Dynamic

Gang Tool Lathe

with

Fagor 8055T control

Operation & Programming Manual

Table of Contents

Written By **R.W Turney**

Table of Contents

The GTS27 & FX (Gang Tool, 27mm through spindle capacity) is a complete, state of the art, high speed, high accuracy turning system that incorporates the production rates of an automatic screw machine with the versatility of a lathe, while utilizing the flexibility of a modern industrial hardened, CNC controller.

This machine tool system, like all machine tool systems, is made up of a marriage of a mechanical and an electronic system. In the GTS27, the components in both systems were chosen for their quality, durability and accuracy, not their cost.

Spindle Configurations

The GTS27, in its Standard Configuration, provides all the necessary options and accessories required for a high rate production turning system. There are, however, two configuration options that increase the specialized application efficiency of the system without compromising the standard turning efficiency. The two optional configurations are: Indexing Spindle, which provides a position and hold capability for the spindle with 1^{O} positioning at 0.08^{O} accuracy and repeatability, Full "C" Axis capability with the Spindle being a fully interpolated third axis.

Guide-way Configuration

The mechanical system is based on NSK 25mm, super precision, four circuit linear bearings, NSK Angle Load Thrust Bearing assemblies supporting ultra accurate five pitch, four circuit ballscrews. Direct coupled to high torque, rare earth servo motors. The entire assembly is covered and protected from direct contact with either chips or coolant by a combination of ridged and moving covers. Each guide way is equipped with a pre-lubricated system.

Servo and motor Configuration

The electrical / electronics systems begin with the high speed, high torque, rare earth servo motors driven by high powered, pulse width modulated servo power amplifiers which provide axis rapid speeds of 800 inches per minute and peak thrust in excess of 900 pounds. The servo motors have the tachometer and 1000 line rotary encoders incorporated in the servo motor assembly which completes the axis positioning system feedback to the CNC Turning controller. This combination provides 0.00005" inch resolution for positioning accuracy of +/- 0.0001 inch and repeatability of +/-0.00005" inch.

Spindle motor Configuration

The spindle drive inverter is a variable speed, constant 5 hp, 3 phase AC powered unit in the standard Lathe, and an AC flux vector technology drive inverter in the Indexing Spindle and "C" Axis configurations. All lathe configurations have 6000 rpm capability.

FX configured machines have either a 3C or 5C sub spindle capable of full synchronization with the main spindle. And a maximum RPM of 4500RPM

All GTS27 & FX lathes have a CNC controller, automatic collet closers, high volume coolant pumps, electronic handwheels, full chip enclosures, internal waterproof work light and extra M Function outputs as standard equipment.

Standard Features

- Fagor 8055 T-A controller (GTS 27)
- Fagor 8055 T-B controller (GTS-FX)
- 5C collet nose
- Air powered drawbar actuator
- Work light
- Stainless steel telescoping way covers
- Coolant system (15 gallon capacity)
- Full chip and coolant enclosures
- Leveling bolts and plates
- Leveling Pads
- Operation and maintenance manuals
- Parts list and electronic drawings
- One year machine warranty
- One year control warranty

Optional Features

- Positioning spindle (1^{O} increment)
- Full C axis spindle (Allows contouring of shapes with live tools)
- Parts catcher
- Parts conveyor
- Sub spindle (Standard FX)
- Overhead cutoff slide (Y axis) (Standard FX)
- Back working tool slide (Standard FX)
- Live tools (NSK air or electric)
- Foot Pedal
- Bar feeder interface
- Chip conveyor
- Auto lubrication
- Dead length collet nose
- Threaded end collet nose

Machine specifications

CAPACITY	GTS27	
Maximum Bar Capacity	1.0625”	(27.0mm)
Maximum Turning Length	7.5"	(190.5mm)
Maximum Turning Diameter	10.0"	(254.0mm)
Maximum Swing over ways	13.0"	(330.2mm)
MAIN HEADSTOCK		
Spindle Nose	5C(bolt on adaptor)	
Hole Through Spindle	1.25"	(31.75mm)
Max. Chuck Diameter	5"	(127.0mm)
Spindle Speed	60-6000 rpm	
Spindle Motor	5 hp high torque	(3.7kw)
SUB HEADSTOCK 3C		
Spindle Nose	3C (bolt on adaptor)	
Hole Through Spindle	0.5"	(12.7mm)
Max. Chuck Diameter	3"	(75.0mm)
Spindle Speed	45-4500 rpm	
Spindle Motor	1.5 hp high torque	
SUB HEADSTOCK 5C		
Spindle Nose	5C (bolt on adaptor)	
Hole Through Spindle	1.125"	(27.0mm)
Max. Chuck Diameter	5"	(127.0mm)
Spindle Speed	45-4500 rpm	
Spindle Motor	1.5 hp high torque	
X AXIS		
Travel	15"	(381.0mm)
Rapid Traverse Rate	800"/ min.	(20M/min)
Max. programmable feedrate	400”/ min.	(10M/min)
AC Servo Motor	900”/ lbs.	
Y AXIS (overhead cutoff)		
Travel	1.25"	(31.75mm)
Rapid Traverse Rate	400"/ min.	(10M/min)
Max. programmable feedrate	200”/ min.	(5M/min)
AC Servo Motor	600”/ lbs.	
Z-AXIS		
Travel	7.5"	(190.5mm)
Rapid Traverse Rate	800”/ min.	(20M/min)
Max. programmable feedrate	400”/ min.	(10M/min)
AC Servo Motor	1200”/ lbs.	
MACHINE DIMENSIONS		
Height	70"	(1778mm)
Width	51"	(1296mm)
Depth	53”	(1347mm)
Weight	5650lb (5780lb FX)	

Section 1

Safety

Safety Instructions

- Please do not operate your machine until you have studied the machine and have read and understand this manual.
- Always close and lock the electrical cabinets prior to operating the machine, and only allow qualified electrical technicians to perform service on these electrical connections.
- The high voltage power supply to your machine can **KILL** or severely injure you. Always turn off the main breaker before opening cabinets.
- **NEVER** attempt to service or repair electrical circuit's unless you are qualified to work safely on electronic circuit's, or are a CMS authorized Technician or dealer technician.
- Revolving parts and devices inside the working envelope of the machine can cause severe injury if not used properly, use caution when operating this machine.
- Do not remove any covers or panels from the machine, rotating pulleys and slide can cause severe injury.
- Compressed air is connected to this machine for machine use only. Compressed air can KILL or seriously injure you. **NEVER** work on compressed air systems without disconnecting the compressed air source. Prior to work always bleed down any residual air in the systems. Only qualified technicians familiar with the compressed air systems should perform work on these compressed air systems.
- Always wear safety glasses around this equipment.
- Never wear loose clothing or jewelry which may become entangled on moving or rotating components.
- Never extend bar stock past the left end of the spindle. This can cause bar whip at high RPM.and can result in serious injury or death.
- **MODIFICATIONS, ALTERATIONS AND CHANGES TO THE MACHINE REQUIRE THE WRITTEN AUTHORIZATION OF COMPACT MANUFACTURING SYSTEMS.,**

Address any questions to CMS. Attention to the Service Department Manager.

- **UNAUTHORIZED CHANGES MAY LEAD TO HAZARDOUS CONDITIONS.**

DANGER

DANGER - Immediate hazards which WILL result in severe personal injury or death.

WARNING

WARNING - Hazards of unsafe practices which COULD result in severe personal injury or death.

CAUTION

CAUTION - Hazards of unsafe practices which COULD result in minor personal injury or product or property damage.

Any machine that is operated in a careless or improper manner can cause serious injury or death and damage to the machine itself. The warnings in this manual follow accepted industry safety guidelines.

CMS cannot have control of individual company applications or production methods, and is not responsible for injuries or equipment damage when the machine is not used according to safe industry practices.

The safe use and disposal of toxic or hazardous chemicals or materials processed on a machine are the responsibility of the user.

It is the sole responsibility of the end user of the machine to comply with all local, state, and federal safety laws and regulations applicable to their machine and its use.

Section 2

Machine Installation

Machine Packaging

When your GTS27 arrives, it will either be in a large plywood box or on a heavy wood skid and wrapped with plastic stretch wrap. The shipping preparations will depend on the shipping method and distances involved. Actually, the packaging is the same except for the addition of the plywood sides and top when the machine is being transported on an open trailer or if the transportation will take over one day.

If your machine arrives with the plywood protection, remove the drywall screws holding the plywood together, starting with removal of the top, then the sides .

The machine is now visible through the stretch wrap and a careful inspection as the unpacking continues is essential.

If shipping damage is discovered:

1. Stop unpacking and contact CMS or your dealer to determine the appropriate course of action to follow, as it will vary, depending on the type and extent of damage.

2. Document the damage with a clear, complete description of the conditions and the names of people present, and take photographs if possible.

3. If there are multiple incidence of damage, document each incidence individually, as if it were the only one.

4. Remember, shipping damage **is not** covered under the warranty, and is generally an issue that will have to be resolved between you and the carrier, with CMS or dealer support and assistance.

Normal Unpacking of the machine:

- Carefully cut and remove the stretch wrap and inspect the machine again for any signs of damage.
- Referring to your sales order and the packing list, take complete inventory of the items shipped with the machine to be sure the shipment is complete, and includes all hardware, options and documentation listed on the sales order and packing list. Report any shortages to CMS and the dealer as soon as they are identified.
- Note that the machine is secured to the skid by four bolts.
- The machine can be moved by forklift by inserting the forks into the gap under the machine provided by the 2 wood blocks.
- Add the 5 leveling screws prior to placing the machine.

Selecting a Location

When selecting the location for the machine, there are a few considerations that will make the neighborhood more comfortable for all. Probably most important is open access to all areas around the machine so regular inspection of the machine and utilities is available. The area should be dry and have good ventilation, and the floor should be reasonably flat, especially if a bar feed may be a consideration.
The facilities / utilities required for the machine are: 3 phase power of 230 VAC @ 45 amps or 460 VAC @ 25 amps and clean dry compressed air at 80 psig minimum, approximately 10 cfpm, with no air tools.
A solid earth ground to a ground stake or reliable earth ground such as metal plumbing or construction metal in solid ground is required for a frame (chassis) ground to the machine. This earth ground must have a standing potential of less than 1VAC to a qualified earth ground point when in full operation.
NOTE : REMOVE THE BUS GROUND ON THE MACHINE AFTER CONNECTING THE EARTH GROUND

Leveling The Machine

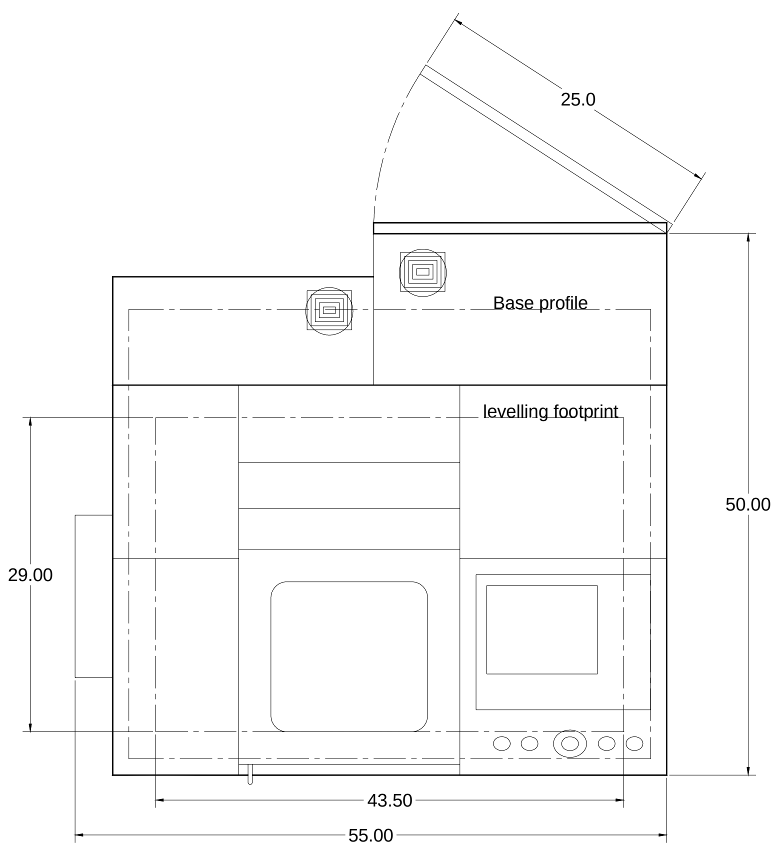
After completing the previous sections, the machine is free of shipping damage, the shipment is complete and the machine has been placed in an appropriate location, with the required facilities and utilities at hand. Assuming these conditions, continue with the set-up and start-up as follows:

- The machine has 6 bolt holes in the base of the machine, insert at least 5 of these bolts, prior to placing the machine on the floor. Normally the 4 corner bolts and a central rear bolt are needed. (the center rear is to support the extra weight of the control / electrical box.)
- Place the flat leveling pads under the bolts, and lower the machine into place taking extreme car not to place hands of arms under the machine.
- If the machine is to be interfaced to a barfeed, check with the barfeed requirements for leveling and level the machine to the same tolerances to assure trouble free interfacing.

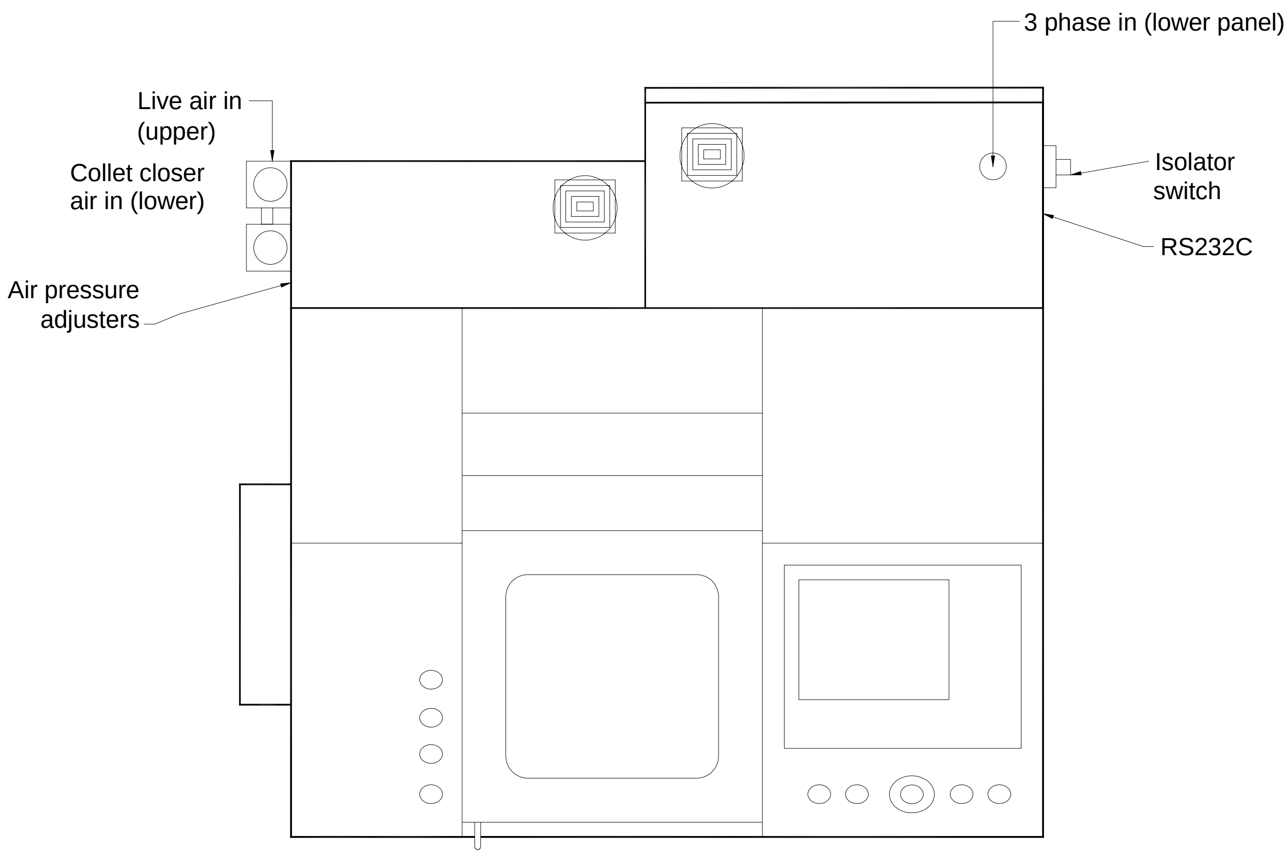
Machine floor layout

Footprint dimensions (fig2.1)

The basic machine hold down bolts are located on a 29"x43.5" footprint. The lower left hole is 2.5"x2.5" from the mainframe corner.



Locating the utility connection locations



Machine top view (fig 2.2)

Air adjusters

The Main collet clamp adjuster controls the grip pressure for the drawbar. The range of adjustment is 20PSI to 110PSI.
The Sub collet clamp adjuster controls the grip pressure for the sub drawbar. The range of adjustment is 20PSI to 75PSI.
The Sub slide adjuster controls the advance pressure for the sub tools slide. The range of adjustment is 20PSI to 100PSI.
The Sub spindle air blow adjuster controls the blow pressure for the sub spindle. The range of adjustment is 10PSI to 110PSI.

Air input live tooling

The air input is for a quick disconnect fitting. This air in is supplied to a water separator & lubricator for NSK live **AIR** tools. The air spindles require paraffin lubrication,
The ASTRO “E” electric spindles require air for cooling only,
Do not use oil for electric spindles.

Air input Collet closer

The air input is for a quick disconnect fitting. This air in, is supplied to a water separator & lubricator for the collet closer.
The collet closer requires oil lubrication. Type DTE 11 or equivalent.

Connecting the Utilities

There are two **utility** connections required for the machine, they are:

- Three phase AC Power of 208 or 230 VAC (or optional 460 VAC)
- A ground stake or solid earth ground from a non-electrical component.

There is one **facility** requirement. It is:

- clean, dry compressed air at 80 to 120 psig, with a volume of about 10 cfpm , average, for a machine with no air tools.

For a machine with air tools, a volume of 25cfpm is recommended.

Connecting the Power and Air

- Locate the safety switch on the right side of the machine and make sure it is turned **OFF**. Following the local electrical codes, construct the three phase power service to the terminal block located in the lower left corner of the electrical cabinet. The three phase power should be routed through a **45 amp, three phase, ganged breaker.**
- After making sure the circuit breaker is **OFF**, connect the three phase power and earth ground conductors to the safety switch fuse lugs and ground buss respectively. Leave the safety switch turned **OFF**.
- Plumb as necessary, in compliance with local codes, to provide clean, dry, compressed air to the separator / filter air input at the quick connect fitting. The air should be between 80 and 120 psig with a volume capacity of 10 cfpm, average. Do **not** supply air at this time.

Testing the Connections

Apply air to the air connection. Check for leaks at this time.

Switch on the main wall breaker to the machine, and switch on the Safety switch on the right of the machine.

Press the Control On button. Wait approx. 20 seconds, the machine control will power up.

If it does not check the power supply for any interruption or cable damage.

Coolant Tank

Premix coolant, or add oil to the 17.5 gallon capacity.

Phase test on a Fagor 8055T

The easiest way to test phase is with the coolant pump rotation. The coolant pump is located in the coolant tank. The top of the pump has a plastic window, the impeller shaft has a dot on the top of the shaft, and the housing has an arrow cast into the top face. When the pump rotates the dot will spin and the direction of the shaft can be seen with the movement of the dot. If the movement is opposite to the arrow, Power down the 3 phase supply and reverse any 2 wires on the 3 phase breaker.

To activate the pump press this sequence of buttons.

Power On (wait for the display to show)

- Check that the emergency stop button is not depressed.
- Check that coolant switch is set to automatic.

On the control panel key in this sequence:

- MAIN MENU
- F4(jog)
- F4 (MDI)
- M8(on keypad)
- Cycle start (green button on control)

The pump will rotate now. To stop pump rotation turn coolant switch to off.

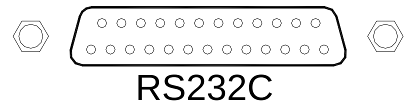
Other connections

If a barfeed is to be connected please refer to the connection information supplied with the bar feed.

Please refer to the section on live tooling for information on this subject.

RS232 Connections

Data connection port



The control can be linked to a separate computer, so that programs, parameters, and offsets can be sent and received. This connection is through the DB25 connector on the side of the control box.

Parameter Configuration

The standard machine set up :

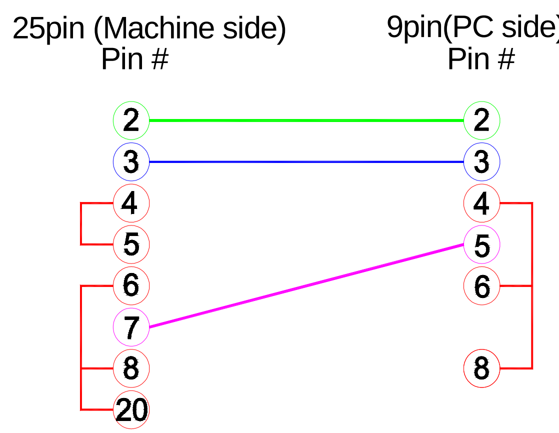
To access the parameters press:

Main Menu
F7 (+)
F4 (Machine parameters)
F4 (Serial ports)
F2 (Serial 2)

Description	Value	Parameter value
Baud rate	9600	P000 = 7
# of bits	7 or 8	P001 = 1
Parity	No	P002 = 0
Stop bits	1	P003 = 0
Protocol	DNC	P004 = 0

See the Fagor installation manual for complete descriptions on these parameter settings

Cable Wiring



The cable configuration is shown.

The cable requires a male connector for the machine end, see your computer port for the correct type of connector at the PC end.(Com port)

Programming Considerations

The program **must** begin with a **%** sign followed by a part program description, a comma, MX and a final comma : %test program,MX,

(MX is required for the file protection system)

and the program must end with either:

an <EOT> (ASCII 4) character

or <ESC> (ASCII 27) character

or 20 <NUL> (ASCII 0) characters.

Example:
%test program,MX, <LF>
N1 G00 X0 <LF>
X1.5 <LF>
M30 <LF>
<EOT>

Most commercial software allows the addition of ASCII codes on the end of file transmissions. Please see your software vendor for specific instructions.

Please refer to section 8 (Fagor 8055 operation) for sending and receiving programs.

Section 3

Fagor 8055 T Screen Layout

&

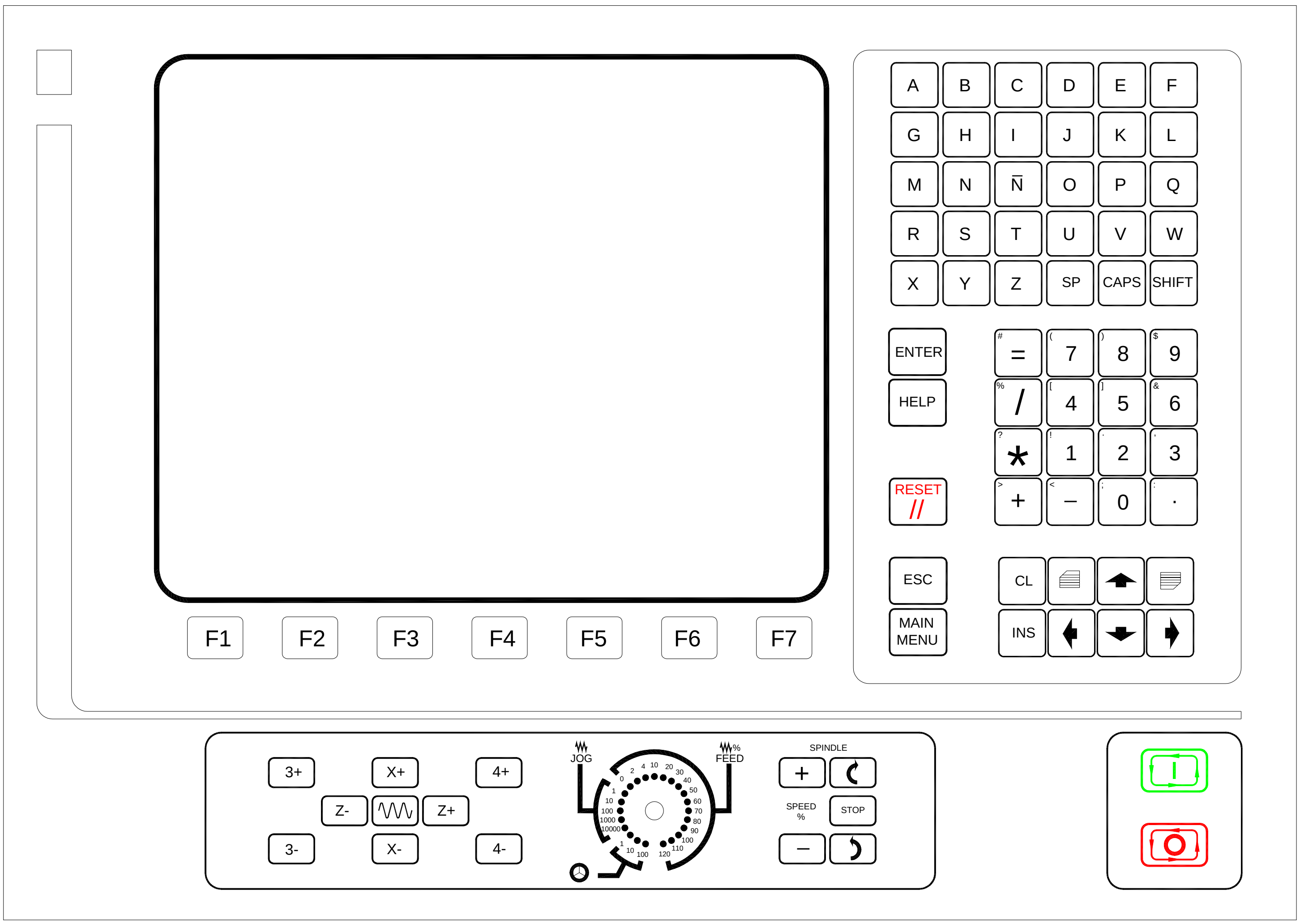
Operator panel

Screen and keyboard layout

The control is laid out in a simple to understand format comprised of:

1. **LCD Screen**
2. **Alpha keypad**
3. **Numeric keypad**
4. **Cursor keys**
5. **Function keys**
6. **Jog keys**
7. **Spindle keys**
8. **Cycle Start and Feed Hold keys**
9. **Reset key**
10. **Main Menu key**
11. **Enter key**
12. **Feedrate override knob**

The basic layout of the control is shown below.



Control panel layout(Fig4.1)

The following section will cover each area of the control.

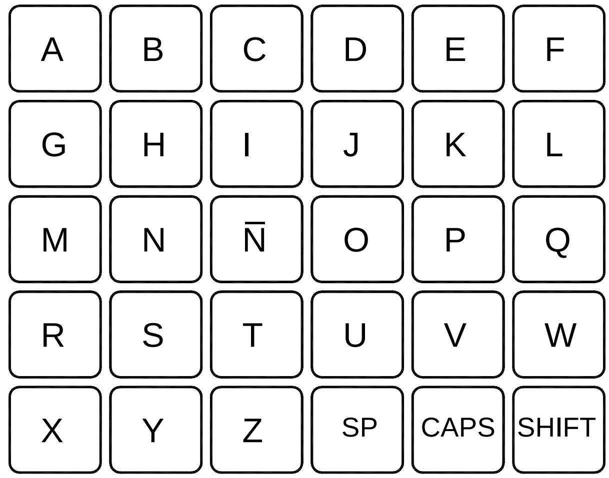
LCD Screen

The screen area changes depending on the Function key selected. Each Function can display a number of pages and the cursor up and down keys can access these pages.(see Function keys)

Alpha keypad

The Alpha keypad is used to input programs and messages and select axes for the control.

The 3 keys at the bottom right are used for messages and are described below.

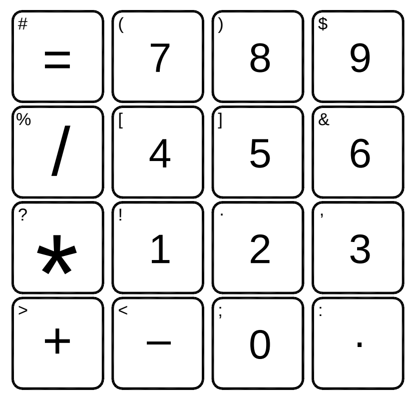


SP Space key for adding spaces to messages

CAPS Capitals key for using upper and lower case letters in messages.

SHIFT Shift key is for accessing the special function keys on the numeric keypad.(see numeric keypad below).

Numeric keypad



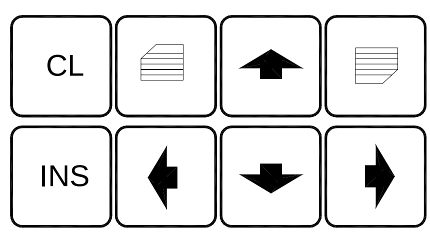
The numeric keypad is used to enter numbers and symbols into the programs, and to enter information when setting tools or offsetting. Or any time that a number is required by the control for an operation.

To access the special symbols on the upper right of the keys press the **SHIFT** key on the alpha keypad,

Cursor keypad

This area is used to scroll through programs or sections of the screen. When editing a program the left and right keys move you through the program line. The 2 keys at the top are used for editing and are grouped in this area for ease of access.

INS This key is the **INSERT** key. Pressing this key in edit mode gives a highlighted cursor and moves all existing information to the right as you input new information to the left. A complete explanation is include in the section, How to edit a program.



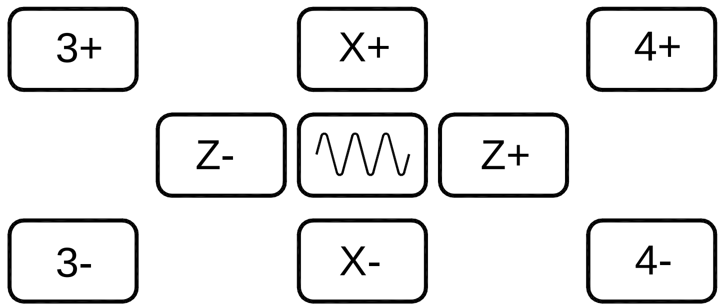
CL The **CLEAR** key removes information. Place the cursor on the code you wish to remove and press CL.

Function Keys

These keys provide different screen depending on the **OP MODE** selection

F1 F2 F3 F4 F5 F6 F7

Jog Keys

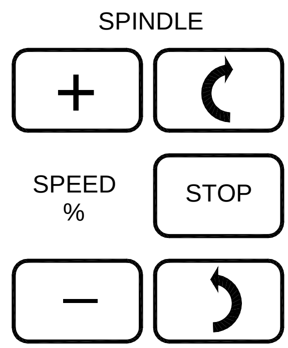


The jog keys move the machine in jog mode X moves the slide across the spindle and Z moves the slide towards or away from the spindle. The speed of movement can be controlled by the position of the jog override knob.

The X & Z axis keys reflect the movement Directions of each axis.

The 3 & 4 keys are for the extra possible axes. On the standard configuration machine these keys are not functional.

Spindle Keys

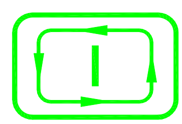


The spindle keys start and stop the spindle in either a clockwise or counter-clockwise direction. The percentage override range is 50% to 120% of programmed speed.

These keys are active in Execution and Jog mode. In MDI or Handle mode always esc back to the jog screen prior to using these keys.

(If the spindle will not respond to the Start Reverse & stop keys check that you are in JOG mode and a spindle speed is active.)

Cycle start and Feed Hold keys



The Cycle start (upper key) starts the machine cycle in Automatic and dry run modes, and is used to activate tool initiation in setting mode.

The Feed Hold key (lower key) halts operation of the control and stops axis motion.

When the Feed Hold is pressed the screen flashes “Interrupted”

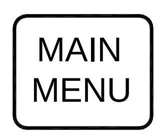
Reset key

The Reset key sets any mode in operation to an off status.

