PRODIGY CNC LATHE

GT-27

Operators' Quick Reference

SNK AMERICA, INC.

ADDRESS : 1800 Howard Street
Elk Grove, Il. 60007, U.S.A.

•Internet: www.snkamerica.com • E-Mail: bwhelan@snkamerica.com
mrogosienski@snkamerica.com

•Telephone: (847) 364-0801 • Fax: (847) 364-4363

Preface:

This document provides a quick reference for shop-floor personnel setting up and operating the Prodigy GT-27 lathe and is intended to be a companion to, not a replacement for the GT-27 Operators' Manual and/or Fanuc Operators' Manual. All information in this manual can be found within the GT-27 Operators' Manual and/or the Fanuc Operators' Manual along with details not found in this document. It is important that you take the time to familiarize yourself with the Operators' Manual(s) before attempting to operate the lathe.

This manual's focus is to get your Prodigy lathe up and running quickly and efficiently. The methods demonstrated here are not the only ways to accomplish this goal.

Safety:

WARNING! Users of the Prodigy GT-27 lathe must be aware of the dangers associated with machine operation. The GT-27 is shipped with guards and door interlocks provided to protect the operator from harm. Always wear safety glasses when setting up or operating this machine and never sacrifice safety to save time by disabling or removing the door interlocks.

Table of Contents

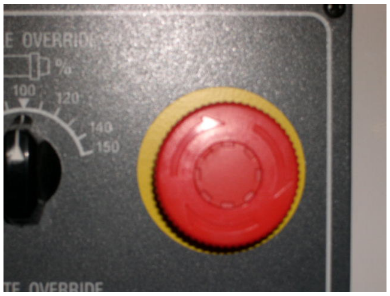
Release Record

Revision	Date	Revised by	Comments
A	01/02/2005	Michael Rogosienski	Initial release

Start up

Verify that all safety devices are in place and in good working condition.
Verify that shop air is supplied to the machine
Verify that the machine is level to within 30 microns per meter.
Turn on the Main power circuit breaker

Verify that E-Stop is not active by turning E-Stop button clockwise.   If E-Stop is active, the large, red button will "pop-up".



Turn on the "Power" switch to "On"

Shut down

Wait until the machine finishes cycling
When all motion of the machine stops, turn the "Power" switch to "Off"
Turn off the Main power circuit breaker

Reference Return

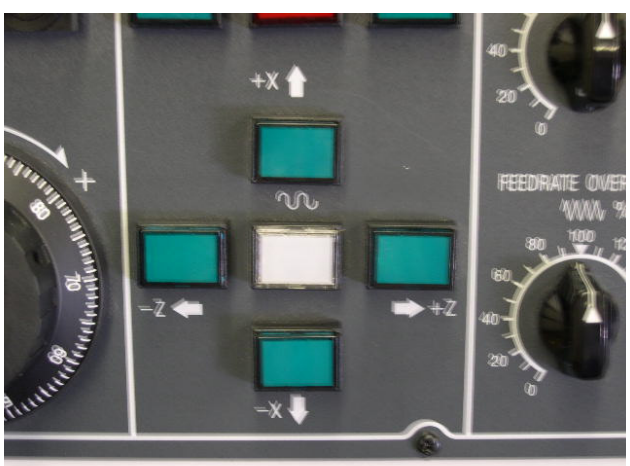
What is it?

When C.N.C. machines are built, a physical location in the machine is defined as a Reference position. The Prodigy lathe needs to be sent to that position after starting up so the machine knows where it is. The machine can then accurately position itself as required.

Performing Reference Return

1. Perform Start up as described in Section A of this document.
2. Wait for the feed hold button to stop blinking. The machine is "booting up."
3. Set Mode Selector to "Jog."

Use Jog Buttons to move both the X-axis and Z-axis off of Reference Return position if required. It does not matter which direction is chosen. Use care not to jog cutting tools into the work-piece.



Set Mode Selector to "Ref.Rtn."

Display the position page on the screen by pressing the "Pos" button.

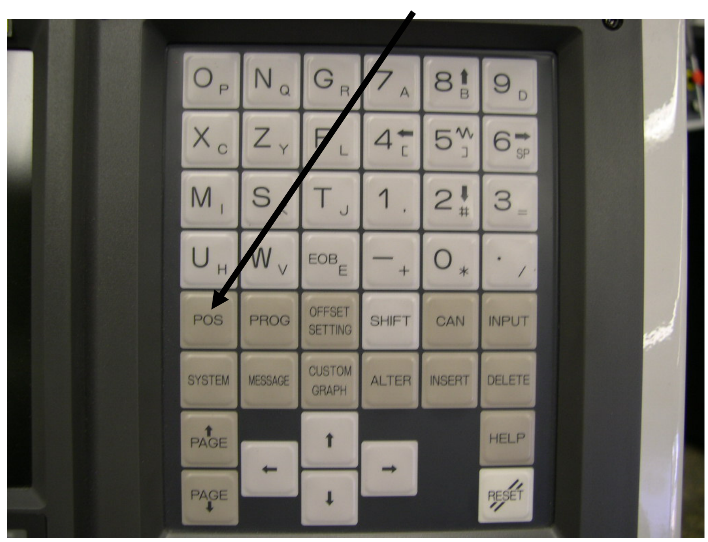
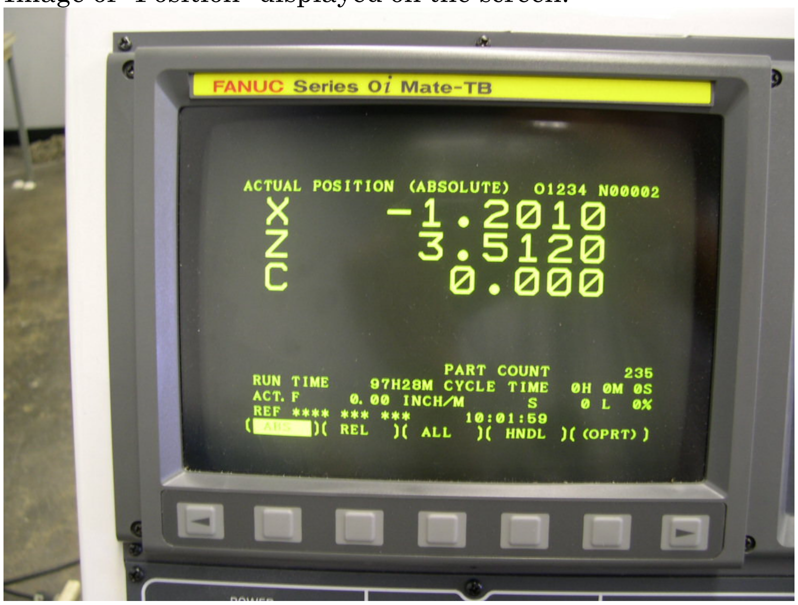
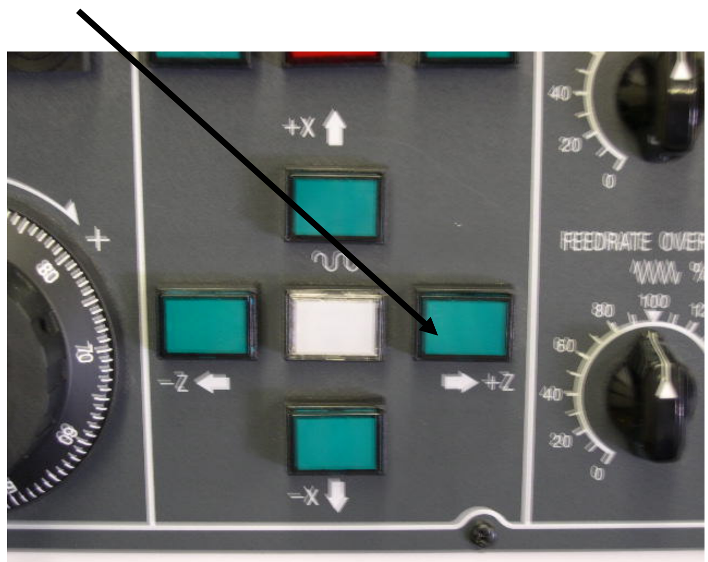


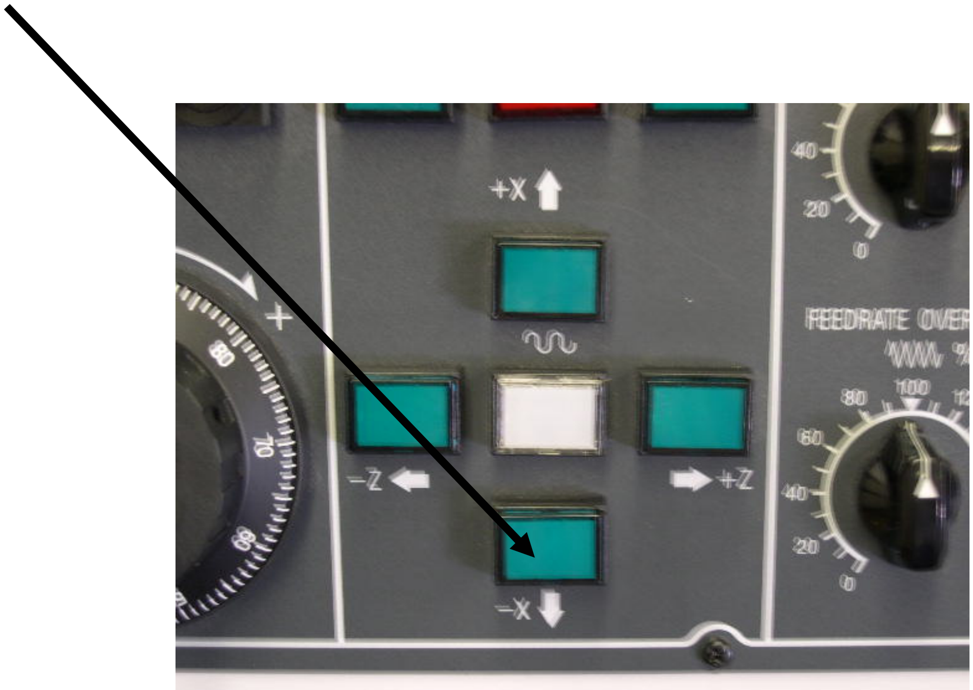
Image of "Position" displayed on the screen.



It is safer to return Z-axis first. Press and hold the Z-plus button until the position display indicates motion has stopped.



