displayed on the screen.

After the corner has been computed, pressing or selecting the **Insert WO** softkey will insert the corner values into the selected work offset number. Pressing or selecting the **Move to WorkOffset** softkey anytime during the setup will move to all the axes the current fixture offset locations. When the softkey is pressed, a confirm message will be display

"Pressing YES will cause all axes to move to current offset."

Press **YES** softkey will cause the axes to rapid to the current offset locations.

2.7.3 Recovery Softkey

Pressing or selecting Recovery softkey will bring up a recovery window as below.

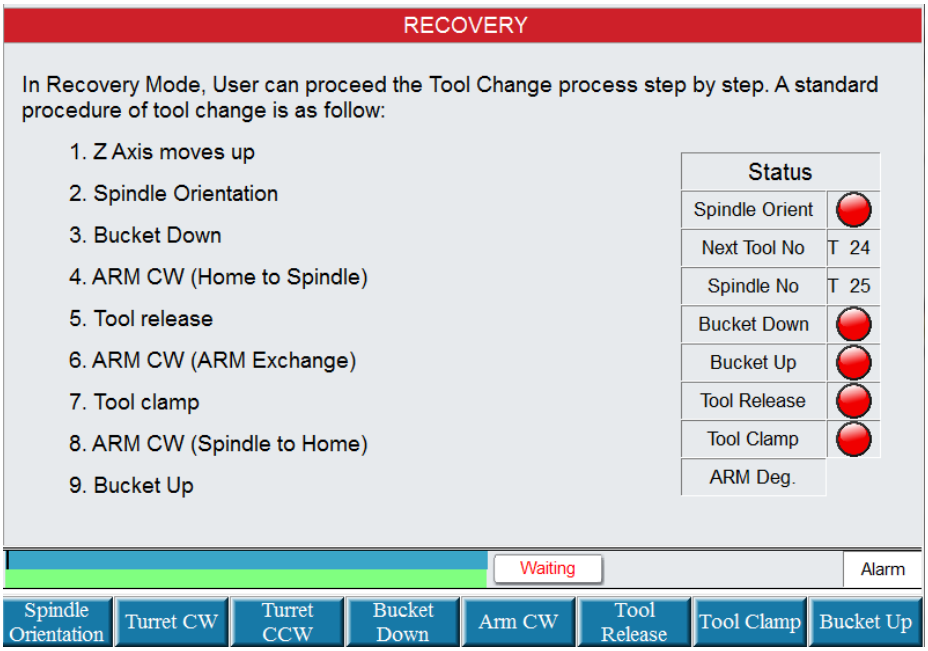


Figure 2-40 Utilities – Recovery

This screen will guide the user to do the tool change procedure manually. Press the coresponding buttons by following the instruction you can finish a tool change.

By pressing right-arrow button you can achieve DATC single step function. In this function, you don't need to press coresponding buttons. All you need to do is press DATC single step then hit cycle start. It will execute tool change step by step.

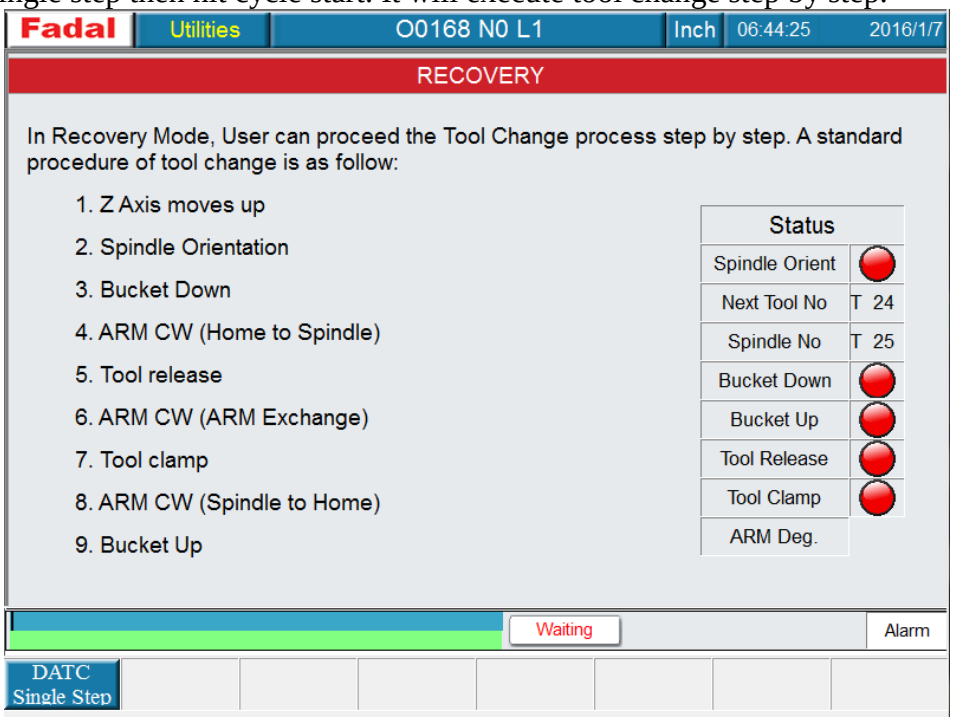


Figure 2-41 Utilities – Recovery-DATC single step

2.7.4 Probe Function

Probe function is for setting a work offset by auto-probing. There are 9 options for auto-probing:

i. Bore
ii. Boss
iii. Rectangle I.D.
iv. Rectangle O.D.
v. Web
vi. Pocket
vii. Corner Finding
viii. Single Touch
ix. Angle Measurement

2.7.4.1 Bore

The screen of **Bore** cycle is as below:

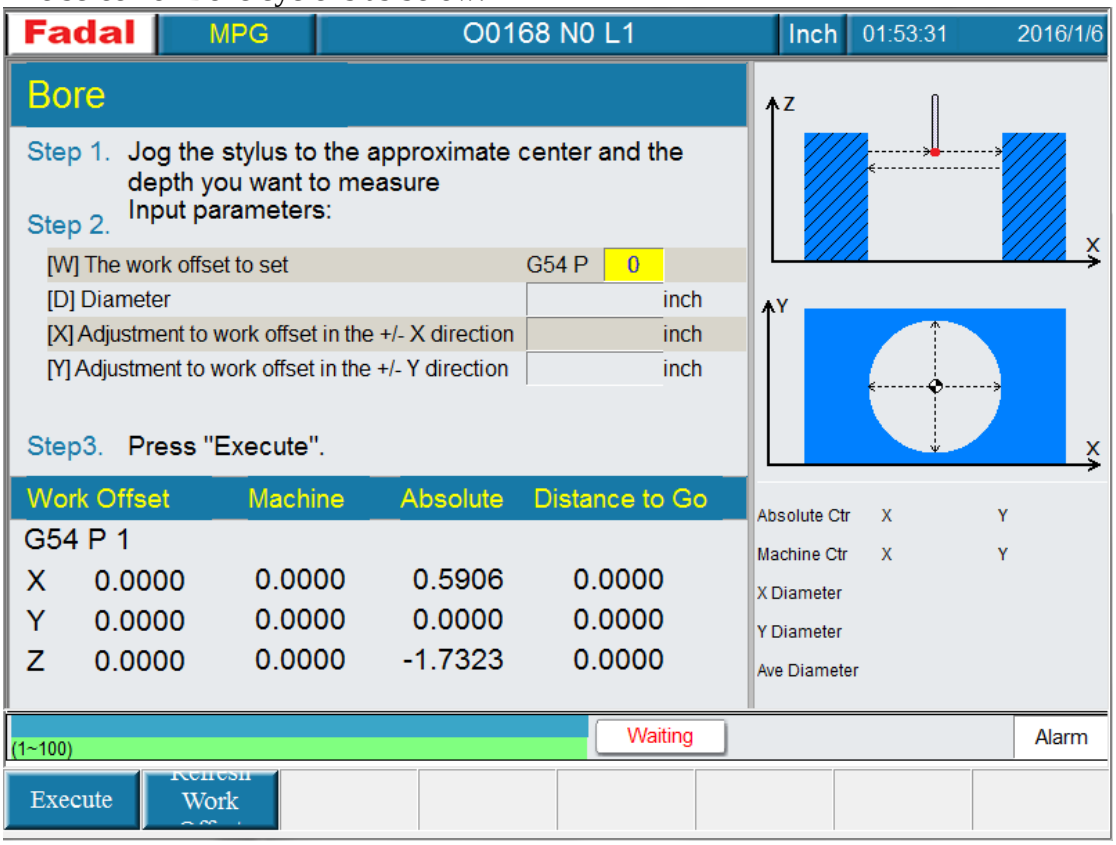


Figure 2-42 Probe Function– Bore

The **Bore** cycle is for setting the work offsets for a fixture or part with a circular pocket. The work offset zero will be the center of the pocket.

Before using this cycle:

- The probe must be in the spindle and calibrated.

Step 1:

- Please release emergency stop before jogging the stylus.
- The stylus must be positioned to the approximate center of the work piece and the depth you want to measure.

Step 2:

- For parameter [W], please input the work offset number that you would like to set.
- For parameter [D], please input the maximum possible diameter for the work piece. Default value is 3.0000 inch.
- For parameter [X], please input the adjustment to work offset in +/-X direction, it's optional.
- For parameter [Y], please input the adjustment to work offset in +/-Y direction, it's optional.

Step 3:

- Make sure all the parameters are set properly.
- Press "Execute", control will pop up a confirm window.
- Select "OK" to start probing.

After using this cycle:

—When auto-probing is finished, the value of work offset will be updated automatically.

2.7.4.2 Boss

The screen of **Boss** cycles is as below:

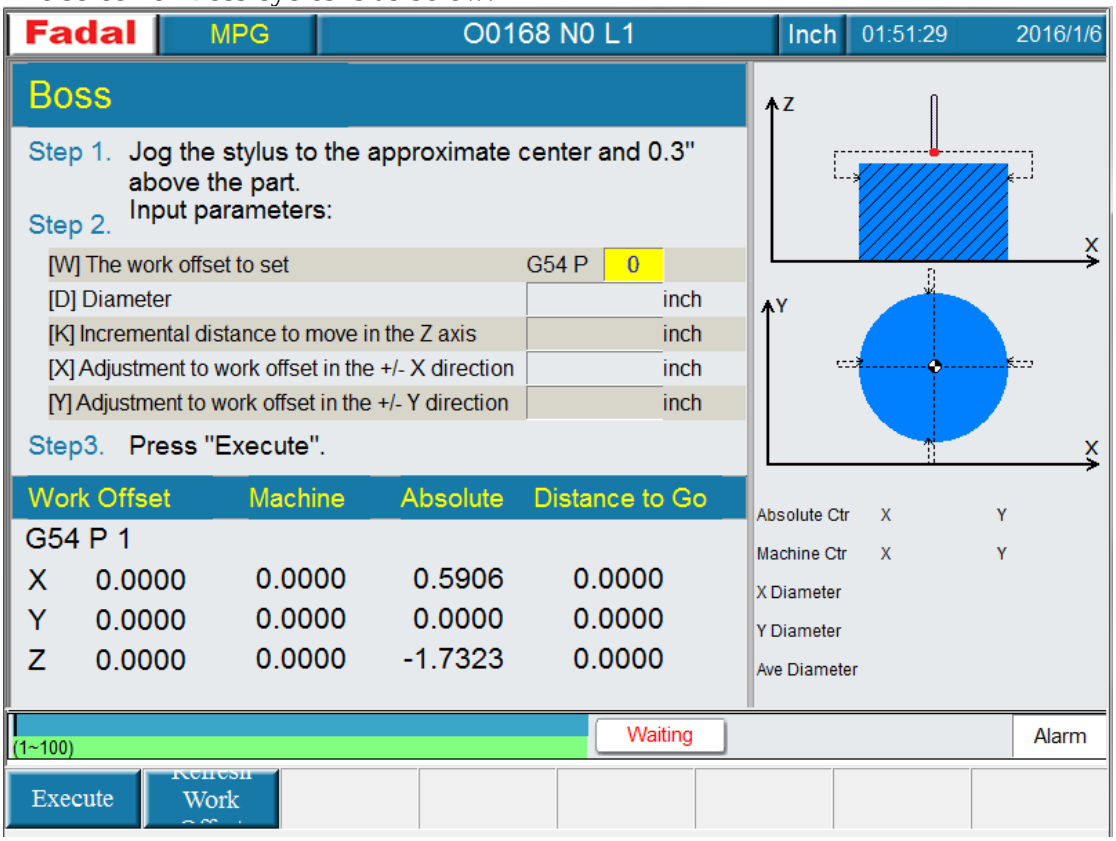


Figure 2-43 Probe Function– Boss

The **Boss** cycles is for setting the work offsets for a circular part. The work offset zero will be the center of the part.

Before using this cycle:

- The probe must be in the spindle and calibrated.

Step 1:

- Please release emergency stop before jogging the stylus.
- The stylus must be positioned to the approximate center of the work piece and approximately 0.3” above the top surface.

Step 2:

- For parameter [W], please input the work offset number that you would like to set.
- For parameter [D], please input the maximum possible diameter for the work piece. Default value is 3.0000 inch.
- For parameter [K], please input the incremental distance to move in Z-axis, it’s optional.
- For parameter [X], please input the adjustment to work offset in +/-X direction, it’s optional.
- For parameter [Y], please input the adjustment to work offset in +/-Y direction, it’s optional.

Step 3:

- Make sure all the parameters are set properly.
- Press “Execute”, control will pop up a confirm window.
- Select ”OK” to start probing.

After using this cycle:

—When auto-probing is finished, the value of work offset will be updated automatically.

2.7.4.3 Rectangle I.D.

The screen of **Rectangle I.D.** is as below:

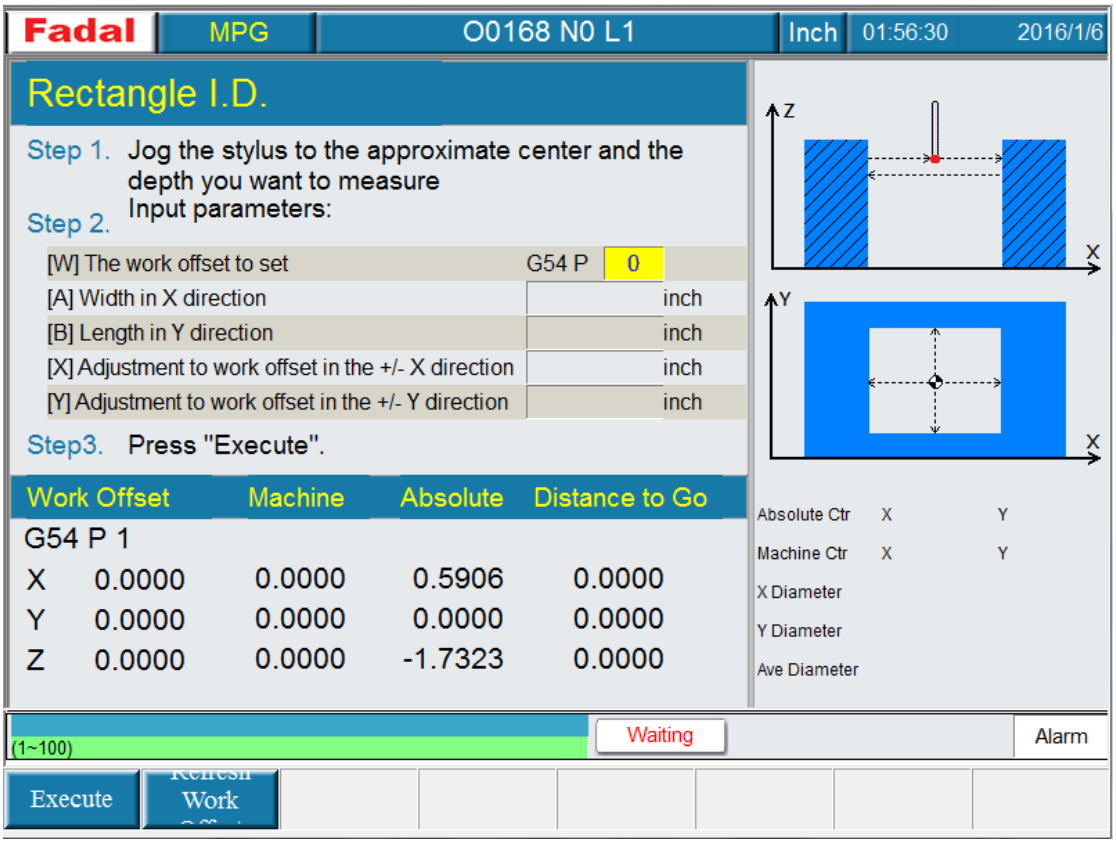


Figure 2-44 Probe Function– Rectangle I.D.

The **Rectangle I.D.** is for setting the work offsets for a fixture or part with a rectangular pocket. The work offset zero will be the center of the pocket.

Before using this cycle:

- The probe must be in the spindle and calibrated.

Step 1:

- Please release emergency stop before jogging the stylus.
- The stylus must be positioned to the approximate center of the work piece and the depth you want to measure.

Step 2:

- For parameter [W], please input the work offset number that you would like to set.
- For parameter [A], please input the maximum possible width in X direction for the work piece.
- For parameter [B], please input the maximum possible length in Y direction for the work piece.
- For parameter [X], please input the adjustment to work offset in +/-X direction, it's optional.
- For parameter [Y], please input the adjustment to work offset in +/-Y direction, it's optional.

Step 3:

- Make sure all the parameters are set properly.
- Press “Execute”, control will pop up a confirm window.
- Select ”OK” to start probing.

After using this cycle:

—When auto-probing is finished, the value of work offset will be updated automatically.

2.7.4.4 Rectangle O.D.

The screen of **Rectangle O.D.** is as below:

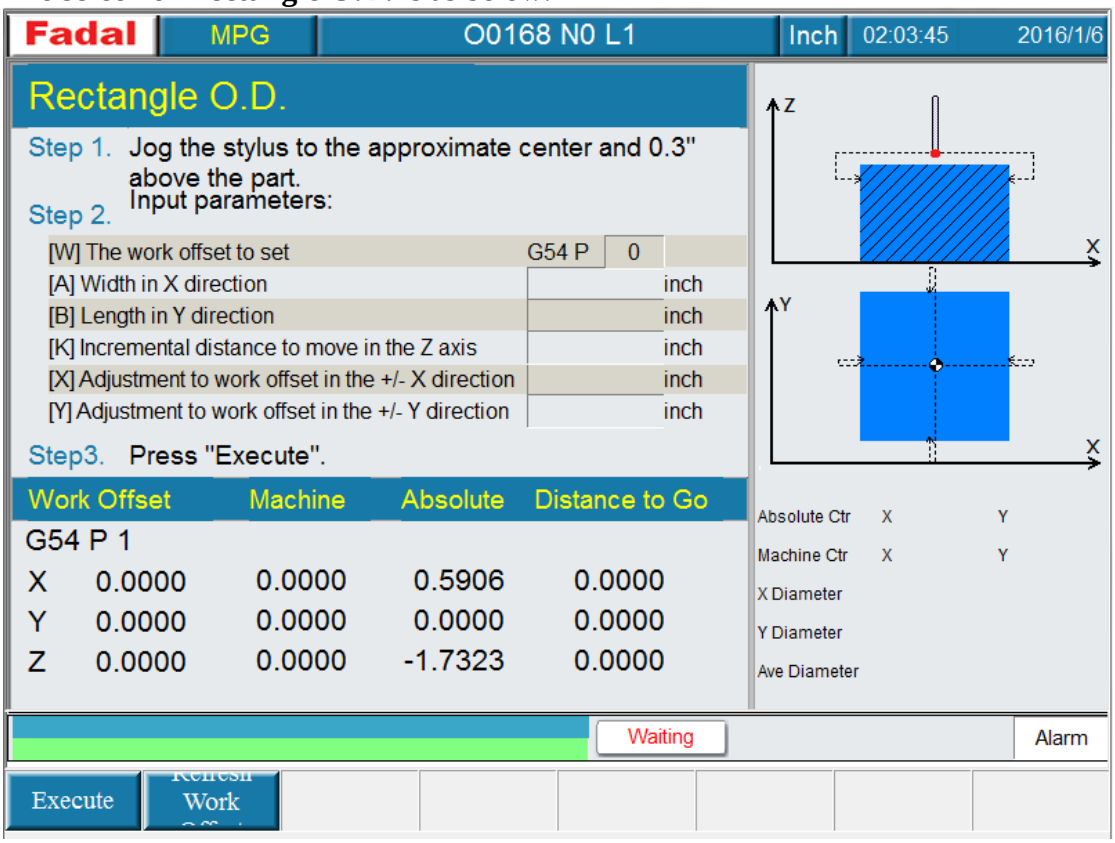


Figure 2-45 Probe Function– Rectangle O.D.

The **Rectangle O.D.** is for setting the work offsets for a rectangular pocket. The work offset zero will be the center of the pocket.

Before using this cycle:

- The probe must be in the spindle and calibrated.

Step 1:

- Please release emergency stop before jogging the stylus.
- The stylus must be positioned to the approximate center of the work piece and approximately 0.3” above the top surface.

Step 2:

- For parameter [W], please input the work offset number that you would like to set.
- For parameter [A], please input the maximum possible width in X direction for the work piece.

- For parameter [B], please input the maximum possible length in Y direction for the work piece.
- For parameter [K], please input the incremental distance to move in Z-axis, it's optional.
- For parameter [X], please input the adjustment to work offset in +/-X direction, it's optional.
- For parameter [Y], please input the adjustment to work offset in +/-Y direction, it's optional.

Step 3:

- Make sure all the parameters are set properly.
- Press "Execute", control will pop up a confirm window.
- Select "OK" to start probing.

After using this cycle:

—When auto-probing is finished, the value of work offset will be updated automatically.

2.7.4.5 Web in X(Y)

The screen of **Web in X(Y)** is as below:

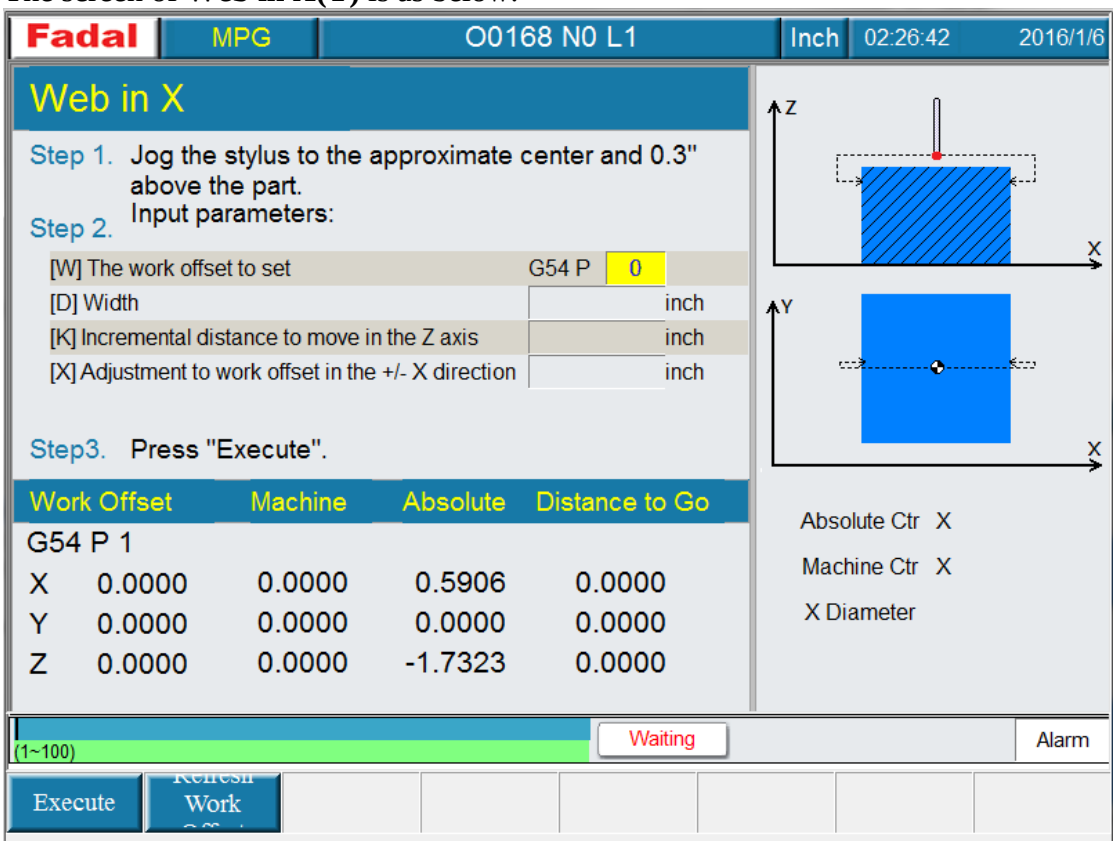


Figure 2-46 Probe Function– Web in X(Y)

The **Web in X(Y)** is for setting the work offsets for a fixture or part with a web in X(Y)-direction. The work offset zero will be the center in (X)Y-direction of the web.

Before using this cycle:

- The probe must be in the spindle and calibrated.

Step 1:

- Please release emergency stop before jogging the stylus.
- The stylus must be positioned to the approximate center of the work piece and approximately 0.3” above the top surface.

Step 2:

- For parameter [W], please input the work offset number that you would like to set.
- For parameter [D], please input the maximum possible diameter for the work piece. Default value is 3.0000 inch.
- For parameter [K], please input the incremental distance to move in Z-axis, it’s optional.
- For parameter [X]([Y]), please input the adjustment to work offset in +/- X(Y) direction, it’s optional.

Step 3:

- Make sure all the parameters are set properly.

- Press “Execute”, control will pop up a confirm window.
- Select ”OK” to start probing.

After using this cycle:

—When auto-probing is finished, the value of work offset will be updated automatically.

2.7.4.6 Pocket in X(Y)

The screen of **Pocket in X(Y)** cycle is as below:

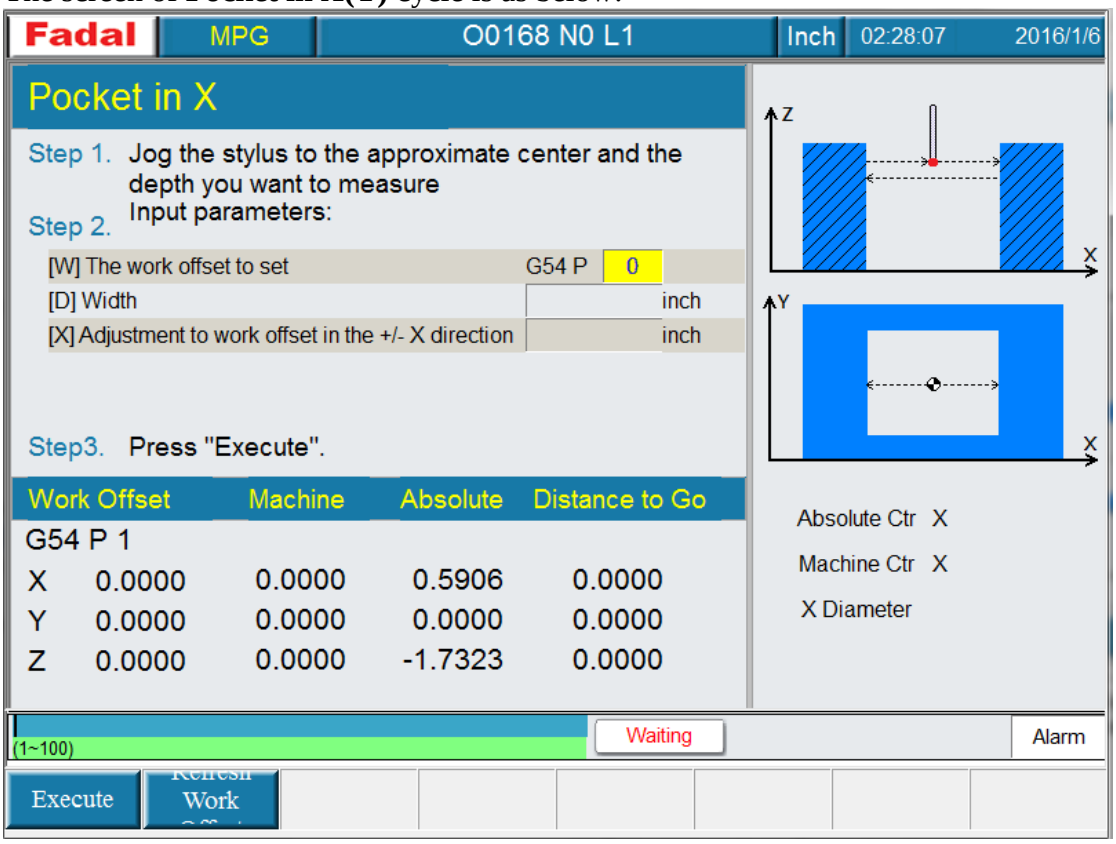


Figure 2-47 Probe Function– Pocket in X(Y)

The **Pocket in X(Y)** cycle is for setting the work offsets for a fixture or part with a circular/ rectangular pocket. The work offset zero will be the center in X(Y)-direction of the pocket.

Before using this cycle:

- The probe must be in the spindle and calibrated.

Step 1:

- Please release emergency stop before jogging the stylus.
- The stylus must be positioned to the approximate center of the work piece and the depth you want to measure.

Step 2:

- For parameter [W], please input the work offset number that you would like to set.
- For parameter [D], please input the maximum possible diameter for the work piece. Default value is 3.0000 inch.

- For parameter [X]([Y]), please input the adjustment to work offset in +/- X(Y) direction, it's optional.

Step 3:

- Make sure all the parameters are set properly.
- Press "Execute", control will pop up a confirm window.
- Select "OK" to start probing.

After using this cycle:

—When auto-probing is finished, the value of work offset will be updated automatically.

2.7.4.7 Finding Inside (Outside) Corner

The screen of **Finding Inside(Outside) Corner** is as below:

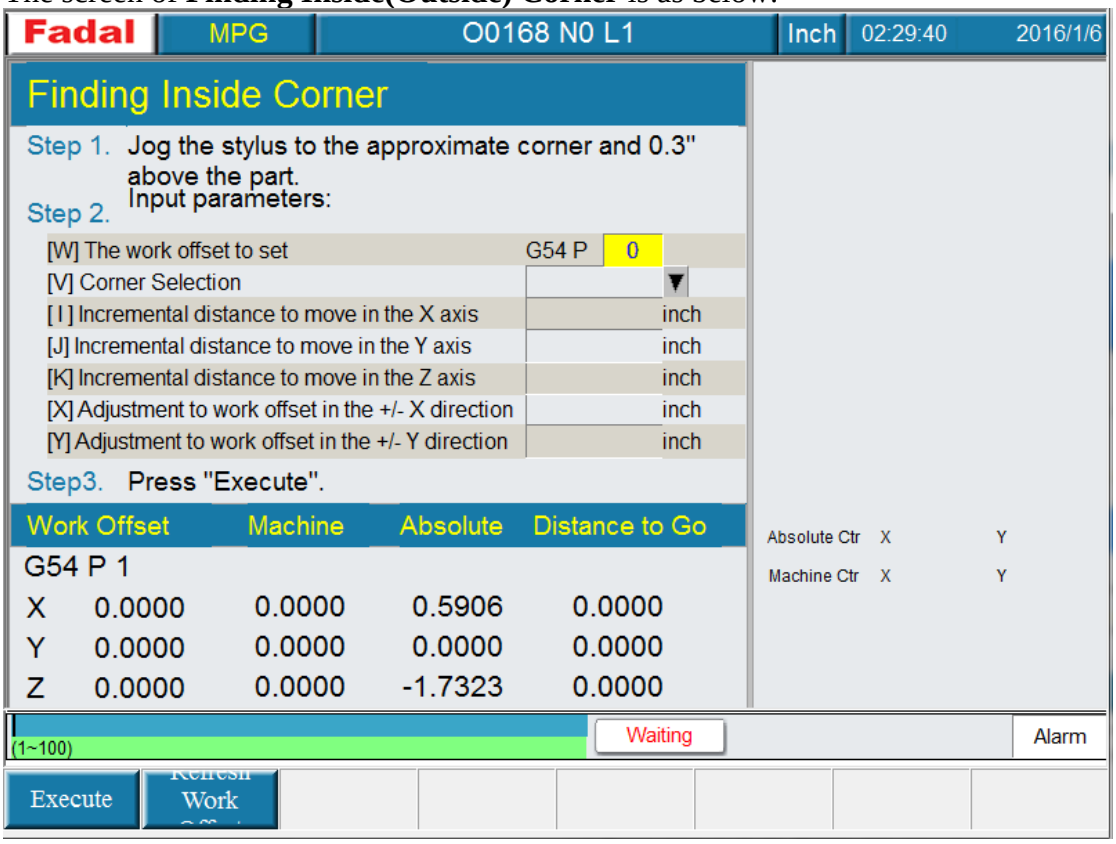


Figure 2-48 Probe Function–Finding Inside (Outside) Corner

The **Finding Inside(Outside) Corner** is for setting the work offsets for a inside (outside) corner of a part. The work offset zero will be at the corner.

Before using this cycle:

- The probe must be in the spindle and calibrated.

Step 1:

- Please release emergency stop before jogging the stylus.
- The stylus must be positioned to the approximate corner and approximately 0.3" above the top surface.

Step 2:

- For parameter [W], please input the work offset number that you would like to set.
- For parameter [V], please select the corner type.
- For parameter [I], please input the incremental distance to move in X-axis, it's optional.
- For parameter [J], please input the incremental distance to move in Y-axis, it's optional.
- For parameter [K], please input the incremental distance to move in Z-axis, it's optional.
- For parameter [X], please input the adjustment to work offset in +/-X direction, it's optional.
- For parameter [Y], please input the adjustment to work offset in +/-Y direction, it's optional.

Step 3:

- Make sure all the parameters are set properly.
- Press "Execute", control will pop up a confirm window.
- Select "OK" to start probing.

After using this cycle:

—When auto-probing is finished, the value of work offset will be updated automatically.

2.7.4.8 Single Touch

The screen of **Single Touch** is as below:

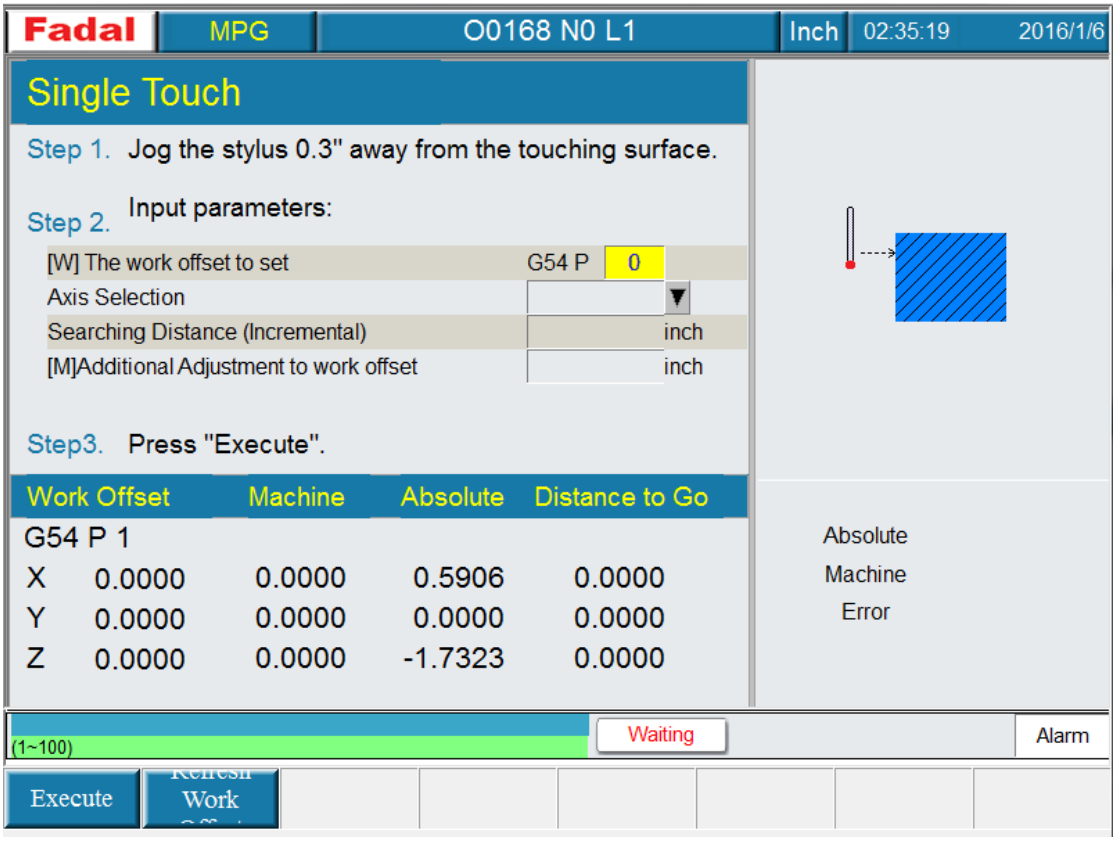


Figure 2-49 Probe Function–Single Touch

The **Single Touch** is for finding a surface. The work offset for specified axis will be at the surface.

