SL Series
CNC MACHINE

HAAS
Haas Automation Inc.

PROGRAMMING WORKBOOK

HAAS AUTOMATION, INC.
2800 Sturgis Rd.
Oxnard, CA 93030

JANUARY 2006

.

HAAS AUTOMATION INC.

2800 Sturgis Road
Oxnard, California 93030
Phone: 805-278-1800
www.HaasCNC.com

The information in this workbook is reviewed regularly and any necessary changes will be incorporated in the next revision. This material is subject to change without notice.

Warning: This workbook is for the exclusive use of Haas Customers, Distributors and Trainers and is protected by copyright law. The reproduction, transmission or use of this document or its contents for profit is not permitted.

All content is the property of Haas Automation, Inc., copyright 2006. This Workbook may not be copied,distributed or reproduced for profit, in full or in part, without written permission from Haas Automation, Inc. This training information is being supplied for free to all Haas customers and schools that are learning to use Haas equipment. Haas information should never be modified unless you have written permission by Haas Automation.

Enquiries to Haas Automation about training information or a letter of authorization to copy, contact ebowman@haascnc.com

CUSTOMER SATISFACTION PROCEDURE

Dear Haas customer,

Your complete satisfaction and goodwill are of the utmost importance to both Haas Automation, Inc., and the Haas distributor where you purchased your equipment. Normally, any concerns you may have about the sales transaction or the operation of your equipment will be rapidly resolved by your distributor.

However, if your concerns are not resolved to your complete satisfaction, and you have discussed your concerns with a member of the dealership's management, the General Manager or the dealership's owner directly, please do the following:

Contact Haas Automation's Customer Service Center by calling 800-331-6746 and ask for the Customer Service Department. So that we may resolve your concerns as quickly as possible, please have the following information available when you call:

- Your name, company name, address and phone number
- The machine model and serial number
- The dealership name, and the name of your latest contact at the dealership
- The nature of your concern

If you wish to write Haas Automation, please use this address:

Haas Automation, Inc.
2800 Sturgis Road
Oxnard, CA 93030
Att: Customer Satisfaction Manager
e-mail: Service@HaasCNC.com

Once you contact the Haas Automation Customer Service Center, we will make every effort to work directly with you and your distributor to quickly resolve your concerns. At Haas Automation, we know that a good Customer-Distributor-Manufacturer relationship will help ensure continued success for all concerned.

NOTE: Should you have a problem with your machine, please consult your operator's manual first. If this does not resolve the problem, call your authorized Haas distributor. As a final solution, call Haas directly at the number indicated below.

Haas Automation, Inc.
2800 Sturgis Road
Oxnard, California 93030-8933   USA
Phone: (805) 278-1800

CONTENTS

CONTENTS

CONTENTS

INTRODUCTION

This workbook provides basic programming principles necessary to begin programming the HAAS C.N.C. Lathe.

In a "CNC" (Computerized Numerical Control) machine, the tool is controlled by a computer and is programmed with a machine code system that enables it to be operated with minimal supervision and with a great deal of repeatability.

The same principles used in operating a manual machine are used in programming a CNC machine. The main difference is that instead of cranking handles to position a slide to a certain point, the dimension is stored in the memory of the machine control **once**. The control will then move the machine to these positions each time the program is run.

In order to operate and program a CNC controlled machine, a basic understanding of machining practices and a working knowledge of math is necessary. It is also important to become familiar with the control console and the placement of the keys, switches, displays, etc., that are pertinent to the operation of the machine.

This Haas lathe workbook can be used for both operator's and programmer's. It is meant as a supplementary teaching aid for the user of HAAS Lathes. The information in this workbook may apply in whole or in part to the programming of other CNC machines. It is intended to give a **basic** understanding of CNC programming and it's applications. It is not intended as an in-depth study of all ranges of the machine use, but as an overview of common and potential situations facing CNC programmers. Much more training and information is necessary before attempting to program on the machine. For a complete explanation and an in-depth description, refer to the Programming and Operation Manual that is supplied with your HAAS lathe.

HAAS LATHE MACHINE SPECIFICATIONS

	MINI	SL-10	SL-20	TL-15*	SL-30	TL-25*	SL-40
CAPACITY							
Chuck Size	5C Collet	6.5"	8.3"	8.3"	10.0"	10.0"	15.0"
Standard Bar Capacity	1.06"	1.75"	2.0"	2.0"	3.0"	3.0"	4.0"
Between Centers	N/A	16.5"	24.0"	12.7"	39.0"	39.0"	51.0"
Max. Cutting Dia.		11.0"	10.3"	8.2"	17.0"	16.0"	25.5"
Max. Cutting Length		14.0"	20.0"	17.5"	34.0"	34.0"	44.0"
SPINDLE							
Peak Horsepower	7.5Hp	15Hp	20Hp	20Hp	30Hp	30Hp	40Hp
Max. RPM	6,000 rpm	6,000 rpm	4,000 rpm	4,000 rpm	3,400 rpm	3,400 rpm	2,400 rpm
Spindle Nose	A2-5	A2-5	A2-6	A2-6	A2-6	A2-6	A2-8
Spindle Bore Dia.	1.38"	2.31"	3.00"	3.00"	3.50"	3.50"	4.62"
Draw Tube Bore Dia.	1.18"	1.81"	2.06"	2.06"	3.03"	3.03"	4.06"
SWING DIAMETER							
Over front apron		16.25"	23.0"	20.0"	30.0"	30.0"	40.0"
Over cross slide	5.8"	8.0"	9.5"	9.5"	14.5"	14.5"	25.0"
TRAVELS							
X-Axis	12.0"	6.25"	8.45"	7.5"	11.3"	11.3"	17.0"
Z-Axis	12.0"	14.0"	20.0"	17.5"	34.0"	23.0"	44.0"
FEED RATES							
Axis Motor's Max. Thrust	2000 lbs.	3400 lbs.	3400 lbs.	3400 lbs.	For SL-30, 25, 40 X=3400 lbs. Z=5400 lbs.		
Rapid - AC Brushless	600 ipm	710 ipm	710 ipm	710 ipm	710 ipm	710 ipm	710 ipm
TOOLS	Gang style	Turret	Turret	Turret	Turret	Turret	Turret
Number Of Tools	up to 10	12	10	12	12	12	10
MISCELLANEOUS							
Coolant (U.S. Gallons)	24	15	40	40	50	50	77
TAILSTOCK							
Taper	N/A	MT3	MT4	N/A	MT4	N/A	MT5
Travel	N/A	4.0"	20"	N/A	33.5"	N/A	44"
Thrust	N/A	225-900 lbs.	300-1500 lbs.	N/A	300-1500 lbs.	N/A	300-1500 lbs.

*Live Tooling Lathe

THE COORDINATE SYSTEM

The first diagram that were concerned with are called a NUMBER LINES. This number line has a reference zero point that is called ABSOLUTE ZERO and may be placed at any point along the number line.

The number line also has numbered increments on either side of absolute zero. Moving away from zero to the right are positive increments. Moving away from zero to the left are negative -" increments. The "+", or positive increments, are understood, therefore no sign is needed. The "-" sign is always needed if it's a negative value. We use positive and negative along with the increment's value to indicate its relationship to zero on the line. In the case of the previous line, if we choose to move to the third increment on the minus (-) side of zero, we would call for -3. If we choose the second increment in the plus range, we would call for 2. Our concern is with distance and direction from zero.

Remember that zero may be placed at any point along the line, and that once placed, one side of zero has negative increments and the other side has positive increments.

The next illustration shows the two directions of travel on a lathe. To carry the number line idea a little further, imagine such a line placed along each axis of the machine.
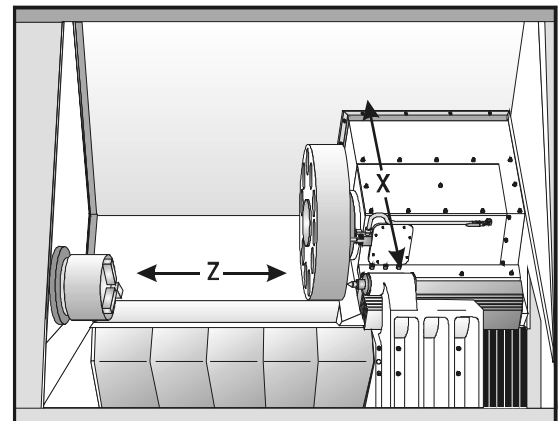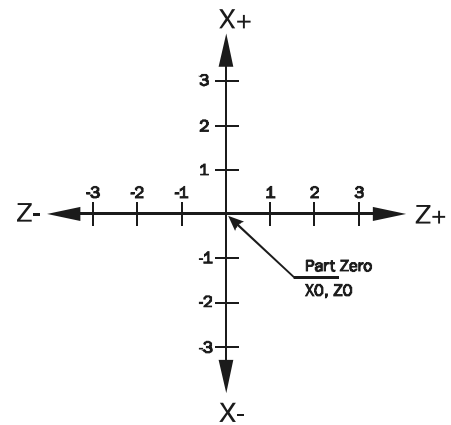
Haas Lathe X and Z axis lines

The first number line is easy to conceive as belonging to the left-to-right, or "Z", axis of the machine. If we place a similar number line along   the front-to-back, or "X" axis, the increments toward the operator are the negative increments, and the increments away from the operator are the positive increments.

The increments on a number line on the HAAS lathe equals .0001 inches. While a line theoretically has infinite length in either direction, the two lines placed along the X and Z axes of the machine do not have unlimited accessibility. That is to say, we are limited by the range of travel on the machine. For the HAAS SL-20 for example, we have access to 8.45 inches in the X axis and 20 inches in the Z axis.

Remember, when we are moving the machine, we are concerned with positioning the turret around our workpiece. And our coordinates for positioning the turret are based off a floating zero point or our part origin.

Note: The Haas lathe use X dimensions based on the part diameter, not the radius. Thus an X move from 0. to 1.0 (X1.0) will only move the tool up .5 on the X axis.

The zero position may be placed at any point along each of the two number lines, and in fact will probably be different for each setup of the machine. It is noteworthy to mention here that the X-axis is usually set with the machine zero position on the center line of the spindle, while the Z axis zero is usually set at the finished right end surface of the part being machined. This places all X axis cutting in a positive range of travel, whereas the Z axis cutting would be in the negative range of travel.
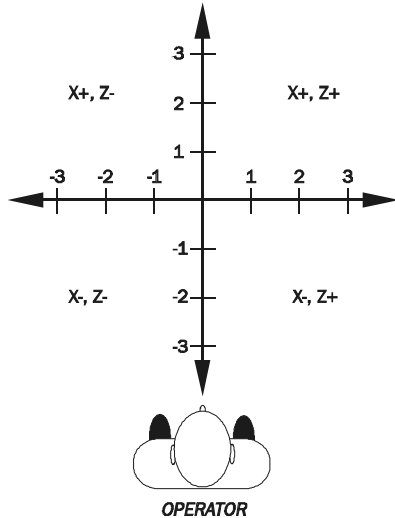
The diagram at left shows a front view of the grid as it would appear on the lathe. This view shows the X and Z axes as the operator faces the lathe. Note that at the intersection of the two lines, a common zero point is established. The four areas to the sides and above and below the lines are called "QUADRANTS" and make up the basis for what is known as rectangular coordinate programming.

QUADRANT 1 is on the Top Right at X+, Z+
QUADRANT 2 is on the Top Left at X+, Z-
QUADRANT 3 is on the Bottom Left at X-, Z-
QUADRANT 4 is on the Bottom Right at X-, Z+

Whenever we set a zero somewhere on the X axis and somewhere on the Z axis, we have automatically caused an intersection of the two lines. This intersection where the two zeros come together will automatically have the four quadrants to its sides, above, and below it. How much of each quadrant that is accessible is determined by where we place the zeros on the travel axes of the lathe.

For example, if we set zero exactly in the middle of the Z axis and if we set the X axis zero on the spindle center line, we have created four quadrants. For an SL-20 for example, the upper two quadrants of the Z travel is 10 inches and the X travel is 7.45 inches. The lower two quadrants will have Z travel of 10 inches and X travel of 1 inch. The HAAS lathes have 1 inch of negative travel beyond the center line of the spindle.

MACHINE HOME

The principal of machine home may be seen when doing a reference return of all machine axes at machine startup. A zero return (POWER UP/RESTART) is performed when you power on machine. All axes will then move to the furthest positive locations, to the upper right of machine, until the limit switches are reached. When this condition is satisfied, the only way to move any of the axes is in the negative direction. This is because the machine zero, is set to the furthest positive point to the upper right of the machine, when the machine was sent Home with a POWER UP/RESTART.

Machine Home is placed at the edge of each axes travel. In effect, now the positive quadrants cannot be reached, and all the X and Z moves will be found to be in the X-, Z- quadrant. It is only by setting a new location with, Tool Geometry and Work Zero Offsets somewhere within the travel of each axis that other quadrants are able to be reached.
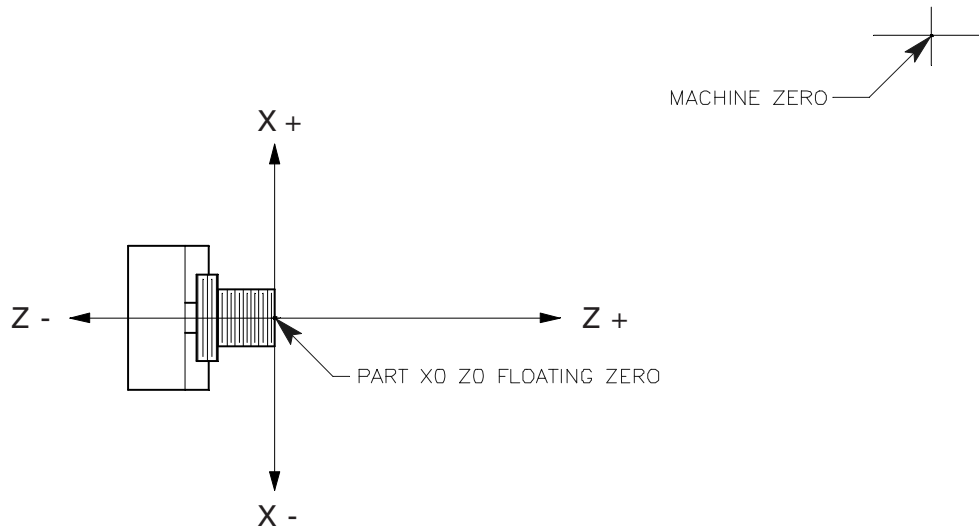
It "would not" be convenient to program our parts from the machine zero, so secondary floating zero point is established with offsets. This floating zero is referred to by either, PART ZERO or PART ORIGIN, both having the same meaning.

To create the new part zero location, each tool is manually touched off of the part being setup, on the diameter and the length. Then through a series of control key strokes, that distance from machine zero to the part zero is stored for X & Z axes, in tool offsets, and activated later, from the part program, when that tool is used for cutting a part.

Centerline of the lathe spindle will always be "X" zero and the "Z" zero location will "float" to the face on the part that reflects most of the part length dimensions. Normally the front face is used, because it's usually easier to access for touch-off procedure and also easier to program.

ABSOLUTE AND INCREMENTAL POSITIONING

Up to this point, we have defined a system of positioning the tool that is known as absolute programming. In absolute, all coordinate points are given, in X and Z axis, with regard to their relationship to a fixed part zero or an origin point. This is the most common type of positioning.

Another type of positioning is called incremental. Incremental positioning is defined using U and W. The "U" character is used to specify incremental motion in the X-axis, and the "W" character is used to specify incremental motion in the Z axis. Both "U" and "W" define distance and direction from a point to point location. An incremental coordinate position is entered, using U and W, in terms of its relationship to the previous position, and not from a part zero or part origin point. In other words, after a block of information has been executed, the position that the tool is now at is the new zero point for the next move to be made. The "U" address is used to specify an incremental move along the "X" axis and the "W" address is used to specify an incremental move along the "Z" axis.

An example of the use of the incremental system is shown below. Note that to move from Z-3.375 to Z-.625 on the scale, a positive incremental move of W2.750 was made, even though the move W still places the tool on the minus side of the Z scale. Therefore the move was determined from the start point position, with no regard for the fixed zero reference point. The **+** and **-** signs are used in terms of direction from the starting point, and are not defined in regard to the part zero point.
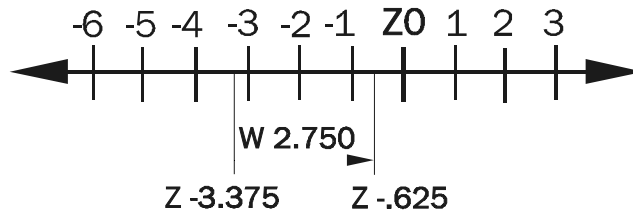
An example of an incremental move.

Keep in mind that when positioning in **absolute**, we are concerned with distance and direction from a fixed part zero reference point, and when positioning in **incremental** we are concerned with distance and direction from the last point.

Absolute mode should be your positioning mode of choice for most applications. There are times when incremental mode can be quite helpful. Repeating motions within a subroutine, for example, is one excellent example. If you have six identical grooves to turn on a Haas lathe, you can save programming effort if you specify the motions incrementally to machine one groove. Then just call up the subroutine again to repeat the commands to do another groove at a new location.

There are even times when it is helpful to command one axis move in the absolute mode while another moves in incremental mode. Any turning center using U and W to specify incremental motions in X and Z easily allows this. Say you're experiencing some unwanted taper on a diameter and you want to program a tapering movement to counteract the problem. In the command that turns the diameter that is experiencing unwanted taper, you can specify the Z endpoint in absolute mode from program zero and the X endpoint as an incremental move. Here is an example:

N040 G01 U0.002 Z-2.5 F0.005

In this command, the tool will move 0.002 inch, on the diameter, in the X positive direction while the Z axis moves to an endpoint of minus 2.5 inches relative to program zero.

Note: The Haas lathe used absolute X dimensions based on the part diameter, not the radius. Thus an X move from 0. to 1.0 (X1.0) will only move the tool up .5 on the X axis. Like X axis absolute moves, incremental U movements are specified for the part diameter, not the radius. Thus a U move from 0. to 1.0 (U1.0) will only move the tool up .5 incrementally on the X axis.

TYPICAL LATHE PART

We can now take this part and show it in a TYPICAL LATHE SETUP, This would include showing the part, soft jaws, and chuck. Starting with this drawing, you will see this symbol

⊕ shown on the parts, this target is to indicate where the FLOATING ZERO has been established by the programmer.

CHUCK
JAWS

PART

4.000

1.000

R .100

.25 x 45 Deg.

2.000

1.500
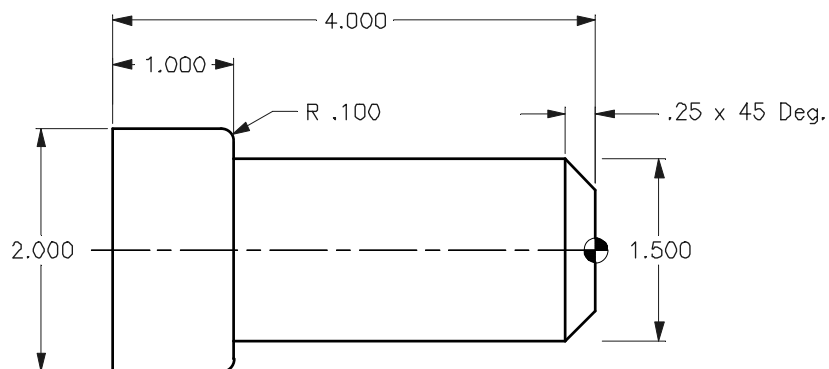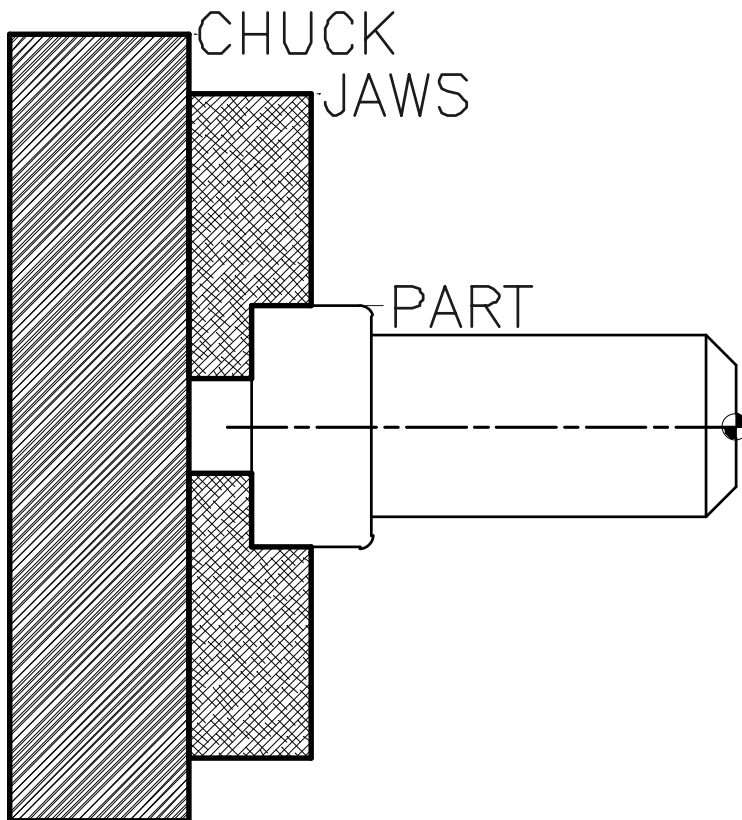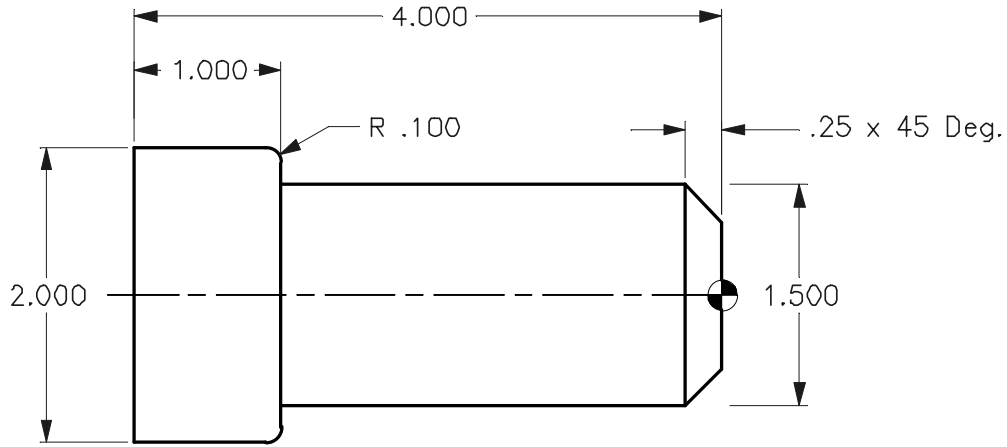
We can now take our TYPICAL LATHE PART and record the geometry points needed to program this part. Diameters are defined as the actual diameter value and not the radius from center.
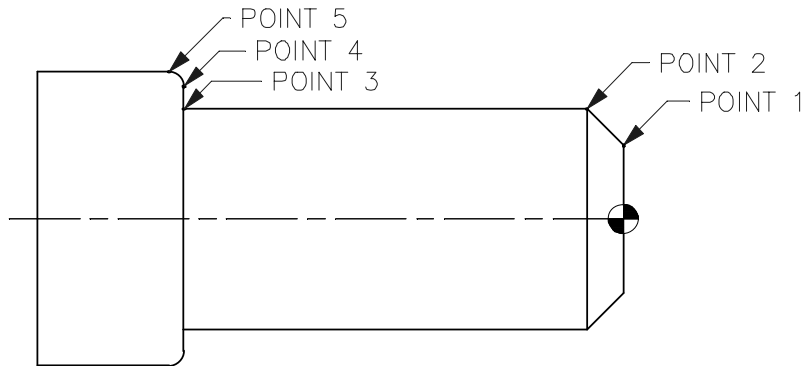
Values for the Points are :

$$X \qquad Z$$

(Diameter not Radius)

Point 1   _____   _____

Point 2   _____   _____

Point 3   _____   _____

Point 4   _____   _____

Point 5   _____   _____

Calculate and record the X-axis diameters and Z-axis lengths for the part shown below.

The values between (parenthesis) would be points where the tool is already located, and those points in a program, would not need to be defined again.
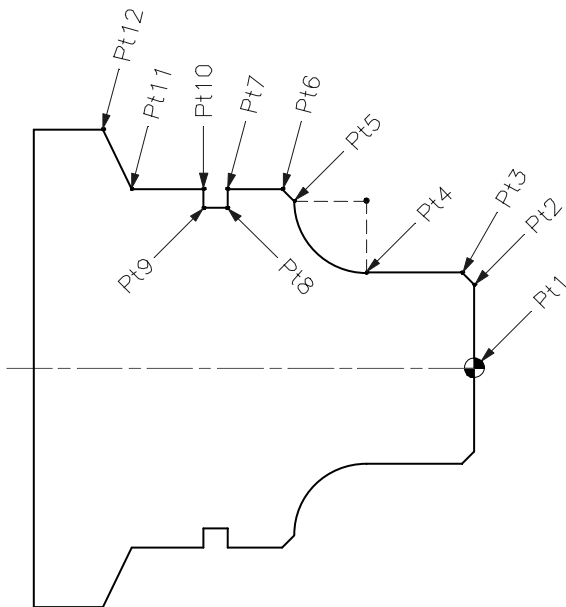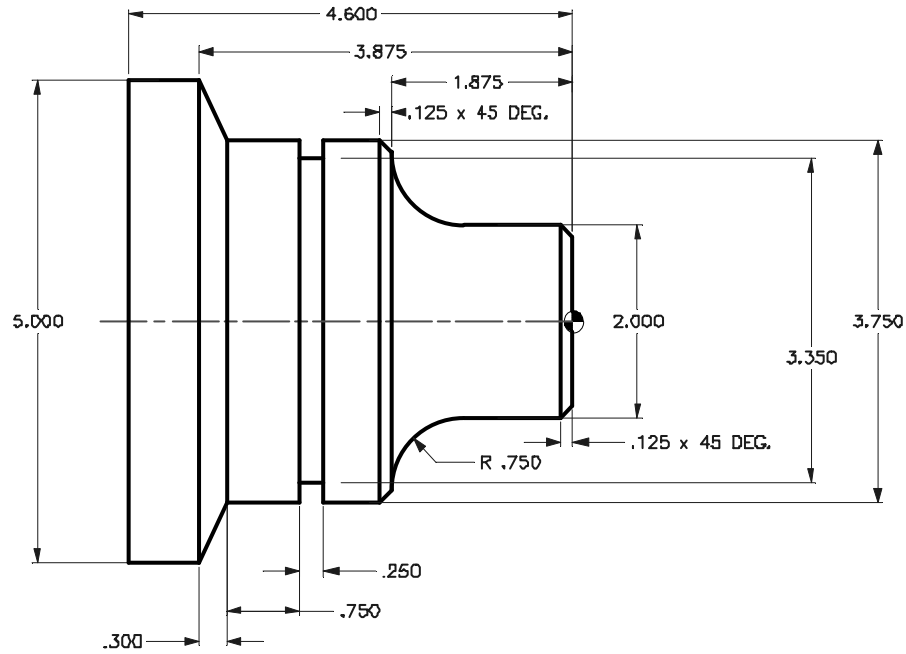
	X	Z
Pt 1	_____	_____
Pt 2	_____	(_____)
Pt 3	_____	_____
Pt 4	(_____)	_____
Pt 5	_____	_____
Pt 6	_____	_____
Pt 7	(_____)	_____
Pt 8	_____	(_____)
Pt 9	(_____)	_____
Pt 10	_____	(_____)
Pt 11	(_____)	_____
Pt 12	_____	_____

PROGRAMMING WITH CODES

The definition of a part program for any CNC consists of movements of the tool and speed changes to the spindle RPM. It also contains auxiliary command functions such as tool changes, coolant on or off commands, or external M code commands.

Tool movements consist of rapid positioning commands, straight line moves or movement along an arc of the tool at a controlled speed.

The Haas lathe has two (2) linear axes defined as **X** axis and **Z** axis. The **X** axis moves the tool turret toward and away from the spindle center line, while the **Z** axis moves the tool turret along the spindle axis. The machine zero position is where the tool is at the upper right corner of the work cell farthest away from the spindle axis. Motion in the **X** axis will move the turret toward the spindle centerline with negative values and away from spindle center with positive values. Motion in the **Z** axis will move the tool toward the spindle chuck with negative values and away from the chuck with positive values.

A program is written as a set of instructions given in the order they are to be performed. The instructions, if given in English, might look like this:

> **LINE #1 =** SELECT CUTTING TOOL.
> **LINE #2 =** TURN SPINDLE ON AND SELECT THE RPM.
> **LINE #3 =** RAPID TO THE STARTING POSITION OF THE PART.
> **LINE #4 =** TURN COOLANT ON.
> **LINE #5 =** CHOOSE PROPER FEED RATE AND MAKE THE CUT(S).
> **LINE #6 =** TURN THE SPINDLE AND COOLANT OFF.
> **LINE #7 =** RETURN TO CLEARANCE POSITION TO SELECT ANOTHER TOOL.

and so on. But our machine control understands only these messages when given in machine code, also referred to as G and M code programming. Before considering the meaning and the use of codes, it is helpful to lay down a few guidelines.

PROGRAM FORMAT

There is no positional requirement for the address codes. They may be placed in any order within the block. Each individual can format their programs many different ways. But, Program format or program style is an important part of CNC machining. There are some program command formats that can be moved around, and some commands need to be a certain way, and there are some standard program rules that are just good to follow. The point is that a programmer needs to have an organized program format that's consistent and efficient so that any CNC machinist in your shop can understand it.

Some standard program rules to consider are:

Program X and Z in alphabetical order on any block. The machine will read Z or X in any order, but we want to be consistent. If both X and Z are on a command line in a program, they should be listed together and in order. Write X first, and Z second.

You can put G and M codes anywhere on a line of code. But, in the beginning when N/C programming was being developed G codes had to be in the beginning of a program line and M codes had to be at the end. And this rule, a lot of people still follow and is a good standard to continue.