Press and hold the X-minus button until the position display indicates motion has stopped.



The “Reference Return” function is now completed and the machine is ready for production operation.

Reference Return using program commands.

Once you have established a manual reference return at start-up, it is convenient to use this location to safely position the tool plate in preparation for running the next tool in a process. The Prodigy GT-27 is equipped with the ability to position the Z-axis at one of two reference positions with G-codes. These commands are useful, and safer, if used at the start and end of each process in a program.

G28, Return to reference position.

To position the tool plate at the reference position of the Z-axis using Manual Data Input, use the following procedure:

1. Set the "Mode" switch to "MDI."
2. Press "Prog" to display the program.
3. On the keypad, enter the following: "G28W0 (zero);" (The button labeled "EOB" is used to enter a colon (;) at the end of a block. This is why "EOB" creates the "end of block" symbol.)
4. Press "Insert", the command will appear at the top of the display and the machine is now ready to execute the command.
5. Press "Start" two times to execute the command. The machine will move to the Z-axis reference position.

IMPORTANT! Do not use "G28Z0 (zero);" The Z-axis will move towards the spindle instead of to the reference position.

To position the tool plate at the reference position of the X-axis, use the following procedure:

1. Set the "Mode" switch to "MDI."
2. Press "Prog" to display the program.
3. On the keypad, enter the following: "G28U0 (zero);"
4. Press "Insert", the command will appear at the top of the display and the machine is now ready to execute the command.
5. Press "Start" two times to execute the command. The machine will move to the X-axis reference position.

G30, Return to second reference position.

Additionally, you may want to position the tool plate further away from the spindle than the Z-axis reference position attained with the G28 command due to the use of a tool exceeding 2" in length. The available option allows positioning 49mm (1.929") further away from the spindle with the program command: "G30W0".

To position the tool plate at the second reference position of the Z-axis using Manual Data Input, use the following procedure:

1. Set the "Mode" switch to "MDI."
2. Press "Prog" to display the program.
3. On the keypad, enter the following: "G30W0;"
4. Press "Insert", the keyed in text moves to the top of the display.
5. Press "Start" two times to execute the command. The machine will move to the Z-axis reference position.

IMPORTANT! Do not use "G30Z0;" The Z-axis will move towards the spindle instead of to the reference position.

The X-axis has no second reference position.

Cartesian Coordinate System

What is it?

The Cartesian coordinate system was developed by the mathematician Descartes during an illness. As he lay in bed sick, he saw a fly buzzing around on the ceiling, which was made of square tiles. As he watched he realized that he could describe the position of the fly by the ceiling tile he was on. After this experience he developed the coordinate plane to make it easier to describe the position of objects. A C.N.C. machine tool uses this system to describe part geometry to be machined. This definition of the Cartesian coordinate system was provided by the Shodor Education Foundation, Inc. at: www.shodor.org

Coordinate plane (Cartesian)

What is it?

A plane, with a point selected as an origin, and some length selected as a unit of distance. Coordinates of a point are determined by the distance of this point from the origin, and the signs of the coordinates are determined by whether the point is in the positive or in the negative direction from the origin. In a part program, each axis of movement requires a coordinate plane. The Prodigy lathe uses three axes of motion, so three coordinate planes are required.

The Prodigy lathe uses industry standards to describe motion along each axis.

A traditional lathe is designed to rotate a cylinder (bar-stock, part blank, etc.) while a stationary tool moves along an axis to make a cut. Motion parallel to the centerline (length of the part) is commanded using either the letter Z or W and a numerical value. Motion perpendicular to the centerline (diameter) would require a command using either the letter X or U and a numerical value.

The Prodigy lathe has the capability to use rotating (live) tools to perform milling functions. This third axis of motion used when performing processes using rotating tools would require a command using either the letter C or H and a numerical value to position the spindle in degrees of rotation.

It is possible to command in *absolute* and *incremental* modes. (details of *absolute* and *incremental* commands are addressed later in this section.) therefore, there are six possible unique commands that govern axis motions.

Absolute mode.

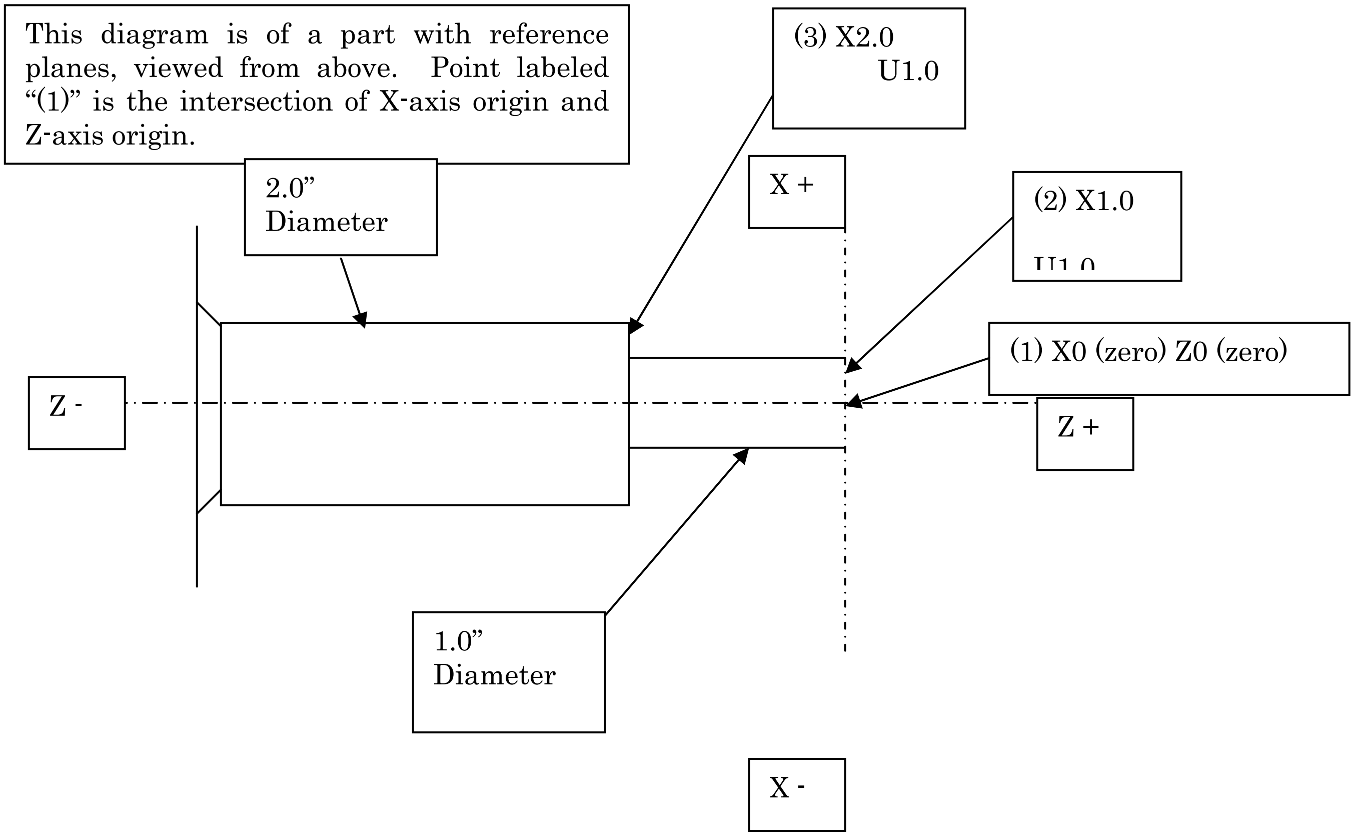
An absolute command moves an axis a described distance from a plane's origin.

Incremental mode.

An incremental command moves an axis a specific distance from where it is presently located, regardless of how far away from the origin point the start of the move is.

Axis Origin

The programmer may choose any location for an axis' origin point. It is desirable from a programmer's point of view to define origin points so that part-print dimensions are usable to define positioning. This is accomplished on a lathe by defining the origin of the X-axis to be the centerline of a cylinder and the origin of the Z-axis to be in line with the surface of the part positioned furthest away from the collet. The origin points are often referred to as "X0 (zero)" and "Z0 (zero)". The illustration on the next page demonstrates the relationship between absolute and incremental positioning of the X-axis. The point labeled "1)" shows the intersection point of X0 and Z0. If you wanted to position a tool at this location, the program would read "X0Z0".



Diameters: X (absolute) or U (incremental)

If you wish to command your tool to position to machine a 1" diameter, and your tool is positioned at the origin point of X0 (Point labeled "(1)") as shown there are two commands that would work: X1.0 (absolute) and U1.0 (incremental).

The point labeled "(2)" shows the tool positioned at X1.0. If you then wish to move to a diameter of 2", the two commands would be X2.0 (absolute) or U1.0 (incremental). The U1.0 statement is an *incremental* command that moved the X-axis a distance of 1.0" from where it was previously (1.0" diameter.) The point labeled "(3)" shows the tool positioned at the absolute position of X2.0 after executing the command.

Lengths: Z (absolute) or W (incremental)

The above theory applies to each axis' incremental command. If you are positioned at Z0 and want to move 1" in the negative direction, the two commands would be Z-1.0 (absolute) or W-1.0 (incremental). Just as above, once you are positioned other that at axis origin, the Z coordinate would be the distance from Z origin point and the W command would be the distance from wherever the tool is. When positioned at Z-1.0, a move of 1" in the negative direction would be commanded as either Z-2.0 (absolute) or W-1.0 (incremental).

Degree of Rotation: C (absolute) or H (incremental)

The relationship of absolute and incremental commands also applies to the axis that controls rotation of the work-piece during milling operations.