Before using this cycle:

- The probe must be in the spindle and calibrated.

Step 1:

- Please release emergency stop before jogging the stylus.
- The stylus must be positioned to approximately 0.3” away from the surface.

Step 2:

- For parameter [W], please input the work offset number that you would like to set.
- For axis selection, please select the direction you would like to measure.
- For searching distance, please input the maximum possible distance from stylus to surface.
- For parameter [M], please input the adjustment to work offset in the measured direction, it’s optional.

Step 3:

- Make sure all the parameters are set properly.
- Press “Execute”, control will pop up a confirm window.
- Select ”OK” to start probing.

After using this cycle:

—When auto-probing is finished, the value of work offset will be updated automatically.

2.7.4.9 Angle Measurement

The screen of **Angle Mrasurment** is as below:

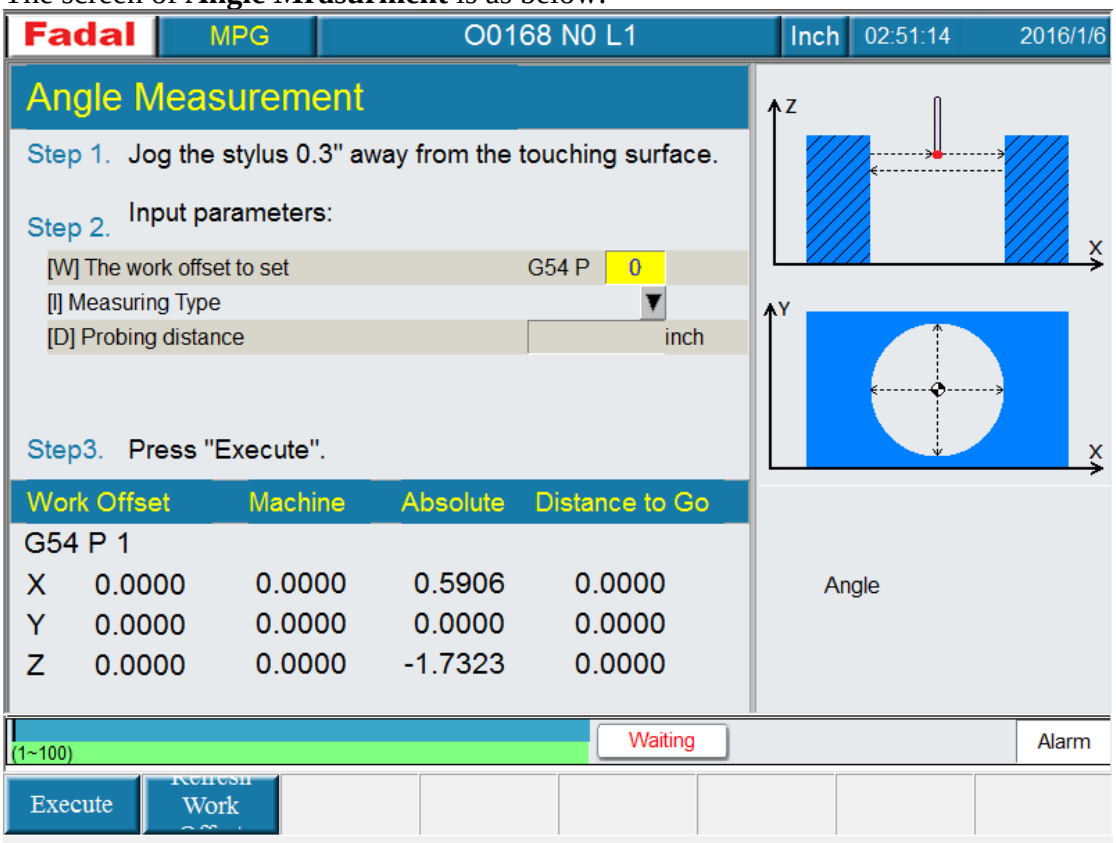


Figure 2-50 Probe Function–Finding Inside (Outside) Corner

The **Angle Measurement** is for finding a angle offset for a specified surface. The result will be shown on the right bottom corner of screen.

Before using this cycle:

- The probe must be in the spindle and calibrated.

Step 1:

- Please release emergency stop before jogging the stylus.
- The stylus must be positioned to approximately 0.3” away from the surface.

Step 2:

- For parameter [W], please input the work offset number that you would like to set.
- For parameter [I], please select the direction you would like to measure.
- For parameter [D], please input the maximum possible distance from stylus to surface.

Step 3:

- Make sure all the parameters are set properly.
- Press “Execute”, control will pop up a confirm window.

- Select ”OK” to start probing.

After using this cycle:

—When auto-probing is finished, the result will be shown on the right bottom corner of screen.

2.8 Set Parameters Screen

Set parameters screen can be reach by pressing or selecting **Set Parameters** softkey. The screen is similar as Figure 2-45.

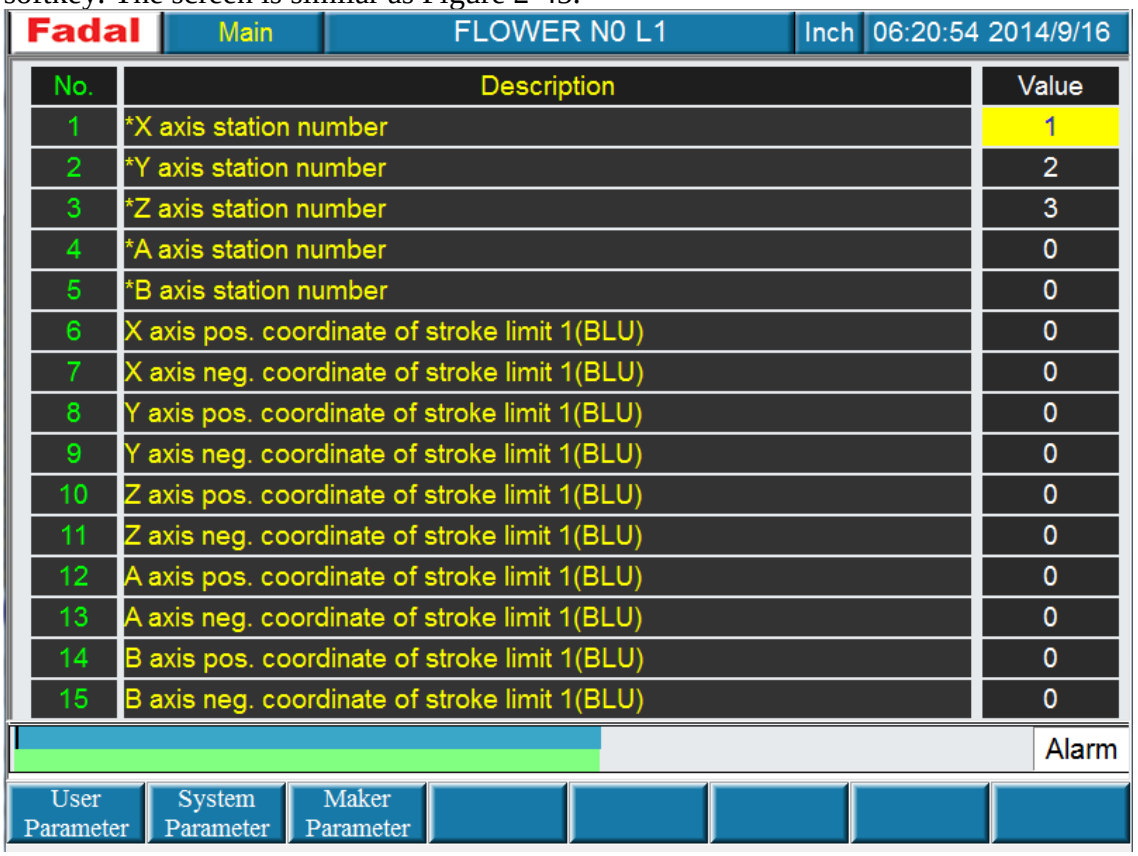


Figure 2-51 Set Parameters Screen

The parameters are divided into:

i. User Parameters
ii. System Parameters
iii. Maker Parameters

Only user parameters and system parameters will be described in this manual.

2.8.1 User Parameters

Parameter No	Name	Description
1-5	Axis station number	Defines the station number of each axis.
6-10	Axis pos. coordinate of stroke limit 1(BLU)	Software pos. stroke limit. Enabled after machine homed. For absolute encoder, enabled once power up.

		Note: BLU = 0.001mm
11-15	Axis neg. coordinate of stroke limit 1(BLU)	Software neg. stroke limit. Enabled after machine homed. For absolute encoder, enabled once power up. Note: BLU = 0.001mm
16	Check spindle speed arrival (0:Disable; 1:Enable)	This parameter used to enable checking spindle speed arrival function.
17	Coolant preference	0:M7 Flood Coolant, M8 Mist 1:M7 Mist, M8 Flood Coolant
18	*Initial Command Mode (0:default; 1:G90; 2:G91)	Default is G90

2.8.2 System Parameters

Parameter No	Name	Description
19	Cutting acceleration time (ms)	Together with parameter no.22, define the maximum cutting acceleration. $A_{max} = \frac{\frac{\text{No. 22}}{60}}{\frac{\text{No. 19}}{1000}} (mm/sec^2)$
20	Acceleration accelerated to 1G time (Jerk) (ms)	$J_{max} = \frac{9.8}{\frac{\text{No. 20}}{1000}} (mm/sec^3)$
21	Post acceleration bell-shaped time (ms)	This parameter smooth the command of the control, recommended value is 10~30ms
22	Maximum cutting federate (mm/min)	Together with parameter no.19, define the maximum cutting acceleration. $A_{max} = \frac{\frac{\text{No. 22}}{60}}{\frac{\text{No. 19}}{1000}} (mm/sec^2)$
23	Maximum corner reference feedrate (mm/min)	Maximum feedrate as refer to 120 deg. corner
24	Arc cutting reference federate at radius 5 mm (mm/min)	Maximum cutting feedrate as refer to arc of 5mm radius
25-29	Axis rapid travel F0 feedrate (mm/min)	Axis rapid travel feedrate when the override switch is

		F0. Default 0.
30-34	Axis JOG feedrate (mm/min)	Axis Jog federate
35-39	Axis 2nd reference point (BLU)	BLU–0.0001inch/ 0.001mm / 0.001deg
40	1st spindle zero floating speed (RPM)	When PLC flag C60 is ON, the control will force the spindle to move with this speed. Regardless the spindle override and spindle minimum speed.
41	1st spindle maximum speed (RPM)	-
42	1st spindle motor acceleration time (ms)	-
43	1st Spd. Acc. Acced. to 1000 RPM/s time (Jerk) (ms)	-
44	*Screen saver time latency (min)	Auto enter screen saving mode
45-47	Home Position (mm)	In format 1, once cycle start is pressed the control shall move all axes to home position.
48-50	Load Position (mm)	When **Load Position** softkey in Jog function bar is pressed
51	Homing Speed (mm/min)	In format 1, once cycle start is pressed the control shall move all axes to home position.
52	Format (1 / 2 / 3)	Format 1 : Fadal original Format 1 Format 2 : Fadal original Format 2 Format 3 : Fanuc compatible
53	Imperial (70) / Metric (71)	-
54	Bucket Number	Total pot number in tool magazines
55	Tool change Z Aixs position (M6)	-
56	Tool change Z Axis position check	-
57	Tool release sensor check time (0.01s)	Used in M6
58	Tool clamp sensor check time (0.01s)	Used in M6
59	ARM 60 deg. sensor check time (0.01s)	Used in M6
60	ARM 240 deg. sensor check time (0.01s)	Used in M6
61	X Axis Rapid Travel F0 Feedrate	-

	(mm/min)	
62	Y Axis Rapid Travel F0 Feedrate (mm/min)	-
63	Z Axis Rapid Travel F0 Feedrate (mm/min)	-
64	Engineering Mode – Mode Selection	0 : Off 1 : ON 2 : Observe

2.9 Diagnostic Screen

Diagnostic screen can be reach by pressing or selecting **Diagnostic** softkey.

Diagnostic screen consists of four different function:

i. Status – Observe the I/O status
ii. Calibration – Perform mechanism calibration
iii. Faults – To diagnose system faults information
iv. Admin – System Admin
v. ABOUT –Controller Software version check

2.9.1 Status Softkey

Pressing of selecting **Status** softkey will bring up a window to show the machine I/O real time status. Figure 2-46 shows an example of one of the status pages.

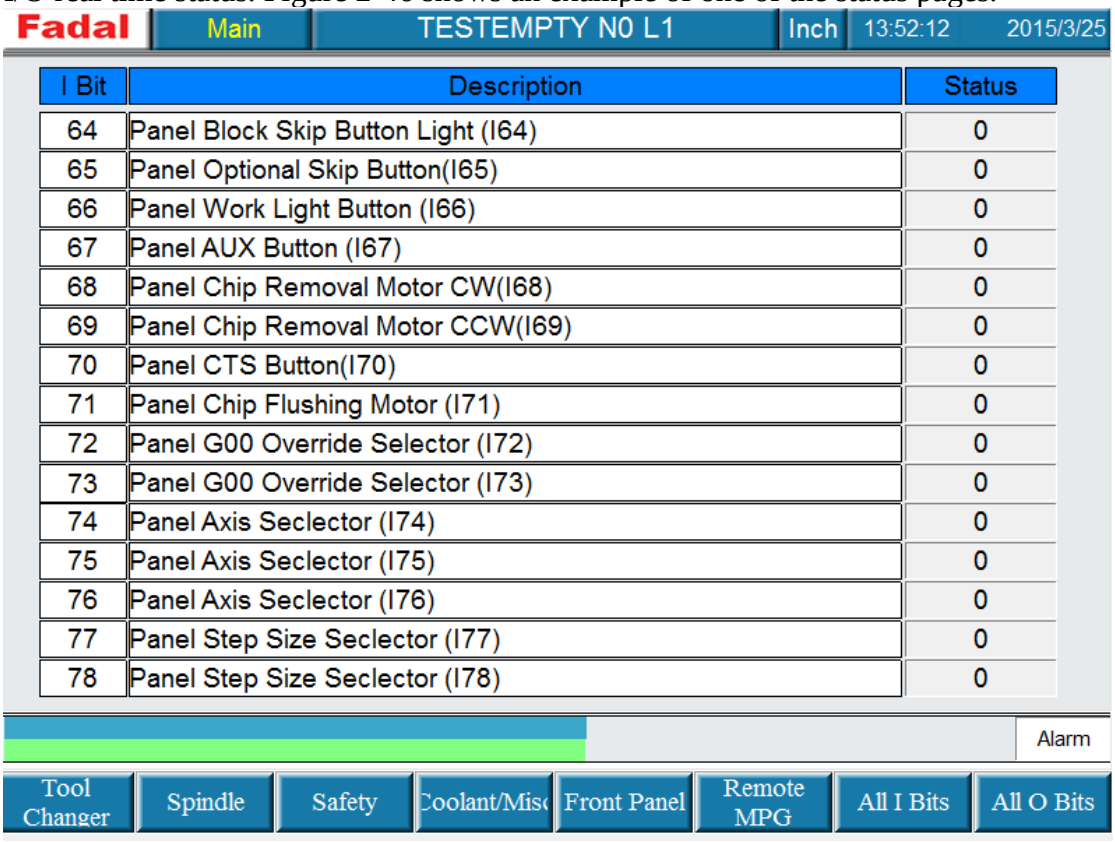


Figure 2-52 Diagnostic - Status

2.9.2 Calibration Softkey

Pressing of selecting **Calibration** softkey will bring up a window to show the mechanism calibration information. Figure 2-47 shows the main page of the calibration screen.

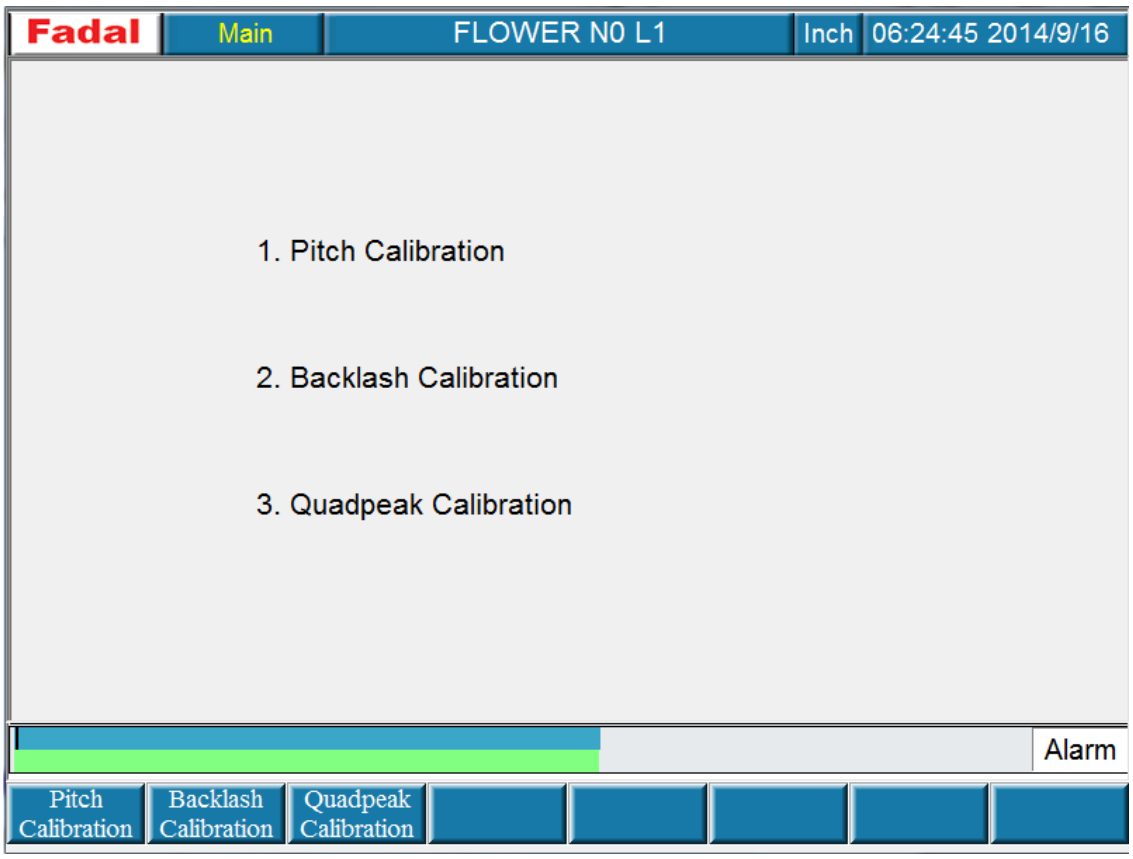


Figure 2-53 Diagnostic – Calibration

The Fadal 64 MP control offer three way to calibrate the mechanism error, includes:

i. Pitch Calibration
ii. Backlash Calibration
iii. Quadpeak Calibration

Details calibration method is not introduced in this user manual.

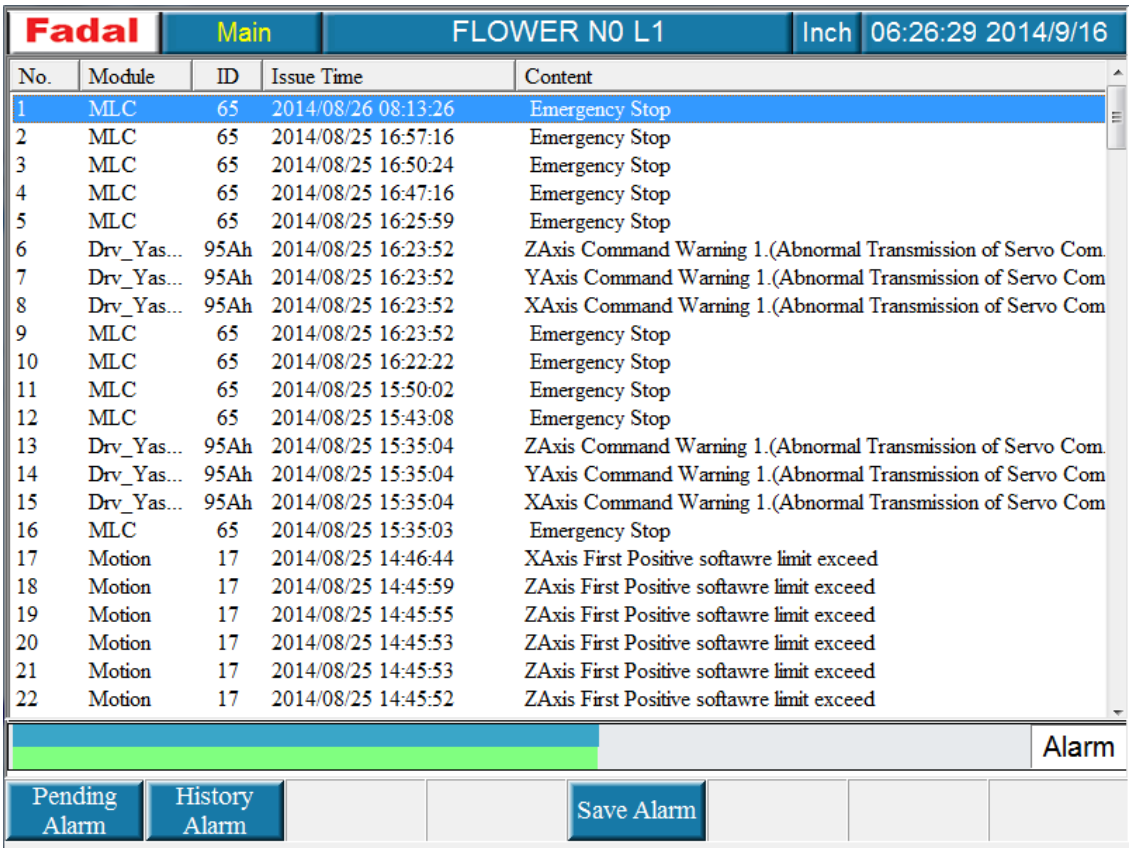
2.9.3 Faults Softkey

Pressing of selecting **Faults** softkey will bring up a function bar which consists two softkey as shown in Figure 2-48.

Figure 2-54 Diagnostic – Faults

2.9.3.1 Faults Log Softkey

Pressing of selecting **Faults Log** softkey will bring up a window to show the current/history faults. As seen in Figure 2-49.



No.	Module	ID	Issue Time	Content
1	MLC	65	2014/08/26 08:13:26	Emergency Stop
2	MLC	65	2014/08/25 16:57:16	Emergency Stop
3	MLC	65	2014/08/25 16:50:24	Emergency Stop
4	MLC	65	2014/08/25 16:47:16	Emergency Stop
5	MLC	65	2014/08/25 16:25:59	Emergency Stop
6	Drv_Yas...	95Ah	2014/08/25 16:23:52	ZAxis Command Warning 1.(Abnormal Transmission of Servo Com.
7	Drv_Yas...	95Ah	2014/08/25 16:23:52	YAxis Command Warning 1.(Abnormal Transmission of Servo Com
8	Drv_Yas...	95Ah	2014/08/25 16:23:52	XAxis Command Warning 1.(Abnormal Transmission of Servo Com
9	MLC	65	2014/08/25 16:23:52	Emergency Stop
10	MLC	65	2014/08/25 16:22:22	Emergency Stop
11	MLC	65	2014/08/25 15:50:02	Emergency Stop
12	MLC	65	2014/08/25 15:43:08	Emergency Stop
13	Drv_Yas...	95Ah	2014/08/25 15:35:04	ZAxis Command Warning 1.(Abnormal Transmission of Servo Com.
14	Drv_Yas...	95Ah	2014/08/25 15:35:04	YAxis Command Warning 1.(Abnormal Transmission of Servo Com
15	Drv_Yas...	95Ah	2014/08/25 15:35:04	XAxis Command Warning 1.(Abnormal Transmission of Servo Com
16	MLC	65	2014/08/25 15:35:03	Emergency Stop
17	Motion	17	2014/08/25 14:46:44	XAxis First Positive softawre limit exceed
18	Motion	17	2014/08/25 14:45:59	ZAxis First Positive softawre limit exceed
19	Motion	17	2014/08/25 14:45:55	ZAxis First Positive softawre limit exceed
20	Motion	17	2014/08/25 14:45:53	ZAxis First Positive softawre limit exceed
21	Motion	17	2014/08/25 14:45:53	ZAxis First Positive softawre limit exceed
22	Motion	17	2014/08/25 14:45:52	ZAxis First Positive softawre limit exceed

Figure 2-55 Faults – Faults Log

Each fault consist of the following information:

i. Module : MLC/MACRO/OP/MOT/SPD/COM/COR
   - ⇨ MLC : PLC alarm
   - ⇨ MACRO : MACRO alarm
   - ⇨ OP : Operation alarm
   - ⇨ MOT : Motion alarm
   - ⇨ SPD : Spindle alarm
   - ⇨ COM : Compiler alarm
   - ⇨ COR : Coordinate alarm

ii. ID : The alarm ID

iii. Issue Time : The time and date when the alarm occur

iv. Content : The information of the alarm

When an alarm occur, please search for a troubleshooting method through troubleshooting manual by the module and alarm ID.

2.9.3.2 Operation Log Softkey

Pressing of selecting **Operation Log** softkey will bring up a window to show the operation record of user. This information is always useful for troubleshooting.

3 Basic Operation of the CNC Control

3.1 Power-Up the Machine

The general procedures for power-up the machine are as:

1. Turn on the main power of the machine
2. Push the "Power" button to power off the control
3. Reset the "Emergency Stop" button (it should be pushed in last power off procedure)
4. The control should be in "Waiting" status when there is no faults occur

3.2 Homing the Machine

Homing the machine is a procedure that will take the axes to a known position on the machine. This known position is typically called the Home position.

Some machines are equipped with absolute feedback. This allows the machine to start up and remember its current machine coordinates. Machines so equipped do not require a Homing procedure.

If Homing is required it must be completed before the CNC can execute a part program in Auto or MDI mode.

3.3 Jogging the Machine

Generally, the machine must be homed before it can be jogged. The completion of the homing sequence results in the creation of software travel limits. These limits allow the CNC to stop the motion an axis prior to a hardware travel limit, which generally results in an Emergency Stop condition. The general procedure for jogging the machine:

1. Reset the Emergency Stop push button if it is lock on.
2. Go to the JOG Mode. (See section 2.4)
3. Select the axis to be jogged.
4. Select the Step Size.
5. Select the active direction (if *JOG* is used instead of the Handwheel)
6. Rotate the Handwheel in the desired direction/ Press the *JOG* on the keyboard.

3.4 Setting the Tools

3.4.1 Tool length setting

Generally, the tool length does not refer to the physical length of the tool, but the machine coordinate of touching the part surface with that tool. To general for setting up a tool length:

1. Bring the desired tool to spindle
2. Go to the JOG Mode. (See section 2.4)
3. Jog the machine to the surface of the part
4. Pressing **Set Tool** softkey and enter the current machine coordinate
5. Pressing *Enter* on the Fadal Keyboard
6. The tool length will be set.

Sometimes it requires to setting up various tool lengths, a more efficient method to do that is to use the tool setup function (see section 2.6.1.5). Tool setup function

enhances the operator to setting up various tool lengths continuously.

3.4.2 Tool Diameter setting

The tool diameter refers to the physical diameter of the tool. To set the tool diameter into the control, follow up the procedures as below:

1. Bring up the tool offset setting screen (See section 2.6.1)
2. Move the cursor to the tool radius input box of the desired tool
3. Enter the tool radius and press *Enter*
4. The tool radius will be set.

Note: Tool radius is used instead of tool diameter.

3.5 Setting Up the Work Offsets

A work offset generally refer to the offset of the part. The operator can set up the work offset manually through the work offset setting screen (see section 2.6.2). Instead of manually setting up work offset, the control also offers several general ways to set up the work offset (see section 2.7.2).

3.6 Running a Program with the Machine

This section describes the procedure to managing and running the programs in CNC.

3.6.1 Create a New Program

To create a new program in CNC, the general procedures are as:

1. Bring up the edit screen (see section 2.5)
2. Press the **Directory** softkey
3. Press the **New File** softkey and enter an arbitrary name for the new program
4. Pressing *Enter* on the Fadal Keyboard will create a new program

3.6.2 Import/ Export the Programs

The general procedures of import/ export the programs are as:

1. Insert a pen disk or other media into the USB port or connect the CNC to a network
2. Bring up the edit screen (see section 2.5)
3. Press the **Directory** softkey
4. Press the **File Export/Import** softkey
5. Choose the desired programs from the device/CNC
6. Press **Copy** softkey to import/export the selected programs

3.6.3 Delete a program in the CNC

The general procedures of deleting a program in CNC memory are as:

1. Bring up the edit screen (see section 2.5)
2. Press the **Directory** softkey
3. Press the **Delete** softkey
4. Choose the desired programs to be deleted
5. Press **Delete** softkey to delete the selected programs

3.6.4 Run a Program from beginning

The general procedures for running a program on the machine from the beginning are as:

1. Reset the Emergency Stop condition.
2. Bring up the auto screen (see section 2.2)
3. Press the **Directory** softkey
4. Choose the desired programs to be run
5. Select **Single Step/MPG Dry Run** softkey if the program is to be run one block at a time/ run with Handwheel rotating speed
   Note: there will be "Single Step" blinking on the screen
6. Push the "Cycle Start" button
7. As long as the feedrate override pot is not at 0%, the program will continue to its completion. Moving the federate override pot to 0% will place the machine in a feedhold condition. When the override is returned to a position greater than 0% the axis motion and program execution will resume. Pressing the "Slide Hold" button will put the CNC in a "Slide Hold" state. To resume the program, press the "Cycle Start" button. The CNC will be in the "Running" state once the program is executing. The active executed block will be highlighted.
8. When the program has completed, the CNC will back to "Waiting" status. indicating that nothing is happening in Automatic mode. The program that just ran will be rewound to its beginning and the first block will be shown on the top of the NC blocks region of the Auto mode display.

Remarks:
There are a number of features available to run a program in different ways.

- The program can be run to stop on a block with M01 (the default is to ignore that M-code). This feature is enabled by pushing the "Optional Stop" button.
- The program can be run to skip over blocks that start with / (the default is to run those blocks). This feature is enabled by pushing the "Block Skip" button

3.6.5 Run a Program with Mid Program

In some situations the executing program is forced to stop in the mid program, such as tool break during the execution of the program. After the operator dissolves the situation, one might wish to start the program from the mid program instead of the beginning as a reason of time consuming. The general procedures for running a program on the machine from the mid program are as:

1. Reset the Emergency Stop condition.
2. Bring up the auto screen (see section 2.2)
3. Press the **Directory** softkey
4. Choose the desired programs to be run
5. Select **Single Step/MPG Dry Run** softkey if the program is to be run one block at a time/ run with Handwheel rotating speed
   Note: there will be "Single Step" blinking on the screen
6. Press the **Mid Program** softkey (See section 2.2.6)
7. Press the **Search** softkey to search for specific string if needed
8. Press the **Scan/No Scan** softkey to decide mid program start method
9. Press the **Start From Line No.** or **Start From Cursor Line** softkey to start

from mid program
10. Push the “Cycle Start” button to start the program

3.7 Entering MDI

The general procedures for MDI are as:

1. Reset the Emergency Stop condition.
2. Bring up the mdi screen (see section 2.3)
3. Press the **MDI Input** softkey
4. Type the desired commands (M codes, G codes, Axes words, etc)
5. Press *Enter* key on the keyboard. This will move the data block to the top line in the NC region of the MDI display
6. Press the “Cycle Start” button to start the execution of the entered data
7. Type each subsequent block and press *Enter* key on the keyboard. The CNC will continue to execute the data block in the order they were entered.

3.8 Switching Spindle High/Low Range

In Fadal 64MP control, spindle high/low range will be switched automatically according to the programmed spindle RPM. The spec of range switching is as below:

1. Low Range : 0-2500RPM
2. High Range : 0-10000RPM
3. Range switching will be proceed only when the control encounter M03/M04 function.
4. When M03/M04 encounter, with S#, where # is lower than 2500RPM, the control shall stop the spindle, and switch to low range, then proceed spindle rotation to programmed speed.
5. When M03/M04 encounter, with S#, where # is higher than 2500RPM, the control shall stop the spindle, and switch to high range, then proceed spindle rotation to programmed speed.
6. Any other causes that change spindle RPM beside M03/M04 function shall **NOT** switch spindle range. For example, when the spindle is turning with 2200RPM, where the spindle override is at 100%. Switching the spindle override to more than 110% (e.g.200%) will **NOT** cause spindle high/low range changing. The spindle will remain in low range and with its max speed, i.e. 2500RPM.
7. The control shall switch to low range once initialized, i.e. power up and emergency stop push button released.
8. When spindle is at max speed of current range, i.e. 2500RPM for low range and 10000RPM for high range. The spindle override on HMI shall display “MAX” instead of the current spindle override %.

3.9 Connect the CNC to a network

In many circumstances import/export files from CNC to PC is desired. Instead of using an external device such as USB drive, connect the CNC to a network is more convenient for frequent import/export files operator. To connect a CNC to PC, it requires some setting in both sides and will be introduced in this section.

3.9.1 PC setting

Window XP Setting

A. Guest account setting

Login as **"Administrator"**, click on **"Start"** → **"Control Panel"** → **"User Accounts"**, open the guest account.

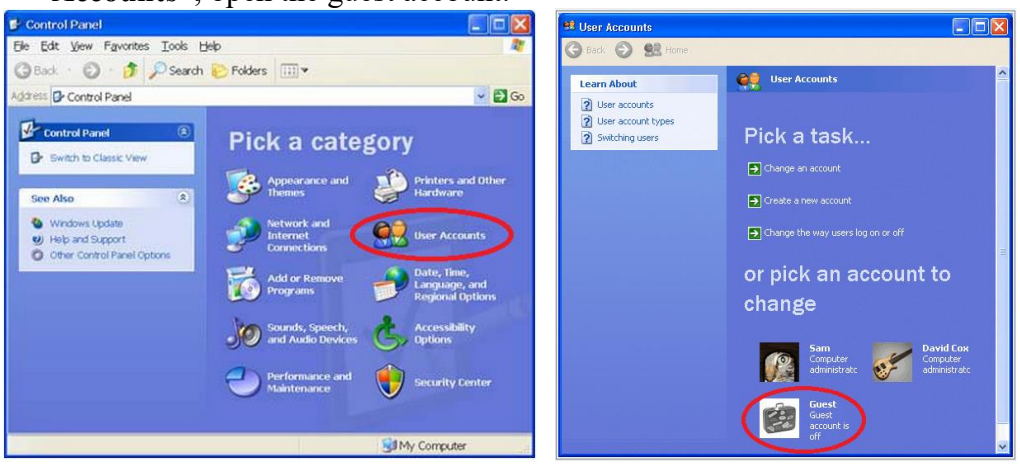


Figure 3-1 Window XP setting (1)

B. Resource files sharing setting

1. Open **"My Computer"** and find the folder that you want to share. Right click on it and select **"Sharing and security".**

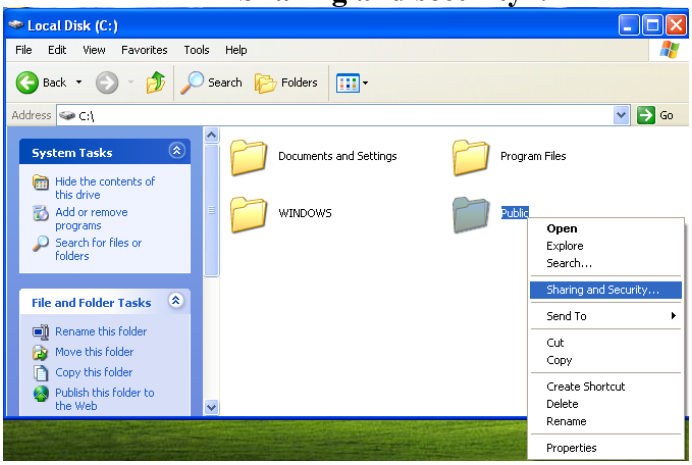


Figure 3-2 Window XP setting (2)

2. Click **"If you understand security risks but want to share files without running the wizard, click here"** and then Click **"OK"** to confirm sharing setting.