Some CNC machines allow you to write more the one M code per line of code and some won't. On the HAAS, only one M code may be programmed per block and all M codes are activated or cause an action to occur after everything else on the line has been executed.

Program format is a series and sequence of commands that a machine may accept and execute. Program format is the order in which the machine code is listed in a program that consist of command words. Command words begin with a single letter and then numbers for each word. If it has a plus (+) value, no sign is needed. If it has a minus value, it must be entered with a minus (-) sign. If a command word is only a number and not a value, then no sign or decimal point is entered with that command. Program format defines the "language of the machine tool."   ...

```
    ...
    G82 Z-0.2 P0.3 R0.1 F0.003
    G80 G00 Z1. M09
    G28
    M01

    N4  (Drill .312 Dia. x 1.5 Depth)
    G28
    T404  (5/16 DIA. DRILL)
    G97 S2400 M03
    G54 G00 X0. Z1. M08
    G83 Z-1.5 Q0.3 R0.1 F0.006
    G80 G00 Z1. M09
    G28
    M01
    …
    …
```

AN EXAMPLE OF THE PROGRAM'S FIRST COUPLE OF LINES:

THE FIRST LINE or block of a program, should be a return to machine zero (using G28 or G51 codes). Any tool change should be after a return to machine zero or a tool change location. Although this is not necessary it is a good safety measure.

THE SECOND LINE of code should apply to any appropriate tool selections and tool geometry offsets or tool shifts.

THE THIRD LINE may optionally contain a spindle speed maximum for the tool being used.

THE FOURTH LINE or block should cancel any constant surface speed mode (G97). And it should specify a constant spindle speed command (S____) along with a spindle ON clockwise command (M03).

THE FIFTH LINE should contain a work offset (G54), a preparatory code (G00) for rapid command with an X and Z location for positioning the turret, and turn on the coolant (M08).

THE SIXTH LINE may choose to, optionally specify a Constant Surface Speed with (G96) and a surface feet per minute (SFM) defined with a (S____) command .

An example of the program's startup lines might look like this:

With Constant
Surface Speed

N11 G28 ;
N12 T101 (O.D. TOOL x .031 TNR);
N13 G50 S2800 ;
N14 G97 S650 M03 ;
N15 G54 G00 X1.85 Z1. M08;
N16 G96 S315 ;

Without Constant
Surface Speed

N21 G28 ;
N22 T101 (.750 DIA. DRILL);
N23 G97 S1600 M03 ;
N24 G54 G00 X0. Z1. M08;

 All the necessary codes for each operation are listed in the following pages. This tool startup format is a good example and defines a commonly used program style.

DEFINITIONS WITHIN THE FORMAT

1. CHARACTER : A single alphanumeric character value or the "+" and "-" sign.

2. WORD : A series of characters defining a single function such as, *G codes, M codes,* an "X" *axis moves,* or "F" *feedrate.* A letter is the first character of a word for each of the different commands. There may be a distance and direction defined for a word in a program. The distance and direction in a word is made up of a value, with a plus (+) or minus (-) sign. A plus (+) value is recognized if no sign is given in a word.

3. BLOCK : Series of words defining a single instruction. An instruction may consist of a single linear motion, a circular motion or canned cycle, plus additional information such as a feedrate or miscellaneous command (M-codes).

4. POSITIVE SIGNS : If the value following an address letter command such as A, B, C, I, J, K, R, U, V, W, X, Y, Z, is positive, the plus sign need not be programmed in.
If it has a minus value it must be programmed in with a minus (-) sign.

5. LEADING ZERO'S : If the digits proceeding a number are zero, they need not be programmed in. The HAAS control will automatically enter in the leading zero's.
        EXAMPLE: G0 for G00 and M1 for M01,
Trailing zeros must be programmed: M30 not M3, G70 not G7.

6. MODAL COMMANDS : Codes that are active for more than the line in which they are issued are called MODAL commands. Rapid traverse, feedrate moves, and canned cycles are all examples of modal commands. A NON-MODAL command which once called, are effective only in the calling block, and then immediately forgotten by the control.

7. PREPARATORY FUNCTIONS : "G" codes use the information contained on the line to make the machine tool do specific operations, such as :
    1.) Move the tool at rapid traverse.
    2.) Move the tool at a feedrate along a straight line.
    3.) Move the tool along an arc at a feedrate in a clockwise direction.
    4.) Move the tool along an arc at a feedrate in a counterclockwise direction.
    5.) Move the tool thru a series of repetitive operations controlled by "fixed
        cycles" such as, spot drilling, drilling, boring, and tapping.

8. MISCELLANEOUS FUNCTIONS : "M" codes are effective or cause an action to occur at the end of the block, and only one M code is allowed in each block of a program.

9. SEQUENCE NUMBERS : N1 thru N99999 in a program are only used to locate and identify a line or block and its relative position within a CNC program. A program can be with or without SEQUENCE NUMBERS. The only function of SEQUENCE NUMBERS is to locate a certain block or line within a CNC program.

The machining cycles G70, G71, G72 and G73 require the use of sequence numbers to call up specified blocks in a program that in it define the part geometry to rough and finish.

SAFE STARTUP LINE???

G18 G20 G40 G54 G80 G97 G99

Do you need a safe startup line to be sure all the commands are canceled before starting a program? Many programs have a G code default line (or the CAD/CAM programming system may output a default line) at the beginning. To be sure machine control is in a safe start condition. Here are the conditions on a Haas control, to help you decide if you need a safe start line at the beginning of a program.

G18 Circular Motion ZX Plane Selection - The G18 is the default condition on a Haas lathe, and is the only one available in the X and Z axis to do an arc. If you try an arc in the G17 XY plane, or the G19 YZ plane, the machine would stop and give you an alarm. There is no need to program a G18, it is already active when you power up.

G20 Inch / G21 Metric Diiensioning - The G codes G20 and G21 are used to select between inch and metric. On the HAAS control, the G20 (inch) and G21 (mm) codes are used to make sure the inch/metric setting (Setting 9) is set correctly for that program.

If you're not switching between inch and metric very much, then why keep putting in these G-codes at the start of every program?
If you did have the wrong condition in Setting 9, graphics display will display whatever Setting 9 is set at. You'll easily see the error when running a program in graphics.

G54 Work Offset Command - Work offsets on a CNC lathe, are not used like they are on a CNC mill. Many lathe users don't even have values in their work offsets, because all the offsets that are needed on a lathe, for most users, are entered in the Tool Geometery display. The only time someone might use a work offset on a lathe, is to shift all the tools in Z axis the same distance.

Example: Lets say you want to move all your tools -.015 in the Z axis, to make the parts that much shorter. Instead of changing all the tool offsets. You could shift all the tools in -0.015 by changing the work offset. **Just be sure to take this offset shift out when you're done using it.**

WORK ZERO OFFSET

G CODE	(X)	(Z)	(B)
G 52	0.	0.	0.
G 54	0.	-0.015	0.
G 55	0.	0.	0.
...			

Another main reason to use a work offset on a lathe, is when you're touching off your tools on a tool probe. The X axis value is usually always good to the center of the spindle using the value touched off in the X axis on a tool probe along with the value in Setting 59 or 60. The problem is the distance from the edge of the tool probe in the Z axis to the face of the part. So compensate for that difference in the Work Zero Offset, that has in it, the extra distance from the edge of the tool probe to the face of the part, in the Z axis.

Be aware that even if you're not using a work offset command in a program, G54 is still active as a default G code. This is why you see G54 is in the program examples of this book, as a reminder that this work offset is active wether you program it in or not. If you're never using work offsets, you may choose to leave this G54 command out.

G40 Cancels Tool Nose Compensation - You may see a G40 in the beginning of every program. And some even put it in at the beginning of every tool, to be sure cutter compensation is off, before they start a program.

You should always cancel cutter compensation (G40) when you're done using it. If you forget to cancel cutter compensation on a lathe, and you run a program in graphics, you'll get a 349PROG STOP W/O CANCEL CUTTER COMP alarm. To let you know, that you ended a program without canceling cutter compensation. Pressing RESET or POWER OFF will also cancel cutter compensation. Because of these conditions that cancel cutter compensation, you don't need to put a G40 at the beginning of every program.

G80 Cancels Canned Cycles - If you forget to cancel a canned cycle with a G80, RESET, G00, G01, M30 and Power Down will also cancel any active canned cycle.

G97 Constant Non-Varying Spindle Speed - For safety reasons you should program in a G97 at the beginning and end of every tool cycle, with a spindle speed.

G99 Feed Per Revolution - This command changes how the "F" address is interpreted. The F command indicates inches per spindle revolution when Setting 9 is set to INCH. If Setting 9 is set to METRIC, F indicates millimeters per revolution. G99 and G98 are modal commands. G99 is the default command, and the one you'll usually want. In certain situations you may choose G98 for Feed Per Minute, and then switch back to G99. But most customers, will already be in G99 and never switch to G98, so there's usually no need to have it in a safe start line. If a mistake is made, using the wrong feed command, you'll easily see it when it happens, when running the program on the machine.
Example: If you program a feed of F0.005 to feed per revolution, and you accidentally program in a G98 (Feed Per Minute), then it would take forever to feed .005 a minute. If you programmed F10. for feed per minute (G98), and you were accidentally in Feed Per Revolution (G99), and you had the spindle on, lets say at a low speed of S200. The calculated feedrate for this would be 2000ipm, which is above the machine maximum feed rate. The machine would stop and give you an alarm.

There might be good reasons why you want a safe start line at the beginning of every program, and some programers like having it at the beginning of every tool, as a safety precaution. You may have other machines running these same programs that may need safe start up lines. But for many, it may not be necessay. Because of these reasons listed above. It's up to the programer to decide if they want safe start up lines in their programs.

OFTEN USED PREPARATORY "G" CODES

G00   Rapid traverse motion; Used for non-cutting moves of the machine in positioning quick to a location to be machined, or rapid away after program cuts have been performed. Maximum rapid motion (I.P.M.) of a Haas machine will vary on machine model.

G01   Linear interpolation motion; Used for actual machining and metal removal. Governed by a programmed feed rate in inches (or mm) per revolution (G99). Maximum feed rate (I.P.M.) of a Haas machine will vary on machine model. (*Inch Per Minute = R.P.M. x Inch Per Revolution*).

G02   Circular Interpolation - *Clockwise.*

G03   Circular Interpolation - *Counterclockwise.*

G28   Machine Home (*Rapid traverse*)

G40   Tool Nose Compensation CANCEL

G41   Tool Nose Compensation LEFT of the programmed path

G42   Tool Nose Compensation RIGHT of the programmed path

G50   Spindle Speed Maximum RPM limit

G70   Finishing *Cycle*

G71   O.D./I.D. Stock Removal *Cycle*

G72   End Face Stock Removal *Cycle*

G76   O.D./I.D. Thread Cutting *Cycle*

G80   Cancel *Canned Cycle*

G81   Drill *Canned Cycle*

G82   Spot Drill *Canned Cycle*

G83   Peck Drill *Canned Cycle*

G84   Tapping *Canned Cycle*

G96   Constant Surface Speed On

G97   Constant Surface Speed Cancel

G98   Feed Per Minute

G99   Feed Per Revolution

OFTEN USED MISCELLANEOUS "M" CODES

M00   The M00 code is used for a Program Stop command on the machine.
It stops the spindle, turns off coolant and stops lookahead processing.
Pressing CYCLE START again will continue the program on the next
block of the program.

M01   The M01 code is used for an Optional Program Stop command.
Pressing the OPT STOP key on the control panel signasl the machine
to perform a stop command when the control reads an M01 command. It will
then perform like an M00.

M03   Starts the Spindle FORWARD. Must have a spindle speed defined.

M04   Starts the Spindle REVERSE. Must have a spindle speed defined.

M05   STOPS the Spindle

M08   Coolant ON command

M09   Coolant OFF command

M30   Program End and Reset to the beginning of program.

M97   Local Subroutine Call

M98   Subprogram Call

M99   Subprogram Return (M98) or Subroutine Return (M97), or a Program Loop.

*NOTE: Only one "**M**" code can be used per line. And the "**M**" codes will be the last
command executed in a line, regardless of where it's located in that line.*

PROGRAM STRUCTURE

A CNC part program consists of one or more blocks of commands. When viewing the program, a block is the same as a line of text. Blocks shown on the CRT are always terminated by the ";" symbol which is called an EOB (End Of Block). Blocks are made up of alphabetical address codes which are always an alphabetical character followed by a numeric value. For instance, the specification to move the **X**-axis would be a number proceeded by the **X** symbol.

Programs must begin and end with a percent (%) sign. After the first percent (%) sign with nothing else on that line, the next line in a program must have a program number beginning with the letter O (not zero) and then the number that defines that program. Those program numbers are used to identify and select a main program to be run, or as a subprogram called up by the main program. The % sign will "not" be seen on the control. But they must be in the program when you load a program into the control. And they will be seen when you download a program from the machine. The % signs are automatically entered in for you, if you enter a program in on the HAAS control.

A program may also contain a " **/** " symbol. The " **/** " symbol, sometimes called a slash, is used to define an optional block. If a block contains this symbol, any information that follows the slash in a program block, will be ignored when the **BLOCK DELETE** button is selected when running a program.

On the following page is a sample program as it would appear on the CRT. The words following the ":" are not part of the actual program but are put there as further explanation.

This program will rough and finish turn and face for a part with two diameters along with drilling and tapping for a 3/8-16 x 1.0 deep threaded hole one end.

%	:Program must begin and end with a %
O00018 ;	:Letter "O" and a five digit program number
(CNC LATHE PROGRAM EXAMPLE) ;	:Comment statement between parenthesis
N1 (Rough O.D.) ;	:First operation
G28 ;	:Return to machine zero for a tool change, cancel tool offset
T101 (O.D. TOOL x .031 TNR) ;	:Select tool 1 with offset 1
G50 S2600 ;	:Set spindle speed max. clamp 2600 RPM
G97 S414 M03 ;	:Cancel CSS, 415 spindle speed, on forward
G54 G00 X3.6 Z0.1 M08 ;	:Work offset, rapid X, Z axes, coolant on
G96 S390 ;	:CSS on at 390 SFM, coolant on
G00 Z0.005 ;	:Rapid to .005 from the end of part
G01 X-0.063 F0.005 ;	:Rough face end of part
G00 X3.6 Z0.1 ;	:Rapid to start point above part
G71 P10 Q20 U0.01 W0.005 D0.1 F0.01 ;	:Rough turning G71 canned cycle using the path defined :between N10 thru N20, leaving .010 stock on the X-axis :diameters, .005 stock on the linear faces, with 0.1 depth :of cut each pass, feeding .010 per revolution.
N10 G42 G00 X0.82 ;	:N10 is the starting block called by the P block in the G71
G01 Z0. F0.004 ;	:line that difines in it the geometery to rough out.

```
X0.9  ;                          :Lines that defines the part geometery to rough out
G03 X1. Z-0.05 R0.05  ;              "        "        "        "        "        "
/ G01 Z-1.75  ;                      "        "        "        "        "        "
/ X1.75  ;                           "        "        "        "        "        "
/ G03 X2.25  Z-2. R.25  ;            "        "        "        "        "        "
G01 Z-3.25 F0.004  ;                 "        "        "        "        "        "
X2.94  ;                             "        "        "        "        "        "
X3. Z-3.28  ;                        "        "        "        "        "        "
Z-4.1  ;                             "        "        "        "        "        "
N20 G40 X3.6  ;                  :End of geometry with Q20 in G71 line, cancel cutter comp.
G97 S414 M09  ;                  :Cancel CSS, coolant off
G28  ;                           :Return to machine zero for a tool change, cancel tool offset
M01  ;                           :Optional program stop
;
;
N2  (FINISH O.D.)  ;             :Second operation
G28  ;                           :Return to machine zero for a tool change, cancel tool offset
T202  (O.D. TOOL x .031 TNR)  ; :Select tool 2 with offset 2
G50 S2600  ;                     :Spindle speed max. of 2600 RPM
G97 S1354 M03  ;                 :Cancel CSS, 1350 spindle speed, on forward
G54 G00 X1.1 Z0.1 M08  ;         :Work offset, rapid X and Z axis, coolant on
G96 S390  ;                      :CSS on at 390 SFM
G00 Z0.  ;                       :Rapid to end of part
G01 X-0.032 F.003  ;             :Finish face end of part
G00 X3.6 Z0.1  ;                 :Rapid to start point above part
G70 P10 Q20  ;                   :Finishing cycle calling N10thru N20 to do a finish pass
G97 S414  ;                      :Cancel CSS, 475 spindle speed
G00 Z1.0 M09  ;                  :Rapid Z axis, coolant off
G28  ;                           :Return to machine zero for a tool change, cancel tool offset
M01  ;                           :Optional program stop
;
N3  (Drill .750 Dia. x 2.5 Depth)  ; :Third operation
G28  ;                           :Return to machine zero for a tool change, cancel tool offset
T303  (3/4 DIA. DRILL)  ;        :Select tool 3 with offset 3
G97 S1986 M03  ;                 :Cancel CSS, 1950 spindle speed, on forward
G54 G00 X0. Z1. M08  ;           :Work offset, rapid X and Z axis, coolant on
G83 Z-2.5 Q0.3 R0.1 F0.005  ;    :Deep hole peck drill 2.5 deep with a 0.3 peck
G80 G00 Z0.1 M09  ;              :Cancel canned cycle, rapid Z axis, coolant off
G28  ;                           :Return to machine zero for a tool change, cancel tool offset
T100  ;                          :Select tool 1 to get ready for the next part
M30  ;                           :Stop program, rewind to beginning
%                                :Program must end with a %
```

It is common to begin each tool in a part program with preparatory codes, turning on commands associated for that tool, and then ending by returning to machine home, or a safe location, to position for a tool change. There might be a number of commands that are repeated throughout the program. This is done for safety to insure that the proper commands are attained if the operator has to begin, at the start of a tool in the middle of a program, in the event of tool breakage, to rerun a tool, or finish a part after powering up the machine. This is a common programming practice.

MISCELLANEOUS FUNCTION "M" CODES

All M codes are activated or cause an action to occur after everything else on a block has been completed. Only one M code is allowed per block in a program.

If there is a (Setting number) listed next to an M code, that setting will in some way relate to that M code.

The following list is a summary of Haas M codes. A double asterisk (**) indicates options available.

M00 Program Stop (Setting 42)
M01 Optional Program Stop (Setting 17)
M02 Program End
M03 Spindle On Forward (S) (Setting 144)
M04 Spindle On Reverse (S) (Setting 144)
M05 Spindle Stop
M08 Coolant On (Setting 32)
M09 Coolant Off
M10 Chuck Clamp (Setting 92)
M11 Chuck Unclamp (Setting 92)
M12** Auto Air Jet On
M13** Auto Air Jet Off
M14** Main Spindle Clamp
M15** Main Spindle Unclamp
M17 Rotate Turret Forward (T) (Setting 97)
M18 Rotate Turret Reverse (T) (Setting 97)
M19** Orient Spindle (R, P)
M21** Tailstock Advance (Setting 93, 94, 106, 107, 121, 145)
M22** Tailstock Retract (Setting 105)
M23 Angle Out of Thread On (Setting 95, 96)
M24 Angle Out of Thread Off
M30 Program End and Reset (Setting 2, 39, 56, 83)
M31** Chip Auger Forward (Setting 114, 115)
M33 Chip Auger Stop
M36** Parts Catcher On
M37** Parts Catcher Off
M41 Spindle Low Gear Override
M42 Spindle High Gear Override
M43 Turret Unlock (For Service Use Only)
M44 Turret Lock (For Service Use Only)
M51-M58 Optional User M Code Set
M59 Output Relay Set
M61-M68 Optional User M Code Clear

M69   Output Relay Clear
M76   Program Displays Active
M77   Program Displays Inactive
M78   Alarm if skip signal found
M79   Alarm if skip signal not found
M85** Automatic Door Open (Setting 131, 51)
M86** Automatic Door Close (Setting 131, 51)
M88** High Pressure Coolant On (Setting 32)
M89** High Pressure Coolant Off
M93** Axis Position Capture Start
M94** Axis Position Capture Stop
M95   Sleep Mode (hh:mm)
M96   Jump If No Signal (P,Q)
M97   Local Sub-Routine Call (P,L)
M98   Sub-Program Call (P,L)
M99   Sub-Program/Routine Return Or Loop (Setting 118)
M110** Tailstock Chuck Clamp (Setting 122)
M111** Tailstock Chuck Unclamp (Setting 122)
M119** Sub-Spindle Orient (P,R)
M121-M128   Optional User M Code Interface with M-Fin Signal
M133** Live Tool Drive Forward (P)
M134** Live Tool Drive Reverse (P)
M135** Live Tool Drive Stop
M143** Sub-Spindle Forward (P)
M144** Sub-Spindle Reverse (P)
M145** Sub-Spindle Stop
M154** C-Axis Engage (Setting 102)
M155** C-Axis Disengage
      ** Options

PREPARATORY FUNCTIONS "G" CODES

1) G Codes come in groups. Each group of G codes will have a specific group number.

2) A G code from the same group can be replaced by another G code in the same group. By doing this the programmer establishes modes of operation. The universal rule here is that codes from the same group cannot be used more than once on the same line.

3) There are Modal G codes which once established, remain effective until replaced with another G code from the same group.

4) There are Non-Modal G codes (Group 00) which once called, are effective only in the calling block, and are immediately forgotten by the control.

The rules above govern the use of the G codes used for programming the Haas Lathe. The concept of grouping codes and the rules that apply will have to be remembered to effectively program the Haas Mill. The following is a list of Haas G codes. If there's a (Setting number) listed next to a G code, that setting will in some way relate to that G code. A single asterisk (*) indicates that it's the default G code in a group. A double asterisk (**) indicates available options.

Code	Group	Function
G00*	01	Rapid Motion (Setting 10, 56, 101)
G01	01	Linear Interpolation Motion
G01	01	Linear Interpolation Motion with Chamfering and Corner Rounding
G02	01	CW Interpolation Motion
G03	01	CCW Interpolation Motion
G04	00	Dwell (P) (P=seconds"."milliseconds)
G05**	00	Fine Spindle Control Motion (Live Tooling)
G09	00	Exact Stop, Non-Modal
G10	00	Programmable Offset Setting
G14**	00	Main Spindle/Sub Spindle Swap
G15**	00	Main Spindle/Sub Spindle Swap Cancel
G17**	02	Circular Motion XY Plane Selection (G02, G03) (Live Tooling)
G18*	02	Circular Motion ZX Plane Selection (G02, G03) (Setting 56)
G19**	02	Circular Motion YZ Plane Selection (G02, G03) (Live Tooling)
G20*	06	Verify Inch Coordinate Positioning (Setting 9 will need to be INCH) (Setting 56)
G21	06	Verify Metric Coordinate Positioning (Setting 9 will need to be METRIC)
G28	00	Rapid to Machine Zero Thru Reference Point, Cancel Offsets (Fanuc)
G29	00	Move to Location Thru G29 Reference Point (Fanuc)
G31**	00	Feed Until Skip Function
G32	01	Thread Cutting Path, Modal
G40*	07	Tool Nose Compensation Cancel (Setting 56)
G41	07	Tool Nose Compensation Left (Setting 43, 44, 58)
G42	07	Tool Nose Compensation Right (Setting 43, 44, 58)

SL Series WORKBOOK

<u>Code</u>	<u>Group</u>	<u>Function</u>
G50	11	Spindle Maximum RPM Limit
G51	11	Rapid to Machine Zero, Cancel Offsets (YASNAC)
G52	00	Work Offset Positioning Coordinate (Setting 33, YASNAC)
G52	00	Global Work Coordinate System Shift (Setting 33, FANUC)
G53	00	Machine Zero Positioning Coordinate Shift, Non-Modal
G54*	12	Work Offset Positioning Coordinate #1 (Setting 56)
G55	12	Work Offset Positioning Coordinate #2
G56	12	Work Offset Positioning Coordinate #3
G57	12	Work Offset Positioning Coordinate #4
G58	12	Work Offset Positioning Coordinate #5
G59	12	Work Offset Positioning Coordinate #6
G61	13	Exact Stop, Modal
G64*	13	Exact Stop G61 Cancel (Setting 56)
G65 **	00	Macro Sub-Routine Call
G70	00	Finishing Cycle
G71	00	O.D./I.D. Stock Removal Cycle (Setting 72, 73)
G72	00	Face Stock Removal Cycle (Setting 72, 73)
G73	00	Irregular Path Stock Removal Cycle
G74	00	Face Grooving Cycle or High Speed Peck Drill Cycle (Setting 22)
G75	00	O.D./I.D. Grooving Cycle (Setting 22)
G76	00	Threading Cycle, Multiple Pass (Setting 86, 95 ,96, 99)
G77 **	00	Flatting Cycle (Live Tooling)
G80*	09	Canned Cycle Cancel (Setting 56)
G81	09	Drill Canned Cycle
G82	09	Spot Drill / Counterbore Canned Cycle
G83	09	Peck Drill Deep Hole Canned Cycle (Setting 22, 52)
G84	09	Tapping Canned Cycle
G85	09	Bore in~Bore out Canned Cycle
G86	09	Bore in~Stop~Rapid out Canned Cycle
G87	09	Bore in~Manual Retract Canned Cycle
G88	09	Bore~Dwell~Manual Retract Canned Cycle
G89	09	Bore~Dwell~Bore out Canned Cycle
G90	01	O.D./I.D. Turning, Modal
G92	01	Threading Cycle, Modal (Setting 95, 96)
G94	01	End Facing Cycle, Modal
G95 **	09	End Face Rigid Tap (Live Tooling)
G96	12	Constant Surface Speed, CSS On
G97	12	Constant Non-Varying Spindle Speed, CSS Off (Setting 56)
G98	05	Feed Per Minute
G99*	05	Feed Per Revolution (Setting 56)
G100	00	Mirror Image G101 Cancel
G101	00	Mirror Image (Setting 45, 47)

Code	Group	Function
G102	00	Programmable Output to RS-232
G103	00	Limit Block Lookahead (P0-P15 max. for number control looks ahead)
G105	00	Servo Bar Command
G110	12	Work Offset Positioning Coordinate #7
G111	12	Work Offset Positioning Coordinate #8
G112**	00	Live Tooling Cartesian to Positioning Polar Transformation
G113**	00	Live Tooling Cartesian to Positioning Polar Transformation Cancel
G114	12	Work Offset Positioning Coordinate #11
G115	12	Work Offset Positioning Coordinate #12
G116	12	Work Offset Positioning Coordinate #13
G117	12	Work Offset Positioning Coordinate #14
G118	12	Work Offset Positioning Coordinate #15
G119	12	Work Offset Positioning Coordinate #16
G120	12	Work Offset Positioning Coordinate #17
G121	12	Work Offset Positioning Coordinate #18
G122	12	Work Offset Positioning Coordinate #19
G123	12	Work Offset Positioning Coordinate #20
G124	12	Work Offset Positioning Coordinate #21
G125	12	Work Offset Positioning Coordinate #22
G126	12	Work Offset Positioning Coordinate #23
G127	12	Work Offset Positioning Coordinate #24
G128	12	Work Offset Positioning Coordinate #25
G129	12	Work Offset Positioning Coordinate #26
G154	12	Select Work Offset Positioning Coordinate P1-99 (P)
G159**	00	Background Pickup / Part Return
G160**	00	APL Axis Command ON
G161**	00	APL Axis Command OFF
G184**	00	Reverse Tapping Canned Cycle
G186	00	Live Tooling Reverse Rigid Tapping
G187**	00	Accuracy Control for High Speed Machining (Setting 85)
G194	00	Sub-Spindle / Tapping Canned Cycle
G195**	00	Live Tooling Radial Tapping
G196**	00	Live Tooling Radial Tapping Reverse
G200	00	Index on the fly

*Defaults
** Options

Each **G** code is a part of a group of **G** codes. The Group 0 codes are nonmodal; that is, they specify a function applicable to this block only and do not affect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal **G** code applies to all subsequent blocks so those blocks do not need to re-specify the same G code.

There is also one case where the Group 01 **G** codes will cancel the Group 9 (canned cycles) codes. If a canned cycle is active (G81 through G89), the use of G00 or G01 will cancel the canned cycle.

MACHINE DEFAULTS

A **DEFAULT** is an automatic function of the machine tool control. After powering up the machine, the control will recognize the default "**G**" code values. The machine will go to the part zero that was entered in for **G54** if no other work coordinate code was specified in the actual program, because the machine automatically recognizes the **G54** column upon start-up. That is a **DEFAULT**. The defaults for the Haas mill are indicated by an asterisk (*****) in the "Preparatory Functions (**G** Codes)" list of this workbook.

The control automatically recognizes these **G** codes when your HAAS lathe is powered up:

G00   Rapid Traverse
G18   XZ Circular Plane Selection
G40   Cutter Compensation Cancel
G54   Work Coordinate Zero #1 (1 of 26 available)
G64   Exact Stop Cancel
G80   Canned Cycle Cancel
G97   Constant Surface Speed Cancel
G99   Feed Per Revolution

There is no default feedrate (F code) or spindle speed (S code) , but once an F or S code is programmed, it will apply until another feedrate or spindle speed is entered or the machine is turned off.

MACHINE CYCLES FOR THE LATHE

Machining cycles are used to simplify the programming of a part. Machine cycles are used for the most common machining operations, and they're two types: They're machining cycles for turning and grooving and they're canned cycles for drilling, tapping and boring.

MACHINING CYCLES FOR TURNING & GROOVING

The following is a list of the canned cycles that can be used for turning and grooving for the HAAS lathe controls.

<u>MACHINING CYCLES</u>

G70	Finishing Cycle
G71	O.D./I.D. Stock Removal Cycle
G72	End Face Stock Removal Cycle
G73	Irregular Path Stock Removal Cycle
G74	End Face Grooving or Z axis turn with a Chip Break Cycle
G75	O.D./I.D. Grooving or X axis turn with a Chip Break Cycle
G76	Thread Cutting Cycle, Multiple Pass
G90	O.D./I.D. Turning Cycle Modal
G92	Thread Cutting Cycle Modal
G94	End Face Cutting Cycle Modal

A Machine Cycle is used to simplify programming of a part. Machine Cycles are used for the most common repetitive operations such as turning, facing, threading, and grooving. They're both modal and non-modal machine cycles. Modal cycles **G90, G92** and **G94** remain in effect after they are defined. After any subsequent X or Z axis positioning, the canned cycle is executed again. Modal machining cycles remain in effect until canceled by a **G80**, **G00**, an End of Program (M30), or RESET. Non-modal **G71, G72, G73, G74, G75** and **G76** Machine cycles are effective for only the block that contains them, but will be performing a series of machining moves to execute that command block.