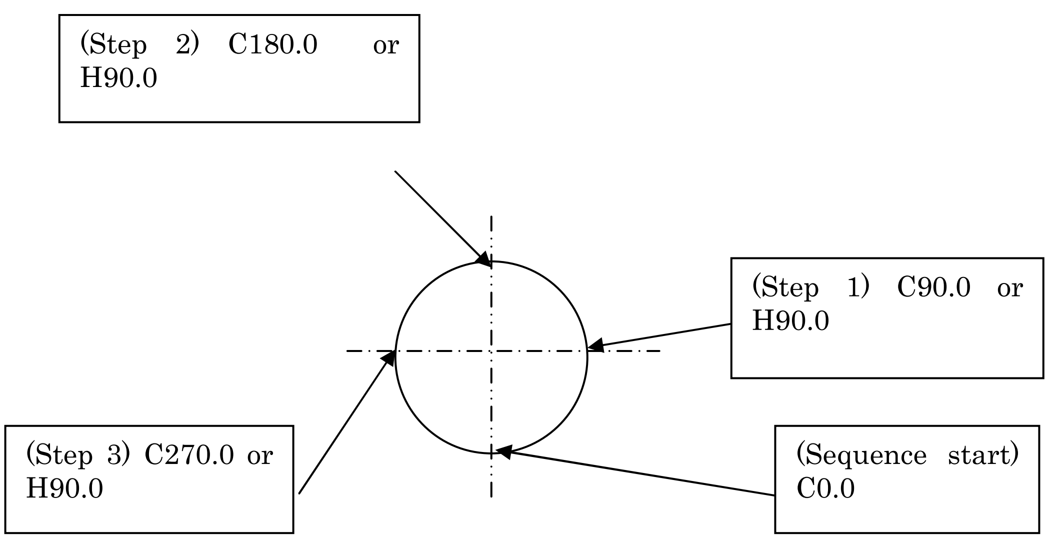
1 If your live tool is positioned at C0.0 degrees (axis origin) and you want to position 90 degrees in the positive direction, the two commands would be C90.0 (absolute) or H90.0 (incremental).
2 If your tool is positioned at C90.0 degrees, and your next position needs to be 180 degrees in the positive direction from axis origin, the two commands would be C180.0 (absolute) and H90.0 (incremental). The *incremental* command of H90.0 moves the C-axis 90 degrees in the positive direction from its present *absolute* position of 90 degrees (C90.0). The following illustration shows the C-axis positions defined above.

The diagram below illustrates the C-axis rotation programmed using absolute and incremental commands. The cutting tool is positioned at coordinate C0 (axis origin) at Sequence start.

At Step 1 the C-axis has rotated 90 degrees in the positive direction. The distance from axis origin is the same amount as the distance from the tool's previous location. This is why the absolute C90.0 command contains the same numerical value as the incremental H90.0 command.

At Step 2 the C-axis has rotated another 90 degrees in the positive direction. The absolute C180.0 command reflects the distance the axis has positioned from axis origin. The incremental H90.0 command reflects the distance the axis has rotated from the tool's previous position.

Step 3 illustrates another 90 degrees of rotation from the tool's previous position. The incremental H90.0 command does not consider where axis origin is, but the absolute C270.0 command references axis origin.



Note: An *incremental* command of H0.0 while milling will result in no motion.

Review:

The Prodigy lathe uses three axes of motion. These three axes of motion can be commanded in absolute or incremental modes. When commanding a diameter, use either X or U. When commanding a length, use either Z or W. When commanding degrees of rotation, use either C or H. An absolute command moves a distance measured from the an axis' zero point, an incremental command moves a distance measured from where a tool is presently located regardless of the present distance from zero point.

Offset

What is it?

Offsets are electronic adjustments that allow the user to inform the machine where a cutting tool is located.

Practical Use of Offsets

Each cutting tool requires a unique set of offset data. This adjustment tells the machine how far the cutting point of the tool is from the reference position of a reference plane.

The Prodigy lathe has several types of offsets designed to accomplish different tasks. They are: Work coordinate, geometry, wear, and radius compensation. We shall discuss them in the order in which they are typically used during setting up the lathe.

Work Coordinate

This offset is used to tell the machine how far the tool-plate must move from "home position" to where it is determined that Z0 (zero) of the part is located. Home position is where the tool-plate positions when you start-up the machine and perform a Reference Return as instructed in Section B of this document. As different length parts are run, this offset is adjusted to compensate. Because the distance from the X-axis reference position to the centerline of the spindle never changes and the location of C0 (zero) never changes, a Work Coordinate is only required for the Z axis.

Geometry

Each tool is placed on the tool-plate with a unique distance from its cutting point to the reference position of both the X and Z-axes. Because the distance from the reference positions to a tool's cutting point in both X axis and Z axis is unique, each tool must have both an X and Z geometry offset. Typically, the geometry offset chosen will be the same as the tool number assigned to a tool in the program.

Wear

A wear offset is used to make small adjustments without disturbing the geometry offset. This feature allows the operator to recover quickly from mistakes made when making small (typically less than .005") offset changes and can be useful in tracking tool wear. The geometry and wear offsets are added to the work coordinate to identify the tool's cutting point's distance from the X axis and Z axis reference positions.

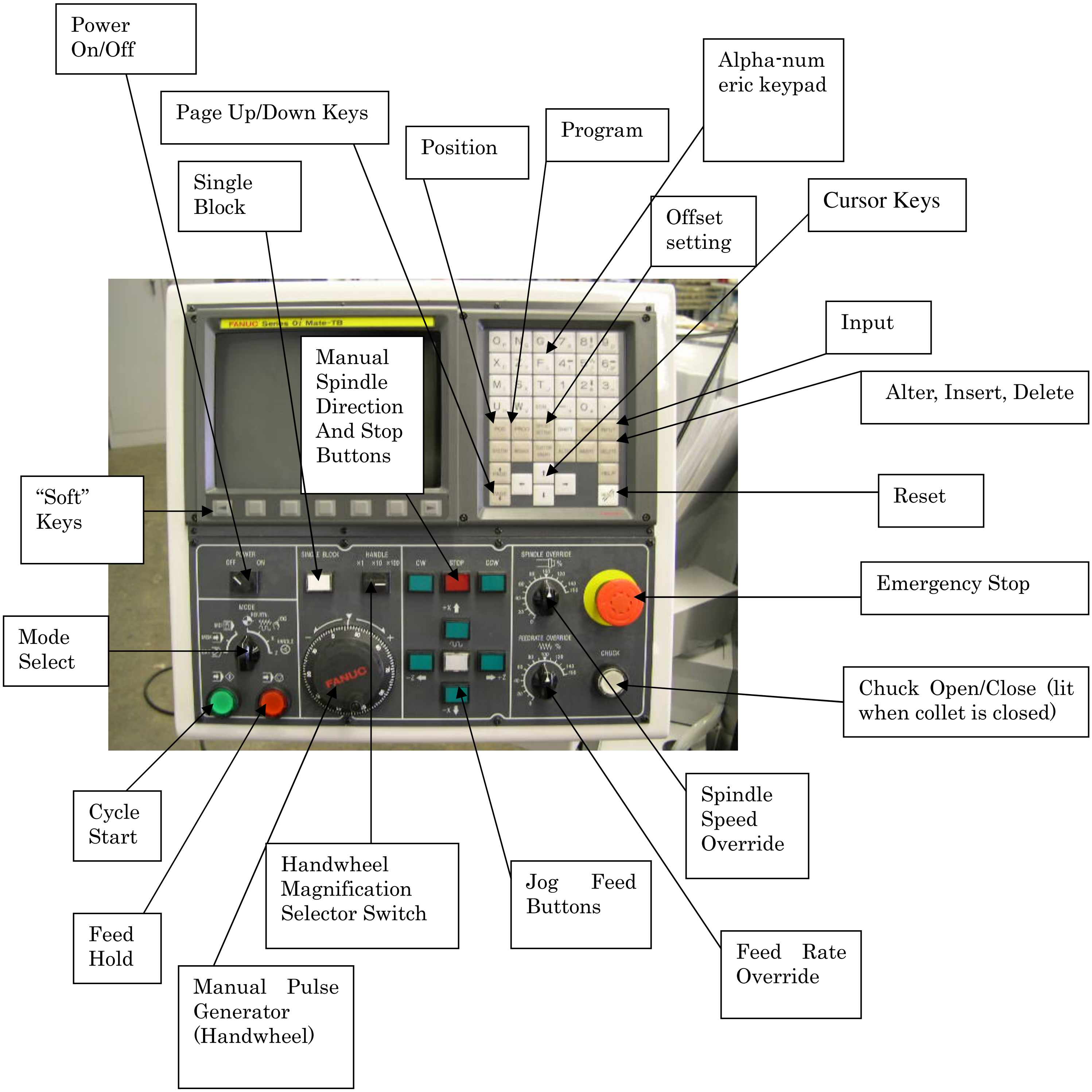
Radius Compensation

This offset tells the machine what radius the cutting tool has and in which direction the tool is facing. More detail concerning use of offsets can be found in Sections F and G.

Operators’ Controls

Located on Control

Operator controls located on the control pendant are illustrated below. Refer to the Prodigy Operators’ Manual for further details.



Additional controls are accessed via “soft” keys. The following steps will display additional controls:

- Press “Offset Setting” to access work, geometry and wear offsets.
- Press the far-right “soft” key repeatedly until (seting) becomes available above one of the soft keys. (seting) means “setting” and is abbreviated due to space constraints.
- Press the “soft” key below (seting). Use Page up or Page down as needed until Setting (Handy) screen is displayed. Communication Parameters are located here.
- Press Page Down twice until Setting (Timer) screen is displayed. You can now access the Parts Counter.
- Press the far-right soft key until (opr) becomes available above one of the “soft” keys. Press the “soft” key below (opr) to access the Rapid Override function.
- Press Page Down once to access the following options: Block Skip, Single Block, Machine Lock, Dry Run, and Protect Key. (Protect Key disables the operator panel’s program editing buttons.)
- Press Page Down once to access the following options: Optional Stop, Coolant On/Off and DNC & Tape.
- Use the Cursor Left/Right to enable/disable any option desired.

Setting Reference Plane (Work Coordinate of Z Axis)

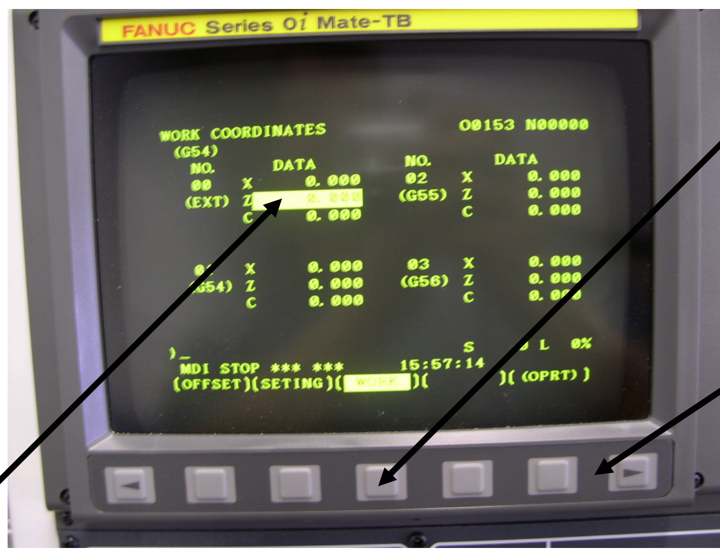
IMPORTANT! Before attempting to set the work coordinate, verify that you have performed a Reference Return after starting up the machine.