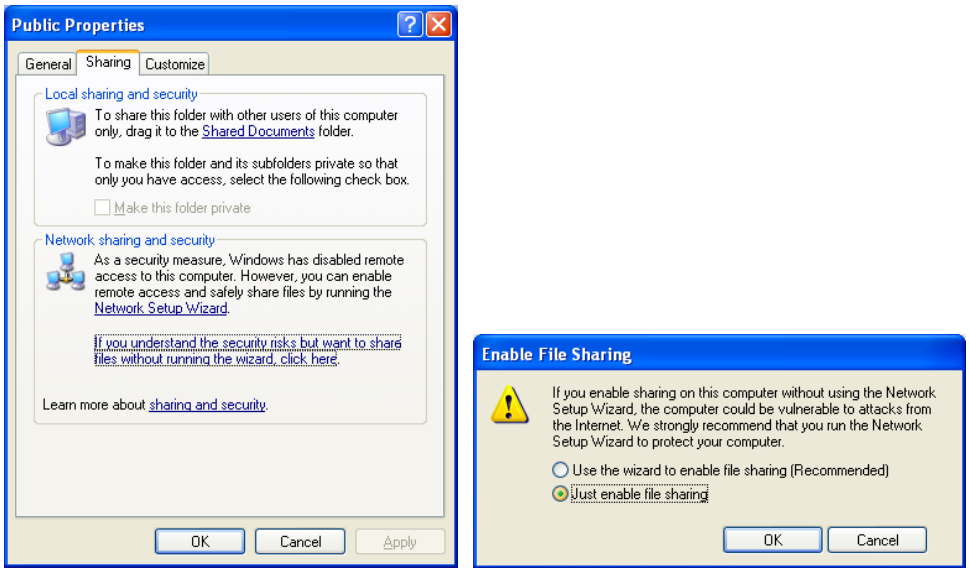


Figure 3-3 Window XP setting (3)

3. Select **"Share this folder on the network"** and **"Allow network users to change my files"** and then key folder name and click **"OK"**.

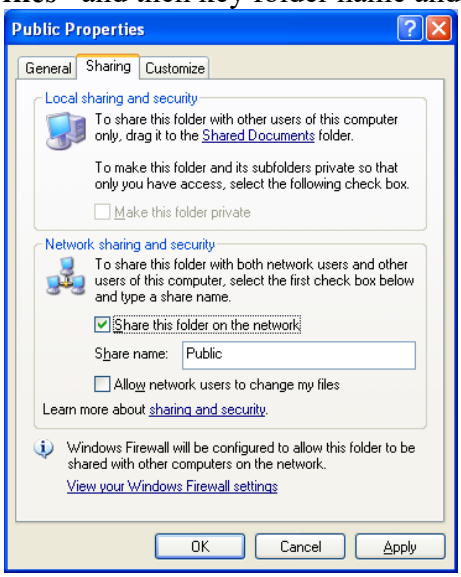


Figure 3-4 Window XP setting (4)

C. PC information acquire

Click **"Start"** → **"Control Panel"** → **"System"** → **"Computer Name"** and then write down the **"Computer Name"** and **"Working Group"** which are necessary information during CNC setting.

D. TCP/IP setting

1. Login as **"Administrator"**, click on **"Start"** → **"Control Panel"** → **"Network Connections"**, double click **"Internet Protocol (TCP/IP)".**
2. Jump Wire (without HUB)
   Select **"Use the following IP address"** and fill in **"IP Address"** which's the fourth IP code has to be different from CNC setting and **"Subnet mask"** which should be identical to CNC setting.

3. General network wire (with HUB)
   Select **"Obtain an IP address automatically"**.

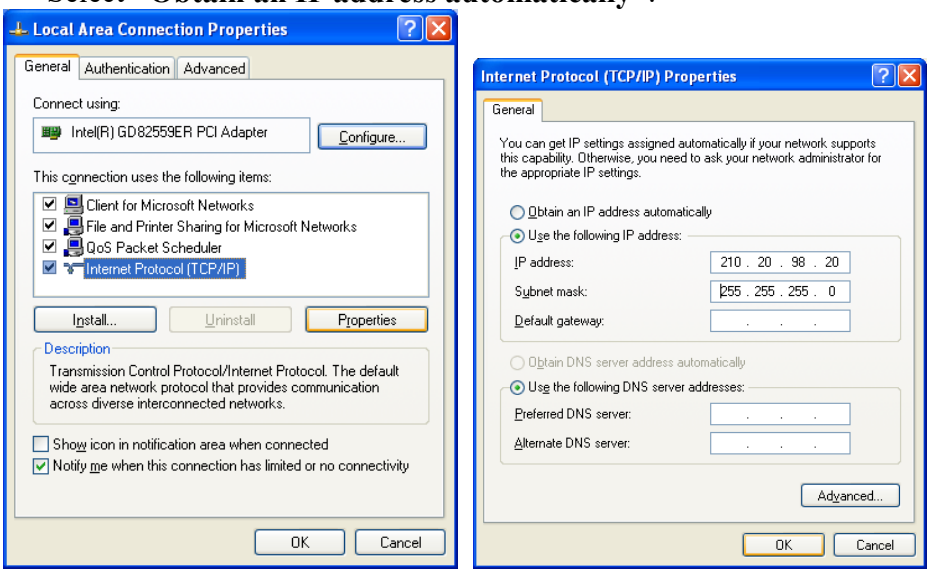


Figure 3-5 Window XP setting (5)

VISTA Setting

A. Guest Account Setting

Login as **"Administrator"**, click on **"Start"** → **"Control Panel"** → **"User Accounts"**, open the guest account.

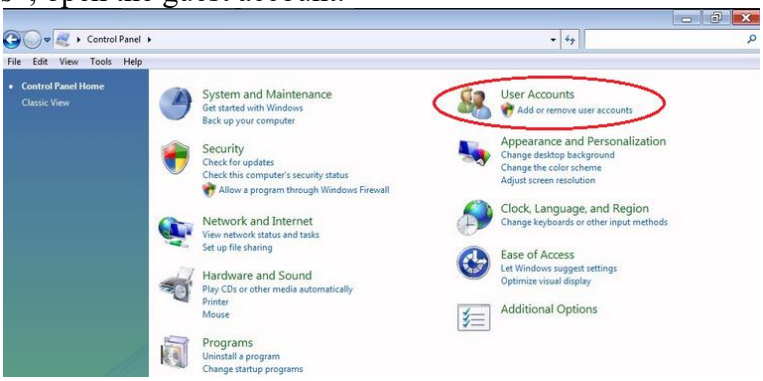


Figure 3-6 Vista setting (1)

B. Resource files sharing setting

1. Open **"My Computer"** and find the folder that you want to share. Right click on it and select **"Properties"**. Click on **"Advanced Sharing"**under the **"Sharing"**, as shown below.
2. Select **"Share this folder"**.
3. Click on **"Permissions"** to add a new group named **"Guest"**, and click on **"OK".**

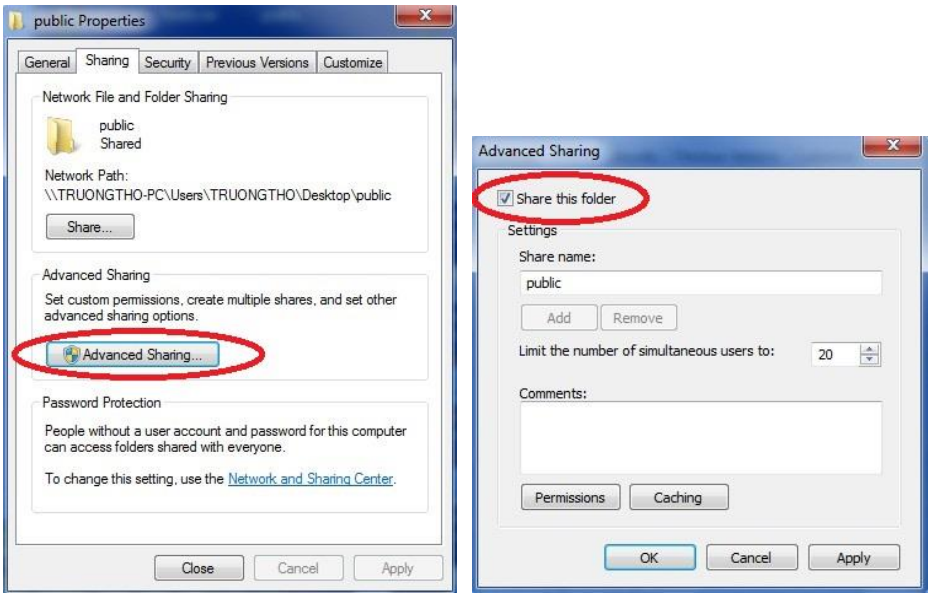


Figure 3-7 Vista setting (2)

C. Extend Security Setting

Similarly open the **"Properties"**, click on **"Edit"** under the **"Security"**, then add a new group named **"Guest"**. Extend the permission of **"Guest"** to maximum.

D. PC information acquire

Click **"Start" → "Control Panel" → "System" → "Computer Name"** and then write down the **"Computer Name"** and **"Working Group"** which are necessary information during CNC setting.

E. TCP/IP Setting

1. Login as **"Administrator"**, click on **"Start" → "Control Panel" → "Network and Dial-up connections"**, double click **"Local Area connection" → "Properties".**
2. Double click **"(TCP/IPv4)"**.
3. Jump Wire (without HUB)
   Select **"Use the following IP address"** and fill in **"IP Address"** which's the fourth IP code has to be different from CNC setting and **"Subnet mask"** which should be identical to CNC setting.
4. General network wire (with HUB)
   Select **"Obtain an IP address automatically"**.

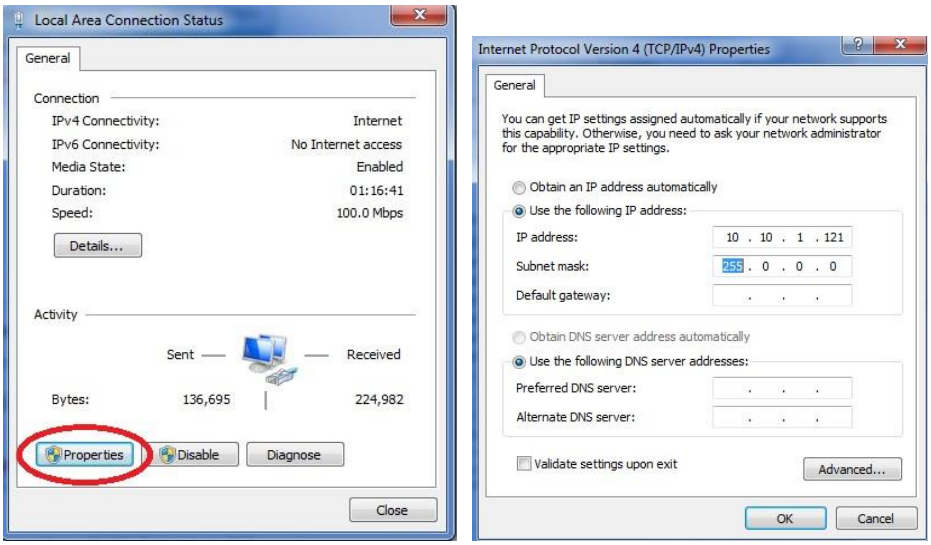


Figure 3-8 Vista setting (3)

Win7 Setting

A. Resource files sharing setting

1. Open **"My Computer"** and find the folder that you want to share. Right click on it and select **"Properties"** and then click the tab **"Sharing"**.
2. As shown below, click the button **"Share"** to change the Setting for sharing this folder for everyone.

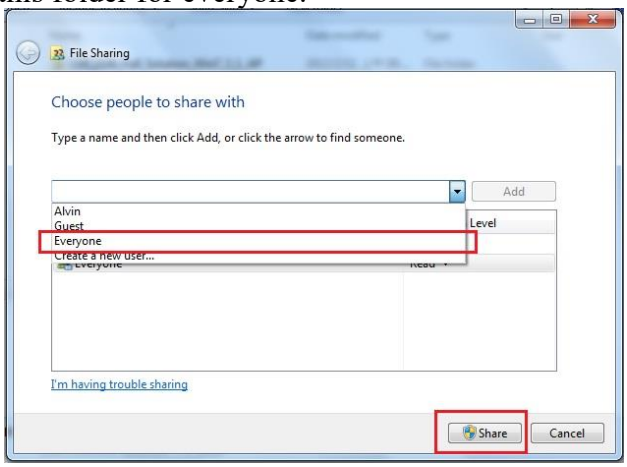


Figure 3-9 Win7 setting (1)

3. As shown below, click on **"Advanced Sharing"** and select **"Share this folder"**.

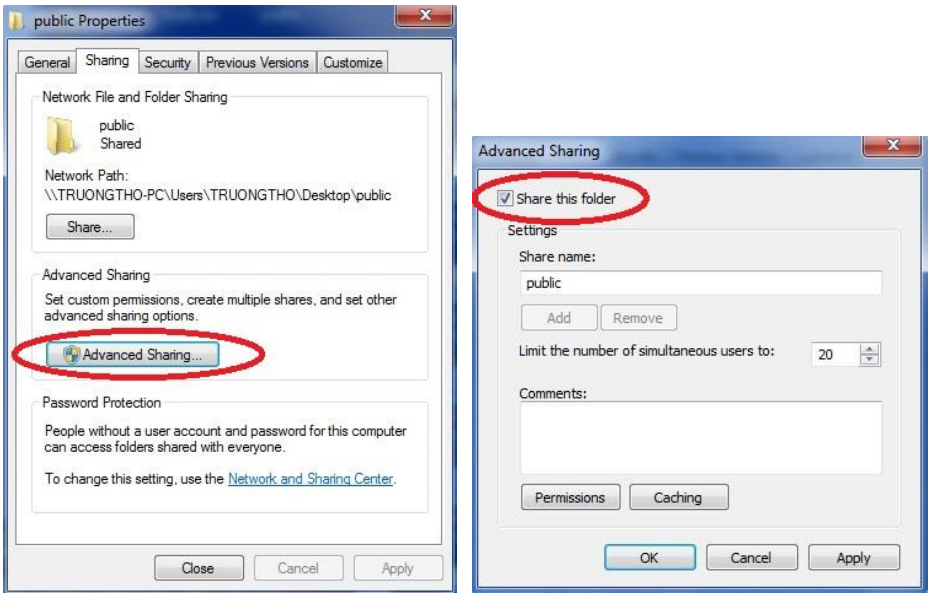


Figure 3-10 Win7 setting (2)

4. Click on **"Permissions"** and allow the permission to maximum.
5. Click on **"Network and Sharing Center"**.
6. Select **"Turn off password protected sharing"** and **"Use user account and passwords to connect to other computers".**

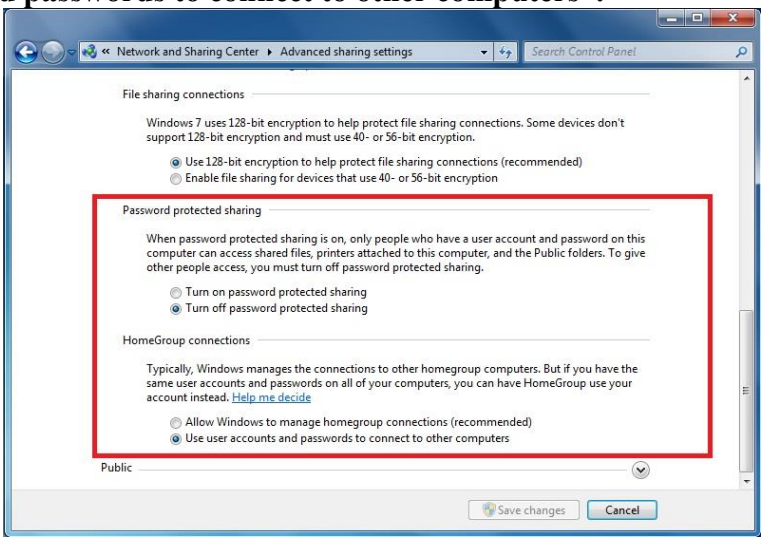


Figure 3-11 Win7 setting (3)

B. PC information acquire

Click **"Start"** → **"Control Panel"** → **"System"** → **"Computer Name"** and then write down the **"Computer Name"** and **"Working Group"** which are necessary information during CNC setting.

C. TCP/IP Setting

1. Click **"Start"** → **"Control Panel"** → **"Network and Internet"** → **"Network and Sharing Center"**.

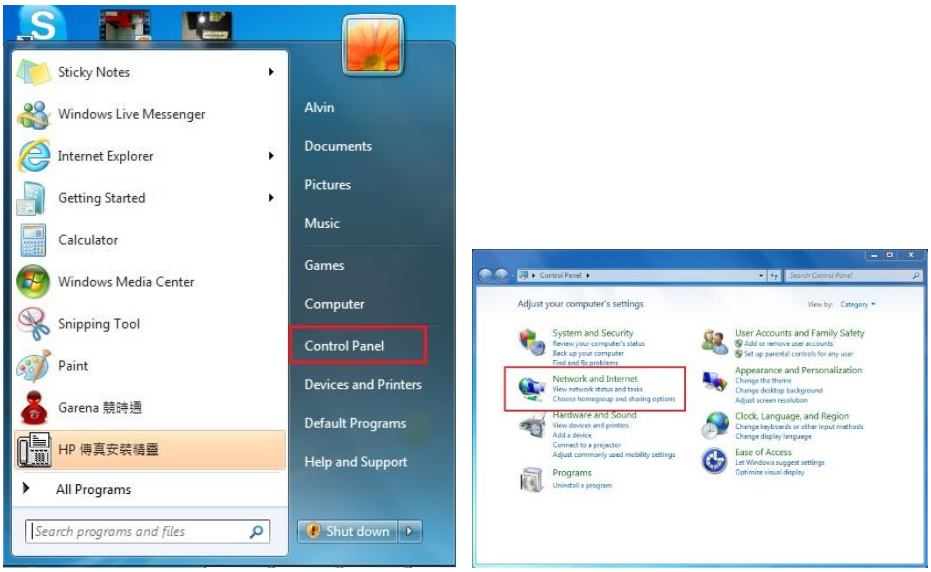


Figure 3-12 Win7 setting (4)

2. Click on **"Change adapter settings"** nad then right click **"Local Area connection"** and select **"Properties".**

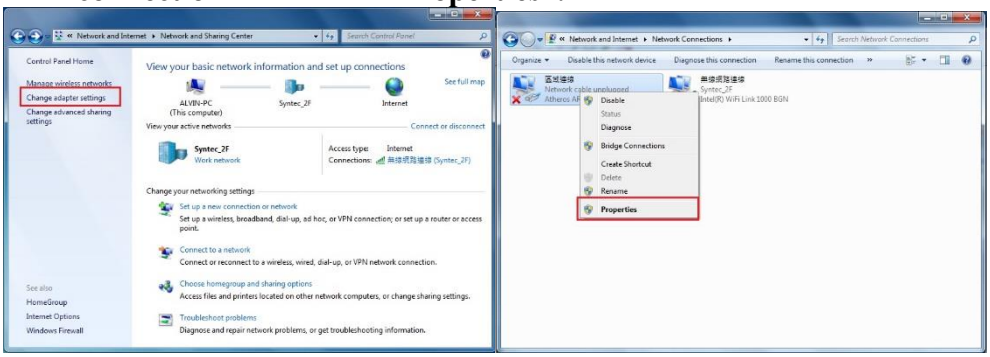


Figure 3-13 Win7 setting (5)

3. Double click **"(TCP/IPv4)"**.
4. Jump Wire (without HUB)
   Select **"Use the following IP address"** and fill in **"IP Address"** which's the fourth IP code has to be different from CNC setting and **"Subnet mask"** which should be identical to CNC setting.
5. General network wire (with HUB)
   Select **"Obtain an IP address automatically"**.

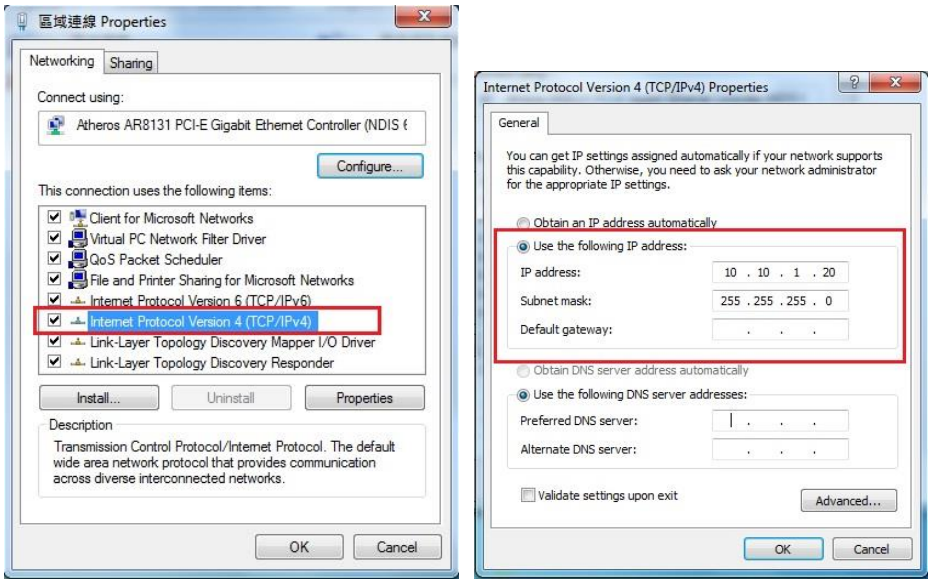


Figure 3-14 Win7 setting (6)

3.9.2 CNC setting

A. On the main screen of the control, press **Set Parameter** softkey

B. Press the **Maker Parameter** softkey and then enter the password

C. Press the "Next" softkey

D. Press the **Network Setting** softkey to bring up the network setting dialog box

E. **IP Address Setting**:

- ⇨ Select **"Specify an IP Address"** if the PC connects to controller directly, i.e., using a jump network wire(without HUB).
- ⇨ Select **"Obtain an IP Address via DHCP"** if using a network connection via Dynamic Host Configuration Protocol, i.e., using a general network wire (with HUB).

F. **IP Address**: If users select "Specify an IP Address", it is necessary to enter the IP address which only the fourth IP code is different from PC setting.

G. **Subnet mask**: If users select "Specify an IP Address", it is necessary to enter subnet mask which must identical to PC setting.

H. **PC Name**: enter the full computer name which must identical to PC setting.

I. **Dir Name**: enter the name of sharing folder which must identical to PC setting.

J. Press **OK** softkey and reboot the controller to activate the internet connection.

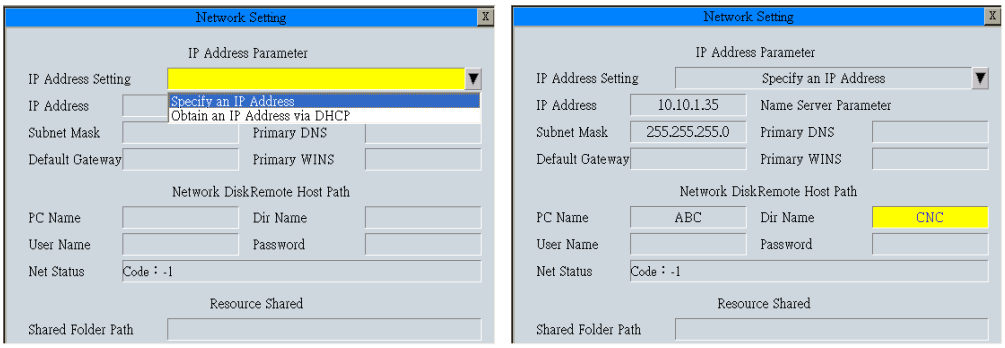


Figure 3-15 CNC setting

3.10 Software Update

In some circumstances system software update/install is required. The procedures of updating/installing the system software are as:

1. Put Update data package into a pen disk or the network sharing folder
2. Insert a pen disk or other media into the USB port or connect the CNC to a network
3. Press the **Diagnostic** softkey on the main function bar
4. Press the **Admin** softkey and enter the password
5. Press the **System Admin** softkey
6. Press the **Install Software** softkey to bring up a dialog box
7. Choose the install package and click **OK** softkey
8. Reboot the control to start install the software

3.11 System Backup/Restore

In some circumstances system backup/restore is required. The procedures are as:

1. Insert USB storate to USB port or connect the CNC to a network
2. Press the Diagnostic softkey on the main function bar
3. Press the Admin softkey and enter the password
4. Press the System Admin softkey and enter the password
5. Choose the System Backup/Restore softkey to bring up a dialog box
6. Choose the location to proceed system backup/restore

3.12 Power Off the Machine

It is always recommended to push the “E-Stop” button before power off the machine. The general procedures for power down the machine are as:

1. Make sure the machine is not in “Running” status
2. Push the “E-Stop” button
3. Push the “Power” button to power off the control
4. Shut down the main power of the machine

4 CNC Programming Manual

This chapter of the manual introduces the basic NC programming of the Fadal 64 MP control. This control supports three different formats of programming language:

1. Format 1 : Fadal format 1 compatible
2. Format 2 : Fadal format 2 compatible
3. Format 3 : Fanuc compatible

4.1 Common G codes

Description	Modal	Format1/2 G Codes	Format 3 G Codes
Rapid Traverse Mode	Yes	G00	G00
Linear Interpolation Mode	Yes	G01	G01
Clockwise Circular Interpolation Motion	Yes	G02	G02
Counter-Clockwise Circular Interpolation Motion	Yes	G03	G03
Non Modal Rapid Traverse	No	G05	-
Dwell	No	G04	G04
High Speed High Precision Interpolation	Yes	G5.1	G05
Acceleration (No Feed Ramps)	Yes	G08	-
Deceleration (Feed Ramps)	Yes	G09	-
In Position Check	No	G04	G09
Programmable Data Input	No	G10	G10
X-Y Plane Selection	Yes	G17	G17
Z-X Plane Selection	Yes	G18	G18
Y-Z Plane Selection	Yes	G19	G19
Imperial Mode Verification	No	G20/G70	-
Metric Mode Verification	No	G21/G71	-
Imperial Mode Define	Yes	-	G70
Metric Mode Define	Yes	-	G71
Return to Home Position	No	G28	G28
Cancel Jog Away	No	G28.1	-
Return from Home Position	No	G29	G29
Cutter Radius Compensation Cancelled	Yes	G40	G40
Cutter Radius Compensation Left	Yes	G41	G41
Cutter Radius Compensation Right	Yes	G42	G42
Positive Tool Length Compensation	Yes	G43	G43
Negative Tool Length Compensation	Yes	G44	G44
Tool Length Offset Single Expansion	No	G45	G45
Tool Length Offset Single Reduction	No	G46	G46
Tool Length Offset Double Expansion	No	G47	G47
Tool Length Offset Double Reduction	No	G48	G48
Tool Length Offset Cancel	Yes	G49	G49
Cancel Mirror Image	Yes	G50.1	G50.1

Enable Mirror Image	Yes	G51.1	G51.1
Axis Scaling	Yes	G51.3	G50
Axis Scaling Disable	Yes	-	G51
Coordinate System Shift	Yes	G52	G52
Machine Coordinates System	No	G53	G53
Apply Work Offset	Yes	G54~G59	G54~G59
Modal Subroutine Call	Yes	G66	G66
Cancel Modal Subroutine	Yes	G67	G67
Coordinate System Rotation	Yes	G68	G68
Coordinate System Rotation Cancel	Yes	G69	G69
Absolute Positioning	Yes	G90	G90
Incremental Positioning	Yes	G91	G91
Absolute Preset	Yes	G92	G92
Inverse Time Feedrate	Yes	G93	G93
Feedrate	Yes	G94	G94
Return to Initial Plane	Yes	G98	G98
Return to Rapid Plane	Yes	G99	G99

4.1.1 Rapid Traverse Mode

Command form:

G00 X_ Y_ Z_ ;

X, Y, Z: Specified point

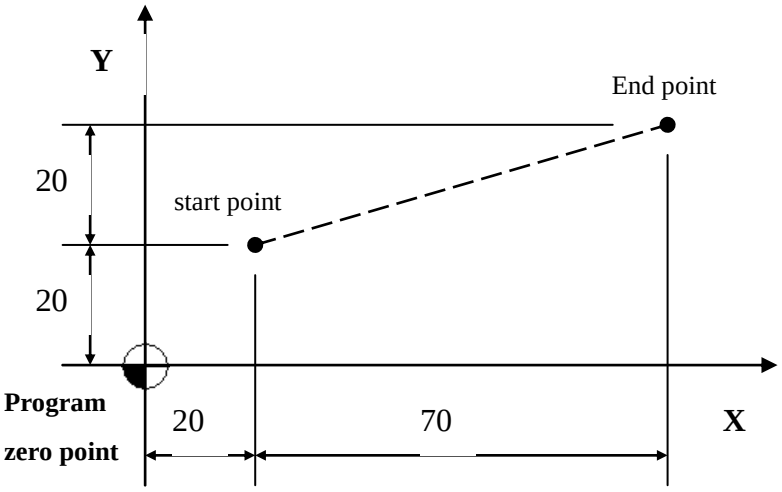
Description:

All axes move to appointed point with no interpolation status, use G90/G91 to decide absolute or increment value.

``

In format1/2, when the Z axis is to move in the positive direction, it moves prior to X, Y, A, and B axis motion. When the Z axis is to move in the negative direction, it moves after X, Y, A, and B axis motion.

Example:



1. Absolute input: G90 G00 X90.0 Y40.0；

2. Incremental input: G91 G00 X70.0 Y20.0；

4.1.2 Linear Interpolation Mode

Command form:

G01 X_ Y_ Z_ F_；

X, Y, Z: Specified point
F: Feed rate, Unit: mm/rev (inch/rev) for G95
mm/min (inch/min) for G94←default mode

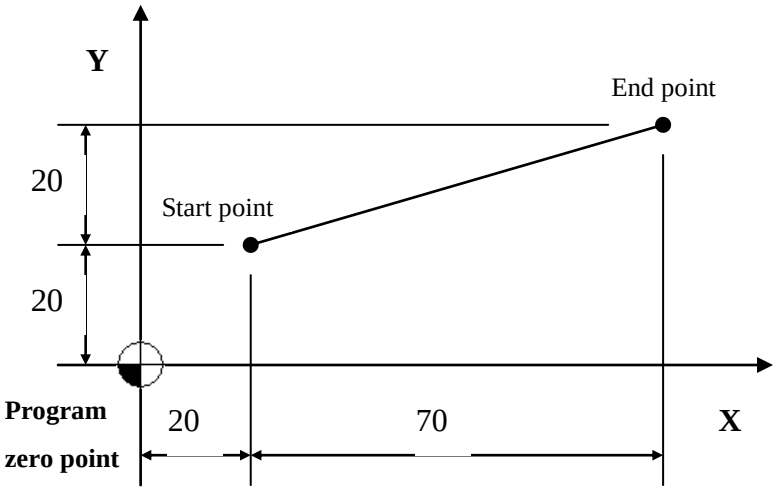
Description:

G01 executes linear interpolation, it can be used with G90/G91 to decide absolute or incremental mode, use the feedrate provided by **F** word to move to the specified position.

Note:

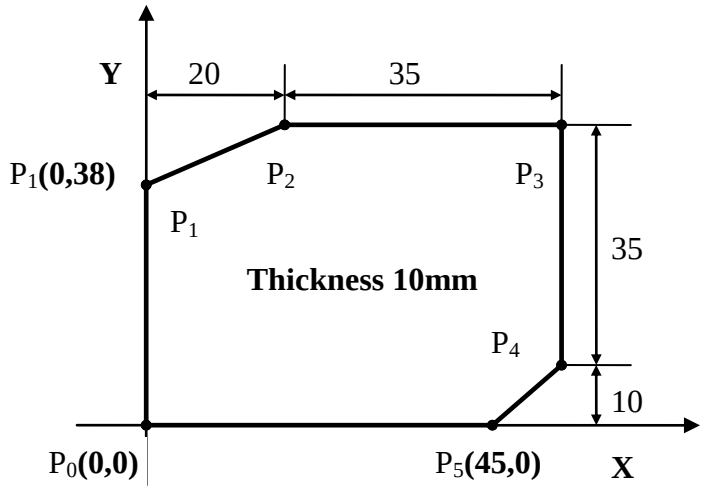
- The max. feedrate of G01 is defined by user parameter no.22-maximum cutting feedrate or maker parameter (Pr621~Pr636)-each axis maximum cutting feed rate
- Default value F: 1000mm/min(inch/min) for G94 mode and 1.mm/rev(inch/rev) for G95 mode
- Default mode G94/G95 can be changed by maker parameter Pr3836 (reboot controller to activate setting).

Example 1:



1. Absolute input: G90 G01 X90.0 Y40.0；
2. Incremental input: G91 G01 X70.0 Y20.0；

Example 2:



1. Absolute input:

N001 G90 G00 X0.0 Y0.0 Z10.0；//positioning to above of P_0

N002 G90 G01 Z-10.0 F1000；//straight interpolation to bottom of workpiece, speed 1000mm/min

N003 Y38.0；//P_0 → P_1

N004 X20.0 Y45.0；//P_1→ P_2

N005 X55.0；//P_2 → P_3

N006 Y10.0；//P_3 → P_4

N007 X45.0 Y0.0；//P_4 → P_5

N008 X0.0；//P_5 → P_0

N009 G00 Z10.0；//positioning back to above of P_0

N010 M30；//program end

2. Incremental input:

N001 G90 G00 X0.0 Y0.0 Z10.0；//positioning to above of P_0

N002 G91 G01 Z-20.0 F1000；//straight interpolation to bottom of workpiece, speed 1000mm/min

N003 Y38.0；//P_0 → P_1

N004 X20.0 Y7.0；//P_1→ P_2

N005 X35.0；//P_2 → P_3

N006 Y-35.0；//P_3 → P_4

N007 X-10.0 Y-10.0；//P_4 → P_5

N008 X-45.0；//P_5 → P_0

N009 G00 Z20.0；//positioning back to above of P_0

N010 M30；//program end

4.1.3 Circular Interpolation Motion

Command form:

1. X-Y plane circular interpolation:

$$G17 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X\_ \ Y\_ \left\{ \begin{matrix} R0\_ \\ I\_ \ J\_ \end{matrix} \right\} F\_;$$

2. Z-X plane circular interpolation:

$$G18 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} X\_ \ Z\_ \left\{ \begin{matrix} R0\_ \\ I\_ \ K\_ \end{matrix} \right\} F\_;$$

3. Z-X plane circular interpolation

$$G19 \left\{ \begin{matrix} G02 \\ G03 \end{matrix} \right\} Y\_ \ Z\_ \left\{ \begin{matrix} R0\_ \\ J\_ \ K\_ \end{matrix} \right\} F\_;$$

X, Y, Z: Specified point

I, J, K: the vector value that starting point of arc to the center of a circle(center of a circle - starting point)

R0: Radius of arc

F: Feed rate

G90/G91 decide absolute or increment

Note:

In format3, use R instead of R0.