When the Feed Hold is pressed the screen flashes “Interrupted” to end the cycle press **RESET** .

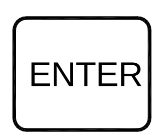
To reset the control after changing parameters press the SHIFT & RESET key.

Main Menu key



The Main Menu key returns to the original power on screen.

Enter key

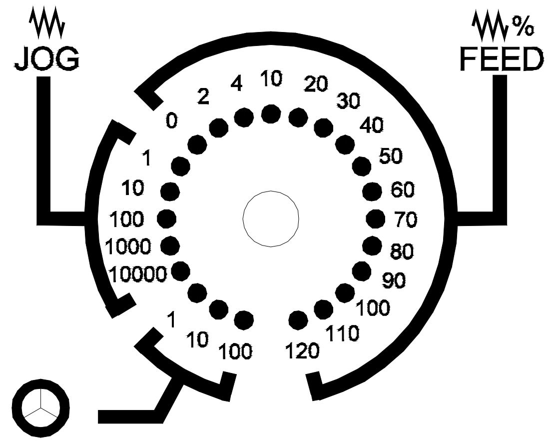


The Enter key is used to accept any information that will be written to the control memory.

Feedrate / Jog Override knob

The jog override knob has a number of settings when moving the machine slides manually.

Feedrate override



The feedrate override section (0-120%) is only enabled when in execute, and jog mode.

This mode allows the operator to slow down or stop the slide motion, including rapid moves.

The operator can safely run a program and check the feedrates for the program, and easily determine how much the program feedrates need to be adjusted to complete the parts safely and with the correct finishes.

This also overrides the rapid traverse speeds, when running a program.

Jog override

The jog mode override is used in jog, and teach in mode. Each position allow the machine to be jogged that amount. in inch mode each press of the axis jog button moves the selected axis this amount:

Switch location	X axis	Z axis
1	0.0002”	0.0001”
10	0.0020”	0.0010”
100	0.0200”	0.0100”
1000	0.2000”	0.1000”
10000	2.0000”	1.0000”

Warning:

Be very careful when close to the collet or chuck if 10000 is selected and you are within 1” of the collet or chuck and Z- is pressed a collision is likely. An operator will not be able to stop the machine in time to avoid a collision.

Handle override

The handle mode allows the axes to be moved by using the manual pulse generator(MPG).

Turn the knob to this section and press the X or Z on the alpha key pad(to the right of the screen)

The axis will display in reverse video(Highlighted) on the screen. Each position represents a different amount of movement.

The amount of movement is determined by the parameters in the control and the axis drives.

The amounts equal:

1 =0.0001”

10 =0.0010”

100 =0.0100”

Secondary control panel

The switches & buttons on the left of the controller are for coolant, collet, door interlock, Option stop & Home search operation.

The buttons and handle on the right hand side of the machine are for the MPG handle, emergency stop, and warning light. These buttons and switches may vary by machine.

Optional Stop button (Blue)

This blue push button is for activating the M01 optional stop feature of the control.
When lit, an M01 in the program when executed will cause the execution of the program to stop.
The spindle & coolant if on will continue to run.

Coolant switch

The coolant switch controls whether the coolant pump is activated when programmed, if it is set in the OFF position, the coolant pump will not activate when an M08 code is commanded. But if turned to Auto when an M08 is active the coolant pump will start.

If the switch is in the AUTO position the coolant will be activated by a M08 and deactivated by a M09.

Collet switch (Main Spindle)

The collet switch controls whether the collet is activated when programmed, if it is set in the OPEN position, the collet will open, when in AUTO mode the collet will close and an M10 will open the collet. and an M11 will close the collet.

If the cycle start button is pressed and the collet is open it will close regardless of the switch position.

Collet switch (Sub Spindle)

This collet switch controls whether the sub collet or chuck is activated when programmed, if it is set in the OPEN position, the collet will open, when in AUTO mode the collet will close and an M21 will open the collet. and an M20 will close the collet.

Door Interlock switch

This switch activates the door interlock switch and disables the cycle start buttons and the jog axis & spindle buttons.

Home search button (Orange)

Any time that the machine axes have to be homed this button must be held down.

Any time that an axis is over traveled and generates an emergency stop this button must be held down while jogging the axis off its hard stop.

Inverter Reset button (Orange)

Any time that the spindle inverter requires a reset, press this button to reset.

Cycle Start button (Green) Optional

A 2nd cycle start button may be present on the right hand control panel, press this to button to start operation in execute, simulate or MDI mode.

Manual pulse generator

This handle will move the X or Z axis of the machine when in handle mode.

Please refer to the prior page for explanations of use and axis and amount selection, for this handle.

Section 4

Fagor 8055TG Operation

Turn on the main power & Home the Machine

1. Verify the emergency stop button is off (out)
2. Press **MAIN MENU** key.
3. Press **F4** to go to jog mode, to home the machine
4. Press **F1 (Reference search)**
5. Press **F7 (All axis)**
6. Keep the Yellow HOME Button pressed and then press Cycle Start

The machine slides will move to their home positions, A message is displayed in the upper left corner of the screen.

Keep the yellow home button held down until this message area clears.

Home Sequence sub routine

The **ALL AXES** button activates a hidden sub program #9000 hidden in the m/c subroutine file.
This program is a series of commands that home the machine axes and on C axis and FX lathes, the spindle or spindles.

This program sets the tool offset to zero, & cancels any G54-G59 work offset.

To home just the spindles on an FX machine you can use the M96 command.

Starting the Machining sequence

1. Press **MAIN MENU** and then **F1** this will put you in execute mode.
2. Cursor to the program you wish to run, Press **ENTER**

Note! always make sure the program is at the correct block # before pressing the start key.

Press **RESET** to reset the program to the starting block, Press **CYCLE START**

1. If the program appears to be running correctly, move the feedrate override switch to 100%.

   Start (the green key) will restart the program

   Feed hold (the red key) will freeze the slides in position.

   Reset when pressed will stop the program.

You must reset to the beginning of the program when you restart the program.

Before you set tools

Always ensure that the X & Z axes have been homed prior to setting tools.

It is always recommended to reset any G54 thru G59 offset to zero by using the G53 offset.

The following example will automatically set the X and Z tool coordinate for a standard right hand turning tool.

Setting a tool

You should have all tools set on the tool plate prior to tool setting. Decide which numbers are to be assigned to each tool.
In this example it is assumed that T1 is to be set. Check that the jog knob is set to feedrate override mode.

Press this sequence of buttons to set a tool at **T1**

MAIN MENU
F4 (Jog)
F4 (MDI)
T1
Cycle Start

The screen will now display **T001 D001**

Starting the spindle before cutting

You are still in MDI mode, press

S1200
Cycle Start

This will set a spindle RPM

Press **ESC** to return to jog mode

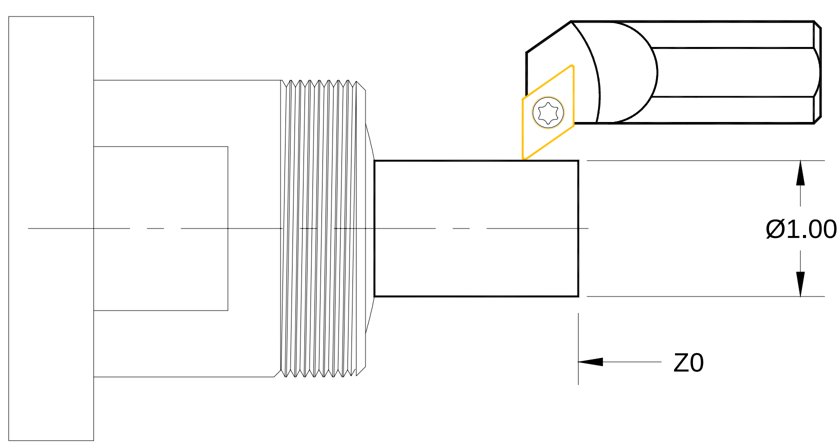
Select a spindle direction with the spindle jog keys.

Setting the X coordinate

Jog the tool in the X direction to touch to the outside diameter of the stock.
Jog off the stock in the Z+ direction without moving X. In this example the material is 1.00 diameter and the tool is in line with that OD. enter the value as follows.

F4 (Tool Calibrate)
F1 (X) 1. , Enter
F4 (X load)

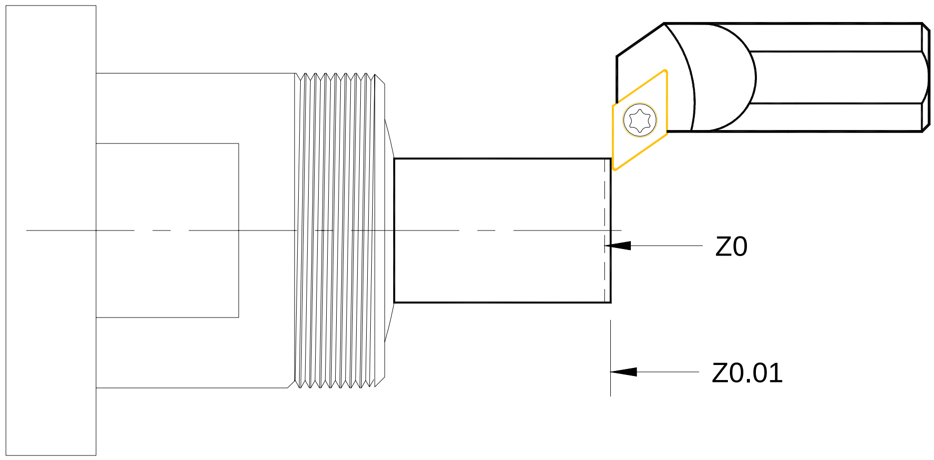
the X value is now recorded and is displayed on screen.



Setting the Z coordinate

Jog the tool in the Z-direction until you touch the end of the stock.
Then without moving Z, jog off in the X+ direction until the tool is clear. In this example we are allowing .010" facing stock on the end of the material so we will enter the value as follows:

F2 (Z) 0.01 , enter
F5 (Z Load)
The Z value is now recorded and is displayed on screen



To load the next tool you must enter the new tool number and go through the same procedure.

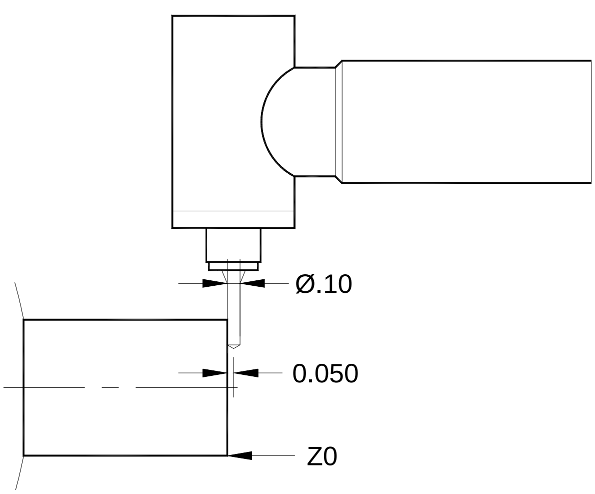
Notes:

- The control stores the last entered value for X & Z. This is useful when entering Z0 values for multiple tools.
- Always remember that X can be positive or negative, enter the correct value including – if the value is negative.

Setting Live tools

Always set the tool centerline for end mills in an axial orientation. C axis will not give correct results if this setting is incorrect.

To set the drill center line in radial applications. Touch off the side of the tool with Z0 and tell the control the radius of the drill or end mill.



F2 (Z) 0.05 , enter
F5 (Z Load)

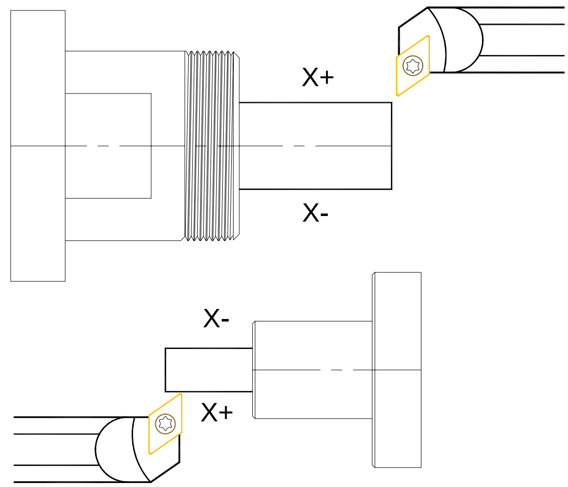
The control knows that the center is now 0.05 from Z0

Setting Sub spindle tools

When setting the tools on the sub plate the X values change sides only.

The Z offset values are still at Z0.

It is recommended that a G54 offset is used for main spindle tools, and a G55 offset is used for the sub tools.
This allows shifting of the Z0 point for length control.

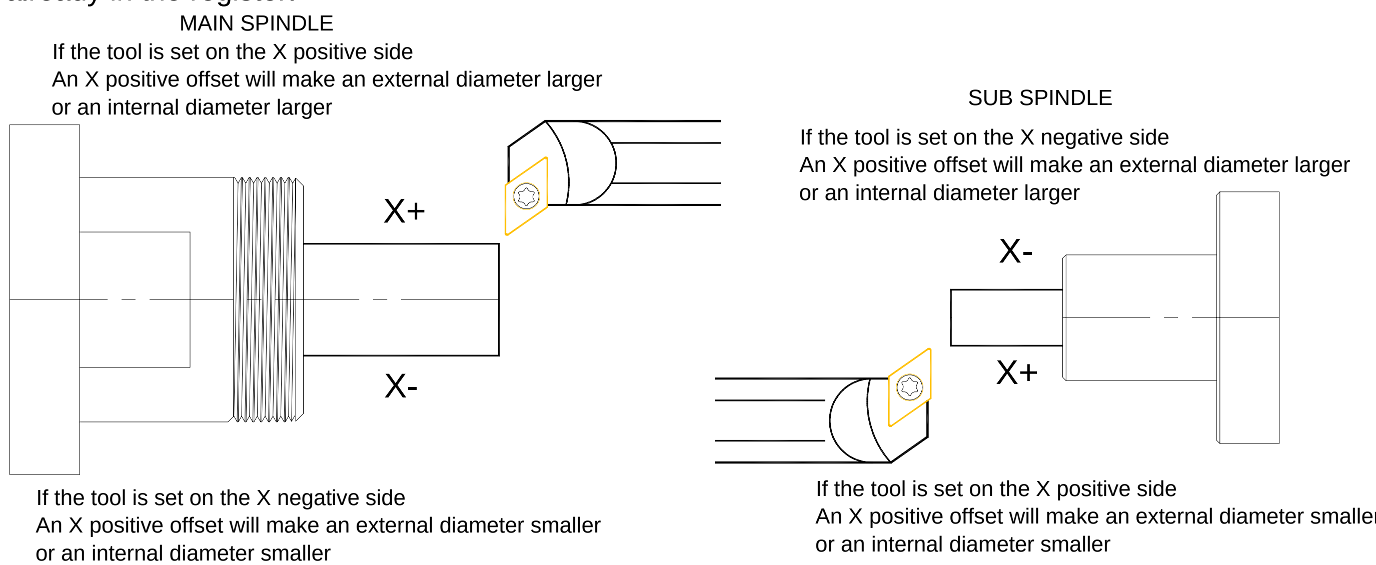


Offsetting the tool in Execution mode

In Execution mode there is a Tool Offset choice on the screen.

Press **F5 (Tools)**
Press **F5 (Tables)**
Press **F2 (Tool offsets)**

To make an offset to a tool
press **F1 EDIT** (The Lower screen will display a D)
Type in the tool offset #, press I or K and type in the offset amount.
If an offset is already in the register, you have to add or subtract the new offset amount to the offset already in the register.



Offset directions(fig 8.1)

Screen layout for editing

The editing functions and program management and display methods will be covered in this section.
You can:

- Create new programs
- Create sub programs
- Create parametric sub programs
- Edit existing programs
- Delete existing programs
- Rename existing programs
- Copy an existing program to create a copy that can be edited without changes to the original.
- Copy all or parts of a program to a new program
- Paste all or part of a program into an existing program.

The 8055 editor is a powerful full featured editor. The purpose of this section is an introduction to the editor. Please refer to the Fagor operating manual for a full explanation of features.

Press this sequence of buttons to view the editing section:

MAIN MENU
F3 (Edit)
A screen displaying the stored programs is displayed.
This area is used to create and edit new or existing programs.

The **UTILITIES** section is for management of the programs.

Starting a new program

Press this sequence of buttons to start a new program:

MAIN MENU
F3 (Edit) The screen will show a list of existing programs,

Enter a program number that is not currently used.
If a number entered is not already used the control will start a new program.
To add a title to the program press the new comment key, enter a 16 digit or less title.
Pressing enter opens a new program or loads an existing program.

The screen will now show a split screen with the program number displayed in the top
You can now enter a program by typing in the required codes.
There are 3 choices of program data entry.

1. **CNC LANGUAGE**
2. **TEACH-IN**
3. **INTER-ACTIVE**

To type in a new program using G code press F1 (CNC Language).
This allows you to enter G codes in a simple format.
The function keys displayed, allow fast entry of codes.

NOTE: If no codes are entered and you exit the edit mode the program will not be stored.

Writing a program using INTERACTIVE mode

When in edit mode you can type in a canned cycle G code and press the HELP key.
This will activate the INTERACTIVE screen for the G code entered.

If a standard G code is entered and HELP is pressed a screen giving help on that G code is displayed.
These interactive screens are designed to help format the G codes easily, and are a very powerful feature of the editor.

Accessing and editing an existing program

Press this sequence of buttons to access an existing program:

MAIN MENU
F3 (Edit)
You can now see the list of programs in the control memory cursor to & highlight the program you wish to edit.
Press ENTER
The screen will now show a split screen with the program displayed in the upper section.
You can cursor or page down through the program.
To edit a line of code follow these steps:
F2 (Modify) This will display the line of code in the lower section of the screen.
Cursor across and type changes to the line of code.
Press ENTER to overwrite the old line with the new line.

Adding a new line to an existing program

Press this sequence of buttons to add a new line to an existing program:
Cursor to the line before the new line location

Press CNC language (F1)
Enter the new line of code
Press enter.
To escape from this mode **press ESC**

Deleting, Moving & copying blocks

All of these functions operate in the same way, The basic rules are, Select the first line of code. Cursor to the last line of code and press enter. The softkey selected initially set the operation type.

Deleting blocks

To delete only one block, just position the cursor over it and press **ENTER**.
To delete a group of blocks, indicate the first and last blocks to be deleted. To do so, follow these steps:

- Position the cursor over the first block to be deleted and press the “INITIAL BLOCK” softkey.
- Position the cursor over the last block to be deleted and press the “FINAL BLOCK” softkey.

If the last block to be deleted is also the last one of the program, it can also be selected by pressing the “TO THE END” softkey.

- Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to delete them.

Moving blocks

- Position the cursor over the first block to be moved and press the **INITIAL BLOCK** softkey.
- Position the cursor over the last block to be moved and press the **FINAL BLOCK** softkey.
- If the last block to be moved is also the last one of the program, it can also be selected by pressing the **TO THE END** softkey.
- To move only one block, the "initial block" and the "final block" will be the same one.
- Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to move them.
- Then, indicate the block after which this group of blocks must be placed.
- Press the **START OPERATION** softkey to carry out the move.

Copying blocks within a program

To access the copying features of the control you must press the **F7 +** softkey

- Press **Copy block** softkey
- Position the cursor over the first block to be copied and press the **INITIAL BLOCK** softkey.
- Position the cursor over the last block to be copied and press the **FINAL BLOCK** softkey.
- If the last block to be copied is also the last one of the program, it can also be selected by pressing the **TO THE END** softkey.
- To copy only one block, the "initial block" and the "final block" will be the same one.
- Once the first and last blocks are selected, the CNC will highlight the selected blocks requesting confirmation to copy them.
- Then, indicate the block after which this group of blocks must be placed.
- Press the **START OPERATION** softkey to carry out this command.

Copying blocks to a new program

- Press **Copy To Program** softkey
- Position the cursor over the first block to be copied and press the **INITIAL BLOCK** softkey.
- Position the cursor over the last block to be copied and press the **FINAL BLOCK** softkey.
- If the last block to be copied is also the last one of the program, it can also be selected by pressing the **TO THE END** softkey.
- To copy only one block, the "initial block" and the "final block" will be the same one.
- Once the first and last blocks are selected, the CNC will highlight the selected blocks and will execute the command.
- If the destination program already exists, the following options will be displayed:
- Write over the existing program. All the blocks of the destination program will be
- erased and will be replaced by the copied blocks.
- Append (add) the copied blocks behind the ones existing at the destination program.
- Abort or cancel the command without copying the blocks.

Copying a program into program

Once this option is selected, the CNC will request the number of the source program to be merged. After keying in that number press **ENTER**.
Next, indicate with the cursor the block after which the source program will be included.
Finally, press the **START OPERATION** softkey to execute the command.

Renaming an existing program

Press this sequence of buttons to change the number of an existing program to a different number :

MAIN MENU
F6 (Utilities)
F4 (Rename)
The control enters a P, type in the program # to be changed
Press TO (F1)
Press either the [NEW NUMBER] or the [NEW COMMENT] softkey
Key in the new number or the new comment
Press ENTER

Copying an existing program

Press this sequence of buttons to make a duplicate copy of an existing program to a different number:

MAIN MENU
F6 (Utilities)
Press the (COPY) softkey
Indicate the location of the program to be copied (Memory, CARD A, HD or DNC)
key in the program number to be copied
Press the (IN) softkey
Indicate the destination of the copy (RAM memory, CARD A, HD or DNC)
Key in the program number
Press (ENTER)

Deleting an existing program

Press this sequence of buttons to delete an existing program:

MAIN MENU
F6 (Utilities)
Press the (DELETE) softkey
Indicate the location of the program to be deleted (MEMORY, CARD A, HD or DNC).
Key in the number of the program to be deleted or place the cursor over it.
Press [ENTER]

NOTE: All the examples above show the full keystrokes required from the power up passed screen. You can move between screens in edit mode by pressing the continue button.

Using the Simulate function to test new programs

The Simulate function on the 8055TG controller is very useful for testing new programs, this function is explained below.

Using the Graphics function

There are 2 choices that are normally used when in simulate mode:

- **G functions**
- **Theoretical Path**

Each selection affects the actuation of the simulate operation differently:

G functions

This function allows the control to execute the program activating all G codes including Tool nose radius compensation,(G40,G41,G42)

Theoretical functions

This function allows the control to execute the program not using cutter compensation, and displays the actual programmed tool path.

Press this sequence of buttons to use the simulate mode:

MAIN MENU
Simulate (F2)
Select a program from the displayed list Press **ENTER**
Theoretical path
The control displays the operation screen that is similar to Execute mode

You can run the program and check for alarms.

Note :
The control checks tool offsets, and displays errors if the tools are not set.
The control will check program format.

Using the Graphics function to test new programs

The graphics function on the 8055TG controller is very useful for testing new programs, this function is explained below.

Graphics function

In Execute & Simulate modes there is a Function key Graphics (F6) available.
when F6 is pressed the control will display a graphics screen:
Select the type of graphics to be displayed by pressing **F1**

A set of softkeys allows line or solid graphics in a range of axis selections.

Press Type of graphics (F1)
Press X-Z (F1)
Press CYCLE START
A simulation of cutter path will be displayed.

If you press:
MAIN MENU
SIMULATE
G CODE
The graphic screen will show cutter compensated path if G42,G41 are used

Solid Graphics function

To use solid graphics or simulation of canned cycles, it is very important that the graphic variables for the tools are defined.

Press this sequence of buttons to find the graphics definition page.

Tables (F5)
Tools (F3)
Geometry (F7)

The page displayed, shows the tool # and X-Z R & F offset data on the left side,and on the right side are shown 3 fields.

NOSEA (Nose angle)
NOSEW (Nose width)
CUTA (Cutting angle)
In the center of the screen is a graphic showing the tool tip and how the variables are used.

Page up & page down select the tool number to be edited (or press FIND)
Pressing Edit (F1) will allow you to enter these 3 fields.

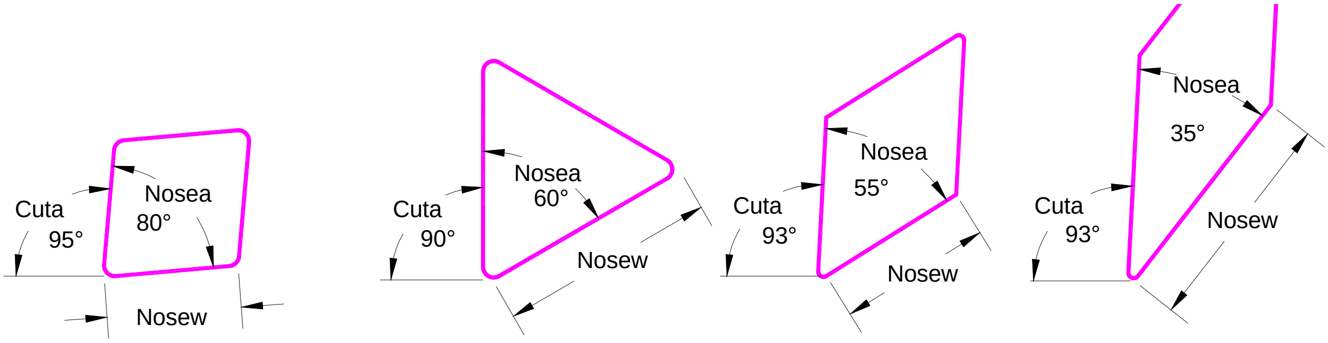
F1 = NOSEA (Nose angle)
F2 = NOSEW (Nose width)
F3 = CUTA (Cutting angle)

The R & F codes have to be entered on the tool offset page.

The following pages will show the most popular tool types and the variables to describe them

Turning tools

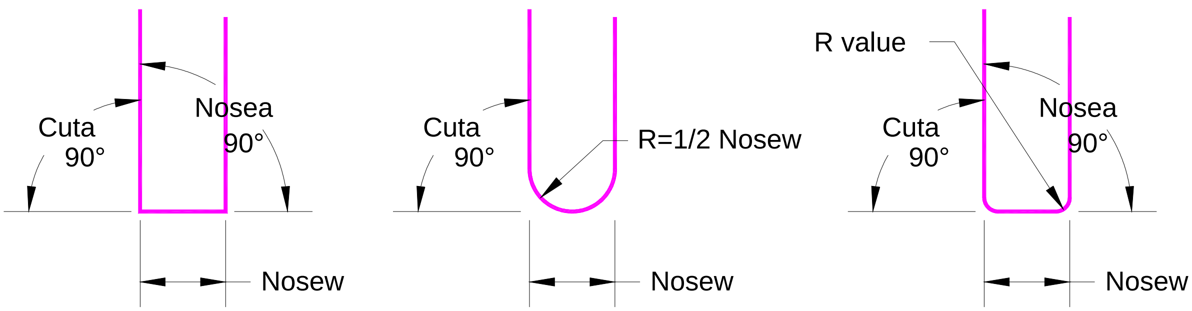
The typical turning tool is an 80^{O} diamond, 60^{O} triangle, 55^{O} diamond, and 35^{O} diamond.



The values for CUTA, and NOSEA are constant, the value of NOSEW varies depending on the size of the insert.
The F value in the tool offset page determines the orientation, not the CUTA value.
Usually F3 or F5 is used for this type of tool

Grooving tools

For groove tools use the F value for the tip registration point not for the graphics value



Notice that both the CUTA & NOSEA are 90^{O} This applies to all grooving tools.

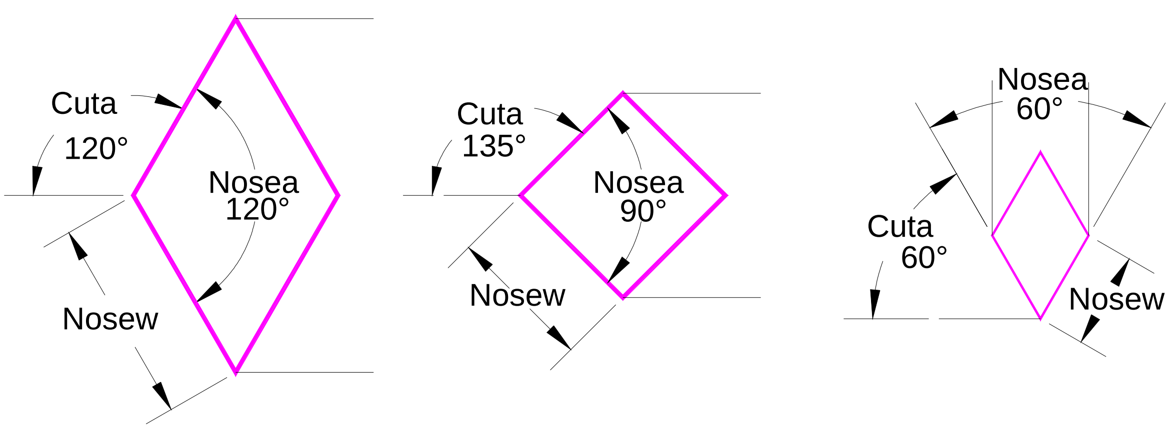
A radius value can be applied to the edge of a groove tool. This can be compensated for in a roughing canned cycle.

Drilling & Threading

Drills use F4 as an orientation code and threads use F2 or F6, but their definition is similar

Shown is a standard drill point and an NC spot drill point of 90^{O}.

The thread tool is almost universal.



Connecting to peripheral devices

Press this sequence of buttons to display the choices for communications:

MAIN MENU
Utilities (F6)

This displays an area where you can alter the status of programs, You can copy, delete, rename, Or set status with protection.

If the software you are using is not FAGOR software, you must set serial parameter P4 to 0

Sending an existing program to a PC

Press this sequence of buttons to send an existing program to a PC:

Utilities (F6)
Copy (F2)
Memory (F1)
Enter program # to copy
To (F6)
RS232 (F4)

The control will now send the program to the RS232 port,
When the program has been transmitted the control displays:

PROGRAM SENT

Notes:

You must set the PC ready to receive before sending a program.

Software for a personal computer may be set differently,

Check with you software vendor for any configuration setting before attempting to connect this controller to a computer.

Receiving a program from a PC

The controller must be set up before sending a program from a PC

Press this sequence of buttons to receive a program from a PC:

Utilities (F6)
Copy (F2)
RS232 (F4)
Enter program number (if not using DNC50 enter 1)
To (F6)
Memory (F1)
Enter program number for new program
ENTER

On the lower green line the control will display **Transmitted blocks…0**

Go to the PC and send the program.

The Fagor 8055TG controller requires very specific data formats. The program must start with a % and a 6 digit program #, followed by a comma and MX and a final comma.

Example: **%123456,MX,**

When the controller receives a % it starts to create a new program with the new number.
If this number is already in memory the control will display:

P###### exists, Replace (enter/esc)

You can delete or not by choosing enter or esc.

Select a different program #

Notes:

You must set the Controller ready to receive before sending a program from a PC.

The software must be able to send special ASCII codes at the end of data transmissions.

Check with you software vendor for any configuration setting before attempting to connect this controller to a computer.

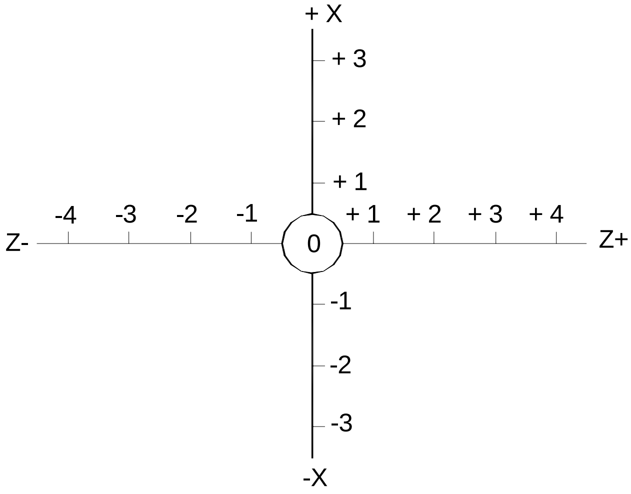
EVERY CONTROL IS TESTED PRIOR TO LEAVING OUR FACILITY FOR UPLOAD AND DOWNLOAD.