CANNED CYCLES FOR DRILLING TAPPING AND BORING

These cycles are used to define and simplify programming for the most common Z-axis repetitive operations such as drilling, tapping, and boring. Once selected, a canned cycle is active until canceled with a **G80** code. There are six operations involved in every canned cycle:

1.) Positioning of **X** and **Z** axes.
2.) Rapid traverse to the reference **R**-plane.
3.) Drilling, boring, or tapping action.
4.) Operation at the bottom of the hole.
5.) Retraction to the reference **R**-plane.
6.) Rapid traverse to the initial starting point.

These cycles are Modal, which remain in effect after they are defined and are executed in the Z axis for each positioning of X axes in a program. Some of the cycle command values can also be changed after these cycle have been defined. The command values most often changed during a cycle are the R plane value and the Z depth value. These modal cycles will be canceled with the G80, G01 or G00 commands. The X axis moves in these modal machine cycles are performed as a rapid moves.

The operation of a canned cycle will vary according to whether incremental (U,W) or absolute (X,Z) is specified. Incremental motion is often useful in a canned cycle. If a loop count (Lnn code number) is defined within the block, the canned cycle will repeat that many times with an incremental U (X axis) move between each cycle.

The following is a list of the canned cycles that can be used on the HAAS lathe.

CANNED CYCLES

G80 * *Canned Cycle* **Cancel**
G81 Drill *Canned Cycle*
G82 Spot Drill *Canned Cycle*
G83 Peck Drill *Canned Cycle*
G84 Tapping *Canned Cycle*
G85 Bore in Bore out *Canned Cycle*
G86 Bore in Rapid out *Canned Cycle*
G87 Bore with Manual Retract *Canned Cycle*
G88 Bore in Dwell with Manual Retract *Canned Cycle*
G89 Bore in Dwell Bore out *Canned Cycle*

ALPHABETICAL ADDRESS CODES

The following is a list of the Address Codes used in programming the Lathe.

B   LINEAR B-AXIS MOTION (Tailstock) (Setting 93, 94, 105, 106, 107, 121, 145)

The **B** address character is currently reserved for the tailstock. It is used to specify absolute position or motion for the tailstock along the B axis. B axis commands in the negative direction moves the tailstock toward the spindle, and a B axis command in the positive direction moves it away from the spindle.

F   FEED RATE (Setting 19, 77)

The F address character is used to select feed rate applied to any interpolating G codes or canned cycles. This command value is in, inches per revolution or mm per revolution. Inches per revolution (G99) is the default. But it can be changed to units/minute with G98. Traditionally, the F code was capable of only 4 fractional position accuracy; but on this control you can specify F to six place accuracy. Code E and F are equivalent.

G   PREPARATORY FUNCIONS (G codes)

The G address character is used to specify the type of operation to occur in the block containing the G code. The G is followed by a two or three digit number between 0 and 187. Each G code defined in this control is part of a group of G codes. The Group 0 codes are non-modal; that is, they specify a function applicable to this block only and do not effect other blocks. The other groups are modal and the specification of one code in the group cancels the previous code applicable from that group. A modal G code applies to all subsequent blocks so those blocks do not need to re-specify the same G code. More than one G code can be placed in a block in order to specify all of the setup conditions for an operation.

I   CIRCULAR INTERPOLATION / CANNED CYCLE DATA

The I address character is used to specify data used for some canned cycles and circular motions. It is either in inches with four fractional positions or mm with three fractional positions.

J   CANNED CYCLE DATA

The J address character is used to specify data used for some canned cycles.

K   CIRCULAR INTERPOLATION / CANNED CYCLE DATA

The K address character is used to specify data used for some canned cycles and circular motions. It is formatted just like the I data.

L   LOOP COUNT TO REPEAT A COMMAND LINE

The L address character is used to specify a repetition count for some canned cycles and auxiliary functions.

M MISCELLANEOUS FUNCTIONS (M Codes)

The M address character is used to specify an M code for a block. These codes are used to control miscellaneous machine functions. Note that only one M code is allowed per block of the CNC program and all M codes are performed at the end of the block.

N LINE/BLOCK NUMBER

The N address character is entirely optional. The only function of a N number is to identify and locate a certain block or line within a program.

O PROGRAM NUMBER (PROGRAM name in parenthesis)

The O address character is used to identify a program. It is followed by a number between 0 and 99999. A program saved in memory always has a Onnnnn identification in the first block. Altering the Onnnnn in the first block causes the program to be renumbered. If you enter a program name (Name) between parenthesis in the first three lines of a program, that program name will also be seen in your list of programs. You can have up to 500 program numbers (200 programs on an older machine) in your List of Programs. You can delete a program number from the LIST PROG display, by cursor selecting the program, and pressing the ERASE PROG key. You can also delete a program in the advanced editor using the menu item DELETE PROGRAM FROM LIST.

P DELAY OF TIME / M97 SEQUENCE NUMBER CALL / M98 PROGRAM NUMBER CALL / LIVE TOOLING SPINDLE SPEED

The P address character is used as a delay of time in seconds for a dwell command, or as a P number to search for a sequence number in a local subroutine call, or as a P number to search for a program number in your list of programs for a subprogram call. P is also defined with Q, and is used in canned cycles G70, G71, G72 and G73 to specify the starting block number of the part geometry defined for machining with these cycles.

Q CANNED CYCLE DATA

The Q address character is used in a G83 canned cycle and is a positive number for the peck amount. Q is also defined with P, in the canned cycles G70, G71, G72 and G73 to specify the ending block number of the part geometry defined for machining with these cycles.

R CIRCULAR INTERPOLATION / CANNED CYCLE DATA (Setting 52)

The R address character is used in canned cycles and circular interpolation. It is usually used to define the reference plane for canned cycles.

S SPINDLE SPEED COMMAND (Setting 20, 144)

The S address character is used to specify the spindle speed. The S command does not turn the spindle on or off; it only sets the desired speed. By default, S specifies RPM. When used with G96, S specifies surface feet peer minute.

T   TOOL SELECTION CODE (Setting 42, 87, 97)

The T number address calls a tool and an offset when initiating a tool change. Txxyy is the T command format. The first two digits (xx) specifies the turret position and is used to call up a tool that's between 1 and the number of tool turret positions on the machine. The second two digits (yy) calls up a tool geometry/wear offset that are going to be used for that tool and will be a number between 1-50.

The **T**xxyy code selects the tool and offset number that are going to be used. The **T** numbers differ slightly depending on if Setting 33, Coordinate System, is set for FANUC or YASNAC . Most people set it asFANUC and is what we ship the Haas machines out as from Haas. When set to FANUC, the "xx" calls up a tool number (any xx leading zero numbers are ommitted); The yy specifies tool geometry and wear offset that are going to be used for that tool. A T101 would be recognized as tool 1 offset 1; T323 would be recognized as tool 3 offset 23. If you command T300, that would call tool 3 and cancel any active offsets. When you rapid to a location from machine home with no offsets, it'll be a minus position value in X and Z axis or, it'll be from a part zero origin location, that'll be using a tool geometry "**T**xxyy" offset commanmd

U   INCREMENTAL X AXIS MOTION

The U address character is used to specify motion for the X-axis. It specifies an incremental position or distance along the X-axis relative to the current machine position. It is defined either in inches with four fractional positions or in mm with three fractional positions.

W   INCREMENTAL Z AXIS MOTION

The W address character is used to specify motion for Z-axis. It specifies an incremental position or distance along the Z-axis relative to the current machine position. It is defined either in inches with four fractional positions or in mm with three fractional positions.

X   ABSOLUTE X AXIS MOTION (Setting 45)

The X address character is used to specify absolute motion for the X-axis. It specifies a position or distance along the X-axis. It is either in inches with four fractional positions or mm with three fractional positions.

Z   ABSOLUTE Z AXIS MOTION (Setting 47)

The Z address character is used to specify absolute motion for the Z-axis. It specifies a position or distance along the Z-axis. It is either in inches with four fractional positions or mm with three fractional positions.
Note: For all motion commands, if the decimal point is omitted, the value is in units of .0001" (or .001mm)

Note: For all motion commands, if the decimal point is omitted, the value is in units of .0001" (or .001mm)

RAPID POSITION COMMAND

G00 RAPID POSITIONING MOTION
  * **X** Absolute X-axis positioning command
  * **Z** Absolute Z-axis positioning command
  * **U** Incremental X-axis positioning command
  * **W** Incremental Z-axis positioning command
  * **B** Absolute tailstock command
  * indicates optional

This G code is used for rapid traverse of the two axes of the machine from one program point to the next program point. The auxiliary (tailstock) B axis, can also be moved in rapid with a G00. This G code is modal so that all following blocks will be in rapid motion until another group 01 G code is specified. Generally, rapid motion done in both X and Z axes will not be in a straight line from one program location to the next program location. All the axes specified are moved at the maximum speed and will not necessarily complete each axis move at the same time. So you need to be careful of any obstructions to avoid with this type of rapid move. The tool will first move from the current position in a straight line along a 45 degree angle to an intermediate location, until one of these axes has completed its move. Then the machine will move parallel to the X or Z axis to complete the rapid move to the final location. These rapid moves may be in ABSOLUTE or INCREMENTAL coordinate command values which will change how those values are interpreted. The "U" letter address relates to X-axis incremental moves and the "W" letter address relates to Z-axis incremental moves.

Pt. "A" X4.0 Z.1

CUTTER PATH

Pt. "B" X2.2 Z3.0
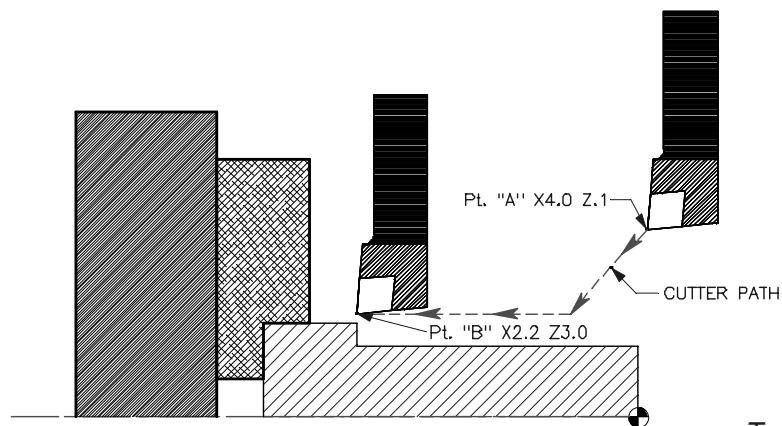
*To move from point "**A**" to point "**B**", the programmed line can be either:*

	G00 X2.2 Z-3.0
or	**G00 U-1.8 W-3.1**
or	**G00 X2.2 W-3.1**
or	**G00 U-1.8 Z-3.0**

LINEAR INTERPOLATION COMMAND

G01 LINEAR INTERPOLATION MOTION
   * **X** Absolute X-axis motion command
   * **Z** Absolute Z-axis motion command
   * **U** Incremental X-axis motion command
   * **W** Incremental Z-axis motion command
   * **F** Feed rate in inches (or mm) per minute
   indicates optional

This G code provides for straight line (linear) motion from point to point. Motion can occur in 1 or 2 axes. Both axes will start and finish motion at the same time to move the tool along a straight line path parallel to an axis or at a slope (angled) line. The speeds of all axes are controlled so that the feedrate specified is achieved along the actual path. The F (Feedrate) command is modal and may be specified in a previous block. These moves may be made in ABSOLUTE or INCREMENTAL coordinate command values which change how those values are interpreted. The "**U**" letter address relates to X-axis incremental moves, and the "**W**" letter address relates to Z-axis incremental moves. Only those axes specified are moved in either absolute **X Z**, or incremental **U W** commands.

When feeding to a location using G01, it should be using a previously defined tool geometry "**T**xxyy" offset commanmd. The **T**xxyy code defines the tool and offset that are being used. This **T** command differs slightly depending on Setting 33 FANUC or YASNAC coordinate system. Most people set if for FANUC. When set to FANUC, xx is the tool number (any xx leading zero is ommitted); and yy is the tool geometry and wear offset that is being used for that tool. T101 would be recognized as tool 1 offset 1 and T323 would be recognized as tool 3 offset 23.

G00 X0.9106 Z0.1
  (ABSOLUTE)
G01 X1.5 Z-1. F0.006
Z-3.
X2.
	G00 X0.9106 Z0.1
	  (INCREMENTAL)
	G01 U.5894 W-1.1 F0.006
	W-1.9
	U0.5
		G00 X0.9105 Z0.1
		  (ABS. AND INC.)
		G01 X1.5 W-1.1 F0.006
		Z-3.
		U0.5
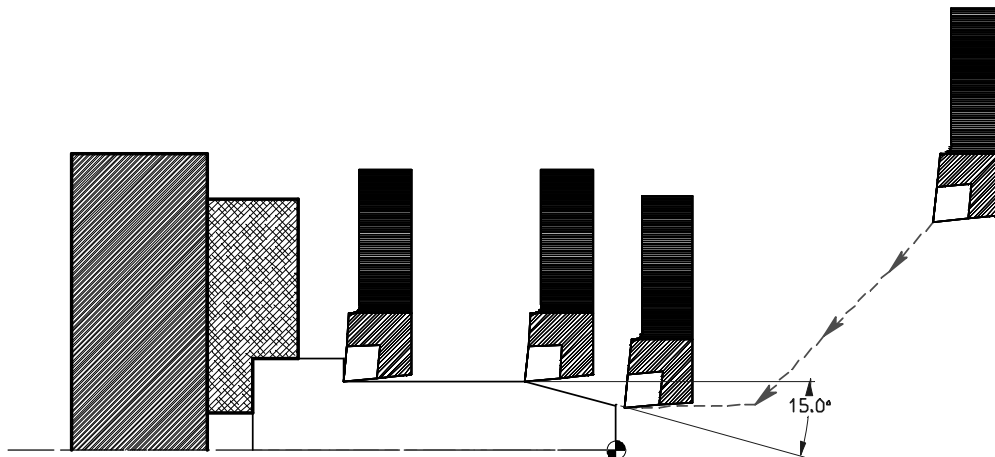
LINEAR INTERPOLATION EXERCISE

Linear interpolation to feed in a straight line motion from point to point for machining diameters, faces and corner chamfering in Absolute or Incremental. With what we have learned about **G01**, we can move the edge of a cutter along the part profile, by a series of Absolute **X** and **Z** axis movements, or Incremental **U** (X axis) and/or incremental **W** (Z axis) movements. The part can be programmed with both **X** and **Z** absolute movements and/or incremental **U** and **W** movements. You can define the actual points around part using both incremental and absolute.
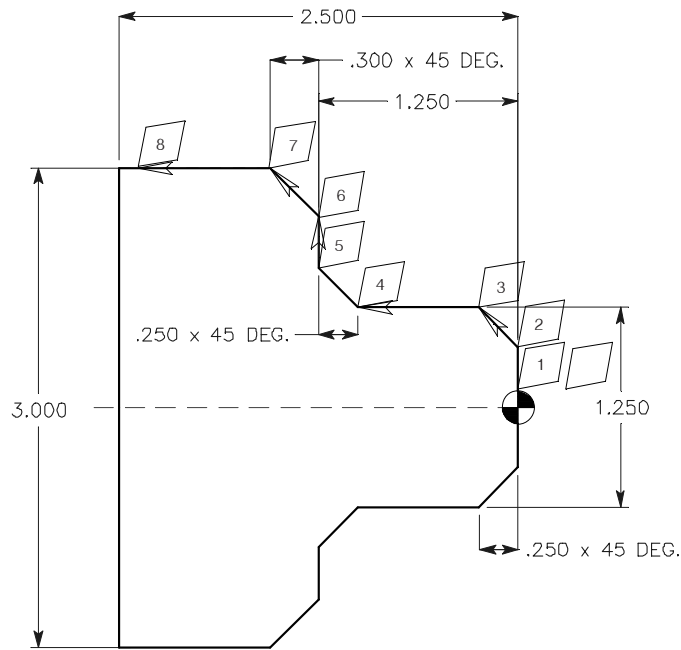
ABSOLUTE PROGRAMMING

```
N... G00 X0.25 Z0.1
N11 G___ Z0. F0.006
N12 X_____
N13 X_____    Z-_____
N14 Z-_____
N15 X_____    Z-_____
N16 X_____
N17 X_____    Z-_____
N18 Z-_____
N19 G00 X3.01 Z0.1
```

ABSOLUTE AND INCREMENTAL PROGRAMMING

```
N... G00 X0.25 Z0.1
N21 G___ Z0. F0.006
N22 X_____
N23 U_____    W-_____
N24 W-_____
N25 U_____W-_____
N26 U_____
N27 U_____    W-_____
N28 W-_____
N29 G00 U0.01 Z0.1
```

CIRCULAR INTERPOLATION COMMANDS

G02 CIRCULAR INTERPOLATION MOTION CW

* **X** Absolute circular end point X-axis motion command
* **Z** Absolute circular end point Z-axis motion command
* **U** Incremental circular end point X-axis motion command
* **W** Incremental circular end point Z-axis motion command
* **I** X-axis Incremental distance from the start point to arc center (If R is not used)
* **K** Z-axis Incremental distance from the start point to arc center (If R is not used)
* **R** Radius of the arc (If I and K are not used)
* **F** Feed rate in inches (or mm) per minute
indicates optional

G03 CIRCULAR INTERPOLATION MOTION CCW

* **X** Absolute X-axis arc end point motion command
* **Z** Absolute Z-axis arc end point motion command
* **U** Incremental X-axis arc end point motion command
* **W** Incremental Z-axis arc end point motion command
* **I** X-axis incremental distance from the start point to arc center (If R is not used)
* **K** Z-axis incremental distance from the start point to arc center (If R is not used)
* **R** Radius of the arc (If I and K are not used)
* **F** Feed rate in inches (or mm) per minute
indicates optional

G03 will generate counterclockwise circular motion but is otherwise defined the same as G02 clockwise circular motion.

Circular interpolation commands are used to move a tool along a circular arc to the commanded end position. Five pieces of information are required for executing a circular interpolation command: Plane selection, arc start position coordinates, rotation direction, arc end position coordinates, and arc center coordinates or arc radius.

There are two ways to define a circular interpolation move. The first way is using I and K method, which defines the distance and direction from the start point to the arc center. The R method, is easier to define, and is the actual arc radius value your interpolating.

When using I and K, it does not need to be entered in the porgram if either one has a zero value. It will be recognized as zero if it's not listed in the G02 or G03 line.

R: Using an R is the distance from the starting point to the arc center. With a positive R value, the control will generate a circular path of 180 degrees or less, but to generate a circular path of over 180 degrees, you specify a negative R.

When feeding to a location using G02 or G03, it should be using a previously defined tool geometry "Txxyy" offset commanmd. The Txxyy code defines the tool and offset that are being used.

G02 CIRCULAR INTERPOLATION MOTION CW

X Z END POINT

R I CENTER OF ARC

X Z START POINT

K
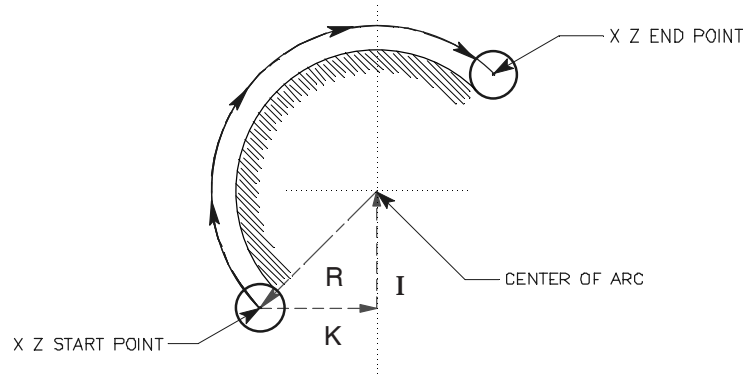
Five pieces of information are required for executing a circular interpolation command.

	ITEM	COMMAND	Remark
1.	Plane selection command Plane selection command Plane selection command	G17 G18 G19	Arc parallel to X Y-plane (NOT AVAILABLE) Arc parallel to Z X-plane (DEFAULT) Arc parallel to Y Z-plane (NOT AVAILABLE)
2.	Arc start position coordinate	X,Z	Coordinates to the start position of arc
3.	Rotation direction	G02 G03	Clockwise interpolation direction Counterclockwise interpolation direction
4.	Arc end position Absolute Arc end position Incremental	X,Z U,W	End position coordinates of the of arc defined from part origin zero point. End point, incremental distance and direction defined from the start point of arc to the end point in the X and Z axis.
5.	I and K method (arc center coordinates) ("I" is the X axis incremental distance and direction from the arc start point to the center of the radius) ("K" is the Z axis incremental distance and direction from the arc start point to the center of radius) R method (actual part arc path radius value)	I,K R	Incremental distance and direction from start point of arc to the arc center for X and Z axes using "I" for the X axis, and "K" for the Z axis. Arc radius value of toolpath. Easier to define then I & K

G03 CIRCULAR INTERPOLATION MOTION CCW

X Z END POINT

CENTER OF ARC I R

K

X Z START POINT
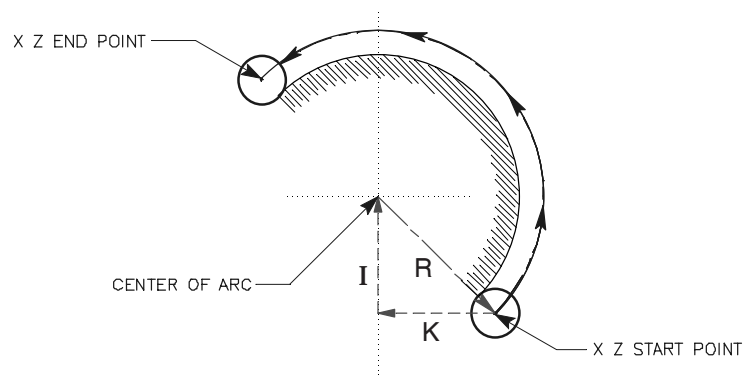
CIRCULAR INTERPOLATION MOTION EXAMPLE
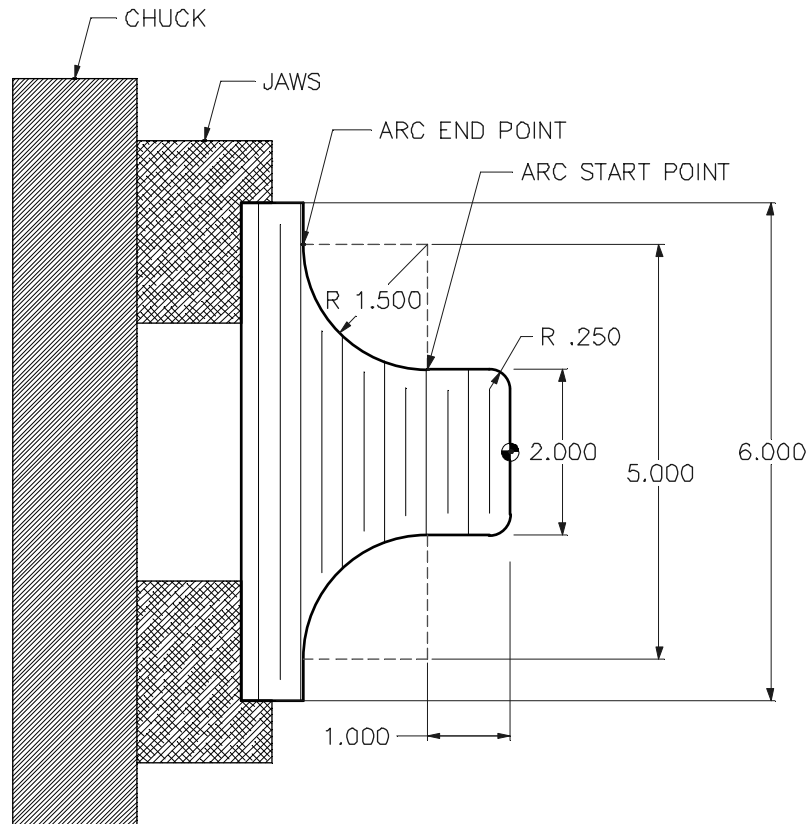
This program example starts machining with the tool at the beginning of the .250 radius to machine in a counterclockwise direction. Then moves cutter to the start of 1.500R and around in a clockwise direction from START POINT to END POINT. For each radius, the programming code would look like this:

```
N3 ……                        (Tool nose compensation  is not being used in this example. This
N4 ……                         program example is defined to a sharp tool point.)
N5 G00 X1.4 Z0.1              (Tool rapids to X1.4 Z0.1 in front of part.)
N6 G01 Z0. F0.008            (Tool feeds to Z0. face of part.)
N7 X1.5                       (Tool feeds up to 1.5 dia. in X-axis to the start of .250 radius.)
N8 G03 X2. Z-0.25 R0.25      (CCW .25 Radius to the arc end point using R.)
   or G03 X2. Z-0.25 I0. K-0.25  (or, CCW .25 Radius to the arc end point X2. Z-.25 with I and K)
N9 G01 Z-1.                   (Machines the 2 dia. to the arc start point at Z-1.)
N10 G02 X5. Z-2.5 R1.5        (CW 1.5 Radius to the arc end point using R.)
   or G02 X5. Z-2.5 I1.5 K0.  (or, CW 1.5 Radius to the arc end point X5. Z-2.5 with I and K)
N11 G01 X6.0                  (Feeds up X-axis to 6. dia.)
N12 G00 Z0.1
N…
```

CIRCULAR INTERPOLATION EXERCISES

EXERCISES FOR G01 AND G02 USING I AND K
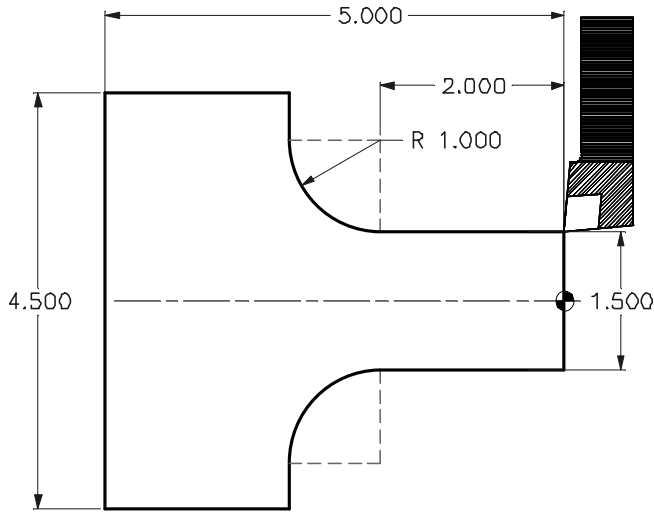
From the tools current position, program the tool path to turn the 1.500 dia. to the START POINT of the 1.000 radius and then interpolate the radius. Finish the cut by feeding up to the 4.500 dia.

("I" is for the X axis incr. distance and direction from start point to arc center, and "K" is for the Z axis incr. distance and direction from start point
to arc center.)

N11 G___ Z_____   F0.01
N12 G___ X_____ Z_____ I_____ K_____
N13 G___ X_____
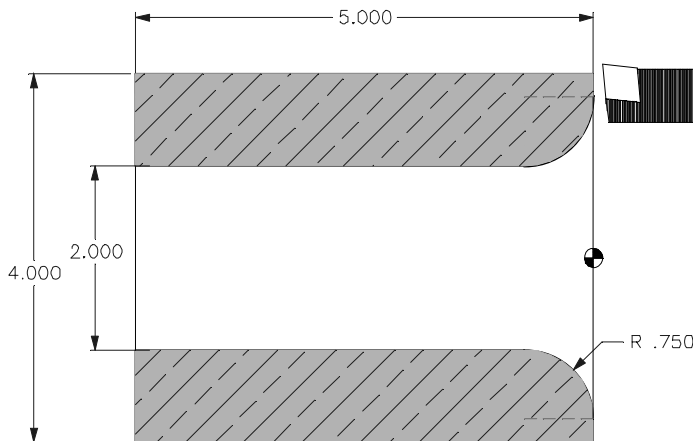
From the tools current position, feed into the face and down to the START POINT of radius. Then machine the radius down into the bore, and bore thru to the end of part.

N21 G___ Z0. F0.01
N22 X_____
N23 G___ X_____ Z_____ I_____ K_____
N24 G___ Z_____

EXERCISES FOR G01 AND G03 USING I AND K
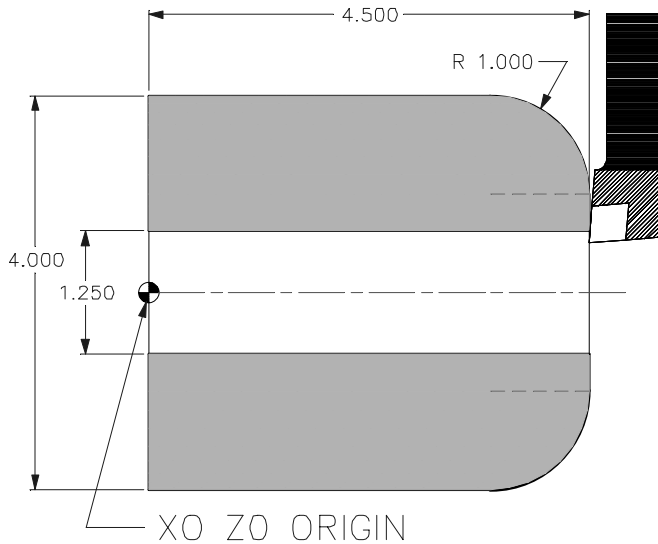
From the tools current position, program the tool path to feed up the face to the START POINT of the 1.0 radius. Then machine radius and turn the 4.000 diameter to the end of part.