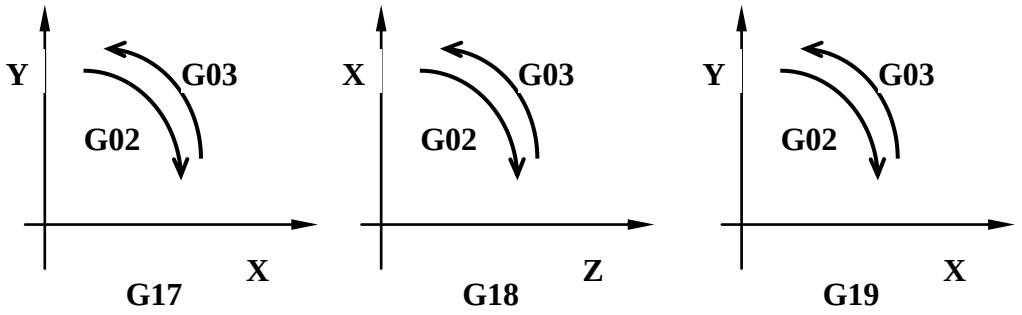
Description:

G02, G03 do circular interpolation according to appointed plane, coordinate system, size of arc, and speed of interpolation. The rotate direction is decided by G02(CW)、G03(CCW). Description of the command format as below:

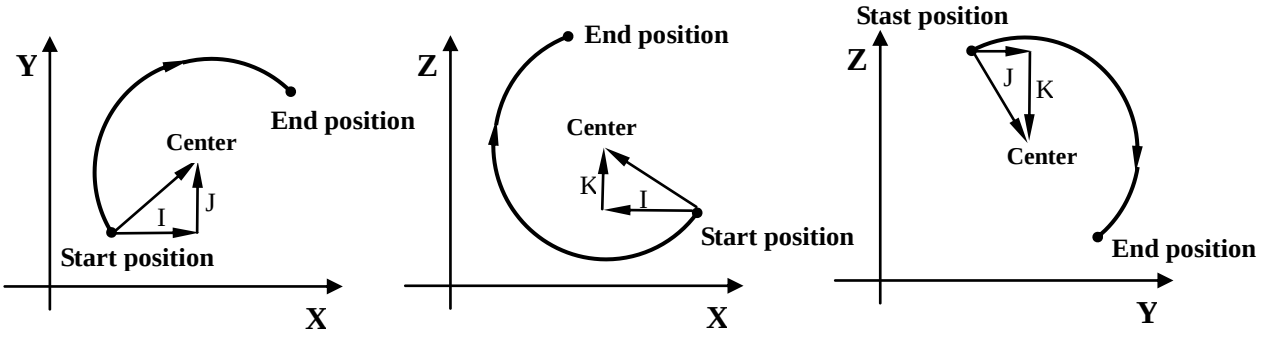
Setting Data			Command	Definition
1	Plane selection		G17	X-Y plane setting
			G18	X-Z plane setting
			G19	Y-Z plane setting
2	Direction		G02	Clockwise direction (CW)
			G03	Counterclockwise direction (CCW)
3	End position	G90	Two axes of X, Y, Z	End coordinate of arc
		G91	Two axes of X, Y, Z	Vector value from start point to end point

4	Distance from start point to center of circle	Two axes of I, J, K	Vector value from start of arc to center of circle
	Radius of arc	R	Radius of arc
5	Speed of feed (feedrate)	F	Feedrate along the arc

4. G02, G03 direction



5. I, J, K definition:



6. Use Radius Designation:

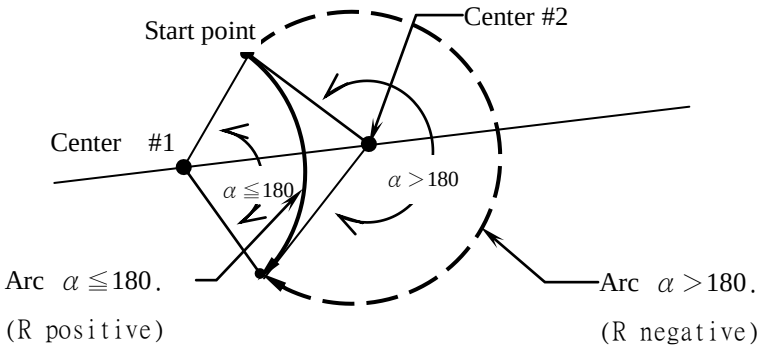
- When θ≦180 degree, R is positive.

$$\left\{\begin{matrix} G02 \\ G03 \end{matrix}\right\} \; X\_ \;\; Y\_ \;\; R25.0;$$

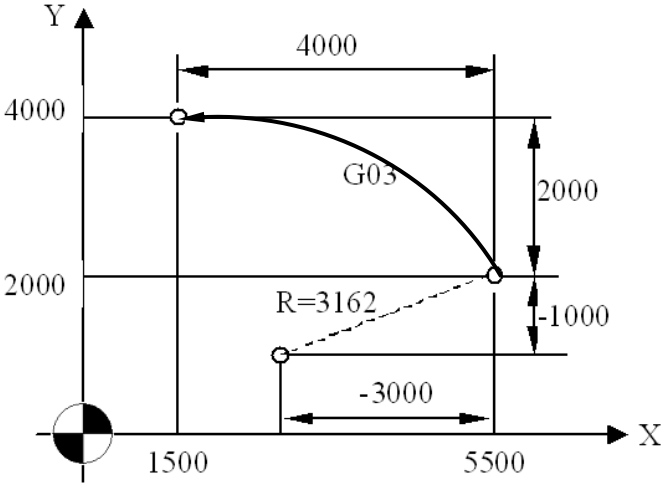
- When 180 degree < θ < 360 degree, R is negative.

$$\left\{\begin{matrix} G02 \\ G03 \end{matrix}\right\} \; X\_ \;\; Y\_ \;\; R-25.0;$$

- When θ=360 degree, only use **I、J、K**.

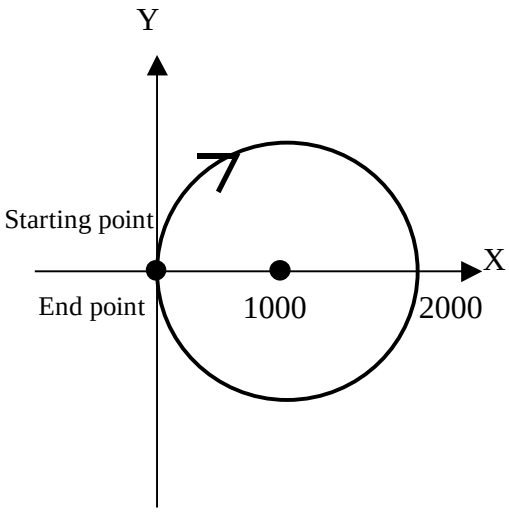


Example 1:



G90 G00 X5500 Y4000;//positioning to start point of arc
G17 G90 G03 X1500 Y4000 I-3000 J-1000 F200;
//absolute command
(G17 G91 G03 X-4000 Y2000 I-3000 J-1000 F200;
//increment command)

Example 2: (interpolate a full circle)



G90 G00 X0 Y0;
G02 I1000 F100; //interpolate a full circle

4.1.4 Non Modal Rapid Traverse

Command form:

G05 X_ Y_ Z_ ;

X, Y, Z: Specified point

Description:

It exhibits the same motion as G00, however, this code will only affect the line in which it exists.

Note:
In format3, G05 has a different function.

Example:

X250. G01 F2000.
G05 Z10.
X300. Y100. //The G01 is still in effect from above

4.1.5 Dwell

4.1.5.1 Format 1/2 – G04

Command form:

*G*04 *P* _ ;

P: specific time (ms, decimal point not permitted)

Note:

The use of G04 with P66000 forces an endless dwell or a program stop,

placing the machine in the waiting state. When in the "Waiting" state the spindle and coolant will remain on, as opposed to M00 and M01 which turn them off. To continue program execution press the "Cycle Start" button.

Description:

By specifying a dwell, the execution of the next block is delayed by the specified time. In addition, a dwell can be specified to make an exact check in the cutting mode.

Example:

G04 X2500;//delay 2.5 sec
G04 X2.5;//delay 2.5 sec
G04 P2500;//delay 2.5 sec
G04 P2.5;//delay **2** ms

4.1.5.2 Format 3 – G04

Command form:

$$G04 \left\{ \begin{array}{l} X\_ \\ P\_ \end{array} \right\} ;$$

X: specific time (s, decimal point permitted 0.001 ~ 9999.999s)

P: specific time (ms, decimal point not permitted)

Description:

By specifying a dwell, the execution of the next block is delayed by the specified time. In addition, a dwell can be specified to make an exact check in the cutting mode.

Example:

G04 X2500;//delay 2.5 sec
G04 X2.5;//delay 2.5 sec
G04 P2500;//delay 2.5 sec
G04 P2.5;//delay **2** ms

4.1.6 High Speed High Precision Interpolation

Command form:

$$G05 \quad P \left\{ \begin{array}{c} 10000 \\ 1 \\ 2 \\ 3 \\ 4 \\ 5 \end{array} \right\} ; \ // \text{Start HSHP interpolation}$$

G01 X__Y__Z__F__;
G02 X__Y__Z__R__;

G00 X__Y__Z__;

G05 P0; // Cancel HSHP interpolation

P: Multiple motion parameters
X, Y, Z: Specific coordinate point
F: Max feedrate (mm/min)

Note: Apply only in Format3

Note:

In Format 1/2, use G05.1 instead of G05 for HPCC function.

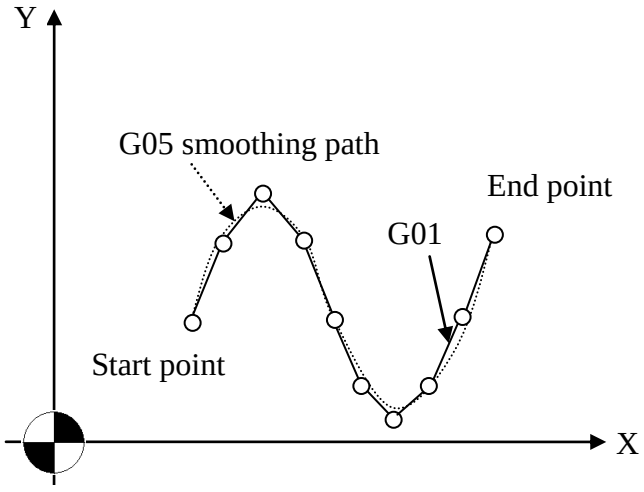
Description:

G05 provides one default parameter, P10000, and five other parameters, P1~P5, for users. Interpolation commands execute the mode of smoothing curve by processing program. G90/G91 decides absolute or increment mode. Feedrate is decided by **F** code for high speed & high precision interpolation.

Condition:

- On high speed & high precision interpolation (G05 P__) mode, M code and MPG simulation of negative direction are invalid.
- On high speed & high precision interpolation (G05 P__) mode, if cutter compensation(G40/G41/G42) and tool length compensation (G43/G44/G49) are used, the program can cancel G05 mode until G40/G41/G42 or G43/G44/G49 ending. It is not recommended to do that unless necessary.

Example:



G0 X3. Y4. Z0.
G05 P10000 //Start high speed & high precision interpolation
G01 X3.8 Y6.1 F5000.
X4.6 Y7.

```
X5.4 Y6.1
X6.1 Y4.
X6.9 Y1.9
X7.7 Y1.
X8.5 Y1.9
X9.3 Y4.
X10. Y6.1
G05 P0    // Cancel high speed & high precision interpolation
M30
```

4.1.7 Acceleration (No Feed Ramps)

Command form:

G8

Description:

In G8 mode, tool does not decelerate on the end of path, and continue to execute next path after to specified point.

4.1.8 Deceleration (Feed Ramps)

Command form:

G9

Description:

In G* mode, tool decelerates at the end of corner. When tool arrived at the terminal, a feedback signal is sent to ensure the position is in the setting range. The next path is executed after the feedback control.

4.1.9 In Position Check

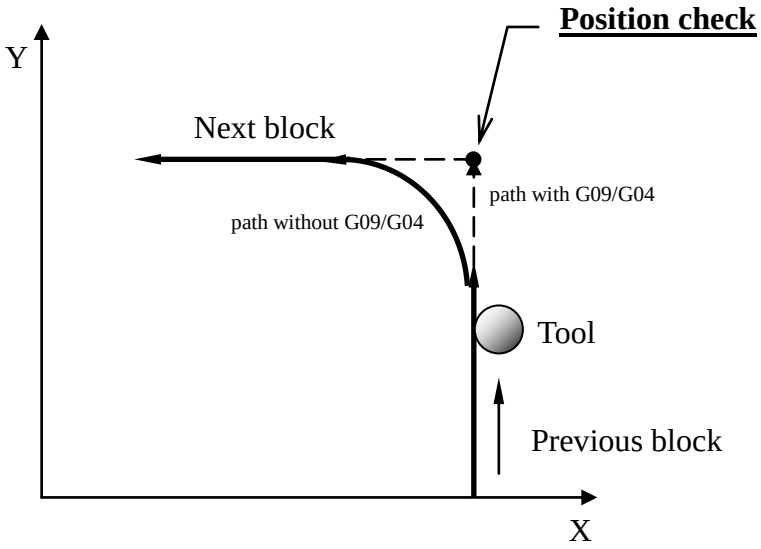
Command form:

G04

Description:

In some circumstances, high precision rectangular is need, G09 or G04 can be used to perform in position check. The tool will be slow down when approaching to the corner. G04 exact stop only effected in one block which has G04.

Example:



4.1.10 Programmable Data Input

Command form:

$$G10 \begin{Bmatrix} L10 \\ L11 \\ L12 \\ L13 \end{Bmatrix} P\_ \quad R\_ \quad ;$$

L10: for tool length(H) geometric compensation value
L11: for tool length(H) wear compensation value
L12: for tool diameter(D) geometric compensation value
L13: for tool diameter(D) wear compensation value

P: tool NO.
R: compensation value(data of tool length or tool diameter)

Description:

G10 command: it can directly use program command to enter tool compensation value.

In absolute mode (G90), value of G10 is the new compensation value; in increment mode (G91), value of G10 is the sum of the value of the moment with the new compensation value.

4.1.11 Plane Selection

Command form:

G17; X-Y plane selection

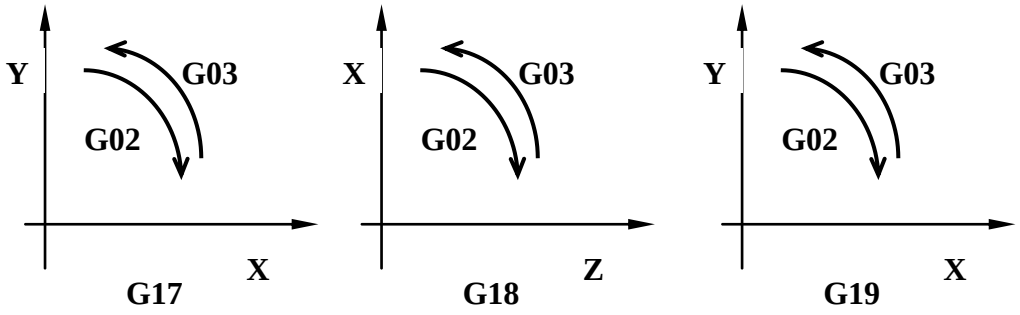
G18; Z-X plane selection

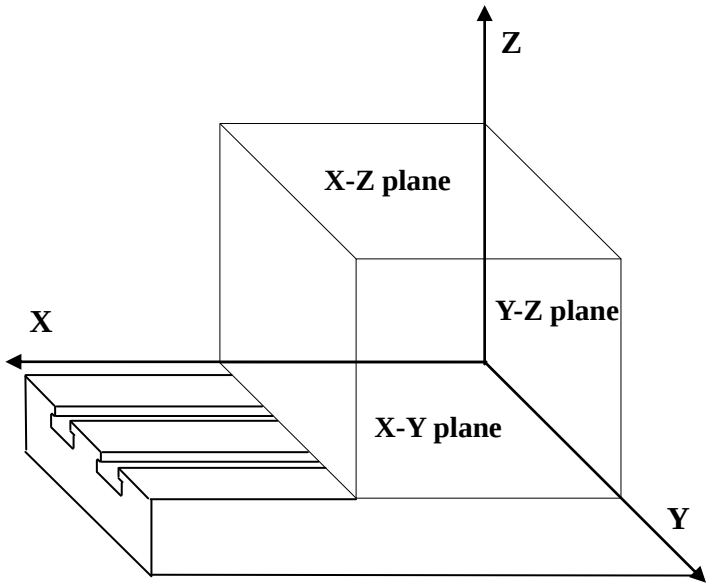
G19; Y-Z plane selection

Description:

When circular interpolation is in used, tool radius compensation or polar coordinate command, need to use G17, G18, or G19 to define the cutting plane or working plane (default is G17).

Example:





4.1.12 Flat Cam (Cam Wrapping) Programming

Command form:

G17 Q _ (P1);

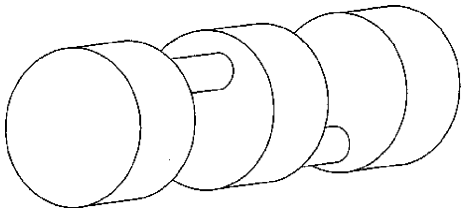
Q: counts constant = A axis ratio / (5 * pi * (cam diameter in inches)) 5

Note: The axis ratio should be 90.

P: Use G17 Q word P1 on the same line in the program for YB wrapping. All information for Y to A wrapping applies to X to B wrapping.

Description:

Flat cam programming is used when an XY program needs to be "wrapped" around the circumference of the part. This function is designed to convert Y axis motion into A axis motion. XA conversion is used when the A axis is the rotary table, YB conversion is used when the B axis is the rotary table.



The conversion from Y to A axis moves is defined in the program by using a G17 and Q word in the same line. The Q word represents a number used by the control for converting the Y or X axis moves to A or B axis moves.

Directly after the unwrapping move, cancel the flat cam programming function by coding a G17 on a line by itself.

Example:

```
N2 M6 T1
N3 (TOOL #1, 1/2 2 FL E.M. USE .5 IN THE TOOL TABLE
N4 G0 G90 S2000 M3 X0 Y0 A0
N4.5 G51 .1 Y0 It is important to mirror the Y axis
N5 H1 Dl M8 Z.1
N6 G17 Q1.4324 //This line starts Flat cam conversion (see Q word)
N7 X1.125 Y-2.125
N8 G1 Z-.27 F25.
N9 G1 G42 X2.125 //To maintain a climb cut on a mirrored path, use G42
N10 Y-.25
N11 X1.875 Y0 I-.25 G3
N12 X.25
N13 X0 Y-.25 J-.25 G3
N14 Y-4.5
N15 X.322 Y-5.2437 I1.02 G3
N16 X.625 Y-5.9437 I-.657 J-.7 G2
N17 Y-6.5
N18 X2.125 I.75 G3
N19 Y-3.375
N20 X1.125 G40
N21 Z.1 G0
N22 Y0 //Return to original Y position (unwrapping move)
N22.5 G50.1 //Turn off mirror       FULL RM.)
N23 G17 //This line cancels the Flat cam conversion
N24 M5 M9
N25 G90 G0 H0 Z0
```

4.1.13 Imperial/Metric Mode Verification

Command form:

G20/G70 : Imperial mode verification
G21/G71 : Metric mode verification

Note : Apply only in Format1/2

Description:

When G20/G70 is encountered, if the system is not in imperial mode alarm will occur and then control will stop at that block.

When G21/G71 is encountered, if the system is not in metric mode alarm

will occur and then control will stop at that block.

4.1.14 Imperial/Metric Mode Define

Command form:

G70 : Imperial mode define
G71 : Metric mode define

Note : Apply only in Format3

Description:

When G70 is encounter, the system will be defined as imperial mode. When G71 is encounter, the system will be defined as metric mode.

4.1.15 Return to Home Position

Command form:

G28 X__Y__Z__ ;

X, Y, Z: mid-point position (absolute value in G90 mode, increment value in G91 mode)

Description:

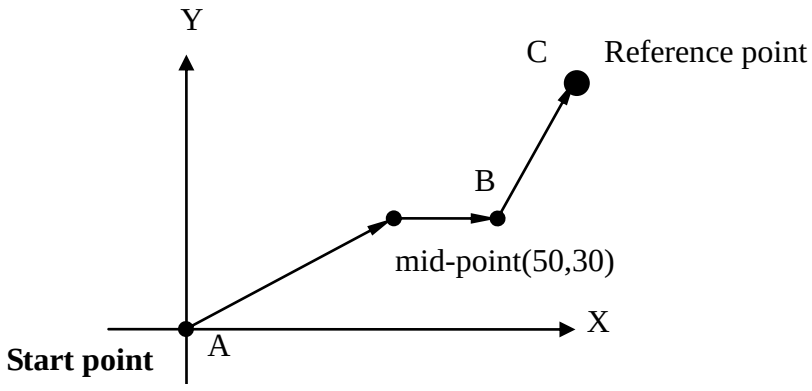
It will move to the specified safety mid-point first and then return to origin point or reference point.

Note:

This command usually use in auto tool exchange. For safety, before doing G28, it is recommended to cancel tool compensation

Example 1:

G90 **G28** X50.0 Y30.0; //A→B→C, mid-point(50,30)



Example 2:

G28 X0; //only X axis return to reference point
G28 Y0; //only Y axis return to reference point
G28 Z0; //only Z axis return to reference point

4.1.16 Cancel Jog Away

Command form:

G28.1

Description:

It will cancel the jog away amount.

Example:

Programmed Position	Machine Position	Jog Away offset
X300.0 Y300.0	X300.100 Y300.100	X0.100 Y0.100
G28.1 X0	X300.000 Y300.100	X0.000 Y0.100

4.1.17 Return from Home Position

Command form:

G29 X__Y__Z__;

X, Y, Z: specified coordinate；

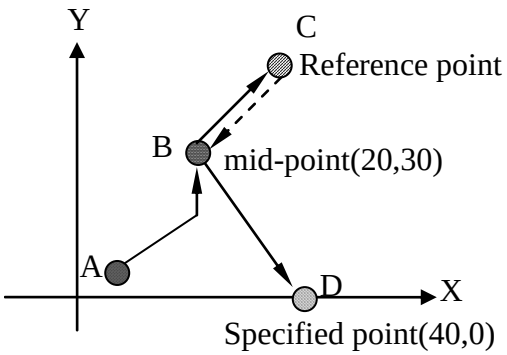
(absolute value in G90 mode, increment value in G91 mode)

Description:

G29 will move machine to specified point through mid-point to which set in G28. G29 cannot be used alone, because G29 does not specify mid-point, G29 use the mid-point from G28, therefore, G29 must be called after G28.

In G90 mode, the specified point is the absolute coordinate; In G91 mode, the specified point is the incremental distance from mid-point.

Example:



7. Absolute command:

N001 G90 G28 X20.0 Y30.0;

//A→B→C, mid-point(20,40), in absolute command mode

N002 M06;//change the tool

N003 **G29** X40.0 Y0.0;

// C→B→D, the specified point is absolute coordinate

8. Increment command:

N001 G91 G28 X20.0 Y40.0;
//A→B→C, mid-point(20,40), in increment command mode
N002 M06;//change the tool
N003 **G29** X40.0 Y-40.0;
//C→B→D, the specified position is the increment value from mid-point to specified point

4.1.18 Cutter Radius Compensation

Command form:

$$\left\{ \begin{array}{l} G41 \\ G42 \end{array} \right\} \; X\_ \;\; Y\_ \;\; Z\_;$$

G40；

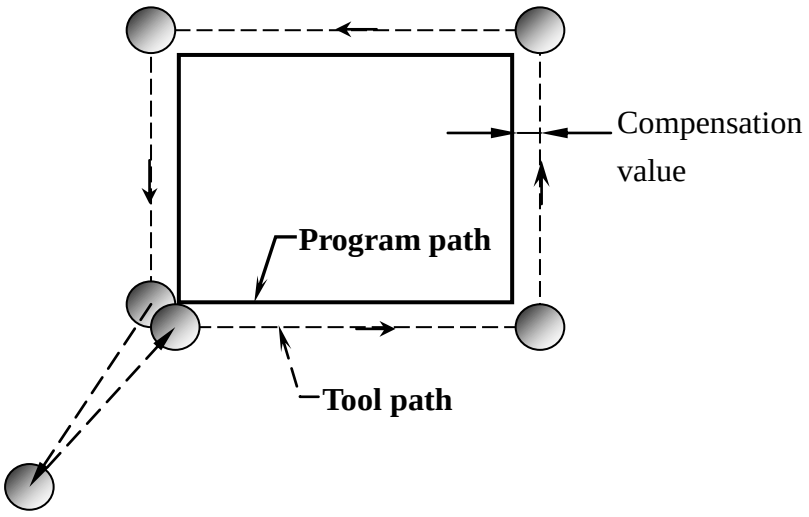
G41: cutter compensation left.
G42: cutter compensation right.
G40: cutter compensation cancel.
X, Y: the end coordinate of each axis.
D: code for specifying as the cutter compensation value.

Description:

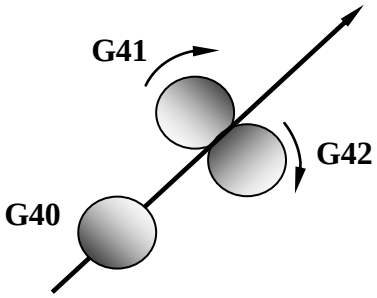
Generally, there will be overcut if the tool path is moved along with the programmed path. Therefore, a cutter radius compensation is required to shift the tool path according to the tool radius.

Example:

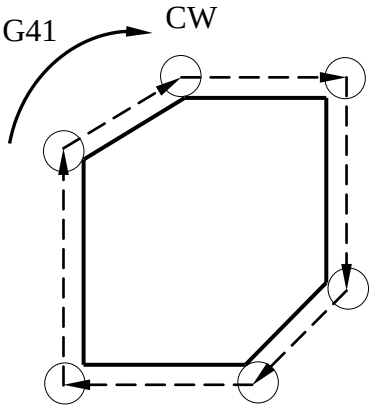
9. Cutter compensation:



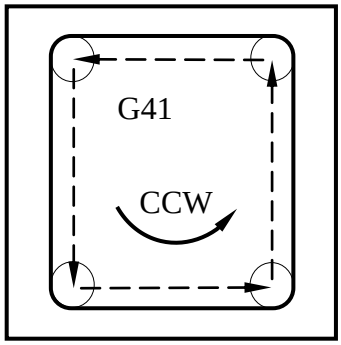
10. Direction decision of cutter compensation:



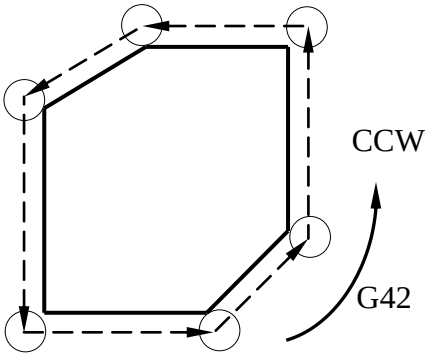
Compensation value	Positive	Negative
G41	Compensation left	Compensation right
G42	Compensation right	Compensation left



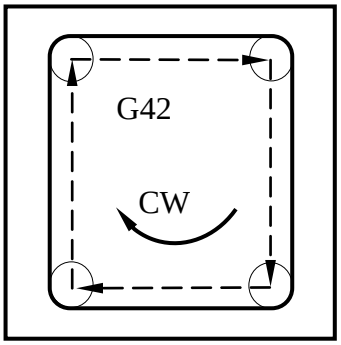
a. G41-outline cut (CW)



b. G41-inline cut (CCW)



c. G42-outline cut (CCW)

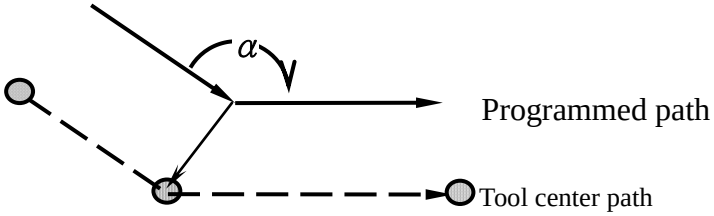


d. G42-inline cut (CW)

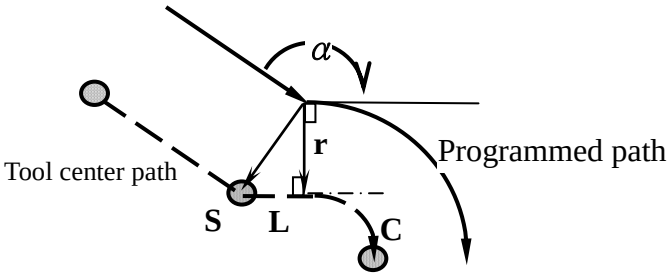
11. cutter compensation of corner interpolartion:

■ When the corner: 90∘≦α < 180∘

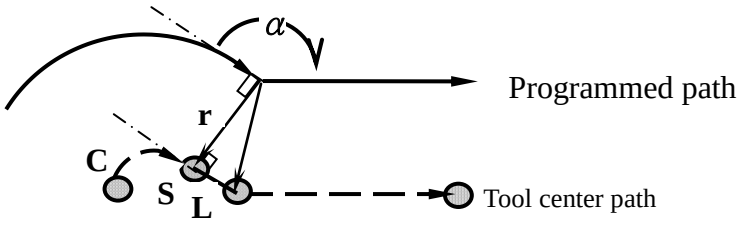
12. straight line → straight line



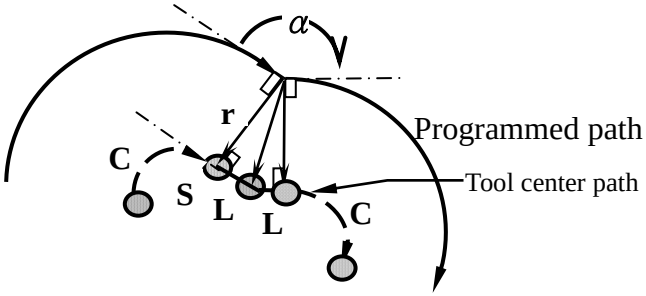
13. straight line → arc



14. arc → straight line

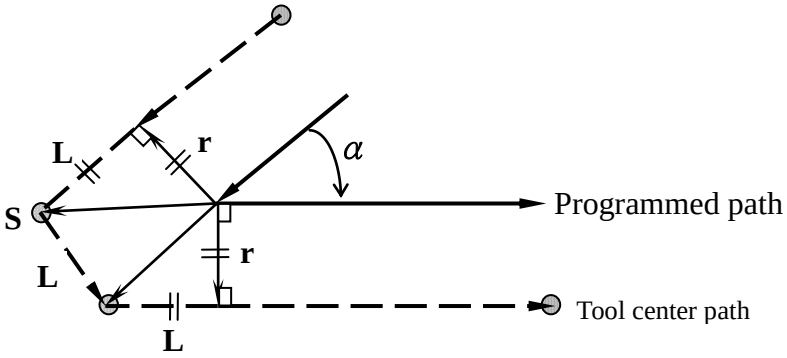


15. arc → arc

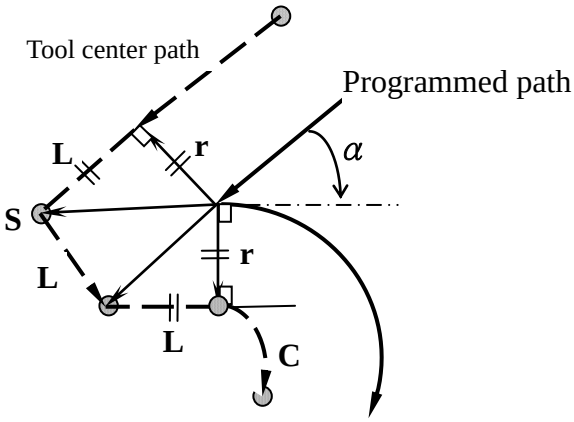


■ When corner α < 90∘

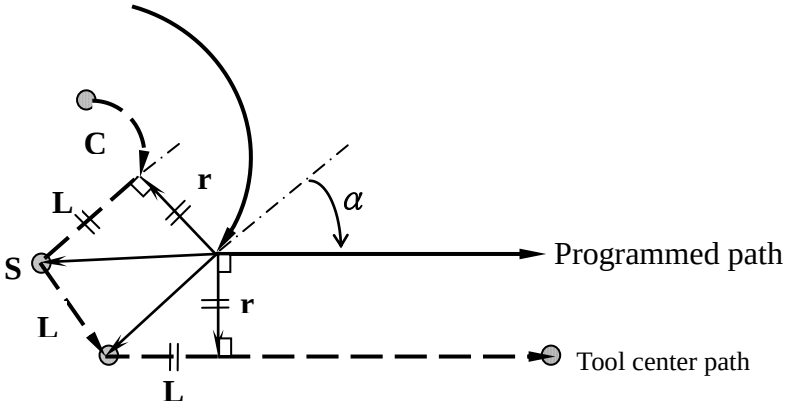
16. straight line → strainght line



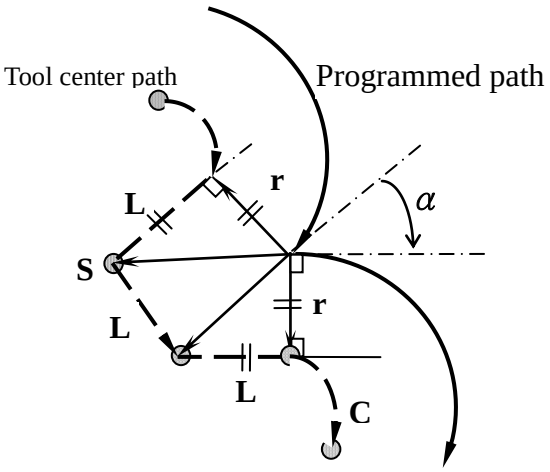
17. straight line → arc



18. arc → straight line

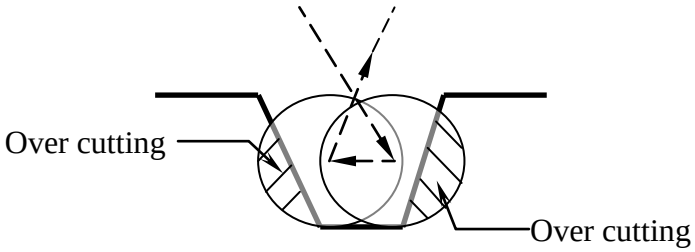


19. arc → arc



Notes:

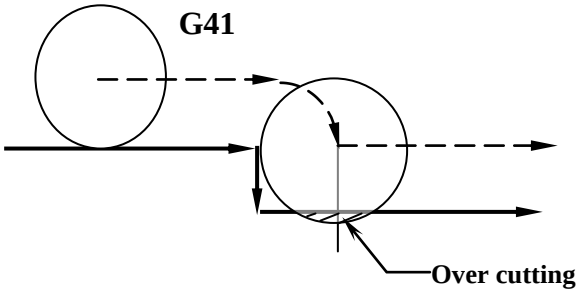
When process a fillister, if the width less than twice of tool, the control will send an alarm to avoid overcutting.



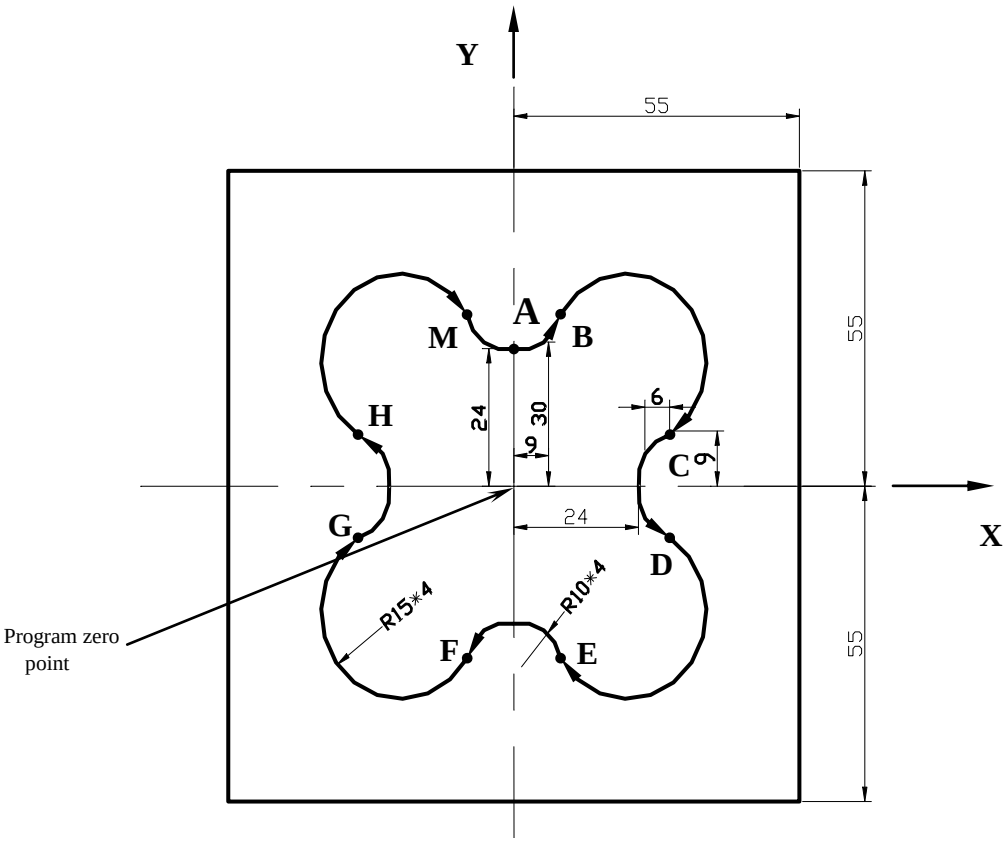
Cutter radius compensation function is not supported in MDI mode.

G41/G42 and G40 cannot be used with G02/G03 at the same block, only can be used with G00/G01 at the same block.

When process a step shape workpiece, if the step higher than workpiece radius, the control will send an alarm to avoid overcutting.