CABLE CONFIGURATIONS ARE CRITICAL TO SUCCESSFUL TRANSMISSIONS.

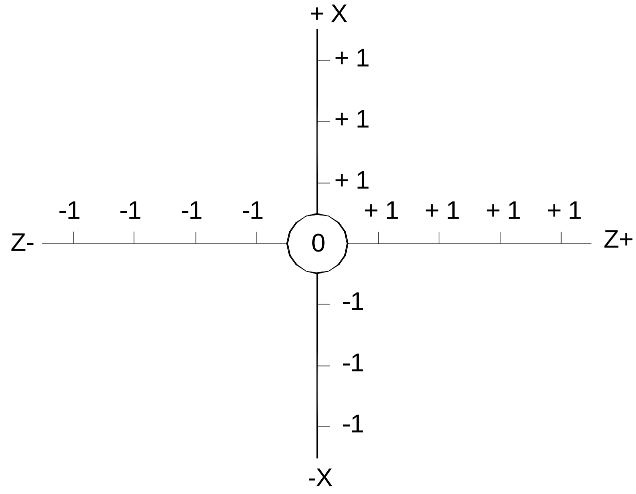
90% OF ALL PROBLEMS RELATED TO RS232C ARE PC OR CABLE RELATED.

Section 5

M & G codes



G90 Absolute system (Fig 5.1)



G91 Incremental system (Fig 5.2)

G90 Absolute Position

Absolute positioning uses a known fixed zero point as a reference for the commanded positions.

A move from X0 Z0 of X1.0 Z1.0 would move the tool to position X1.0 Z1.0, if a command of X3.0 Z-2.0 is made the tool will move to the grid position X3.0 Z-2.0. This is the normal method of programming and directly relates to most machine part prints.

The grid has a specific location for any point and the X and Z locations call these points.

G91 Incremental Position

Incremental coordinates use the last position of movement to define the next move.

If a move from X0 Z0 of X1.0 Z1.0 is commanded, the new coordinate would be X1.0 Z1.0, if the next move was X2.0 Z1.0 the new coordinate would be X3.0 Z2.0(X2.0+X1.0)(Z1.0+Z1.0).

As you can see this uses the last position as the next position zero point.

This option is very useful when programming the same feature on a part a number of times. (grooves) or a grid of holes when using a live tool, and a sub program is called. You would not have to define a new zero position, as each call moved from the last absolute commanded position.

Radius and Diameter Programming

A program can be created using either Radius or Diameter co-ordinates but cannot be switched between types in a program. The machine parameters must be set for the type of programming method required.

The most common method of programming a lathe is in diameter mode, and this method is the one that all explanations will be given in.

The most basic difference is that in Radius mode a required part diameter of 1.0” would be called out as X.5

In Diameter mode the same part dimension would be called as X1.0

As you can see, this will make your programs much easier to read and trouble shoot.

In some of the canned cycles values are requested as Radius values, This is to assist in ease of programming.

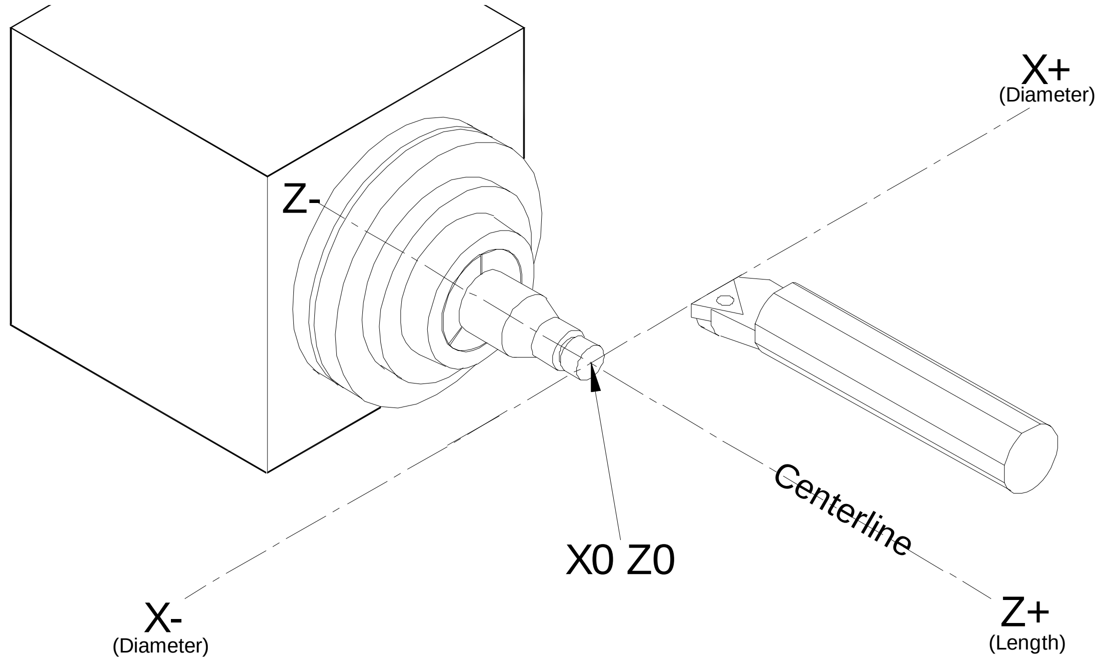


Table axis reference system(Fig 5.3)

Axis direction

The axis direction on a lathe is controlled by a variety of parameters in the controller.
These parameters can be set to allow different axis directions. The most often changed axis is the X direction. The Z axis direction is normally not changed, and its movement is generally as shown in fig 5.3.

Tool Hand directions

When using a gang tooled lathe both left and right handed tools can be used to machine parts. The advantage of this is the ability to set tools so that fast cycles can be achieved. Most tools used on a gang tooled lathe are boring bars as this allows more tools on the tool plate, and the ability to adjust the length of the tool to suit the job being machined.

The GTS27 is setup so that a right handed boring bar when located with the insert face up, is programmed with a positive X axis co-ordinate. Offsetting the tool for wear is correct (X+ offset and the diameter increases.)

Drilling takes place on the X axis centerline and is unaffected by the axis direction. Most live tools are set up at the top of the table and again allow positive X axis values for positive offsets.

When a program example is given it will assume this format. When threading any special needs will be explained, with examples.

Construction Of The Program

After the proper coordinates have been assigned to the drawing, the programming consists of turning on or off the proper accessories and sending the tools on the path you have determined necessary to complete the part.

A rough layout of the tools on the slide or on a piece of paper should be completed before you start to write the program. Take into consideration the length of the tools and the proximity of one tool in respect to another while it is cutting the parts. Always leave room for the various tools. After a tool is clear of the workpiece the exit move should always be in the Z+ direction to avoid crashing other tools in line on the slide. Determine the tool number for each tool that you are going to use then you are ready to start writing the program.

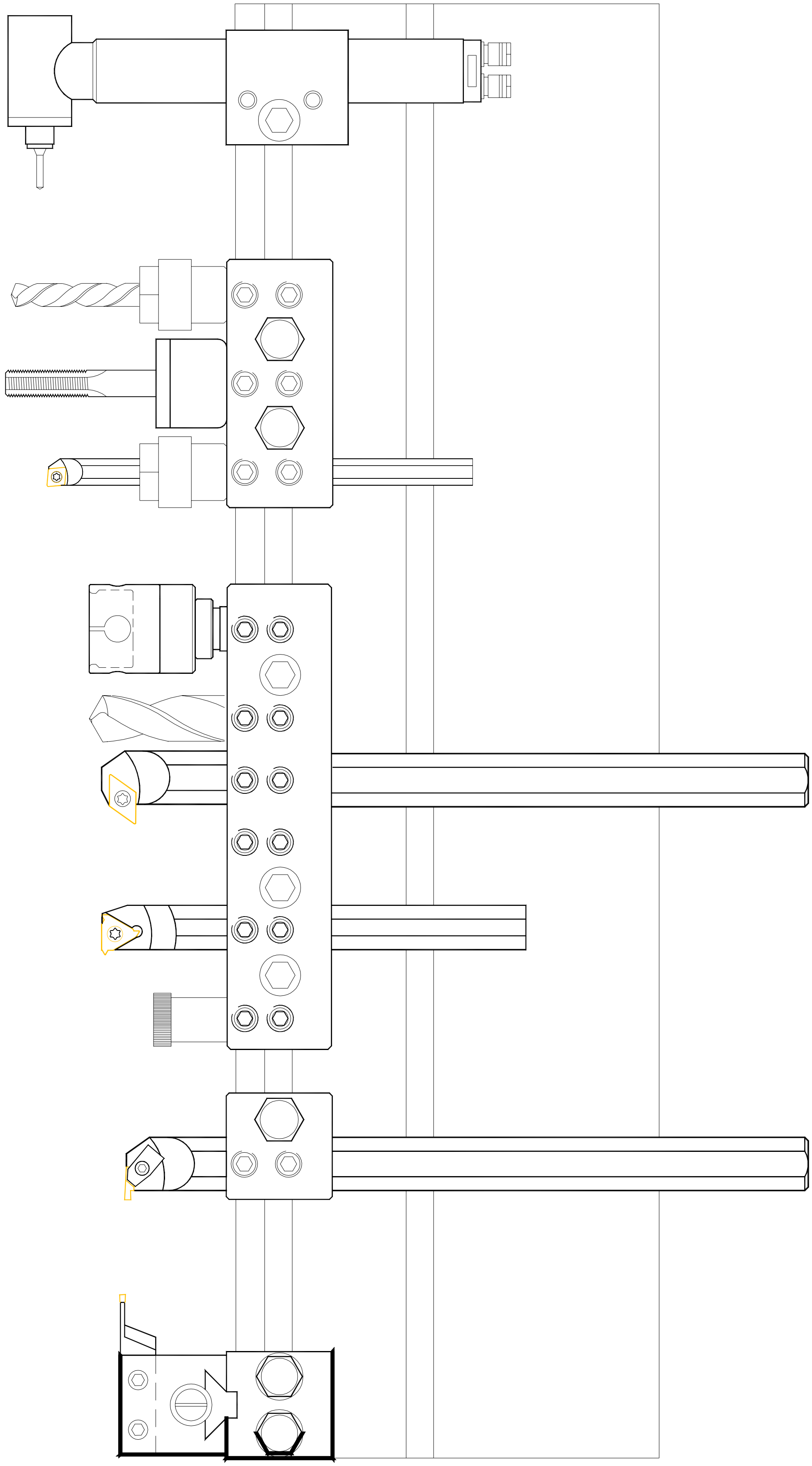


Table layout(Fig 5.4)The tooling plate shown is displayed to demonstrate a possible layout of tools.

Miscellaneous Functions

M codes

M codes are commands that turn on or off mechanical switches, these can be Relays or programmed operations.

The M codes are grouped:

M00 thru. M30 are standard M codes.

M31 thru. M99 are optional M codes, depending on the options supplied with each machine.

M100 up are customizable functions used for parametric and Macro use.

Standard M code list

M00	**Program hold**	**M11**	**Main Collet open**
M01	**Optional program stop**	**M13**	**Spindle forward / coolant on**
M03	**Spindle forward**	**M14**	**Spindle reverse / coolant on**
M04	**Spindle reverse**	**M15**	**Spindle stop / coolant off**
M05	**Spindle stop**	**M19**	**Orientate spindle**
M08	**Coolant on**	**M20**	**Sub Collet close**
M09	**Coolant off**	**M21**	**Sub Collet open**
M10	**Main Collet close**	**M30**	**Program end (rewind)**

M code Descriptions

M00 Program Stop

The M00 code is used to stop operation of a program. It causes the spindle and coolant to be switched off. A cycle start will resume operation of the program from the cursor position.

M01 Optional Program Stop

The M01 code is used to stop operation of the program selectively by activating the optional stop button. The spindle and coolant are not switched off. A cycle start will resume operation of the program from the cursor position.

M03 Spindle Forward

The M03 code will start the spindle in a clockwise direction at the programmed spindle speed. The machine waits until the spindle reaches a percentage of the programmed speed. Direction of the spindle is determined by standing at the left side of the machine and looking towards the tool slide M03 is considered clockwise.

M04 Spindle Reverse

The M04 code will start the spindle in a counter clockwise direction at the programmed spindle speed. The machine waits until the spindle reaches a percentage of the programmed speed. Direction of the spindle is determined by standing at the left side of the machine and looking towards the tool slide M04 is considered counter clockwise.

M05 Spindle Stop

The M05 code stops the spindle.

M08 Flood Coolant On

The M08 code will start the main coolant pump. This code is activated at the end of a block . The coolant switch if OFF overrides the coolant pump activate.

M09 Coolant Off The M09 code switches off the main coolant pump.

M10 Collet close

The M10 code is used to retract the draw bar into the spindle(close the collet or chuck)

M11 Collet Close

The M11 code will advance the drawbar from the spindle.(open the collet or chuck)

M19 Index spindle (Optional)

The M19 code will set the spindle controller in index mode, specify an absolute angle that the spindle is to index to. Example M19 S90(see Live Tools for a further explanation).

M20 Sub Collet close (FX only)

The M20 code is used to retract the draw bar into the spindle(close the collet or chuck)

M21 sub Collet open (FX only)

The M21 code will advance the drawbar from the spindle.(open the collet or chuck)

M30 Program Rewind

The M30 code is used as a program end. It will stop the spindle and turn off the coolant.

The cursor will be reset to the beginning of the program.

Option M code list

These M codes are grouped into areas based on type of option. Only the activate codes are listed,

To turn off these codes use the next higher M code,

Example : M50 live tool 1 on, M51 live tool 1 off

Air Live Tools

The first group controls live tool functions, Depending on machine configuration these codes may vary. When air tools only are used the format is as follows:

M50 Live tool 1 on (Air)
M52 Live tool 2 on (Air)
M54 Live tool 3 on (Air)
M56 Live tool 4 on (Air)
M58 Live tool 5 on (Air)

Electric Live Tools

This group controls Electric live tool functions, Depending on machine configuration these codes may vary. When only one electric tool is used the format is as follows:

M60 Turn live tool 1 on (Electric)
M61 Turn live tool 1 off (Electric)

If more than one electric live tool is used, a selector unit is installed. The first tool is started and stopped with the control box activate M code. For the 2^{nd} through 5^{th} tool you must first select the tool to use and activate the control box.

M60 Turn live tool 1 on or turn on tools 2 thru 5 (Electric)
M61 Turn live tool 1 off or turn off tools 2 thru 5 (Electric)
M50 Select live tool 2 on (Air)
M52 Select live tool 3 on (Air)
M54 Select live tool 4 on (Air)
M56 Select live tool 5 on (Air)

Miscellaneous Functions

Bar feeder codes

The second group controls bar feeder operation.

M70	**End of bar check on**	**M71**	**End of bar check off**
M72	**Feed bar on**	**M73**	**Feed bar off**
M74	**Load new bar start**	**M75**	**Load new bar end**

M76 thru M78 available for other bar feed functions

Automation Codes

The third group controls machine automation functions.

To turn off these codes use the next higher M code,

M80	**Operator door Close**
M82	**Part catcher open, or conveyor door open**
M84	**Conveyor on**
M86	**Robot activate**
M88	**Available code**

Miscellaneous Codes

The fourth group controls Machine Miscellaneous functions.

M90	**Advance sub tool slide**	**M91 Retract sub tool slide**
M92	**Airblow on (sub spindle)**	**M93 Airblow off**
M94	**Available**	
M96	**Available**	

Parametric or user defined Codes

The fifth group are M codes that can be used to call macros or predefined sub-routines.

These macros are stored in the “User subroutines” program.

To automate bar feeder operation some predefined M codes are used.

These are stored in the “Bar Feeder subroutines” program.

To edit these M codes, read the notes stored in the subroutines.

M96	**Home spindles command**
M98	**Tool change position (Use P299 for retract value)**
M99	**End of program position**
M100	**Automated bar feed for magazine applications (See embedded subroutines)**
M102	**Automated part unloading for conveyor & sub spindle applications**
M103	**Increment part counter when using loop commands**

This is a full list of available G codes for the Fagor 8055T. Not all codes are explained in this manual

G code	Description	Machine type
G00	Rapid travel	all
G01	Linear interpolation	all
G02	Clockwise circular interpolation	all
G03	Counter-clockwise circular interpolation	all
G04	Dwell/block preparation stop	all
G05	Round corner	all
G06	Absolute arc center coordinates	all
G07	Square corner	all
G08	Arc tangent to previous path	all
G09	Arc defined by three points	all
G10	Mirror image cancellation	all
G11	Mirror image on X axis	all
G12	Mirror image on Y axis	overhead cutoff only
G13	Mirror image on Z axis	all
G14	Mirror image in the programmed directions	all
G15	C axis activate	C axis only
G16	Selection of main plane in two directions	all
G17	Main plane X-Y and longitudinal Z	all
G18	Main plane Z-X and longitudinal Y	all
G19	Main plane Y-Z and longitudinal X	all
G20	Definition of lower work zone limits	all
G21	Definition of upper work zone limits	all
G22	Activate/cancel work zones	all
G28	Selection of the second spindle	FX
G29	Selection of the main spindle	FX
G30	Spindle synchronized offset	FX
G32	Feedrate "F" as an inverted function of time	all
G33	Thread cutting	all
G36	Automatic radius blend	all
G37	Tangential entry	all
G38	Tangential exit	all
G39	Automatic chamfer blend	all
G40	Cancellation of tool radius compensation	all
G41	Right-hand tool radius compensation	all
G42	Left-hand tool radius compensation	all
G50	Controlled corner rounding	all
G51	Look ahead	all
G52	Movement to hard stop	all
G53	Program coordinates with respect to home	all
G54	Absolute zero offset 1	all
G55	Absolute zero offset 2	all
G56	Absolute zero offset 3	all
G57	Absolute zero offset 4	all
G58	Additive zero offset 1	all
G59	Additive zero offset 2	all
G60	Axial drilling / tapping canned cycle	all
G61	Radial drilling / tapping canned cycle	all
G62	Longitudinal slot milling canned cycle	all

G63	Radial slot milling canned cycle	all
G66	Pattern repeat canned cycle	all
G68	Stock removal canned cycle along X axis	all
G69	Stock removal canned cycle along Z axis	all
G70	Programming in inches	all
G71	Programming in millimeters	all
G72	General and specific scaling factor	all
G74	Machine reference search	all
G75	Probing until touching	optional
G76	Probing while touching	optional
G77S	Synchronized spindle	FX
G78S	Synchronized spindle cancellation	FX
G81	Turning canned cycle with straight sections	all
G82	Facing canned cycle with straight sections	all
G83	Drilling canned cycle	all
G84	Turning canned cycle with circular sections	all
G85	Facing canned cycle with circular sections	all
G86	Longitudinal thread cutting canned cycle	all
G87	Face thread cutting canned cycle	all
G88	Grooving canned cycle along X axis	all
G89	Grooving canned cycle along Z axis	all
G90	Programming in absolute	all
G91	Programming in incremental	all
G92	Coordinate preset/spindle speed limit	all
G93	Polar origin	all
G94	Feedrate in millimeters (inches) per minute	all
G95	Feedrate in millimeters (inches) per revolution	all
G96	Constant Surface Speed	all
G97	Spindle speed in rpm	all

Preparatory Functions

G codes are the basic codes that command a CNC controller to move, or prepare the control for a type of move, or set the control in a mode to be able to interpret a method of operation.

Modal operation

The advantages of modal G codes is that you do not have to type or enter a repeated code on every line.

G00 X1. Z.1	**G00 X1. Z.1**
G01 Z0	**G01 Z0**
G01 X1.5 Z-.5	**X1.5 Z-.5**
G01 Z-2.	**Z-2.**
G00 X3. Z1.	**G00 X3. Z1.**

Both of these programs will operate exactly the same.

G00 or G0 Type format

The format for a G code can be G00 or G0 both will work with the controls.

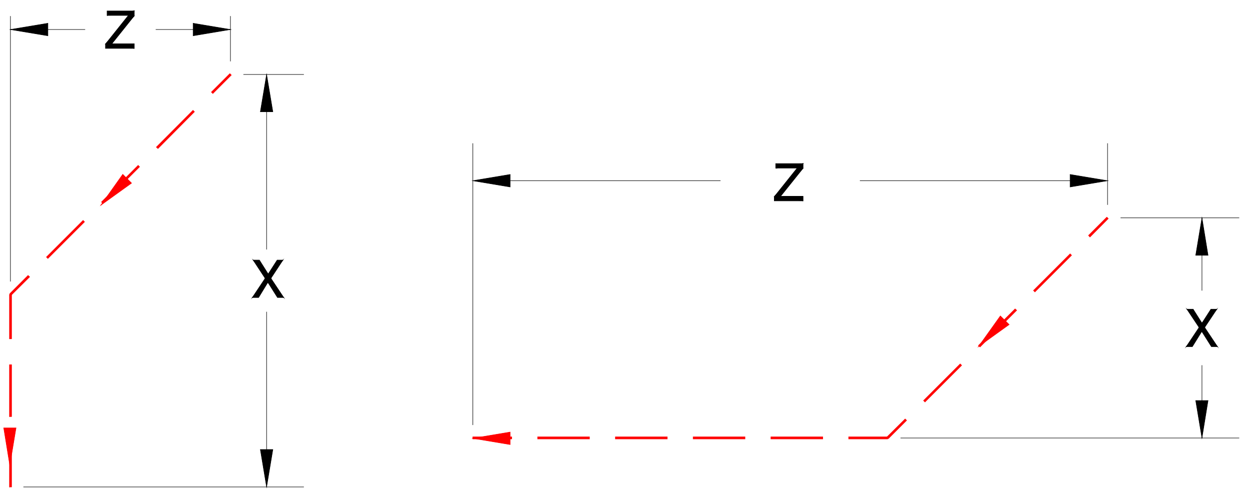
G00 Rapid Travel

This code causes machine motion to occur at the maximum traverse rate of the machine tool.

When only one axis is programmed the move will be a straight line, when both axis move simultaneously, each axis moves at its maximum speed, until one axis end point arrives at the programmed location. In the case of an X and Z rapid move, the axes would move along a 45 degree angle until either axis reached its programmed location. The other axis would continue to its programmed location.

In the figure below the left example shows a rapid move when the X axis move is greater than the Z axis move.

The figure on the right shows a rapid move when the Z axis move is greater than the X axis move.

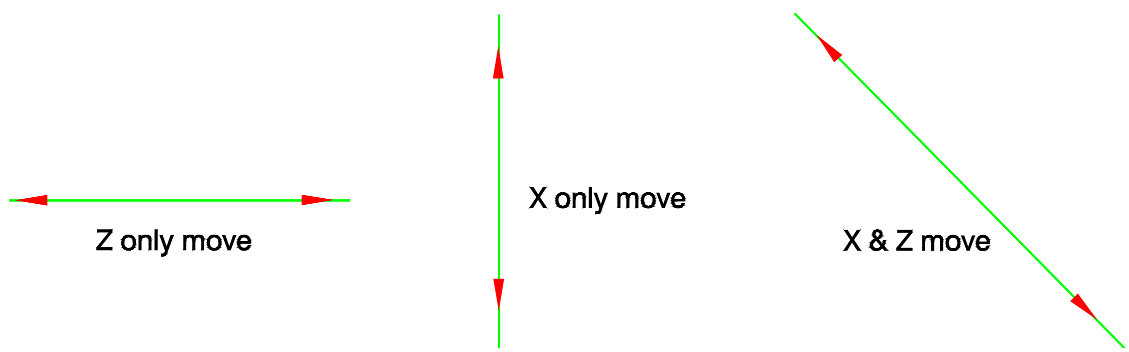


Rapid moves in X & Z that are unequal(fig 5.5)

G01 Linear Interpolation

A mode of movement in which, one or two axes move simultaneously to the programmed end point. The programmed feed rate in moves with X or Z motion are maintained along the line determined by the X and Z dimension words.

A single axis move moves the machine in one direction only. Either X or Z. These moves are considered axis parallel. When a taper is to be programmed both axis must be commanded.



G1 moves(fig 5.6)

G02 Circular Interpolation Arc Clockwise

A clockwise arc generated by the coordinated motion of two axes. The arc is defined as G2 when the tool motion, relative to the work piece, is clockwise. An arc is defined with the start point, the end point, the incremental distance from the start point to the center of the arc (I and K), and the direction of the arc, or an R value equal to the required radius.

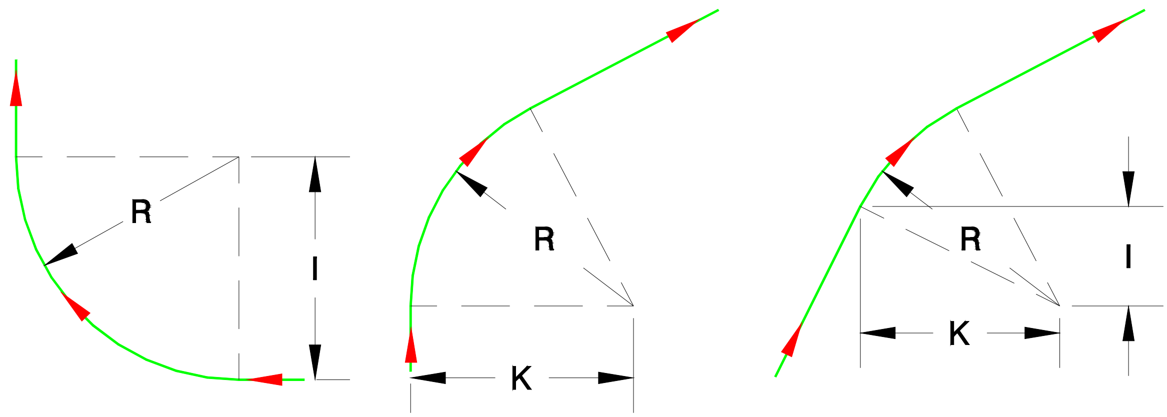
In the example to the left either an R value or an I value both equal to the radius would be programmed, K is zero and it is recommended that it is included in the line.

In the example in the middle either an R value or a K value both equal to the radius would be programmed, I is zero and it is recommended that it is included in the line.

In the example to the right either an R value, or both I & K values are required.

Remember I & K values are distances from the start position to the center of the radius and can be positive or negative.

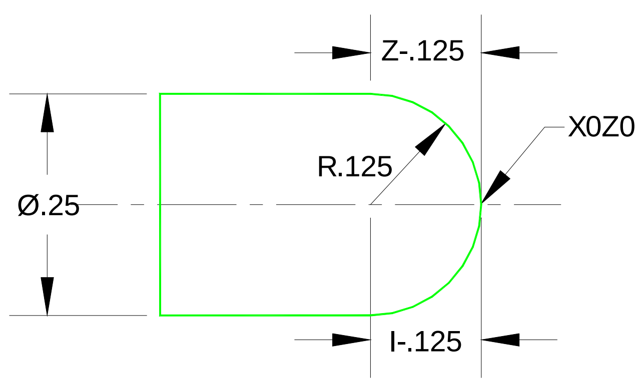
In G68 roughing cycles arcs must be programming using I & K



G2 Moves(fig 5.7)

Example of a 1/8" radius from the X0 Z0 centerline

N30 G2 X.25 Z-.125 R.125 or N30 G2 X.25 Z-.125 10 K-.125



I&K or R values(fig 5.8)

Note: Arc centers are defined using I as the X axis plane and K as the Z axis plane.arc centers are always incremental from the start point of the arc.

G03 Circular Interpolation Arc Counterclockwise

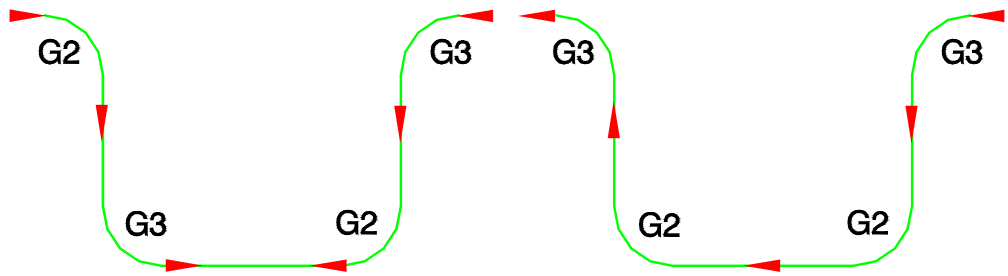
A counterclockwise arc generated by the coordinated motion of two axes. The arc is defined as G3 when the tool motion, relative to the work piece, is counterclockwise An arc is defined with the start point, the end point, the distance from the start point to the center of the arc (I and K), and the direction of the arc, or an R value equal to the required radius. The examples in G2 can be used to explain the I, K and R values.

Directions when programming arcs.

When programming external or internal arcs the basic rule of clockwise and counterclockwise always applies.

The same arc can be programmed with either G02 or G03 depending on the direction of the tool motion.

Grooving is a good example of this.



Different side approach direction Same direction approach (Fig5.9)

G04 Dwell

A timed delay of programmed duration. The time in milliseconds is coded with the K word. For example, G4 K200 is a time delay equal to 2 seconds.

G05 Block Deceleration checking off

Corner rounding is proportionate to speed. The faster the feed, the bigger the error at a change of direction will be. This is used to decrease the overall cycle time, by reducing error checking when accuracy is not need. (roughing, tool change movements etc.)

G07 Block Deceleration checking on

This G code forces the control to check for accurate positioning and ensures that an end point is reached before the control allows a change of direction. This format is the machine default, and should be used for all accurate machining.

G06-G08-G09 Refer to Fagor programming manual

G10 thru G14 Refer to Fagor programming manual

G15 C axis enable

Program G15 on its own to activate the C axis on "C" axis machines. Please refer to the section "C axis programming" for a complete description of this programming method.

G16 thru G22 Refer to Fagor programming manual

G28-G29 Spindle selection

These codes are used to select the 2 separate spindles on a GTS FX. Each spindle can be programmed independently. To specify the main spindle use G29. To select the sub spindle use G28. Please see G77S for synchronization information.

G30 Refer to Spindle synchronization section

G32 Refer to Fagor programming manual

G33 Refer to the section "Single point thread cutting." in this manual.

G36 Automatic Radius blend

G36/G39 Automatic corner radius & chamfering

G36 is a radius at the end of the cut,

G39 is a chamfer at the end of a cut.

This code saves programming time when specific radii or chamfers are desired. This can be used when non axis parallel moves are required.

Defining G2 or G3 and arc centers are not necessary to utilize this feature.

The format has to be as follows:

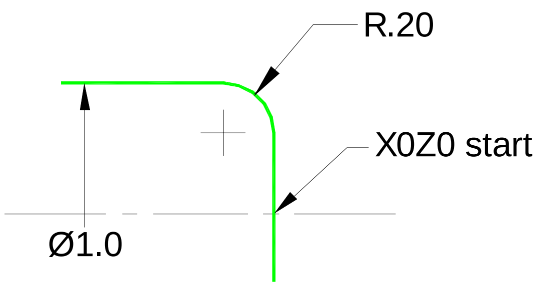
G1 G36 R?? X?? or Z?? or G1 G39 R?? X?? or Z??

If this format is not followed the control will not allow its input into a line.

The following examples show how this is used.

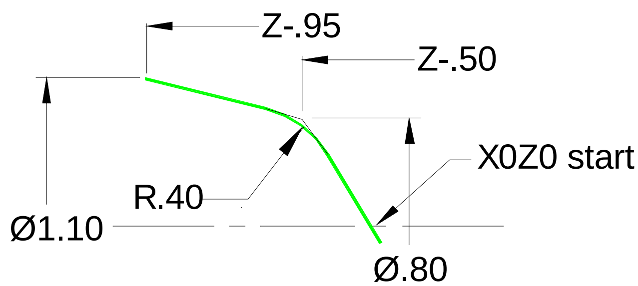
Lines only

Line to line moves can be in X,Z or both.



Example 1 line parallel to axis(fig 5.15)

N1 G90 G95 S1200 T1 M3
G00 X0
Z0
G01 G36 R.2 X1.0
G01 Z-1.