("I" is for the X axis incr. distance and direction from start point to arc center, and "K" is for the Z axis incr. distance and direction from start point to arc center.)

N31 G____ X_____ F0.01
N32 G____ X_____ Z_____ I_____ K_____
N33 G____ Z_____
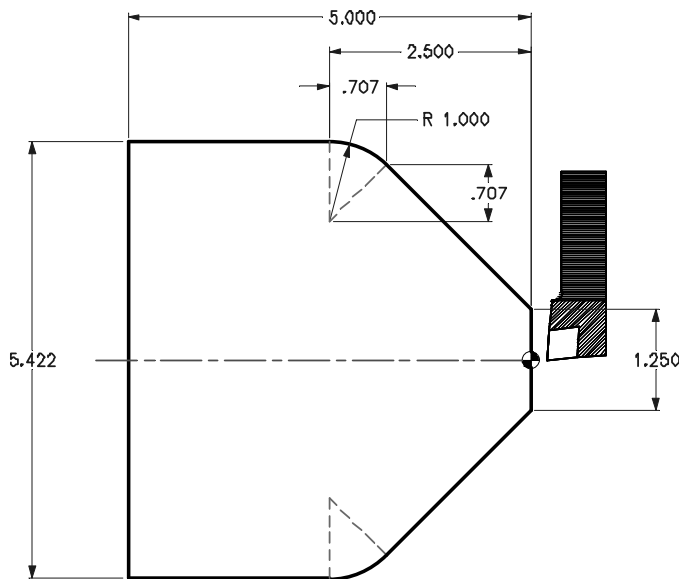
From the tools current position, feed into the face of the part. Feed up the face to the 45 degree angle to the START POINT of radius. Then machine 1.00 radius and the 5.422 dia. to the end of part.

N41 G01 Z_____ F0.01
N42 X_____
N43 X_____ Z_____
N44 G____ X_____ Z_____ I_____ K_____
N45 G____ Z_____

EXERCISES FOR G02 AND G03 USING R

FORMAT: G02 and G03 (Circular Interpolation Clockwise/Counterclockwise)
   using "R" (radius) in place of "I" and "K".

It is sometimes easier to program the "R" (radius) word in place of the "I" and "K" words. The "I" and "K" words are used to define the incremental distance and direction from START POINT to the ARC CENTER. This can also be done by using the "R" word in place of "I" and "K".

NOTE: Any radius being cut using the "R" word needs a positive (+) value for a radius that is 180 degrees or less, and for a radius that is larger than 180 degrees it needs to have a minus (-) sign with the radius value.
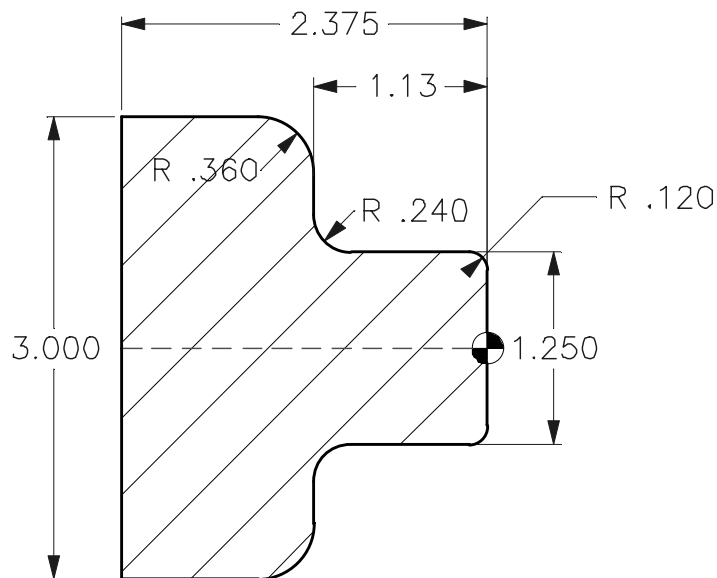
Program example
for G02 and G03 using "I" and "K"

N11 G01 Z0. F0.005
N12 X1.01
N13 G03 X1.25 Z-0.12 I_____ K_____
N14 G01 Z-0.89
N15 G02 X1.73 Z-1.13 I_____ K_____
N16 G01 X2.28
N17 G03 X3. Z-1.49 I_____ K_____
N18 G01 Z-2.375

Program example
for G02 and G03 using "R"

N21 G01 Z0. F0.005
N22 X1.01
N23 G03 X1.25 Z-0.12 R_____
N24 G01 Z-0.89
N25 G02 X1.73 Z-1.13 R_____
N26 G01 X2.28
N27 G03 X3. Z-1.49 R_____
N28 G01 Z-2.375

CHAMFER AND CORNER ROUNDING WITH G01

G01 CHAMFER OR CORNER RADIUS WITH LINEAR G01 MOTION

* **X** Absolute X-axis motion command
* **Z** Absolute Z-axis motion command
* **U** Incremental X-axis motion command
* **W** Incremental Z-axis motion command
* **I** 45 degree chamfer size from Z into X axis command (I+ or I- for direction into X)
* **K** 45 degree chamfer size from X into Z axis command (K+ or K- for direction into Z)
* **R** Corner rounding radius command (R+ or R- for direction into X or Z)
* **A** Angle of chamfer command (Angle is from 3'oClock, used with only one X, Z, U, or W)
* **F** Feed rate in inches (or mm) per minute

*indicates optional

This G code provides for straight line (linear) motion from point to point that can have a chamfer or radius between 2 blocks of linear motion that intersect at a 90 degree angle. This type of command is programmed with a chamfer or corner rounding move by providing a letter command of I, K, R, or A in the G01 command block. The speeds of all axes are controlled so that the feed rate specified is achieved along the actual path. The F command is modal and may be specified in a previous block. Only the axes specified are moved and the incremental (U, W) or absolute (X, Z) commands will change how those values are interpreted.

When specifying an angle with A use only one of the other axes, the corresponding X or Z destination is calculated based on the angle from the start point. Angles are define from 0 to 360 degrees from 3 o'clock going counterclockwise. Minus values can be used to reverse this definition clockwise from 3 o'clock. Decimal points are needed for degrees.

CAUTION: This G01 Code using I, K, R, or A, to do an arc or angle is NOT recognized in the roughing passes of the G71 or G72 canned cycles. However, the **last pass** of the G71/G72 will be executed with the radius or chamfer defined with I, K, R, or A, in the G01 command. Smaller outside corner radius or chamfers could be done on the last pass. But fillet, inside corner radius or chamfers would not be there. What would be there is the radius left behind by the tool nose. The G73 or G70 commands will support this type of chamfer or radius command.

AUTOMATIC G01 CORNER ROUNDING
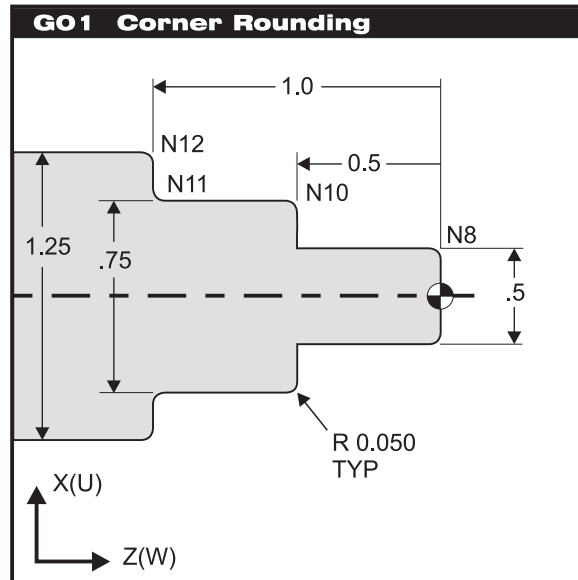
AUTOMATIC CORNER ROUNDING PROGRAM USING R EXAMPLE

```
O00042 (Linear G01 with Radius using R)
N1 G53 G00 X0. Z0. T0
N2 T101 (O.D. TOOL x .031 TNR)
N3 G50 S3000
N4 G97 S3000 M03
N5 G54 G00 X0.3 Z0.1 M08
N6 G96 S390
N7 G42 G01 Z0. F0.01
N8 G01 X0.5 R-0.05
N9 G01 Z-0.5
N10 G01 X0.75 R-0.05
N11 G01 Z-1.0 R0.05
N12 G01 X1.25 R-0.05
N13 G01 Z-1.5
N14 G40 G00 U0.01 Z0.1 M09
N15 G53 G00 X0. Z0. T0
N16 M30
```

(The R command is for a 90-degree corner radius from X into Z, or Z into X.)

(R+ is to machine a radius into the plus direction on an axis.)

(R- is to machine a radius into the minus direction on an axis.)

Rules:
1) The linear G01 block must be a single X(U) or Z(W) move with an R that is perpendicular to the previous move for corner rounding.
2) When using R for corner rounding, do not use I, K, or A.
3) Caution: This Radius (R) command is not supported in any of the roughing passes of a G71 or G72 canned cycles. The **last pass** of the G71 and G72 will be executed with the radius defined with R in the G01 command .
4) A G70 or G73 will support this type of radius command.
5) The + or - sign define the direction of the axis your moving into at the end of the arc move.

AUTOMATIC G01 CHAMFERING 45 DEGREE ANGLES

G01 Chamfering 45 Dgrees
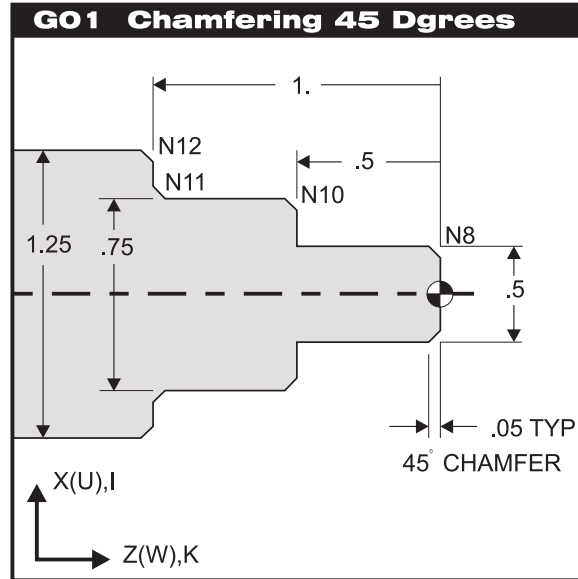
AUTOMATIC CHAMFERING PROGRAM EXAMPLE

O00043 (Linear G01 with 45 Degree Chamfer using I or K)
N1 G53 G00 X0. Z0. T0
N2 T101 (O.D. TOOL x .031 TNR)
N3 G50 S3000
N4 G97 S3000 M03
N5 G54 G00 X0.3 Z0.1 M08
N6 G96 S390
N7 G42 G01 Z0. F0.01
N8 G01 X0.5 K-0.05
N9 G01 Z-0.5
N10 G01 X0.75 K-0.05
N11 G01 Z-1.0 I0.05
N12 G01 X1.25 K-0.05
N13 G01 Z-1.5
N14 G40 G00 U0.01 Z0.1 M09
N15 G53 G00 X0. Z0. T0
N16 M30

(**I** is for the 45-degree chamfer size from Z into X. I+ is up, I- is down into the X axis.)

(**K** is for the 45-degree chamfer size from X into Z. K- is left, K+ is right into the Z axis.)

Rules:
1) The linear G01 block must be a single X(U) or Z(W) move with an I or K that is perpendicular to the previous move for chamfering.
2) When using I or K for chamfering, do not use A or R.
3) Caution: This chamfer (I,K) command is not supported in any of the roughing passes of a G71 or G72 canned cycles. The **last pass** of the G71 and G72 will be executed with the chamfer defined with I and K in the G01 command.
4) A G70 or G73 will support this type of chamfer command.
5) The + or - sign defines the direction of the axis your moving into at the end of chamfer move.

AUTOMATIC G01 CHAMFERING WITH ANGLE

G01 Chamfer Angle with A

——————— Feed

— — — — Rapid

Rapids to
X1.25 Z0.1

0.5"

Angle Ending
Diameter 1.5

1.5" Diameter

A = 170.
From 3 o'Clock
for a 10° Angle

Angle Starting
Point 1.375 Dia.

C L
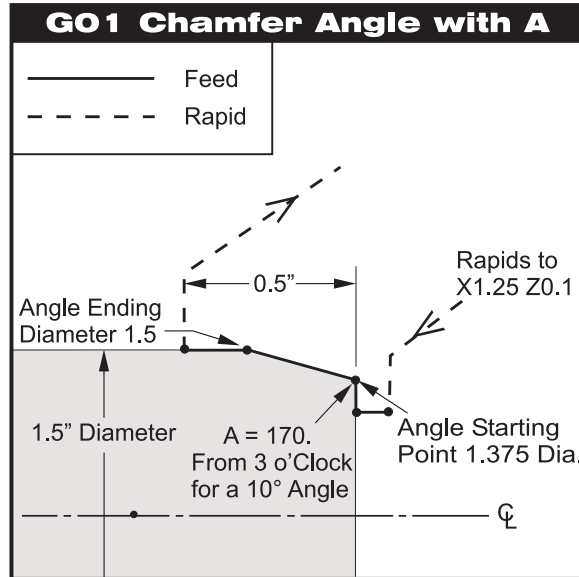
AUTOMATIC CHAMFERING PROGRAM EXAMPLE

```
O00044 (Linear G01 Chamfer 10 Degree Angle using A)
N1 G53 G00 X0. Z0. T0
N2 T101 (O.D. TOOL x .031 TNR)
N3 G50 S2800
N4 G97 S1490 M03
N5 G54 G00 X1.25 Z0.1 M08
N6 G96 S390
N7 G42 G01 Z0. F0.01
N8 G01 X1.375 (Start point)
N9 G01 X1.5 A170. (10 Degree Angle using A)
N10 Z-0.5
N11 G40 G00 U0.01 Z0.1 M09
N12 G53 G00 X0. Z0. T0
N13 M30
```

Rules:
1) The linear G01 block must be a single X(U) or Z(W) move that is perpendicular to the previous move with an A to do a specific angle.
2) When using A for an angle, do not use I, K or R.
3) Caution: This angle (A) command is "not supported in any of the roughing passes of a G71 or G72 canned cycles, though the **last pass** in the G71 and G72 will be executed with the angle defined with A in the G01 command.
4) A G70 or G73 will support this type of chamfer command.
5) You can use a minus value to define an angle clockwise from three o'clock: A-30. = A150. and A-45. = A135.
6) Be sure to enter in a decimal point for angles.

CHAMFER & CORNER ROUNDING WITH G01 EXERCISE
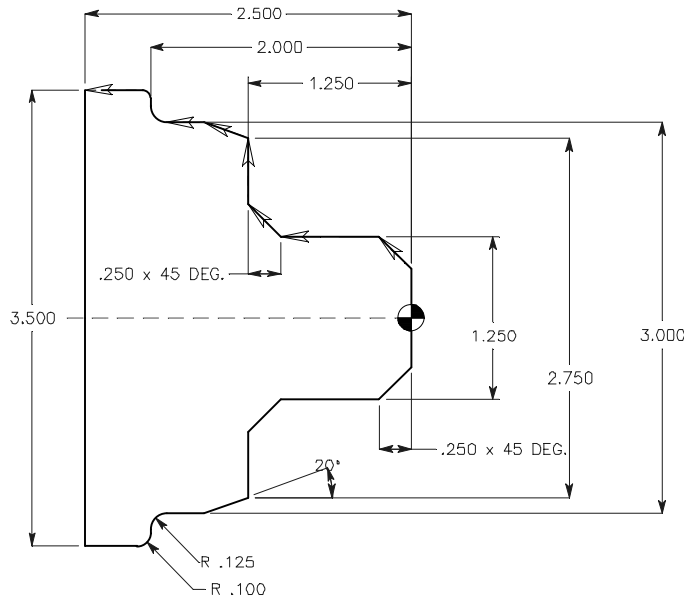
This G code provides for straight line (linear) motion from point to point that can have a chamfer or radius between 2 blocks of linear motion that intersect at a 90 degree angle. This type of command is programmed with a chamfer or corner rounding move by providing a letter command of I, K, R, or A in a G01 command block.

Note: This G01 Code using I, K, R, or A, to do an arc or angle, is not recognized in the roughing passes of the canned cycles G71 or G72.

Interpolation with G01, G02 and G03

```
O00045
N11 G53 G00 X0. Z0. T0
N12 T101 (O.D. TOOL x .031 TNR)
N13 G50 S2600
N14 G97 S2057 M03
N15 G54 G00 X0.65 Z0.1
N16 G96 S350
N17 G01 G42 Z0. F0.006
N18 X.75
N19 X1.25 Z-0.25
N20 Z-1.0
N21 X1.75 Z-1.25
N22 X2.75
N23 X3.  Z-1.5934 (Calculated Z move)
N24 Z-1.875
N25 G02 X3.25 Z-2.0 R0.125
N26 G01 X3.3
N27 G03 X3.5 Z-2.1 R0.1
N28 G01 Z-2.375
N29 G00 G40 U0.01 Z0.1 M09
N30 G53 G00 X0. Z0. T0
```

Interpolation with G01 using I, K, R & A

```
O00050
N31 G53 G00 X0. Z0. T0
N32 T101 (O.D. TOOL x .031 TNR)
N33 G50 S2600
N34 G97 S2057 M03
N35 G54 G00 X0.65 Z0.1
N36 G96 S350
N37 G42 G___ Z0. F0.006

N38 X_____  K_____  (K+ or - ?)

N39 Z_____ I_____  (I+ or - ?)
N40 X_____ (To X start point for A)
N41 X_____ A_____  (A+ or - ?)

N42 Z_____ R_____  (R+ or - ?)

N43 X_____ R_____  (R+ or - ?)
N44 Z_____
N45 G40 G00 U0.01 Z0.1 M09
N46 G53 G00 X0. Z0. T0
```

MISCELLANEOUS G CODES

G04 Dwell
    P   The dwell time in seconds or milliseconds

G04 is used to cause a delay or dwell in the program. The block containing G04 will delay for the time specified in the P code, coolant and spindle will remain on. If the P has a fraction part, the delay is in milliseconds (0.001 seconds); otherwise the delay is in seconds.

        EXAMPLE: G04 P1.0 (for a delay of 1.0 second)
                       -or-
                  G04 P2.5 (for a delay of 2.5 seconds)

G20 INCH / G21 METRIC SELECTION

The G codes G20 and G21 are used to select between inch and metric. In the HAAS control, the G20 (inch) and G21 (mm) codes can only be used to make sure that the inch/metric setting is set correctly for that program.

Selecting between inch and metric programming can only be done from the Setting page, Setting 9. Changing the setting from inches to metric or back again will interpret the content of any program listed in memory. You must load in a program with metric values after changing this setting to MM.

REFERENCE POINT RETURN COMMANDS
G28   FANUC RETURN TO MACHINE ZERO THROUGH REFERENCE POINT

The G28 code is used to return to the machine zero position on all axes. If an **X** or **Z** axis is specified on the same block, only those axes will move and return to the machines' zero point. If **X** or **Z** specifies a different location than the current position, then the movement to machine zero will be through the specified point. If no **X** or **Z** is specified, all axes will be moved directly to machine zero, along with the tailstock, **B** axis (for the machines that have a tailstock). If you do not want to position through an intermediate point while specifying an **X** or **Z** axis to position to machine home, than use an incremental **U0** and/or **W0** command for the specific axis that you want to send to machine zero. This will command those axis specified to position incrementally to a zero distance as an interme-diate point, and then those axes specified will go directly to machine zero. G28 will not recognize any tool length offsets in this move, though the offset is not cleared. The tool offsets will not be cleared until a T0, M30, Reset, or a new offset is called up.

The G28 command will send turret in the X and Z axes to machine home along with tailstock. If the tailstock is home not being used, then this is not a problem. If tailstock is being used, and is in holding a part, you may not want to send the tailstock home when a G28 is on a line in a program. To command only the turret to go home, and not the tailstock, program in **G28 U0 W0** to send only the X and Z axes home and the tailstock will remain in place.

G51   YASNAC RETURN TO MACHINE ZERO AND CANCEL OFFSET

G51 is used on a YASNAC control, to cancel out any existing tool shift and tool wear offsets and then return to machine zero position.

SPINDLE SPEED COMMANDS

G50 SPINDLE SPEED CLAMP

Spindle speed, on any CNC lathe, is limited by the maximum permissible speed of the power chuck. The effects of centrifugal force on its gripping force, unbalanced condition of the workpiece causing machine vibration, and so on, can be constrained by programming a G50 code at the start of each tool.

G50 can be used to clamp the maximum spindle speed. The control will not allow the spindle to exceed the **S** address value specified in the most recent **G50** command. This is most often used in constant surface feed mode G96.

N1   G50 S2500 ;   (SPINDLE RPM WILL NOT EXCEED 2500 RPM)
N2   G96 S390 M03 ;   (ENTER CONSTANT SURFACE SPEED MODE, SPINDLE ON)

The MAXIMUM spindle speed is designated by the "**S**" word along with the preparatory **G50** command with the "**S**" word in the same block with no other commands.

Once the MAXIMUM spindle speed is established, any direct RPM programmed in G97 (direct revolution per minute) or control calculated RPM from the G96 (constant surface footage mode) that exceeds the RPM established by the **G50** block, is ignored and the **G50 S** word spindle speed is used.

The second use of G50 is for older equipment that does not have work zero and geometry offset capabilities. This will not be covered in this workbook because it's not desirable anymore because it's not as easy to use as what is now available.

G96 CONSTANT SURFACE SPEED ON

A G96 commands the control to maintain a constant surface speed, with the spindle speed and where the tool tip is relative to the part diameter. Surface speed is based on the distance of the tool tip to the spindle centerline. This is the radius of cut. Surface speed is maintained by adjusting the spindle speed based on the radius of cut. The current **S** code is used to determine the surface speed. **G96** is modal.

The formula the machine performs while machining up or down a diameter with G96 on a part is calculating this formula: (SFM x 3.82) $\div$ Dia. = RPM
Example on a 3/4 part diameter: (225 x 3.82) $\div$ .750 = 1146

G97 CONSTANT SURFACE SPEED OFF / REVOLUTION PER MINUTE ON

A G97 commands the control to **not** adjust the spindle speed based on the radius of cut. This command will cause the spindle to be at a constant speed. It will cancel any current G96 command. When **G97** is actively on, any **S** command is in revolution per minute (RPM). **G97** is modal.

The formula to find out what surface speed when machining on a part diameter, when the machine is at a constant spindle speed: Diameter x .2618 x RPM = SFM
The surface speed for a 5/8 drill at a 1400 spindle speed is: .625 x .2618 x 1400 = 229.075

WORK COORDINATE SELECTION COMMANDS

The HAAS CNC lathe control supports both YASNAC and FANUC coordinate systems. Work coordinates together with tool offsets can be used to position a part program anywhere within the work cell with great flexibility.

G52 SET LOCAL COORDINATE SYSTEM FANUC

This code selects the user coordinate shift value. It's non-modal. FANUC compatible.

G53 NON-MODAL MACHINE COORDINATE SELECTION

This code temporarily cancels work coordinates offset and uses the machine zero coordinate location. It's non-modal, so the next block will revert to the work coordinate that was previously active.

G54-59 WORK COORDINATE OFFSET SYSTEM #1 - #6

These codes select one of the six work coordinate (part zero) offsets that are entered in and stored on the offsets display page . All subsequent references to axes' positions will be interpreted with the new coordinate value.

G110-G129 WORK COORDINATE OFFSET SYSTEM #7-26

These codes are the same as work offsets G54 to G59.

G154 P1-99 WORK COORDINATE OFFSET SYSTEM #7-105

This was added in Lathe software version 5.02 and above. This feature has 79 additional work offsets. In all previous versions, the user was limited to a maximum of 27 work offsets. These included the standard work offsets (designated G54 through G59), and twenty additional work offsets (G110 through G129). Now with G154 P1 to P99, your able to activate additional work offsets. For example G154 P10 will select work offset 10 from the list of additional work offset. Note that G110 to G129 refer to the same work offsets as the G154 P1 through P20. I.e., they can be selected by using either method. The Work Offset display screens have been adjusted accordingly. The Position page display has also been enhanced so that when a G154 work offset is active, the heading in the upper right work offset will show the G154 P value.

FEED COMMANDS

G98 FEED PER MINUTE

This command changes how the F address code is interpreted. The value of F indicates inches per minute when Setting 9 is set to INCH, and F indicates millimeters per minute when Setting 9 is set to METRIC. This code is modal.

$$IPM = CURRENT\ RPM \times IPR$$

G99 FEED PER REVOLUTION

This command changes how the F address is interpreted. The value of F indicates inches per revolution of the spindle when Setting 9 is set to INCH, while F indicates millimeters per revolution of the spindle when Setting 9 is set to METRIC. This code is modal, and is the default feed mode.

$$IPR = IPM : CURRENT\ RPM$$

MANUALLY PROGRAMMING TOOL NOSE COMPENSATION FOR A RADIUS

If you decided to manually calculate tool nose compensation for a radius on your part, It would be a hassle. Our suggestion is, if you want to save time and money, learn how to use TNC, it will help you in programming your part. This section briefly defines what you would need to calculate if you didn't use tool nose compensation.

MANUALLY CALCULATING COMPENSATION FOR A RADIUS ON YOUR PART

When you program a straight line in either X or Z the tool tip touches the part at the same point where you touched your original tool offsets in X and Z. However, when you program a radius, the theoretical tool tip does not touch the part radius. Where the tip actually touches the part is dependent upon the radius of the tool and the point around the radius being cut. If you were to try and program your part without using any compensation and you programmed to the finish radius size of your part, you would see a smaller radius on the outside corners of your part. And you would see larger radiuses on the fillet radiuses of your part. The amount that will be either a smaller corner radius or a larger fillet radius will be the amount that is the radius of the tool that you are machining with.

Refer to the illustrations below while reading the text:

For a 90 degree corner radius, to calculate the correct tool path position for a tool with a .031 TNR in your program. Manually calculate the compensated path by adding .031 to the radius to be machined. You will also need to recalculate the start point and end point of this larger programmed radius.

SHIFT Z AXIS

ARC COMMANDED BY PROGRAM W/O COMPENSATION

DESIRED PART RADIUS

TNR

TNR

SHIFT X AXIS
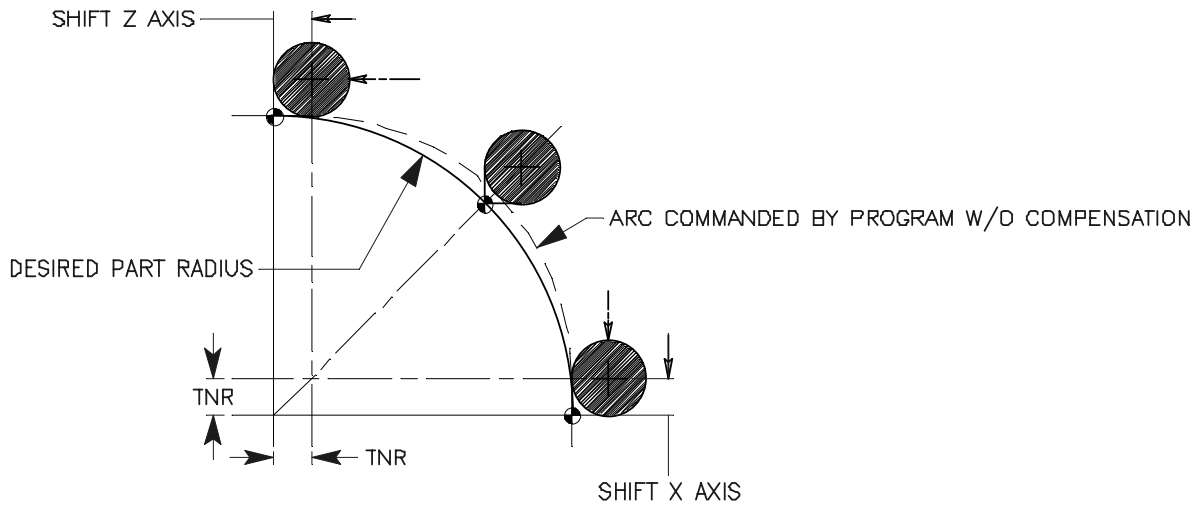
For a 90 degree fillet radius, to calculate the correct tool path position for a tool with a .031 TNR in your program. Manually calculate the compensated path by subtracting .031 from the radius to be machined. You will also need to recalculate the start point and end point of this smaller programmed radius.