Example:



N001 T1 S1000 M03；//tool NO.1(diameter 10mm), spindle 1000rpm (CW)

N002 G00 X0.0 Y0.0 Z10.0；//positioning above programmed zero point

N003 M08；//open cutting liquid

N004 G90 G01 Z-10.0 F600；//linear interpolation to bottom of workpiece, feedrate 600mm/min

N005 G42 Y24.0 D01；//cutter compensation left, program zero point→**A**

N006 G03 X9.0 Y30.0 R10.0；//**A→B** circular interpolation (CCW)

N007 G02 X30.0 Y9.0 R15.0；//**B→C** circular interpolation (CW)

N008 G03 X30.0 Y-9.0 R10.0；//**C→D** circular interpolation (CCW)

N009 G02 X9.0 Y-30.0 R15.0；//**D→E** circular interpolation (CW)

N010 G03 X-9.0 Y-30.0 R10.0；//**E→F** circular interpolation (CCW)

N011 G02 X-30.0 Y-9.0 R15.0；//**F→G** circular interpolation (CW)

N012 G03 X-30.0 Y9.0 R10.0；//**G→H** circular interpolation (CCW)

N013 G02 X-9.0 Y30.0 R15.0；//**H→M** circular interpolation (CW)

N014 G03 X0.0 Y24.0 R10.0；//**M→A** circular interpolation (CCW)

N015 G00 Z10.0；//Z axis rise, return to start point

N016 G40 X0.0 Y0.0；//cutter interpolation cancel, return to start point

N017 M09；//cutting liquid OFF

N018 M05；//spindle stop

N019 M30；//program end

4.1.19 Tool Length Compensation

Command form:

$$\left\{ \begin{array}{l} G43 \\ G44 \end{array} \right\} Z\_ \quad H\_;$$

G49；

G43: compensation along positive direction；

G44: compensation along negative direction；

G49: compensation cancel；

Z: Z axis end coordinates；

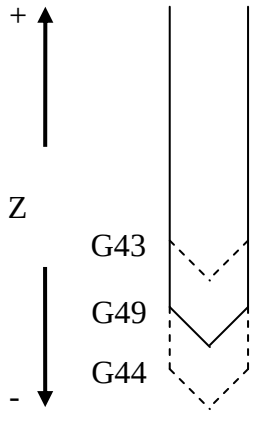
H: tool number；

Description:

Tool length compensation (G43/G44) is used to compensate the Z axis position to confirm the program zero position is on the surface of the part.

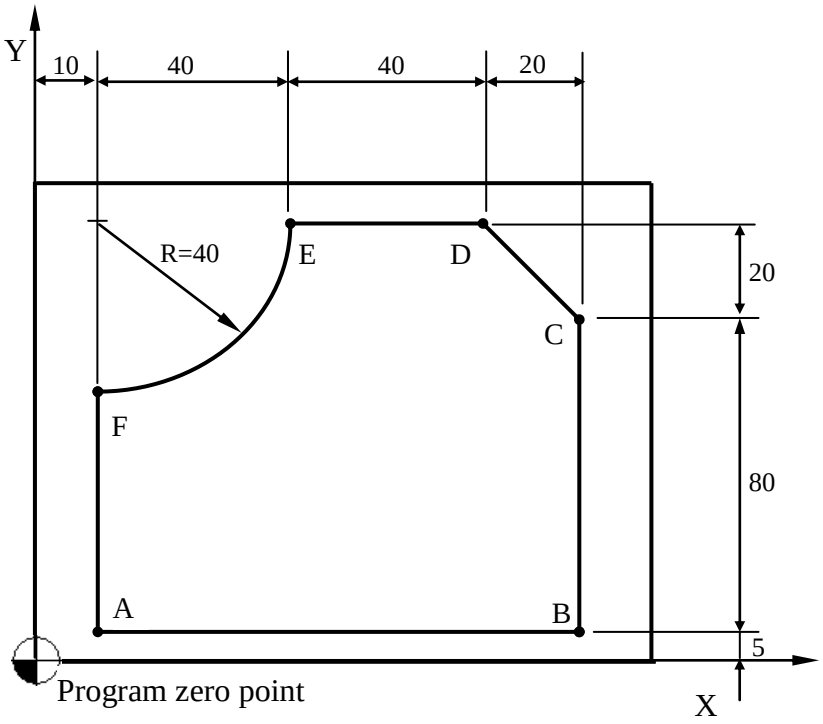
Note:

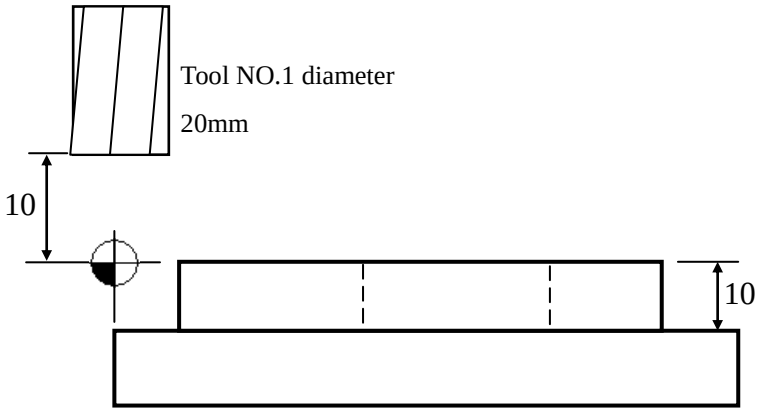
In Format 1, H_ = G43 H_ and H0 = G49.



Compensation value	Positive	Negative
G43	Positive direction	Negative direction
G44	Negative direction	Positive direction

Example:





T1 S1000 M03 ; //use tool NO.1(diameter 20mm), spindle 1000rpm(CW)

G42 D01；//tool radius compensation right(D01=10)

G00 X10.0 Y5.0 Z15.0；//positioning above A point

G43 H01；//tool length compensation positive(H01=-10)

G01 Z-10.0；//linear interpolation to bottom of A point

X110.0；//A→B

Y85.0；//B→C

X90.0 Y105.0；//C→D

X50.0；//D→E

G02 X10.0 Y65.0 R40.0；//E→F

G01 Y5.0；//F→A

G00 Z15.0；//positioning return above A point

G40 G49；//compensation cancel

M05；//spindle stop

M30；//program end

4.1.20 Tool Length Offset Single Expansion/Reduction

Command form:

G45 X__Y__Z__ H_；(Single Expansion)

G46 X__Y__Z__ H_；(Single Reduction)

X, Y, Z: end point position (absolute value in G90 mode, increment value in G91 mode)
H_: offset value of tool#

Description:

This code is used for extending the programmed axis move by a value stored in the tool offset table. The value is determined by an H word. Program the H word in the same block with the G45/G46 code and an axis move. Only the block containing the G45/G46 code is extended.

Note:

The G45-G48 codes may only be used in X only, Y only, or quarter arc moves. No angular movements of full circles are allowed.

Example:

G91 G0
G45 X100. H1

The above example extends the 100. axis move by the tool length value of offset #1. **Tool length offset is not applied to the Z axis.**

4.1.21 Tool Length Offset Double Expansion/Reduction

Command form:

G47 X__Y__Z__ H_；(Double Expansion)

G48 X__Y__Z__ H_；(Double Reduction)

X, Y, Z: end point position (absolute value in G90 mode, increment value in G91 mode)

H_: offset value of tool#

Description:

This code is extending/reducing the program axis move by a value stored in the tool table. It is similar in function to a G45/G46, but the value determined by the H word is double.

4.1.22 Mirror Image Enable/Cancel

Command form:

G51.1 X___Y___Z___;
G50.1;//programmable mirror image cancel

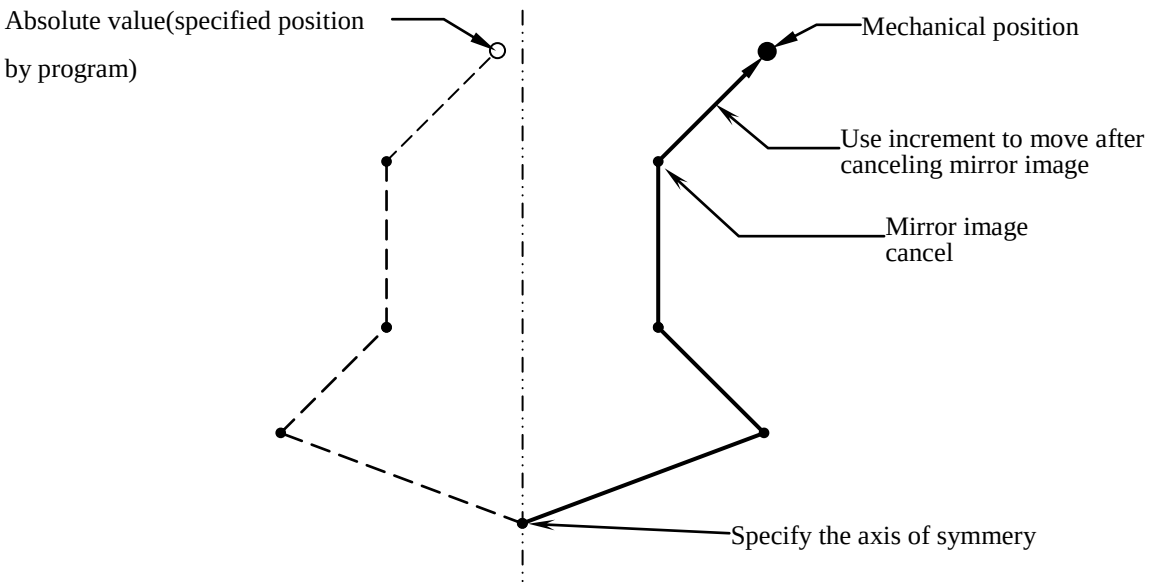
X, Y, Z: mirror point (axis) coordinate value.

Description:

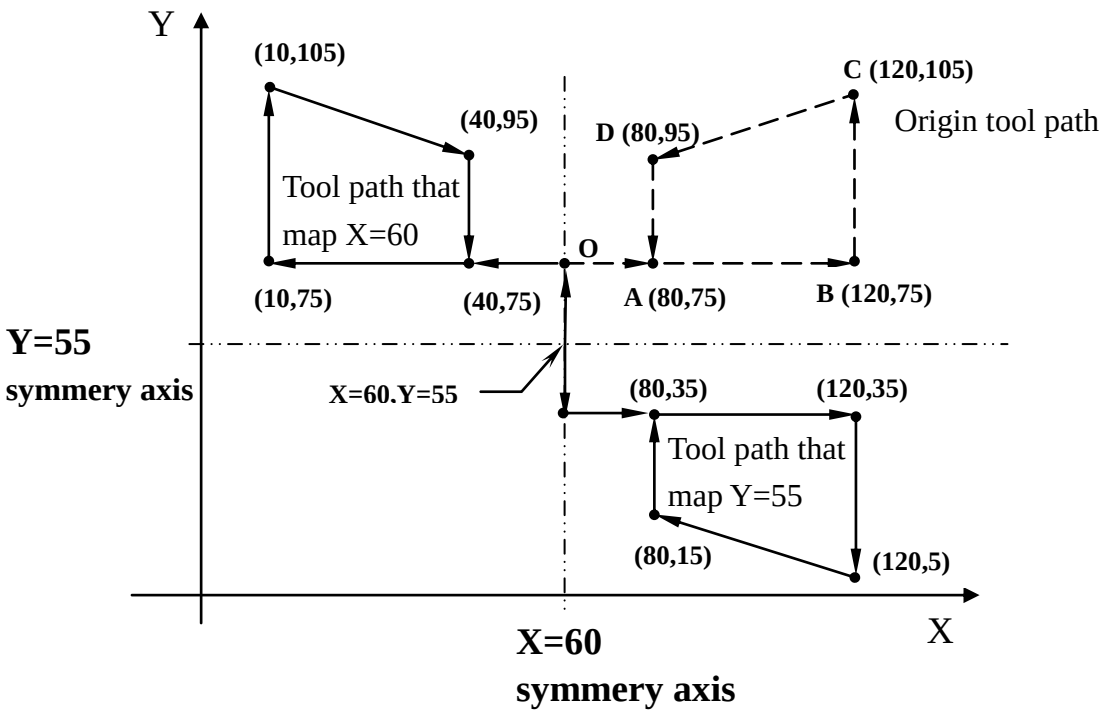
In symmetry shape cutting circumstances, mirror image modes help the user to simplify the NC programming. G51.1 activates the mirror image mode. G50.1 deactivates the mirror image mode. The axes to be mirrored are identified in the same block with the G51.1 code.

Note:

This function is identified in program coordinates, once encounter reset or work offset changes, the center of mirror image is changed.



Example 1:



N001 T1 S1000 M03；//use tool NO. 1, 1000rpm(CW)

N002 M98 H100；//execute sub-program

N003 G51.1 X60.0；//execute programmable mirror image that symmery axis X=60

N004 M98 H100；// execute sub-program

N005 G50.1；//programmable mirror image cancel

N006 G51.1 Y55.0；//execute programmable mirror image that symmery axis Y=55

N007 M98 H100；// execute sub-program

N008 G50.1；// programmable mirror image cancel

N009 M05；//spindle stops

N010 M30；//program ends

N100；//sub-program list

G00 X60.0 Y55.0；//positioning to specified point

G01 Y75.0；//linear interpolation to O point

X80.0；//O→A

X120.0；//A→B

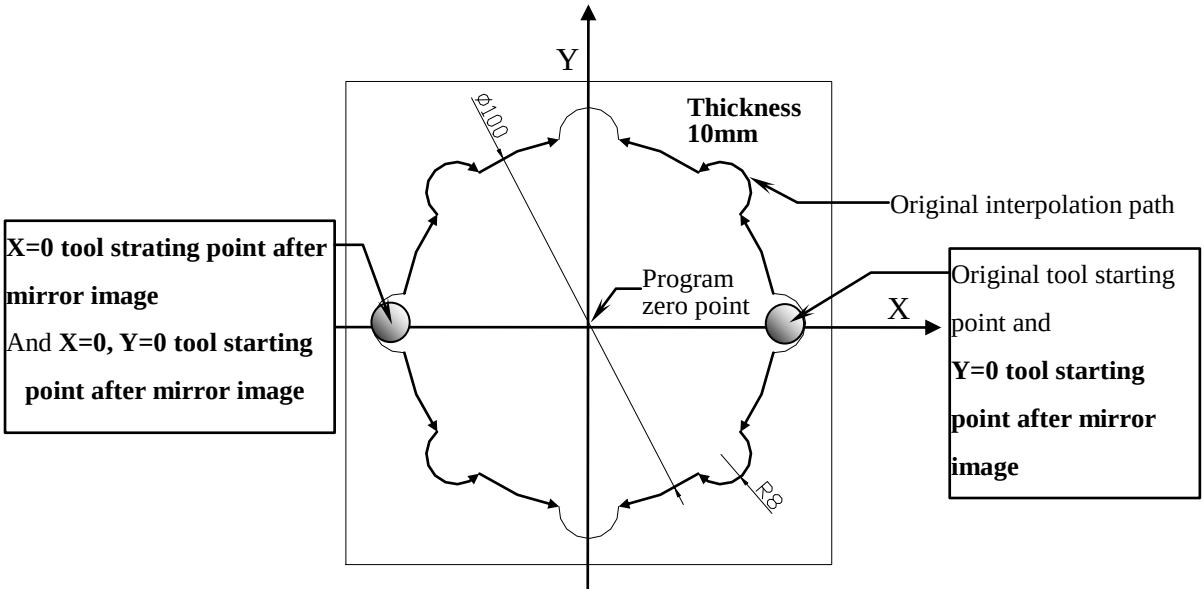
Y105.0；//B→C

X80.0 Y95.0；//C→D

Y75.0；//D→A

M99；//sub-program ends

Example 2: processing example



N001 T1 S1000 M03；//tool No.1(diameter 10mm), 1000rpm(CW)

N002 G41 D01；//set cutter compensation left of tool No.1(D01 = 5)

N003 M98 H100；//execute sub-program

N004 G51.1 X0.0；//execute programmable mirror image at symmery axis X=0

N005 M98 H100；//execute sub-program

N006 G50.1；//programmable mirror image cancel

N007 G51.1 X0.0 Y0.0；// execute programmable mirror image at symmery point X=0, Y=0

N008 M98 H100；// execute sub-program

N009 G50.1；// programmable mirror image cancel

N010 G51.1 Y0.0；// execute programmable mirror image at symmery axis Y=0

N011 M98 H100；// execute sub-program

N012 G50.1；// programmable mirror image cancel

N013 G40；//cutter compensation cancel

N014 M05；//spindle stops

N015 M30；//program ends

Sub-program

N100；sub-program list

G00 X58.0 Y0.0 Z10.0；//positioning to the above of starting position

G01 Z-10.0；//linear interpolation to bottom of workpiece

G03 X49.36 Y7.9744 R8.0；//circular interpolation(CCW), radius 8mm

G03 X40.5415 Y29.2641 R50.0；// circular interpolation(CCW), radius 50mm

G03 X29.2641 Y40.5415 R8.0；// circular interpolation(CCW), radius 8mm

G03 X7.9744 Y49.36 R50.0；// circular interpolation(CCW), radius 50mm

G03 X0.0 Y58.0 R8.0；// circular interpolation(CCW), radius 50mm

G00 Z10.0；//positioning to above of end point

M99 ；//sub-program end, continue to execute main program

4.1.23 Axis Scaling Enable/Disable

4.1.23.1 Format 1/2 G51.3

Command form:

G51.3 R1_ R2_ R3_ R4_;

R1: scale all axes
R2: scale only the X axis
R3: scale only the Y axis
R4: scale only the Z axis

Description:

This code allows the programmer to scale all or individual axis dimensions. The number with R1~R4 represents a percentage to scale. The percentage is represented in the decimal form. For example 2.0 would double the size, .5 would half the size.

Note:

Circular moves will be scaled according to the axis being scaled. If the X axis is scaled, the I for the circle center description will be scaled in the same proportion. The same would apply for the Y and Z axis. When the circles are to be scaled, it is suggested that all axes of the plane selection be scaled proportionally. For example, in G18 the X and Z axes should be scaled at the same percentage.

Example:

N001 O1;
** Cut part **
N002 M6T1; //tool #1
N003 G0 G90 S2500 M3 E1 X0Y0;
N004 H1 D1 Z.1;
N005 G51.3 R1+2.; //scale all axes by 2 times

** Cut part **
N006 G51.3 R1+1.; //Cancel scaling of all axes
N007 G0 G90 H0 Z0;

4.1.23.2 Format 3 G50/G51

Command form:

$$G50 \quad X\_ \; Y\_ \; Z\_ \begin{cases} I\_J\_K\_ \\ P\_ \end{cases}$$

G51 : cancel scaling

X, Y, Z: center coordinate value of scaling；

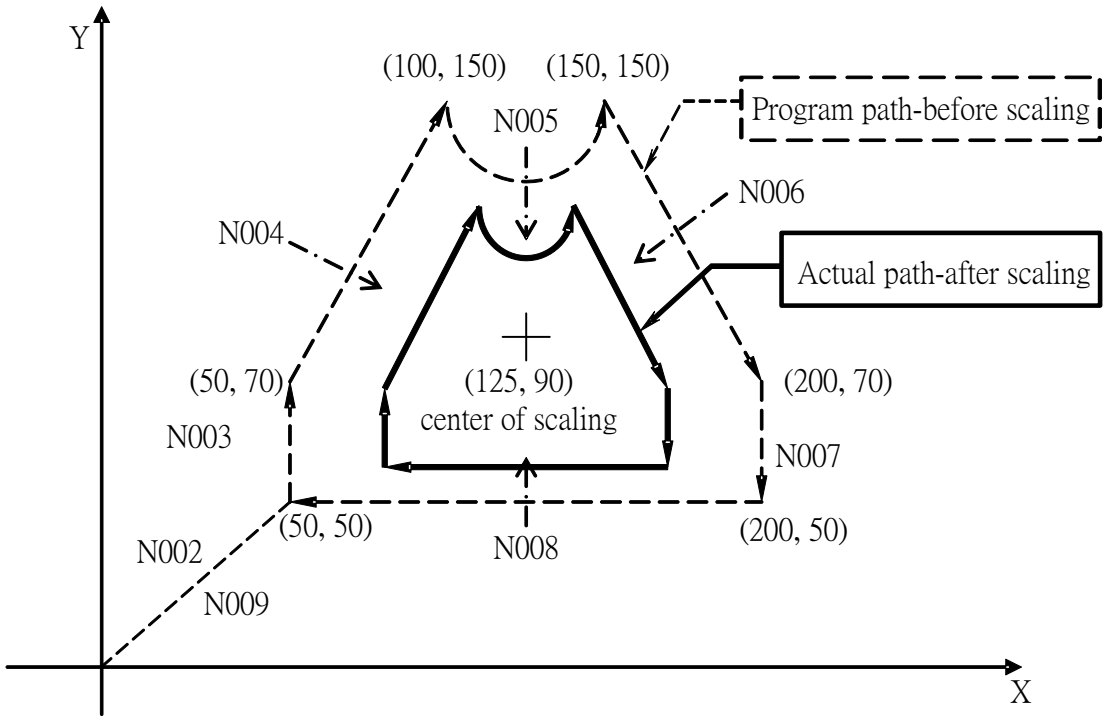
I, J, K: scaling factor for X axis Y axis and Z axis respectively；

P: scaling factor for X axis Y axis and Z axis are the same magnification；

Description:

This code allows the programmer to scale all or individual axis dimensions. The number with I, J, K represents a percentage to scale. The percentage is represented in the decimal form. For example 2.0 would double the size, .5 would half the size.

Example:



N001 G00 X50.0 Y50.0；//positioning

N002 G51 X125.0 Y90.0 P0.5；//define center of scaling X125,Y90 scaling factor 0.5, enable scaling in steps N003~N009

N003 G01 Y70.0 F1000；//linear interpolation, feedrate 1000mm/min

N004 X100.0 Y150.0；

N005 G03 X150.0 I25.0；//circular interpolation, radius 25mm；

N006 G01 X200.0 Y70.0；// linear interpolation

N007 Y50.0；

N008 X50.0；

N009 G00 X0.0 Y0.0；//return

N010 G50；//scaling cancel

N011 M30；//program end

4.1.24 Coordinate System Shift

Command form:

G52 X_ Y_ Z_ ;

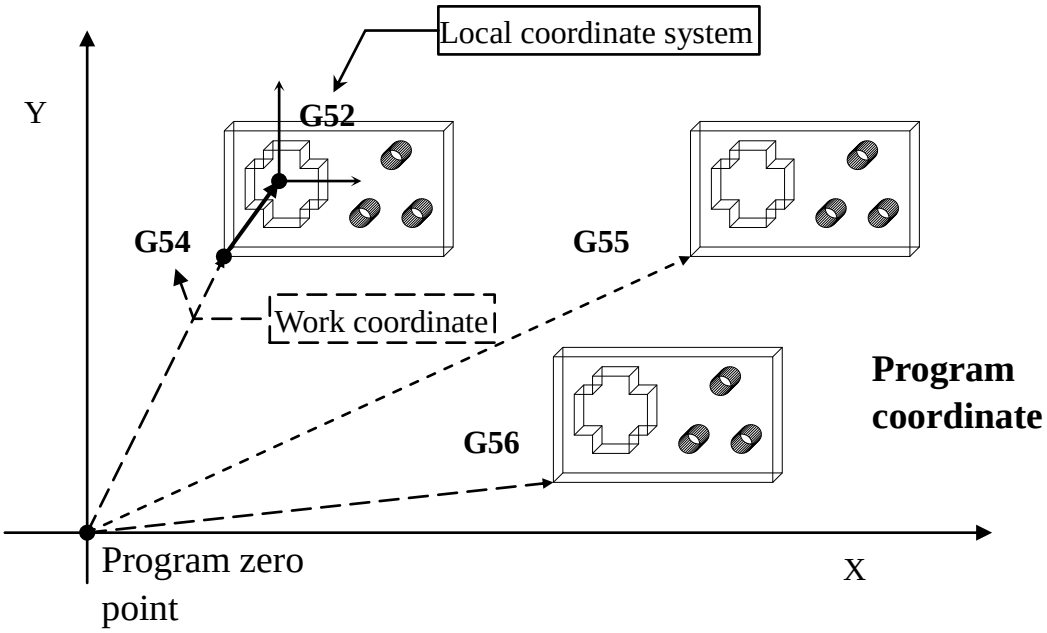
G52 X0.0 Y0.0 Z0.0: cancel the coordinate system

X, Y, Z: coordinate values

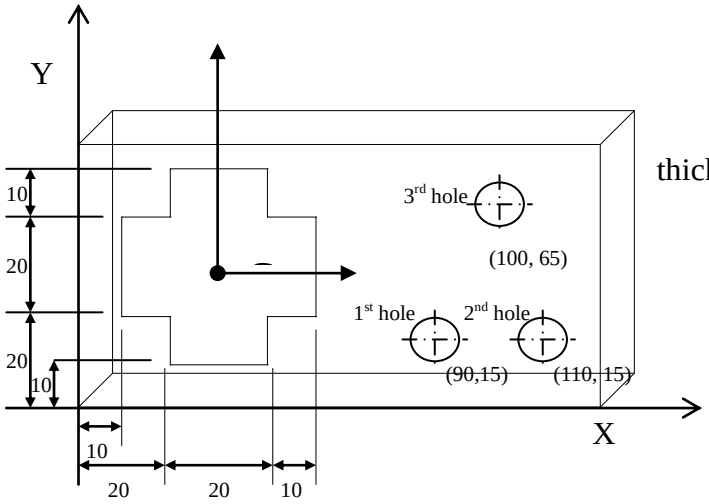
Description:

This code is used to define a local coordinate offset upon the work offset (G54~G59). This code is used when an absolute subroutine or subprogram needs to be used at different locations. Whereas an incremental subroutine or subprogram can be repeated from any location.

Coordinate system:


Local coordinate system
Y
G52
G54
G55
Work coordinate
Program
coordinate
G56
Program zero
point
X

Example:



thickness10mm

N001 T1 S1000 M03；//tool #1(diameter 10mm), spindle 1000rpm

N002 G54 X0.0 Y0.0 Z0.0；//define work offset (G54 or E1)

N003 G00 X90.0 Y15.0 Z10.0；//positioning to above of specified position

N004 G43 H01；//tool length compensation (tool #1)

N005 G99 G81 Z-15.0 R2.0 F1000；//execute drilling cycle, stop at R point when return, feedrate 1000mm/min, drill 1st hole

N006 X110.0；//drill 2nd hole

N007 X100.0 Y65.0；//drill 3rd hole

N008 G80；//cancel cycle

N009 M05；//spindle stops

N010 G28 X0.0 Y0.0 Z10.0 ； //reference point return, X0.0,Y0.0,Z10.0 to be center point

N011 T2 M06 S1000 M03；//tool change (tool #2 diameter 10mm)

N012 G52 X30.0 Y30.0 Z0.0；//define local coordinate zero point to the work offset (G54) of X40.0, Y40.0, Z0.0

N013 G00 X0.0 Y0.0 Z10.0；//position to local coordinate

N014 G01 Z-12.0；//linear interpolation to bottom of the hole

N015 G17 G41 D02；//cutter compensation left (tool No.2)

N016 G91 X20.0；//incremental mode

N017 Y10.0；

N018 X-10.0；

N019 Y10.0；

N020 X-20.0；

N021 Y-10.0；

N022 X-10.0；

N023 Y-20.0；

N024 X10.0；

N025 Y-10.0；

N026 X20.0；

N027 Y10.0；

N028 X10.0；

N029 Y10.0；

N030 G90 G00 Z10.0；//absolute mode

N031 G52 X0.0 Y0.0 Z0.0；//cancel local coordinate

N032 G40 M05；//cancel compensation, spindle stops

N033 M30；//program ends

4.1.25 Machine Coordinate System

Command form:

G53 X_ Y_ Z_ ;

X: move to specified machine coordinate of X position.
Y: move to specified machine coordinate of Y position.
Z: move to specified machine coordinate of Z position.

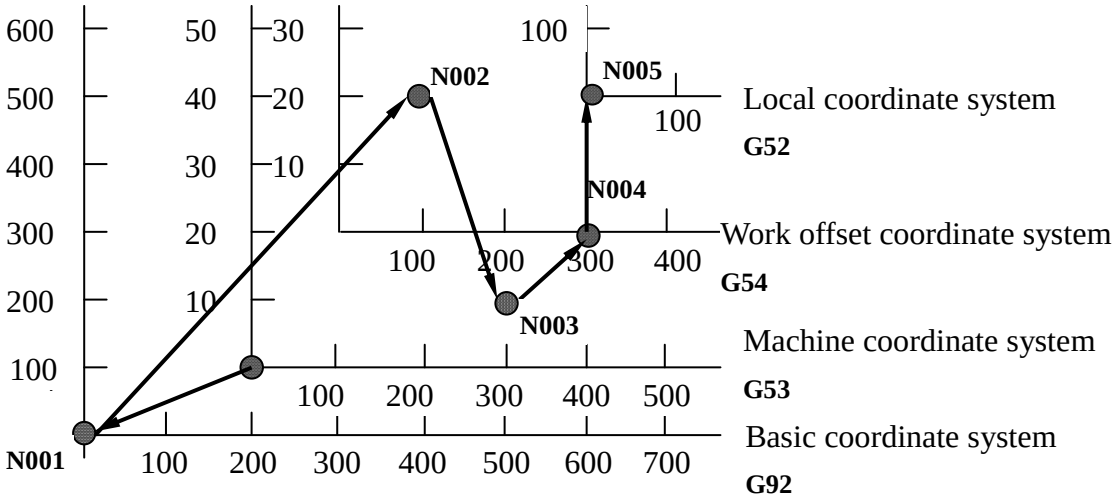
Description:

This code causes the control to use the machine coordinate system, this coordinate system is fixed according to the machine zero.

Notes:

20. G53 only effective in the block in which it exists;
21. G53 only effects in absolute mode (G90), not effects in incremental mode (G91);
22. Before using G53, one must cancel related cutter radius compensation ,tool length compensation or position compensation；

Example:



N001 G92 X-200.0 Y-100.0；//specify to basic coordinate system

N002 G54 G90 X100.0 Y200.0；//define G54 work offset

N003 G53 X300.0 Y100.0 ； //move to specified machine coordinate point

N004 X300.0 Y0；

//as G53 only effective in one block, this block will continue using G54 coordinate system

N005 G52 X300.0 Y200.0；//set local coordinate

N006 X0.0 Y0.0；

4.1.26 Apply Work Offset

Command form:

$$\left\{\begin{array}{l} G54 \\ G55 \\ G56 \\ G57 \\ G58 \\ G59 \\ G59.1 \\ G59.2 \\ \quad : \\ \quad : \\ \quad : \\ G59.9 \end{array}\right. \quad X\_Y\_Z\_;$$

G54: 1st work offset

:

:

G59: 6th work offset
G59.1: 7th work offset
:
:
G59.9: 15th work offset

X, Y, Z: move to specified position on defined work offset；

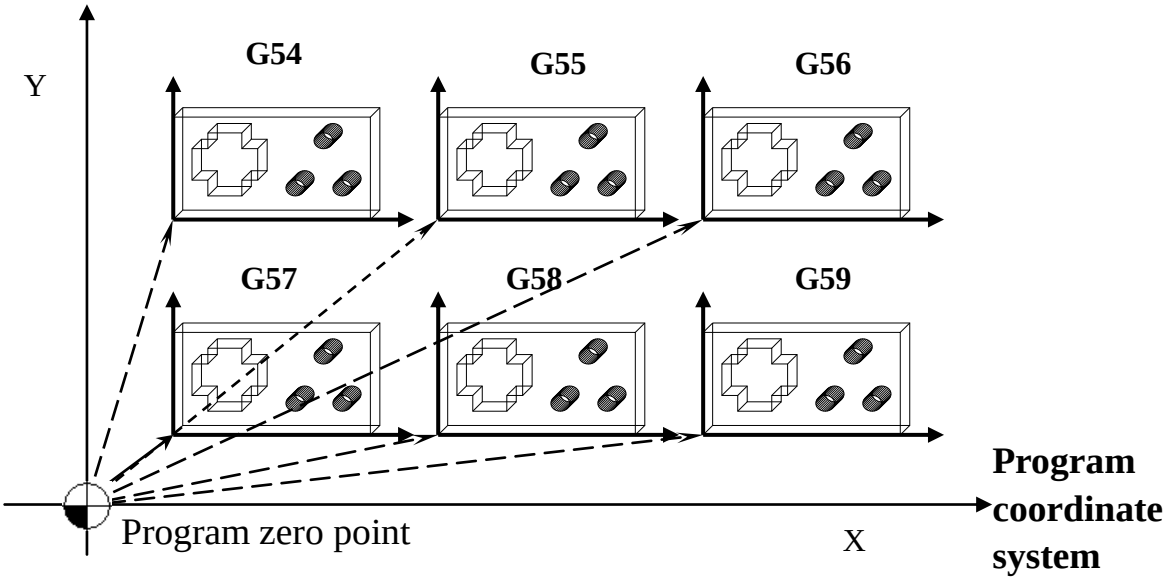
Description:

In general when operating numerical machine, if there are many work offset required, one can use work offset system G54 to G59 six G codes G59.1~G59.9 to present 15 different coordinate systems.

Note:

In Format 1/2, E_ = G54 P_, e.g. E1 represents G54 or G54 P1.

Example:



4.1.27 Modal Subroutine Call/Cancel

Command form:

G66 P_ L_ ; modal subroutine call

G67 ; modal subroutine cancel

P: number of the subroutine to call ;

L: repetition count ;

Description:

After G66 is called, P_ is called to execute and L__ indicates repeating times. If there is a moving block, G66 block will be executed after moving block ends, until G67 to cancel it.

Example:

G91
G66 P10 L2 X10.0 Y10.0 //repeat twice calling sub-program O0010 and set X=10.0 and Y=10.0 into sub-program.
X20.0 //Move to position X=20.0. After moving, call G66 P10 L2 X10.0 Y10.0.
Y20.0 //Move to position Y=20.0. After moving, call G66 P10 L2 X10.0 Y10.0.
G67 //Cancel macro call mode.

4.1.28 Coordinate System Rotation/Cancel

Command form:

(G17) G68 X_ Y_ R_; // start coordinate rotation
(G18) G68 Z_ X_ R_;
(G19) G68 Y_ Z_ R_;

G69; // Disable coordinate rotation

X_, Y_, Z_: absolute coordinate of center of rotation
R_: angle of rotation

Note:

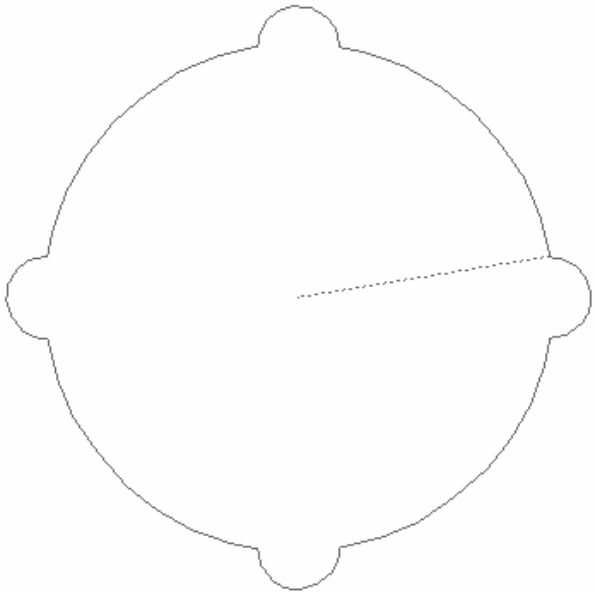
In Format 1/2, R0_ represents R_, e.g. R0+30. represents R30.

Description:

When coordinate rotation enable, all movement command will rotate according to the specified rotation center. Rotation center will only be effective in absolute command, if all command is incremental, the actual rotation center is the starting point of path.