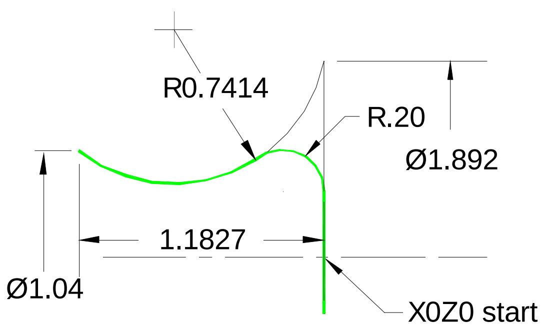


Example 2 lines non-parallel to axis(fig 5.16)

N1 G90 G95 S1200 T1 M3
G00 X0
Z0
G01 G36 R.4 X.8 Z-.5
G01 X1.1 Z-.95

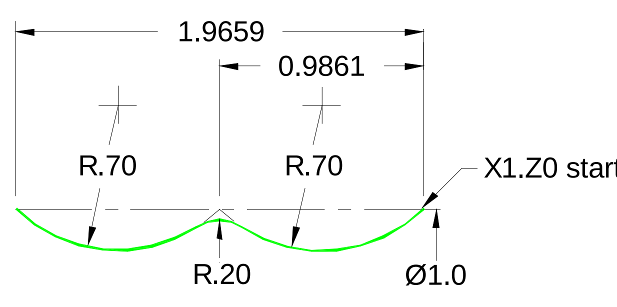
Arcs and lines, arc to arcs

Line to arc, and arc to arc moves can be in many configurations.



Example 3 line to arc (fig 5.17)

N1 G90 G95 S1200 T1.1 M3
G00 X0
Z0
G01 G36 R.2 X1.892
G02 X1.04 Z-1.1827



Example 4 arc to arc (fig 5.18)

N1 G90 G95 S1200 T1.1 M3
G00 X1.0
Z0
G02 G36 R.2 X1.0 Z-.9861
G02 X1.0 Z-1.9659

G39 Automatic chamfer blend

This feature can be used in a similar manner to G36 except that only lines may be chamfered.

G37/G38 Tangential approach and exit

This code allows facing to centerline or approaching centerline without back-cutting to remove centerline dimples that are sometimes left on a face. Using the code saves programming time.

G40/G41/G42 Tool nose radius compensation

These G codes allow the actual part profile to be entered as the program. Editing the tool table for radius, rather than changing the program can accommodate any variation in tool nose radius.

The main rules are that the proper tool location codes are installed on the tool-offset page and that the entry and exit moves are at least double the radius value entered in the table.
See Using and Calculating tool nose radius compensation for a full explanation of these codes.

G50-G52 Refer to Fagor programming manual

G53-G59 Work co-ordinate offsets

These are position offsets for moving the tool offset table by a fixed amount. They can be used for cutting multiple parts.

If a part is to be turned around in the collet and have a 2nd operation completed, the G55 offset could be used to provide a 2nd Z zero location for the 2nd operation. Without changing the tool offsets used for the 1st operation.

The G54 thru G57 offset are absolute offsets, G58 & G59 are additive offsets, and will add or subtract their value from a G54 thru G57 offset. These offsets remain active even after power off. The reference search reset the control to G53 (Cancel work offsets).

A positive offset moves the tool away from the spindle.
A negative offset moves the tool towards the spindle.

G60 thru G69 Canned cycles Refer to the section "Canned cycles" in this manual.

G70 Inch programming

Programming in inches. The machine is set-up in inch mode. This code is not required

G71 Metric programming

If you wish to program in millimeters, use G71 at the beginning of the program.

Please refer to the parameter tables and change the machine co-ordinate system to metric If you wish to have the machine display change also.

G72 Scaling

This code switches on scaling. Please refer to the Fagor programming manual for a full description of this powerful feature.

G74 Home search

This code sends the programmed axis to its home position.

Example: G74 X

G75-G76 Probing Refer to Fagor programming manual

G77S& G78S Synchronizing spindles

There are a number of requirements that have to be met for correct synchronization.
This command has to be programmed on a line by its self.
The recommended way to activate this feature is as follows :
Select the main spindle G29, Stop RPM on the spindle, issue a G77S, command an rpm & direction.
The Main & sub spindles will activate on this command.
If a Sub spindle speed & direction are active, they will be overridden by the G77S command.
When G78S is commanded the sub spindle will resume the previously commanded rpm & direction.

Notes:

- The maximum synchronized speed is 3500 rpm.
- M19 indexing is synchronized, but C axis is not.
- Offsetting of the spindles is allowed. (see below)

G30 D##.## is used to set an offset between the main & sub spindles, If live tools are to be used on the rear side, it is important to realize that, the orientation of front to back is achieved at handoff by setting G30 D to the required offset amount.
D is an angular value programmable to 2 decimal places.

G81-G89 Canned cycles Refer to the section “Canned cycles” in this manual.

G90 Absolute co-ordinates

This code is used for programming in the absolute program format.
(Please see machine co-ordinates).

G91 Incremental co-ordinates

This code is used for programming in the incremental program format.
(Please see machine co-ordinates).

G92 Maximum spindle speed clamping / Work reference point

This code is used for constant surface speed maximum RPM clamping.
If a surface speed of 600 SSFM(surface speed in feet per minute) is commanded for cutting but a maximum RPM of 1200RPM is required a G92 S1200 would be commanded on a line prior to the spindle speed command. (See G96)

G92 can be used to set a new zero position, for a work co-ordinate system.

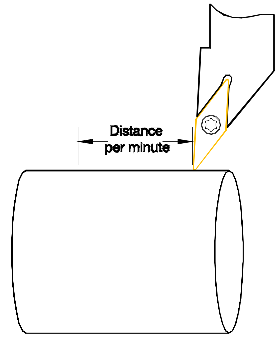
Axes X,Y,Z,& C are programmable

G94 Feed Rate Specification IPM, MMPM

This mode allows the feed rate to be specified by Millimeters Per Minute or Inches Per Minute. Normally this is used when no spindle speed is active and machine motion has to be controllable(Not rapid). Such as bar feeding.

This method of programming is the recommended method when using a live tool. No decimal point is allowed.

F80 = 8 inches per minute
F800= 80 inches per minute



Feed in inches per minute (G94)
(fig 5.11)

G95 Feed Rate Specification IPR, MMPR

This mode allows the feedrate to be controlled in relation to the commanded spindle speed.

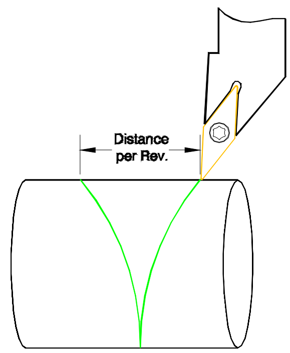
The feedrate is based on feed per revolution(FPR)

In inch mode the feedrate would be programmed as :

F.002(.002” per rev)

or in metric:

F.02(.02mm per rev).



Feed per revolution (G95)
(fig 5.12)

In order to program finer feedrates than the minimum allowed by G95, it may be necessary to switch to G94 and specify a small feed in Inch per minute, to achieve a similar slow feed rate.

Example: 3000rpm x 0.15” per minute =0.00001” per rev

F Feed rate address

This code is used to control how fast or slow the machine executes an axis move, depending on the G code selected this code can be in inches per revolution or inches per minute.

Feedrates are modal commands and once an amount is commanded, that feed rate is active until a new feedrate is programmed. This number can be up to 4 decimal places (F.0001)

Normally in lathe programming the setting is for inches per revolution.

Example: N1 G1 X.5 Z-.1 F.002 = .002”/rev

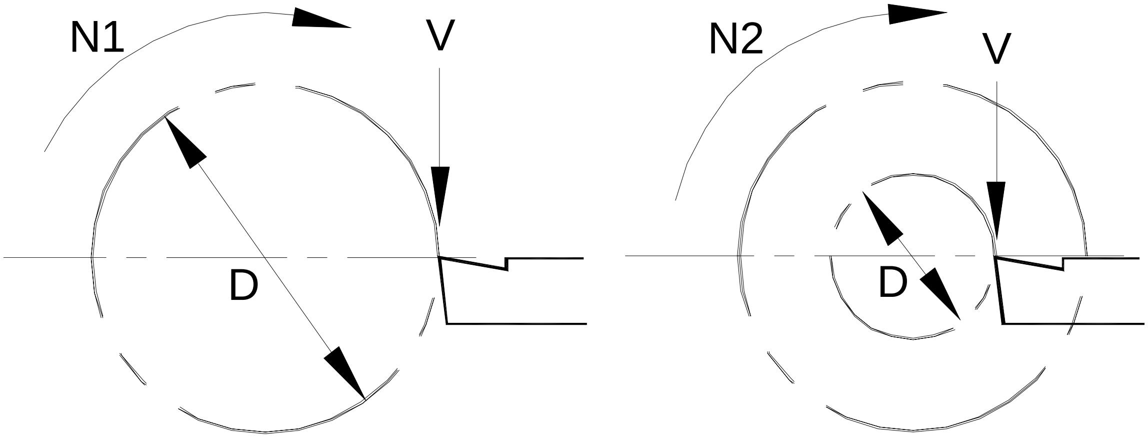
N2 G1 X.5 Z-.1 F.010 = .010”/rev

Line N1 commands a feed of .002 inches per rev.

Line N2 commands a feed of .010 inches per rev., This feed rate is 5 times faster than line N1.

G96 Constant surface speed mode (CSS)

This code controls cutting speeds base on surface speed and diameter being cut.
A given surface speed will produce a variable RPM depending on the diameter programmed.
The example below demonstrates this:



RPM based on surface speed (fig 5.10)

$$\textbf{RPM(N)} = \frac{\textbf{SSFM(V) * 3.82}}{\textbf{Work Diameter(D)}}$$

Given the same SSFM(V) 2 different diameters will give 2 different speeds in RPM

600 SSFM at a 2” diameter = 1146RPM
600 SSFM at a 1” diameter = 2292RPM

If a 2” part was to be faced to 1” diameter using CSS the machine would increase RPM from 1146RPM to 2292RPM gradually as the cut decreased in diameter.

If the same calculation is used on a .1 diameter you can see that the RPM exceeds the spindle capabilities of the machine.

$$\text{RPM} = \frac{\text{SSFM * 3.82}}{\text{Work Diameter}}$$

600 SSFM at a .1” diameter = 22,920 RPM

This is the reason why a G92 code is required when using CSS

Example G92 S4500 (Max speed preset)
G96 S250 M3 (SSFM and Spindle On)

G97 Constant Revolution per minute mode

This code allows programming of RPM as a fixed amount such as for threading, drilling, reaming, etc.
Example: G97 S750 M3 (750RPM, spindle forward)

Activating spindle orientation

To activate the spindle as a vector spindle capable of indexing issue a M19 to the control for each index you require. An S value is also required to specify where to index.

M19 S0 This will index to the marker pulse location
M19 S90 This will index clockwise to the 90^{O} position
M19 S-90 This will index counter clockwise to the 90^{O} position

In this example 3 holes are to be drilled at 120 degrees to each other.

The drilling moves are located in a sub program at N1000 to N1025

G94 feed in inches per minute is programmed as no spindle speed is being used.

```
%00101,MX,
N01 G90 G94 T7 M5 M8 ; 0.068 Cross Drill
G00 X0.6
Z0.1
M19 S60 ; 1st spindle position
G01 Z -1.0 F250 ; Index to 1st Position
(RPT N1000,N1001) ; Drill 1st hole
M19 S180 ; 2nd spindle position
(RPT N1000,N1001) ; Drill 2nd hole
M19 S300 ; 3rd spindle position
(RPT N1000,N1001) ; Drill 3rd hole
G00 Z2.0 M5
M30
N1000 G01 X0.1 ; Cross drill sub
G01 X0 F20.
G00 X0.1
X0
G01 X0.1
N1001 G00 X 0.6 F250. ; Safe index position
```

To program a spindle movement with an axis movement requires a full C axis spindle.

D Address

The D word is used for tool offsetting. Please refer to tool offsetting.

I & K Incremental value for position of radius center point from start point.

See the explanation of I and K in G2 Circular interpolation.

N Block numbers

Block numbering is optional. The 8055 control has an auto-numbering feature that will add numbering to new lines that are input.

P Address

The P word is used for parametric programming, and is the variable descriptor.

Q Address

The Q word is used for angular programming. Please refer to advanced program features.

R address

The R word is used for programming radius values in G2 and G3 and chamfer and radius values in G36-G39

S Spindle speed address

This a 4 digit number that is used to specify the spindle speed or surface speed required.

See G96 and G97 for specific details.

T Tool number address

This a number between T0 and T100 that is used to specify the tool required.

The format for this number is T000 The digits are for the tool offset.

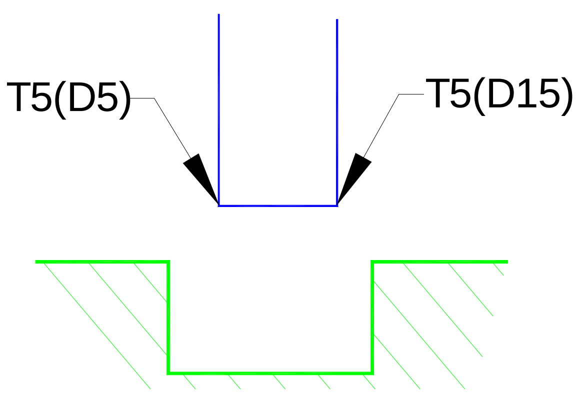
The tool table sets an offset value to the tool number using a D address, this address is linked to the tool number. T1 calls offset D1 etc, through T100

Using dissimilar tool offset numbers And D Offsets

These numbers do not have to correspond. On most machines 100 offsets are provided, This allows you to assign the extra offset values to multiple tool offsets.

Example : Give 2 offsets to a groove tool.

T5(D5) would be the left corner and **T5(D15)** would be the right corner. This allows the operator to adjust the width of a groove without changing the program.



(fig 5.13)

X & Z Absolute addresses

These addresses command axis movement in absolute mode. This is the recommended style of programming. X is a diameter designation and Z is Linear.

Section 6

Tool nose radius compensation

G40, G41, G42 Compensation codes

These 3 codes define how the machine calculates toolnose radius compensation (TNRC).

G40 Compensation code

G40 cancels the active G41 or G42 mode.

R and F Compensation registers

In the Tool Offset pages there are registers for X and Z, I and K, and R and F.

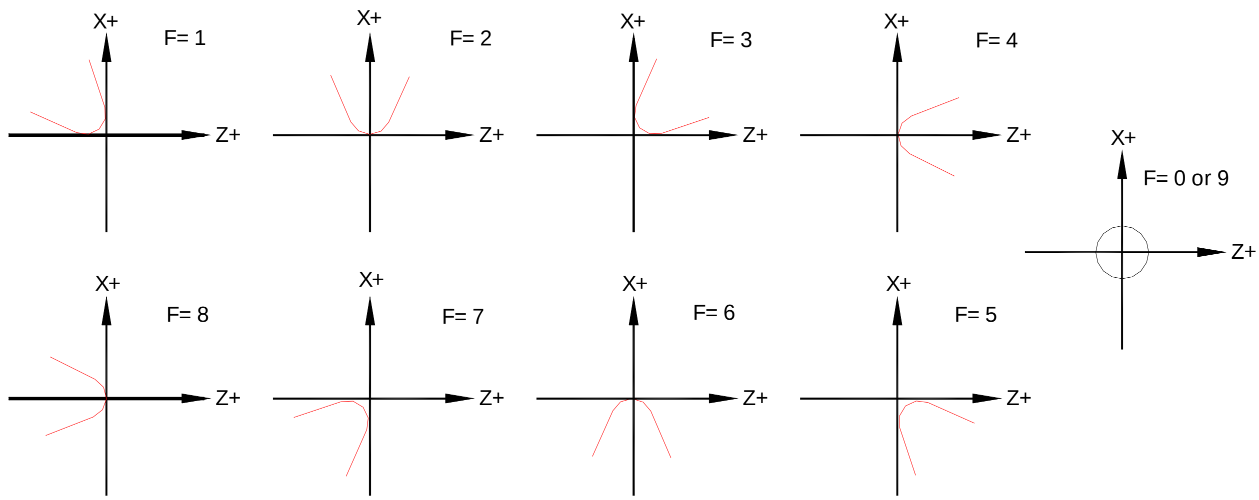
The R and F registers are critical to the correct function of TNRC.

R is the register for the radius of the tool tip. All commercially available tools have a radius, It may be as small as 0.004” or as large as 0.500”. This tool nose radius is the reason for TNRC.

The F register is used to help the control decide how to compensate for the radius. If the control knows how to apply the direction of compensation, the program will produce the correct shape.

Defining F register values

There are 9 initial values for the F number 0-9. (0 and 9 mean the same value, tool radius center)



Tool tip codes(fig 5.19)

These codes are defined by the X axis positive direction, If this is reversed the F code values change.

As figure 5.19 shows, the numbers correspond to a position around an XZ reference point.

In general use the tool codes would represent the following tool types.

- F1 would be the right side of a grooving tool or a back turning tool on X+
- F3 would be an outside turning tool on X+ or a boring bar on X-
- F4 would be a drill or facing tool with the center on the middle of the radius
- F5 would be a boring bar on X+ or an outside turning tool on X-
- F0 or F9 are used when programming the tool radius center.

Note:

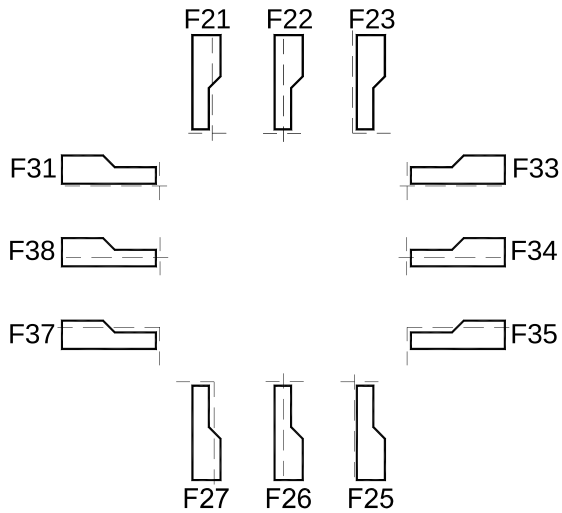
When threading you would not use TNRC and there for do not need an F code for the tool tip, It is a common mistake to apply an F2 or F6 to a threading tool tip.

Defining F register values for groove tools

The software for the 8055 controller allows the designation of groove tool edges.

These codes are intended for better visual aid in proofing programs, and for correct application in grooving and roughing canned cycles.

The controller is capable of compensating for both width and radius in roughing cycles.

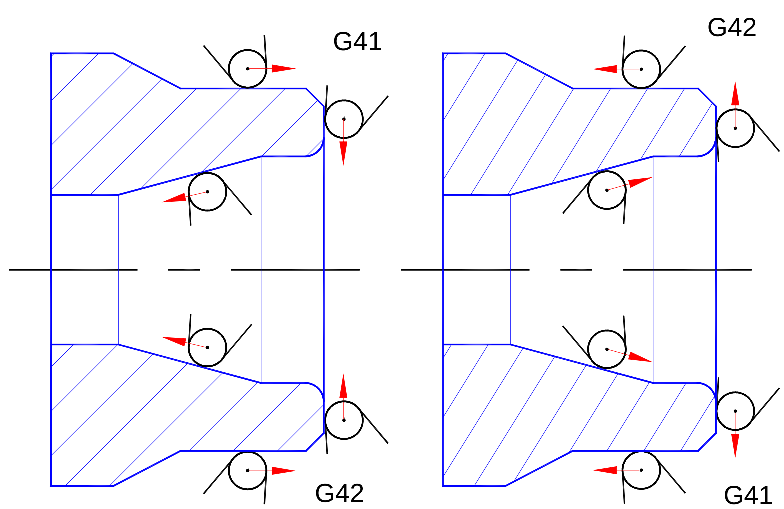


Compensation Directions

When machining a profile to the actual drawing dimensions the control has to calculate the amount of change that is required to compensate for TNRC. It also need to know on which side of the profile it has to compensate.

As can be seen in fig 5.20 profiling towards the chuck on the X+ side would require a G42 code with a value of F3, While boring on the X+ side would require a G41 code with a Value of F5.

If a profile was generated on the X- side the G41 would be used for an outside profile but would require a Value of F5, While a boring bar would require a G42 with a value of F3.



Compensation side (Fig5.20)

Entrance and exit amounts for TNRC

For TNRC to be active the controller requires that the approach distance be at least 2 times the radius value.

For a complete explanation of limits with TNRC please refer to the Fagor 8055 programming manual.

Please refer to the section " Manual tool nose radius compensation"

Section 7

Programming

Basic Format for a Program

Start-Up Block

A start-up block is recommended at the beginning of each tool cutting sequence. By programming each tool as a separate program, the tools can be easily adjusted or changed individually without running through the whole program.

A typical start-up block would be as follows:

N10 G90 G95 S2000 T1 M3 M8

N10	= Block #
G90	= Absolute coordinates
G95	= Feed in Inches per revolution
S2000	= Spindle RPM
T1	= Call up Tool #1
M3	= Spindle Forward
M8	= Coolant On

N	Block number (start with N1)
G	G codes or Prep Functions
C,X,Z,	Axis coordinates in that order
F	Feedrate
S	Spindle Speed
T	Tool Number
M	M Functions

Coordinate moves and feed rates are usually not included in the start-up block.

After the 1st block, the next block normally contains an axis movement.

It is strongly recommended that only an X axis move be programmed, On a Gang tooling lathe the main cause of tool crashes is a Z move after a tool clearance move.

Start-Up Block for constant surface speed

If Constant surface speed is to be programmed the 2nd line normally should be G92 and a maximum spindle speed.

Example:
```
N1 G90 G95 T1 M8
G92 S3500
G96 S500 M3
G00 X1.
```

A prior spindle speed can be programmed to start the spindle and decrease cycle time

Example:
```
N1 G90 G95 S2500 T1 M3 M8
G92 S3500
G96 S500
G00 X1.
```

Basic Format for a Program

Ending a tool sequence

An ending block is recommended at the end of each tool cutting sequence. By programming a safe rapid distance for the next tool so that it can rapid to its start point without any interference, and finally an M1 at the end of the sequence.

Example:

N50 G00 Z1.5
N60 M01

Ending a cycle sequence

At the end of a program cycle a different sequence is required. Normally turn the coolant and spindle off prior to the final line.

Example:

N50 G00 Z1.5 M09 M05
N60 M30

Active error checking

The controller will some times refuse to accept a block of code when using the editor, This is caused by a format error, either a bad or missing code or a block in the wrong order.
If this happens re read the line and see if it does not conform to the outline suggested.
The control will prompt for correct format by displaying the correct format on the lower line of the screen.

Generating programs

When programming it is possible to create a working program where all the moves are programmed line by line, every machine movement is commanded by either a G00, G01, G02, G03, G04 this is effective but will lead to long programs that require much thought and are difficult to modify.

Canned cycles are a way to simplify programming, by making routines that require a large amount of code and create a cycle using a fixed group of variables to do the same cycle. These are known as canned cycles.

In the Canned cycle section a brief overview of the most used cycles is presented.

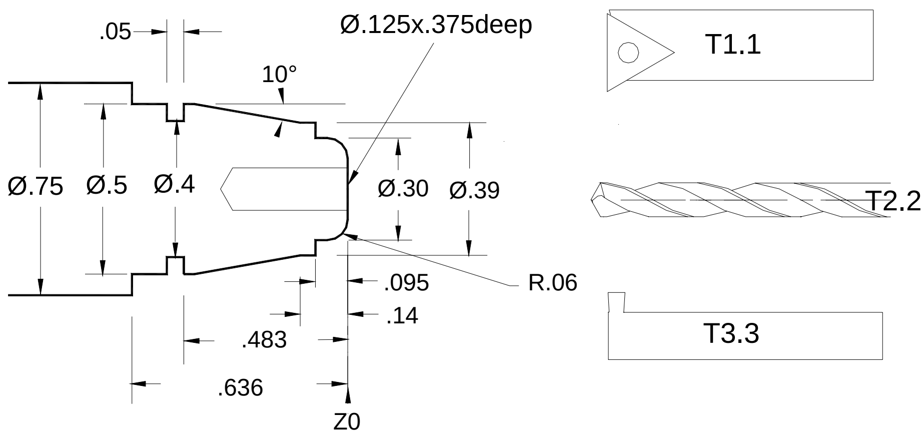
These do not present all the cycles available, but provide information on the most used.

Sample programs

In the following pages a sample program explains the main usage of different methods of generating a program, using both long code(No cycles) and canned cycles to simplify programming.

Long coding

This example shows a part generated with long code, using 3 tools.



Sample part drawing and tool list (fig 5.21)

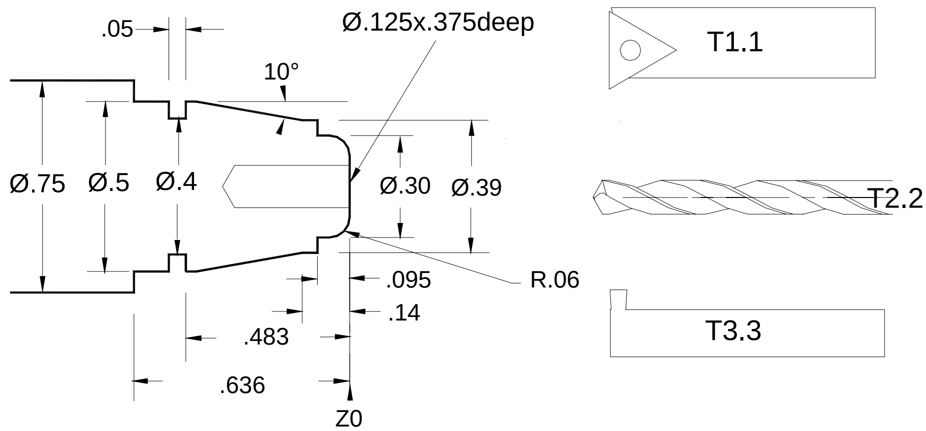
```
%000110,MX,
N1 G90 G95 S3000 T1 M04;TURNING TOOL
G00 X.77
G00 Z0
G01 X-.01 F.003
G00 X.52 Z.01
G01 Z-.634  F.006
G00 X.55  Z.010
G00 X.41
G01 Z-.092
G00 X.43 Z.01
G00 X0
G01 Z0 F.003
G01 X.18
G03 X.3 Z-.06 R.06
G01 Z-.095
G01 X.39
G01 Z-.14
G01 X.5  Z-.3119
G01 Z-.636
G01 X.75
G00 Z1.
M01
N2 G90 G95 S1200  T2 M13;0.125 DRILL
G00  X0
G00 Z.05
G01  Z-.125  F.003
G00 Z.1
G00 Z-.115
G01 Z-.250
G00 Z.1
G00 Z-.240
G01 Z-.375
G00 Z.1
G00 Z1.
M01
N3 G90  G95  S1000  T3  M13;0.05 WIDE GROOVE TOOL
G00  X.625
G00 Z-.533
G00 X.515
G01 X.400 F.0015
G00 X.515
G00 X.75  Z1.0 M15
M30
```

Explanation of sample program

%000110,MX,	
N1 G90 G95 S3000 T1 M14	Absolute IPR spindle 3000rpm Tool 1 direction counterclockwise
G00 X.77	Rapid feed to X+.77 diameter
G00 Z0	Rapid to face Z0
G01 X-.01 F.003	Facing cut to X-.01 at .003 IPR
G00 X.52 Z.01	Rapid to X+.52 Z+ .01
G01 Z-.634 F.006	Feed to Z-.634 AT .006 IPR
G00 X.55 Z.01	Rapid to X+.55 Z+.01
G00 X.41	Rapid to X+.41
G01 Z-.092	Feed to Z-.092
G00 Z.01	Rapid to Z+.01
G00 X0	Rapid to X0
G01 Z0 F.003	Feed to Z0 AT .003 IPR, beginning of finish pass
G01 X.180	Feed to X.18, start of radius
G03 X.3 Z-.06 R.06	Generate a .06 radius
G01 Z-.095	Feed to Z- .095
G01 X.39	Feed to X+.39
G01 Z-.14	Feed to Z-.14
G01 X.500 Z- .3119	Feed along 10^{O} taper
G01 Z-.636	Feed to Z-.636
G01 X.75	Feed off material to X+.75
G00 Z1.	Rapid to Z1., Safe position for next tool
M01	Optional stop
N2 G90 G95 S1200 T2 M13	Start-up block for tool #2 .125 Drill 1200 rpm clockwise
G0 X0	Rapid to X0
Z.1	Rapid to Z.1
G01 Z-.125 F.003	Feed to Z-.125 at .003 IPR
G00 Z.1	Rapid out to Z+.1
G00 Z-.115	Rapid into hole +.01 from last position
G01 Z-.250	Feed to Z-.250 at .003 IPR
G00 Z.1	Rapid out to Z+.1
G00 Z-.240	Rapid into hole +.01 from last position
G01 Z-.375	Feed to Z-.375 at .003 IPR
G00 Z.1	Rapid out to Z+.1
G00 Z1.	Rapid to Z1., Safe position for next tool
M01	Optional stop
N3 G90 G95 S1000 T3 M13	Start-up block for tool #3 Groove tool
G00 X-. 625	Rapid to X-.625
G00 Z-.533	Rapid to Z-.533, groove position
G00 X-.515	Rapid to safe position above .5 turned diameter
G01 X-.40 F.0015	Feed to X-.40 at .0015IPR
G00 X-.515	Rapid out of groove to X-.515
G00 X-.75 Z2.0 M15	Rapid to safe Z position turn off spindle and coolant
M30	End of program

Canned cycle coding

This example shows canned cycles and shortcut in code

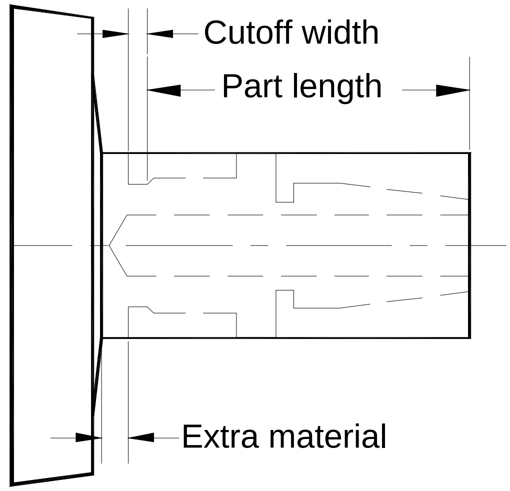



```
%00111,MX,
N1  G90  G95  S3000 T1  M14;Turning Tool 0 radius
G0 X.77
Z0
G1 X-.01 F.003
G0 X.76 Z.01 F.007
G68  X0  Z0  C.06  L.01 M.005 H0.004 S10 E20
(GOTO N21)
N10 G1 G36 R.06 X.18
Z-.095
X.39
Z-.140
Q170. X.50
Z-.636
N20  X.75
N21 G0 Z1.
M1
N2 G90 G95 S1200  T2.2  M13;.125 Stub Drill
G0  X0
Z.1
G1  Z.05  F.003
G83  X0 Z0 I.375 B0.1
G0 Z1.
M1
N3 G90  G95  S1000  T3.3  M13;0.05 Wide Groove Tool
G0  X.625
Z-.533
X.515
G1 X.4 F.0015
X.515
G0 X.6  Z1. M15
M30
```

Explanation of sample program

As can be seen 12 blocks of code were removed from the program, as well as repetitive G codes, refer to the canned cycles section for explanations of the 2 cycles used.