DESIRED PART RADIUS

SHIFT Z AXIS

ARC COMMANDED BY PROGRAM W/O COMPENSATION

SHIFT X AXIS
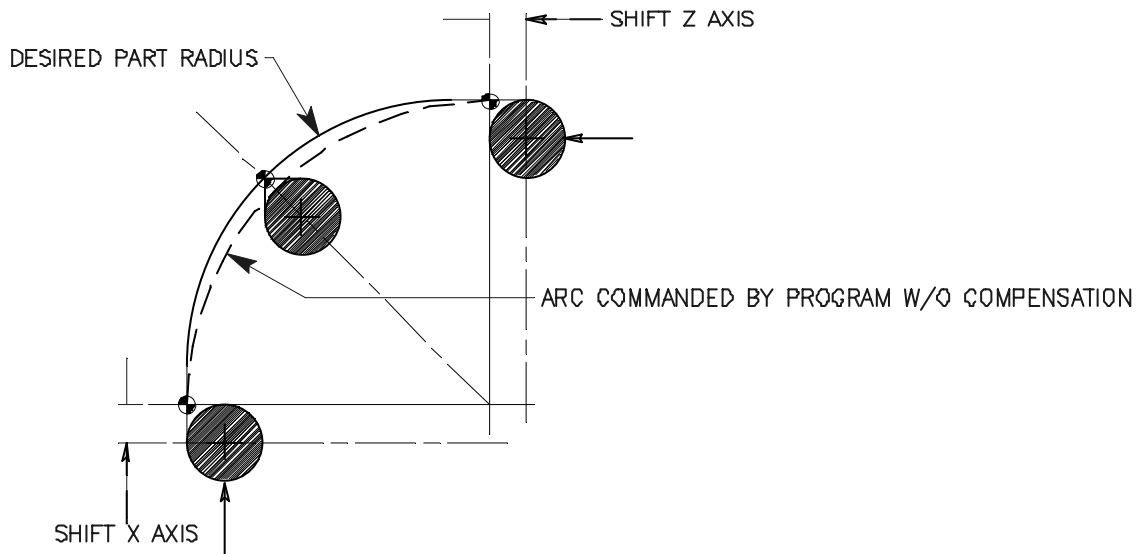
MANUALLY ADDING RADIUS COMPENSATION FOR CIRCULAR MOVES EXAMPLE

Manually Calculating Compensation for a Lathe tool with a .031 tnr to machine a radius on your part.
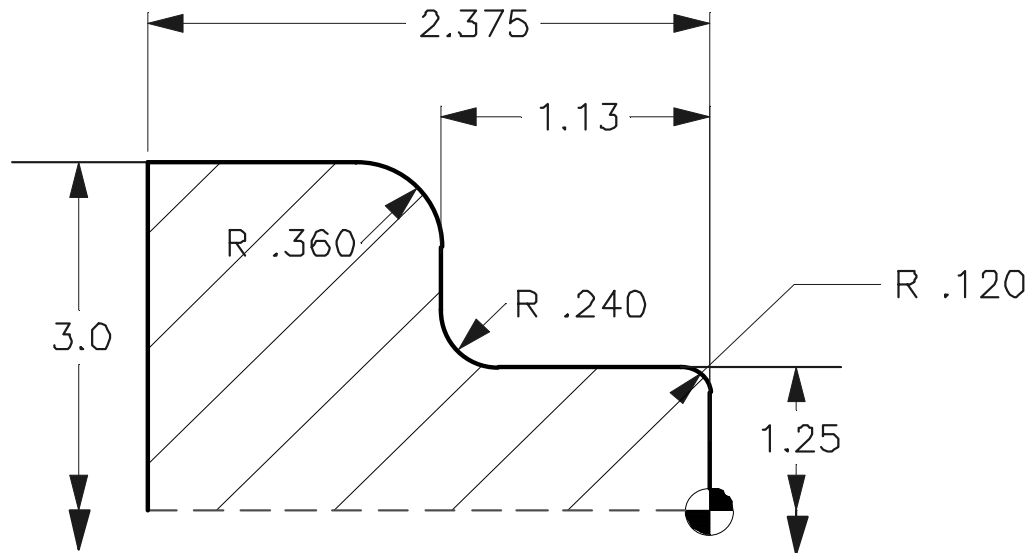
Manual Program and compensation of a part radius with a tool that has .031 TNR

For G02 and G03 using "I" and "K":   For G02 and G03 using "R":

```
O00052
N101 G53 G00 X0. Z0. T0
N102 T101 (O.D. TOOL)
N103 G97 S1450 M03
N104 G54 G00 X0.91 Z0.1
N105 G01 Z0. F0.01
N106 X0.948 F0.006
N107 G03 X1.25 Z-0.151 I0. K-0.151
N108 G01 Z-0.921
N109 G02 X1.668 Z-1.13 I.209 K0.
N110 G01 X2.218
N111 G03 X3. Z-1.521 I0. K-0.391
N112 G01 Z-2.375
N113 G00 U0.01 Z1.0
N114 G54 G00 X0. Z0. T0
N115 M30
```

```
O00052
N201 G53 G00 X0. Z0. T0
N202 T101 (O.D. TOOL)
N203 G97 S1450 M03
N204 G54 G00 X0.91 Z0.1
N205 G01 Z0.  F0.01
N206 X0.948 F0.006
N207 G03 X1.25 Z-0.151 R0.151
N208 G01 Z-0.921
N209 G02 X1.668 Z-1.13 R0.209
N210 G01 X2.218
N211 G03 X3. Z-1.521 R0.391
N212 G01 Z-2.375
N213 G00 U0.01 Z1.0
N214 G54 G00 X0. Z0. T0
N215 M30
```

MANUALLY PROGRAMMING TOOL NOSE COMPENSATION FOR AN ANGLE

If you decided to manually calculate tool nose compensation for an angle on your part, It would be a hassle. Our suggestion is, if you want to save time and money, learn how to use TNC, it will help you in programming your part. This section briefly defines what you would need to calculate if you didn't use tool nose compensation.

MANUALLY CALCULATING COMPENSATION FOR "AN ANGLE" ON A PART

When you program a straight line in either X or Z axis, the tool tip touches the part at the same point where you touched your original tool offsets in X and Z. However, when you program a chamfer or an angle, the tip does not touch the part at those same points. Where the tip actually touches the part is dependent upon the degree of angle being cut and the size radius on the cutting tool your using. If you were to try and program your part without using any compensation you would see over and undercutting on your part and it could wind up being no good.

The following pages contain tables and illustrations on how to manually calculate tool nose compensation of an angle in order to program your part accurately.

TOOL NOSE RADIUS CALCULATION DIAGRAM

Machining a chamfer, when cutter compensation is NOT used on the control, requires that calculations must be made for the tool tip geometry for the programmed moves on your part angles.
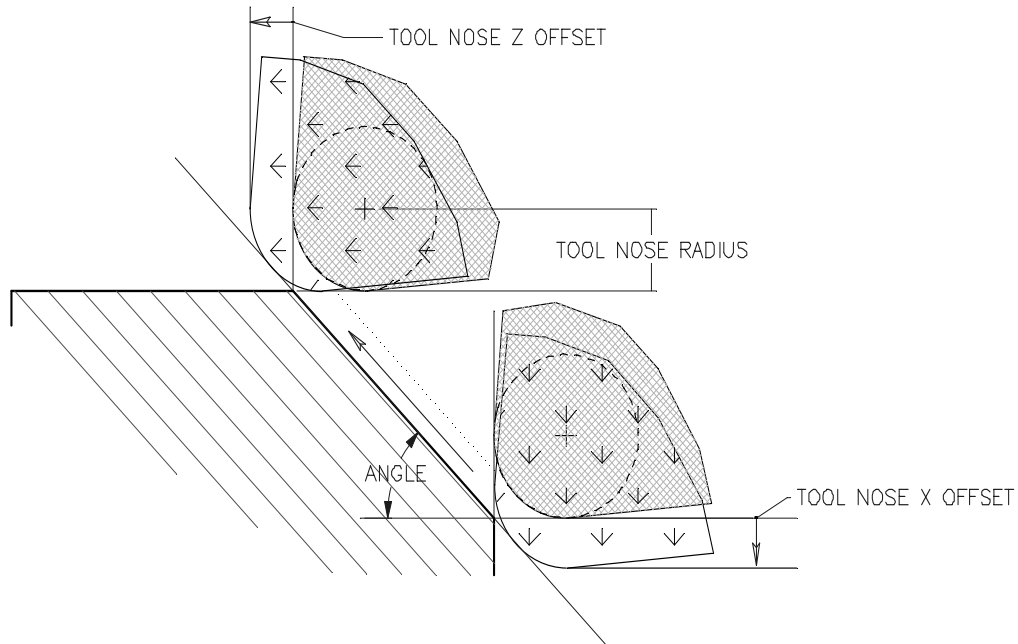
In the Lathe programming and operation manual there is a section for manually calculating tool nose radius compensation with various charts and illustrations for different angles for 1/32 and 1/64 insert radiuses.

ANGLE	X-OFFSET .015 TNR	Z-OFFSET .015 TNR	X-OFFSET .031 TNR	Z-OFFSET .031 TNR
15	.0072	.0136	.0146	.0271
30	.0132	.0114	.0264	.0229
45	.0184	.0092	.0366	.0183
60	.0228	.0066	.0458	.0132

MANUALLY ADDING RADIUS COMPENSATION FOR ANGLES MOVES EXAMPLE
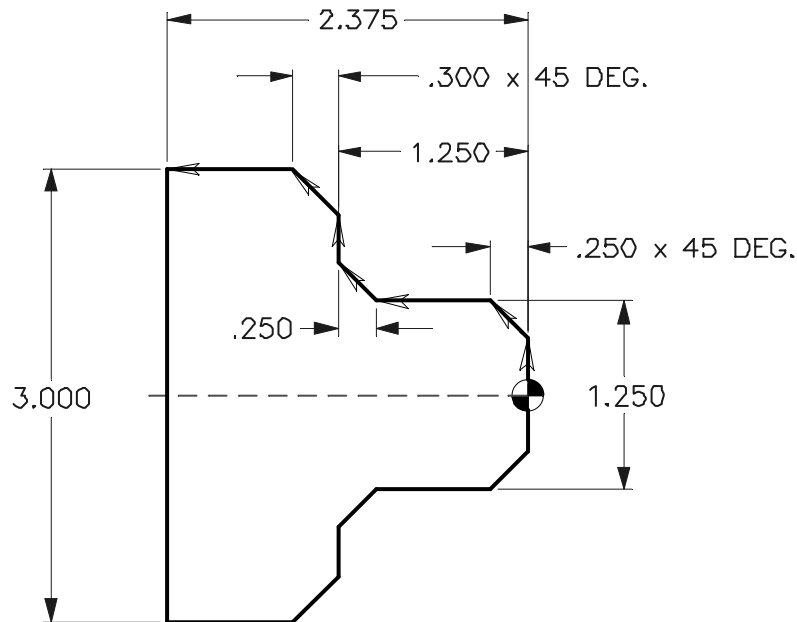
Program is manually adding in compensation for a finish pass using a tool with a .031 TNR

```
O00055 (LINEAR INTERPOLATION "WITHOUT" CUTTER COMP.)
N1 G53 G00 X0. Z0. T0
N2 T101 (O.D. TURNING TOOL .031 TNR)
N3 G97 S1450 M03
N4 G54 G00 X0.85 Z0.1 M08
N5 Z0.
N6 G01 X-.062 F0.01   (Face down end of part)
N7 G00 X.7134 Z0.02  (Rapid to start point of angle subtracting X comp. value)
N8 G01 Z0. F0.006   (Feed into face)
N9 X1.250 Z-.2683   (Feed up angle adding Z compensation amount)
N10 Z-1.0183  (Feed to angle adding Z compensation amount)
N11 X1.7134 Z-1.250  (Feed up angle subtracting X compensation amount)
N12 X2.3634  (Feed up to angle subtracting X compensation amount)
N13 X3.0  Z-1.5683  (Feed up angle adding Z compensation amount)
N14 Z-2.375  (Feed to finish to end in Z axis)
N15 G00 U0.01 Z1.0 M09
N16 G53 G00 X0. Z0. T0
N17 M30
```

TOOL NOSE COMPENSATION

OVERVIEW

Tool nose compensation is a feature that allows the user to adjust the programmed tool path on a lathe, in response to the tool nose radius at the end of a turning tool. The user can do this by entering specific G-code commands to offset and position for the tool nose radius value entered in the TOOL GEOMETRY offset display under RADIUS.

WHEN TO USE TOOL NOSE COMPENSATION

Tool nose compensation is used when the tool nose that you're cutting with has a radius. Tool nose compensation does not need to be used when the programmed cuts are solely along the X axis (diameters) or Z axis (faces). Cutter compensation comes into effect on angled cuts and radius cuts. The program path without cutter compensation is positioning the tool around the part geometry to the theoretical sharp tool nose point. But a perfectly sharp tool nose does not exist. What is there is a radius that is farther back from where the theoretical sharp tool nose tip would be. And so the tool, is taking off less material for what is programmed on the part radiuses and angles, and as a result will have an undersize or oversize condition on radiuses and angles.
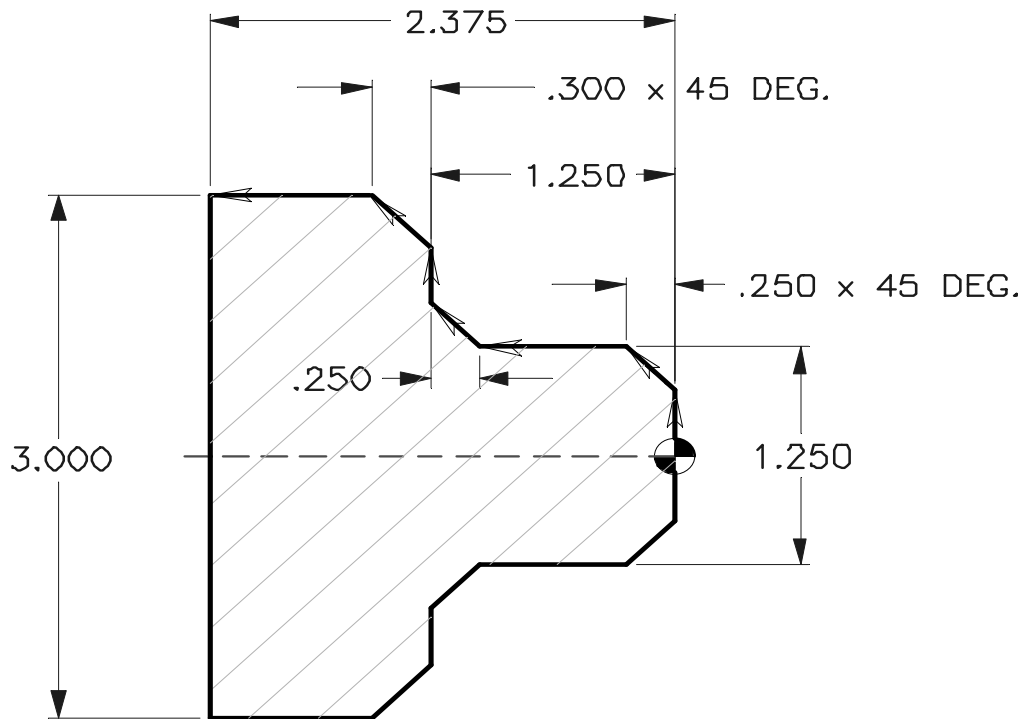
A Programmed part line not using cutter compensation showing cutting error

Note: The figure above shows that the programmed path coincides with the final part dimension, and the radiuses and angles are not quite right. Although parts don't have to be programmed using tool nose compensation, it's the preferred method because it's easier to program a part and problems are easier to detect and resolve.

TOOL NOSE COMPENSATION CONCEPTS

Tool nose compensation accomplishes it's task by reading ahead one or two blocks to determine how it must modify the current programmed block of code. This is done, so that the control can calculate in advance, how to position around radiuses and angles with a specified tool nose radius. This is referred to as BLOCK LOOK-A-HEAD or LOOK-A-HEAD PROCESSING.

When the control is first powered on or in the reset condition, tool nose compensation is not active. Tool nose compensation is turned on in a program by programming in a G41 or G42 command. When this command is executed, the control will look ahead in the program to determine where the first compensated move will be. The first move, when turning on cutter compensation, is generally a move from a non-compensated position to a compensated position and therefore this move is unusual. This first move is called the APPROACH move and is required when using tool nose compensation. Similarly, a DEPARTURE move is also required. In a departure move, the control will move from a compensated position to a non-compensated position. A departure move occurs when the tool nose compensation is cancelled with a G40 command or T0 command. Although approach and departure moves can be precisely planned, they're generally uncontrolled moves and the tool therefore, should not usually be in contact during these moves.

IMAGINARY TOOL TIP AND DIRECTION

Tool nose compensation calculates a compensated tool path for a tool that has a radius at the tool tip. This tool tip is known as the TOOL NOSE RADIUS (TNR). For a lathe it's not easy to determine where the edge around the tool nose radius is. The cutting edges are set when a tool is touched off with the edge of the radius in the X and Z axes to record the tool geometry offsets. The X and Z axes geometry offsets intersect at a point, called the IMAGINARY TOOL TIP, refer to the figure below, that also shows the tool tip direction. The tool tip direction is determined by a vector originating from the center of the tool radius and extending to the imaginary tool tip. The control can calculate where the center of the tool radius is by using the tool geometery X Z offsets, tool radius offsets, which side of the line the cutter is on left (G41) or right (G42), and the direction tool tip is pointing (1-9).

TIP DIRECTION IS 3

Tool nose radius
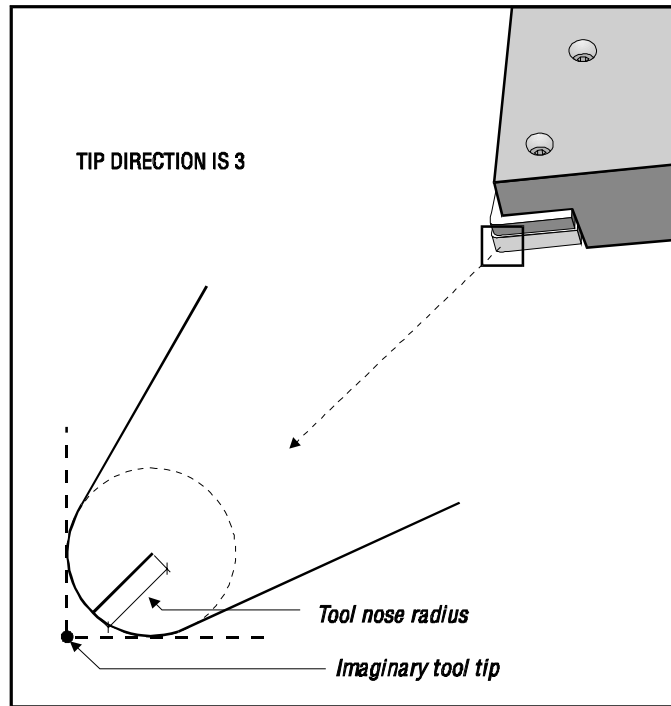
Imaginary tool tip

Imaginary Tool Tip

USING TOOL NOSE COMPENSATION

Tool nose compensation works by programming the TOOL PATH part line and defining what side of the line the cutter is on, right or left. The programmer will usually program the tool path to the current part print finish size. When tool nose compensation is used, the control will compensate for a tool's radius based on special instructions written into the program. Two G-code commands are used for tool nose compensation, G41 and G42. G41 commands the control to compensate for the tool nose radius positioning to the left of the programmed path, and G42 commands the control to compensate for the tool nose radius positioning to the right of the programmed path. The third command, G40, is used to cancel any tool nose compensation made by G41 and G42 .

Tool path and direction tools travel relative to workpiece

TIP = 3

G42 TNC Right

G41 TNC Left

TIP = 2
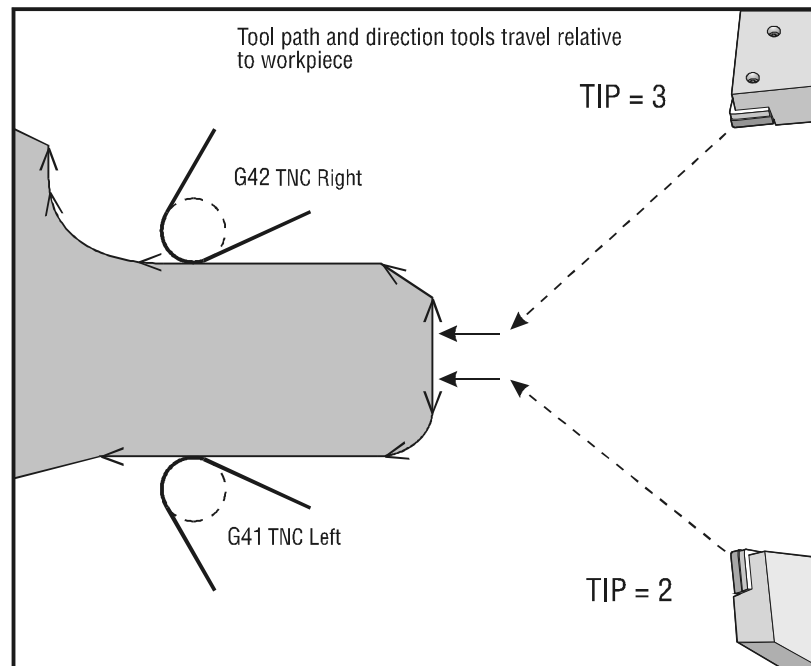
Tool path and shift direction.

Compensating the tool path on either the left or right is based on the direction of the tool movement relative to the part, and which side of the part it's on. When thinking about which side of the part the compensation will occur, imagine yourself standing on the line of the part to be cut, looking in the direction the cutter is traveling, what side of the line will the cutter pass you on, left or right. Programming G41 will compensate for the tool tip on the left and a G42 will compensate for the tool tip on the right. For a lathe, this means that normal O.D. turning, which is machining from the face of the part toward the chuck, will require a G42 cutter right of the program path, while normal I.D. turning will require a G41 cutter left of the program path.

Does it sound complicated? It's not if you follow the next seven steps when using tool nose compensation.

SEVEN STEPS FOR USING TOOL NOSE COMPENSATION

1. APPROACH MOVES

Program an approach move for each tool path that needs tool nose compensation and determine if G41 or G42 is to be used.

2. DEPARTURE MOVES

Be sure there is a departure move for each compensated tool path by using a G40 command to cancel tool nose compensation.

3. TOOL GEOMETRY OFFSETS AND TOOL WEAR OFFSETS

When setting up the tools for a part program, zero any Tool Geometery, Tool Wear, and Work Zero offsets that remain from an earlier job. Then you must touch off and enter the tool geometry (distance from machine home to part zero) length offsets for each tool being used.

4. TOOL NOSE RADIUS GEOMETRY

Select a standard insert (with a defined radius) that will be used for each tool that is using tool nose compensation. Enter the tool nose radius of each compensated tool in the TOOL GEOMETRY offset display under RADIUS.

5. TOOL TIP DIRECTION

Input the tool tip direction, in the TOOL GEOMETRY offset display under TIP, for each tool that is using tool nose compensation, G41 or G42.

6. TEST RUN COMPENSATION GEOMETRY

Run the program in graphics mode and correct any tool nose compensation geometry problems that may occur. A problem can be detected in two ways: either an alarm will be generated indicating compensation interference, or you will see the incorrect geometry generated and seen in graphics mode.

7. RUN AND INSPECT FIRST PART

Carefully cycle through the program to machine part. After running part, check and adjust wear offsets for the part to bring it within size.

Each of the above steps is described in detail in the following sections.

1. APPROACH MOVES

The first X or Z motion in the same line that contains a G41 or G42 is called the APPROACH move. The first move must be a linear move, that is a G01 or G00. At the start of an approach move the current position is not compensated. At the end of the approach move the machine position will be fully compensated.
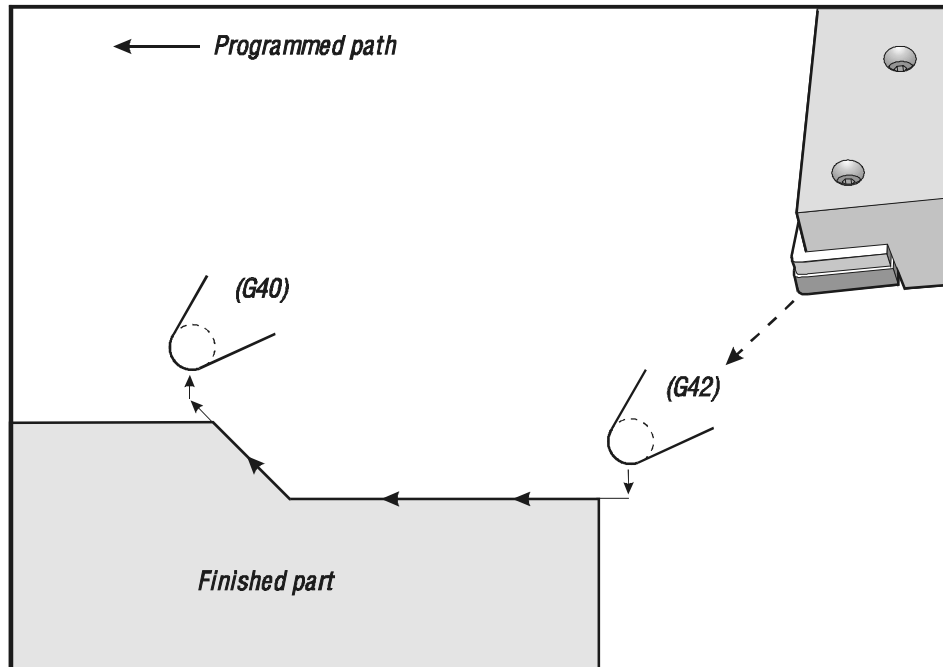
Approach and Departure moves.

2. DEPARTURE MOVES

Any line that contains a G40 will cancel tool nose compensation and is called the DEPARTURE move. The last move must be a linear move, that is a G01 or G00. At the start of a departure move, the current position is fully compensated. At the end of the departure move the machine position is not compensated.

Note: You cannot activate or cancel cutter compensation in a G02 or G03 circular move, it must be turned on or off in a linear Feed (G01) or Rapid (G00) straight line move.

G41   TOOL NOSE COMPENSATION LEFT

*X   Absolute X-axis approach move
*Z   Absolute Z-axis approach move
*U   Incremental X-axis approach move
*W   Incremental Z-axis approach move
*F   Feed rate
   * indicates optional

G41 will select tool nose compensation left; that is, the tool is programmed to the left of a tool path part line, to compensate for the tool tip radius. A tool offset must be selected with a Tnnxx code, where xx corresponds to the offsets that are to be used for the nn tool turret location.
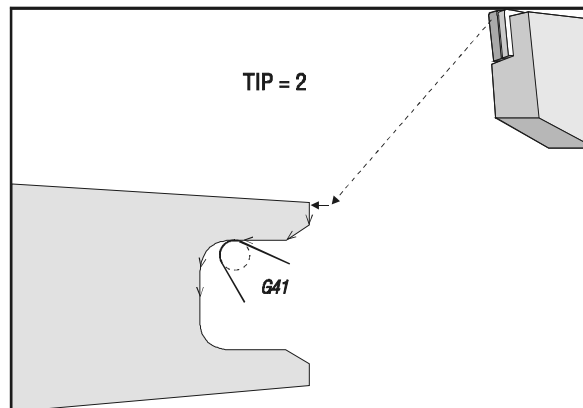
G41 TNC Left of the programmed part line.

G42 TOOL NOSE COMPENSATION RIGHT

*X Absolute X-axis approach move
*Z Absolute Z-axis approach move
*U Incremental X-axis approach move
*W Incremental Z-axis approach move
*F Feed rate
* indicates optional

G42 will select tool nose compensation right; that is, the tool is programmed to the rightof a tool path part line, to compensate for the tool tip radius. A tool offset must be selected with a Tnnxx code, where xx corresponds to the offsets that are to be used for the nn tool turret location.

TIP = 3

G42
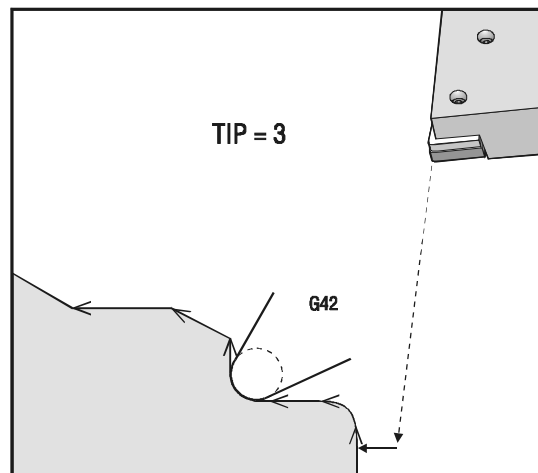
G42 TNC Right of the programmed part line.

G40 TOOL NOSE COMPENSATION CANCEL

*X Absolute X-axis departure move
*Z Absolute Z-axis departure move
*U Incremental X-axis departure move
*W Incremental Z-axis departure move
*I   X axis intersection vector direction, (radius)
*K Z axis intersection vector direction
*F Feed rate
* indicates optional

G40 cancels G41 or G42. Programming Txx00 or just T0 will also cancel tool nose compensation. You must always cancel tool nose compensation before the end of a program.

The G40 departure move, off of the part, usually does not correspond with any point on the part. In many cases overcutting or undercutting could occur if tool is still on the part when you cancel cutter compensation.
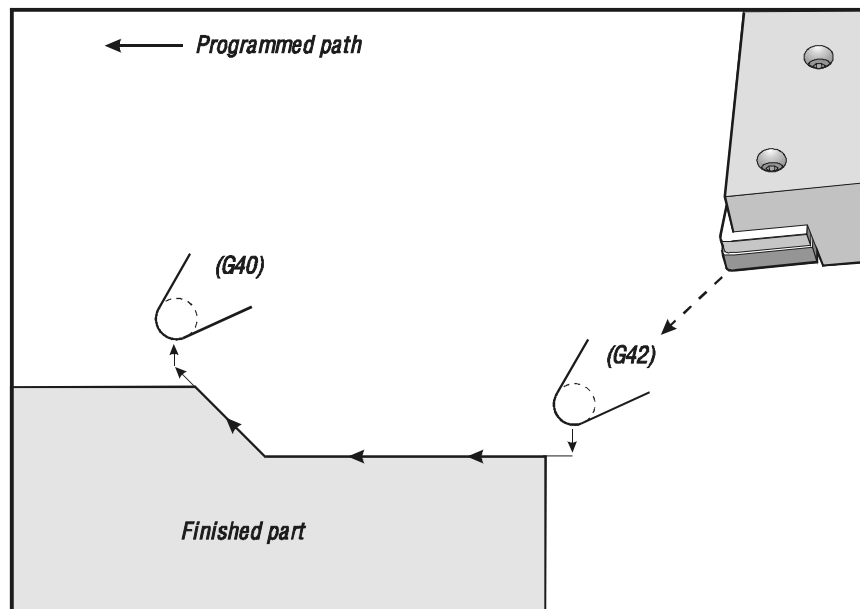
G40 Cancel TNC of the programmed part line.