Decide where you want Z0 (zero) to be. If setting up for the first time or if all tools have been removed, use Method 1. If changing to a part of a different length and tools that you want to re-use are mounted, use Method 2.

Method 1

Set value in Work Coordinates to be 0 (zero) using the procedure below:

1) Press "Offset Setting"
2) Press the far-right "soft" key as many times as it takes for "work" to appear on the screen above one of the soft keys, then press the soft key below "work" to get to the Work Coordinates screen displayed below.

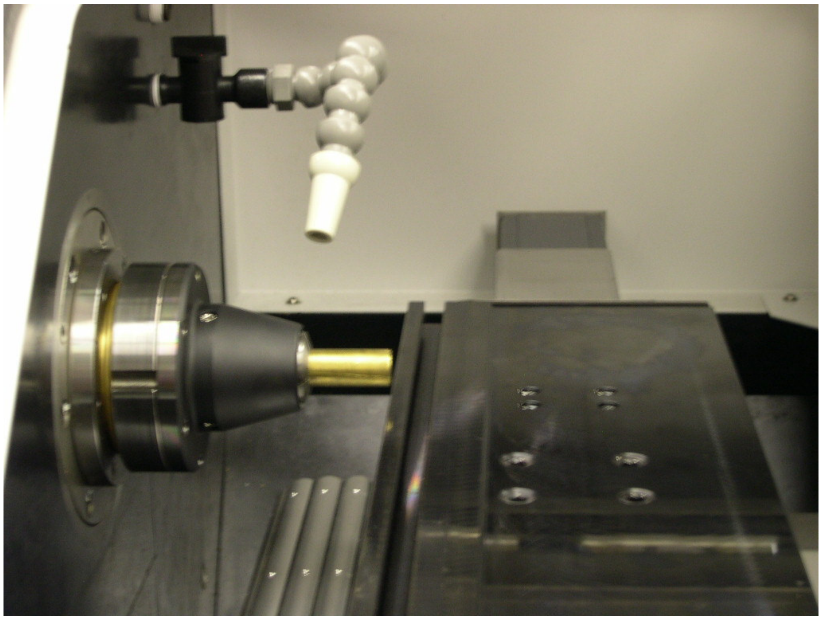


"Soft" key directly below "work" displayed on the screen

Far-right soft key

3) Use "cursor-down" key to highlight "(EXT) Z" as shown above.
4) Next, press "0"(zero), then "input" to set the work offset to zero.

Position Z-axis so the front face of the tool plate is where you have decided to locate Z0 (zero).



If you have changed the display, repeat steps above to restore the Work Coordinates page.
Press “Z”, then “0” (zero), then press the “Measur” soft key.
Perform a manual Reference Return of the Z-axis as shown in Section B of this manual.

Method 2

1. Calculate the distance you want to move Reference Zero of the Z-axis.
2. (Example: If the part to be set up next is one inch **longer** that the previous one, you would want to move Reference Zero one inch in the **positive** direction.)
3. Verify that (EXT) Z is highlighted, if not, use cursor keys to make it so.
4. Key in “1.0” then press the (+ INPUT) soft key to shift Z0 1” in the **positive** direction.

Setting Cutting Tools

Geometry Offsets

IMPORTANT! Before attempting to set the geometry offsets, verify that you have performed a Reference Return after starting up the machine as described in Section B of this manual and have set a work coordinate offset as described in Section F in this manual.

Each tool used needs a geometry-offset in both the X and Z-axes. All offset functions can be accessed by pressing the "Offset Setting" key. Refer to Section E, Operators' Controls.

Setting Z-axis Geometry offsets

1. Position the tool to be aligned with the end of the work-piece. It is a good practice to use a piece of paper or a thin shim placed between the tool tip and the work-piece to lessen the chance of damaging the cutting tool.
2. Use the axis direction keys to get the tool close to the work-piece.
3. Use the pulse generator (hand-wheel); set the resolution to X10.
4. Select the Z-axis.
5. Hold the shim/paper between the tool and the work-piece. While slowly turning the hand-wheel in the minus direction (counter-clockwise), wiggle the shim/paper until the tool pinches the shim/paper against the work-piece. Wiggling the shim/paper will help you feel when the tool is close to, but not touching the work-piece.
6. Cursor up or down to the desired geometry offset to set.
7. Key in "Z", (width of paper or shim used), (Measur) soft key.

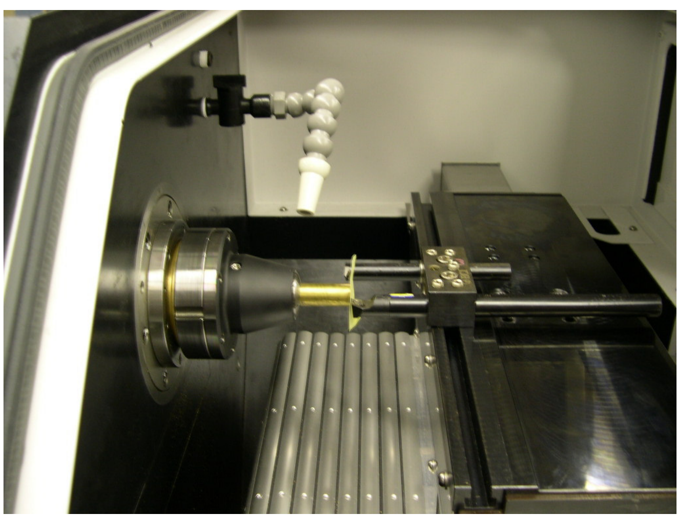


Image of a tool positioned prior to setting a Z-axis geometry offset.

Setting the X-axis geometry offsets

1. Rotate the spindle in MDI mode as follows:
2. Place mode selector to MDI
3. Press "Prog"
4. Use the keypad to enter "G97S1000": then enter "M3" or "M4" to select forward or reverse as needed.
5. Enter "EOB"
6. Press "Insert", the command seen at the bottom of the screen will move to the top of the screen.
7. Press "Start": The spindle rotates as commanded. Once this step is completed, you can re-start and stop rotation as needed by using the spindle rotation buttons on the control in any Manual mode.
8. Take a light skim cut on the diameter of the work-piece using the hand-wheel moving in the Z minus direction, then move the tool off of the work-piece in the Z plus direction. Do not move the X-axis during this sequence.
9. Stop the spindle and measure the diameter you just cut.
10. Cursor to the desired geometry offset to set.
11. Key in "X", "(diameter)", (Measur) soft key.

<u>Caution!</u> The Prodigy GT-27, like all gang-tool lathes, allows the operator to cut on either side of the centerline of the Z-axis by programming in either X-plus or X-minus. If the tool is positioned closer to you than the work-piece, the tool is positioned on the X-minus side of Z-axis centerline. When this is the case, you must include a minus sign when keying in the "(diameter)". [Example: If the diameter is .500", key in "X", "-.500", (Measur)] and the program's X-axis coordinates must also be minus values.

If you are setting a non-revolving tool such as a drill, tap or reamer to cut along the centerline, the keyed in "(diameter)" will be 0 (zero).

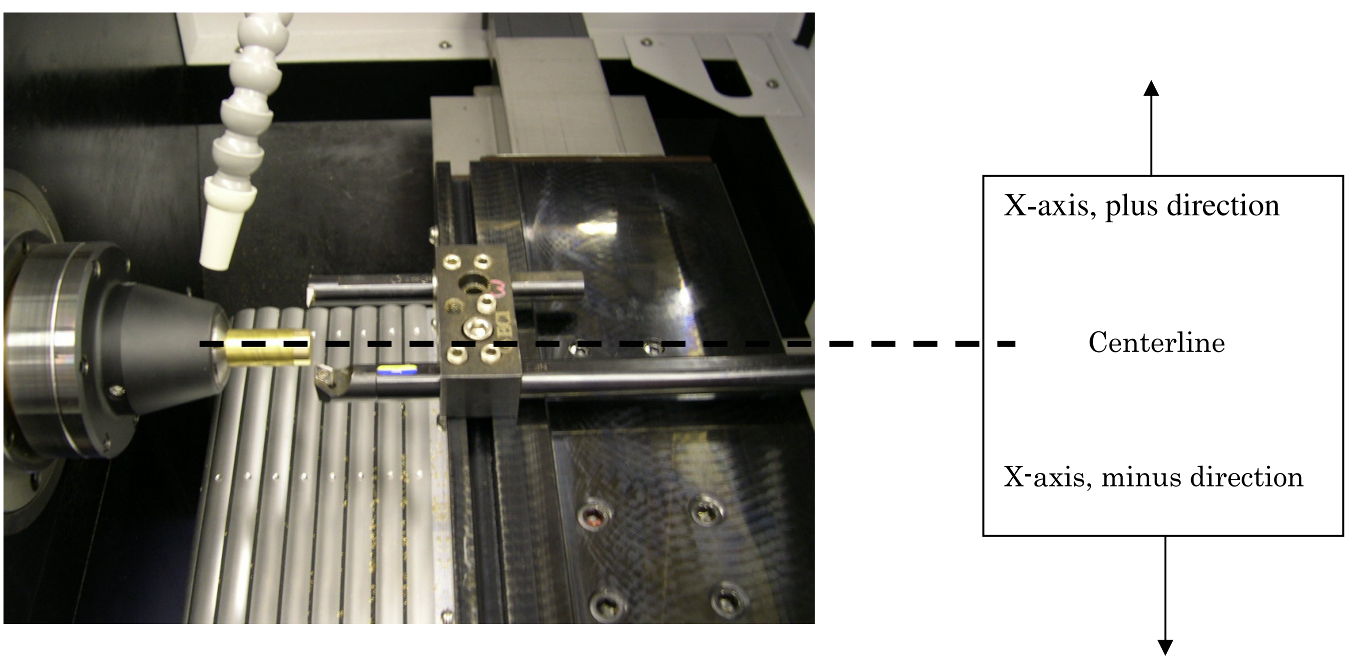


Image of a tool positioned to set an X-axis geometry offset on the X minus side of part centerline.

Wear Offsets

Wear offsets are used to make small adjustments needed due to tool wear or deflection.