Variables related to Bar feeding



There are three variables related to barfeeding:

- Part Length
- Cutoff tool width
- Amount of material left out of collet

Auto Barfeeding with a Bar Pusher

Using a barstop, and assuming Z0 for the end of the part and no barfeed interface for magazine loading. The program would be formatted like this:

```
%000112,MX,
N10 G90 G94 T32 M5      Absolute, Inch/min, T32 offset and spindle stop
G0 X0                   Rapid to X0
Z-??                    Rapid to Part length
M11                     Open collet
G4 K75                  Dwell for ¾ seconds
G1 Z.015 F80.0          Feed bar out against stop at 80 IPM
M10                     Close collet
G4 K75                  Dwell for ¾ seconds
G0 Z1.                  Rapid out to a safe Z position
M1                      Optional stop
```

Bar pulling with a Bar Puller

A bar puller requires an extra variable:

- Amount of material to be gripped

Using a bar puller, and assuming Z0 for the end of the part. The program would be formatted like this:

```
%000113,MX,
N1 G90 G94 T32 M5       Absolute, Inch/min, T32 offset and spindle stop
G00 X??                 Rapid to X size allowing clearance for fingers over stock
Z.1                     Rapid to safe Z point
G01 Z-??                Feed to Z-(part length + grip distance)
X0                      Feed to X0 to grip bar
M11                     Open collet
G04 K75                 Dwell for ¾ seconds
G01 Z.015 F80.0         Feed bar out against stop at 80 IPM
M10                     Close collet
G04 K75                 Dwell for ¾ seconds
G00 X??                 Rapid out to safe X position
G0 Z1.                  Rapid out to a safe Z position
M1                      Optional stop
```

Section 8

Fagor 8055TG

Canned cycle programming

Most commonly used canned cycles

These are the commonly used canned cycles.

Please refer to the Fagor programming manual for a full list of cycles.

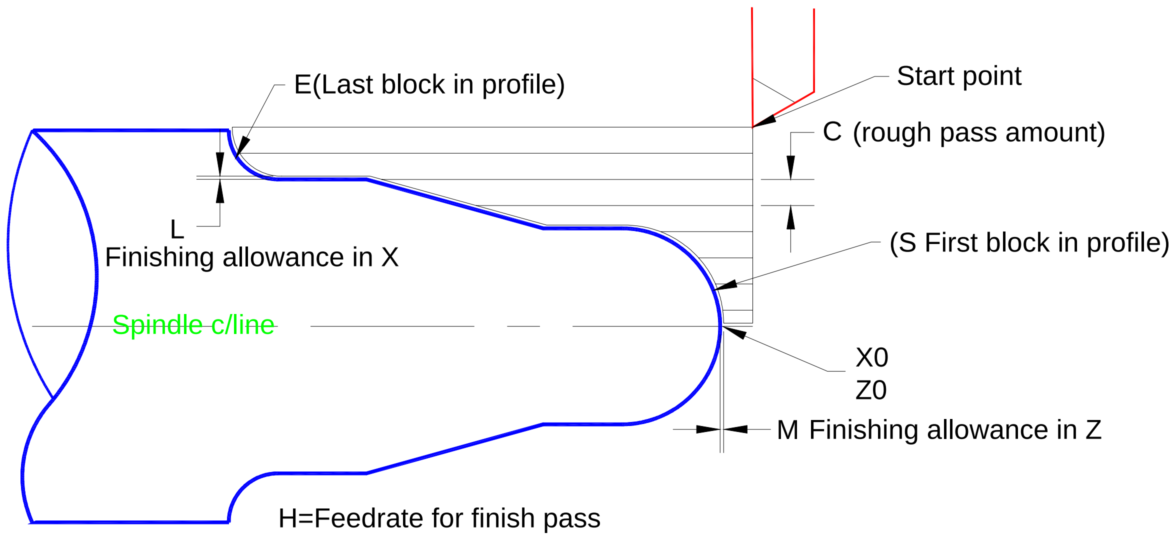
G68 Roughing in Z axis

G83 Hole drilling canned cycle

G86 Threading canned cycle

Note When entering a block containing a canned cycle a short cut is provided after entering the Gxx code. press soft key "help" then enter the values as you are prompted.

Always precede a canned cycle with a feed rate, this set the operating feed rate for the canned cycle.



Parameters for G68(fig 6.1)

Format:

G86 X Z C D K M L F H S E Q

Meaning of parameters:

X The start point of the final pass in X

Z The start point of the final path in Z

C The stock removed per pass

M The stock left on the X dia's for finishing (in radius)

L The stock left on the Z faces for finishing

H The feedrate of the finishing pass (if zero is entered no finish will occur)

S The first block of the profile. X and Z will put the tool in position to begin in the first block.

E The last block of the profile. the tool will automatically return to the start point when the cycle is complete.

Other Parameters

D Retract amount. If not programmed the cycle traces the profile

K If valleys are present in the profile K is the penetration feed rate.

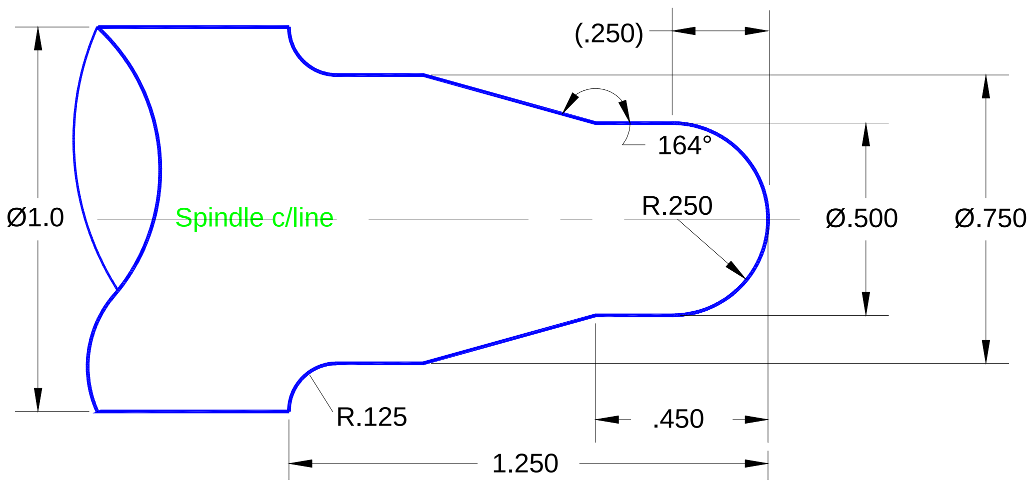
F If D is used and steps in the profile remain F is the semi-finish feedrate.

Q If a profile is not defined by S & E use Q for sub program #

Notes:

1. The profile can only use lines or arcs
2. Always position the tool outside the material (start point) before calling G68.
3. when using G41 or G42 the value of C (roughing pass) must be 2.1 times the tool nose radius. (in example above if tool has 0.03 radius then C must be at least 0.062)

Sample program

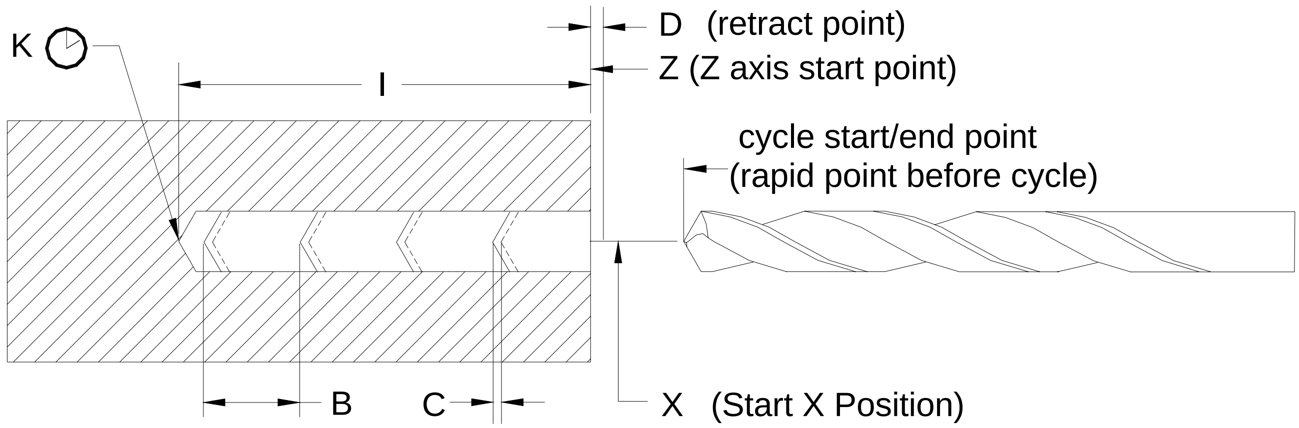


Sample profile(fig 8.2)

```
%000114,MX,
N1 G90 G95 S1000 T1 M3 M8
G42 G0 X1.1 Z0.1
G1 F.008 ;Feed Rate For Roughing
G68 X0 Z0 C.062 M.010 L.002 H.002 S2  E3
G40 G0 Z1.0  ;Clear Tool And Exit Tool Nose Radius Comp
X1.5 M5
M30
N2 G3  X.500  Z-.250  R.250 ;   Treat this final path as a subroutine.
G1  Z-.450
Q164.0 X.750  ; angular co-ordinates
Z-1.125
G2  X1.00  Z-1.25  R.125
N3 G1  Z-1.25
```

The parameters for the G68 cycle are:

X0	X0 start point of profile
Z0	Z0 start point of profile
C.062	0.062 depth of cut (Radius value)
M.010	0.010 stock on X diameters(Radius value)
L.002	0.002 stock on Z lengths
H.002	Finish feedrate for final pass
S2	N2 block start profile description
P3	N3 block end profile description



Hole Drilling Cycle(Fig 6.3)

Format:

N4 G83 X Z I B D H K C

Meaning of parameters:

X Absolute X value of the point where the drilling or circular groove is desired (if different from zero) in radius or diameters

Z Absolute Z value of the point where the drilling is to rapid to.

I Total depth of the hole. It will have positive value when drilled towards the negative direction of the Z axis and vice versa. If it is zero, error code 3 will be displayed.

B Max. peck size. The CNC will execute the minimum number of equal passes (B) until the total depth, defined by I, is reached.
If B is 0 the cycle becomes a tapping cycle and the spindle will reverse at depth and retract the tap.

D Safety distance. It defines the distance to the part from the point where the tool ends the positioning approach

H It indicates the incremental value of the G00 movement after each pass. If it is zero, this movement will be executed up to the start of drilling point. (D)
If it has a value the tool will rapid out of the hole by the amount given.

K Dwell. It identifies the value in seconds of the dwell at the bottom of the hole.

C It indicates the safety distance between the bottom of the previous penetration and the point where the tool ends the rapid approach for a subsequent penetration.

The machining conditions (feedrate, spindle rotation, etc.) must be programmed before calling the cycle. The parameters can be programmed in the calling block or in previous blocks.

- The cycle does not alter the calling parameters and thus they can be used for future cycles.
- The exit conditions are G00, G07, G40 and G90.
- The cycle starts with a G00 approach to point D and ends at D as well.

G83 Constant Depth Peck Drilling Cycle

G83 peck drilling with full retract

In this sample program, ZO(zero) is the face of the part and the final depth of the hole is 1.0 inches.

```
%000115,MX,
; Peck drill example
G90 G95 S1200 T12 M3 M8
G00 X0
Z.1 F.003
G83  X0  Z0  I1.0  B0.2 H0 C.01
G0 Z1. M9 M5
M30
```

G83 peck drilling with full retract & Dwell

In this sample program, ZO(zero) is the face of the part and the final depth of the hole is 1.0 inches.

```
%000115,MX,
; Peck drill example
G90 G95 S1200 T12 M3 M8
G00 X0
Z.1 F.003
G83  X0  Z0  I1.0  B0.2 H0 K0.5 C.01
G0 Z1. M9 M5
M30
```

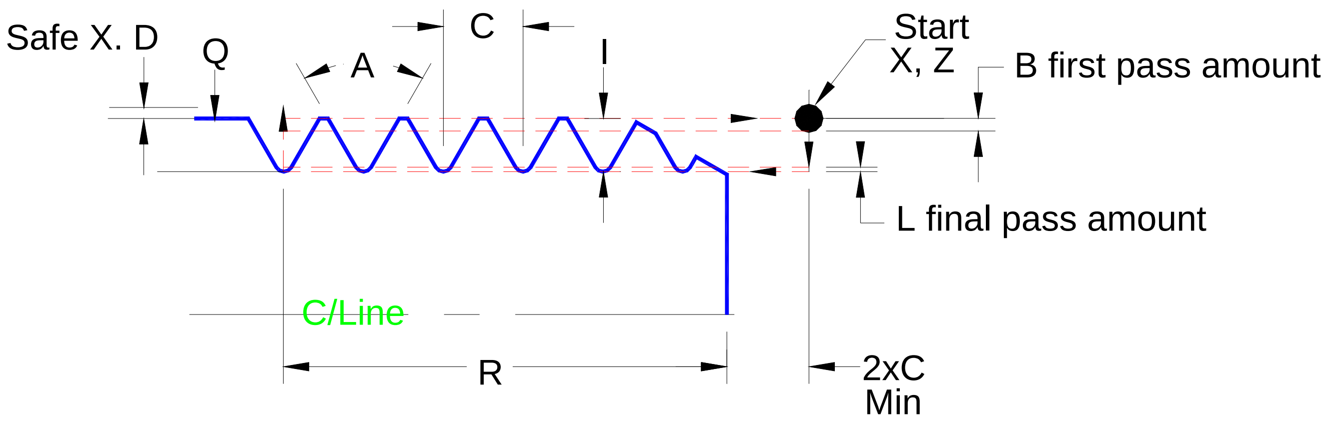
Adding the K value will dwell at the bottom of the hole for 0.5 seconds

G83 peck drilling with controlled retract

In this sample program, ZO(zero) is the face of the part and the final depth of the hole is 1.0 inches.

```
%000116,MX,
; Peck drill example
G90 G95 S1200 T12 M3 M8
G00 X0
Z.1 F.003
G83  X0  Z0  I1.0  B0.2 H0.1 C0.01
G0 Z1. M9 M5
M30
```

H has been changed to 0.1 this forces the control to retract the drill 0.1 out of the hole and then back to the last depth less the C value 0.01.



G86 Cycle Format(fig 6.6)

Parameter definitions

X	Major diameter at start of thread
Z	Start point in Z axis
Q	Major diameter at end of thread (See note 1)
R	Absolute position at end of thread (See note 2)
I	Total depth of thread per side (See note 3)
B	Depth of first pass and factor for remaining passes.(See note 4)
D	Safety distance for X axis retract at end point of thread
L	Final pass as a radius value.(See note 5)
C	Thread lead (Pitch)
J	Exit length
A	Included angle of threading tool.(See note 6)

Notes:

1. For straight threads use same value as X. For taper threads use different value in Q
2. Include sign for R minus (absolute position)
3. For external threads this will be a positive number. For internal threads it will be negative.
   (If the X value is negative reverse note 3)
4. If the sign is positive the control will calculate the remaining depths of cut.
   If the sign is negative the control uses this value for all cuts.
   If the value is zero an error will be issued.
5. If the value is zero the previous pass is repeated.
   (If the value is positive the pass will be infed at P12/2)
   If the value is negative the infeed will be a plunge pass taking equal amounts off each flank.
6. This value allows you to either infeed at an angle, or plunge into the thread taking equal amounts of both sides.(See compound infeed threading)
7. Spindle speed must be programmed prior to the cycle.
8. The feedrate override knob cannot adjust the feedrate during threading.

Thread a 1/2" x 13UNC thread 1" long

```
%000117,MX,
N10 ; 1/2x13UN thread
G90 G95 S1000 T1  M3 M8
G00 X-0.500 Z0.200
G86  X    = -0.500      (Major diameter at start of thread)
     Z    = 0.200       (Start point Z)
     Q    = -0.500      (Major diameter at end of thread)
     R    = -1.00       (Length of thread)
     I    = 0.0471      (Total depth of thread per side)
     B    = -0.010      (Depth of first pass and factor for remaining passes)
     D    = 0.020       (Safety distance at start of thread)
     L    = 0.0005      (Final pass)
     C    = 0.07692     (Thread lead)
     J    = 0.154       (Exit length)
     A    = 60          (Included angle of threading tool)
G00  Z2.0 M5 M9
M30
```

Notes:

A minor diameter is not programmed for the Fagor control. The final pass location is calculated from the start position in P0 and the Value of P4

MINOR DIAMETER = X-(B*2)

The rapid location is normally the major diameter.

If you are threading on the minus X(X-) side of the spindle the P4 value has to be - to ensure that you move in the correct direction.

Internal threading

The most common problem encountered with internal threading is the starting position programmed. It should be the minor diameter.

Example of a 1/2"x13UN internal thread

```
%000118,MX,
N10 ; 1/2x13UN internal thread
G90 G95 S1000 T1  M3 M8
G00 X-.416 Z.200
G86   X  =  -0.416    (Minor diameter at start of thread)
      Z  =  0.200     (Start point Z)
      Q  =  -0.416    (Minor diameter at end of thread)
      R  =  -1.00     (Length of thread)
      I  =  -0.0416   (Total depth of thread per side)
      B  =  -0.010    (Depth of first pass and factor for remaining passes.)
      D  =  0.020     (Safety distance at start of thread)
      L  =  0.0005    (Final pass)
      C  =  0.07692   (Thread lead)
      J  =  0.154     (Exit length)
      A  =  60        (Included angle of threading tool)
G00  Z2.0 M5 M9
M30
```

Section 9

General Threading information

General Information

This section is provided to help explain some of the questions when producing threads, The charts and tables are provided for reference only and *CMS* is not responsible for errors.

Maximum feedrate related to spindle speed

The feedrate for precision threading should be lead limited to 200 inches [5080 mm] per minute. Above this value, to the maximum machine feedrate of 600 inches per minute, the lead error should be checked to make certain it does not exceed specification for the individual thread being produced.

It is the programmer's responsibility to ensure that the combination of lead and spindle speed does not exceed a feedrate which produces threads that are not within drawing specifications.

The following chart shows the maximum lead in inches that can be cut with a given spindle speed without exceeding the maximum recommended feedrate of 200 IPM [5080 MMPM]. The values are derived from the formulas:

$$\text{LEAD (IPR)} = \frac{200\ \text{IPM}}{\text{RPM}} \qquad \text{LEAD (MMPR)} = \frac{5080\ \text{MMPM}}{\text{RPM}}$$

Spindle RPM	Maximum programmable Lead	
	Inches per Rev. (Threads per inch)	Millimeters per Rev.
50	2.0000 (0.25)	101.6
100	1.0000 (0.5)	50.80
150	0.6666 (.75)	33.86
200	0.5000 (1.0)	25.40
250	0.4000 (1.25)	20.32
300	0.3333 (1.5)	16.93
500	0.2000 (2.5)	10.16
750	0.1333 (3.75)	6.77
1000	0.1000 (5)	5.08
1500	0.0666 (7.5)	3.38
2000	0.0500 (10)	2.54
2500	0.0400 (12.5)	2.03
3000	0.0333 (15)	1.69
3500	0.0285 (17.5)	1.45
4000	0.0500 (20)	1.27
4500	0.04444 (22.5)	1.12

Thread depths for UN & UNJ Threads

This chart is provided to help speed up programming and give a reference to new programmers.

The values in thread depths can be used for address **I** in the G68 canned cycle.

The values in the Pitch column can be used for address **C** in single start threads.

		UN Threads		UNJ Threads	
Threads per inch	**Pitch (Feedrate)**	**External Thread depths**	**Internal thread depths**	**External Thread depths**	**internal thread depths**
4	0.250000	0.1578	0.1353		
5	0.200000	0.1262	0.1082		
6	0.166666	0.1052	0.0902		
7	0.142857	0.0902	0.0773		
8	0.125000	0.0789	0.0676		
9	0.111111	0.0701	0.0601		
10	0.100000	0.0631	0.0541		
11	0.090909	0.0574	0.0492		
12	0.083333	0.0526	0.0451	0.0504	0.0363
13	0.076923	0.0485	0.0416		
14	0.071428	0.0451	0.0386	0.0432	0.0311
16	0.062500	0.0394	0.0338	0.0378	0.0272
18	0.055555	0.0350	0.0300	0.0336	0.0242
20	0.050000	0.0315	0.0270	0.0303	0.0218
24	0.041666	0.0263	0.0225	0.0252	0.0181
28	0.035714	0.0225	0.0193	0.0216	0.0155
32	0.031250	0.0197	0.0169	0.0189	0.0136
36	0.027777	0.0175	0.0150	0.0168	0.0121
40	0.025000	0.0157	0.0135	0.0151	0.0109
44	0.022727	0.0143	0.0123	0.0137	0.0100
48	0.020833	0.0131	0.0112	0.0126	0.0090

Notes:

- For the bore of an internal thread, program the major diameter less 2 x the I value
- The feedrate for a thread is determined by dividing 1 by the pitch
- Internal UNJ threads have a larger minor diameter.

UN & UNJ External root radii values

This chart is provided to point out the differences between UN and UNJ minor root radii.

		UN External Threads		UNJ External threads	
TPI	**Pitch (Feedrate)**	**Min. Root Flat**	**Max. Root Radius**	**Min. Root Radius**	**Max. Root Radius**
80	0.0125	0.0016	0.0018	0.0019	0.0023
72	0.0139	0.0017	0.0020	0.0021	0.0025
64	0.0156	0.0020	0.0023	0.0023	0.0028
56	0.0178	0.0022	0.0026	0.0027	0.0032
48	0.0208	0.0026	0.0030	0.0031	0.0038
44	0.0227	0.0028	0.0033	0.0034	0.0041
40	0.0250	0.0031	0.0036	0.0038	0.0045
36	0.0278	0.0035	0.0040	0.0042	0.0050
32	0.0312	0.0039	0.0045	0.0047	0.0056
28	0.0357	0.0045	0.0052	0.0054	0.0064
24	0.0416	0.0052	0.0060	0.0063	0.0075
20	0.0500	0.0063	0.0072	0.0075	0.0090
18	0.0556	0.0069	0.0080	0.0083	0.0100
16	0.0625	0.0078	0.0090	0.0094	0.0113
14	0.0714	0.0089	0.0103	0.0107	0.0129
13	0.0769	0.0096	0.0111	0.0115	0.0139
12	0.0833	0.0104	0.0120	0.0125	0.0150
11	0.0909	0.0114	0.0131	0.0136	0.0164
10	0.1000	0.0125	0.0144	0.0150	0.0180
9	0.1111	0.0139	0.0160	0.0167	0.0200
8	0.1250	0.0156	0.0180	0.0188	0.0226

Notes:

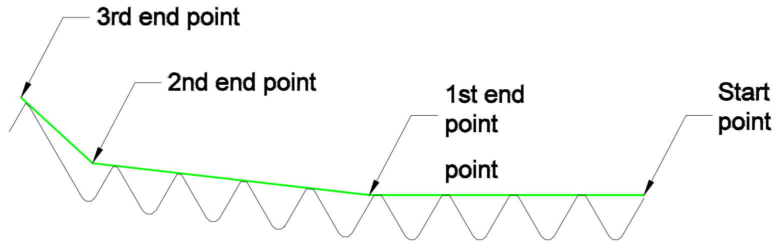
- These values are useful in determining the correct feedrate and insert shape for the threading of these threads.
- Most insert manufacturers make inserts for these threads, and it is recommended that the correct insert for each thread is used.
- Full profile inserts provide the correct radii and top profile for a given UNJ thread.
- Refer to MIL-S-8879A for a full specification on this critical threading application.

How the machine can repeat a thread

All lathes feature an accurate encoder that is geared to the spindle through a timing belt. The encoder monitors RPM during a threading pass and when feeding in Inches/mm per Revolution (G95) The encoder sends data controlling X&Z Axis velocity, as determined by the X and Z distance commands, to the control system.

With the Single Block Thread cutting feature, the programmer can cut a thread in any desired number of passes using the G33 preparatory command.

- The G33 command controls single direction per block moves.
  These moves can be straight or tapered. Another block is programmed to retract the tool tip out of the cut and another block to return it to its start position.
- G33 blocks programmed consecutively can be used to change thread direction.(fig 7.1).
  (use G5 for continuous block processing)



(Fig 7.1)

Pitch and Lead

Thread pitch is the axial distance from the center of one thread to the center of the next. Lead is the distance the screw will advance when turned one revolution. On a single thread screw, the pitch and lead are equal since a screw will advance an amount equal to the pitch when turned one revolution. On a double thread the screw will advance two threads or twice the pitch in one revolution. Therefore, the programmed lead is twice the pitch.

Program the spindle speed for a threading operation in a block of data preceding the thread cutting calling block (G33). This will allow time for the spindle speed to stabilize before entering the thread cutting mode.

The % Feedrate Override switch is not active during a G33 G68 thread cutting pass unless it is set to 0%. When the % Feedrate Override switch is set to 0%, axis motion WILL STOP at the end of the single cycle.

The Spindle INCREASE and DECREASE knob is active.
FEED HOLD is not active during the thread cutting pass, but is active on the return pass.

Acceleration and deceleration of machine axis

For accurate thread leads it is essential that the per revolution feedrate of the tool is held constant during the threading pass. The location of the start point for each threading pass is important in that sufficient distance must be provided to accelerate the tool from its Z axis velocity at the end of the infeed to the proper threading velocity.

Due to the nature of the servo-controlled axis drive system, provide a minimum of three pitches or 0.250 inch, whichever distance is greater, between the first thread to be cut and the start point for the threading pass. (this amount can be cut to 1.5 pitches if a slower RPM is used).

The X axis start point should be equal to the diameter of the work piece plus an amount of clearance for cycle retract, normally 0.02”-0.04” (The control will use the D address if the major diameter is the start point.

Compound infeed and acceleration

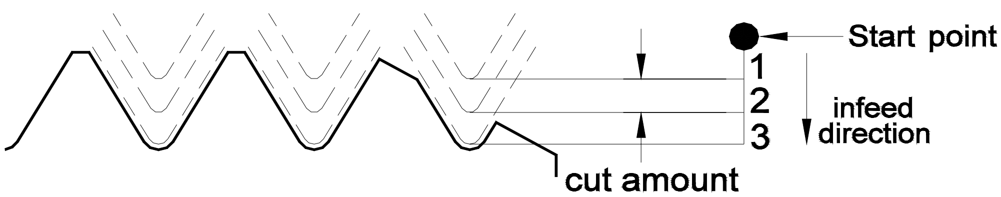
This minimum clearance must be provided for all threading passes. If a compound infeed is used, (See “Compound Infeed Threading”) work backwards to calculate the start point for the cycle. Beginning with the last threading pass, calculate the Z axis motion during infeed for the first pass. Add this distance to the Z axis clearance (three leads or 0.250 inch whichever is greater). This gives the Z axis position of the start point for the cycle relative to the first thread to be cut.

Multi-start threads

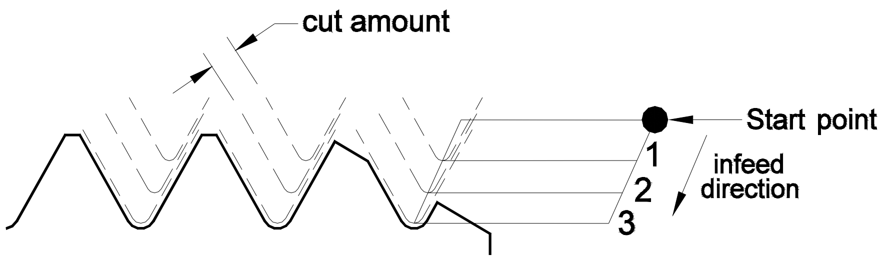
When threading multi start threads the canned cycle start point sets the location for the other passes, each new “start” must move away from the original position by the pitch amount.

Plunge and compound infeed

When machining a material that presents threading difficulties due to its toughness or when cutting a coarse thread of extreme depth, it is often desirable to infeed the tool so that the leading edge of the tool cuts the major portion of the material. This reduces deformation of the tool nose due to pressure and heat, thus adding to the tool life. To accomplish this the Z axis position of the tool at the start point of each pass is altered by in-feeding both the X and Z axes to produce the desired infeed angle.



Plunge Infeed
(Fig7.2)



Compound Infeed
(Fig 7.3)

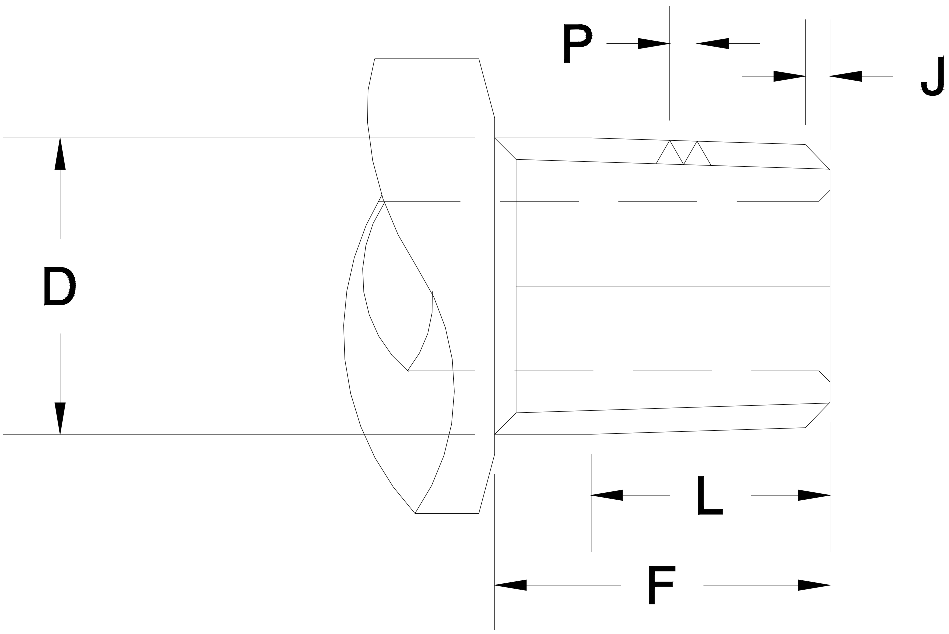
During a compound infeed, (fig 7.3), the tool moves along the X and Z axes from the start point of the threading cycle to the start point for the current threading pass. The infeed can be any angle relative to the face of the part up to 1/2 the included angle of the thread. Axis travel during infeed is as follows: Let X equal the distance traveled along the X axis during infeed. The corresponding distance traveled along the Z axis is equal to the TAN X of the required angle in degrees.

Overview of pipe thread sizes

The sample programs in this section are derived from MS33677 for hexagon bar stock

Material is assumed to be steel.

All ANPT pipe threads are the same taper (1.78^{O}) and this is the basic recalculation in these programs.



Pipe thread variables(Fig 7.4)

Pipe Size	Thread Size	D Min Dia	F +/-.015	J +/- .016	L Effective length	P Pitch
1/16	1/16 x 27	.312	.391	.031	.2611	.037037
1/8	1/8 x 27	.405	.391	.031	.2639	.037037
1/4	1/4 x 18	.540	.594	.047	.4018	.055555
3/8	3/8 x 18	.675	.609	.047	.4078	.055555
1/2	1/2 x 14	.840	.781	.062	.5337	.071428
3/4	3/4 x 14	1.050	.797	.062	.5457	.071428
1	1.00 x 11-1/2	1.315	.984	.078	.6828	.086956
1-1/4	1.25 x 11-1/2	1.660	1.016	.078	.7068	.086956
1-1/2	1.50 x 11-1/2	1.900	1.031	.078	.7235	.086956
2	2.00 x 11-1/2	2.375	1.062	.078	.7565	.086956
2-1/2	2.50 x 8	2.875	1.562	.109	1.1375	.125000
3	3.00 x 8	3.500	1.625	.109	1.2000	.125000

MS33677 Pipe Chart provided for reference only

Left and Right Hand Threads

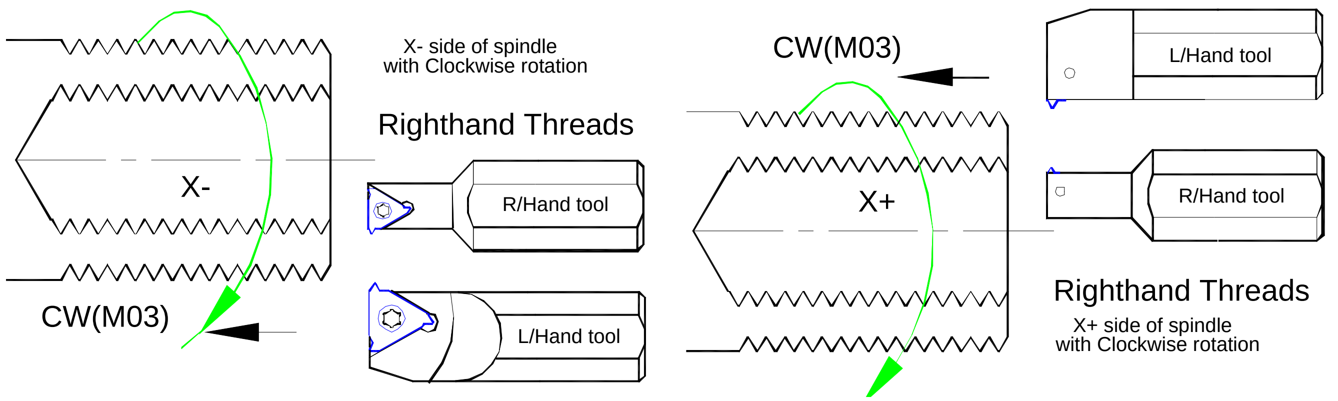
Tool hand and direction factors when threading

Gang tool lathes allow you to work on both sides of the spindle, this feature has to be considered when producing threads. The following examples show the variations for different choices.