NOTE: Some conditions of a cancellation of tool nose compensation geometries will result in over or undercutting of the part. The programmer can control this by including an I and K in the G40 cancellation block. The I and K address codes in a G40 block define a vector that the control will use in determining the compensated target position of the previous block. The vector is usually aligned with an edge or wall of the completed part. The Figure below shows how I and K can correct undesired cutting in a departure move.
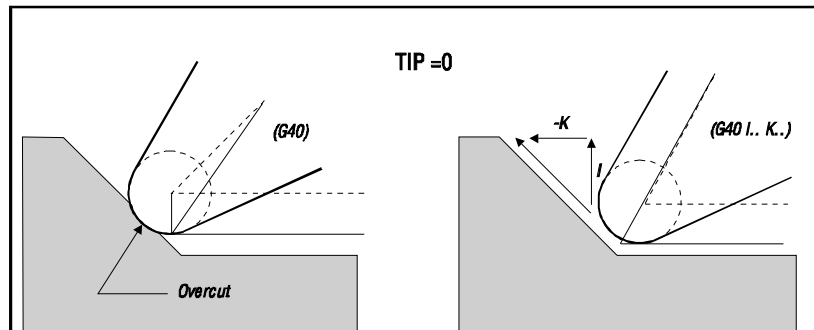
Figure showing Use of I and K in a G40 block.

When address codes I and K are used in a G40 departure move, the control will use these values as an intersection vector for the end point of the last completely compensated motion stroke. The illustration below shows where I and K lie in relation to the departure stroke. Usually I and K lie along a face of a machined part.
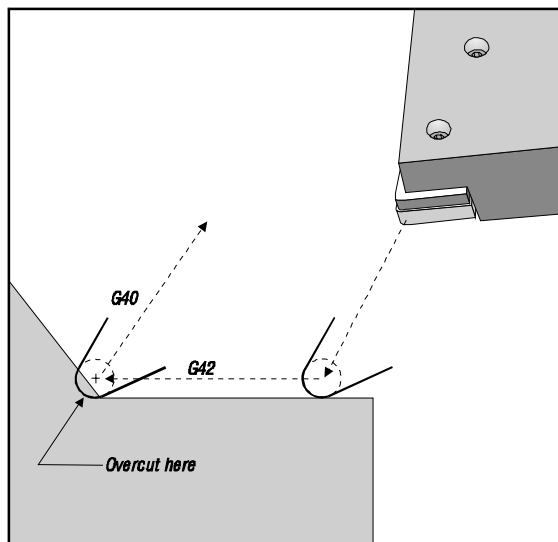
G40 without I and K
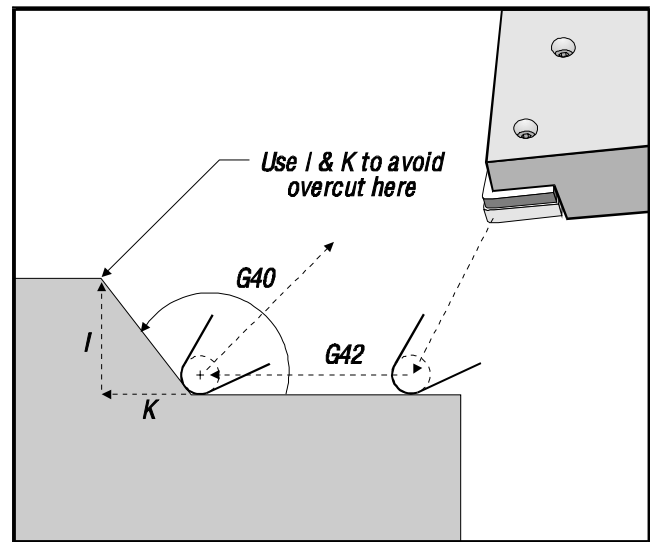
G40 with I and K

3. TOOL GEOMETRY AND TOOL WEAR OFFSETS

The tool length geometry offsets, for each tool, that use tool nose compensation are set up in the same manner as tools not using radius compensation. Refer to the "Setup Procedures" section of your operation manual for details on touching off tools and recording tool length geometries.

When a part is set up using new tools, the tool wear offsets should be zero. You can zero all wear offsets at once by going to the TOOL WEAR offset display and pressing ORIGIN, then Y to accept zeroing all the offsets in that display (any Lathe control 3.0 and above).

Small adjustments to the radius offset during production runs should be placed in the wear offset page. This allows the operator to easily track the wear for a given tool. Most often, a tool will exhibit uneven wear. This occurs when particularly heavy cuts are done on one edge of the tool. In most cases it is desirable to adjust that axis that is affected X or Z in the TOOL WEAR offsets rather than the RADIUS TOOL WEAR offsets. When adjusting the X or Z tool GEOMETRY/WEAR offsets, the operator is compensating for uneven tool nose wear. The tool geometry X or Z TOOL WEAR offsets will shift ALL dimensions for that single axis. If the part dimensions being machined, need the operator to compensate for the tool wear in both axes. The tool is wearing evenly around the radius, and the dimensional changes on the part in the X and Z axes are the same and suggests that the RADIUS wear offsets should be increased. But if the wear is on one axis only, then the length geometry wear should be adjusted. One can determine which wear offset to adjust by checking several X and Z dimensions on a finished part.

When replacing a worn tool with a new one, that wear offset row should be cleared by cursoring to each offset item in that row and pressing 0 then F1 to clear that offset.

4. TOOL NOSE RADIUS GEOMETRY AND WEAR OFFSETS

Each turning tool that uses tool nose compensation requires a TOOL NOSE RADIUS. The tool nose radius specifies how much the control is to compensate for a given tool. It is determined by the geometry of the tool tip. If standard inserts are being used for the tool, then the tool nose radius is simply the tool tip radius of the insert.

Associated with each tool on the geometry offsets page is a TOOL NOSE RADIUS OFFSET. The column labeled RADIUS is where the tool nose radius of each tool is placed. If the value of any tool nose radius offset is set to zero, no compensation will be generated for that tool.

Associated with each radius offset is a RADIUS WEAR OFFSET. It is located on the wear offset page. The control adds the wear offset to the radius offset to obtain an effective radius that will be used for generating compensated values.

Small adjustments to the radius offset during production runs should be placed in the wear offset page. This allows the operator to easily track the wear for a given tool. As a tool is used, the insert will generally wear so that there is a larger radius at the end of the tool. This should place positive values in the wear column. When replacing a worn tool with a new one, the wear offset should be cleared to zero.

When replacing a worn tool with a new one, that wear offset row should be cleared by cursoring to each wear offset item in that row and pressing 0 and F1 to clear that offset.

5. TOOL TIP DIRECTION

The tool tip direction of each tool is coded as a single integer from 0 to 9. The tip direction code is found next to the radius offset on the geometry offsets page. It is recommended that a tip direction be specified for all tools using tool nose compensation. Figure 14 is a summary of the tip coding scheme along with cutter orientation examples.

Note that the tip indicates to the setup person how the programmer intends the tool offset geometry to be measured. For instance, if the setup sheet shows tip direction 8, the programmer intends the tool geometry to be at the edge of and on the centerline of the tool insert.

Note: It is important to remember that tool nose compensation values are in tearms of radius rather than diameter. This is important in blocks where tool nose compensation is cancelled. If the incremental distance of a departure move in a compensated path is not twice the radius of the cutting tool's radius, overcutting will occur. Always remember that programmed paths are in terms of diameter and allow for twice the tool radius on a departure move.

TOOL TIP CODE #	TOOL TIP DIRECTION ORIENTATION	TOOL TIP POINT DIRECTION LOCATION
0		Zero (0) indicates no specified direction. It is usually not used when tool nose compensation is desired.
1		TOOL TIP OFF IN THE X+,Z+ DIRECTION
2		TOOL TIP OFF IN THE X+,Z- DIRECTION
3		TOOL TIP OFF IN THE X-,Z- DIRECTION
4		TOOL TIP OFF IN THE X-,Z+ DIRECTION
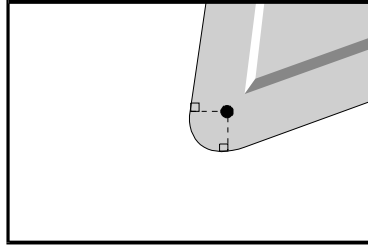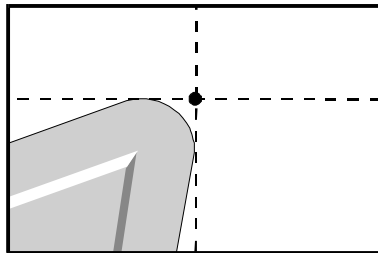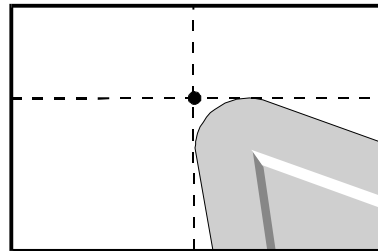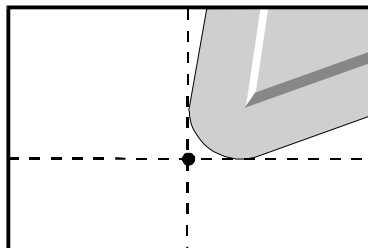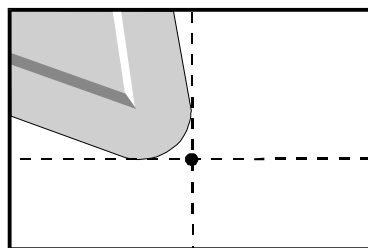
TOOL TIP CODE #	TOOL TIP DIRECTION ORIENTATION	TOOL TIP POINT DIRECTION LOCATION
5		TOOL EDGE Z+ DIRECTION
6		TOOL EDGE X+ DIRECTION
7		TOOL EDGE Z- DIRECTION
8		TOOL EDGE X- DIRECTION
9		SAME AS TIP 0
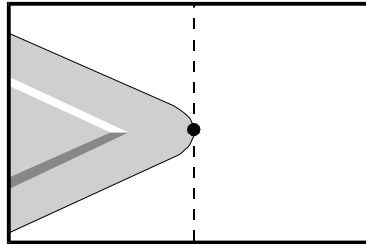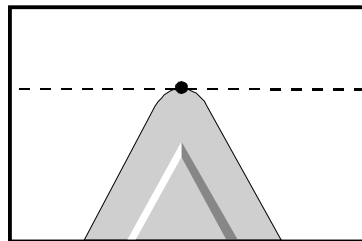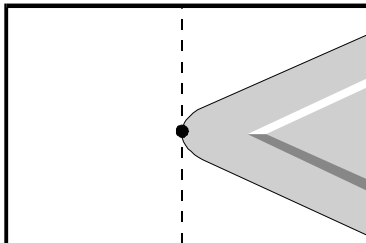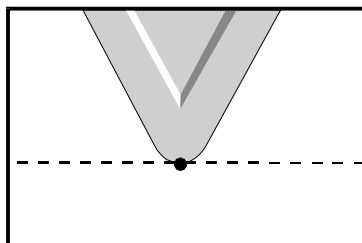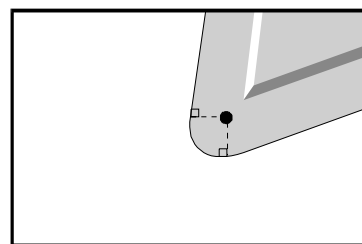
6. TEST RUN COMPENSATION GEOMETRY

Debug the program in graphics mode and correct any tool nose compensation geometry problems that may occur. A problem can be detected in two ways: either an alarm will be generated indicating compensation interference, or the incorrect geometry will be generated and seen in graphics mode.

7. RUN AND INSPECT FIRST PART

Adjust wear offsets for the setup part.

ADVANTAGES OF CUTTER COMPENSATION

1. The mathematical computations to define a tool path for angles and arcs is greatly simplified.

SOME RESTRICTIONS WITH CUTTER COMPENSATION

1. A cutter compensation command (G41/G42) must be on the same block with an X and/or Z linear move when *ramping onto* the part.

2. Cutter compensation cancel command (G40) must be on the same block with an X and/or Z linear move when *ramping off of* the part.

3. You cannot turn on or off cutter compensation in a G02 or G03 circular move, it must be in a linear Feed (G01) or Rapid (G00) straight line move.

WHEN ACTIVATING CUTTER COMPENSATION

1. Define a clearance distance without cutter compensation to a start point in "**X**" and "**Z`**" axis double the tool radius off the part before you initiate cutter compensation.

2. Make an **X** and/or **Z** axis move with a **G41** or **G42** call-out on the same line, with a tool tip radius and tip direction entered in the TOOL GEOMETRY offset display register under the RADIUS and TIP column for that tool being used.

WHEN DEACTIVATING CUTTER COMPENSATION

1. Cancel of cutter compensation may be a one or two axis move.

2. Select a clearance point, that's at least double the tool radius off the part in the "X" and/or "Z" axis.

3. "DO NOT" cancel cutter compensation on any line that is still cutting the part.

AN EXAMPLE OF INCORRECTLY TURNING OFF CUTTER COMPENSATION

It is important to remember that tool nose compensation values are in terms of radius rather than diameter. This is important in blocks where tool nose compensation is cancelled. If the incremental distance of a departure move in a compensated path is not twice the radius of the cutting tool nose radius, overcutting may occur. Always remember that programmed paths are in terms of diameter and allow for twice the tool radius on departure moves. The Q block of canned cycles (G71, G72 or G73) that require a PQ sequence can often be a departure move.

The following example illustrates how incorrect programming will result in overcutting.

	(X)	(Z)	(RADIUS)	(TAPER)	TIP
SETTING 33 is FANUC:					
TOOL GEOMETRY Offset 8:	-7.6543	-8.7654	0.0156	0.	2

```
O00072
(An example of incorrectly canceling cutter compensation) ;
N1 G28
N2 T808 ;  (BORING BAR .015 TNR)
N3 G97 S2400 M03 ;
N4 G54 G00 X0.49 Z0.1 ;
N5 G41 G01 X0.5156 F0.02 ;
N6 Z-0.05 F0.004 ;
N7 X0.3438  Z-0.25 ;
N8 Z-0.5 ;
N9 X0.33 ;          (NOTE: Move is less than .031, which is the value required to avoid cut-in
                    with a departure move before TNC is cancelled.)
N10 G40  G00  X0.25 ;
N11 Z0.05 ;
N12 G28 ;
N13 M30 ;
```
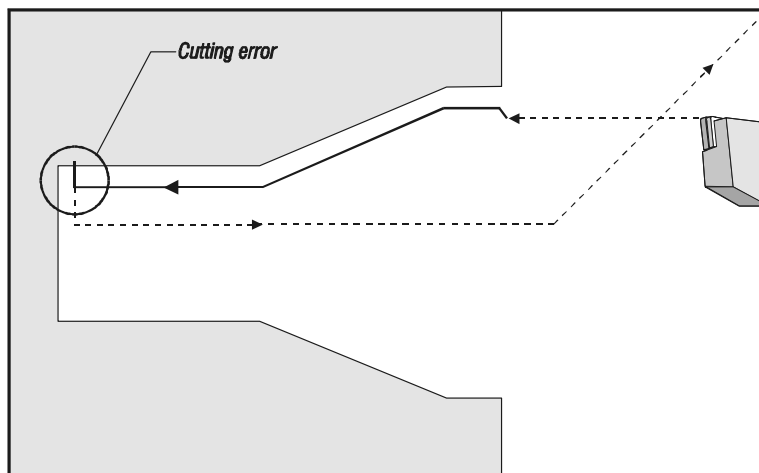

An invalid program depature move using TNC

THE FOLLOWING CANNED CYCLES WILL IGNORE TOOL NOSE COMPENSATION

It is recommended that tool nose compensation be cancelled prior to executing any of these canned cycles.

G74   END FACE GROOVING CYCLE, PECK DRILLING
G75   O.D./I.D. GROOVING CYCLE
G76   THREAD CUTTING CYCLE, MULTIPLE PASS
G92   THREAD CUTTING CYCLE, MODAL

THE FOLLOWING CANNED CYCLES PERFORM THEIR OWN SPECIAL CUTTER COMPENSATION

G90   O.D./I.D. MODAL TURNING CYCLE

G90 performs tool nose compensation independently. Since G90 executes only linear moves, some of the compensation overhead can be eliminated to produce more accurate results on tapered cuts.

G94   END FACE CUTTING CYCLE

G94 performs tool nose compensation independently. Since G94 executes only linear moves, some of the compensation overhead can be eliminated to produce more accurate results on tapered cuts.

EXAMPLE PROGRAMS USING TOOL NOSE COMPENSATION

This section is comprised of example programs that use tool nose compensation.

EXAMPLE USING TOOL NOSE COMPENSATION

This example illustrates tool nose compensation using standard interpolation modes G01/G02/G03.
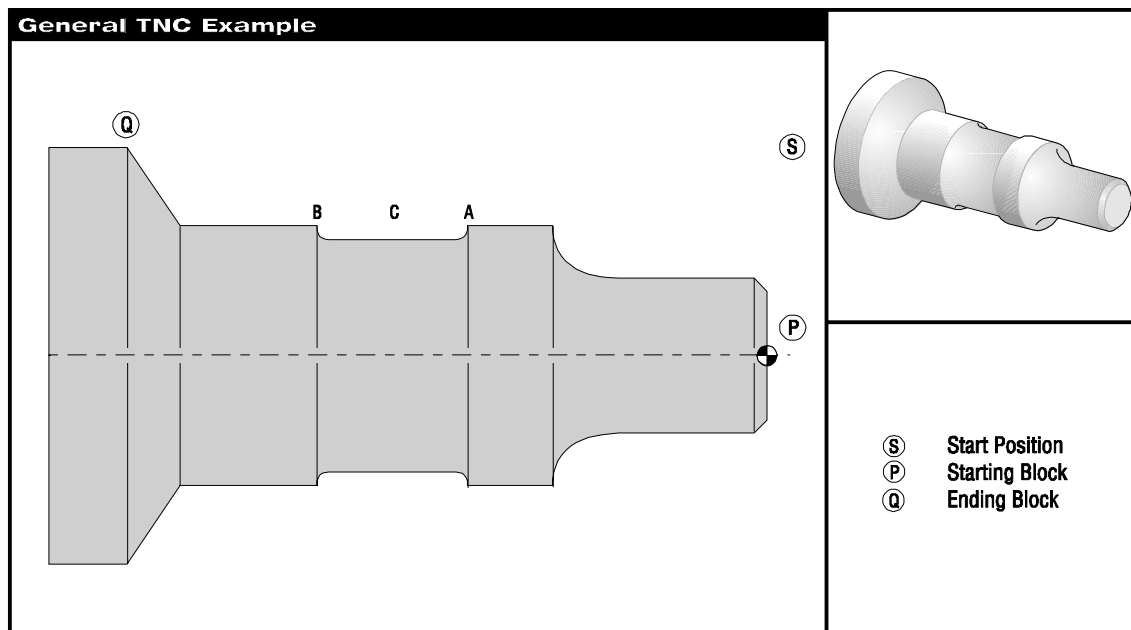
General TNC Example

PREPARATION

TOOL DESCRIPTION
TOOL 1 - O.D. Rough Tool with a 55 Degree Insert and a .0312 radius.
TOOL 2 - O.D. Finish Tool with a 55 Degree Insert and a .0312 radius.
TOOL 3 - .250 Wide grooving tool with .016 corner radius.

SETTING 33 IS FANUC

TOOLS	OFFSETS	X	Z	RADIUS	TIP
T1	01	-7.9650	-9.8470	.0312	3
T2	02	-7.9010	-9.8450	.0312	3
T3	03	-7.8400	-9.5880	.016	4

PROGRAM EXAMPLE   **(DESCRIPTION)**
%
O00075
(GENERAL TNC EXAMPLE)
(T101 - O.D. Rough Tool)
(55 Deg. x .031 TNR, TIP Direction 3)
(T202 - O.D. Finish Tool)
(55 Deg. x .0312 TNR TIP Direction 3)
(T303 - O.D. Groove Tool)
(.250 wide x .016 Radius Corner Radius)
(TIP Direction 4)

N101 (ROUGH O.D.)
N102 G28   *(Return to Machine Home reference point)*
N103 T101 (55 Deg. O.D. TOOL x .0312 TNR)*(Select tool 1, offset 1)*
N104 G50 S2500
N105 G97 S591 M03
N106 G54 G00 X2.1 Z0.1 M08   *(Rapid to start point)*
N107 G96 S325
N108 Z0.005
N109 G01 X-0.063 F0.01
N110 G00 X2.1 Z0.1
N111 G71 P112 Q124 U0.02 W0.005 D0.1 F0.012 *(Rough P to Q using G71 and TNC)*
                                        (Define part path PQ sequence)
N112 **G42** G00 X0.55 Z0.1   *(P) (G71 Type II, TNC approach)*
N113 G01 Z0. F0.004
N114 X0.65
N115 X0.75 Z-0.05
N116 Z-0.75
N117 G02 X1.25 Z-1. R0.25
N118 G01 Z-1.5
N119 Z-1.72 X1. F0.006
N120 G01 Z-2.5
N121 G02 X1.25 Z-2.625 R0.125
N122 G01 Z-3.5 F0.004
N123 X2. Z-3.75 F0.008
N124 **G40** G00 X2.1   *(Cancel TNC Departure move)*
N125 G97 S591
N126 M09
N127 G28   *(Return to Machine Home reference point)*
N128 M01

```
N201 (FINISH O.D.)
N202 G28                              (Return to Machine Home reference point)
N203 T202 (55 Deg. O.D. TOOL x .0312 TNR) (Select tool 2, offset 2)
N204 G50 S2500
N205 G97 S1655 M03
N206 G54 G00 X0.75 Z0.1 M08          (Rapid to point S)
N207 G96 S325
N208 Z0.
N209 G01 X-0.063 F0.01
N210 G00 X2.1 Z0.1
N211 G70 P112 Q124                   (Finish turn P to Q using G70)
N212 G97 S591
N213 M09
N214 G28                             (Return to Machine Home reference point)
N215 M01

N301 (O.D. GROOVE)
N302 G28                             (Return to Machine Home reference point)
N303 T303 (.250 WIDE x .016R GROOVE TOOL)    (Select tool 3, offset 3)
N304 G50 S2500
N305 G97 S920 M03
N306 G54 G00 X1.35 Z0.1 M08          (Rapid to start point)
N307 G96 S325
N308 G00 Z-1.8                       (Rapid to point A)
N309 G41 G01 X1.25 F0.02             (TNC approch move)
N310 X1. F0.006                      (Move to point B with TNC approch move)
N311 Z-1.625 F0.003
N312 G03 X1.25 Z-1.5 R0.125          (Move to point A)
N313 G40 G00 X1.35                   (Cancel TNC with a departure move)
N314 G97 S920
N315 M09
N316 G28                             (Return to Machine Home reference point)
N317 M30
%
```

Note: Tip Direction 4, for tool 3 using cutter compensation, since the corner that is cutting is pointing down in the X- Z+ direction .

MACHINE CYCLES FOR TURNING AND GROOVING

A machine cycles is used to simplify the programming of a part. machine cycles are defined for the most common machining operations. They can be divided into two types. There are machining cycles for turning and grooving. And there are canned cycles for drilling, tapping and boring. These cycles can be either single block cycles or modal cycles.

A modal cycle remains in effect after they are defined and are executed for each positioning of an axes. Once a cycle is defined, that operation is performed at every X-Z position subsequently listed in a block.

The machining cycles G70, G71, G72, and G73 cannot be in DNC while running a program, since these G-codes require the control to look ahead.

These G-codes, (G71, G72, and G73) do not force feed moves within the PQ block. To prevent rapid motion from occurring when feed moves are intended, a G01 should be defined, when needed, near the beginning of the P block.The feed rate that it'll do throughout the cycle is the one listed on the canned cycle line. And the G70 will perform the feed rates that are listed on the lines between the PQ blocks.

Canned cycles G70, G71, G72 and G73 should not be executed after a tool nose compensation command (G41 or G42). If tool nose compensation is desired, it should be defined in the P block called up by the canned cycle.

The following is a list of the canned cycles that can be used for turning and grooving on HAAS lathe controls.

G70	Finishing Cycle	Group 00
G71	O.D./I.D. Stock Removal Cycle	Group 00
G72	End Face Stock Removal Cycle	Group 00
G73	Irregular Path Stock Removal Cycle	Group 00
G74	End Face Grooving Cycle, or Peck Drilling	Group 00
G75	O.D./I.D. Grooving Cycle	Group 00
G76	O.D./I.D. Thread Cutting Cycle, Multiple Pass	Group 00
G90	Turning Cycle, Modal	Group 01
G92	Thread Cutting Cycle, Modal	Group 01
G94	End Face Cutting Cycle, Modal	Group 01

G71   O.D./I.D. STOCK REMOVAL CYCLE

P   Starting block number of part path to machine

Q   Ending block number of part path to machine

U* Finish stock remaining with direction (+or -), X-axis *diameter* value

W* Finish stock remaining with direction (+or -), Z-axis value

I*   Last pass amount with direction (+or -), X-axis *radius* value

K* Last pass amount with direction (+or -), Z-axis value

D* Depth of cut stock removal each pass, positive *radius* value (Setting 72)

F   Roughing passes feed rate throughout this cycle

R1*YASNAC type II roughing (only if setting 33 is on Yasnac)

S** Spindle speed in this cycle

T** Tool and offset in this cycle

   * Indicates optional

  ** Rarely defined in a G71 line
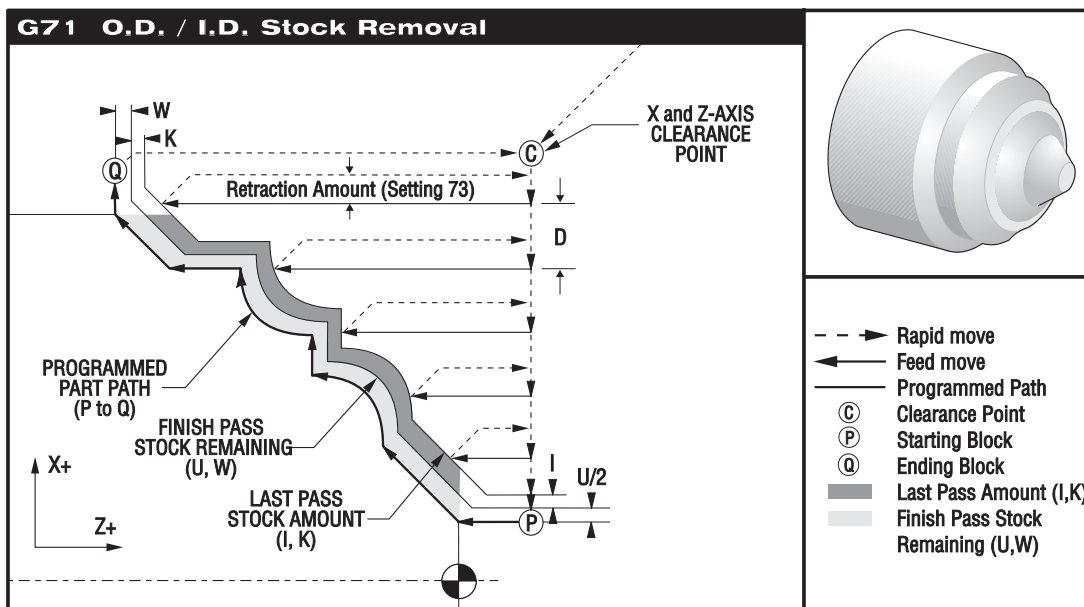
This canned cycle will rough out material on a part defining the finished part shape. All a programmer needs to do is to define the shape of a part by programming the finished tool path and then submitting the part path definition to the G71 call by means of a PQ block designation. Any feeds, spindle speed, or tool offset commands on the blocks defining the path are ignored by the G71 call. Any F, S or T commands on the G71 line, or the active ones at that time of the G71 call, are used throughout the G71 roughing cycle. A call to the same PQ block using a G70 will finish the part path with the programmed feeds, speed and offsets that are defined within the PQ block definition.

There are two ways to define the part geometry for machining a path using a G71 command. The first type is a TYPE I geometry path and is when the X-axis of the

programmed path does not change direction. This type of path is called monotonic. The second type is a TYPE II geometry path and allows the X-axis to change direction. For both types the programmed path for the Z-axis, must be monotonic, that is it cannot change direction in the Z axis. For FANUC, the Type I path is selected by having only an X-axis motion command in the N block that's specified by the P in the G71 command line. For FANUC, the Type II path is selected by having both an X-axis and Z-axis motion command in the N block that's specified by the P in the G71 command line, then TYPE II roughing is recognized. When in the YASNAC mode, Type II roughing is selected by including an R1 on the G71 command line.

NOTE: A part using Tool Nose Compensation with a G71 will usually be a Type I path. When using TNC it is very difficult to have a Type II path. This is because for the machines current compensation methods, you can only compensate for a tool tip in one direction. Because of this, the Type II geometry using TNC for a part program may not be accurate. This is because the tip direction would need to change to another tip direction during the cycle. So to be able to use TNC for a Type II part program you may need to modify the program for this TNC situation by editing your program geometry to bring this condition within size.

G71 consists of a roughing phase and a finishing phase. The roughing and finishing phase are handled slightly differently for types I and types II. Generally the roughing phase consists of repeated passes along the Z-axis at the specified G71 command line feed rate. The finishing phase consists of a pass along the programmed tool path to remove excess material left by the roughing phase, with perhaps a finishing tool to finish machine using a G70 command block that will use the programmed feeds, speeds and offsets defined within the PQ block definition. The final motion in either type I or Type II is a return to the starting clearance point "C".

The starting clearance point "C" is the position of the tool at the time of the G71 is called. This point is usually above the outside diameter or below the inside diameter and in front of that area you want to rough machine away using this G71 cycle .

G71 TYPE I ROUGHING DETAILS

When Type I is specified by the programmer it is assumed that the tool path is monotonic in the X-axis. Prior to any roughing motion the tool path designated by PQ is checked for monotonicity and G code compliance. An alarm is generated if a problem is found.

Roughing begins by advancing from the start position S and moving to the first roughing pass. All roughing passes start and end at the Z clearance plane. Each roughing pass X-axis location is determined by applying the value specified in D to the current X location. The direction that D is applied is determined by the signs of U and W. The nature of the movement along the Z clearance plane for each roughing pass is determined by the G code in block P. If block P contains a G00 code, then movement along the Z clearance plane is a rapid mode. If block P contains a G01 then movement will be at the G71 feed rate. Roughing continues until the X-axis position in block P is exceeded.

Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material, at a 45 degree angle by the distance specified in setting 73. The tool then moves in rapid mode to the Z-axis clearance plane.

When roughing is completed the tool is moved along the tool path to clean up the rough cut. If I and K are specified an additional rough finish cut parallel to the tool path is performed.