Setting the X-axis geometry offset

1. Press the “Offset Setting” button to display Offsets
2. Change the display to the Wear offset page by using the “soft” keys.
3. Cursor up or down to the desired offset to change.
4. Cursor left or right to select the desired axis to offset.
5. Key in the desired adjustment and press the (+ Input) soft key.

Caution! The (+ Input) soft key will add or subtract from the current value for you. The (Input) soft key will over-write the current value.

Radius Compensation Offset

When the program is written using part-print dimensions, the machine assumes that all tools used have a zero radius at the cutting point. In fact, most cutting tools used to turn a part, and all end-mills used to mill a part have a radius on the tip. When programming an angle or radius arc with a turning tool, or using the “polar” milling feature to machine a hexagon or other shape, the program must be altered to allow for the fact that a tool has a radius. The programmer would need to re-calculate all coordinates for geometry defining angles and radii to produce a part to print.

The radius compensation feature can save much time by letting the machine’s controller do these calculations. The Radius Compensation Offset is used to tell the machine what the radius of the cutting tool is and which direction the cutting point is facing.

Setting the radius compensation offset

Select the Geometry Offsets page. At the far-right of the display are columns labeled “R” and “T”. The “R” column is used to define the radius of the tool and the “T” column is used to define the direction of the cutting point. This data is only used when tool radius compensation is active (G41 or G42)

1. Cursor up or down to select the desired tool number.
2. Cursor right or left to the “R” column.
3. Key in the RADIUS of the tool tip or end mill and press the (Input) soft key.
4. Cursor right to the “T” column.
5. Key in the direction and press the (Input) soft key. Refer to the Fanuc Operators’ Manual for details on setting the cutting direction.

Programming

What is it?

A program directs the machine tool to perform functions and motions required to machine a part. A C.N.C. program is a language, and like any language it has words that are grouped together to communicate.

In programming, a *word* is a combination of a letter and number(s). A *block* is a word or group of words separated from other blocks by what is known as an "end of block". The "end of block" is a semi-colon (;) and is always the last symbol seen in a block. If a block contains more than one word, all commands in that block are completed before the program moves on to the next block. Blocks of words send commands to the various motors, servos and devices of the machine tool. In most cases, blocks of commands are executed in the order that they appear on the screen. The machine's controller is not able to think for itself, it only does what it is told to do. The Prodigy lathe is equipped with a Fanuc controller using industry-standard commands.

Some commands use only one word while others use additional words to modify commands. The number associated with the word's letter causes the machine to perform a unique function from within a group. Some commands are modal, meaning that they remain active until replaced by another command from within the same group. In the case of axis motion commands, the numbers will be dimensions. Below is a table illustrating examples of this relationship. For a complete list and further detail, refer to the Prodigy Operators' manual and the Fanuc Operators' Manual.

Letter of Word	Group
G	General
M	Miscellaneous
S	Speed
F	Feed
X,Z,C,U,W,H	Axis of motion
N	Place Holder; not used to command motions or functions.

M and G code list

The tables below document a list of M-codes and G-codes typically used by the Prodigy GT-27 lathe. The Prodigy GT-27 uses G-code Type A. It is acceptable to use a word without leading zeros. Example: G00 and G0 are both acceptable. For uniformity, all words listed below contain leading zeros.

G-Codes	
G00	Rapid Interpolation (traverse)
G01	Linear Interpolation (feed)
G02	Circular Interpolation (clock-wise arc)
G03	Circular Interpolation (counter-clock-wise arc)
G04	Dwell (paused motion)
G07.1(G107)	Cylindrical Interpolation
G10	Programmable data input
G11	Programmable data input cancel
G12.1(G112)	Polar coordinate interpolation mode
G13.1(G113)	Polar coordinate interpolation mode cancel
G17	X-Y plane selection
G18	Z-X plane selection
G19	Y-Z plane selection
G20	Inch Mode
G21	Metric Mode
G22	Stored stroke function on
G23	Stored stroke function off
G25	Spindle speed fluctuation detection off
G26	Spindle speed fluctuation detection on
G28	Return to Reference position
G30	2nd. Reference position return
G32	Thread Cutting
G34	Variable-lead Thread Cutting
G40	Tool Nose Radius Compensation Off
G41	Tool Nose Radius Compensation Left
G42	Tool Nose Radius Compensation Right
G50	Coordinate System Setting or max. spindle speed setting
G52	Local coordinate system setting
G53	Machine coordinate system setting
G54	Workpiece coordinate system 1 selection
G55	Workpiece coordinate system 2 selection
G56	Workpiece coordinate system 3 selection
G57	Workpiece coordinate system 4 selection
G58	Workpiece coordinate system 5 selection
G59	Workpiece coordinate system 6 selection
G65	Macro calling
G66	Macro modal call
G67	Macro modal call cancel
G70	Finishing cycle
G71	Stock removal, turning

G72	Stock removal, facing
G73	Pattern Repeating
G74	End Face Peck Drilling
G76	Multi-pass threading, canned cycle
G80	Canned Cycle Cancel
G83	Face Drilling Cycle (used with live and stationary tooling)
G84	Face Tapping (used with live tooling)
G87	Side Drilling Cycle (used with live tooling)
G90	Turning Cycle
G92	Threading canned cycle
G94	Facing Cycle
G96	Constant Surface Speed Mode
G97	Revolutions per Minute Mode
G98	Inches/millimeters per Minute Mode
G99	Inches/revolution Mode

Refer to the Fanuc Operators' Manual for additional G-codes and uses.

M-codes	
M00	Unconditional Stop
M01	Optional Stop
M02	Program Stop
M03	Spindle rotation, forward
M04	Spindle rotation, reverse
M05	Spindle rotation stop
M08	Coolant On
M09	Coolant Off
M10	Collet / chuck unclamp
M11	Collet / chuck clamp
M19	Orient Spindle (C-axis)
M20	2-Axis Mode (used for turning)
M21	3-Axis Mode (used during C-axis work)
M29	Rigid Tapping Mode
M30	Program stop and re-wind
M31	Interlock Bypass
M42	Auto Door Open
M43	Auto Door Close
M44	Parts Counter
M50	Live Tool 1 On
M51	Live Tool 1 Off
M52	Live Tool 2 On
M53	Live Tool 2 Off
M54	Live Tool 3 On
M55	Live Tool 3 Off
M56	Live Tool 4 On
M57	Live Tool 4 Off
M60	Bar Feeder COM 1
M61	Bar Feeder COM 2
M62	Bar Feeder COM 3
M63	Spindle Forward and Coolant On
M64	Spindle Reverse and Coolant On
M74	Error Detect On
M75	Error Detect Off
M76	Chamfer On (Threading)
M77	Chamfer Off (Threading)
M98	Call Sub-program
M99	Return from Sub-program

G02, G03; Circular Interpolation

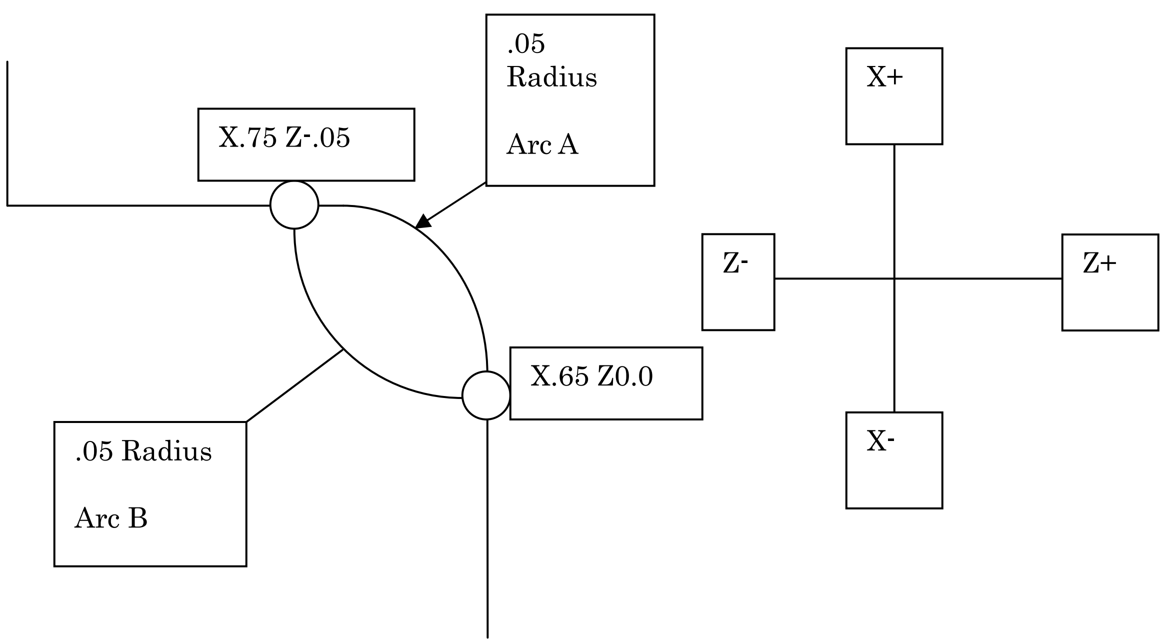
Arcs and radii are programmed using these two G-codes.

G02 will generate a clockwise arc.
G03 will generate a counter-clockwise arc.

When you generate a radius, the control must know:

The starting coordinates of the arc.
The ending coordinates of the arc.
The radius of the arc.
Clockwise or counter-clockwise motion while generating the arc.

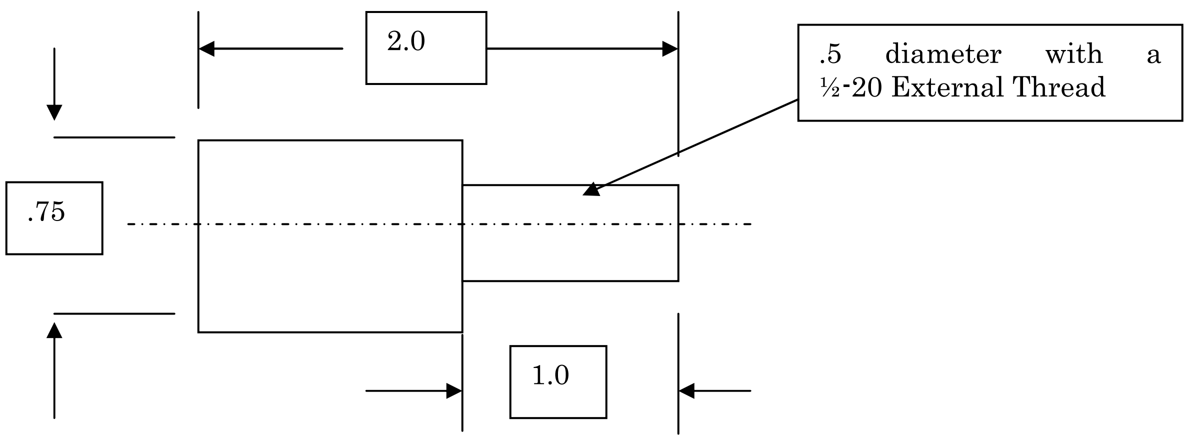
The sketch below illustrates how to use these two codes. Refer to the Fanuc Operator's Manual for more details.