Internal and External threads

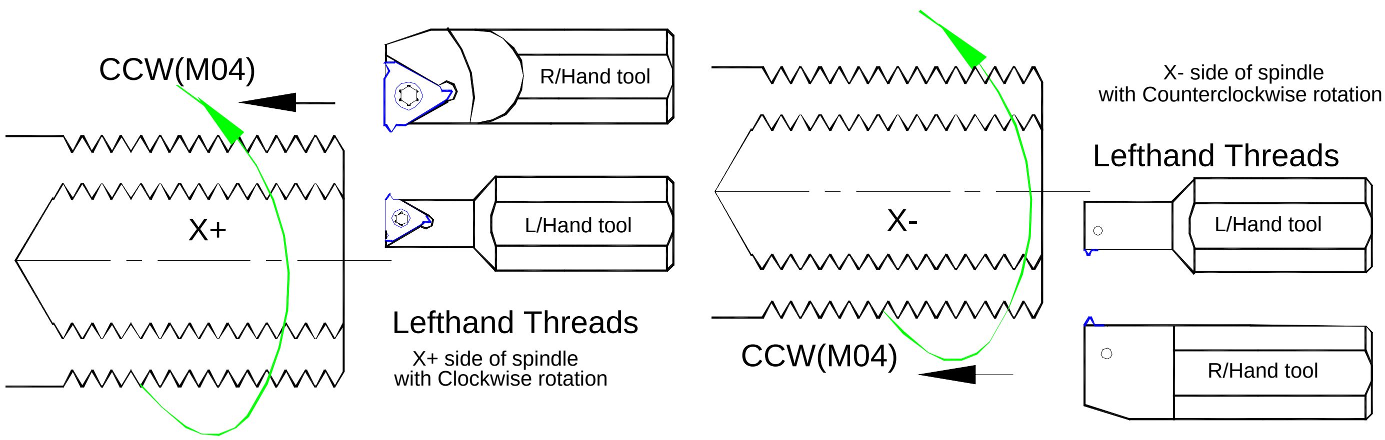
See the figure below to explain these choices.

Right hand threads



Right hand thread cutting(fig 6.26)

Left hand threads



Right hand thread cutting(fig 6.27)

Section 10

Trigonometry For Programming

Triangle Calculation

When programming CNC controls, at some point, a calculation of a triangle will be necessary.

This could be a simple taper, or an intersecting radius. The rules of trigonometry help us find a solution to these problems.

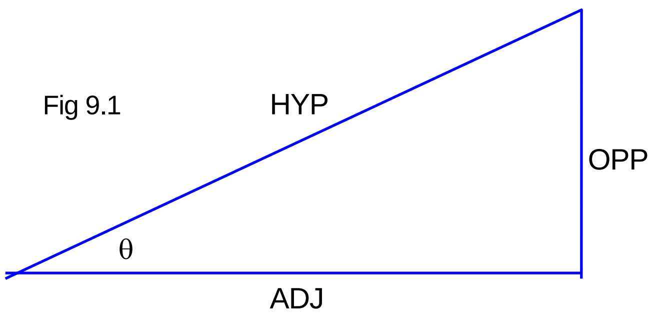
The following chapter is designed to help a beginning programmer understand the basics of these trigonometry rules.

Some program calculations are based on right angles triangles, These calculations can appear in many forms. To simplify the solution of these triangles a basic overview is outlined below.

Right Angle Laws

1. One angle is always 90^0
2. The sum of the 3 angles in a triangle always equal 180^0
3. If one side and one angle are known, the triangle can be solved.
4. If 2 sides of a triangle are known, the triangle can be solved.
5. A triangle consists of 3 angles and 3 sides.
6. If the angle(θ) and one side is known any other angle or side can be found.
7. If two sides are known the angle and third side can be found.

Fig 9.1



Sin θ = Opp ÷ Hyp

Cos θ = Adj ÷ Hyp

Tan θ = Opp ÷ Adj

$Hyp^2 = Opp^2 + Adj^2$

No.	To Find	Known	On Scientific Calculator
1	θ	Hyp & Opp	Opp ÷ Hyp = Arc/Inv Sin
2	θ	Hyp & Adj	Adj ÷ Hyp = Arc/Inc Cos
3	θ	Opp & Adj	Opp ÷ Adj = Arc/Inv Tan
4	Opp	θ & Hyp	θ Sin x Hyp =
5	Opp	θ & Adj	θ Tan x Adj =
6	Opp	Hyp & Adj	Hyp x^2 - Adj x^2 = √
7	Adj	θ & Hyp	θ Cos x Hyp =
8	Adj	θ & Opp	Opp ÷ (θTan) =
9	Adj	Hyp & Opp	Hyp x^2 - Opp x^2 = √
10	Hyp	θ & Opp	Opp ÷ (θSin) =
11	Hyp	θ & Adj	Opp ÷ (θCos) =
12	Hyp	Opp & Adj	Opp x^2 + Adj x^2 = √

Note: Nos. 1,2,3. Arc/Inv. Use whichever is shown on you model calculator

Example 1

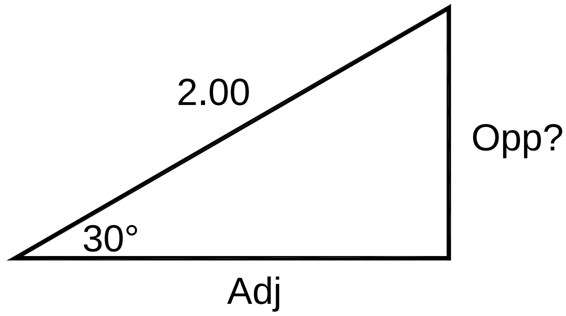


Fig 9.2

To find the side OPP of sample 1. We look down the second column (To Find) and we see lines 4,5, and 6 contain OPP. We then look in the third column (Known) to see which of lines 4, 5, and 6 contain the remaining 2 elements (θ and HYP).

This narrows our search down to line 4.

On our calculator we enter the angle θ(30^o), press SIN, press the multiply sign X enter the HYP (2), press =. We can now read off the value of the OPP which is 1.0

OPP = 30 SIN x 2.0 = 1.00

Example 2

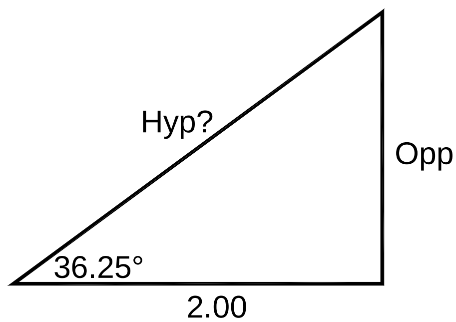


Fig 9.3

To find the HYP of your sample 2. We look down the (To Find) column and we see lines 10,11 and 12 contain HYP. We check the (Known) column to find the other two elements (θ and ADJ).

This narrows our search down to line 11.

On the calculator enter the ADJ(2), press ÷, press(enter the angle θ)36.25, press COS, press), press =

We can now read off the value of the HYP which is 2.4800

HYP= 2÷(36.25 COS)=2.4800

Example 3

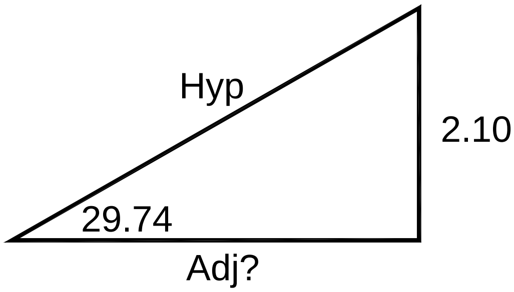


Fig 9.4

To find the **ADJ** of our sample 3. We look down the (To Find) column and see lines 7,8 and 9 contain **ADJ**. We check the (Known) column to find the other two elements (θ and OPP).

This narrows our search down to line 8.

On our calculator we enter the length (OPP) 2.1,press ÷, press 29.74, press TAN, press) press =.

We can now read off the value 3.6756

ADJ = OPP ÷ (θTan) = 3.6756

NOTE:

On modern calculators it may not be necessary to use the() keys since these calculators are capable of selecting mathematical priorities automatically.

You will notice a few "tenths" error in the results. This is due to rounding off the angles to 2 decimal places used in the calculations, the more decimal places used in calculations, the more accurate the results.

Creating Decimal values for Degrees

On most scientific calculators an angle must be input as a decimal.
On newer calculator a Degree, Minute, Second. to Decimal Degree function is included.
If this is not available use this method to convert.

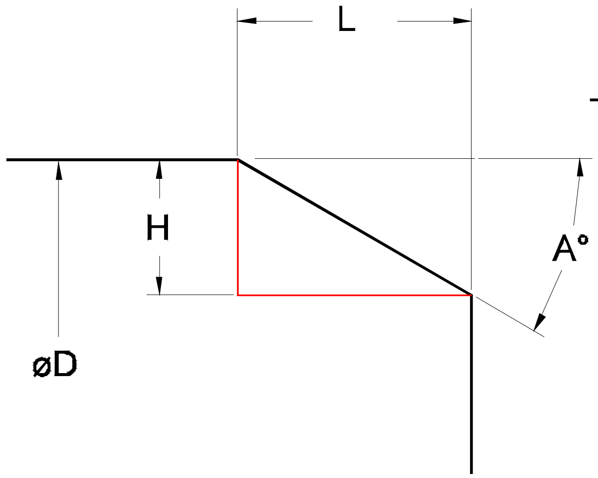
30^0 27' 18" **= 30.0 + (27/60) + (18/3600)**

= 30.0 + (0.45) + (0.005)

= 30.455^0

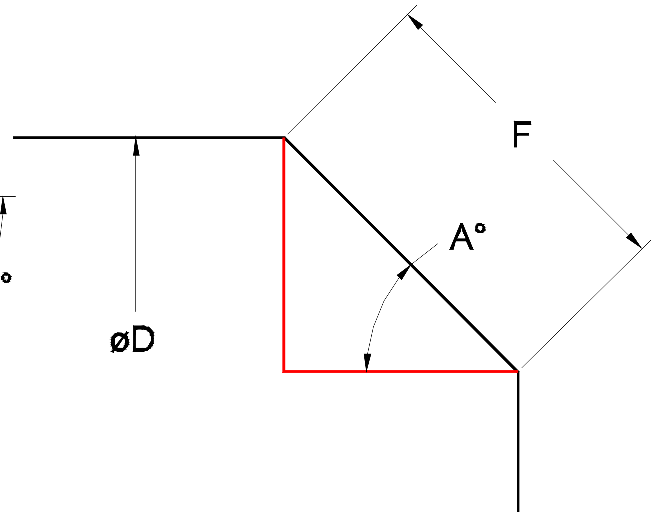
Taper and chamfer calculations

In lathe programming a typical taper is defined by a selection of methods. The most common being described below.



A taper and its variables

Fig 9.5



A chamfer and a method of dimensioning

Fig 9.6

To solve for the start and end points, Triangular calculations must be made, and then added or subtracted from the Absolute co-ordinates of the part.

In the examples above a diameter D is given as the absolute diameter. The Z position could be set by the vertical line on the right of the figures.

Fig 9.5 shows a typical taper. Either the height (H) or the length (L) and normally an angle (A) are provided.

Fig 9.6 shows a corner break that has the hypotenuse (F) and an angle (A) provided.

Example 4

We can apply some values to Fig 9.5

The triangle is solved by
OPP = 30 TAN x ADJ
OPP = 30 TAN x 0.25
OPP = 0.57735 x 0.25
OPP = 0.1443

Start position = 1.6 - (.1443 * 2)
= 1.6 - .2886
= 1.3114

NOTE:

The value of **OPP** has to be multiplied by 2 to allow for both sides of the part in diameter style programming.

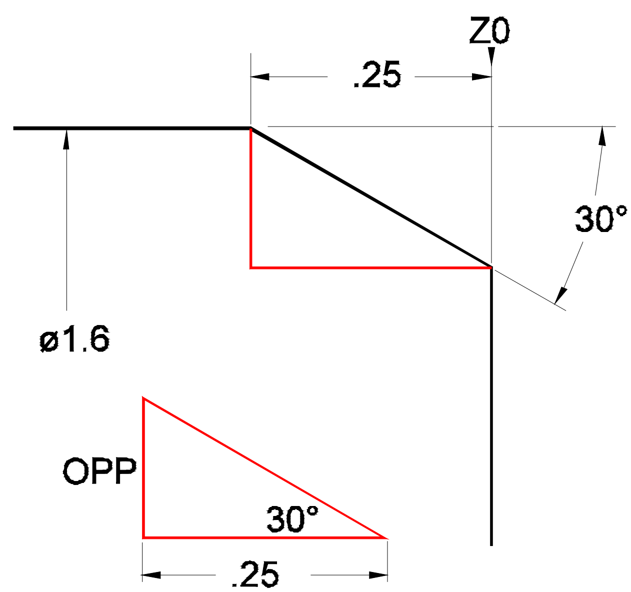


Fig 9.7

Example 5

We can apply some values to Fig 9.6

The triangle is solved by
OPP = 45 SIN * HYP
OPP = 45 SIN * 0.25
OPP = 0.7071 * 0.25
OPP = 0.1767

Start position = 1.6 - (.1767 * 2)
= 1.6 - .3534
= 1.2466

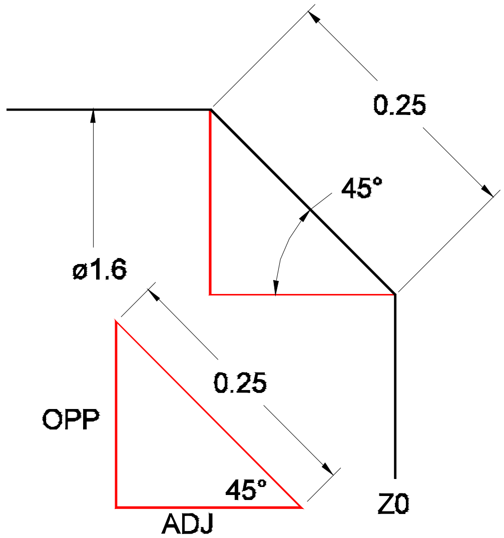


Fig 9.8

NOTE:

The value of **OPP** has to be multiplied by 2 to allow for both sides of the part in diameter style programming.

PYTHAGORAS THEOREM

Pythagoras theorem states that the square of the hypotenuse is equal to the sum of the square of the other two sides.

This is used for calculating the length of any side of a Right Triangle when the other two sides are known.

It is frequently used for double checking Right Triangle calculations when SIN, COS, and TAN have been employed.

$HYP^2 = OPP^2 + ADJ^2$

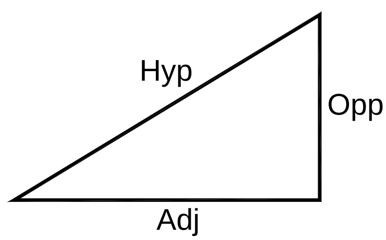


Fig 9.9

Example 6

$1.45^2 = ADJ^2 + .75^2$
$ADJ^2 = 1.45^2 - .75^2$
$\sqrt{ADJ^2} = \sqrt{1.45^2 - .75^2}$
$ADJ = \sqrt{2.1025 - .5625}$
$ADJ = \sqrt{1.54} = 1.2409$

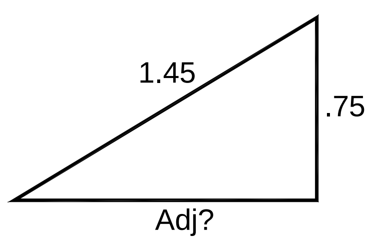


Fig 9.10

To find ADJ we look down the (To Find) column and we see line 7, 8 and 9 contain ADJ. We check the (Known) column to find the other two elements (HYP and OPP).

This narrows our search down to line 9.

On our calculator we enter the HYP(1.45), press the X^2 key. press -, enter the OPP(.75), press the X^2 key, press =. press the √ key. We can now read off the value 1.2409.

Angular Relationships with Tapers and Radii

When a radius breaks an intersection of a taper on a turned part, a relationship is constructed between the angle and the start and end points of the radius.

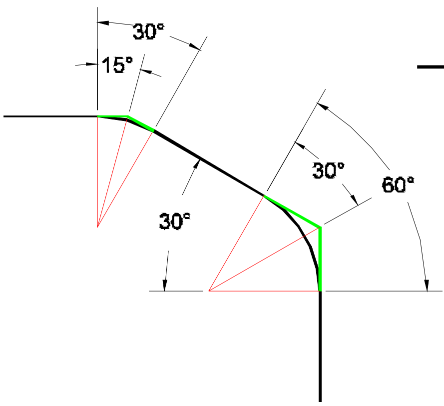


Fig 9.11

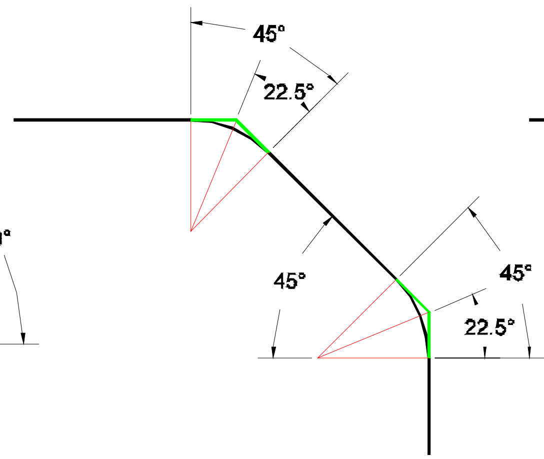


Fig 9.12

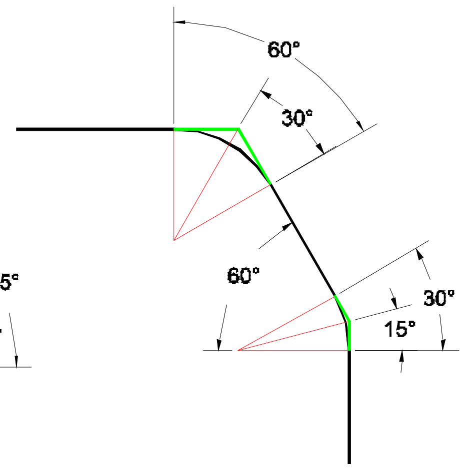
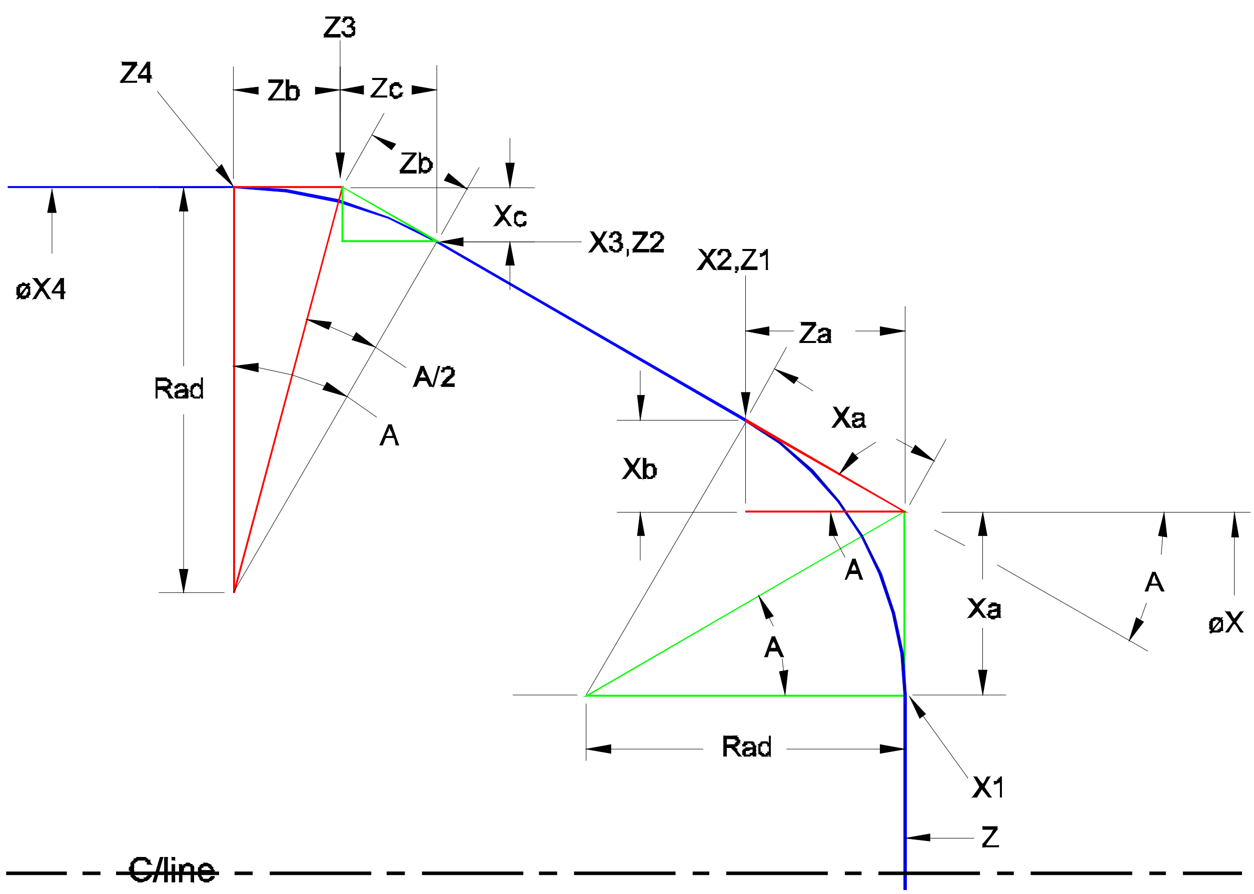


Fig 9.13

3 lines drawn from the center point of the arc, to the intersect points of the arc and corner, create a pair of triangles that are equal and bisect the angle equally. The 3 figures show this relationship.

Calculating Intersect points



Using the rules learned we can define these end points.

Fig 9.14

Defining the points

In Fig 9.14 all of the triangles are displayed that are needed to calculate the start and end co-ordinates for the CNC program.

The main features are Angle A, Diameter X , Diameter X4, and points Z and Z3.

From these points, or a combination of these points, the program can be calculated.

An example (fig 9.15) with dimensions will be used to explain these calculations.

Example 7

Dimensioned part

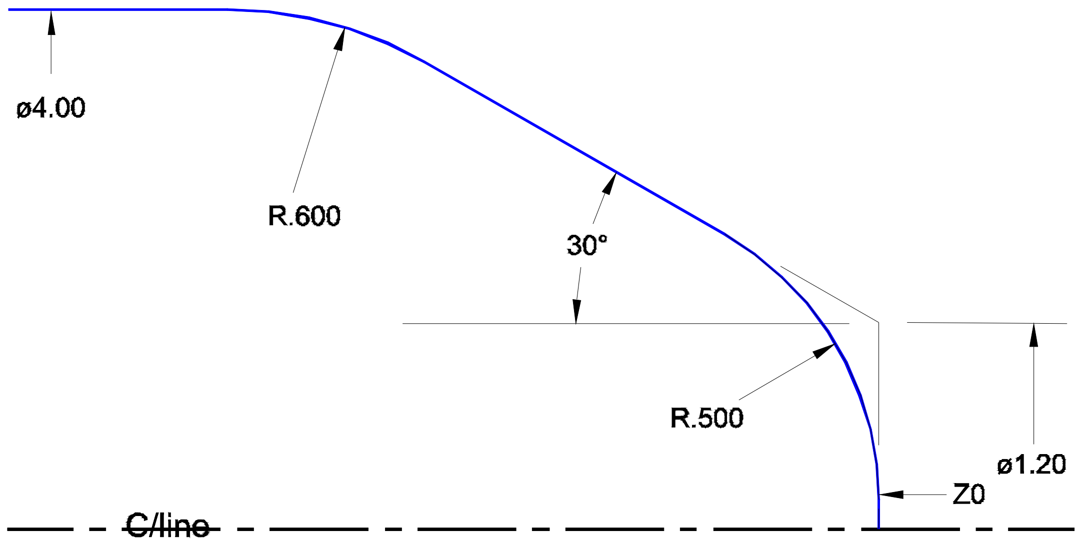


Fig 9.15

This shows a typical part and related dimensions.

The 1st calculation is to determine point Z3

Remember that an X axis triangle side is a radial value. Diameters have to be divided in half.

Opp = (X4-X)/2
= (4.00-1.200)/2
= 2.8/2
Opp = 1.4

Adj = Opp ÷ Tan A
= 1.40 ÷ Tan 30
= 1.40 ÷ 0.5773
Adj = 2.4250

Z3 = Z3 - Z
= 2.425 - 0

Z3 = 2.425

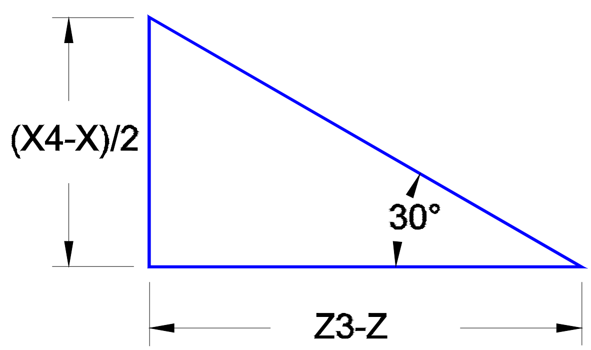


Fig 9.16

Defining the 1st corner points

The 2nd calculation is to find point X1

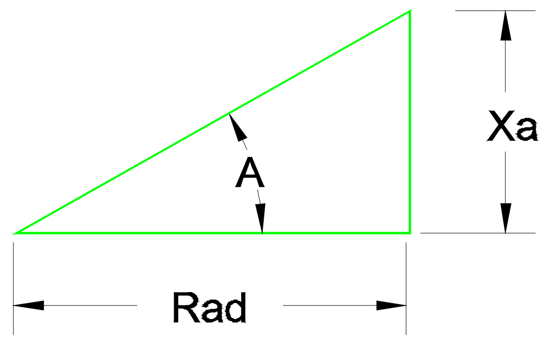


Fig 9.17

Xa = Opp

Use this solution

Opp = Tan A x Adj
Xa = Tan 30 x .5
= .5773 x .5

Xa = 0.2886751

Therefore

X1 = X - 2(Xa)
= 1.2 - 2(0.2886751)
= 1.2 - 0.5773
X1 = 0.6227

Because the 2 triangles are equal that bisect the radius. The Opp side Xa is the Hyp side in the 2nd triangle.

The 3rd calculation is to find point Za

Za = Adj

Use this solution

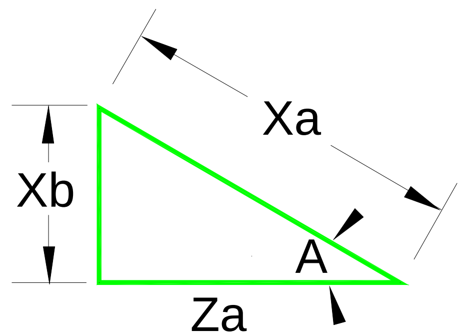


Fig 9.18

Adj = Cos A x Hyp

= Cos 30 x Xa

= 0.8660 x 0.2886751

Adj = .2500

Therefore

Z1 = Z + Za
= 0 + 0.25
Z1 = 0.25

The 4th calculation is to find point Xb

Xb = Opp

Use this solution

Opp = Sin A x Hyp
= Sin 30 x Xa
= 0.5000 x 0.2886751
Opp = .1443

Therefore

X2 = X + 2(Xb)
= 1.2 + 2(0.1443)
= 1.2 + 0.2886
X2 = 1.4886

Defining the 2nd corner points

The main difference in the next group of calculations is that the angle A is divided in half.

If you study the figures 9.11 through 9.13 you will see that as the angle increases to 45^O the relationship between the part angle and the bisecting triangles change. Before 45^O the 1st angle stays at the part angle, After 45^O the 1st angle is half the part angle.

In the Calculations that follow the angle will be 15^O

The 5th calculation is to find point Zb

Zb = Opp

Use this solution

Opp = Tan A x Hyp
= Tan 15 x Rad
= 0.26794 x 0.600
Opp = 0.1607

From the 1st calculation Z3 = 2.425

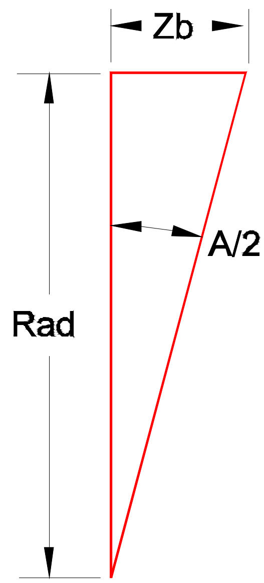


Fig 9.19

Therefore Z4 = Z3 + Zb
= 2.425 + 0.1607
Z4 = 2.5857

The 6th calculation is to find point Xc

Xc = Opp

Use this solution Opp = Tan A x Hyp
= Tan 30 x 0.1607
= 0.5773 x 0.1607
Opp = 0.09278

Therefore X3 = X4 - 2(Xc)
= 4.00 - 2(0.09278)
= 4.00 - 0.18556
X3 = 3.8144

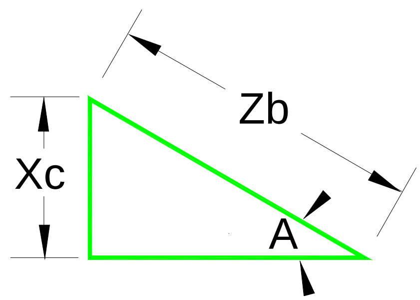


fig 9.20

Finally To find point Zc

Zc = Adj

Use this solution

Adj = Cos A x Hyp
= Cos 30 x 0.1607
= 0.8660 x 0.1607
Adj = 0.1391

Therefore

Z2 = Z3-Zc
= 2.425 - .1391
Z2 = 2.2858

Creating the Program

The co-ordinates that have been calculated can now be use to generate the program.

%00112	%00112
N00 G90 G95 S1200 T1.1 M3	N00 G90 G95 S1200 T1.1 M3
N10 G00 X0.2 Z1.0	N10 G00 X0.2 Z1.0
N20 G42 X0 Z0.5	N20 G42 X0 Z0.5
N30 G01 **Z** F.005	N30 G01 Z0 F.005
N40 G01 **X1**	N40 G01 X0.6227
N50 G03 **X2 Z1** R	N50 G03 X1.4886 Z-.25 R0.5
N60 G01 **X3 Z2**	N60 G01 X3.8144 Z-2.2858
N70 G03 **X4 Z4** R	N70 G03 X4.00 Z-2.5857 R0.6
N80 G00 G40 X?	N80 G00 G40 X?

Non Axis parallel angles

When programming non axis parallel moves, additional calculations have to be made this example shows the extra Zc amount that has to calculated. Other calculations also have to be made to allow for the angular position of the intersect points.