G71 O.D. TYPE I ROUGHING USING TNC WITH 1 TOOL

G71 Type I O.D. with TNC Stock Removal Example

Setup information
Setting 33: FANUC

Tool 1 - Roughing with an
80 Deg. x .031 radius insert

Tool	Offsets	Radius	Tip
1	01	.031	3
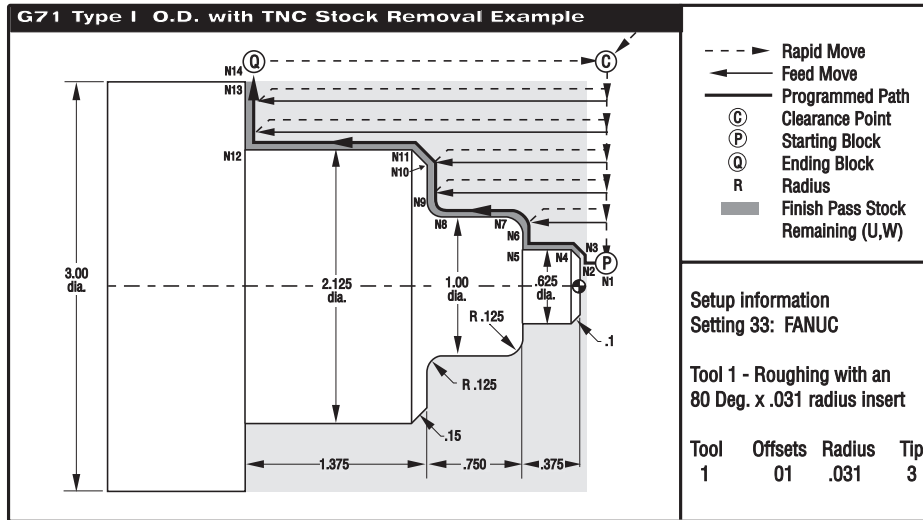
O00082
N1 G28 (G71 ROUGHING O.D.)
N2 T101 (O.D. TOOL x .032 TNR)
N3 G50 S3000
N4 G97 S450 M03
N5 G54 G00 X3.1 Z0.1 M08 (Rapid to a start point above part stock)
N6 G96 S370
N7 Z0.005   (Position to face part)
N8 G01 X-0.062 F0.01 (Face down part adding in compensation for the tool radius)
N9 G00 X3.1 Z0.1
N10G71 P11 Q24 U0.01 W0.005 D0.125 F0.012 (G71 Roughing Cycle with TNC)
N11 G42 G00 X0.325 (**P**) (G42 with the start of geometry with the P number in G71)
N12 G01 Z0. F0.01
N13 X0.425
N14 X0.625 Z-0.1 F0.005
N15 Z-0.375
N16 X0.75
N17 G03 X1. Z-0.5 R.125
N18 G01 Z-1.
N19 G02 X1.25 Z-1.125 R.125
N20 G01 X1.825
N21 X2.125 Z-1.275
N22 Z-2.5
N23 X3.
N24 G40 G00 X3.1 (**Q**)   (End of geometry defined by Q number in G71 and a G40)
N25 G97 S450 M09
N26 G28   (Return to Machine Zero)
N27 M01   (Optional Stop)

G70 FINISHING CYCLE

P   Starting block number of the part geometry to machine
Q   Ending block number of the part geometry to machine

The G70 Finishing cycle can be used to finish cut paths that are defined and roughed out with stock removal cycles G71, G72 and G73.

The G70 requires that a beginning block number (P code) and an ending block number (Q code) be specified for the machine code that defines the part geometry to be machined.

The G70 cycle is usually used after a G71, G72 or G73 has been performed using the blocks specified by P and Q. All codes in the block defined by P and Q are executed. Any F, S or T codes between the P and Q block are effective. The PQ sequence is searched for in the current program starting from the beginning of the program. The current machine position is saved and remembered as the start position. Then the block starting at P is executed. Processing continues in a normal fashion with blocks following P until a block is found and executed that contains an N code that matches the Q code in the G70 calling block. After execution of the Q block, a rapid (G00) is executed returning the machine to the start position that was saved earlier during G70 initialization. The program then returns to the block following the G70 call.

EXAMPLE: G70 Finishing Cycle

A traditional calling sequence using G70 appears below.

Rough cycle feedrate to rough out part between N10 and N80 for the G71 roughing cycle will be F.012 which is defined in the G71 block.

Finish feed rate for the G70 finish cycle will be whatever is defined between N10 and N80.

```
G71 P10 Q80 U.01 W.005 D.08 F.012  (Roughing cycle feedrate will be F.012
                                    which is defined in this block.)
N10 G00    (machine code that defines part path geometry)
N20 G01....   "          "            "         F.005
N30 ....      "          "            "
N40 G03....   "          "            "
N50 G01....   "          "            "
N60 G02....   "          "            "         F.003
N70 G01....   "          "            "         F.008
N80 ....      "          "            "
G70 P10 Q80(Finish cycle feedrate is defined on lines between N10 to N80).
```

This example demonstrates tool nose compensation with the G70 finishing cycle. The basic shape has been roughed out using a G71 roughing cycle.

G71 & G70 O.D. TYPE I ROUGHING AND FINISHING USING TNC WITH 1 TOOL

G71 & G70 Type I O.D. with TNC Stock Removal Example

Legend:
- - - → Rapid Move
← Feed Move
— Programmed Path
Ⓒ Clearance Point
Ⓟ Starting Block
Ⓠ Ending Block
R Radius
▬ Finish Pass Stock Remaining (U,W)

Setup information
Setting 33: FANUC

Tool 1 - Rough and Finish
with an 80 Deg. x .031 radius
insert

Tool	Offsets	Radius	Tip
1	01	.031	3
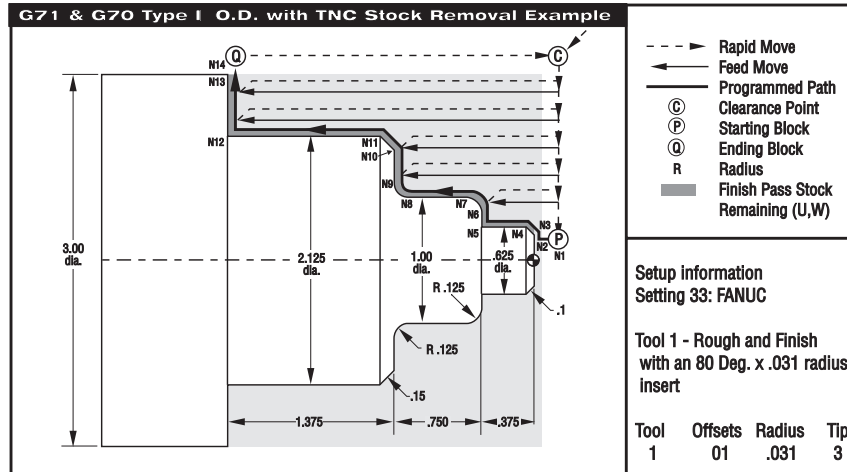
O00084
G28 (G71 ROUGHING O.D.)
T101 (O.D. TOOL x .032 TNR)
G50 S3000
G97 S450 M03
G54 G00 X3.1 Z0.1 M08   (Rapid to start point above part stock)
G96 S370
Z0.   (Position to Z0 to face part)
G01 X-0.062 F0.01   (Face down part adding in compensation for the tool radius)
G00 X3.1 Z0.1
G71 P10 Q20 U0.01 W0.005 D0.125 F0.012   (G71 Roughing Cycle with TNC)
N10 G42 G00 X0.325 (**P**) (Start of geometry defined by P number in G71 and a G42)
G01 Z0. F0.01
X0.425
X.625 Z-0.1 F0.005
Z-0.375
X0.75
G03 X1. Z-0.5 R0.125
G01 Z-1.
G02 X1.25 Z-1.125 R0.125
G01 X1.825
X2.125 Z-1.275
Z-2.5
X3.
N20 G40 G00 X3.1 (**Q**) (End of geometry defined by Q number in G71 and a G40)
G70 P10 Q20   (G70 Finishing O.D.)
G97 S450 M09
G28   (Return to Machine Zero)
M30   (End of Program)

G71 & G70 O.D. TYPE I ROUGHING AND FINISHING USING TNC WITH 2 TOOLS

```
O00085
N1 G28 (G71 ROUGHING O.D.)
N2 T101              (Tool 1 Offset 1)
N3 G50 S3000
N4 G97 S450 M03
N5 G54 G00 X3.1 Z0.1 M08      (Rapid to start point above part stock)
N6 G96 S370
N7 Z0.005                     (Position to face part)
N8 G01 X-0.062 F0.01   (Face down part adding in compensation for the tool radius)
N9 G00 X3.1 Z0.1
N10 G71 P11 Q24 U0.01 W0.005 D0.125 F0.012    (G71 Roughing Cycle with TNC)
N11 G42 G00 X0.325 (P) (Start of geometry defined by P number in G71 and a G42)
N12 G01 Z0. F0.01
N13 X0.425
N14 X.625 Z-0.1 F0.005
N15 Z-0.375
N16 X0.75
N17 G03 X1. Z-0.5 R0.125
N18 G01 Z-1. F0.003
N19 G02 X1.25 Z-1.125 R0.125
N20 G01 X1.825
N21 X2.125 Z-1.275
N22 Z-2.5 F0.008
N23 X3.
N24 G40 G00 X3.1  (Q)  (End of geometry defined by Q number in G71 and a G40)
N25 G97 S450 M09
N26 G28              (Return to Machine Zero)

N27 G28 (G70 FINISHING O.D.)
N28 T202             (Tool 2 Offset 2)
N29 G50 S3000
N30 G97 S1950 M03
N31 G54 G00 X0.725 Z0.1 M08    (Rapid to start point above part stock)
N32 G96 S370
N33 Z0.              (Position to face part)
N34 G01 X-0.062 F0.01    (Face down part adding in compensation for the tool radius)
N35 G00 X3.1 Z0.1
N36 G70 P11 Q24      (G70 Finishing Cycle with TNC in between N10 and N20)
N37 G97 S450 M09
N38 G28              (Return to Machine Zero)
N39 M30              (End of Program)
```

G71/G70 TYPE I ROUGHING & FINISHING AN O.D.
WITH TOOL NOSE COMPENSATION EXERCISE
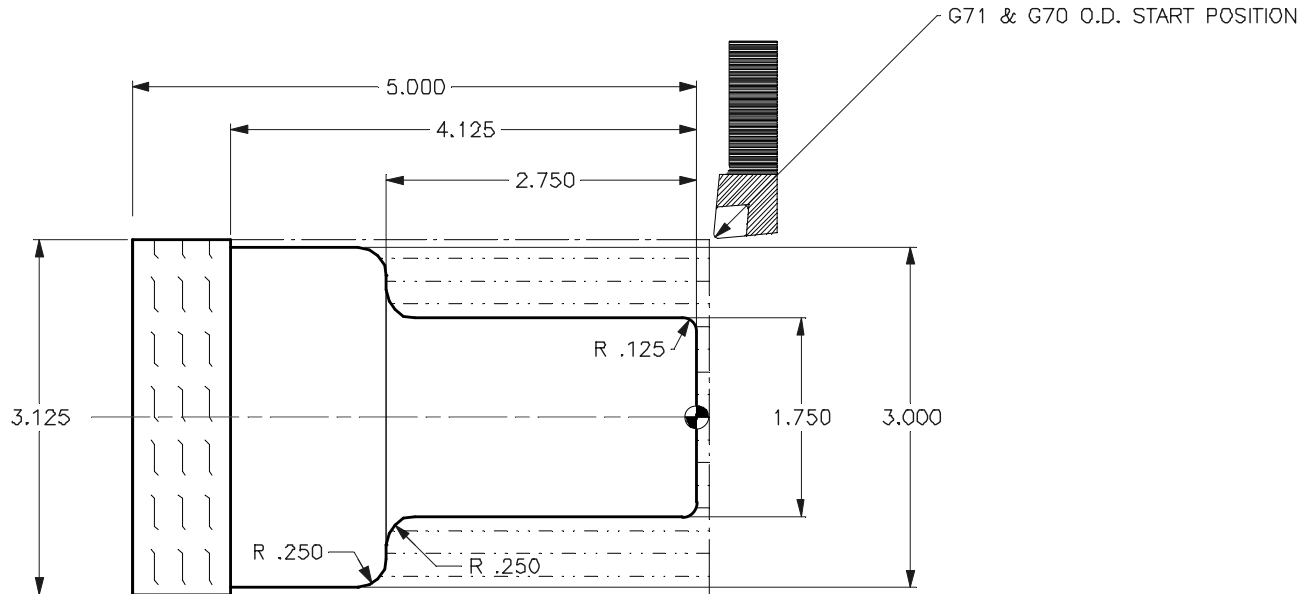
Fill in the blank lines for tool #1 using a G71 Roughing cycle command, to define Roughing passes for the part geometry defined between N10 and N20. Then define a G70 Finishing cycle command, with tool #2, to define a finish pass for the part geometry defined between N10 and N20. On the G71 command line leave .010 stock on diameters and .005 on the faces. Define a .120 depth of cut at .010 roughing feedrate. Use cutter compensation in the controls offset display for both tools #1 and #2 and enter a .031 radius and tip direction.

NOTE: When starting to define part geometry to position the tool nose radius below the radius or angle on an O.D. of a part, or above the radius or angle of an I.D. of a part. The amount to position above or below should be the radius size of tool tip times two (.0315 x 2=.063 above or below). This needs to be done so that the start of the radius of the tool will start at the beginning of the radius on the part at the beginning sequence number after the G71 to start defining the part geometry.

TOOLS	OFFSETS	RADIUS	TIP
T1	01	.0312	3
T2	02	.0312	3

O00060   (Program number)
N1(ROUGH O.D.)   (Rough O.D.)
G53 G00 X0. Z0. T0   (Sending home for a tool change)
T101 (O.D. TOOL x .031 TNR)   (Tool #1 and Offset #1)
G50 S3200   (Spindle speed clamp at 3200 RPM)
G97 S500 M03   (Cancel CSS, 500 RPM, spindle ON forward)
G54 G____ X_____ Z____ M08   (Rapid X3.2, Z0.1 to start point, coolant ON)
G____ S_____   (CSS ON, at 420 surface speed)
Z_____   (Position .005 from end of part)
G____ X_____ F.008   (Feed down X-.063 to face end of part)
G____ X_____ Z_____   (Rapid to X3.2, Z0.1 start point above part)
G____ P___ Q___ U____ W____ D____ F____   (G71 Rough cycle)
N___ G____ G____ X_____   (Pnn start # Cutter Comp. ON, rapid X1.4,)
G____ Z_____ F0.006   (Feed onto face of part a Z0.)
X_____   (Move up in X axis to the start of radius)
G____ X_____ Z_____ R_____   (Define move around .125 Radius)
G____ Z_____   (Define remaining G71 Part Geometry)
G____ X_____ Z_____ R_____   ("   "   "   "   "   "   )
G____ X_____   ("   "   "   "   "   "   )
G____ X_____ Z_____ R_____   ("   "   "   "   "   "   )
G____ Z_____ F.004   ("   "   "   "   "   "   )
N___ G____ X_____ F.02   (Qnn end #, cancel Cutter Comp. feed X to 3.2)
G____ S_____ M____   (Cancel CSS, 500 RPM, Coolant Off)
G53 G00 X0. Z0. T0   (Sending home for a tool change)
M____   (Optional stop command)

N2 (FINISH O.D.)
G53 G00 X0. Z0. T0   (Sending home for a tool change)
T_____ (O.D. TOOL x .031 TNR)   (Tool #2 and Offset #2)
G____ S_____   (Spindle speed clamp at 3200 RPM)
G____ S_____ M____   (Cancel CSS, 890 RPM, spindle on)
G54 G____ X_____ Z_____ M____   (Rapid, X1.8, Z.1 location, coolant ON)
G____ S_____   (Turn on CSS to 420)
Z_____   (Position to Z0 end of part)
G____ X_____ F0.005   (Feed down face of part)
G____ X_____ Z_____   (Rapid to X3.2, Z0.1 start point above part)
G____ P_____ Q_____   (Define a G70 finish pass of part geometry)
G____ S_____ M____   (Cancel CSS, define 500 RPM, Coolant Off)
G53 G00 X0. Z0. T0   (Sending home for a tool change)
M____   (End of program and reset)

G71/G70 TYPE I ROUGHING AND FINISHING AN I.D. WITH TNC EXAMPLE
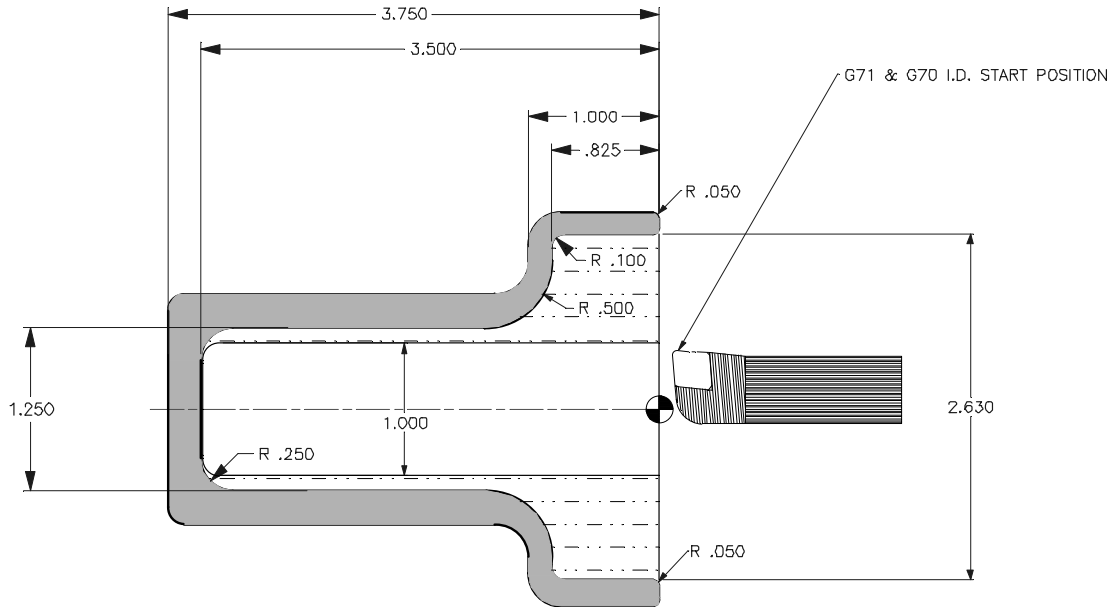
NOTE: Be sure the tools start position is positioned below the diameter of the part you wish to start roughing out, before defining a G71 on an I.D. with this cycle.

TOOL	OFFSET	RADIUS	TIP
4	04	.032	2

```
O00088                    (Example of using a G71 on an I.D. with TNC)
N1 G53 G00 X0. Z0. T0     (Sending home for a tool change)
N2 T404                   (Select Tool 4 Offset 4)
N3 G50 S3000
N4 G97 S1780 M03
N5 G54 G00 X0.9 Z0.1 M08  (Rapid to start point below the I.D. stock diameter)
N6 G96 S420
N7 G71 P8 Q18 U-0.01 W0.005 D0.12 F0.012   (U is minus for G71 I.D. Roughing)
N8 G41 G00 X2.83   (N8, Start of part path geometry defined by P8 in G71 line)
N9 G01 Z0. F0.02
N10 X2.73 F0.005
N11 G02 X2.63 Z-.05 R0.05
N12 G01 Z-.725
N13 G03 X2.43 Z-.825 R.1
N14 G01 X2.25
N15 G02 X1.25  Z-1.325 R0.5
N16 G01 Z-3.25
N17 G03 X.75 Z-3.5 R0.25
N18 G01 G40 X0.7 (N18 End of part path geometry defined by Q18 in G71 line)
N19 G70 P8 Q18
N20 G97 S1780 M09
N21 G53 G00 X0. Z0. T0    (Sending home for a tool change)
N22 M30                   (End of Program)
```

G71 TYPE II ROUGHING DETAILS

When Type II is specified by the programmer the X axis PQ path is allowed to vary non-monotonically. In other words, the X axis can change direction throughout the PQ path. Z must still continue along in the same direction as the initial Z direction. The PQ path is checked prior to the start of any cutting and an alarm is generated if a problem exists.

When Setting 33 is set to FANUC, Type II is specified by placing a reference to both the X and Z axis in the block specified by P.

Roughing is similar to Type I except while roughing, after each pass along the Z axis, the tool will follow the path defined by PQ. The tool will then retract parallel to the X axis by a distance defined in Setting 73 (CAN CYCLE RETRACTION). The Type II roughing method does not leave steps in the part prior to finish cutting and typically results in a better finish.

There is virtually no limit to the number of blocks used to define a Type II PQ path. There is a limit to the number of troughs that can be included in a PQ path definition. A trough can be defined as a change in direction which creates a concave surface in the material being cut. If successive troughs are on the same level, there can be an unlimited number of troughs. When troughs are within troughs (nested), there can be no more than 10 levels of trough nesting. An alarm is generated when this limit is exceeded.

When using TNC it is very unusual to have a Type II path. This is because the current compensation methods can only compensate the tool tip in one direction.

G71 & G70 O.D. TYPE II ROUGHING & FINISHING WITH TNC USING 2 TOOLS

G71 & G70 Type II O.D. with TNC Stock Removal Example

- - - - ▶ Rapid move
◀——— Feed move
——— Programmed Path
Ⓒ Clearance Point
Ⓟ Starting Block
Ⓠ Ending Block
▓ Finish Pass Stock Remaining (U,W)

Setup information

Setting 33: FANUC

Tool 1 - Rough and Finish with a 35 Deg. x .032 radius insert

Tool	Offsets	Radius	Tip
1	01	.032	3
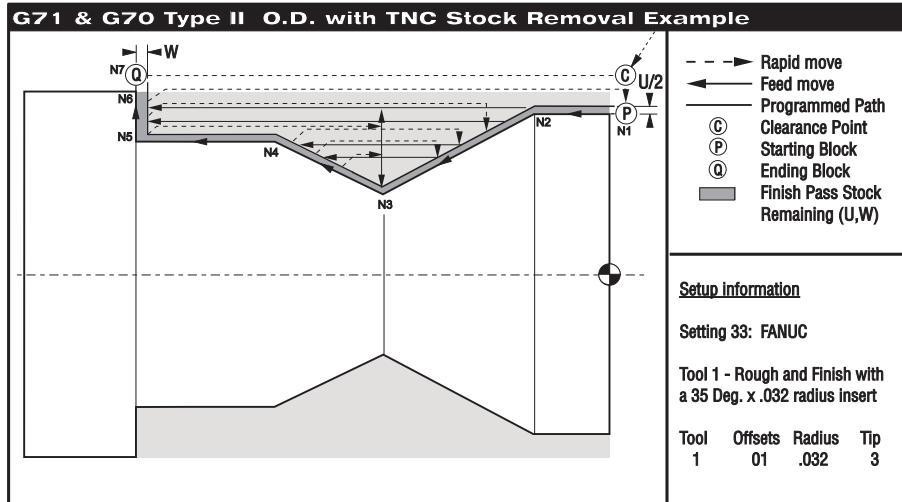
The following program is an example of a G71 Type II program:

O00090
N101 G28 (FANUC TYPE II G71 ROUGHING CYCLE)
N102 T101   (Roughing Tool)
N103 G50 S3000
N104 G97 S746 M03
N105 G54 G00 X2.1 Z0.1   (Start Position)
N106 G96 S410
N107 G71 P108 Q113 U0.02 W0.005 D0.05 F0.01
N108 G00 X1.75 Z0.1   (A W0. will also work, on
N109 G01 Z0-.25 F0.006   the P block number, instead
N110 X1. Z-1.   of using a Z0.1 to define a
N111 X1.5 Z-1.5   G71 type II geometry.)
N112 Z-2.25
N113 G01 X2.1 (Q block number of the PQ part Definition)
N114 G97 S746 M09
N115 G28

N116 G28 (G70 FINISHING O.D.)
N117 T202   (Finishing Tool)   (In the P block number for a
N118 G50 S2500   G71 type-I geometry you
N119 G97 S746 M03   only have a one axis move
N120 G54 G00 X2.1 Z0.1 (Start Position)   in X axis. In a G71 type-II
N121 G96 S410   geometry you have a two
N122 G70 P108 Q113   (G70 Finishing Cycle)   axis moves in both X and Z
N123 G97 S746 M09   in the P block number.)
N124 G28
N125 M30

G71/G70 TYPE II ROUGHING EXERCISE WITH TNC
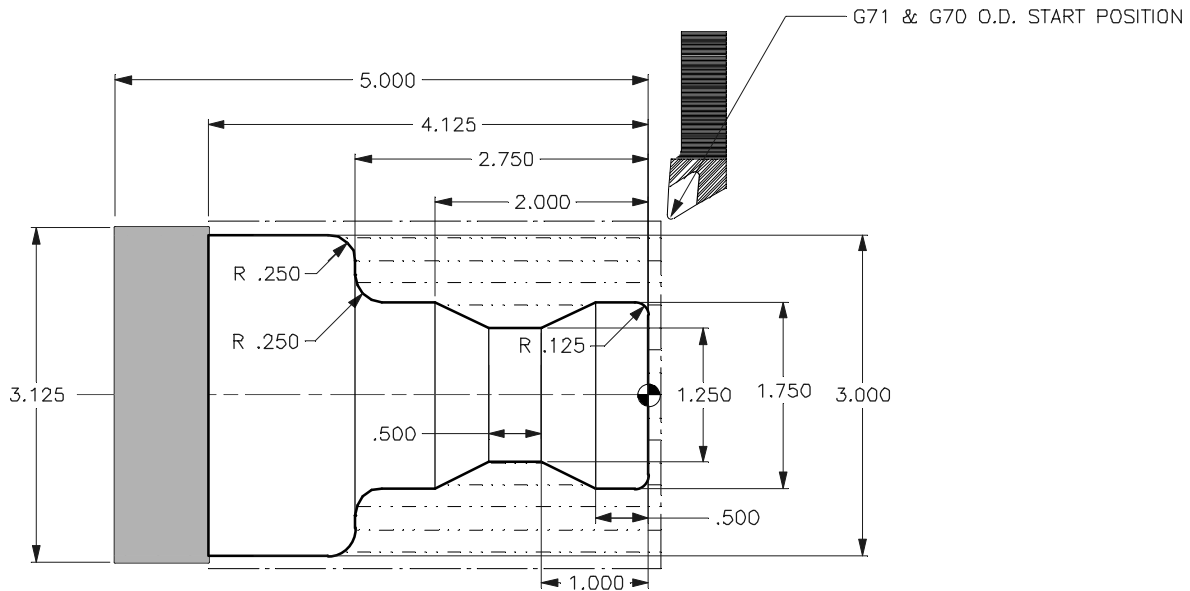
Fill in the blank line for tool #1 using a G71 Roughing Type II cycle command, to define Roughing passes for the part geometry defined between N10 and N20. Then define a finish pass using a G70 Finishing cycle command, with tool #2. Define a finish pass for the part geometry defined between N10 and N20. On the G71 command line leave .010 stock on diameters and .005 on the faces. Take .120 depth of cut at .012 feed.

NOTE: The problem with type II geometry is that the part geometery that you program in with tool nose compensation is compensating with a tip direction 3. And so the angle going from the linear dimension .500 to 1.000 needs to have a tip direction 4. So the linear dimensions for angles or arcs that change tip direction will not be correct since it's not using the tip direction that is needed for those arcs or angles. For this condition, the program will have to be modified (add in the compenation to those program moves to bring within size) for the angles and arcs that change tip direction.

TOOLS	OFFSETS	RADIUS	TIP
T1	01	.0312	3
T2	02	.0312	3

*Duplicate the part program that is similar to this part on page 86 and add in the values to the blank lines for this Type II part geometry.