Start	End	Result
X.65 Z0.0	G03 X.75 Z-.05 R.05	Arc A
X.65 Z0.0	G02 X.75 Z-.05 R.05	Arc B
X.75 Z-.05	G03 X.65 Z0.0 R.05	Arc B
X.75 Z-.05	G02 X.65 Z0.0 R.05	Arc A

G90, G94 and G92 "Canned" Cycles.

These three cycles allow programming of facing, turning and threading processes without the need to key in every coordinate required to perform a function. We will demonstrate these three functions by facing, turning and threading a sample part. Dimensions are in inches. All canned cycles finish with the cutting tool positioned at the same coordinates positioned at before starting the cycle.



G94, Canned cycle for Facing

Program Code	Explanation
G0 G97 G99 S1000 M03	Rapid mode, Inches per revolution feed mode, spindle rotation at 1000 r.p.m.
T0101	Call tool #1 and assign offset #1
X.8 M08	Position at a diameter larger than stock size and turn on coolant.
Z.1	Position the Z-axis .100 away from the face of the part
G94 X-.05 Z.05 F.005	Face past centerline, leaving .050" stock on face, .005" per revolution federate
Z.02	Another face cut, the modal feature allows us to omit G94 and X-.05, but these coordinates are still used during this cut.
Z0	Finish cut, G94 and X-.05 words still valid
G00Z.2	Exit modal command, G94 and X-.05 no longer valid, machine moves to .200" away from face of part

Next, we will turn the part using **G90** command. Program continues from coordinates stated previously: X.800 Z.200 with the spindle still rotating at 1000 revolutions per minute.

G00 Z.03	Rapid position to .030" from the face of the part
G90 X.700 Z-1.000	Turn a .700" diameter 1.0" long
X.65	.700" diameter becomes .650", still 1.0" long, modal command needs no G90 or Z-1.000. Also, trailing zeros are not required.
X.6	.650" diameter becomes .600" still 1.0" long
X.55	
X.5	Final pass at part-print X-axis coordinate.
G00 Z.2 M09	Exit modal command, rapid position to .200" from the face of the part, turn off coolant.
T0	Cancel offset #1 as we prepare to change to a thread tool
G28 W0	Rapid to Z-axis reference position. W0 tells the machine to go exactly to that point. Rapid (G00) still valid because it is modal.
M01	Optional stop allows the operator to stop and check their part, clear chips, etc.

Now we will thread the part using the G**92** command.

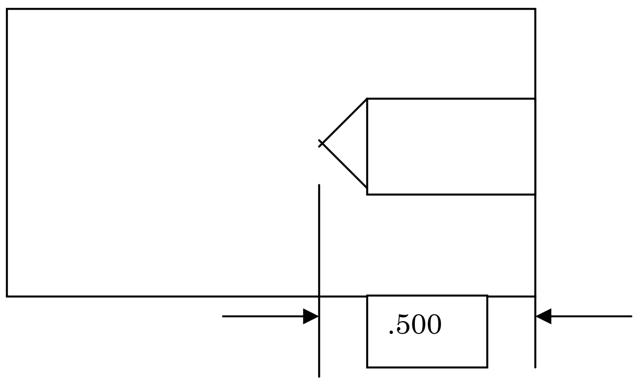
G0 G97 G99 S500 M03	Spindle rotation forward at 500 r.p.m.
T0202	Call Tool #2 and assign offset #2
X.55 Z.2	Position for threading in rapid mode
G92 X.48 Z-.5 F.05	Threading pass: .01" per side cut, .5" deep at .05" per revolution (1.0" / 20 threads per inch = thread pitch for a ½-20 thread.
X.465	.0075" per side cut, modal command needs no G92 or Z-.5
X.455	.005" per side cut
X.445	.005" per side cut
X.441	.002" per side cut
X.4405	Cut at final X coordinate
X.4405	Spring pass

Refer to the Fanuc Operators' manual for further details including tapered facing, tapered turning and tapered threading.

G83 and G87 Peck-drilling Canned Cycles

G83, Peck-drilling using the Z-axis.

Another handy, simple to use cycle is G83. This cycle is used to peck-drill. It can be used for either stationary or live-tool operations. We will discuss using it to use a stationary tool to peck-drill a hole. Because the tool is stationary, our X-axis coordinate must be X0 (zero). Consult the Fanuc Operators' Manual for additional details on these two drilling cycles as well as another alternative for Z-axis processing using a stationary tool: G74



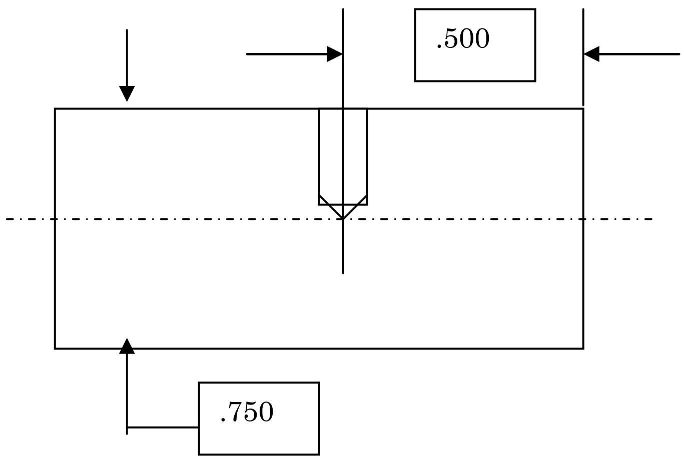
Program Code	Explanation
G0 G97 G99 S1000 M03	Rapid mode, Inches per revolution feed mode, spindle rotation forward at 1000 r.p.m.
G80T0101	Canned cycle for drilling cycle cancel, call tool #1 and assign offset #1
X0	Position the X axis at x-axis origin point
Z.1	Position the Z axis to be .1" from the face of the part
G83Z-.5Q150F.004	Peck drill to .500" deep, using a feed-rate of .004" per revolution and each peck to be .015". The control will calculate how many pecks to take to reach final depth.
G80	Canned cycle drilling cycle cancel
T0	Cancel tool offset
G0G28W0	Rapid mode, Return to reference position
M5	Spindle rotation stop
M30	Program stop and re-wind

Notes:

Peck amounts are defined using the letter "Q" and no decimal point.
Peck-drilling with a live tool requires additional steps:
M05; spindle rotation off must be active prior to drilling.
Because the spindle is stopped, G98, inches per minute feed mode is required.
In G98, feed-rate must be calculated in inches per minute.

G87, Peck-drilling a cross-hole using the X-axis.

Peck-drilling can also be used to drill a cross-hole using a rotating tool. The program format is similar to G83. 3-Axis select must be engaged using the M21 command. Below is an example with final-depth (X-axis coordinate) intersecting the centerline. X coordinates are in diameters; in this case stated as “X0”: drill to centerline.



Program Code	Explanation
G0 G98 M5	Rapid mode, Inches per minute feed, spindle stop.
T0101	Call tool #1 and assign offset #1
M21	3-Axis select
M50	Live-tool #1 on
X.8	Position the X-axis to clear part diameter
Z.1	Position the Z-axis to be .100 away from the face of the part
C0	Orient spindle to zero degrees
Z-.5	Position the Z-axis to part-print coordinate
G87**X0**Q150F5.0	Peck drill to final X-axis coordinate using a peck of .015” per peck and feed-rate of 5 inches per minute
G80	Canned cycle drilling cycle cancel
G0Z.1	Rapid mode, position Z-axis to be .100 from the face of the part
M51	Live tool #1 off
M20	2-Axis select
T0	Cancel offset
G28W0	Return to Reference Position
M30	Program end and re-wind

G70, Finishing Cycle, and G71, Stock Removal, Turning

The G9x cycles discussed previously allow the programmer to accomplish tasks more efficiently by reducing the amount of lines required in a program. The G70, G71 and G72 further save time with additional capabilities. The example used above demonstrating G90, G92 and G94 will be processed below using the G70 and G71 commands, followed by the G76 threading option.

The G71 and G72 Cycles allow the programmer to define complex geometries. The G70 command references the information used in G71 to execute a finish cut with the same, or a different finishing tool without re-defining the geometry. The G9x cycles used in the example above only have the capability to execute straight-line cuts. It is not possible to generate edge breaks or radii with G9x cycles.

In the example below, we will add a chamfer at the threaded end and a .030" edge break to the same part used in the sample above. Tool Radius Compensation will also be incorporated into the following example allowing for further flexibility not found in the G9x cycles.

In the example below, the path of the finish tool is identified immediately following the two lines that call the G71 stock removal cycle. Later in the program, the G70 finishing cycle uses the geometry identified in the G71 stock removal cycle. This re-use of data is both faster and less prone to errors possible during re-keying the geometry to finish the part.

The G71 cycle is written in two blocks, both beginning with G71 to provide the control with adjustable parameters to modify cutting conditions without re-writing the geometry.
The program example below contains all commands needed to machine the part. We will focus on the section of the program relating to the canned cycle.

The first block is written: G71U.05R.005.
U.05 tells the cycle to take a .050" depth of cut each pass. This command is in diameters.
R.005 tells the cycle to retract .005" in the X-axis each pass.

The second block is written: G71P101Q102U-.01W.003F005.
P101 identifies the block beginning with **N101** as the first block containing the part geometry.
Q102 identifies the block beginning with **N102** as the last block containing the part geometry.
U-.01 tells the cycle to adjust the geometry .010" in the negative direction of the X-axis.
W.003 tells the cycle to adjust the geometry .003" in the positive direction of the Z-axis.
F.005 tells the cycle to use a feed-rate of .005" during cutting.