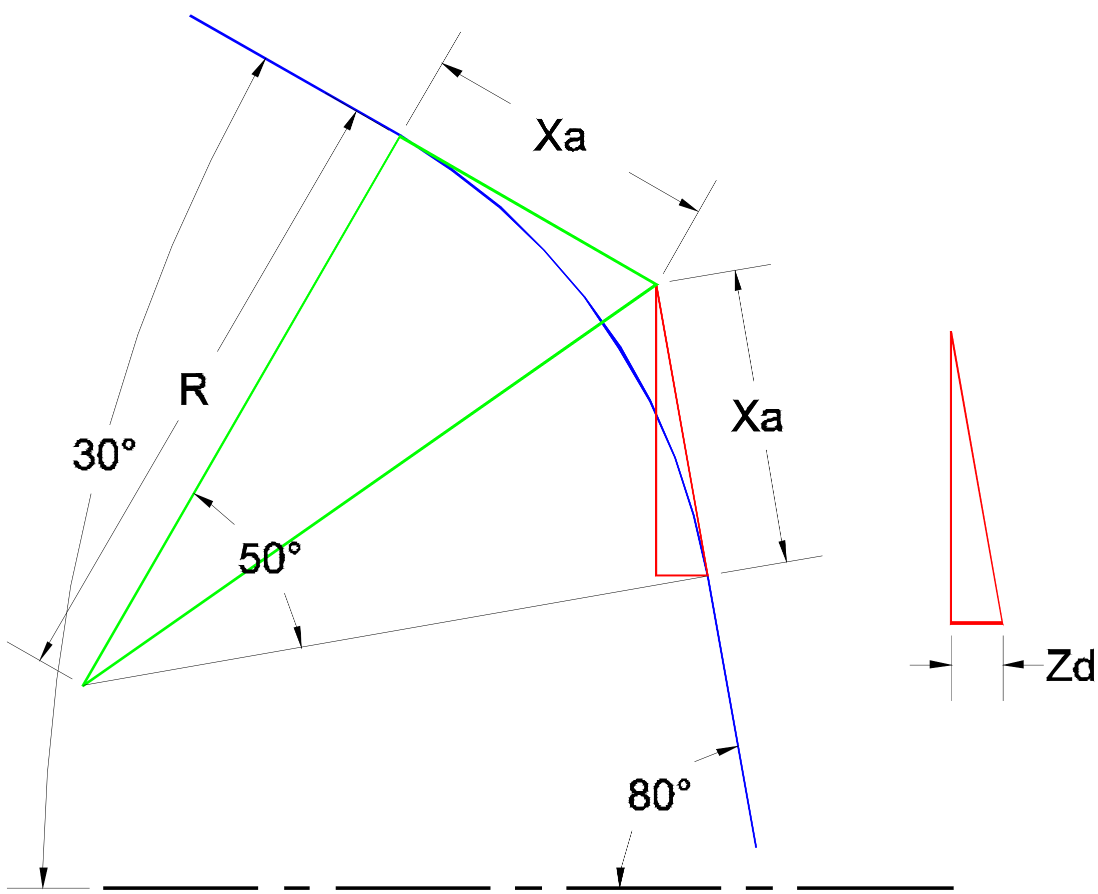


Fig 9.21

Note the relationship that the angled lines form with the interior angles of the bisecting triangle.
If this seems like a lot of programming for such a simple shape,

Circle problems

A common problem in turning is producing a partial radius on a shaft, This can be solved with the formula for circular segments.

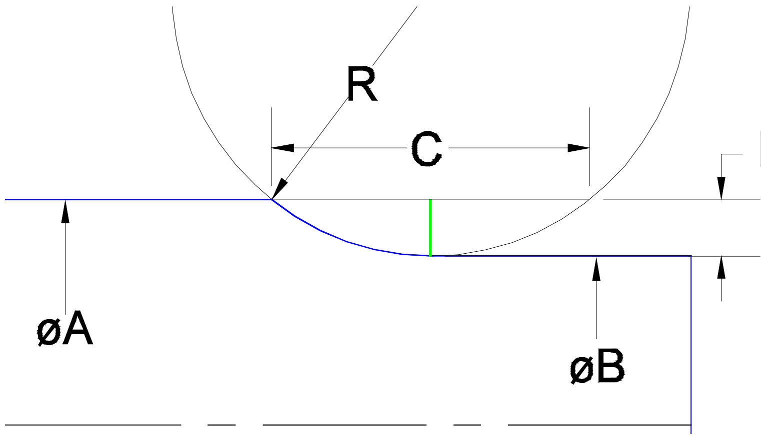


Fig 9.21

If we know the diameters A & B the height h can be simply calculated

h = (A-B)/2

Example 8

A shaft having a Diameter A=1.5 and a diameter B=.75 with a blending radius R=2.0 starting at Z-1.5

Calculate the blend position in Z for the A diameter and the arc R.

Use this solution

h = (A-B)/2
= (1.5-.75)/2
= .375

C = 2 √ h(2R - h)
= 2 √ .375(2x2. - .375)
= 2 √ .375(3.625)
= 2 √ 1.359375
= 2 x 1.165922
= 2.3318

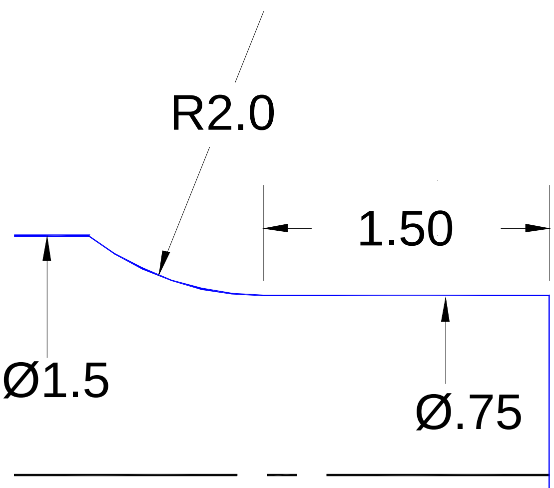


Fig 9.22

Divide C by 2 for half the chord length and add to Z-1.5

Z-1.5+ (-1.1659) = Z-2.6659

Section 11

Advanced Features for Programming

Advanced Features for Position Input

Cartesian Coordinates

In an earlier section we discuss Absolute and Incremental coordinates. This is referred to as a Cartesian coordinate system.

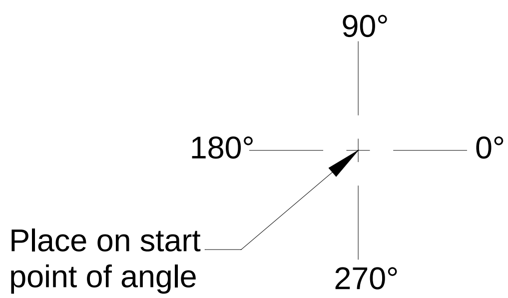
Using the extended features in Cartesian coordinates allow us to provide a method of programming that greatly simplifies the process of cutter path generation.

This section explains this feature and its limitations.

Angles Coordinates

The control allows the input of angles with Cartesian coordinates.

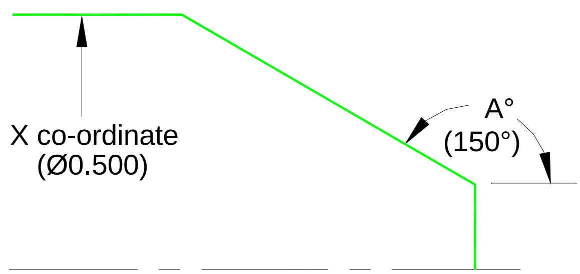
The example below shows the orientation of these angle coordinates:



Angles and X Coordinates

Using this feature in your programs allows the ability to use 1 coordinate position and an angle the example below demonstrates this:

G00 X0 Z0.1
G01 Z0
G01 X0.100
G01 X0.500 A150.
G01 Z-?.

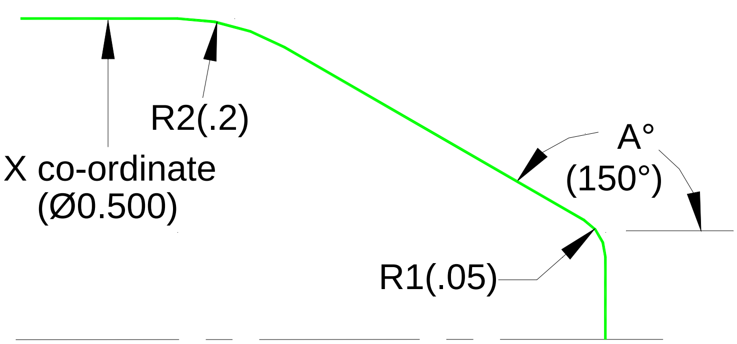


This profile would require a triangle to be calculated for the Z end point of the angle, if using X & Z positions. This may not appear to be of great advantage, but it can be combined with radius corners and chamfering. This is shown in the next example:

Angles, Radii and X Coordinates

G00 X0 Z.1
G01 Z0
G01 G36 R.05 Q90. X0.100
G01 G36 R.2 Q150. X0.500
G01 Z-?.

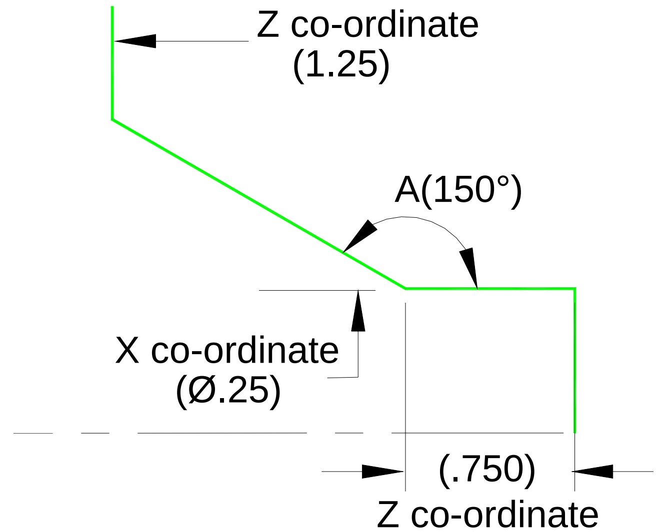
As you can see this reduces the amount of calculation for programming.



Angles and Z Coordinates

Using this feature in your programs allows the ability to use 1 coordinate position and an angle the example below demonstrates this:

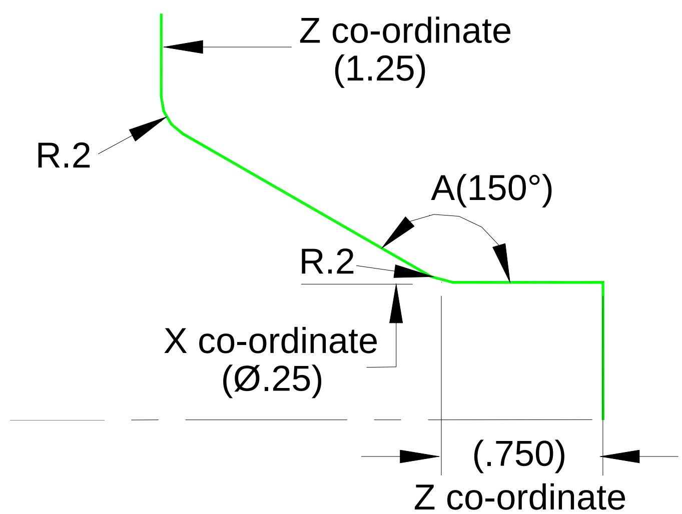
G00 X0 Z.1
G01 Z0
G01 X0.250
G01 Z-.75
G01 Q150. Z-1.250
G01 X?



This profile would require a triangle to be calculated for the X end point of the angle, if using X & Z positions.

Angles, Radii and Z Coordinates

G00 X0 Z.1
G01 Z0
G01 X0.250
G01 G36 R.2 Z-.750
G01 G36 R.2 Q150. Z-1.250
G01 X?



As you can see this reduces the amount of calculation for programming.

Notes:

This feature can produce unexpected results due to incorrect angles, and radii always test the program prior to machining.

If chamfers are required use G39

Operations with parameters

The CNC has 199 parameters (P100-P299) with which the programming of parametric blocks can be performed as well as different types of operations and jumps within a program.

Parameters P290-P299 are used for machine sub-programs and are not to be used as variables.

Parametric blocks can be written in any part of the program.

All math functions should be grouped at the head of the program to make it easier to troubleshoot.

The control itself should be thought of as a parametric control, that is almost every area of the control that stores data, can be accessed through a parametric variable.

This portion of the manual is designed to show some simple examples that you may find useful.

If more information is required, please read the section on High level programming in the Fagor programming manual.

Simple statements

All High level statements are enclosed in () parentheses.

To load a value in a variable use this statement :

(P200 = 2.0) this will load 2 into variable 200

To do math in a statement use this format :

(P200 = 1+1) this will load a value of 2 into P200

You can use variables in math statements :

(P200 = P200/2) this will divide P200 by 2 and load into P200

Math statements can be quite complex :

(P200 = (5 x P201) / (2 x P202)) The rule is every open (must have a close)

Math & Trigonemetry statements

You can use these common math statements :

+	: addition	(P200 = 1**+**1)	P200 = 2
-	: subtraction or a negative number	(P200 = **-**(2 x 3)	P200 = -6
*****	: multiplication	(P200 = 2 **x** 3)	P200 = 6
/	: division	(P200 = 9 **/** 2	P200 = 4.5
MOD	: module (remainder of a division)	(P200 = 7 **MOD** 4)	P200 = 3
EXP	: exponential	(P200 = 2 **EXP** 3)	P200 = 8

You can use these common Trigonometry statements :

SIN	: sine	(P1 = **SIN** 30)	P1 = 0.5
COS	: cosine	(P2 = **COS** 30)	P2 = 0.8660
TAN	: tangent	(P3 = **TAN** 30)	P3 = 0.5773
ASIN	: arc sine	(P4 = **ASIN** 1)	P4 = 90
ACOS	: arc cosine	(P5 = **ACOS** 1)	P5 = 0
ATAN	: arc tangent	(P6 = **ATAN** 1)	P6 = 45
ARG	: ARG (x,y) arc tangent y/x	(P7 = **ARG**(-1,-2))	P7 = 243.4349

Other Math Features

ABS	: absolute value	(P1 = **ABS** –8)	P1 = 8
LOG	: decimal logarithm	(P2 = **LOG** 100)	P2 = 2
SQRT	: square root	(P3 = **SQRT** 16)	P3 = 4
ROUND	: rounding up a number	(P4 = **ROUND** 5.83)	P4 = 6
FIX	: integer	(P5 = **FIX** 5.423)	P5 = 5
FUP	: if integer takes integer	(P6 = **FUP** 7)	P6 = 7
	: if not, takes entire part + 1	(P6 = **FUP** 5.423)	P6 = 6

Logic Features

EQ : equal
NE : different
GT : greater than
GE : greater than or equal to
LT : less than
LE : less than or equal to

Logic or binary operators

NOT, OR, AND, XOR : act as logic operators between conditions and as binary operators between variables and constants.

These features can be used to check counting or make decisions on jumps and cycle calls if a set of circumstances are determined.

Tool Data variables

As stated earlier the control has the ability to access most areas using system variables, The most commonly used are tool data variables.

In all 8055 controls is a user program to store tool offset and geometry data either in a new program on their own, or at the bottom of an existing program. This program is stored as P010000,H-X, This program is normally hidden, but uses these variables along with high level programming to execute.

The values for tools are as follows:

TOXn : This variable is the X value assigned to the selected tool (n) in the offset table.
TOZn **:** This variable is the Z value assigned to the selected tool (n) in the offset table.
TOFn **:** This variable is the F value assigned to the selected tool (n) in the offset table.
TORn **:** This variable is the R value assigned to the selected tool (n) in the offset table.
TOIn **:** This variable is the I value assigned to the indicated tool (n) in the offset table.
TOKn **:** This variable is the K value assigned to the indicated tool (n) in the offset table.

NOSEAn **:** This variable is the tool tip angle value assigned to the indicated tool (n) in the geometry table.
NOSEWn **:** This variable is the cutter width value assigned to the indicated tool (n) in the geometry table.
CUTAn **:** This variable is the cutting angle value assigned to the indicated tool (n) in the geometry table.

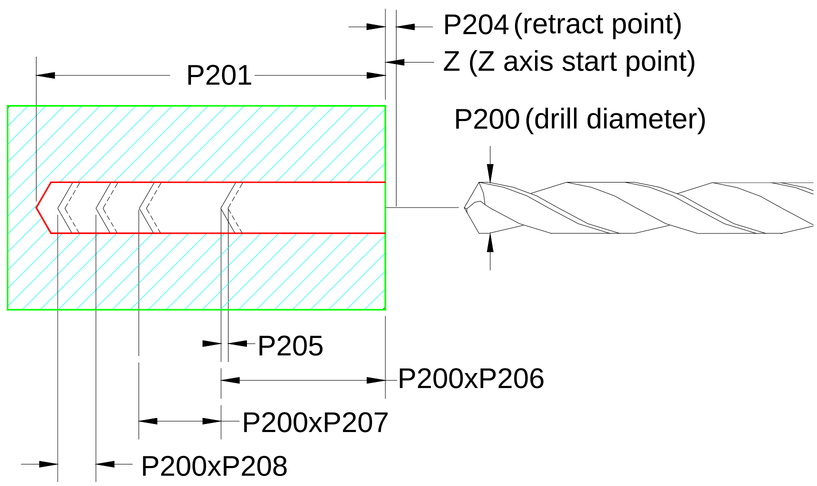
All of these values are READ – WRITE variables, meaning that they can be written to in a program or data can be pulled from these values in the offset & geometry tables tables.

Sample deep hole drilling macro

This program uses a sub program called SUB 101 to control the variables and execute the program. The program is based on the logic of reducing the amount of material cut as the drill goes deeper into the hole, There are three stages, the first depth, second depth, and subsequent pecks to final depth. Each peck is based on a multiplication factor of the drill diameter.
It is assumed that the programmer places the drill in the correct X axis position and that the drilling cycle is into or towards the main spindle. As the programmer is placing the tool in the correct X position this macro will work on live drilled holes off center.

%Deep hole drill,MX,

```
(P200 = 0.500) ;Drill diameter
(P201 = 2.500) ; Drill depth
(P202 = 0.004) ; Feed per rev
(P203 = 0.100) ; Start point
(P204 = 0.100) ; Retract Point
(P205 = 0.010) ; Point clearance
(P206 = 5.0)     ; 1st pass multiplier
(P207 = 3.0)     ; 2nd pass multiplier
(P208 = 1.0)     ; Final peck multiplier
G90 G95 S1200 T3 M3 M8 ; Start block
G54
G0 X0
 (CALL 101) ; Peck drill sub
G0 Z1.0
M1
```



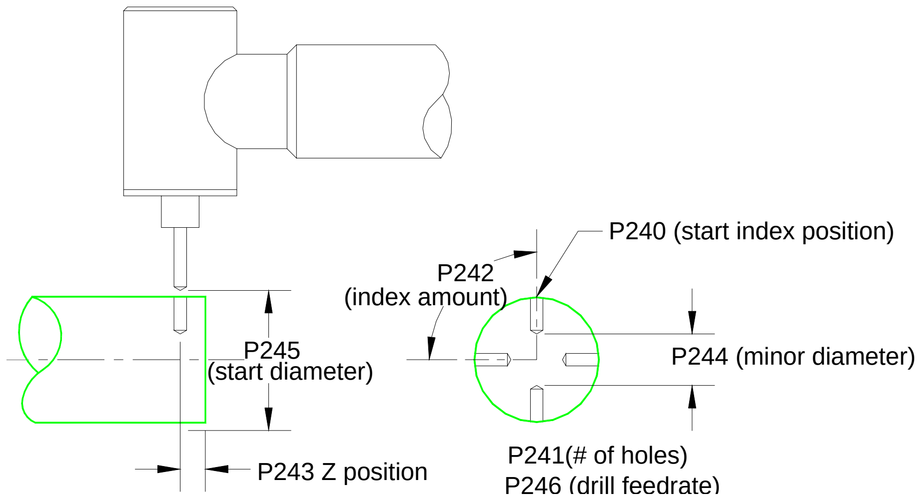
%Peck Sub,MX,

```
(SUB 101)                          ; This is the sub program definition block
(P210=P200*P206)                   ; 1st pass depth
(P211=P210-P205)                   ; Clear pos
(P212=(P210 +(P200*P207)))         ; 2nd Depth
(P214=P212-P205)                   ; 2nd pass clear
(P215=P200*P208)                   ; Remainder peck amount
(P220=P212+P215)                   ; 1st peck depth for remainder
(P216=P220-P205)                   ; Clearance point
G5 ZP203                           ; Rapid Z start point
G1 Z-P210 FP202                    ; 1st pass
G0 ZP204                           ; Retract point
(IF P212 LE P200 GOTO N1 ELSE N20); Depth of hole check logic
N1 Z-P211                          ; Rapid into hole point
G1 Z-P212 FP202                    ; 2nd pass
G0 ZP204                           ; Retract point
(IF P220 LE P200 GOTO N2 ELSE N20); Depth of hole check logic
N2 G0 Z-P214                       ; Next start point
G1 Z-P220 FP202                    ; Next peck point
G0 ZP204                           ; Clearance point
(P214=(P220-P205))                 ; Calculate return point
(P220=(P220+P215))                 ; Calc new depth
(IF P220 LT P201 GOTO N2 ELSE N19); Check depth logic
N19 G0 Z-P214                      ; last peck return
G1 Z-P201 FP202                    ; final depth
G0 ZP204                           ; Clearance point
N20 (RET)                          ; End of routine
```

Sample indexing with live drilling macro

This program uses a sub program called SUB 1001 to control the variables and execute the sub program. The program uses a start drill diameter & end drill diameter, for the drill operation you can specify the angular start position and the number of holes.

Indexing in this sample is for C axis, if done on an indexing machine the G0 C command would be replaced with M19 S



%side drill macro,MX,

```
N1 G90 G94 T9 M5    ;
live side drill
G54
(P240 = 0)          ; Start index position
(P241 = 6)          ; # of holes
(P242 = 360)        ; Angular index amount
(P243 = 0.900)      ; Z hole position
(P244 = 0.200)      ; Minor diameter of hole
(P245 = 0.800)      ; Start Drill Diameter
(P246 = 24.00)      ; Feedrate for drill
G0 G7 X1.2          ; Safe X position
G5 Z0.1             ; safe Z position
Z-P243 M50          ; Live tool on & move to start drill position
(CALL 1001)         ; Call sub program
G0 G7 X1.2          ; Safe X position
Z1.5 M61            ; Live tool off
M1
```

%C Side Drill Sub,MX,

```
(SUB 1001)
(P230=P242/P241)              ; Index amount
(P232=P241)                   ; Hole counter reset
(P239=P240)                   ; Set 1st hole index position
N1000 G15                     ; Activate C axis (only on C machines)
G0 CP239                      ; Index to start position or next index position
G0 XP245                      ; Rapid to start drill position
G1 XP244 FP246                ; Feed to drill end point
G0 XP245                      ; Rapid to start drill position
(P239=P239+P230)              ; Next index position
(P232=P232-1)                 ; Count down holes
(IF P232 GT 0 GOTO N1000)     ; Check for last hole
N1001 (RET)                   ; End of sub return to main program
```

Section 12

Manual tool nose radius Compensation

Manual Calculation

When you select an indexable insert to machine a part, it will normally have a radius on the tip, This tip is called the Tool nose Radius.

As you create a program for a CNC lathe, some moves need tool nose radius compensation.(TNRC) and some do not

In this section we will explain how and when to apply these calculations.

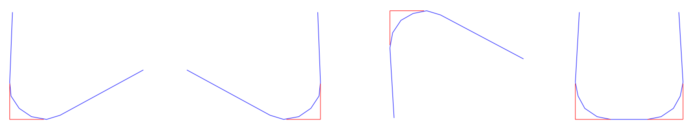
Theoretical Sharp Point

The basic concept of programming is to calculate a tool path based on a known shape.

This shape is the part profile. Without using tool nose radius compensation(G41-G42)you have to alter the tool path to allow for the tool nose radius.

Most lathe programming is done using the concept of theoretical sharp point.

With this method the known point is called the Theoretical sharp point. This is a position axis tangent to the tool radius in X & Z .



Right hand, left hand, boring and grooving tips(fig 10.1)

If you are using a reverse tool or a boring bar, the Theoretical sharp point will be in a relative position to the orientation of the tool. A grooving tool can have 2 points to program.

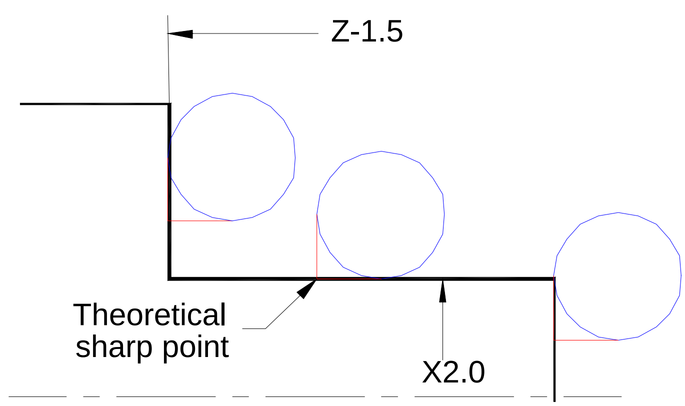
Calculation and application

Any axis parallel moves do not have to have tool nose radius compensation (TNRC) applied to them.

The bottom and side of the insert moves along the desired path and create a part with no angles or radii.

The positions in the program will be the same positions that are on the part drawing, A diameter of 2.0” will be X2.0. A length on Z of 1.5” will be Z-1.5(assuming Z0 is the face of the part)

The example below shows this :



Axis parallel moves (fig 10.2)

Types of Calculation

There are calculations for Radii, Tapers, reverse tapers(rear of the tool tip) and combinations of both these will be explained in the following section.

Radius Calculation

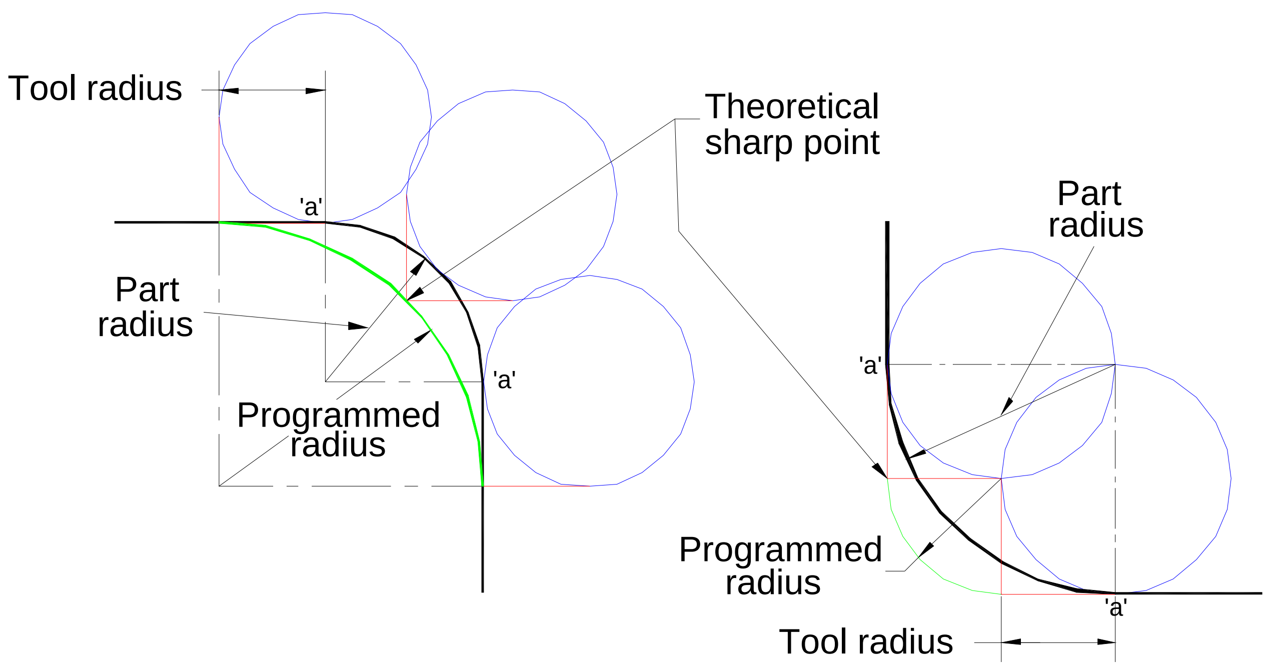
When a radius is required TNRC has to be applied.

The basic rule is as follows :

An external radius has the Tool Nose Radius added to it.

An internal radius has the Tool Nose Radius subtracted from it.

The example below shows this :



External and Internal radius compensation(fig 10.3)

In the example above the Theoretical start and end points are calculated based on the fact that the tangent point for 2 radii is always through their center.(points ‘a’)

In the external radius we add the tool nose radius to the part radius and then calculate the actual start and end points.

External Radius Calculation

If a radius of .25 is required on the external corner of a part and the tool nose radius is .031 the programmed radius will be:

.25+.031=.281

If the radius is on a 1.0 diameter and at the Z0 face of the part the program will follow the example below.

G1 X0 Z0	Start of part profile
G1 X.438	1.0-.(281 x 2)=.438 (see note)
G3 X1.0 Z-.281 R.281	Move to the X and Z axis end point of the arc with the R value.
G1 Z-?	Next axis parallel move

Internal Radius Calculation

If a radius of .25 is required on the internal corner of a part and the tool nose radius is .031 the programmed radius will be :

.25-.031=.219

If the radius is on a 1.0 diameter and at the Z0 face of the part the program will follow the example below.

G1 X1. Z0	Start of part profile
G1 Z-.781	Start of radius
G2 X1.438 Z-1. R.219	1.0+(.219 x 2)=1.438(see note)
G1 X-?	Next axis parallel move

NOTE:

When calculating the end points in a part program you must double the X axis calculation to allow for both sides of the part. Any trigonometry calculation is only generating a radial value for X.

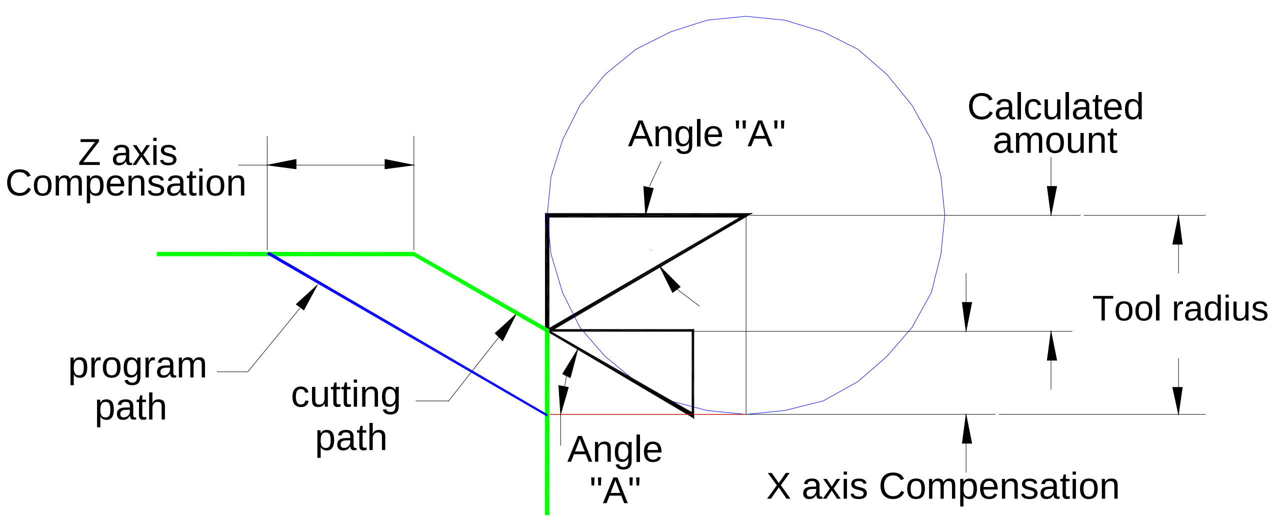
Taper Calculation

To simplify tool radius compensation on tapers a chart is provided that gives a range of radii and angles between 0^{o}-45^{o} If a radius greater than 45^{o} is encountered simply use the opposite angle in the triangle and reverse the Xc and Zc values.