Example 1:

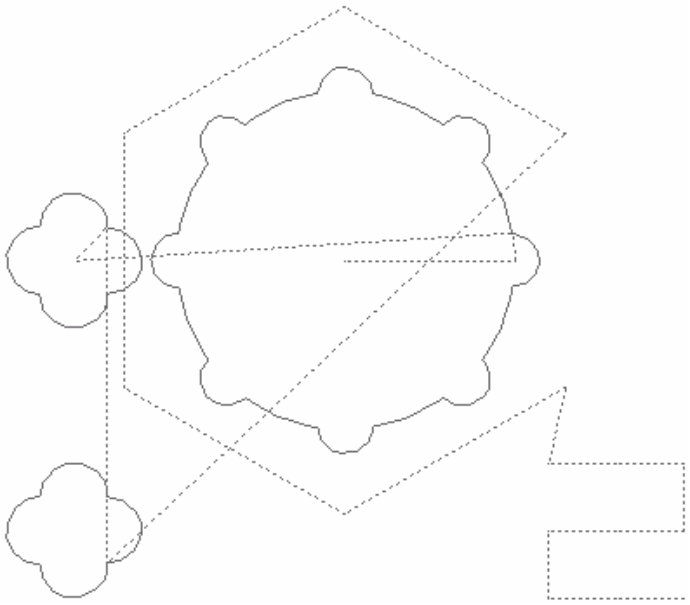
```
G54 X0 Y0 F3000.;
G16;                          // start polar coordinates
G90 G00 X50. Y9.207 R8.;      // positioning to starting point
M98 H100;                     // first process
G68 X0 Y0 R90.;               // coordinate rotates 90∘
M98 H100;                     // second process
G68 X0 Y0 R180.;              // coordinate rotates 180∘
M98 H100;                     // third process
G68 X0 Y0 R270.;              // coordinate rotates 270∘
M98 H100;                     // fourth process
G69;                          // coordinate rotation cancel
G15;                          // polar coordinate cancel
M02;                          // main program end
N100                          // orbit sub-program start
G90 G01 X50. Y9.207 R8.;
G03 X50. Y80.793. R50.;
G03 X50. Y99.207 R8.;
M99;                          // orbit sub-program return
```



Example 2:

```
G54 X0 Y0 F3000.;
G16;                              // start polar coordinate
G90 G00 X50. Y9.207 R8.;          // positioning to starting point
M98 H100;                         // first process
G68 X0 Y0 R45.;                   // coordinate rotates 45∘
M98 H100;                         // second process
G68 X0 Y0 R90.;                   // coordinate rotates 90∘
M98 H100;                         // thied process
G68 X0 Y0 R135.;                  // coordinate rotates 135∘
M98 H100;                         // fourth process
G68 X0 Y0 R180.;                  // coordinate rotates 180∘
M98 H100;                         // fifth process
G68 X0 Y0 R225.;                  // coordinate rotates 225∘
M98 H100;                         // sixth process
G68 X0 Y0 R270.;                  // coordinate rotates 270∘
M98 H100;                         // seventh process
G68 X0 Y0 R315.;                  // coordinate rotates 315∘
M98 H100;                         // eighth process
G69;                              // coordinate rotates cancel
G15;                              // polar coordinate cancel
G00 X-80. Y0.
M98 H200;                         // process first “flower”
G51.1 Y-40.;                      // symmetry axis Y-40.
M98 H200;                         // process second “flower”
G50;                              // mirror image cancel
G90 G81 Z-20. R2. F1000. K0;      // start G81 drilling cycle
G134 X0 Y0 I75. J30. K6;          // circumference hole cycle
G137.1 X60. Y-60. I20. J-20. P3 K3; // chess type hole cycle
G80;                              // drilling cycle cancel
```

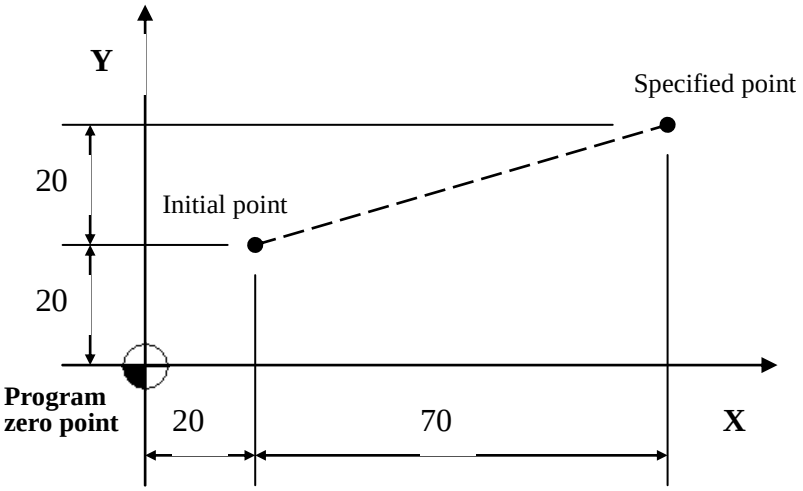
```
M02;                                    // main program end
N100                                    // orbit sub-program
G90 G01 X50. Y9.207;
G03 X50. Y35.793 R50.;
G03 X50. Y54.207 R8.;
M99;                                    // sub-program return
N200                                    // sub-program start (flower)
G90 G00 X-70. Y10.;
G91 G03 X-20. R10.;
G03 Y-20. R10.;
G03 X20. R10.;
G03 Y20. R10.;
M99;                                    // sub-program return(flower)
```



4.1.29 Absolute/Incremental Positioning

Command form:

G90: absolute command.
G91: incremental command.



Absolute: G90 G00 X90.0 Y40.0；

Incremental: G91 G00 X70.0 Y20.0；

4.1.30 Absolute Preset

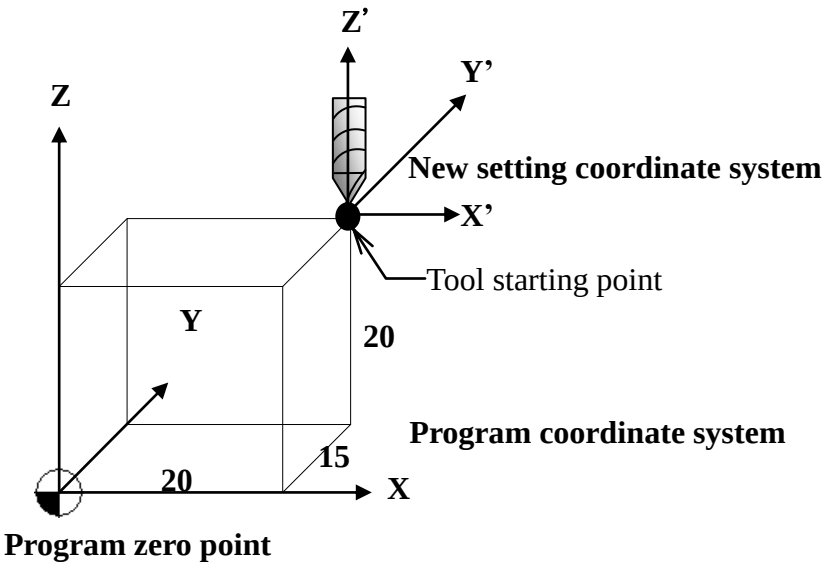
Command form:

G92 X_ Y_ Z_ ;

X, Y, Z: establish the current position of the machine to this value

Description:

The G92 is used to establish a temporary Program Coordinate System (PCS). The axis words coded in the same line with the G92 establish the current axis position to those axis words.



G92 X20.0 Y15.0 Z20.0；

4.1.31 Inverse Time Feedrate

Command form:

G93

Description:

A control mode in which the federate is specified as one divided by the time to complete the move. This value is usually computed by dividing the desired federate by the length of the actual tool path.

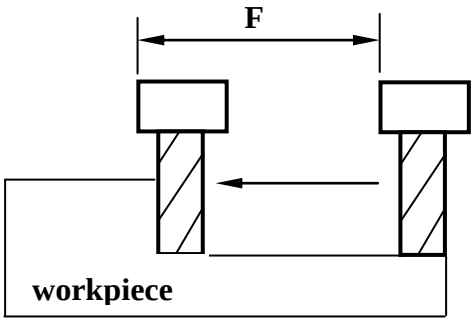
4.1.32 Feedrate Unit Setting

Command form:

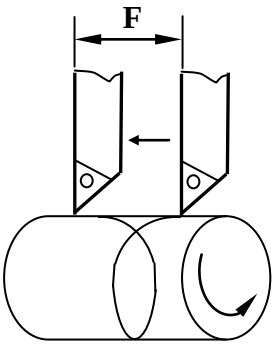
G94 F__;
G95 F__;

Description:

This command set the unit of feedrate of F_ function (tool move distance per unit time or move distance per revolution)；G94 is feed value per minute, unit: mm/min, inch/min, G95 is feed value per revolution, unit: mm/rev, inch/rev.



G94. feed per minute(mm/min or inch/min)



G95. feed per revolution(mm/rev or inch/rev)

Note:

G95 is rarely used in milling machine, more commonly be used in Lathe machine.

4.1.33 Return to Initial/Rapid Plane

Command form:

G98; //return to initial plane
G99; //return to rapid plane

Description:

This is a control mode in which, after performing a fixed cycle, the Z axis is returned to the initial/rapid plane. The initial plane location is identified by the Z axis location prior to a fixed cycle definition. The rapid plane location is identified together with the fixed cycle declaration.

4.2 Common M codes

M codes	Description
M00	Unconditional Stop
M01	Conditional Stop
M02	End of Program
M03	Spindle Clockwise
M04	Spindle Counter-Clockwise
M05	Spindle Stop
M06	Tool Change
M07	Coolant/Mist Enable
M08	Mist/Coolant Enable
M09	Coolant & Mist Disable
M12	Coolant Through Spindle (CTS)
M17	End of Subroutines
M19	Spindle Orientation
M30	End of Program/Subroutines
M36	Auger Forward
M37	Auger Stop
M38	Wash Down
M39	Wash Down off
M40	Probe off
M41	Probe on
M42	Select Spindle Probe
M43	Select Tool Setter
M48	Potentiometer Controls In
M49	Potentiometer Controls Out
M60	Air Brake for 4th Axis
M61	Release Air Brake for 4th Axis
M62	Air Brake for 4th Axis
M63	Release Air Brake for 4th Axis
M68	Coolant Through Spindle (CTS)
M69	Coolant Through Spindle (CTS) Disable

M98	Sub Program Call
M99	End of Subprogram

4.2.1 M00 Unconditional Stop

Description:

M00 temporarily suspends program execution and cancels the spindle and coolant functions. The CNC enters the "Waiting" status until the operator pushes the "Cycle Start" button to continue program execution.

Example:

```
G80
M05 M09
M06 T03 //tool #3
M00
// the program will stop at this line, and will not continue until the "Cycle Start" is pressed
G90 G00 S7500 M03 E1 X1.43 Y-2.7 //E1 = G54
```

Note:

In Format 1/2, another way to accomplish a program stop is to use G04 P66000 on a line in the program. This will cause the control to enter the "Waiting" status. In this case the spindle and coolant will not be turned off.

4.2.2 M01 Conditional Stop

Description:

M01 is similar to M00 with the exception that the program will stop only when the optional stop button is in the ON position. This code could be included in a program for the convenience of the operator to allow the program to stop at certain points.

Example:

```
G80
M05 M09
M06 T03 //tool #3
M01
// the program will stop at this line only if the optional stop button is in the ON position
G90 G00 S7500 M03 E1 X1.43 Y-2.7 //E1 = G54
```

4.2.3 M02 End of Program

Description:

M02 indicates the end of the main program, and reset the control.

4.2.4 M03 Spindle Clockwise

Description:

M03 is used to start the spindle rotation in a clockwise direction.

Example:

M06 T03 //tool #3
M01
G90 G00 S7500 M03 E1 X0 Y0 //the spindle will turn on CW this line

4.2.5 M04 Spindle Counter-Clockwise

Description:

M04 is used to start the spindle rotation in a counter-clockwise direction.

Example:

M06 T03 //tool #3
M01
G90 G00 S7500 M04 E1 X0 Y0 //the spindle will turn on CCW this line

4.2.6 M05 Spindle Stop

Description:

M05 stops the spindle and the coolant. The spindle will neither orient nor lock.

4.2.7 M06 Tool Change

Description:

M06 changes tools in the spindle. The M06 usually appears in a line with a T# code. The T# will specify which tool to pick up next.

Tool change command can be called in the following options:

Command	Machine Action
M06 T#	Call for T# tool, i.e. ready the T# and then perform tool change
T# M06	Call for T# tool, i.e. ready the T# and then perform tool change
T#	Tool magazine will rotate and prepare the T#
M06	Perform tool change

A general action of M06 is as:

1. Prepare the T# in tool magazines (If M06 T# or T# M06 is used)
2. Pot down
3. Z axis move to the tool change position
4. Spindle Orientation
5. Proceed with tool change
6. Pot Up

Example:

M05 M09
G90 G00 G53 Z0
M06 T03 //tool #3

4.2.8 M07/M08 Coolant/Mist Enable

Description:

M07/M08 activates the coolant/mist function. Define by user parameter, see section 2.8.1 user parameter no.17. Default setting is 0, which implies M7 enable flood coolant function and M8 enable mist function.

Example:

```
M06 T03 //tool #3
G90 G00 S7500 M03 E1 X0 Y0
H3 D3 Z-100. M7 // Flood coolant is turned on
```

4.2.9 M09 Coolant/Mist Disable

Description:

M09 deactivates both the coolant and mist function.

Example:

```
E0 X0 Y0
M5 M9 //Cancel both coolant and mist
```

4.2.10 M12 Coolant Through Spindle (CTS)

Description:

M12 activates the CTS function.

Example:

```
M06 T03 //tool #3
G90 G00 S7500 M03 E1 X0 Y0
H3 D3 Z-100. M12 // CTS is turned on
```

4.2.11 M17 End of Subroutines

Description:

The M17 code is used to mark the end of a subroutine. No other coding is allowed on the same line with the M17. A M17 is required to mark the end of a subroutine.

Example:

```
N1 O1234
N2 H1 M7 Z50.
N3 L101
N4 M5 M9
N5 M2
N6 L100
N7 // content of sub #1
N8 M17
```

4.2.12 M19 Spindle Orientation

Description:

The M19 code is used to orientate the Spindle. The Spindle will rotate

at 50 RPM and stop at the position that ready for tool change.

Example:

N1 O1234
N2 M19 //You don't have to put M19 before TxM6.
N3 T6M6
N4 G00 Z0.
N5 M30

4.2.13 M30 End of Program

Description:

The M30 is the same as M02, a mark of end of program and will reset the control.

4.2.14 M36 Auger Forward

Description:

Turn on the auger in forward direction.

Example:

M36 //Auger FW

4.2.15 M37 Auger Stop

Description:

Turn off the auger.

Example:

M37 //Auger Stop

4.2.16 M38 Wash Down

Description:

Turn on the wash down.

Example:

M38 //Wash Down

4.2.17 M39 Wash Down Off

Description:

Turn off the wash down.

Example:

M39 //Wash Down off

4.2.18 M40 Probe Off

Description:

The M40 code is used to turn the spindle probe or tool setter off.

Example:

```
N1 O1234
N2 T6 M15
N3 M41  //Turn spindle probe/tool setter on
N4 G31 Z-5. F60.
N5 G4 X1500
N6 M40  //Turn spindle probe/tool setter off
N7 M30
```

4.2.19 M41 Probe On

Description:

The M41 code is used to turn the spindle probe or tool setter on. See the M40 example above.

4.2.20 M42 Select Spindle Probe

Description:

The M42 code is used to select the spindle probe. Selection is allowed only when the spindle probe and tool setter are off.

Example:

```
N1 O1234
N2 T6 M15
N3 M42  //Select spindle probe
N4 G4 X1000 //Dwell
N5 M41  //Turn the spindle probe on
N6 G31 Z-5. F40.
N7 G4 X1500
N8 M40  //Turn the spindle probe off
N9 M43  // Select tool setter
N10 T1 M15
N11 M41 //Turn the tool setter on
N12 G54 G00 X0. Y20.
N13 G31 Z-10. F40.
N14 G4 X1500
N15 M40 //Turn the tool setter off
N16 M30
```

4.2.21 M43 Select Tool Setter

The M42 code is used to select the tool setter. Selection is allowed only when the spindle probe and tool setter are off. See the M42 example above.

4.2.22 M48 Potentiometer Controls In

Description:

This code enables the operator to override the programmed federate and spindle RPM by use of the potentiometers located on the pendant.

Example:

M49 //**Cancel the operator's ability to override the feed rate and**

RPM
G85 G99 R0+.3 Z-.7 F100 X3.78 Y1
X3. Y5.
M48//Enable the operator to alter the feed rate and RPM

4.2.23 M49 Potentiometer Controls Out

Description:

This code disables the potentiometers located on the pendant. See the M48 example above.

4.2.24 M60 Air Brake for 4th Axis

Description:

The M60 code enables the 4th axis air brake. If there is any axis moving command in the same line. Control will execute axis moving before clamp the 4th axis.

Example:

N1 O1234
N2 T6 M15
N3 G54 G00 X0. Y1. A30. M60
N4 M3 S8000
N5 G81 Z-10. R-5. F60
N6 M61
N7 A60. M60
N8 M61
N9 A90. M60
N10 M80
N11 M61
N12 M30

4.2.25 M61 Release Air Brake for 4th Axis

Description:

This code releases the air brake for 4th axis. See the M60 example above.

4.2.26 M62 Air Brake for 5th Axis

Description:

The M62 code enables the 5th axis air brake. If there is any axis moving command in the same line. Control will execute axis moving before clamp the 5th axis.

Example:

N1 O1234
N2 T6 M15
N3 G54 G00 X0. Y1. B30. M62
N4 M3 S8000
N5 G81 Z-10. R-5. F60
N6 M63
N7 B60. M62

```
N8 M63
N9 B90. M62
N10 M80
N11 M63
N12 M30
```

4.2.27 M63 Release Air Brake for 5th Axis

Description:

This code releases the air brake for 4th axis. See the M62 example above.

4.2.28 M68 Coolant Through Spindle (CTS)

Description:

M12 activates the CTS function.

Example:

```
M06 T03 //tool #3
G90 G00 S7500 M03 E1 X0 Y0
H3 D3 Z-100. M12 // CTS is turned on
```

4.2.29 M69 Coolant through Spindle(CTS) Disable

Description:

M69 deactivates coolant through Spindle(CTS).

Example:

```
E0 X0 Y0
M5 69 //Cancel coolant through Spindle
```

4.2.30 M98 Sub Program Call

Description:

The M98 is used to call for subprogram. The command of M98 is as:

M98 P_ H_ L_

Which, P: subprogram #
H: sequence # (N)
L: repetition

Example 1 :

```
O1234
H1 M7 Z50.
M98 P2 L1 // call for O2 subprogram one time
M5 M9
M30
O2
// content of sub O2
M99
```

Example 2 :

```
O1234
H1 M7 Z50.
M98 H200 L1 // jump to N200 to execute subprogram one time
M5 M9
M30
N200
// content of subprogram N200
M99
```

Example 3 :

```
O1234
H1 M7 Z50.
M98 P2 H200 L1 // jump to N200 to execute subprogram one time
M5 M9
M30
O2
N100
//content of sub#2
N200
//content of sub#2
M99
```

4.2.31 M99 End of Subprogram

Description:

The M99 is used to mark the end of a subprogram and will return to the main program. The command of M99 is as:

M99 or M99 P_ or M99 Q_

Which, P: sequence # (N)
Q: line # (L)

Example 1 :

```
O1234
H1 M7 Z50.
M98 P2 L1 // call for O2 subprogram one time
M5 M9
M30
O2
// content of sub O2
M99 //return to the line after M98
```

Example 2 :

```
N1 O1234
N2 H1 M7 Z50.
N3 M98 P2 L1 // call for O2 subprogram one time
N4 M5 M9
N5 M30
O2
```

```
// content of sub O2
M99 P4//return to the line with N4
```

Example 3 :

```
N1 O1234
N2 H1 M7 Z50.
N3 M98 P2 L1 // call for O2 subprogram one time
N4 M5 M9
N5 M30
O2
// content of sub O2
M99 Q4//return to the line #4, i.e. in this example, the line with N4
```

4.3 Common S Code

Auto mechanical belt change: S_
Manual mechanical belt change: S_.1 / S_.2
S: Spindle speed

Description:

The S code is used to assign the spindle speed command and also execute mechanical belts change if necessary.

For auto belt change, low range will be activated when the speed command is between 0-2500RPM. High range will be activated when the speed command is between 2501-10000RPM.

For the manual belt change, low range will be activated when S_.1 is executed no matter what the speed command is. However, it will stay in 2500 RPM if the speed command for S_.1 is higher than 2500RPM. Similarly, high range will be activated when S_.2 is executed no matter what the speed command is and it will stay in 10000 RPM if speed command is higher than 10000 RPM.

Mechanical belts change includes 3 stages. First, stop the spindle. Second, make a belts change. Third, restore the running status before mechanical belts change. If the spindle was running clockwise before belts change then spindle will run clockwise in the command speed after belts change. If the spindle was stop then control will make a belt change only.

Example 1 :

```
O1234
S6000     //Enable high range
M3        //Run the spindle to 6000 RPM
…
S2400     //Enable low range and run the spindle to 2400 RPM
…
M5
M30
```

Example 2 :

```
N1 O1234
```

```
N2 M3 S1600.2 //Enable high range and run the spindle in 1600 RPM
…
N3 S1000      //Enable high range and run the spindle in 1600 RPM
N4 M5
N5 M30
```

4.4 Fixed Cycles

Fixed Cycles	Format 1/2	Format 3
Peck Drilling	G73	G73
Left Hand Rigid Tapping	G74	G74
Fine Boring	G76	G76
Spot Drilling	G81	G81
Center Drilling	G82	G82
Deep Hole Drilling	G83	G83
Right Hand Rigid Tapping	G84	G84
Bore In, Bore Out	G85	G89
Bore In, Spindle Off, Orient, Rapid Out	G86	G87
Bore In，Bore Out (manual out)	G87	G88
Bore In，Dwell， Bore Out	G88	G89
Bore In，Dwell， Bore Out (manual out)	G89	G88

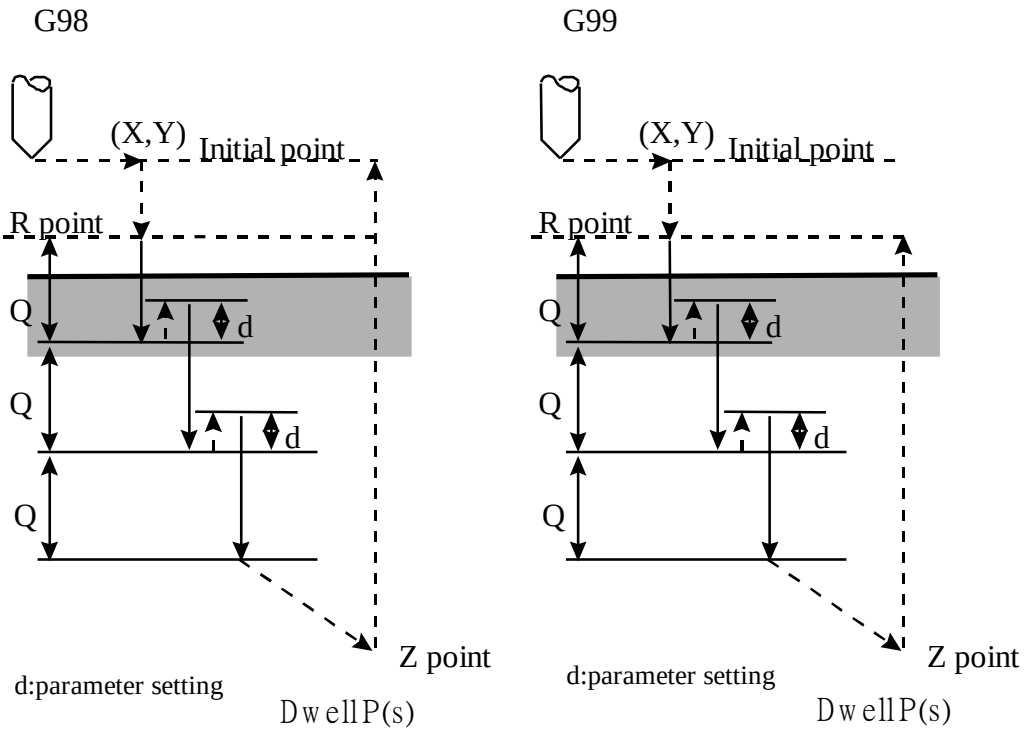
4.4.1 Peck Drilling

Command form:

G73 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_or Y_: hole position data (absolute/increment)
Z_:
G91: the distance from the bottom of the hole to point Z (directional)
G90: program position of point Z
R_:
G91: the distance from initial level to R point level (directional)
G90: program position of point R
Q_: depth of cut for each cutting feed (negative value will be ignore)
P : retract distance, effect in Format1&2
F_: feedrate
K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



Notes:

1. **d** distance in Format 3 has to be defined in the maker parameter No.4002
2. Before using G73, please use M function to turn the spindle
3. G73 is modal G Code, once G73 is defined, it will drill in every XY move until G80 is encounter

Description:

1. use G00 to move to specified (X,Y)
2. use G00 to reach specified R point.
3. use G01 to interpolate a distance Q at the present depth
4. use G00 to return a distance **d** (Format1/2: specified with P word; Format3: maker parameter 4002)
5. repeat drilling hole until reach the Z point
6. use G00 to return initial point(G98) or programmable R point(G99)

Condition:

1. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
2. G00/G01/G02/G03 cannot be in the line in which G73 exist
3. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
M03; // start the drill to turn CW
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
G90 G99; //set the R point, Z point and hole 1, cutting rate 2.0
G73 X5. Y5. Z-10. R-5. Q2.;
X15.; // hole 2
Y15. K2; // hole 3, drill twice
G98 X5.; // hole 4, and return to initial point
X10. Y10. Z-20.; // hole 5, and set new Z point be -20
G80;
M05; // spindle stop
M02;
```

4.4.2 Left Hand Rigid Tapping

Command form:

G74 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ or Y_: coordinates of holes (absolute/increment)

Z_:

G91: the distance from the bottom of the hole to point Z (directional)

G90: program position of point Z

R_:

G91: the distance from initial level to R point level (directional)

G90: program position of point R

P_: dwell time (s)

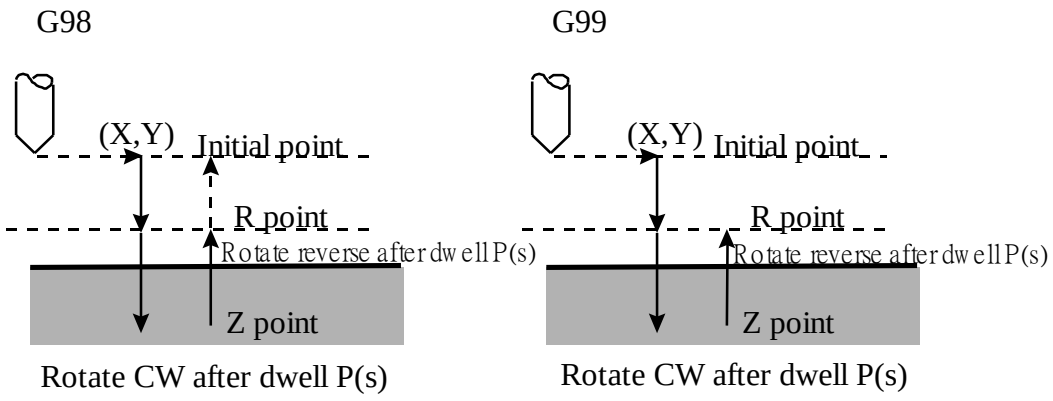
F_:

Format 1: spindle RPM

Format 2/3: feedrate

K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



Rotate CW after dwell P(s) Rotate CW after dwell P(s)

Notes:

1. Before using G74, please use M function to turn the spindle
2. G74 is modal G Code, once G74 is defined, it will tap in every XY move until G80 is encounter
3. Feedrate and spindle switches are not effective during tapping
4. In Format 1/2, G74.1 is equal to G74

Description:

1. use G00 to move to specified(X,Y)
2. use G00 to specified point R.
3. use G01 to tap to the bottom of the hole ,point Z
4. dwell P(s) then reverse the tap
5. use G01 raise to point R
6. dwell P(s) then reverse the tapping
7. use G00 to raise to initial point (G98) or programmable point R(G99)

Condition:

1. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
2. G00/G01/G02/G03 cannot be in the line in which G74 exist
3. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
M04; // start drill to rotate CCW
G90 G99;
//specify point R、 point Z and hole 1 coordinate values, dwell 2 s
G74 X5. Y5. Z-10. R-5. P2.;
X15.; // hole 2
Y15.; // hole 3
G98 X5.; // hole 4, and set to return to initial point
X10. Y10. Z-20.; // hole 5, and set new point Z to be -20.
G80;
M05; // spindle stop
M02;
```

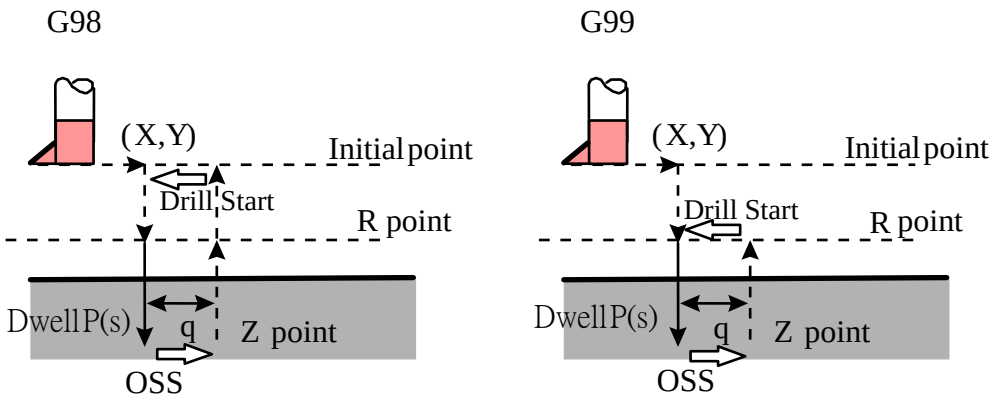
4.4.3 Fine Boring

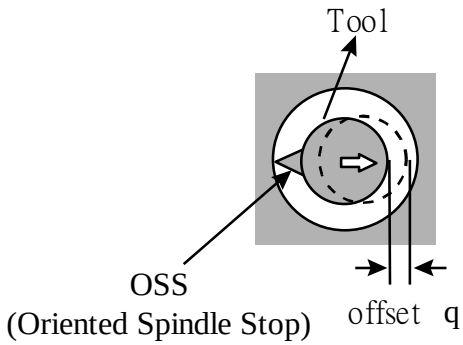
Command form:

G76 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_ or Y_: hole position data (absolute/increment position)
Z_:
G91: the distance from the bottom of the hole to point Z (directional)
G90: program position of point Z
R_:
G91: the distance from initial level to R point level (directional)
G90: program position of point R
Q_: shift amount at the bottom of the hole (negative value will be ignored)
P_: dwell time at the bottom of the hole (s)
F_: feedrate
K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode





Oriented Spindle Stop(OSS)

Notes:

1. **d** distance in Format 3 has to be defined in the maker parameter No.4002
2. Before using G76, please use M function to turn the spindle
3. G76 is modal G Code, once it is defined, it will drill in every XY move until G80 is encounter

Description:

1. use G00 to move tool to specified (X, Y) point
2. use G00 reach the specified R point(not include spindle positioning)
3. use G01 reach point Z at the bottom of the hole, dwell P(s) and spindle positioning and stop the boring
4. shift Q distance
5. use G00 raise to initial point (G98) or programmable point R (G99)
6. shift Q distance in reverse direction
7. bore start

Condition:

1. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
2. G00/G01/G02/G03 cannot be in the line in which G76 exist
3. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
M03; // spindle CW
G90;
G00 X0. Y0. Z10.; // position to initial point
G17;
G90 G99;
//specify point R, point Z, and hole 1, shift amount at bottom of hole2.0, dwell time 5 s
G76 X5. Y5. Z-10. R-5. Q2. P5.;
X15.; // hole 2
Y15.; // hole 3
G98 X5.; // hole 4, and return to initial point
X10. Y10. Z-20.; // hole 5, and specify the new point Z to be -20.0
G80;
M05; // spindle stop
M02;
```

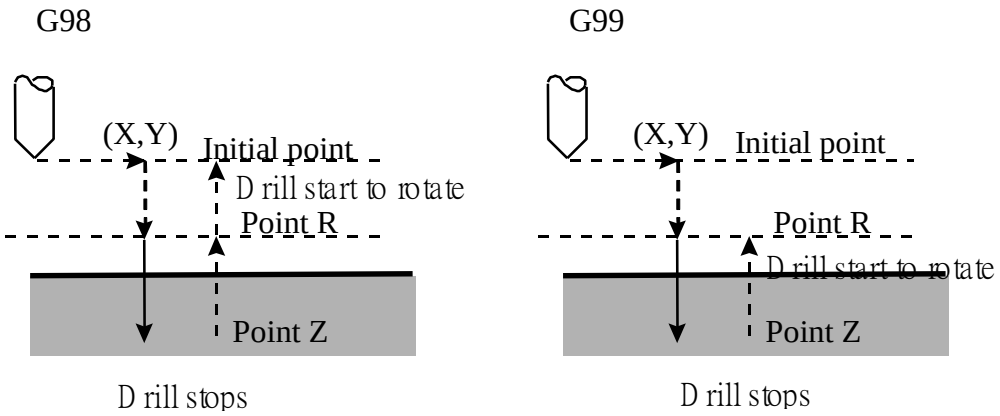
4.4.4 Spot Drilling

Command form:

G81 X_ Y_ Z_ R_ F_ K_ ;

X_ or Y_: coordinates of holes (absolute/increment)
Z_:
G91: the distance from the bottom of the hole to point Z (directional)
G90: program position of point Z
R_:
G91: the distance from initial level to R point level (directional)
G90: program position of point R
F_: feedrate
K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



Notes:

1. Before using G81, please use M function to turn the spindle
2. G81 is modal G Code, once it is defined, it will drill in every XY move until G80 is encounter
3. Feedrate and spindle switches are not effective during tapping

Description:

1. use G00 to positioning to specified (X,Y)
2. use G00 to reach specified point R.
3. use G01 to reach point Z the bottom of the hole,
4. use G00 to raise to initial point (G98) or program point R(G99)

Condition:

1. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
2. G00/G01/G02/G03 cannot be in the line in which G81 exist
3. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
M03; // spindle CW
G90 G99; //specify point R, point Z, and hole 1
G81 X5. Y5. Z-10. R-5.;
X15.; // hole2
Y15.; // hole3
G98 X5.; // hole4, and return to initial point
G80;
M05; // spindle stop
M02;
```

4.4.5 Center Drilling

Command form:

G82 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ or Y_: coordinates of holes (absolute/increment)

Z_:

G91: the distance from the bottom of the hole to point Z (directional)

G90: program position of point Z

R_:

G91: the distance from initial level to R point level (directional)

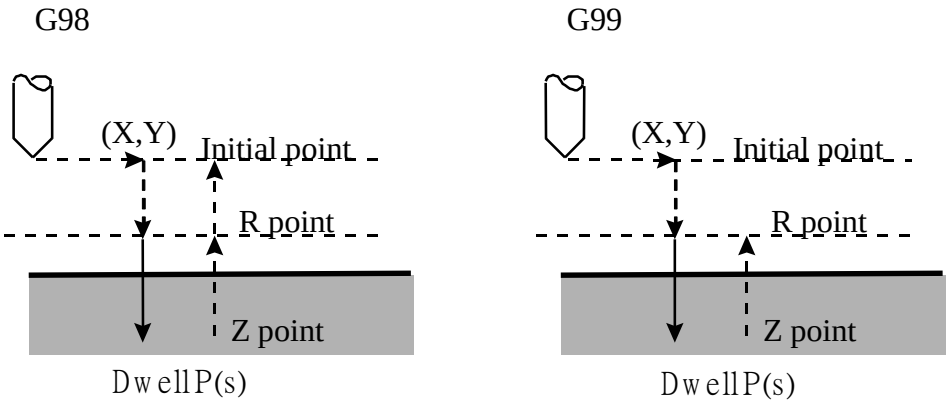
G90: program position of point R

P_: dwell time at the bottom of the hole (s)

F_: feedrate

K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



Notes:

1. Before using G82, please use M function to turn the spindle
2. G82 is modal G Code, once it is defined, it will drill in every XY move until G80 is encounter
3. Feedrate and spindle switches are not effective during tapping

Description:

1. use G00 to positioning to specified (X,Y)
2. use G00 to reach specified point R.
3. use G01 to reach point Z the bottom of the hole,
4. use G00 to raise to initial point (G98) or program point R(G99)

Condition:

1. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
2. G00/G01/G02/G03 cannot be in the line in which G82 exist
3. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
M03; // spindle CW
G90 G99;
//specified point R, point Z and hole 1, dwell time 2 s
G82 X5. Y5. Z-10. R-5. P2.;
X15.; // hole2
Y15.; // hole3
G98 X5.; // hole4, and return to initial point
G80;
M05; // spindle stop
M02;
```

4.4.6 Deep Hole Drilling

Command form:

G83 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_ or Y_: hole position data (absolute/increment)

Z_:

G91: the distance from the bottom of the hole to point Z (directional)

G90: program position of point Z

R_:

G91: the distance from initial level to R point level (directional)

G90: program position of point R

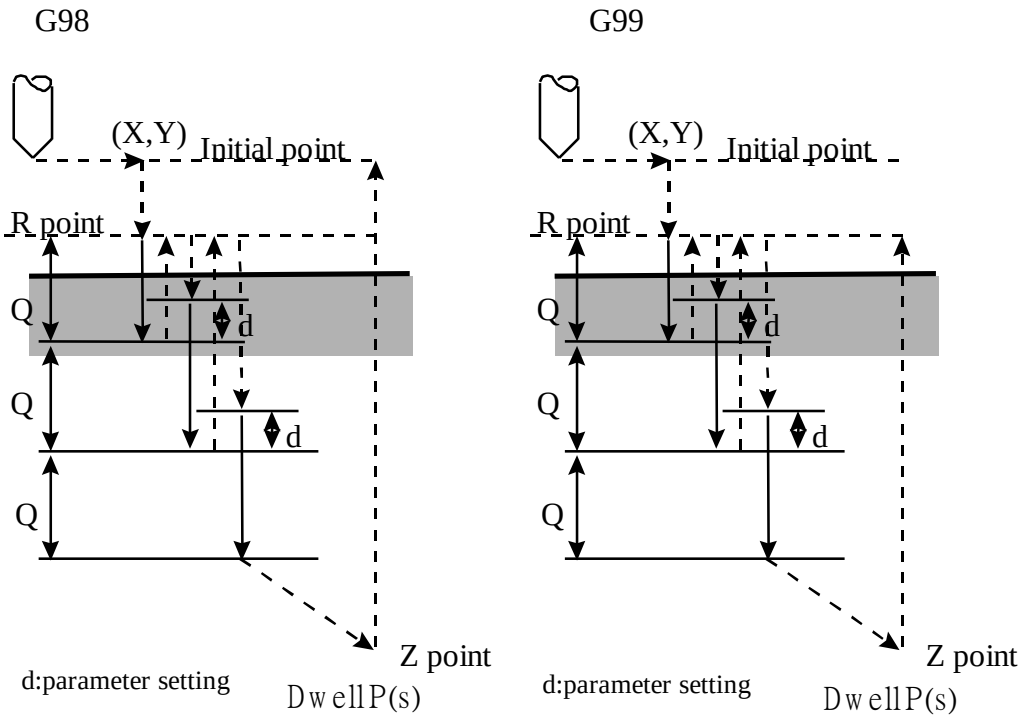
Q_: depth of cut for each cutting feed (negative value will be ignore)

P : retract distance, effect in Format1&2

F_: feedrate

K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



Notes:

1. **d** distance in Format 3 has to be defined in the maker parameter No.4002
2. Before using G83, please use M function to turn the spindle
3. G83 is modal G Code, once G83 is defined, it will drill in every XY move until G80 is encounter

Description:

1. use G00 to positioning to specified (X,Y)
2. use G00 to reach specified point R.
3. use G01 to interpolate a distance Q at the present depth
4. use G00 raise to point R
5. use G00 reach a distance **d**
6. use G01 to interpolate a distance Q at the present depth
7. use G00 raise to point R
8. repeat performing until the bottom of the hole point Z
9. use G00 raise to initial point (G98) or program point R(G99)

Condition:

1. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
2. G00/G01/G02/G03 cannot be in the line in which G83 exist

3. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
M03; // start drill to rotate CW
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
G90 G99; // specify point R, point Z and hole 1, cutting federate 3.0
G83 X5. Y5. Z-10. R-5. Q3.;
X15.; // hole2
Y15.; // hole3
G98 X5.; // hole4, and return to initial point
G80;
M05; // drill stops
M02;
```

4.4.7 Right Hand Rigid Tapping

Command form:

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ or Y_: coordinates of holes (absolute/increment)

Z_:

G91: the distance from the bottom of the hole to point Z (directional)

G90: program position of point Z

R_:

G91: the distance from initial level to R point level (directional)

G90: program position of point R

P_: dwell time (s)

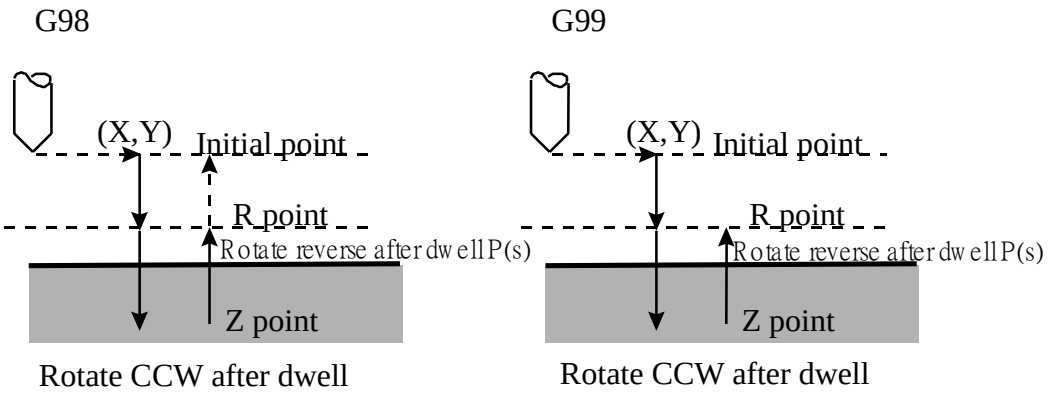
F_:

Format 1: spindle RPM

Format 2/3: feedrate

K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



Rotate CCW after dwell Rotate CCW after dwell

Notes:

1. Before using G84, please use M function to turn the spindle
2. G84 is modal G Code, once G84 is defined, it will tap in every XY move until G80 is encounter
3. Feedrate and spindle switches are not effective during tapping
4. In Format 1/2, G84.1 is equal to G84

Description:

1. use G00 to move to specified(X,Y)
2. use G00 to specified point R.
3. use G01 to tap to the bottom of the hole ,point Z
4. dwell P(s) then reverse the tap
5. use G01 raise to point R
6. dwell P(s) then reverse the tapping
7. use G00 to raise to initial point (G98) or programmable point R(G99)

Condition:

1. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
2. G00/G01/G02/G03 cannot be in the line in which G84 exist
3. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
M03; // start drill to rotate CCW
G90 G99;
//specify point R、 point Z and hole 1 coordinate values, dwell 2 s
```

```
G84 X5. Y5. Z-10. R-5. P2.;
X15.; // hole 2
Y15.; // hole 3
G98 X5.; // hole 4, and set to return to initial point
X10. Y10. Z-20.; // hole 5, and set new point Z to be -20.
G80;
M05; // spindle stop
M02;
```

4.4.8 Bore In, Bore Out

Command form:

Format1/2:

G85 X_ Y_ Z_ R_ F_ K_ ;

Format3:

G89 X_ Y_ Z_ R_ F_ K_ ;

X_ or Y_: coordinates of holes (absolute/increment)

Z_:

G91: the distance from the bottom of the hole to point Z (directional)

G90: program position of point Z

R_:

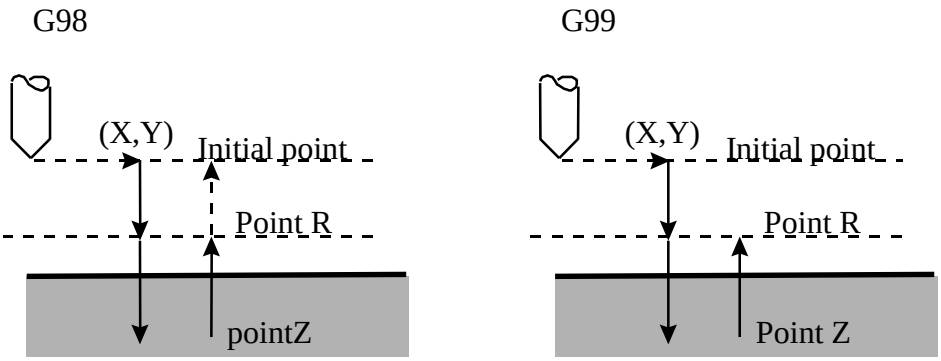
G91: the distance from initial level to R point level (directional)

G90: program position of point R

F_: feedrate

K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



Notes:

4. Before using G85/G89, please use M04 function to turn the spindle
5. G85/G89 is modal G Code, once G85/G89 is defined, it will tap in every XY move until G80 is encounter

6. Feedrate and spindle switches are not effective during tapping

Description:

7. use G00 to positioning to specified (X,Y) when start to perform
8. use G00 to reach specified point R.
9. use G01 to reach point Z the bottom of the hole
10. use G01 to raise to point R
11. use G00 to raise to initial point (G98) or program point R(G99)

Condition:

12. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
13. G00/G01/G02/G03 cannot be in the line in which G85/G89 exist
14. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
M03; // start drill to rotate CW
G90 G99;
//specify point R, point Z, and hole1
G85 X5. Y5. Z-10. R-5.;
X15.; // hole2
Y15.; // hole3
G98 X5.; // hole4, and return to initial point
G80;
M05; // drill stops
M02;
```

4.4.9 Bore In, Spindle Off, Rapid Out

Command form:

Format1/2:

G86 X_ Y_ Z_ R_ F_ K_ ;

Format3:

G87 X_ Y_ Z_ R_ F_ K_ ;

X_ or Y_: coordinates of holes (absolute/increment)

Z_:

G91: the distance from the bottom of the hole to point Z (directional)

G90: program position of point Z

R_:

G91: the distance from initial level to R point level (directional)

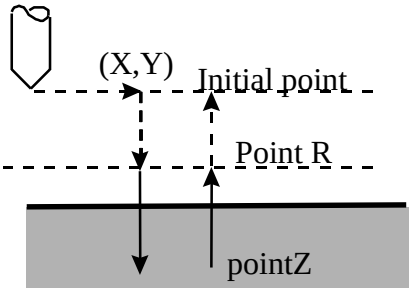
G90: program position of point R

F_: feedrate

K_: number of repetition

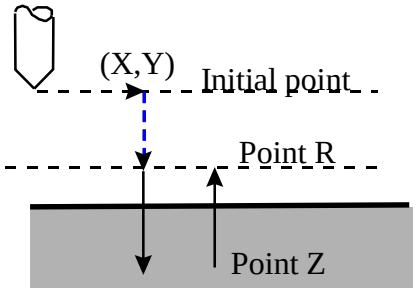
X, Y, Z, R can be used in both G90 and G91 mode

G98



Spindle Stop and Orient

G99



Spindle Stop and Orient

Notes:

15. Before using G86/G87, please use M04 function to turn the spindle
16. G86/G87 is modal G Code, once G86/G87 is defined, it will tap in every XY move until G80 is encounter
17. Feedrate and spindle switches are not effective during tapping

Description:

1. use G00 to positioning to specified (X,Y) when start to perform
2. use G00 to reach specified point R.
3. use G01 to reach point Z the bottom of the hole
4. use G01 to raise to point R
5. use G00 to raise to initial point (G98) or program point R(G99)

Condition:

6. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
7. G00/G01/G02/G03 cannot be in the line in which G86/G87 exist
8. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

F1000. S500;

```
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
G90 G99;
M03; // start drill to rotate CW
//specify point R, point Z and hole 1, shift amount 5.0, dwell time 4.0s
G86 X5. Y5. Z10. R-30.;
X15.; // hole2
Y15.; // hole3
G80;
M05; // drill stops
M02;
```

4.4.10 Bore In, Bore Out (manual out)

Command form:

Format1/2:

G87 X_ Y_ Z_ R_ F_ K_ ;

Format3:

G88 X_ Y_ Z_ R_ F_ K_ ;

X_ or Y_: coordinates of holes (absolute/increment)

Z_:

G91: the distance from the bottom of the hole to point Z (directional)

G90: program position of point Z

R_:

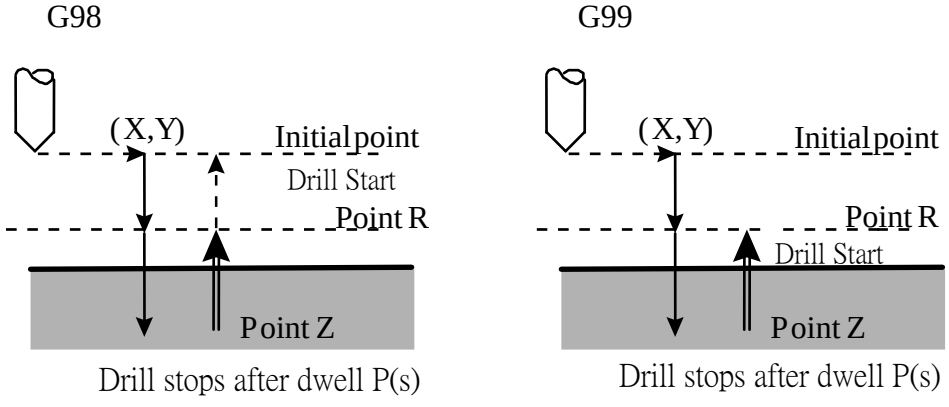
G91: the distance from initial level to R point level (directional)

G90: program position of point R

F_: feedrate

K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



※ ⟹ for positioning by manual.

Notes:

9. Before using G87/G88, please use M function to turn the spindle
10. G87/G88 is modal G Code, once it is defined, it will tap in every XY move until G80 is encounter
11. Feedrate and spindle switches are not effective during tapping

Description:

12. use G00 to positioning to specified (X,Y)
13. use G00 to reach specified point R.
14. use G01 to reach point Z the bottom of the hole,
15. make the tool out of workpiece in manual mode and reset
16. use G01 to move to point R
17. use G00 to raise to initial point(G98) or program point R(G99)
18. drill rotate CW.

Condition:

19. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
20. G00/G01/G02/G03 cannot be in the line in which G86/G87 exist
21. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
M03; // start drill to rotate CW
G90 G99;
//specify point R, point Z and, hole1
G87 X5. Y5. Z-10. R-5..;
X15.; // hole2
Y15.; // hole3
G98 X5.; // hole4, and return to initial point
G80;
M05; // drill stops
M02;
```

4.4.11 Bore In, Dwell, Bore Out

Command form:

Format1/2:

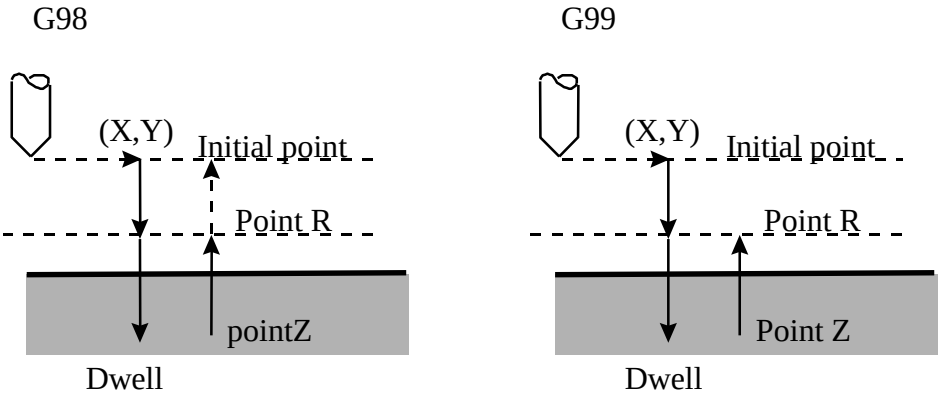
G88 X_ Y_ Z_ R_ P_ F_ K_ ;

Format3:

G89 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ or Y_: coordinates of holes (absolute/increment)
Z_:
G91: the distance from the bottom of the hole to point Z (directional)
G90: program position of point Z
R_:
G91: the distance from initial level to R point level (directional)
G90: program position of point R
P : Dwell time (s)
F_: feedrate
K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



Notes:

1. Before using G88/G89, please use M04 function to turn the spindle
2. G85/G89 is modal G Code, once G88/G89 is defined, it will tap in every XY move until G80 is encounter
3. Feedrate and spindle switches are not effective during tapping

Description:

1. use G00 to positioning to specified (X,Y) when start to perform
2. use G00 to reach specified point R.
3. use G01 to reach point Z the bottom of the hole
4. use G01 to raise to point R
5. use G00 to raise to initial point (G98) or program point R(G99)

Condition:

1. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
2. G00/G01/G02/G03 cannot be in the line in which G88/G89 exist
3. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
M03; // start drill to rotate CW
G90 G99;
//specify point R, point Z and hole1, dwell 2.5s
G88 X5. Y5. Z-10. P2.5 R-5.;
X15.; // hole2
Y15.; // hole3
G98 X5.; // hole4, and return to initial point
G80;
M05; // drill stops
M02;
```

4.4.12 Bore In, Dwell, Bore Out (manual out)

Command form:

Format1/2:

G89 X_ Y_ Z_ R_ P_ F_ K_ ;

Format3:

G88 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ or Y_: coordinates of holes (absolute/increment)

Z_:

G91: the distance from the bottom of the hole to point Z (directional)

G90: program position of point Z

R_:

G91: the distance from initial level to R point level (directional)

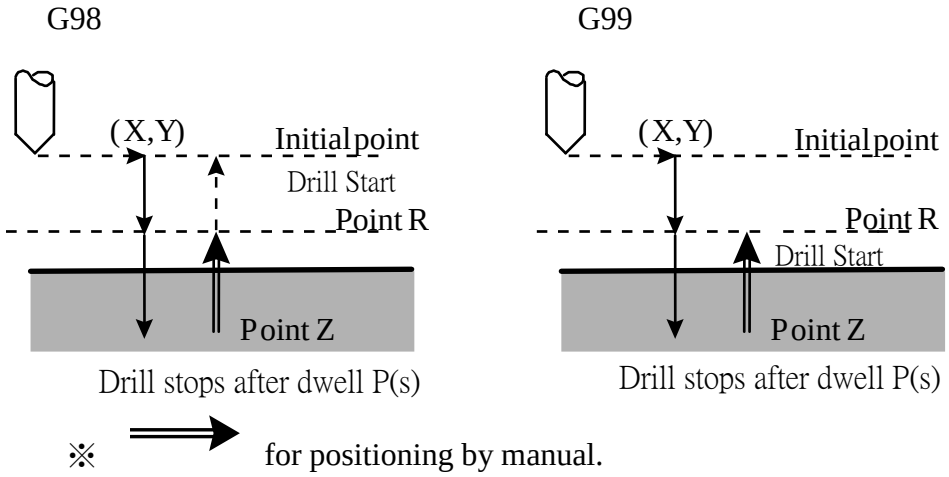
G90: program position of point R

P : Dwell time (s)

F_: feedrate

K_: number of repetition

X, Y, Z, R can be used in both G90 and G91 mode



※ ⟹ for positioning by manual.

Notes:

1. Before using G87/G88, please use M function to turn the spindle
2. G87/G88 is modal G Code, once it is defined, it will tap in every XY move until G80 is encounter
3. Feedrate and spindle switches are not effective during tapping

Description:

1. use G00 to positioning to specified (X,Y)
2. use G00 to reach specified point R.
3. use G01 to reach point Z the bottom of the hole,
4. make the tool out of workpiece in manual mode and reset
5. use G01 to move to point R
6. use G00 to raise to initial point(G98) or program point R(G99)
7. drill rotate CW.

Condition:

1. If a movement command of any axes (X, Y, Z) is not defined in the block, then the drilling will not be executed
2. G00/G01/G02/G03 cannot be in the line in which G86/G87 exist
3. In Fixed Cycle, cutter radius compensation (G41/G42/G40) will be ignored.

Example:

```
F1000. S500;
G90;
G00 X0. Y0. Z10.; // positioning to initial point
G17;
M03; // start drill to rotate CW
G90 G99;
```

```
//specify point R, point Z and, hole1
G89 X5. Y5. Z-10. R-5. P3.;
X15.; // hole2
Y15.; // hole3
G98 X5.; // hole4, and return to initial point
G80;
M05; // drill stops
M02;
```

4.5 Fixed Subroutines

Fixed Subroutines	Format 1/2	Format 3
Engraving Function	L9201	G292
Bolt Circle	L93NN	G293
Mill Boring Counter-Clockwise	L94NN	G178
Mill Boring Clockwise	L95NN	G178
Rectangular Pocket Clean-out Counter-Clockwise	L9601	G175
Rectangular Pocket Clean-out Clockwise	L9701	G175
Circular Pocket Clean-out Counter-Clockwise	L9801	G177
Circular Pocket Clean-out Clockwise	L9901	G177

4.5.1 Engraving Function

Command form:

Format1/2:

L9201 R0_ R1_ R2_ R3_ R4_ Z_ F_ (_ ;

Format3:

G292 R_ W1_ W2_ W3_ W4_ Z_ F_ //_ ;

R0 : This parameter is used to define the clearance plane (R-plane) for the tool to move above the part. The tool retracts to this plane when moving between characters, changing position to continue the same character, or after the last character is engraved.

R1 : 0 – standard font type, 1 – Times New Roman font type

R2 : This parameter represents the height of characters to be engraved minus the tool diameter.

R3 : This parameter represents the angle at chich the characters are to be engraved.

R4 : This parameter represents the incremental value for serialization.

Z : This parameter defines the final depth of cut for the cycle, it's incremental from R plane.

F : Feedrate

(: Words typed after the ((left parenthesis) will be engraved.

Notes:

Parameter of Format 1/2 and 3:

Format 1/2	Format 3

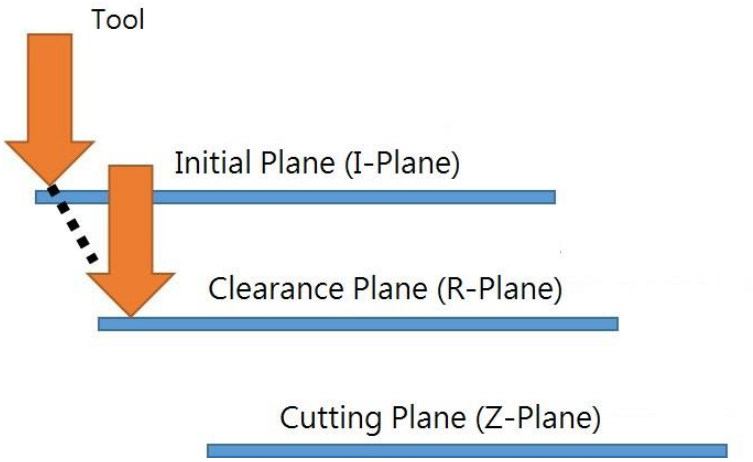
R0	R
R1	W1
R2	W2
R3	W3
R4	W4
Z	Z
F	F
(	//

Description:

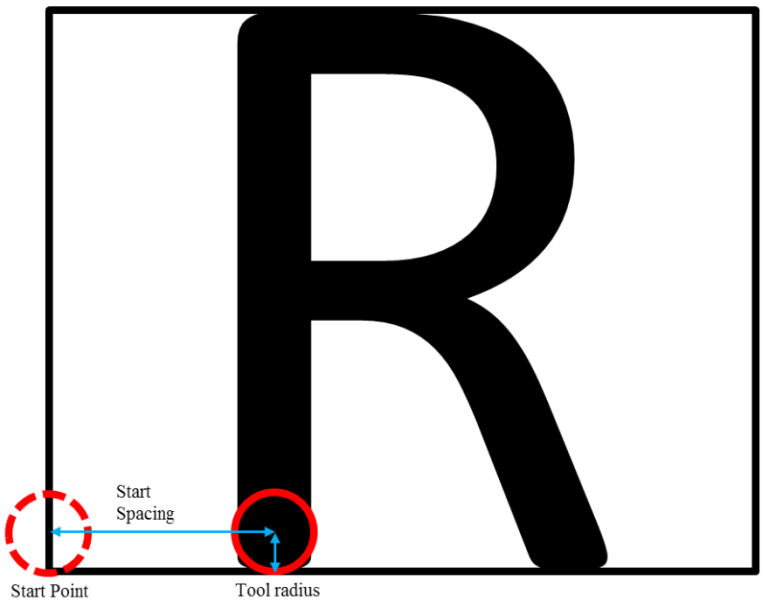
4. The programmer has to define D and H word before using engraving function.
5. The programmer should move the tool to the desired X and Y position before calling this engraving function.
6. Z axis will be moved to R-plane once entered the engraving cycle
7. X and Y axis will be moved to the initial point of first word to be engraved.
8. Z axis interpolate down to the cutting depth defined with Z word
9. Proceed with the word cutting
10. Move Z axis to R-plane between words

Spacing, Positioning, and Tool Path:

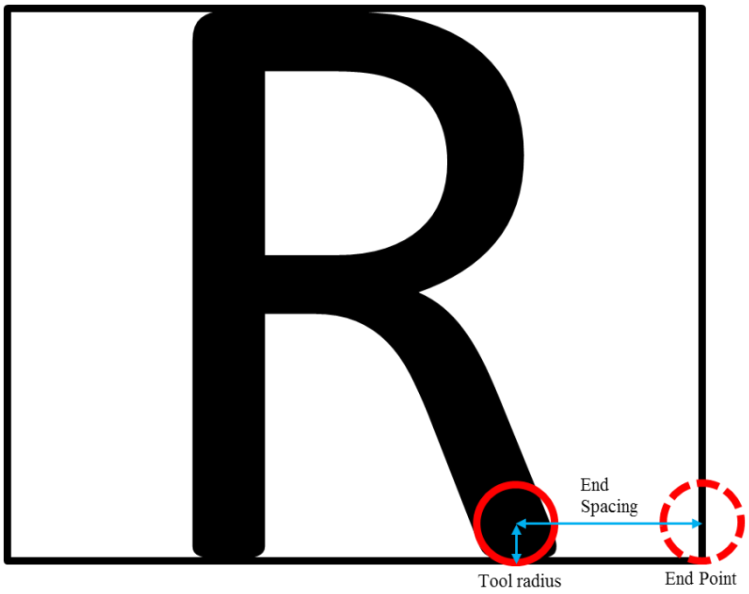
11. Z axis will move to R-plane from I-plane with G00 speed before engrave



12. Actual height = desired letter height – tool diameter
13. Start spacing = actual height*start factor



14. End spacing = actual height*end factor



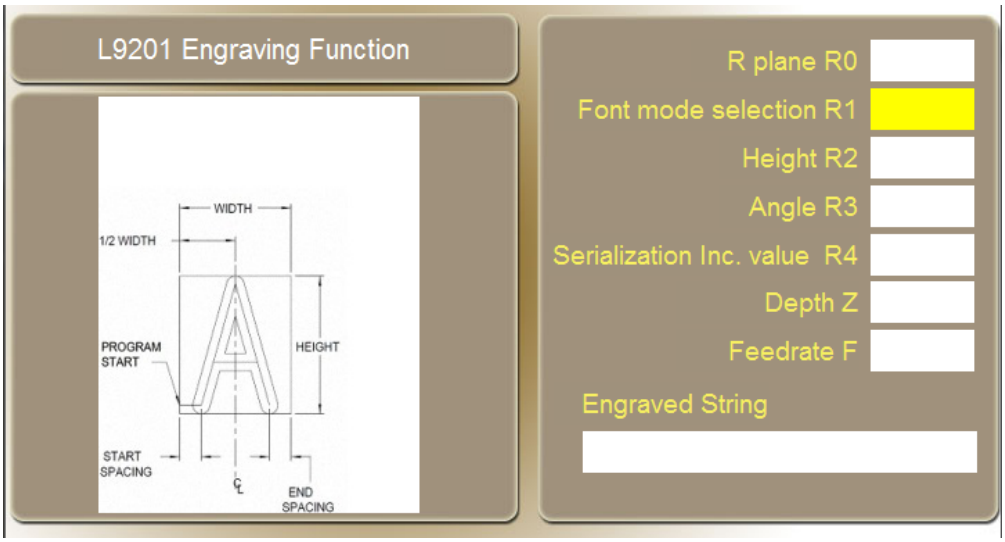
15. Engraving length = actual height*total width factor

Character	Width	Start	End
A	1.0506	.2279	.2279
B	.9455	.2279	.1284
C	.9471	.1837	.2677
D	.9441	.2279	.1927
E	.8853	.2279	.2280
F	.9118	.2279	.2294
G	.9588	.2153	.2282
H	.9706	.2279	.2353
I	.4559	.2279	.2280
J	.95	.2279	.2280
K	1.0249	.2279	.2278
L	.8941	.2279	.2280
M	1.0824	.2279	.2280
N	.9573	.2279	.2279
O	.9647	.1779	.1779
P	.9485	.2279	.2279
Q	.9647	.1779	.1779
R	.9749	.2279	.2278
S	.9853	.2279	.2280
T	.9485	.2279	.2279
U	1.0000	.2279	.2280
V	1.0147	.2279	.2280
W	1.2059	.2279	.2279
X	.9559	.2279	.2280
Y	1.0441	.2279	.2280
Z	.9441	.2279	.2280

Character	Width	Start	End
\	.9559	.2279	.2280
!	.7353	.3676	.3677
#	1.0441	.1926	.1927
$	.9559	.2276	.2280
%	.8676	.2279	.2280
‘	.7353	.3676	.3677
&	.9853	.2345	.1453
(	.5855	.2279	.2279
)	.5855	.2279	.2279
*	.9559	.2279	.2280
	1.1029	.2279	.2280
,	.7353	.2941	.2960
-	1.1029	.2279	.2280
.	.7353	.3676	.3677
/	.9559	.2279	.2280
0	.9647	.1779	.1779
1	.6059	.2279	.2280
2	.8926	.2279	.2271
3	.9632	.2279	.2281
4	1.0779	.2279	.2279
5	.9485	.2279	.2089
6	.9118	.2153	.2267
7	.9691	.2279	.2279
8	.9706	.2271	.2282
9	.9118	.2259	.2123
:	.7353	.3676	.3677
;	.7353	.2941	.2957
“	.7704	.2274	.2274
=	1.1029	.2279	.2280
?	.9853	.2279	.2280
Space	.8823		

Graphic Conversational Input:

Instead of programming the engraving function, Fadal 64 MP control provides an easier way to accomplish the programming, i.e. graphic conversational input.



Use the arrow key on keyboard to move the cursor between input boxes and input the parameters. Pressing **YES** softkey will program the engraving function directly on the editing program.

Condition:

1. The maximum Z depth from R-plane to final is 2.5 inches.
2. Z-plane is not allowed to physically above R-plane.
3. The maximum character height is 2.5 inches.
4. The maximum number of characters (including spaces) that can be engraved in each use of the engraving function is 63.

Example (Format1):

```
N1 O1 (Sample engraving program
N2 M6 T1
N3 (Tool #1 engraving tool .015 center drill
N4 G00 G90 S10000 M3 E1 X0.375 Y-.6175
N5 H1 D1 M8 Z.05 (Diameter 0.015 in offset page
N6 L9201 R0+.05 R1+0 R2+.125 R3+0 Z-.005 F40. (ENGRAVE
N7 M5 M9
N8 G90 G00 H0 Z0
N9 E0 X0 Y0
N10 M2
```

4.5.2 Bolt Circle

Command form:

Format1/2:

L93NN R0_ R1_ R2;

Format3:

G293 R_ W1_ W2_ K_ ;

NN :The number of holes to be drilled. For example, L9304 is for 4 holes.

R0 : This parameter represents the I definition of a circle. This is the **X direction and distance** from the starting position to the center.

R1 : This parameter represents the J definition of a circle. This is the **Y direction and distance** from the starting position to the center.

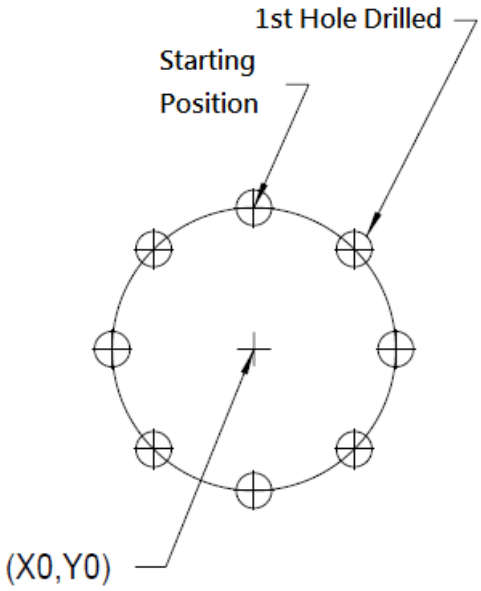
R2 : This parameter represents the angular step between holes. A positive angular step will move CCW around the bolt circle, while a negative angular step will move CW around the bolt circle.

Notes:

Parameter of Format 1/2 and 3:

Format 1/2	Format 3
R0	R
R1	W1
R2	W2
NN	K

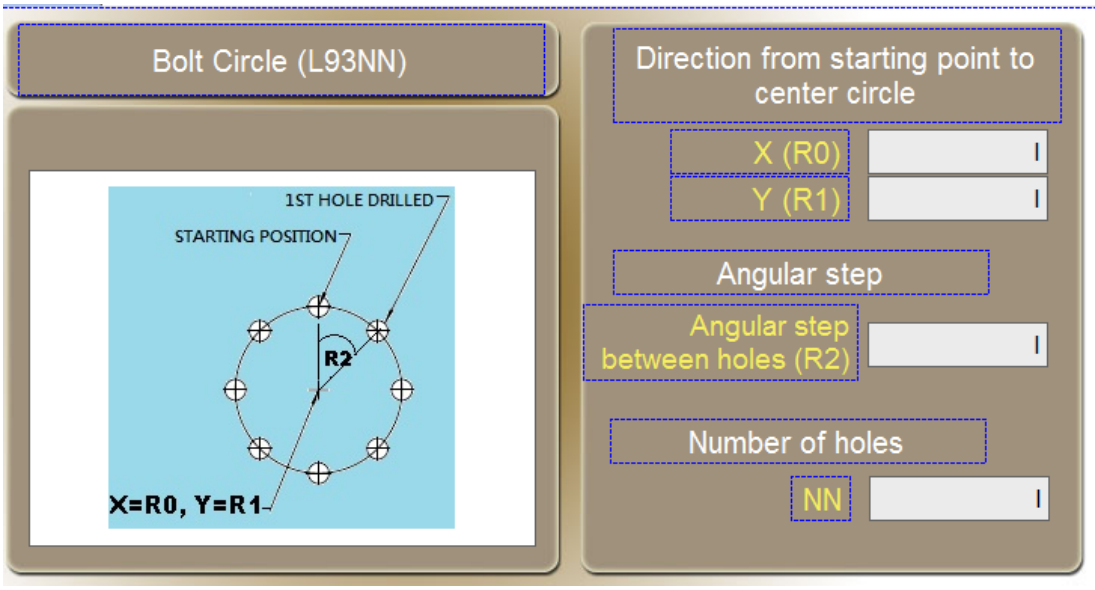
Description:



16. Position the X, Y axes to the starting position and the Z axis to the I-plane.
17. Call for the desired fixed cycle
18. Call for the bolt circle function

Graphic Conversational Input:

Instead of programming the engraving function, Fadal 64 MP control provides an easier way to accomplish the programming, i.e. graphic conversational input.



Use the arrow key on keyboard to move the cursor between input boxes and input the parameters. Pressing **YES** softkey will program the engraving function directly on the editing program.

Condition:

5. R0 and R1 must be input.
6. NN is ranging from 01 to 99.
7. H and D word have to be defined before calling this function.
8. D cannot be 0.

Example (Format1):

N1 O1 (BOLT HOLE EXAMPLE
N2 G0 G90 S2000 M3 X0 Y1.5 (Position to starting point
N3 H1 M7 Z.1
N4 G81 G99 R0+.1 Z-1.0 F10. (Set up fixed cycle
N5 L9308 R0+0 R1-1.5 R2-45. (Call for Bolt Circle
N6 M5 M9
N7 G80

4.5.3 Mill Boring Counter-Clockwise

Command form:

Format1/2:

L94NN R0_ R1_;

Format3:

G294 R_ W1_ K_ ;

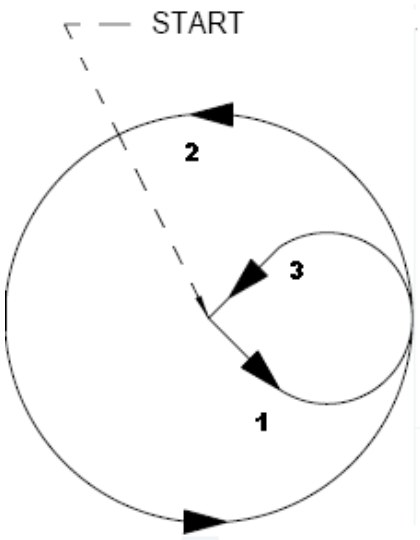
NN :The number of holes to be drilled. For example, L9304 is for 4 holes.
R0 : Feedrate
R1 : This parameter represents the diameter of hole to be bored.

Notes:

Parameter of Format 1/2 and 3:

Format 1/2	Format 3
R0	R
R1	W1
NN	K

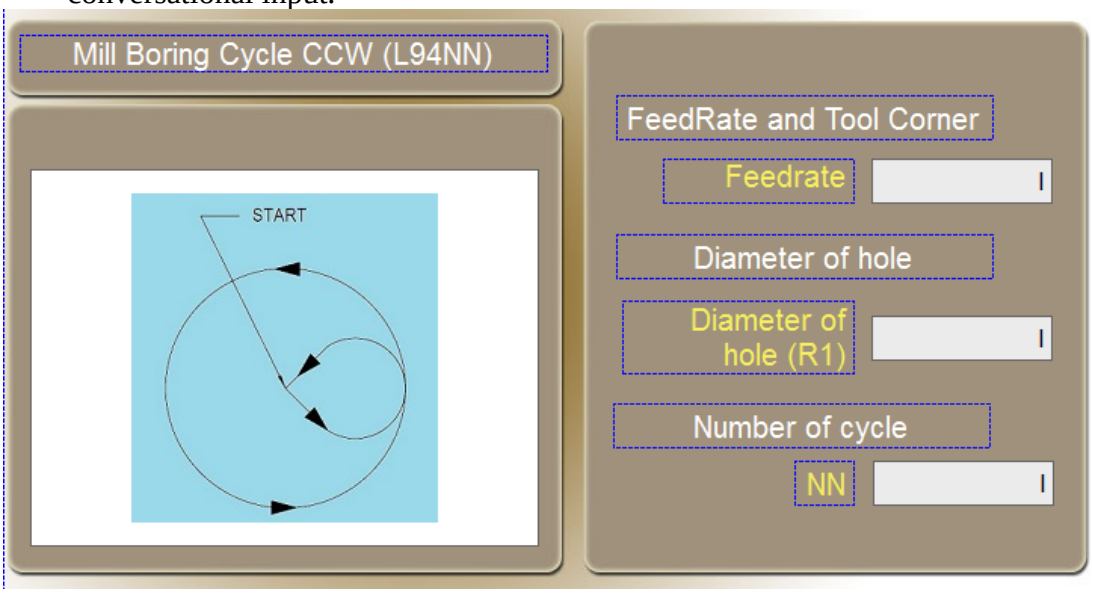
Description:



19. Position the X, Y axes to the center and the Z axis to the finished depth.
20. Call for the Boring function

Graphic Conversational Input:

Instead of programming the engraving function, Fadal 64 MP control provides an easier way to accomplish the programming, i.e. graphic conversational input.