ATTENTION! The sign of the U-word must be adjusted according to two factors: 1: Is X-axis programmed in minus or plus? And 2: Is this process an external (turning) or internal (boring) application? This process is programmed in X-minus and this is an external application so the sign of the U-word is minus.

The "N" words must be placed first in its blocks.

A complete sample program using G70 to rough-turn and G71 with radius compensation to turn a part is below.

```
%
O1999;
(FACE AND TURN);
(K.METAL BAR #NF3-A12S-SDUCL3);
(ISCAR INSERT #DCGT-3-1-AS);
(RAD. COMP OF TOOL #1: R=.0156, T=2)
G28W0;
M20G20;
G0G40T0101;
G97G99S1000M3;
X-.8;
G50S4000;
G96S700M3;
Z.003;
G1X.03F.005;
Z.06;
G0X-.76;

G71U.05R.01;
G71P101Q102U-.01W.003F.005;
N101G0X.05;(First block of Geometry)
G1Z0F.003;
X-.4;
X-.498W-.049;
Z-.5;
X-.68;
G2X-.74W-.03R.03;
G1Z-1.0;
N102X-.8;(Last block of Geometry)

X-.9F.01;(Moves that allow radius comp.)
G0Z.3;(without causing over-cutting)
X-.76;(alarms during radius comp.)

G1G41Z.15F.01;(Engage radius comp.)

G70P101Q102;(G70 cycle copies geometry from G71 cycle)

G0G40Z.2;(Cancel Radius Comp.)

G97S1200;
T0;
/G28W0;
M30;
%
```

Note: Should the operator desire to change the depth of cut during the roughing passes, only one edit is required: In the first G71 block, changing the U.05 is all that is required. Increasing the U-word value will cause less passes to rough out the part, decreasing the U-word value will cause more passes to be created by the control. The end result will be the same, as the G71 cycle will end with a semi-finished part .010” larger in diameter with .003” left on each face.

The G72 cycle uses a similar format for taking multiple facing cuts. It is not discussed here, refer to the Fanuc Operators' Manual for details.

G76, Multiple Threading Cycle.

Just as the G71 is more powerful and flexible for turning than the G90 cycle, the G76 cycle provides options not found in the G92 thread cycle. Manual machinists are trained to set the compound of their lathe to 30 degrees to cut a ½-20 thread. This practice will reduce the width of chip and load on the tool while roughing passes are executed to extend tool life. The machinist would in-feed the X-axis on the last pass to clean up both flanks of the thread. The G76 cycle is written in two blocks, both beginning with G76. As with all canned cycles, the G76 cycle ends with the tool positioned at the coordinates set before beginning the cycle.

The first G76 block is written as follows: G76P010160Q5R5.
P010060 sets three parameters with three pairs of numbers.

> The **first pair**, 01, tells the cycle to execute one spring-pass at finished diameter.
> The **second pair**, 01, sets the amount of chamfer executed at the Z-axis end-point.
>
> (When a relief groove is located at the Z-axis end point, you would set this to 00. 00 will cause the cycle to thread to Z-axis end-point before beginning to retract the X-axis. Setting 01 causes the cycle to begin retracting before reaching the Z axis end-point.)
> This example would create a chamfer at the Z-axis end point .1 (one tenth) as wide as the lead of the thread. The lead of a 20-pitch thread is .050", so the chamfer at the Z-axis end-point will be .005" wide.
> The **third pair**, 60, sets the angle of tool tip. This example is a ½-20 thread. The Included angle of a ½-20 thread is 60 degrees.
>
> (Refer to the Fanuc Operators' Manual, Pg. 152 for tapered threading, Pg 153 for details of cutting, and Pg 154 for details of thread cycle retract.)

Q5 sets the minimum depth of cut. This command uses no decimal point and is expressed as a radius with the sign always being positive. "Q5" sets a minimum thread pass of .001" diameter.
R5 set the depth of cut of the final pass. This command uses no decimal point and is expressed as a radius with the sign always being positive. This last pass ignores the angle of tool tip setting in the "P" word and in-feeds X-axis .001" on only the last pass.

The second G76 block is written as follows: G76X.4408Z-.5P310Q85F.05.
X.4408 sets the final X-axis coordinate.
Z-.5 sets the final Z-axis coordinate.
P310 is the height of the thread. This command, like "Q5" and "R5" above, uses no decimal point and is expressed as a radius with the sign always being positive.
Q85 is the depth of cut of the first pass. This command uses no decimal point and is expressed as a radius with the sign always being positive.

The complete program example below illustrates threading a ½-20 external thread.

```
%
O2999;
(THREAD O.D.);
(1/2-20)
(ALL-5 BAR #DC-35-625)
(ALL-5 INSERT #TTR-35-093)
G28W0;
M20G20;
G0T0202;
G97G99S1000M3;
X.55;
Z.1;

G76P010160Q5R5;
G76X.4408Z-.5P310Q85F.05;

G0Z.2;
T0;
G28W0;
M30;
%
```

Notes: Modifying the cutting parameters of the G76 threading cycle is quick and easy. These are examples only, not specific recommendations.

To add another spring-pass: G76P**02**0160.
To add more roughing passes: G76X.4408Z-.5P310**Q65**F.05.
To not chamfer at the Z-axis end-point: G76P01**00**60.
To decrease the depth of cut of the final pass: G76P010160Q5**R2**.
If the thread were an ACME thread (29 degree included angle): G76P0201**29**.

G84 and M29, Rigid Tapping.

The Prodigy GT-27 is equipped with the capability to rigid tap. The control synchronizes the spindle rotation with the motion of the Z-axis allowing tapping operations to be performed without the need to purchase a tap float. This is accomplished by the use of an M-code along with the G84 command. Below is a program example using the rigid tapping option. The M29 command is NOT modal and is only in effect during the block where it is commanded.

The following conditions must be met for rigid tapping to work correctly:

1. M20, 2-Axis mode is active.
2. G97 mode, Revolutions per minute speed must be active.
3. M03 or M04, Spindle rotation must be active.
4. M29 and speed used during tapping must be the block immediately before the G84 block.
5. The G84 block contains only the Z-axis final position and the pitch of the thread.
6. The G80, Canned cycles for drilling cancel" command must be alone in the block after the G84 command.

The program example below illustrates the rigid tapping using a ¼-20 tap.

```
%
O3999
M20G20; (2-Axis mode)
(1/4 20 TAP)
(HY-PRO #2930001)
G28W0;
G0T0505;
G97G99S300M3; (Spindle rotation forward at 300 revolutions per minute)
X0;(Position at X0 (zero), required for tapping with a stationary tool)
Z.2;(Position Z-axis .200" from the face of the part.)
M29S300; (Non-modal command enabling rigid tapping at 300 r.p.m.)
G84Z-.5R0F.05;(Rigid tap ½" deep at .050 inches per minute feed-rate)
G80;(Canned cycle for drilling cancel)
G0Z.2;
M5;
G28W0;
M30;
%
```

Note: Rigid tapping is only allowed using a stationary tool and the Z-axis with the X-axis positioned at X0 (zero). Rigid tapping is not available for processes using the X-axis in-feeding.

Milling Functions

Manual lathes machine a part by rotating the work-piece and manual mills machine the part by rotating the cutting tool. The Prodigy GT-27 has the capability of performing both types of functions.

Before processing a milling function, the machine must be commanded to enter the 3-Axis mode. In 3-Axis mode, the spindle's indexing motor enables the C-Axis to position at a specific coordinate in rapid mode (indexing) as well as executing C-axis commands at a specified feed-rate (interpolation). Before processing a turning function after a milling function, 2-Axis mode must be restored. This change in functions is controlled by two M-codes. These are both *modal* commands of the same group. A modal command stays in effect until replaced by another modal command of the same group. These commands may be placed in blocks with other words.

M20, 2-Axis machining
M21, 3-Axis machining

This function has requirements that must be met:

1. The M05, spindle rotation stop command must be in effect before entering 3-Axis mode.
2. The *modal* command M21; 3-Axis mode, must be in effect before C-Axis commands will be accepted.
3. M50, M52 or M56; "Live tool rotation on must be in effect before cutting.
4. G98, feed in inches/per minute mode, must be used. Commands issued using the G99, feed in inches per revolution mode, will cause zero motion.

G12.1 and G13.1, Polar Milling.

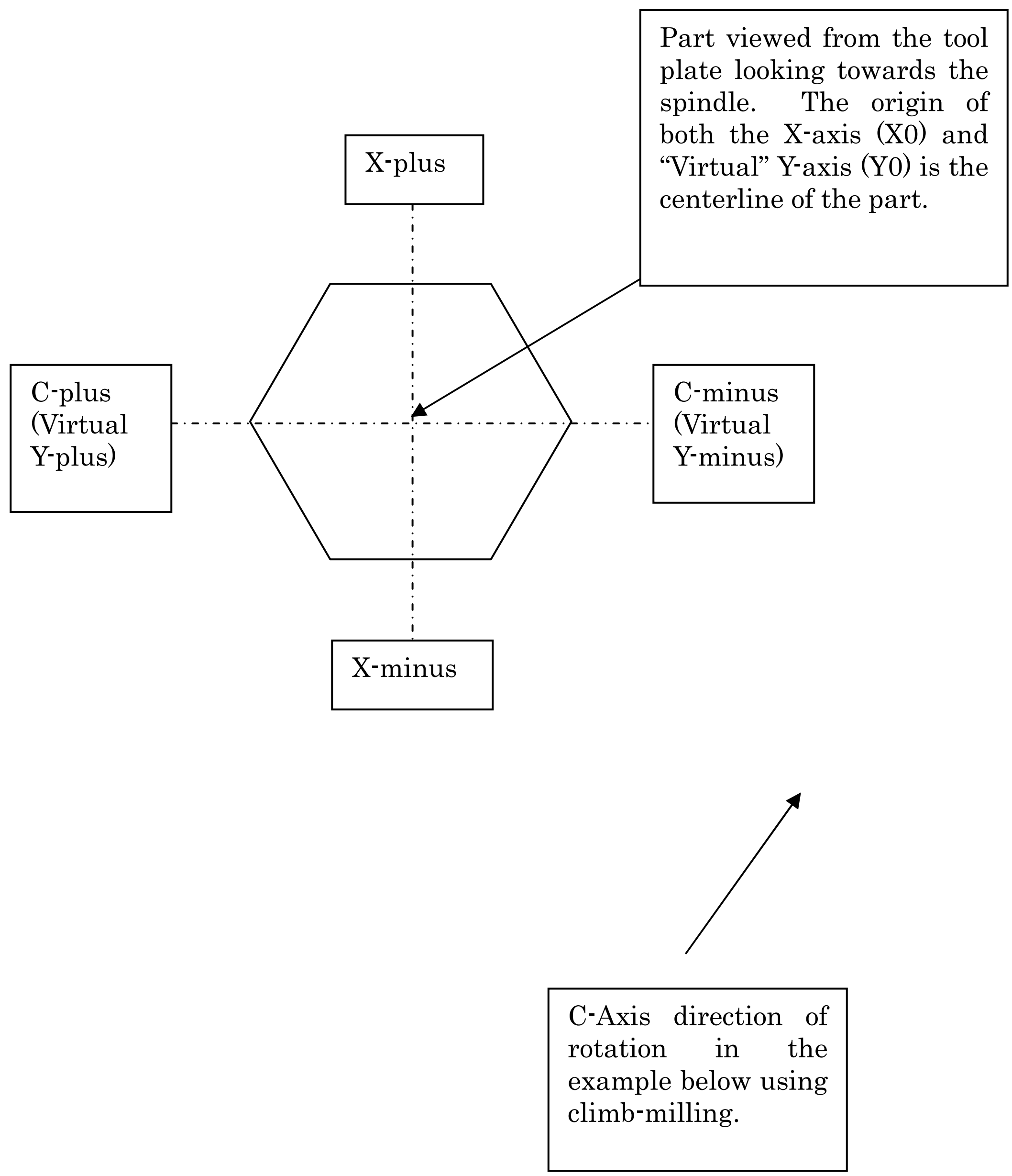
The polar milling option allows the Prodigy GT-27 to perform milling operations of complex geometries using rotating (live) tools. In this mode, the part is programmed as if the lathe were a milling machine with the C-Axis functioning as a “virtual” Y-Axis. Once M21 mode is active no additional commands are required to use the indexing function either before or after using polar milling. This mode is typically used when the live tool’s centerline is parallel to the lathe’s Z-axis.