If an angle has to be calculated the formula is described below :

$$\textbf{TNRC for X= R-(Tan } \theta \textbf{ x R)}$$

Triangle development :



Tool radius comp for angles(Fig10.4)

Taper Calculation

If the angle is 30° and the radius is .03125"

The X axis calculation is : **X= R-(Tan θ x R)**

X=.03125-(Tan30 x .03125)

X=.03125-(.57735 x .03125)

X=.03125-(.01804)

X=.013207

TNRC for Z= $\frac{\text{Compensation in X}}{\text{Tan } \theta}$

The Z axis calculation is : $\mathbf{Z=\frac{.013207}{.57735}}$

Z=.02287

If these calculations are compared to your chart the numbers match.

In practice it is far easier to use a chart and apply one axis of compensation, and then calculate the end point of the other axis. This method ensures that the correct angle is machined, and any rounding of numbers is taken into consideration.

Tool nose compensation chart

	0.005"Radius		0.007"Radius		0.010"Radius		1/64"Radius		1/32"Radius	
ANGLE	Xc	Zc	Xc	Zc	Xc	Zc	Xc	Zc	Xc	Zc
1	.000087	.004956	.000121	.006939	.000173	.009913	.000270	.015489	.000541	.030868
2	.000172	.004913	.000240	.006878	.000343	.009825	.000536	.015352	.001072	.030704
3	.000258	.004869	.000357	.006817	.000510	.009738	.000797	.015216	.001595	.030432
4	.000337	.004825	.000472	.006758	.000675	.009651	.001054	.015079	.002109	.030159
5	.000418	.004782	.000586	.006694	.000837	.009563	.001307	.014943	.002615	.029886
6	.000498	.004738	.000697	.006833	.000996	.009476	.001556	.014906	.003112	.029612
7	.000576	.004694	.000807	.006572	.001153	.009388	.001801	.014894	.003602	.029339
8	.000654	.004650	.000915	.006511	.001307	.009301	.002042	.014532	.004065	.029065
9	.000730	.004606	.001021	.006449	.001459	.009213	.002260	.014395	.004580	.028791
10	.000804	.004563	.001126	.006388	.001609	.009125	.002514	.014258	.005028	.028516
11	.000878	.004519	.001230	.006326	.001757	.009037	.002745	.014121	.005489	.028241
12	.000951	.004474	.001331	.006264	.001902	.008949	.002972	.013983	.005944	.027966
13	.001023	.004430	.001432	.006202	.002046	.008861	.003196	.013845	.006393	.027690
14	.001094	.004386	.001531	.006141	.002187	.008772	.003417	.013707	.006835	.027413
15	.001163	.004342	.001629	.006078	.002327	.008683	.003835	.013568	.007271	.027136
16	.001232	.004297	.001725	.006016	.002464	.008595	.003851	.013429	.007701	.026858
17	.001300	.004253	.001820	.005954	.002600	.008506	.004063	.013290	.008126	.026580
18	.001367	.004208	.001914	.005891	.002735	.008416	.004273	.013150	.008545	.026301
19	.001434	.004183	.002007	.005829	.002867	.008327	.004480	.013010	.008959	.026021
20	.001499	.004118	.002099	.005766	.002998	.008237	.004684	.012870	.009388	.025740
21	.001564	.004073	.002189	.005703	.003127	.008147	.004886	.012729	.009772	.025458
22	.001627	.004028	.002278	.005639	.003255	.008056	.005086	.012588	.010171	.025178
23	.001691	.003983	.002367	.005576	.003381	.007966	.005283	.012446	.010566	.024892
24	.001753	.003937	.002454	.005512	.003506	.007874	.005478	.012304	.010958	.024608
25	.001845	.003892	.002540	.005448	.003629	.007763	.005671	.012161	.011341	.024322
26	.001876	.003846	.002626	.005384	.003751	.007691	.005861	.012018	.011723	.024036
27	.001936	.003800	.002710	.005319	.003872	.007599	.006050	.011874	.012190	.023746
28	.001996	.003753	.002794	.005255	.003991	.007507	.006236	.011729	.012473	.023459
29	.002065	.003707	.002877	.005190	.004109	.007414	.006421	.011584	.012542	.023188
30	.002113	.003660	.002959	.005124	.004226	.007321	.006604	.011438	.013208	.022877
31	.002171	.003613	.003040	.005089	.004342	.007227	.006785	.011292	.013568	.022584
32	.002228	.003586	.003120	.004993	.004457	.007133	.006984	.011145	.013928	.022269
33	.002285	.003519	.003199	.004927	.004570	.007038	.007141	.010997	.014282	.021994
34	.002341	.003471	.003278	.004880	.004683	.006943	.007317	.010848	.014634	.021696
35	.002397	.003424	.003356	.004793	.004794	.006847	.007491	.010699	.014982	.021397
36	.002452	.003375	.003433	.004726	.004905	.006751	.007664	.010548	.015327	.021096
37	.002507	.003327	.003510	.004658	.005014	.006654	.007835	.010397	.015689	.020794
38	.002561	.003278	.003586	.004590	.005123	.006557	.008004	.010245	.016008	.020490
39	.002615	.003229	.003661	.004521	.005230	.006459	.008172	.010092	.016344	.020184
40	.002668	.003180	.003736	.004452	.005337	.006360	.008339	.009938	.016678	.019876
41	.002721	.003131	.003810	.004383	.005443	.006261	.008504	.009783	.017008	.019586
42	.002774	.003081	.003883	.004313	.005548	.006181	.008668	.009627	.017336	.019254
43	.002826	.003030	.003956	.004243	.005652	.006061	.008831	.009312	.017985	.018941
44	.002878	.002980	.004029	.004172	.005755	.005960	.008992	.009312	.017985	.018624
45	.002929	.002929	.004100	.004101	.005858	.005858	.009153	.009153	.018306	.018306

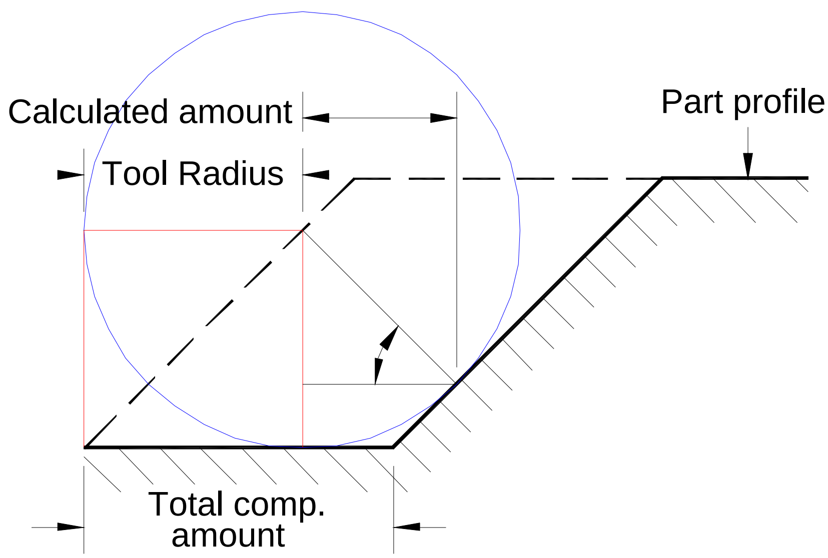
For Lathe compensation MULTIPLY Xc*2 (Diameter programming)For angles over 45^{o} subtract from 0^{o}(Angle = 53^{o} use 37^{o}) (53+37=90) Xc=Zc Zc=Xc

Toolnose compensation when profiling with the rear of the tool

For most rear profiling applications a groove or recess is produced and the compensation takes place in Z Only the programmed end points are modified from the part size. X axis points stay the same.

A simple calculation is all that is needed for rear profiling in Z axis.

Compensation in Z = R+(Sine θ x R)



Rear profile compensation(Fig 10.5)

Rear Taper Calculation

Examples :

Using a **0.03125** Radius tool and a **45°** angle

Compensation in Z = **R+(Sine θ x R)**

0.03125+(Sin45 x .03125)

0.03125+(.7071 x .03125)

Z = 0.0533

Using a **0.015** Radius tool and a **45°** angle

Compensation in Z = **R+(Sine θ x R)**

0.015+(Sin45 x .015)

0.015+(.7071 x .015)

Z = 0.0256

Using a **0.015** Radius tool and a **30°** angle

Compensation in Z = **R+(Sine θ x R)**

0.015+(Sin30 x .015)

0.015+(.5 x .015)

Z = 0.0225

Section 13

Drawbar actuator
&
Collet noses

Description

The air powered rotary actuator operates at speeds up to 6000 RPM, even when powered with maximum air pressure. Special consideration has been given to enhance the durability of your collet closer; this includes hardened wear surfaces, complete sealing of rotating components, and urethane wipers. The bearings are shielded, and the draw tube mount acts as a seal.

The air actuator is designed to mount to rotating devices, most commonly spindles. The unit will provide a push or pull force capable of opening or closing collets, fixtures, and short stroke power chucks.

- Clamping pressure can be adjusted using the air gage on the left side panel and its adjuster.
- Unclamp pressure is the supplied line pressure and cannot be adjusted.
- A safety check is performed on spindle start up, and if the main collet is open the machine will close the collet regardless of the switch setting.
- A PLC switch is provided for reverse clamping operations. (Internal chucking etc.)

Changing collets

The drawbar has a threaded collar that adapts the drawbar thread to the correct thread for a 5C collet.

To change a collet, follow these steps:

1. Remove the set screw in the collet nose.
2. using a collet wrench, unscrew the collet from the adaptor.
3. Change the collet size and thread the collet onto the drawbar. (Turn clockwise)
4. Replace the set screw.

Note : Always clean the threaded adaptor and the collet prior to installation.

How to control the holding force

By using the regulator and gauge, start with air pressure of 60 lbs.; increase pressure gradually until part is held sufficiently for machining operation.

The air actuator permits even gripping pressure although the work piece may have a diameter variation.

The collet closer requires approx. 20 psi of pressure to activate, adjustment of pressure below this limit is not recommended.

Dead Length Collets

The 5C configuration allows the use of 5C dead length collets.

A microcentric dead length collet nose is available for both main & sub spindles.

For the 3C sub spindle, a Schaublin dead length nose is available (F type)

Step collets

Oversize head step collets are available for the 5C system.

Expanding Arbors

Expanding arbors are available for the 5C system.

Section 14

Using Gang tool holders

Using gang tool holders

Gang tooled lathes provide a reliable, fast method of tooling setup, yet very few operators extract the best from this style of machine and tooling.

We hope these pages assist you in optimizing your next set-ups.

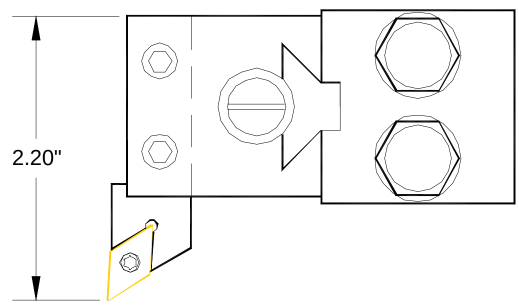
More Tools on the table

When setting up tools for work under one inch in diameter, we can keep tools very close together, this allows you to optimize your table space, and get many tools on the table.

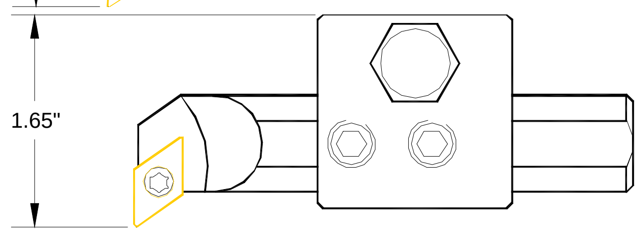
If you can stack the tools closer together for parts under 1" you can :

- Get more tools on the plate
- Change over setups quickly
- Be more flexible on the small jobs

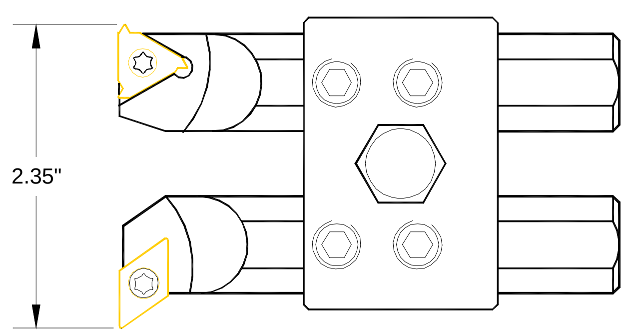
Most turning tools are square shank, but you can get the same results by using boring bars for turning,



There are other benefits also, you can cut inside the part and outside with 1 tool, plus a boring bar takes up a lot less room on the tool plate.



You can stack 2 bars back to back for outside turning and threading.



Consider that most jobs 1" & under, on a gang tool lathe run up to 3" long, and you can overhang a boring bar 4 times its diameter.

A 3/4" boring bar is not over extended for that application. (3/4" x 4 = 3")

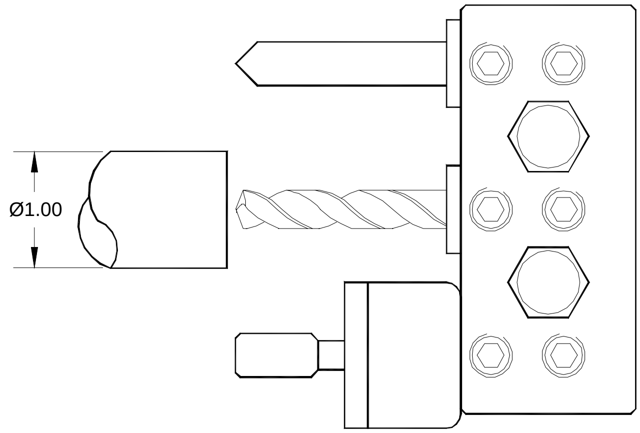
Multiple hole holders

When drilling you can use a multi tool holder and stack your drills close together.

Combining groups of tools simplifies setup greatly.

An NC spot drill (90° point), a drill, and a tap holder provide an ideal 3 position group for many jobs.

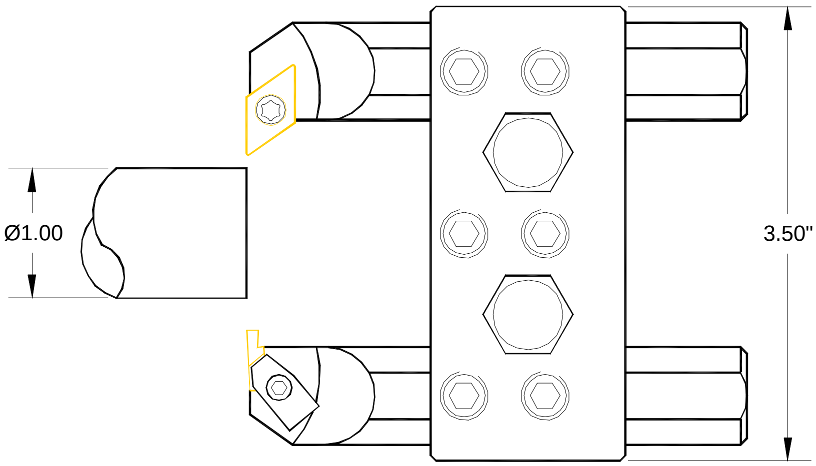
Simply swapping a drill and tap and a quick change over is achieved.



When choosing a threading tool, a grooving style holder (Top notch style) can provide a quick change from grooving to threading inserts.

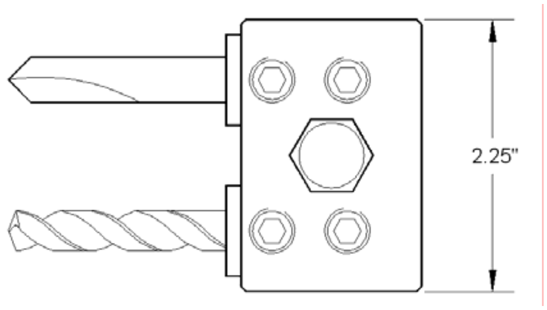
Combining turning & grooving bars in a 3 position holder can allow fast rough and finish of the front & back of a part in a small table space.

The setup shown requires a spindle direction change, this can be solved by using a left hand boring bar running upside down.



A two position holder is ideal for a simple center drill & drill.

This picture shown uses 2 reducer bushings.

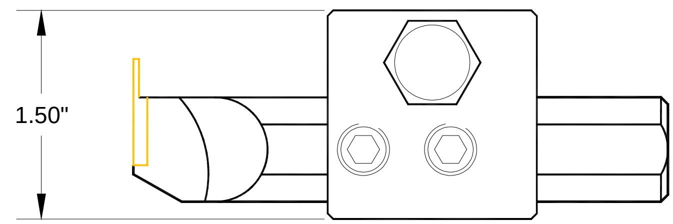


ER collet holders

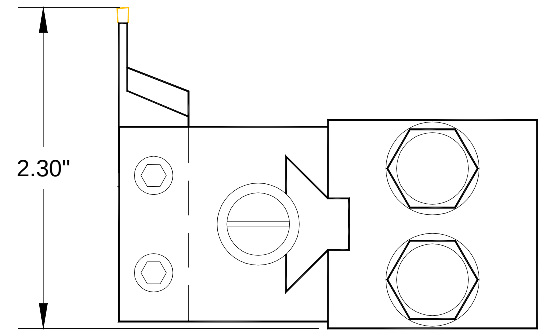
ER11 or ER16 collet holders offer great flexibility for drill holding. These holders are available in 1/2", 5/8" and 3/4" shanks.

Cutoff Tools

Cutoff tools for large diameter holders require a deep throat to accommodate 1” diameters, if your part is less than 0.5” diameter a thin grooving insert works well, (Nikcole manufacture a great range of inserts for grooving and turning) Thin Bit also has a wide range of inserts.



This allows a 5/8” or 3/4" bar for cutoff, saving almost an inch of table space. The boring bar cutoff can also be used in a multi-position holder,



Longer parts

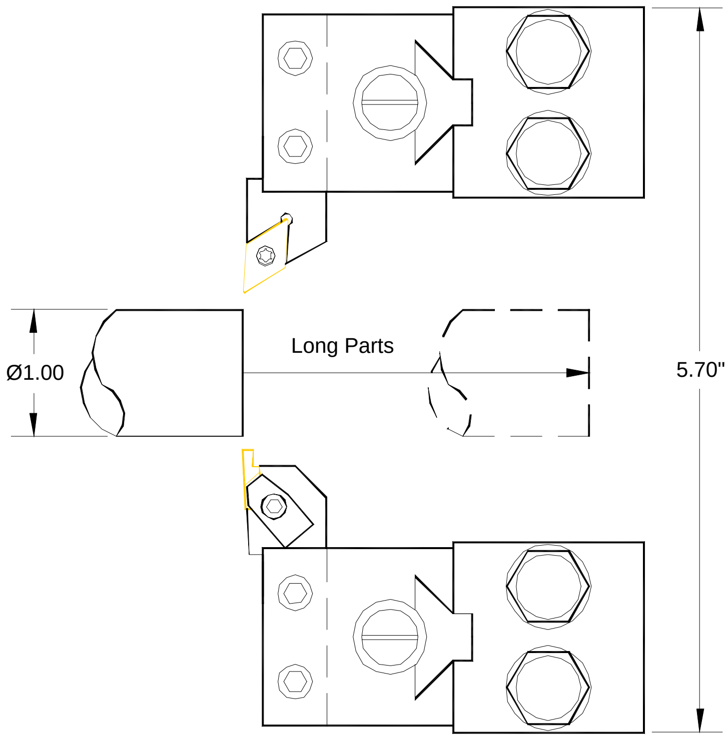
Sometimes a longer part requires machining using 2 holders,

this can be achieved easily.

This example shows the ability to turn a longer part, or perhaps a tough material that required more rigid tooling than a boring bar.

This setup lends itself to form tooling to decrease cycle times even further.

Either tool can be upside down so that a change of spindle direction is not required.



4, 6 & 8 Position holders

If your parts require frequent repeat setups a large capacity holder can save time and money.

Load your tools once in the holder, setup your offsets and at the end remove the tool holder with the tools still mounted. The next time this part is required, simply load the holder onto the plate and locate it against a known stop,

Load the offsets from the last time it was setup. And you are ready to run.

On all 8055 equipped machines a special tool offset saving macro is provided.

This method of setup may seem expensive (saving tools on the shelf) but let us evaluate these numbers:

1 eight position holder	= $420.00
6 tools at $150.00 each	= $900.00
Total tool cost	= $1320.00

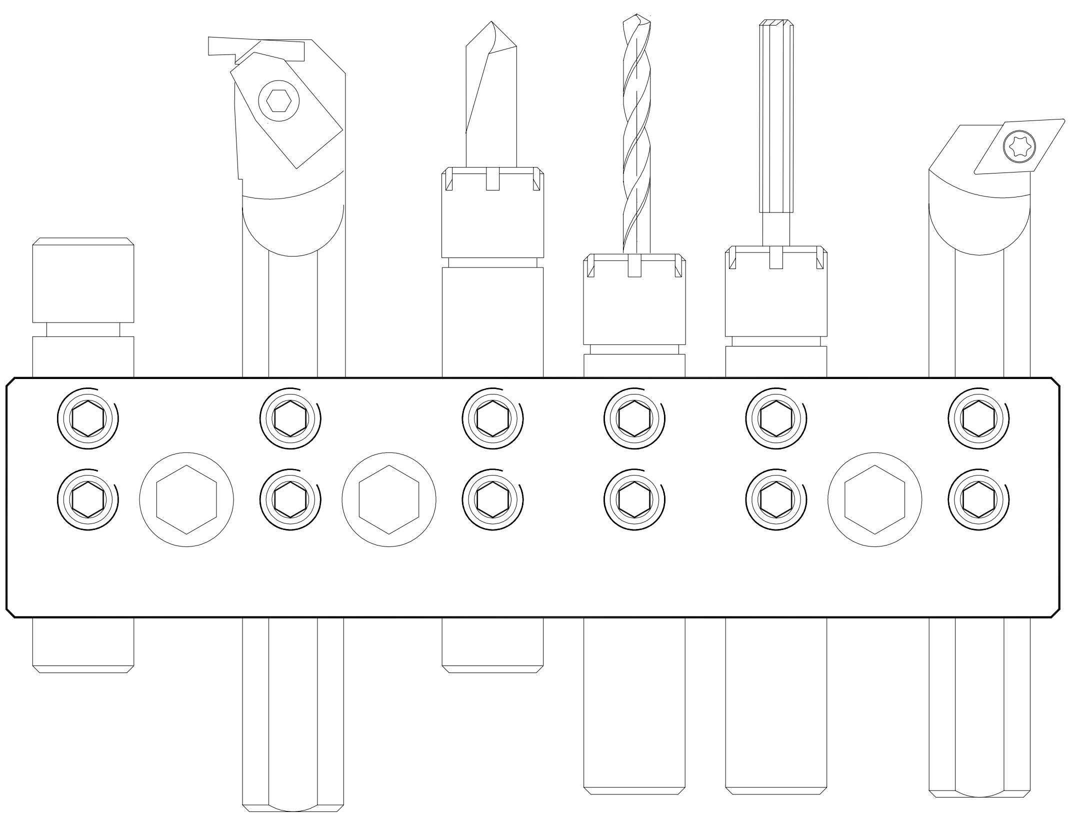
If your setup labor rate is $80.00 hour and a new setup takes 2.5 hours typical setup cost is $200.00

A new setup with would cost $200.00

The next setup should take 10 minutes to load the new tool bar and run offset section of program.

Saving you $190.00 per setup. Pay back is achieved at 6.4 setups. The more complex the setup the faster the payback time. If a 6 position holder is used payback is even faster (lower holder cost).

This example can be used as a basis for your payback time.



Posi-Lock tool holders

The design of this tool holder allows a greater flexibility in tool choices, The adjustable height feature with micro adjust allows the use of both 3/8” or 1/2" center square section holders, also brazed carbide tools can be adjusted for exact center height.

The tool holders are interchangeable with the standard posts, this feature allows quick changeover of tool types, and greater flexibility when changing tool order, to obtain the fastest cycletime on your gang tool lathe. (This feature is a major benefit of gang tooling).

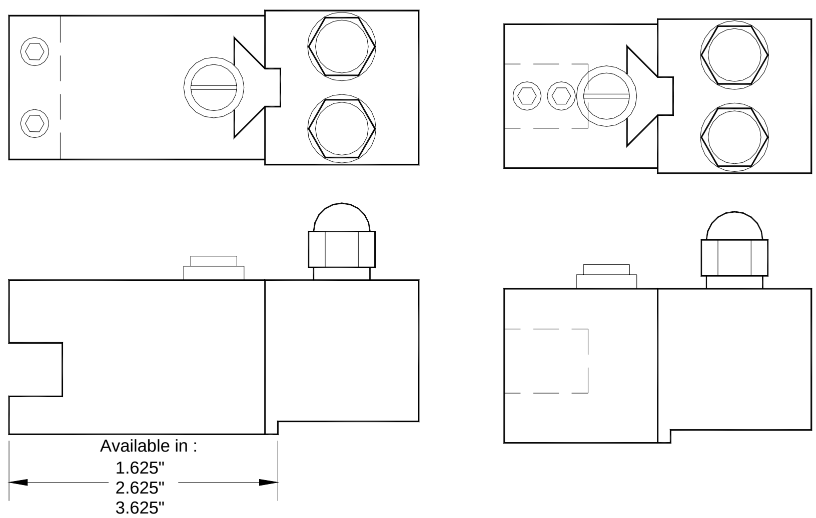
The holders are available in a wide range of styles. Both square shank and round shank holders can be used.

A cutoff holder is available to hold HSS blades.

Extension heads are available in 3 sizes.(1.625”,2.625”, & 3.625”)

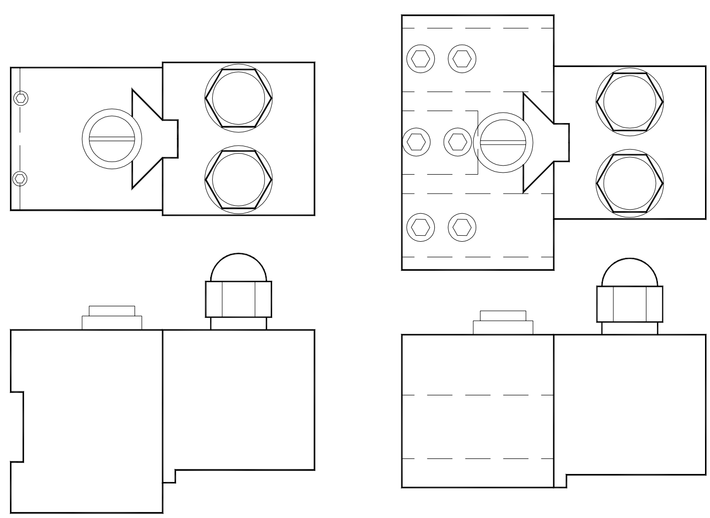
Single & triple bore holders have either 5/8” or 3/4” bore.

Each head can be attached to a standard post, this allows a wide range of setups to be achieved quickly & easily.



Setups can be stored with the tools left on the head. Next time the setup is required simply load the heads onto the pre-positioned posts.

Live holders use the same posts as tool heads, so the system is completely flexible.



Live tool holders

The design of this Posi-Lock holder fits the standard tool post or the vertical (3.5”) high tool post. You can hold live spindles in an axial, radial, or vertical orientation.

These tools when linked to an indexing spindle allows the machining of these types of features:

- Off center holes
- Face Slots
- Face holes
- Side holes
- Saw slots

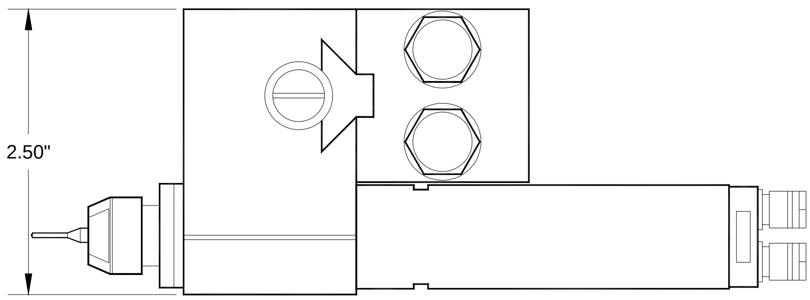
NSK America offer a wide range of live spindles ranging from 2,000 to 120,000 RPM . These spindles are air or electric powered, and are suitable for drilling up to 0.25”.

If you have ever tried to drill a sub 0.03” hole you can see the benefit of these high speed spindles.

Axial live tool holders

This example shows an axial holder and a small drill, with this configuration you can drill bolt patterns

Or mill a straight slot on the face of the part. With a C axis machine, milling profiles on the face is possible.



A fixed center height holder is available.

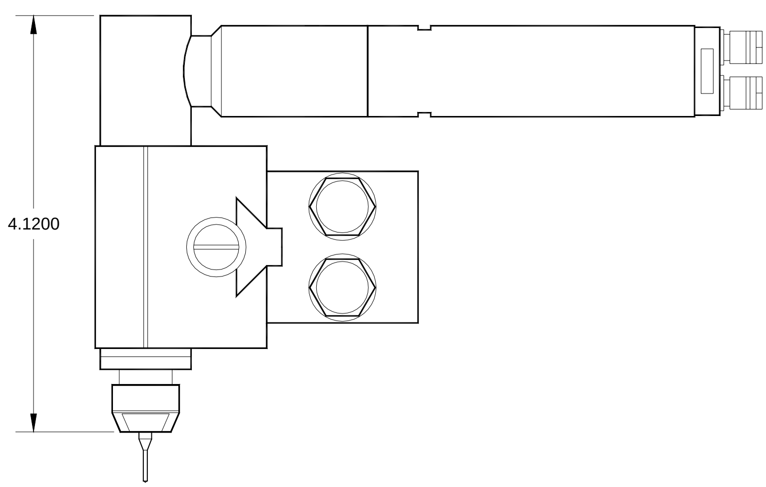
Radial live tool holders

This example shows a radial holder with a small drill for drilling side holes or milling slots along the length of a part.

With a C axis machine it is possible to mill helical slots and engrave part numbers.

Other choices are available for side drilling.

An axial holder with a small head 90^{O} spindle is also available.

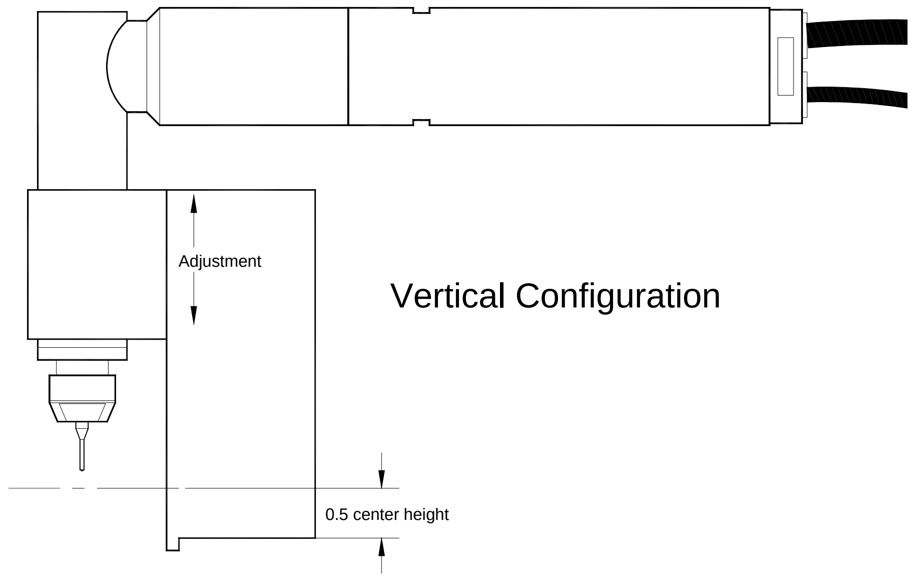


Vertical live tool holders

A tall vertical holder is available to mill flats with the end of an end mill, or to slot longer slots that require a movement in X axis. Profile milling can also be done with type of holder.

A RAS100 spindle (air),
or a RAS150E (Electric)
spindle are used to reduce the height of the live tool package.

The Posi-lock holder allows for fine adjustment of height for the tool.



Angle holes

An adjustable holder base is available to allow drilling of angles into the part. This provides 30^{O} of adjustment between axial and radial orientation.