Use the arrow key on keyboard to move the cursor between input boxes and input the parameters. Pressing **YES** softkey will program the engraving

function directly on the editing program.

Condition:
9. R0 and R1 must be input.
10. NN is ranging from 01 to 99.
11. H and D word have to be defined before calling this function.
12. D cannot be 0.

Example (Format1):

*N1 O1 (*Mill Boring CCW Example
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y-1.0
N4 H1 D1 M7 Z.1
N5 G1 F50. Z-1.
N6 L9401 R0+100. R1+1.5
N7 M5 M9
N8 G0 H0 G90 Z0

4.5.4 Mill Boring Clockwise

Command form:

Format1/2:

L95NN R0_ R1_;

Format3:

G295 R_ W1_ K_ ;

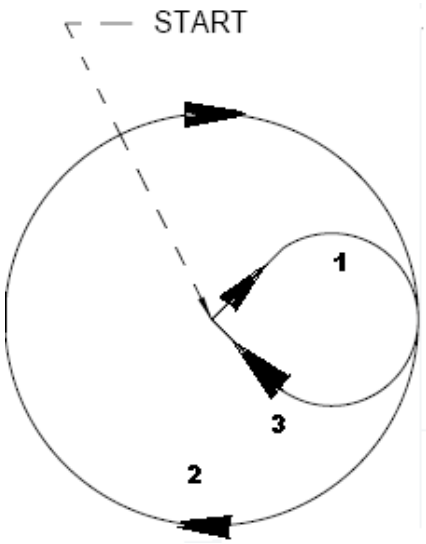
NN :The number of holes to be drilled. For example, L9304 is for 4 holes.
R0 : Feedrate
R1 : This parameter represents the diameter of hole to be bored.

Notes:

Parameter of Format 1/2 and 3:

Format 1/2	Format 3
R0	R
R1	W1
NN	K

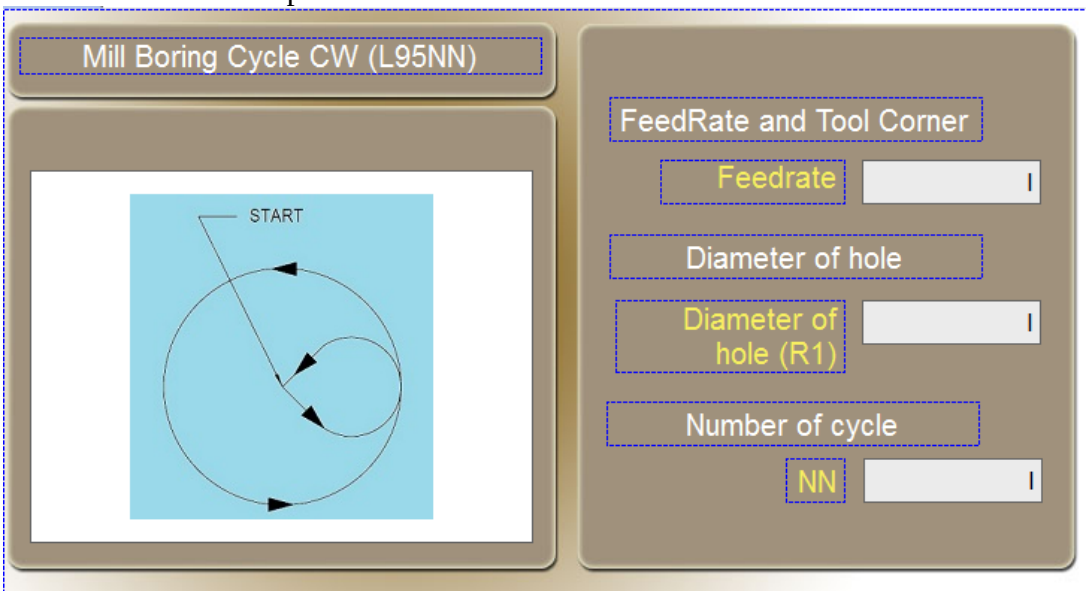
Description:



21. Position the X, Y axes to the center and the Z axis to the finished depth.
22. Call for the Boring function

Graphic Conversational Input:

Instead of programming the engraving function, Fadal 64 MP control provides an easier way to accomplish the programming, i.e. graphic conversational input.



Use the arrow key on keyboard to move the cursor between input boxes and input the parameters. Pressing **YES** softkey will program the engraving function directly on the editing program.

Condition:

13. R0 and R1 must be input.
14. NN is ranging from 01 to 99.
15. H and D word have to be defined before calling this function.

16. D cannot be 0.

Example (Format1):

*N1 O1 (*Mill Boring CW Example
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y-1.0
N4 H1 D1 M7 Z.1
N5 G1 F50. Z-1.
N6 L9501 R0+100. R1+1.5
N7 M5 M9
N8 G0 H0 G90 Z0

4.5.5 Rectangular Pocket Clean-out Counter-Clockwise

Command form:

Format1/2:

L9601 R0_ R1_ R2_ R3_ R4_ R5_ R6_ ;

Format3:

G296 R_ W1_ W2_ W3_ W4_ W5_ W6_ ;

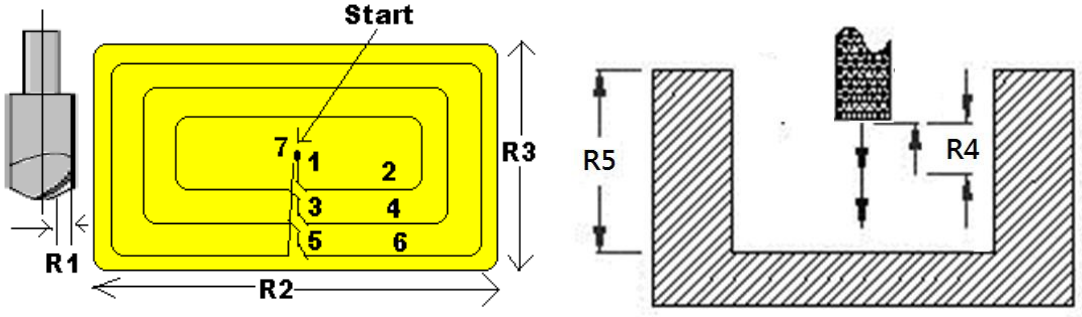
R0 : Feedrate
R1 : Tool Corner Radius
R2 : This parameter represents the overall X dimension.
R3 : This parameter represents the overall Y dimension.
R4 : This parameter represents the Z axis increment value of each cycle
R5 : This parameter represents the overall Z cutting depth
R6 : Z axis feedrate

Notes:

Parameter of Format 1/2 and 3:

Format 1/2	Format 3
R0	R
R1	W1
R2	W2
R3	W3
R4	W4
R5	W5
R6	W6

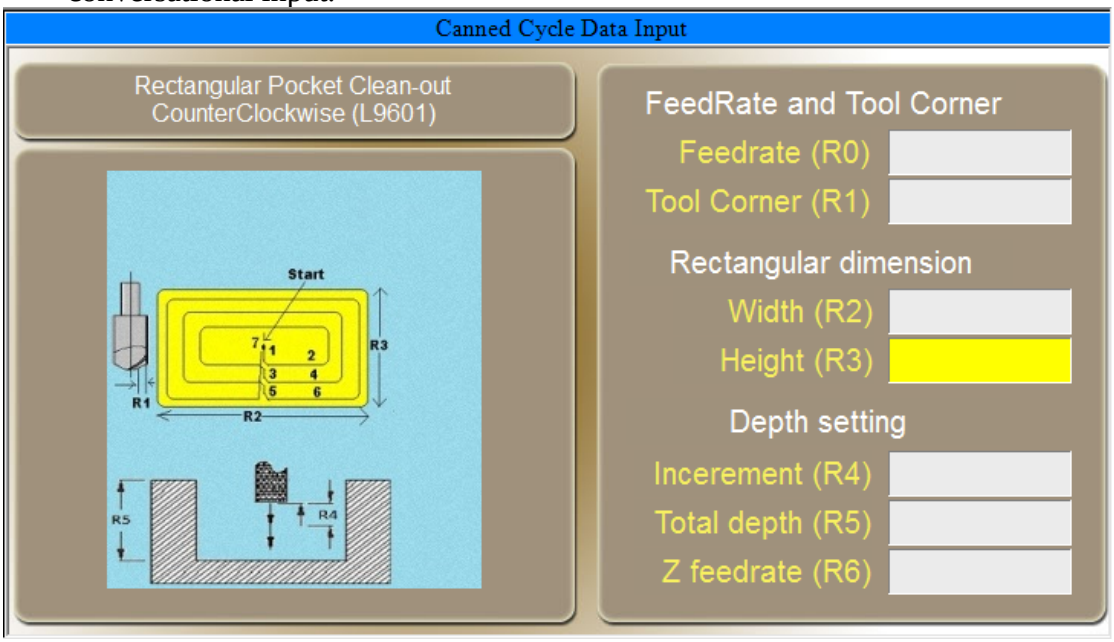
Description:



1. Position the X, Y axes to the center
2. Position the Z axis to the finished depth, if R4~R6 are not input.
3. Call for the rectangular pocket clean-out function

Graphic Conversational Input:

Instead of programming the engraving function, Fadal 64 MP control provides an easier way to accomplish the programming, i.e. graphic conversational input.



Use the arrow key on keyboard to move the cursor between input boxes and input the parameters. Pressing **YES** softkey will program the engraving function directly on the editing program.

Condition:

1. R0, R1, R2, and R3 must be input.
2. H and D word have to be defined before calling this function.
3. D cannot be 0.

Example (Format1):

*N1 O1 (*Rectangular Clean-out CCW example
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y1.0
N4 H1 D1 M7 Z.1
N5 G1 F10. Z-1.
N6 L9601 R0+10. R1+.01 R2+3.25 R3+1.75
N7 M5 M9
N6 G0 G49 G90 Z0

4.5.6 Rectangular Pocket Clean-out Clockwise

Command form:

Format1/2:

L9701 R0_ R1_ R2_ R3_ R4_ R5_ R6_ ;

Format3:

G297 R_ W1_ W2_ W3_ W4_ W5_ W6_ ;

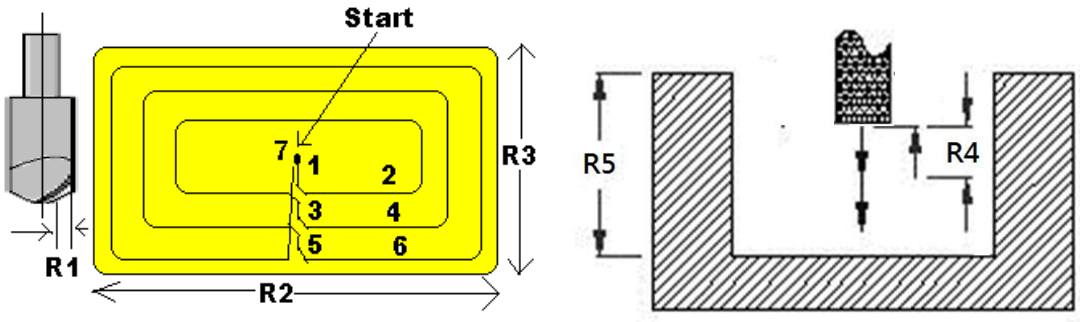
R0 : Feedrate
R1 : Tool Corner Radius
R2 : This parameter represents the overall X dimension.
R3 : This parameter represents the overall Y dimension.
R4 : This parameter represents the Z axis increment value of each cycle
R5 : This parameter represents the overall Z cutting depth
R6 : Z axis feedrate

Notes:

Parameter of Format 1/2 and 3:

Format 1/2	Format 3
R0	R
R1	W1
R2	W2
R3	W3
R4	W4
R5	W5
R6	W6

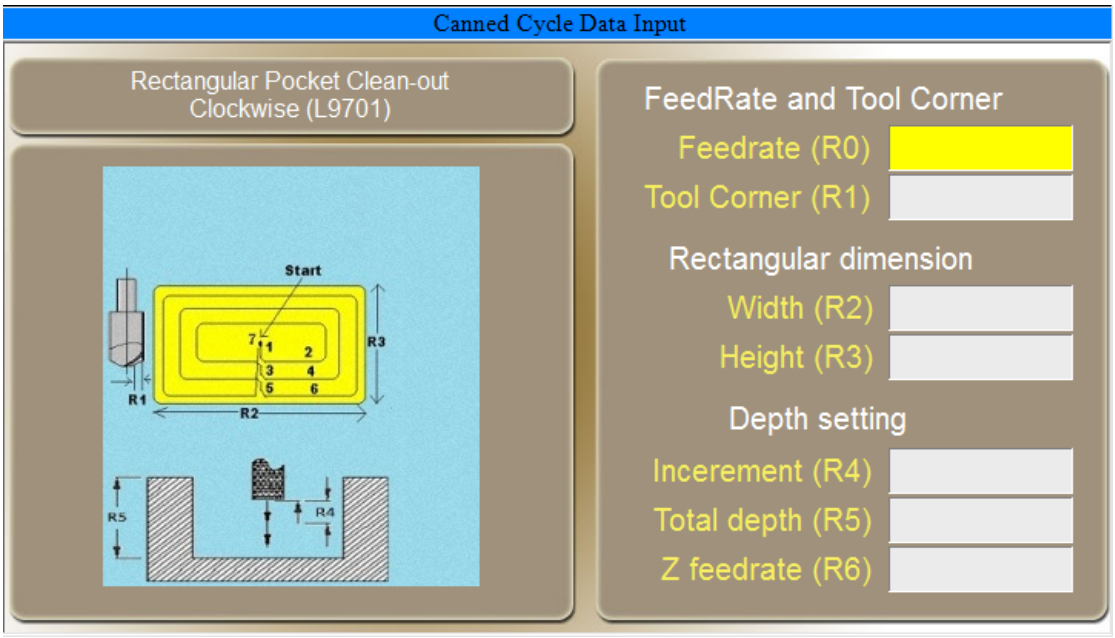
Description:



1. Position the X, Y axes to the center
2. Position the Z axis to the finished depth, if R4~R6 are not input.
3. Call for the rectangular pocket clean-out function

Graphic Conversational Input:

Instead of programming the engraving function, Fadal 64 MP control provides an easier way to accomplish the programming, i.e. graphic conversational input.



Use the arrow key on keyboard to move the cursor between input boxes and input the parameters. Pressing **YES** softkey will program the engraving function directly on the editing program.

Condition:

4. R0, R1, R2, and R3 must be input.
5. H and D word have to be defined before calling this function.
6. D cannot be 0.

Example (Format1):

*N1 O1 (*Rectangular Clean-out CW example
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y1.0
N4 H1 D1 M7 Z.1
N5 G1 F10. Z-1.
N6 L9701 R0+10. R1+.01 R2+3.25 R3+1.75
N7 M5 M9
N6 G0 G49 G90 Z0

4.5.7 Circular Pocket Clean-out Counter-Clockwise

Command form:

Format1/2:

L9801 R0_ R1_ R2_ R3_ R4_ R5_ ;

Format3:

G298 R_ W1_ W2_ W3_ W4_ W5_ ;

R0 : Feedrate
R1 : Tool Corner Radius
R2 : This parameter represents the diameter of the pocket.
R3 : This parameter represents the Z axis increment value of each cycle

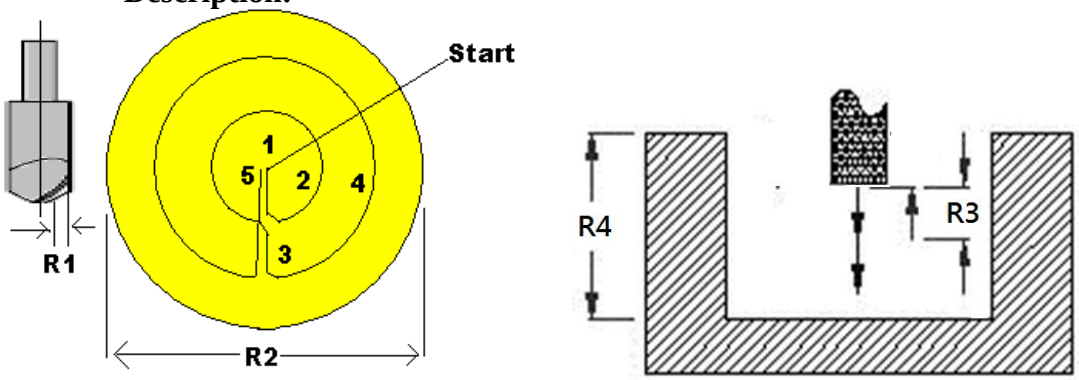
R4 : This parameter represents the overall Z cutting depth
R5 : Z axis feedrate

Notes:

Parameter of Format 1/2 and 3:

Format 1/2	Format 3
R0	R
R1	W1
R2	W2
R3	W3
R4	W4
R5	W5

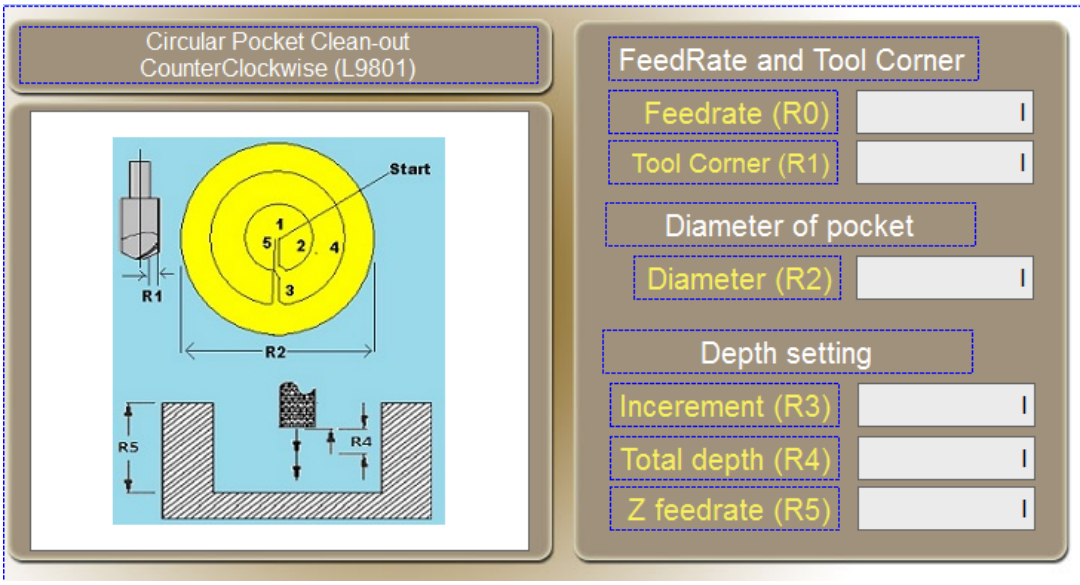
Description:



4. Position the X, Y axes to the center
5. Position the Z axis to the finished depth, if R3~R5 are not input.
6. Call for the circular pocket clean-out function

Graphic Conversational Input:

Instead of programming the engraving function, Fadal 64 MP control provides an easier way to accomplish the programming, i.e. graphic conversational input.



Use the arrow key on keyboard to move the cursor between input boxes and input the parameters. Pressing **YES** softkey will program the engraving function directly on the editing program.

Condition:

7. R0, R1, and R2 must be input.
8. H and D word have to be defined before calling this function.
9. D cannot be 0.

Example (Format1):

N1 O1 (CircularPocket Clean-out Counterclockwise example
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y-1.0
N4 H1 D1 M7 Z.1
N5 G1 F10. Z-1.
N6 L9801 R0+10. R1+.01 R2+1.75
N7 M5 M9
N8 G0 G49 G90 Z0

4.5.8 Circular Pocket Clean-out Clockwise

Command form:

Format1/2:

L9901 R0_ R1_ R2_ R3_ R4_ R5_ ;

Format3:

G299 R_ W1_ W2_ W3_ W4_ W5_ ;

R0 : Feedrate
R1 : Tool Corner Radius
R2 : This parameter represents the diameter of the pocket.
R3 : This parameter represents the Z axis increment value of each cycle
R4 : This parameter represents the overall Z cutting depth

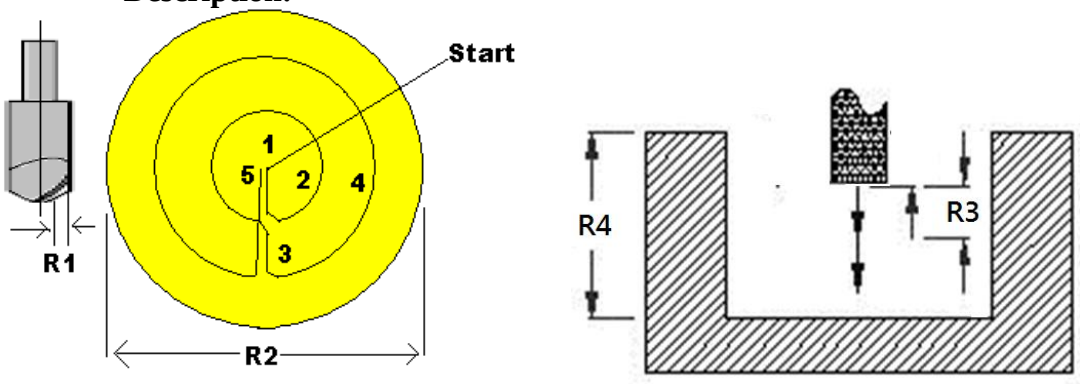
R5 : Z axis feedrate

Notes:

Parameter of Format 1/2 and 3:

Format 1/2	Format 3
R0	R
R1	W1
R2	W2
R3	W3
R4	W4
R5	W5

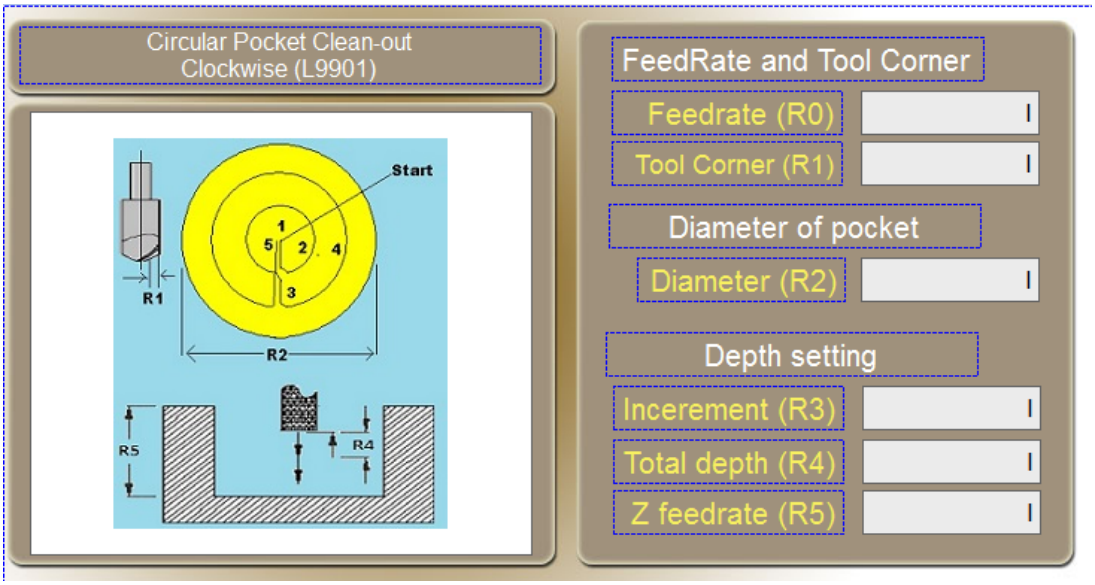
Description:



7. Position the X, Y axes to the center
8. Position the Z axis to the finished depth, if R3~R5 are not input.
9. Call for the circular pocket clean-out function

Graphic Conversational Input:

Instead of programming the engraving function, Fadal 64 MP control provides an easier way to accomplish the programming, i.e. graphic conversational input.



Use the arrow key on keyboard to move the cursor between input boxes and input the parameters. Pressing **YES** softkey will program the engraving function directly on the editing program.

Condition:

10. R0, R1, and R2 must be input.
11. H and D word have to be defined before calling this function.
12. D cannot be 0.

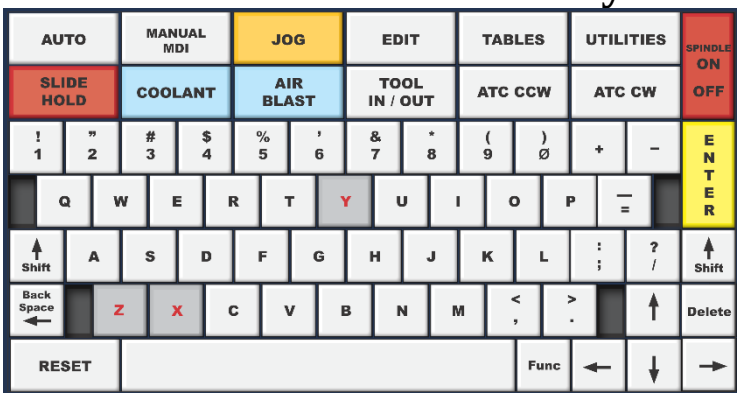
Example (Format1):

N1 O1 (CircularPocket Clean-out Clockwise example
N2 M6 T1
N3 G0 G90 S2000 M3 X1.0 Y-1.0
N4 H1 D1 M7 Z.1
N5 G1 F10. Z-1.
N6 L9901 R0+10. R1+.01 R2+1.75
N7 M5 M9
N8 G0 G49 G90 Z0

5 Miscellaneous Operator Switches and Buttons

This chapter will cover all the buttons and switches found in both the Fadal keyboard and the operator panel.

5.1 Special Buttons on the Fadal Keyboard



There will be 13 special buttons on the Fadal keyboard, which are:

1. AUTO
2. MANUAL MDI
3. JOG
4. EDIT
5. TABLES
6. UTILITIES
7. SLIDE HOLD
8. COOLANT
9. AIR BLAST
10. TOOL IN/OUT
11. ATC CCW
12. ATC CW
13. SPINDLE ON/OFF

5.1.1 AUTO

Pushing this *AUTO* button will switch to auto mode (see section 2.2) and bring up the corresponding screen.

5.1.2 MANUAL MDI

Pushing this *MANUAL MDI* button while the machine is not in running state will switch to MDI mode (see section 2.3) and bring up the corresponding screen.

5.1.3 JOG

Pushing this *JOG* button while the machine is not in running state will switch to JOG mode (see section 2.4) and bring up the corresponding screen.

Pushing this *JOG* button in JOG mode will cause the moving of the machine according to the current active axis and current active direction.

5.1.4 EDIT

Pushing this *EDIT* button while the machine is not in running state will switch to Edit mode (see section 2.5) and bring up the corresponding screen. Pushing this *EDIT* button while the machine is running a program in Auto mode will enable **background edit** function.

5.1.5 TABLES

Pushing this *TABLES* button while the machine is not in running state will switch to Tables mode (see section 2.6) and bring up the corresponding screen.

5.1.6 UTILITIES

Pushing this *UTILITIES* button while the machine is not in running state will switch to Utilities mode (see section 2.7) and bring up the corresponding screen.

5.1.7 SLIDE HOLD

Pushing this *SLIDE HOLD* button while the machine is in running state will cause all axes to decelerate to zero and put the machine in "Slide Hold" status. This *SLIDE HOLD* button is active either in Auto or MDI mode.

5.1.8 COOLANT

Pushing this *COOLANT* button will toggle the flood coolant function on and off. This button is active in all modes.

In Auto mode the flood coolant is typically turned on with a M code (M7 or M8) but can also be turned on and off using this button.

5.1.9 AIR BLAST

Pushing this *AIR BLAST* button will toggle the mist function on and off. This button is active in all modes.

In Auto mode the mist function is typically turned on with a M code (M7 or M8) but can also be turned on and off using this button.

5.1.10 TOOL IN/OUT

Pushing this *TOOL IN/OUT* button while the spindle is not turning will enable/disable the tool clamp function.

5.1.11 ATC CW

Pushing this *ATC CW* button will rotate the turret and allow the operator to manually load tools into the turret. The *ATC CW* will rotate the tool turret or magazine in a clockwise direction.

The tool buckets or pocket count up. Pressing this button once will increment the turret one bucket. Pressing and holding the button will continue

to rotate the turret until the button is released.

5.1.12 ATC CCW

Pushing this *ATC CCW* button will rotate the turret and allow the operator to manually load tools into the turret. The *ATC CCW* will rotate the tool turret or magazine in a counter-clockwise direction.

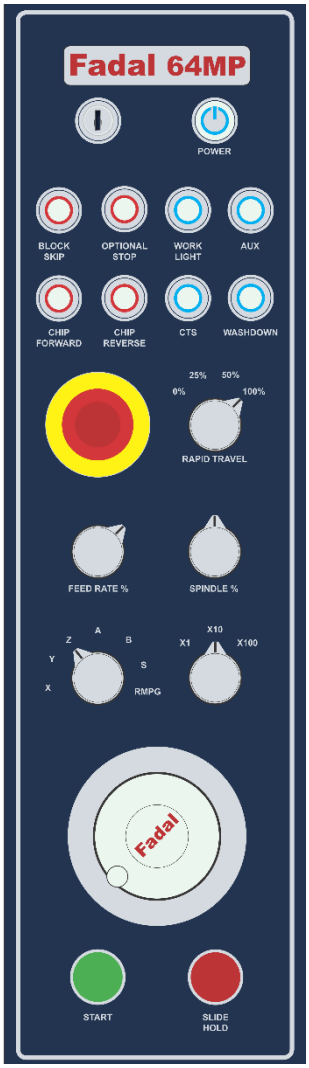
The tool buckets or pocket count down. Pressing this button once will decrement the turret one bucket. Pressing and holding the button will continue to rotate the turret until the button is released.

5.1.13 SPINDLE ON/OFF

Pushing the Fn + *SPINDLE ON/OFF* button will start the spindle rotation when in Jog mode or in Auto or MDI mode if Slide Hold is active. Pushing only the *SPINDLE ON/OFF* button will stop the spindle rotation.

In manual mode, the spindle speed is set from the last programmed speed and direction in Auto or MDI mode. If no speed was programmed in Auto or MDI prior to starting the spindle with this *SPINDLE ON/OFF* button, the spindle will rotate at the minimum RPM.

5.2 Operator Panel



There will be 8 function buttons, 5 switches, 3 push buttons, 1 Handwheel, 1 keylock switch, and 1 power button on the operator panel.

5.2.1 Function Buttons

5.2.1.1 Block Skip

The “Block Skip” button will toggle the block skip feature on and off. This block skip function allows the operator to selectively ignore blocks in the NC part program if the blocks are preceded with a forward slash symbol (/). Pressing the “Block Skip” button will inform the CNC to ignore any NC block that is preceded with a slash code (/).

Example
N110 G0 X4.5 Y6
/N120 M0 (PERFORM A MANUAL TOOL CHANGE TO TOOL 6)
N130 M3 S1000
N140 …

In the above example block N120 will be skipped if the Block Skip function is active and the light is on. If the light is off and thus the feature disabled, block N120 will be executed.

5.2.1.2 Optional Stop

The "Optional Stop" button will toggle the optional stop feature on and off. This optional stop function allows the operator to use the conditional stop code M01. Pressing the "Optional Stop" button will inform the CNC to stop iteration and wait for operator intervention when a M01 code is read and active. When the M01 block is active and the Optional Stop light is on, the CNC will decelerate the axes and spindle to zero and put the system into a Slide Hold state. If the light is off and thus the feature disabled the M01 block will be ignored.

5.2.1.3 Work Light

The "Work Light" button will toggle the work light on and off.

5.2.1.4 Aux

The "Aux" button preserved for further function

5.2.1.5 Chip Forward

The "Chip Forward" button will toggle the chip forward feature on and off. This button will not be active if the "Chip Reverse" button holding on.

5.2.1.6 Chip Reverse

The "Chip Reverse" button will reverse the chip motor. This "Chip Reverse" button will only enable the function when the button is holding on. Once the operator release the button, the chip motor will not reverse.

When the chip motor is moving forward, pressing this "Chip Reverse" button will reverse the motor, once the button is release the chip motor will resume moving forward.

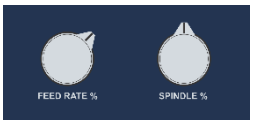
5.2.1.7 CTS

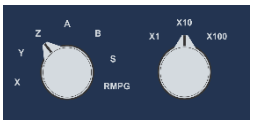
The "CTS" button will toggle the coolant through spindle feature on and off.

5.2.1.8 Washdown

The "Washdown" button will toggle the washdown feature on and off.

5.2.2 Switches





5.2.2.1 Rapid Travel

The “Rapid Travel” switch will change the override of G00 move.

5.2.2.2 Feedrate

The “Feedrate” switch will change the override of G01 move.

5.2.2.3 Spindle

The “Spindle” switch will change the override of spindle RPM.

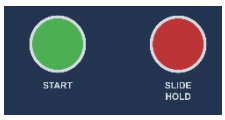
5.2.2.4 MPG Axis Selector

The “MPG Axis Selector” switch will define the current active axis of MPG/JOG.

5.2.2.5 MPG Step Size Selector

The “MPG Step Size Selector” switch will define the current active step size of MPG/JOG.

5.2.3 Push Buttons



5.2.3.1 Emergency Stop

The “Emergency Stop” push button will stop everything of the machine.

5.2.3.2 Cycle Start

The “Start” push button refers to “Cycle Strat”. This button will initiate the start of execution of a program when the system is in Auto or MDI mode and the machine is in “Waiting” state.

5.2.3.3 Slide Hold

The “Slide Hold” will cause all axes to decelerate to zero and put the system into a Slide Hold state. The Hold button is active in either Auto or MDI mode.

5.2.4 Keylock Switch

The Keylock Switch will enable/disable the edit function, i.e. when edit is locked, no program edit can be executed.

Addendum

Below listed all the changes to NC programming of the original Fadal control and the Fadal 64 MP:

1. Command is not supported in Fadal 64 MP
2. The “L” word in fixed cycles should be “K” word instead
3. Only rigid tapping in Fadal 64 MP, i.e. G74 and G84. No G74.1 or G84.1 G codes
4. G73/G83 with I J K function is not supported
5. G75 is not supported

REVISIONS

No	Record	Date	Aurthor	Version
01	First Edition	2014/07/17	Otis Siah	V0.0
02	Modified partial figures	2014/08/08	Otis Siah	V0.1
03	Modified all figures and reviewed chp5	2014/09/18	Otis Siah	V0.2
04	Add chp3.8	2014/09/19	Otis Siah	V0.3
05	Add chp 3.11	2015/03/20	Otis Siah	V0.4
06	Add M19 description Add chp 2.5.5 Update Figure2-18, 2-29, 2-35, 2-41, table 2.82	2015/03/26	Benson Yu	V0.5
07	Add M40/M41/M42/M43 description in chapter 4.2 Add M62/M63/M64/M65 description in chapter 4.2 Fixed the figures for L9601 in chapter 4.4.5 and L9701 in chapter 4.4.6 Add Probe Function in chapter 2.7.4 Modified the description for tool sort in chapter 2.6.1.7 Add Set Load Position softkey description in chapter 2.6.1.7	2015/06/23	Benson Yu	V0.6
08	Add M12, M36~M39	2015/06/29	Otis Siah	V0.7
09	Update G codes list and M codes list (G05.1, G74.1, G84.1)	2015/07/17	Otis Siah	V0.8
10	2.7.4 Probe Function updated Flat Cam description in 4.1.12 added S.1/S.2 description in 4.3 added Modified the description for serialization in 4.5.1 Modified the description for recovery mode in 2.7.3	2016/01/06	Benson Yu	V0.9
11	Description 2.6.2.4 Set M. Coord added. Specified that Z index for Engraving function is incremental from R plane in 4.5.1. Description for M68/M69 added.	2017/01/28	Benson Yu	V1.0