The origin point of the “virtual” Y-Axis is the origin point of the centerline of the part.

This function has requirements that must be met:

1. The M05, spindle rotation stop command must be in effect before entering 3-Axis mode.
2. The *modal* command M21; 3-Axis mode, must be in effect before C-Axis commands will be accepted.
3. M50, M52 or M56, “Live tool rotation on” must be in effect before cutting.
4. G98, feed in inches/per minute mode, must be used. Commands issued using the G99, feed in inches per revolution mode, will cause zero motion.
5. A C-Axis command [usually C0 (zero)] must be executed before G12.1 will be accepted.
6. “Tool Radius Compensation” is required. The radius of the end-mill must be entered into the Geometry Offset page, “R” column for the tool used to mill. The “T” value in the Geometry Offset page is set to 0 (zero).
7. Tool Radius Compensation must be cancelled before activating compensation. Compensation must be cancelled while G12.1 is still active.
8. The modal command M20; “2-Axis mode”, must be restored before an M03 (Spindle Rotation, Forward) or M04 (Spindle Rotation, Reverse) command can be executed.
9. X-Axis coordinates are in diameters and C-Axis (“Virtual” Y-Axis) coordinates are radial (divide diameter by 2).

The program example below illustrates the use of "polar" milling to machine a hexagon that measures 3/8" across the flats followed by machining "scallops" using indexing to position the part. The sketch below illustrates C/X-Axis relationship used to machine the hexagon. The C.N.C. part program on the next page demonstrates polar milling.



```
%
O1234
(MILL 3/8" HEX)
(NSK PNEUMATIC LIVE SPINDLE)
(NIAGARA CUTTER EDP#85671)
G20
G28W0
G0G40T0606
M5 (Spindle rotation stop)
M21 (3-Axis mode)
M56 (Live-tool #3 on)
X-1.1
Z.1
C0 (Orient spindle to C-zero degrees)
G12.1 (Polar milling mode on)
G1G98G41X-.7F10. (Tool Nose Radius Compensation, Left)
Z-.15
X-.385F3.
C.1111F4.5
X0C.2222
X.385C.1111
C-.1111
X0C-.2222
X-.385C-.1111
C0F5.0
X-.375
C.1083
X0C.2165
X.375C.1083
C-.1083
X0C-.2165
X-.375C-.1083
C.13
Z.1F40.
G40X-1.1 (Tool Nose Radius Compensation Cancel)
G13.1 (Polar Coordinate Mode Cancel)
G0Z.5
M57 (Live-tool #3 off)
T0
G28W0M20 (2-Axis mode)
M30
%
```

G07.1 or G107, Cylindrical Machining.

This function is designed for milling complex shapes, such as J-slots or text on a cylinder. This mode is typically used when the centerline of the live tool is parallel to the X-axis.

This function has requirements that must be met:

1. The M05, spindle rotation stop command must be in effect before entering 3-Axis mode.
2. The *modal* command M21; 3-Axis mode, must be in effect before C-Axis commands will be accepted.
3. M50, M52 or M56; "Live tool rotation on" must be in effect before cutting.
4. G98, feed in inches/per minute mode, must be used. Commands issued using the G99, feed in inches per revolution mode, will cause zero motion.
5. A C-Axis command [usually C0 (zero)] must be executed before G07.1 will be accepted.
6. "Tool Radius Compensation" is optional. If you desire to use radius compensation, the radius of the end-mill must be entered into the Geometry Offset page, "R" column for the tool used to mill. The "T" value in the Geometry Offset page is set to 0 (zero).
7. Tool Radius Compensation must be cancelled while G07.1 is still active.
8. The modal command M20; "2-Axis mode", must be restored before an M03 (Spindle Rotation, Forward) or M04 (Spindle Rotation, Reverse) command can be executed.
9. X-Axis coordinates are in diameters and C-Axis ("Virtual" Y-Axis) coordinates are radial (divide diameter by 2).

Refer to the Fanuc Operators' Manual for details and a program example of this function.

Macro

What is it?

As it relates to computers, a “macro” is a command that stands for a sequence of operations.

In C.N.C. usage, a macro is an instruction or series of instructions that repeats a sequence of operations without having to key in each move. This description sounds like a canned cycle because canned cycles use macro technology to function. A macro allows the use of algebraic functions. This ability gives the user of a macro the greatest flexibility of the many types of commands available. The canned cycles are a type of macro in that they take a **variable** value and use it to complete a function. Example: when you change the “Q” word in the G76 threading cycle, the process is altered: More passes are taken when “Q” is decreased. The control uses algebra to calculate how many moves to create based on the present value of several variables.

A variable, as it relates to c.n.c. programming is a symbol in a command or cycle that can be changed by changing value stored in another location. All c.n.c. controls come with locations in memory reserved to store these values. The “#” symbol is used by the control to identify these stored values.

A macro variable can store any numeric value that is to be repeated in a program. Variables are also used to count the number of repetitions of a sequence and then notifying the controller when it is time to move on to the next process.

The values can be negative numbers and fractions stated as decimal equivalents.

Macros use conditional statements with algebraic statements to perform functions. One example is the “counting” function used to control repetitions.

Another use of a macro command allows the programmer to execute commands that are not displayed in sequence on the screen.

Example: The block “If [#1=1]GOTO100;” tells the control that if variable #1=1, skip to the block labeled N100 and continue executing commands from that position in the program. If the variable contains any value other than 1, the control will move on to the next sequence in the program and begin processing commands. By placing a sequence desired to commence between the block containing the “If” statement and the block labeled N100 it is possible to define two different sets of geometry that can be selected by controlling the value of variable #1. This is a powerful feature when programming a “family” of parts with common features.

Program Sample

This section will illustrate a complete sample program. The program uses many of the cycles discussed in previous sections of this manual as well as commands not covered. Refer to the Prodigy Operators' Manual and the Fanuc Operators' Manual for details.

Features used:

Nxx: "N" is used to label a block for searching in edit mode or for use by a canned cycle.

/: "/" Is block skip. With this option set to on via "soft" keys, the program will skip past this command without executing it. In this application, block skip is used to reduce indexing times by inhibiting the G28W0 (Return to Reference Point) command.

G97Sxxxx: G97 (Constant Surface Speed Cancel) causes the spindle to rotate at xxxx revolutions per minute. An M03 (Spindle Rotation Forward), or a M04 (Spindle Rotation Reverse) command is required to engage spindle rotation.

G96Sxxx: G96 (Constant Surface Speed Control) causes the spindle to rotate at xxxx surface feet per minute. This mode causes the spindle to vary speed based upon the X-axis (diameter coordinate) the cutting tool is presently located at.

G94: "Canned" facing cycle.

G70 and G71: "Canned" turning and finishing cycles.

G40 and G41: G40 (Compensation Cancel) and G42 (Compensation, Left) are tool radius compensation commands.

G76: "Canned" threading cycle.

Part made from ¾" diameter brass.

```
%
O1234(DEMO PART)
G50S5500
M20G20
N1(ROUGH AND FINISH TURN)
(ISCAR BAR, NF3-A12S-SDUCL3)
(ISCAR INSERT, DCGT-3-1-AS)
G28W0
G0G40T0101
G97G99S1000M3
X-.78
Z.05M8
G96S600
G94X.05Z.003F.005
G71U.06R.005
G71P101Q102U-.01W.003F.007
N101G0X.05
G1Z0F.003
X-.198
X-.248W-.025
Z-.5F.005
X-.33
G2X-.4W-.035R.035F.0015
G1Z-.75F.003
X-.56,C.025F.0015
Z-1.0F.003
X-.73,R.035F.0015
Z-1.3F.003
N102X-.78
G0Z.15
G1G41Z.05F.01
G70P101Q102
G0G40Z.3
G97S1000
T0
/G28W0
M1
```

(Continued next column)

```
N2(CENTER DRILL)
(MAGAFOR #1, 195-0952)
/G28W0
G0G80T0202
G97G99S2000M3
X0
Z.05M8
G83Z-.12Q90F.003
G80
G0Z1.5
T0
/G28W0
M1

N3(1/4-28 EXTERNAL THREADS)
(ALL-5 BAR, DC-35-625)
(ALL-5 INSERT, TTR-35-093)
/G28W0
G0T0303
G97G99S1000M3
X.4
Z.1
G76P010260Q5R5
G76X.2064Z-.25P230Q70F.0357
G0Z.2M9
T0M5
G28W0
M30
%
```