O00070	(Program number)
N1 (ROUGH O.D.)	(G71 type II roughing operation)
G53 G00 X0. Z0. T0	(Sending home for a tool change)
T101 (O.D. TOOL .031 TNR)	(Tool #1 and offset #1)
G50 S3200	(Spindle speed max. clamp at 3200 RPM)

(Program starts on page 92)

```
G97 S500 M03                    (Cancel CSS, 500 RPM, spindle ON forward)
G54 G00 X3.2 Z1. M08            (Rapid X and Z to start location, coolant ON)
G96 S425                        (CSS on at 420)
Z0.005                          (Position .005 from end of part)
G01 X-0.063 F0.008              (Feed down to rough face end of part)
G00 X3.2 Z0.1                   (Rapid to start position above part)
G71 P10 Q20 U0.01 W0. D0.12 F0.01  (Rough O.D. G71 cycle command)
*N10 G42 G00 X1.4 Z_____        (P start number, G42 cutter comp ON, rapid
                                 with G00 X and Z axis or enter W0 instead of
                                 Z.1 for G71 type II roughing)
G01 Z0. F0.006                  (G71 Part Geometry)
X1.5                            ( "      "        "    )
G03 X1.75 Z-0.125 R0.125        (G71 Part Geometry)
*G_____ Z_____               (Type II G71 Geometry changes direction in X axis)
*X_____ Z_____  F.01          ( "          "        "        "          "    )
*Z_____                        ( "          "        "        "          "    )
*X_____ Z_____                ( "          "        "        "          "    )
Z-2.5 F.006                     (G71 Part Geometry)
G02 X2.25 Z-2.75 R0.25          ( "      "        "    )
G01 X2.5                        ( "      "        "    )
G03 X3. Z-3. R0.25              ( "      "        "    )
G01 Z-4.125 F0.004              ( "      "        "    )
N20 G40 X3.2 F0.02              (Qnn ending number, cancel Cutter Comp. to X3.2)
G97 S500 M09                    (Cancel CSS, 500 RPM, coolant Off)
G53 G00 X0. Z0. T0              (Sending home for a tool change)
M01                             (Optional stop command)

N2 (FINISH O.D.)                (G71 type II finishing operation)
G53 G00 X0. Z0. T0              (Sending home for a tool change)
T202 (O.D. TOOL .031 TNR)       (Tool #2 and Offset #2)
G50 S3200                       (Spindle speed clamp at 3200 RPM)
G97 S890 M03                    (Cancel CSS, 890 RPM, spindle ON forward)
G54 G00 X1.8 Z1. M08            (Rapid, X,Z location, coolant ON)
G96 S420                        (Turn on CSS to 420)
Z0.                             (Position to end of part)
G01 X-.062 F0.006               (Feed down face of part)
G00 X3.2 Z.1                    (Rapid to start position above part)
G70 P10 Q20                     (G70 finish pass using part geometry)
G97 S500 M09                    (Cancel CSS, 500 RPM, coolant Off)
G53 G00 X0. Z0. T0              (Sending home for a tool change)
M30                             (End program and rewind)
```

G72 END FACE STOCK REMOVAL CYCLE

P   Starting block number of part path to machine

Q   Ending block number of part path to machine

U* Finish stock remaining with direction (+or -), X-axis ***diameter*** value

W* Finish stock remaining with direction (+or -), Z-axis value

I*   Last pass amount with direction (+or -), X-axis ***radius*** value

K* Last pass amount with direction (+or -), Z-axis value

D* Stock removal depth of cut each pass, positive value (Setting 72)

R1*YASNAC type II roughing (only if setting 33 is on Yasnac)

F   Roughing passes feed rate throughout this cycle

S** Spindle speed in this cycle

T** Tool and offset in this cycle

   * Indicates optional

  ** Rarely defined in a G72 line
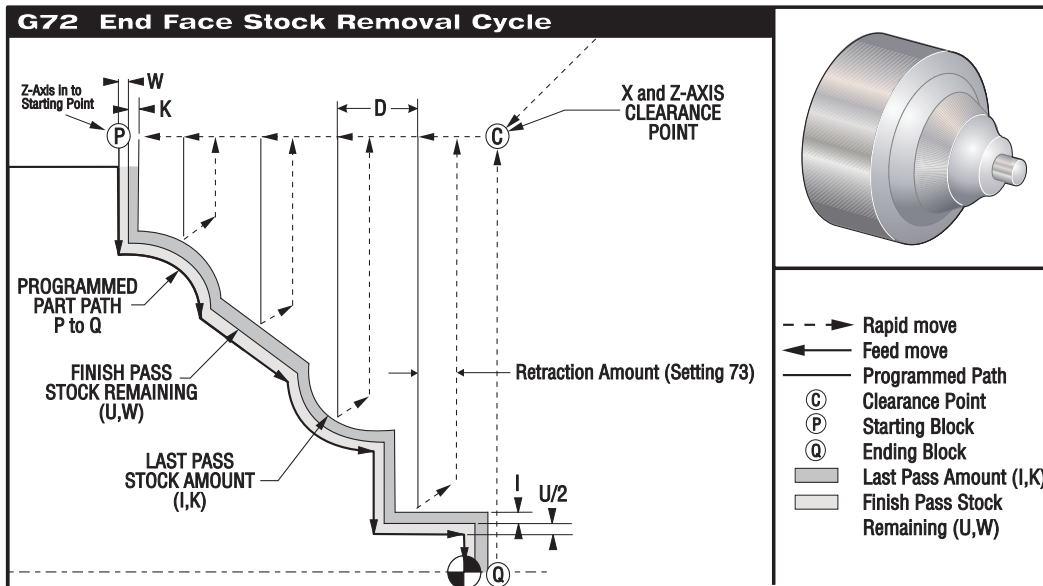
This canned cycle will rough out material on a part given the finished part shape. It is similar to G71 but roughs out material along the face of a part. All a programmer needs to do is to define the shape of a part by programming the finished tool path and then submitting the path definition to the G72 call by means of a PQ block designation. Any feeds, spindle speeds or tools within the block defining the path are ignored by the G72 call. Any F,S or T commands on the G72 line or in effect at the time of the G72 is used throughout the G72 roughing cycle. Usually a G70 call to the same PQ block definition is used to finish the shape using the programmed feeds, speeds tools and offsets.

Two types of machining paths are addressed with a G72 command. The first type of path (TYPE I) is when the Z-axis of the programmed path does not change direction. This type of path is called a monotonic path. The second type of path (TYPE II) allows the Z-axis to change direction. For both the first type and the second type of programmed path the X-axis must be monotonic, that is it cannot change direction. Type I is selected by having only an X-axis motion in the block specified by P in the G71 call. When both an X-axis and Z-axis motion are in the P block then TYPE II roughing is assumed.

The G72 consists of a roughing phase and a finishing phase. The roughing and finishing phase are handled slightly differently for types I and types II. Generally the roughing phase consists of repeated passes along the X-axis at the specified feed rate. The finishing phase consists of a pass along the programmed tool path to remove excess material left by the roughing phase but to leave finish material for a G70 block with perhaps a finishing tool. The final motion in either types is a return to the starting position S.

In the following the start position S is the position of the tool at the time of the G72 call. The X clearance plane is derived from the X-axis start position and the sum of U and optional I finish allowances.

G72 TYPE I ROUGHING DETAILS

When Type I is specified by the programmer it is assumed that the tool path is monotonic in the X-axis. Prior to any roughing motion the tool path is checked for monotonicity and G code compliance. An alarm is generated if a problem is found.

Roughing begins by advancing from the start position S and moving to the first roughing pass. All roughing passes start and end at the X clearance plane. Each roughing pass Z-axis location is determined by applying the value specified in D to the current Z location. The direction that D is applied is determined by the signs of U and W. The nature of the movement along the X clearance plane for each roughing pass is determined by the G code in block P. If block P contains a G00 code, then movement along the X clearance plane is a rapid mode. If block P contains a G01 then movement will be at the G72 feed rate. Roughing continues until the Z-axis position in block P is exceeded.

Each roughing pass is stopped before it intersects the programmed tool path allowing for both roughing and finishing allowances. The tool is then retracted from the material, at a 45 degree angle by the distance specified in Setting 73. The tool then moves in rapid mode to the X-axis clearance plane.

When roughing is completed the tool is moved parallel to the tool path to clean up the rough cut. If I and K are specified an additional rough finish cut parallel to the tool path is performed.

G72 & G70 TYPE I ROUGH AND FINISH FACING USING TNC WITH 2 TOOLS

G72 & G70 Facing with TNC Example

- - - ► Rapid move
──────► Feed move
────── Programmed Path
ⓒ Start Position
ⓟ Starting Block
ⓠ Ending Block
Ⓡ Radius
▮ Finish Pass Stock Remaining (U,W)

Setup information
Setting 33: FANUC

Tool 1 - Roughing with an
80 Deg. x .031 radius insert
Tool 2 - Finishing with an
80 Deg. x .031 radius insert

Tool	Offsets	Radius	Tip
1	01	.031	3
2	02	.015	3
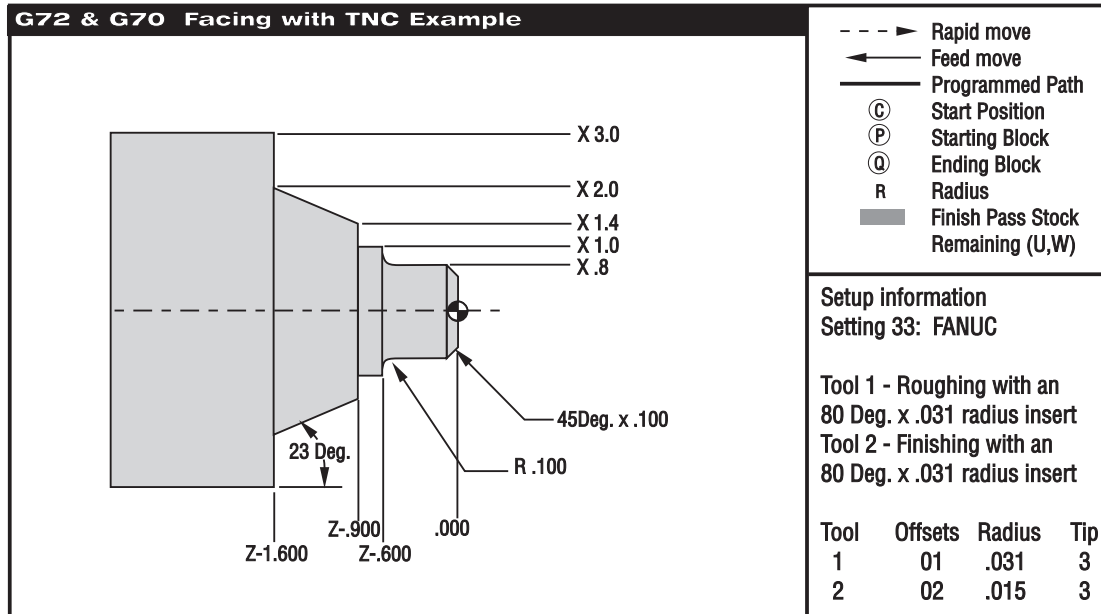
O00096
N1 (G72 ROUGHING FACE)
N2 G53 G00 X0. Z0. T0   (Sending home for a tool change)
N3 T101 (O.D. TOOL x .031 TNR)   (Select Tool 1 Offset 1)
N4 G50 S3000
N5 G97 S450 M03
N6 G54 G00 X3.1 Z0.1 M08   (Rapid to start point above part stock)
N7 G96 S370
N8 G72 P9 Q18 U0.01 W0.01 D0.06 F0.012   (G72 Rough Facing Cycle with TNC)
N9 G41 G00 Z-1.6 (Starting sequence number defined by P8 in G72 and G70)
N10 G01 X2. F0.008
N11 X1.4 Z-0.9
N12 X1.
N13 Z-0.6
N14 G03 X0.8 Z-0.5 R0.1
N15 G01 Z-0.1
N16 X0.6 Z0.
N17 X-0.062
N18 G40 G00 Z0.1 (End of part path geometry defined with P17 in G72 and G70)
N19 G97 S450 M09
N20 G53 G00 X0. Z0. T0   (Sending home for a tool change)
N21 M01 (Optional Stop)

```
N22 (G70 FINISHING FACE)
N23 G53 G00 X0. Z0. T0              (Sending home for a tool change)
N24 T202 (O.D. TOOL x .015 TNR)    (Select Tool 2 Offset 2)
N25 G50 S3000
N26 G97 S450 M03
N27 G54 G00 X3.1 Z0.1 M08          (Rapid to start point)
N28 G96 S420
N29 G70 P9 Q18                     (Finish Facing with G70 Finish Cycle)
N30 G97 S450 M09
N31 G53 G00 X0. Z0. T0             (Sending home for a tool change)
N32 M30                            ( End of Program and Reset)
```

G72 is more efficient to use instead of G71 if the roughing cuts in the X axis are longer than the roughing cuts in the Z axis.

G72/G70 TYPE I ROUGHING & FINISHING A FACE WITH TOOL NOSE COMPENSATION EXERCISE
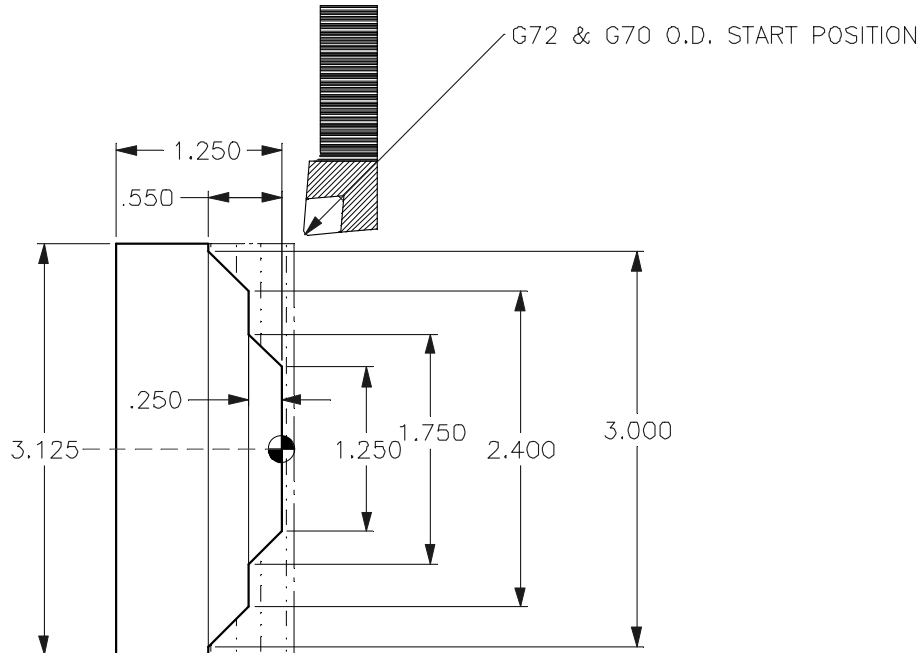
Fill in the blank lines for Tool #1 using a G72 Roughing cycle command to define multiple rough passes facing part geometry defined between N10 and N20. Define tool #2 to do a finish pass using G70 finishing cycle command. For the G72 command line, leave .010 stock on diameter and .010 on the faces. Take .100 depth of cut at .012 feed.

TOOL and OFFSET #	RADIUS	TIP
T101	.0312	3
T202	.0312	3

```
O00080                          (Program number)
N1 (G72 ROUGH FACE)
G53 G00 X-3. Z-4. T0            (Return to tool change location)
T101 (O.D. TOOL x .031 TNR)     (Tool #1 and Offset #1)
G50 S2800                       (Spindle speed clamp at 2800 RPM)
G97 S500 M03                    (Cancel CSS, spindle on, 500 RPM)
G54 G00 X_____ Z_____ M____     (Rapid, X3.2 Z1. location, Coolant ON)
G____ S_____                   (Turn on CSS to 420)
Z_____                         (Position to Z0.1 from end of part)
G___ P___ Q___ U____ W____ D___ F___   (G72 Facing cycle command)
N10 G____ G____ Z_____         (Pnn,  Cutter Comp ON, Rapid to Z-0.55)
G____ X_____ F.006              (Define down part geometry)
X_____ Z_____                (   "       "        "      )
X_____                        (   "       "        "      )
X_____ Z_____                (   "       "        "      )
X_____                         (Face down to X-.063)
N20 G____ G____ Z____           (Ending number, Cancel CC, Rapid Z0.1)
G____ S_____ M____              (Cancel CSS, 500 RPM, coolant off)
G53 G00 X-3. Z-4. T0            (Return to tool change location)
M____                           (Optional stop command)


N2 (G70 FINISH FACE)
G53 G00 X-3. Z-4. T0            (Return to tool change location)
T_____    (O.D. TOOL x .031 TNR)  (Tool #2 and Offset #2)
G____ S_____                    (Spindle speed clamp at 2800 RPM)
G____ S_____ M____              (Cancel CSS, 500 RPM, spindle on CW)
G54 G00 X_____ Z_____ M____    (Rapid to X3.2 and Z1.0, coolant ON)
G____ S_____                    (Turn on CSS at 420)
Z_____                         (Position to Z0.1 from end of part)
G____ P_____ Q_____             (Define a G70 finish pass on part)
G____ S_____ M____              (Cancel CSS, 500 RPM, coolant off)
G53 G00 X-3. Z-4. T0            (Return to tool change location)
M____                           (End of Program and Reset)
```

G73   IRREGULAR PATH STOCK REMOVAL CYCLE

P Starting block number of part path to machine
Q Ending block number of part path to machine
U* Finish stock remaining with direction (+or -), X-axis **diameter** value
W* Finish stock remaining with direction (+or -), Z-axis value
I Distance and direction from first cut to last cut amount, X-axis **radius** value
K Distance and direction from first cut to last cut amount, Z-axis value
D Number of roughing passes, positive number
F Roughing passes feed rate throughout this cycle
S** Spindle speed to use in this cycle
T** Tool and offset to use in this cycle
    * Indicates optional
  ** Rarely defined in a G73 line

G73 Irregular Path Stock Removal Cycle
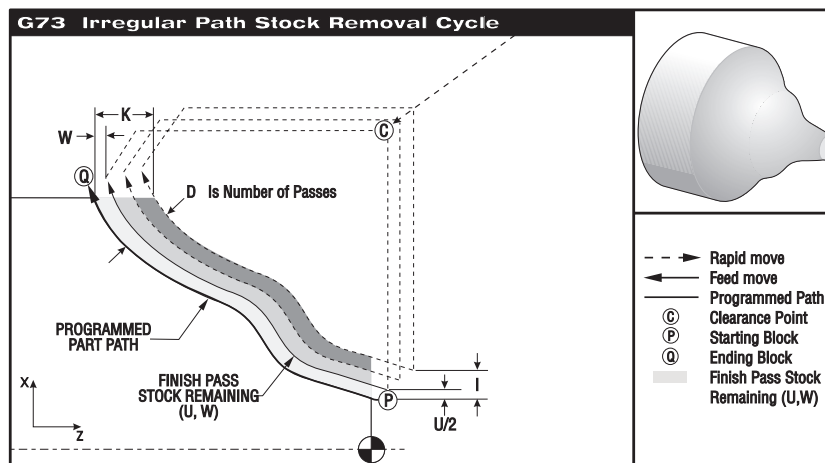
The G73 canned cycle can be used for rough cutting of castings or a forging. The canned cycle assumes that material has been relieved or is missing a certain known distance from the programmed tool path P thru Q.

Machining starts at the start position S, and either rapids or feeds to the first rough cut. The nature of the approach move is based on whether a G00 or G01 is programmed in block P. Machining continues parallel to the programmed tool path. When block Q is reached a rapid departure move is executed to the Start position plus the offset for the second roughing pass. Roughing passes continue in this manner for the number of rough passes specified in D. After the last rough pass, the tool returns to starting position S.

Any feed (F), spindle speed (S) or tool change (T) commands on the lines from P to Q are ignored and any F, S and T prior to or in the G73 block are in effect.

The programmed tool path PQ does not have to be monotonic in X or Z, but care has to be taken to insure that existing material does not interfere with tool movement during approach and departure moves.

This canned cycle is intended for use with the G70 finishing canned cycle.

G73 & G70 O.D. ROUGHING AND FINISHING USING TNC WITH 2 TOOLS

G73 &G70 Type I O.D. with TNC Stock Removal Example
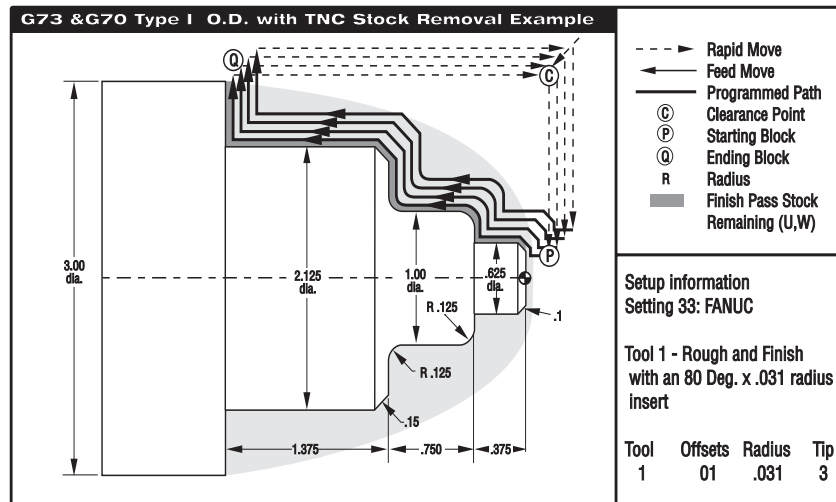
O00101 (G73 O.D. ROUGHING)
N101 G53 G00 X-3. Z-4. T0   (Rapid to tool change location)
N102 T101   (Tool 1 Offset 1)
N103 G50 S3000
N104 G97 S450 M03
N105 G54 G00 X3.1 Z.1 M08   (Rapid to Start Point)
N106 G96 S370
N107 **G73 P108 Q121 U.01 W.005 I0.3 K0.15 D4 F.012** (G73 Stock Removal)
N108 G42 G00 X0.325 (**P**) (Start of geometry P number in G73 line. G42 C.C. right)
N109 G01 Z0. F0.01
N110 X0.425
N111 X.625 Z-0.1 F0.005
N112 Z-0.375
N113 X0.75
N114 G03 X1. Z-0.5 R.125
N115 G01 Z-1. F0.003
N116 G02 X1.25 Z-1.125 R.125
N117 G01 X1.825
N118 X2.125 Z-1.275
N119 Z-2.5 F0.008
N120 X3.
N121 G40 G00 X3.1 (**Q**)   (End of geometry Q number in G71 line. G40 cancels C.C.)
N122 **G70 P108 Q121**   (G70 Finishing cycle N108 thru N121)
N123 G97 S450 M09
N124 G53 G00 X-3. Z-4. T0 (Rapid to tool change location)
N125 M30   (End of Program)

G73/G70 EXERCISE WITH TOOL NOSE COMP.

It's usually best to position the tool above the largest diameter to be machined.

G73 & G70 STARTING POINT

5.000
4.125
2.750
R .250
R .250
R .500
3.125
1.750
3.000
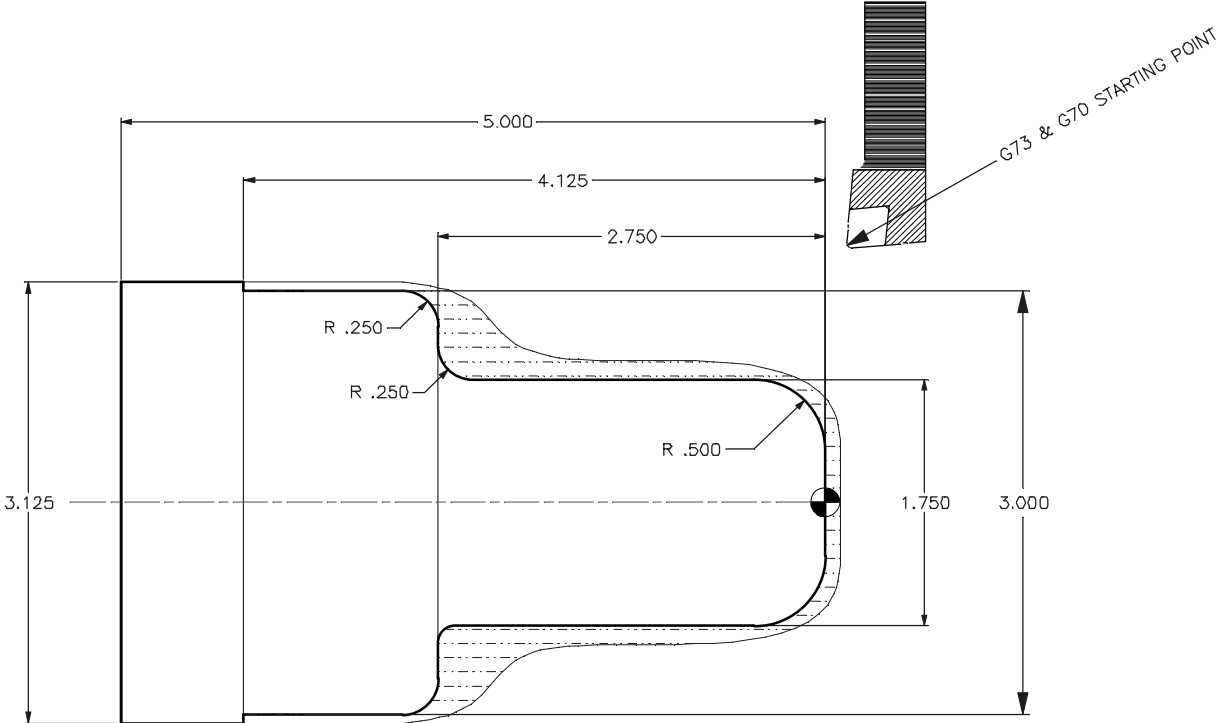
Fill in the blank line for tool #1 using a G73 Roughing cycle command, to define Roughing passes for the part geometry defined between N10 and N20. Define a finish pass using a G70 Finishing cycle command, with tool #2 to do a finish pass of the part geometry defined between N10 and N20. On G73 command line leave .010 stock on diameters and .005 on the face for a G70 finish cycle pass. Take 5 passes on a casting that has .300 stock on the diameter ("I" will be half that amount). Part has .160 stock on linear dimensions defined with K. Use Rough part with .012 feed rate.

TOOLS	OFFSETS	RADIUS	TIP
T1	01	.0312	3
T2	02	.0312	3

```
O00090                                    (Program number)
N1 (G73 O.D. ROUGHING)                    (Rough O.D. with a G73)
G53 G00 X-3. Z-4. T0                      (Rapid to tool change location)
T101 (O.D. TOOL x.031 TNR)                (Tool #1 and Offset #1)
G50 S2800                                 (Spindle speed clamp at 2800 RPM)
G97 S500 M03                              (Cancel CSS, 730 RPM, spindle ON forward)
G54 G00 X3.2 Z0.1 M08                     (Rapid X and Z to start location, coolant ON)
G96 S420                                  (CSS on at 420)
Z0.005                                    (Position .005 from end of part)
G01 X-0.063 F0.01                         (Feed down X-.063 to face end of part)
G00 X3.2 Z0.1                             (Rapid to X3.2, Z0.1 start point above part)
G___P___Q___U____W____I____K____D___F____ (Rough O.D. with G73)
N____G___G____X_____ (P start number, Cutter Comp ON, Rapid to X0.650)
G____Z_____F0.006                         (Feed onto part to Z0.)
X_____                                   (Move up in X axis to the start of radius)
G____X_____Z_____R_____                 (Define move around 0.5 Radius)
G01  Z-2.5                                (Define remaining G73 Part Geometry)
G02 X2.25 Z-2.75 R0.25                    (    "      "      "      "      "      "      )
G01 X2.5                                  (    "      "      "      "      "      "      )
G03 X3. Z-3. R0.25                        (    "      "      "      "      "      "      )
G01 Z-4.125 F0.004                        (    "      "      "      "      "      "      )
N____ G40 G00 X3.2                        (Q Ending number, Cancel Cutter Comp.)
G97 S500 M09                              (Cancel CSS,  500 RPM, Coolant Off)
G53 G00 X-3. Z-4. T0                      (Rapid to tool change location)
M01                                       (Optional stop command)


N2 (G70 O.D. Finish)                      (Finish O.D.)
G53 G00 X-3. Z-4. T0                      (Rapid to tool change location)
T202 (O.D. TOOL x .031 TNR)               (Tool #2 and Offset #2)
G50 S2800                                 (Spindle speed clamp at 2800 RPM)
G97 S890 M03                              (Cancel CSS, turn on spindle 890 RPM)
G54 G00 X1.8 Z1.0 M08                     (Rapid, X,Z location, coolant ON)
G96 S420                                  (CSS on at 420)
Z0.                                       (Position to Z0. face end of part)
G01 X-.062F0.005                          (Feed down the face of part)
G00 X3.2 Z1.0                             (Rapid to clearance point above part)
G____P_____Q_____                         (Define finish pass using part geometry)
G97 S500 M09                              (Cancel CSS, 500 RPM, Coolant Off)
G53 G00 X-3. Z-4. T0                      (Rapid to tool change location)
M30                                       (End of program rewind)
```

G74 End Face Grooving or high speed Peck Drill Cycle

X* X-axis absolute location to the furthest peck, **diameter** value

Z Z-axis absolute pecking depth

U* X-axis incremental distance and direction (+or -) to furthest peck, **diameter** value

W* Z-axis incremental pecking depth

I * X-axis shift increment between pecking cycles, positive **radius** value

K* Z-axis pecking depth increment

D* Tool shift amount when returning to clearance plane (Caution see **NOTE**)

F Feed rate

  * Indicates optional

G74 End Face Grooving Cycle, Peck Drilling
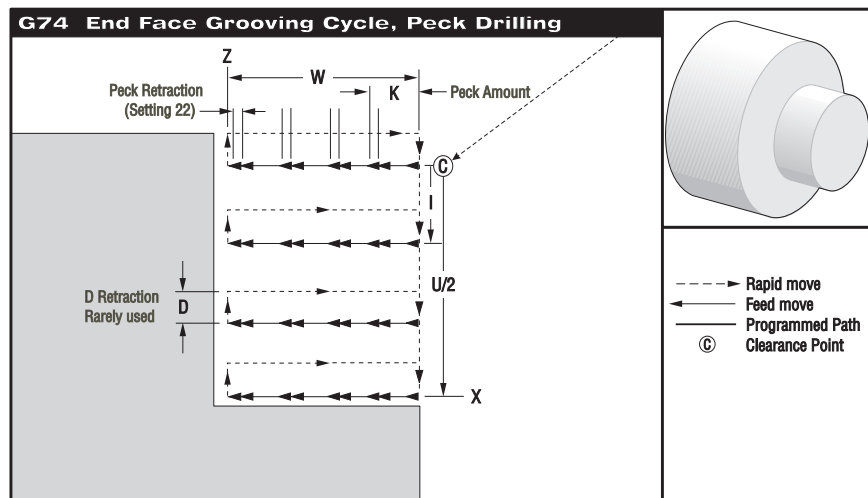
The G74 canned cycle can be used for grooving on the face of a part, turning with a chip break or high speed peck drilling. With this canned cycle either a single groove peck cycle can be executed, or you can perform a series of pecks in this cycle to machine a wide groove, or a high speed peck drilling on the spindle centerline.

When an X, or U, code is added to a G74 block and X is not the current position, then a minimum of two pecking cycles will occur. One at the current location and another at the X location. The I code is the incremental distance between X axis pecking cycles. Adding an I will perform multiple, evenly spaced, pecking cycles between the starting position S and X. If the distance between S and X is not evenly divisible by I then the last interval along X will be less than I.

When K is added to a G74 block, then pecking will be performed at each interval distance specified by K. At each peck depth, a rapid move opposite the direction of feed and the distance the tool pulls back is entered into Setting 22.

Warning: *The D code command is rarely used and should only be used if the wall on the outside of groove does not exist like the diagram above shows.*

The D code can be used in grooving and turning to provide a tool clearance shift, in the X axis, before returning in the Z axis to the "C" clearance point. But, if both sides to the groove exist during the shift, then the groove tool would break. So you wouldn't want to use the D command.

G74 SINGLE PASS FACE GROOVING CYCLE

G74 End Face Grooving Cycle (Single Pass) Example
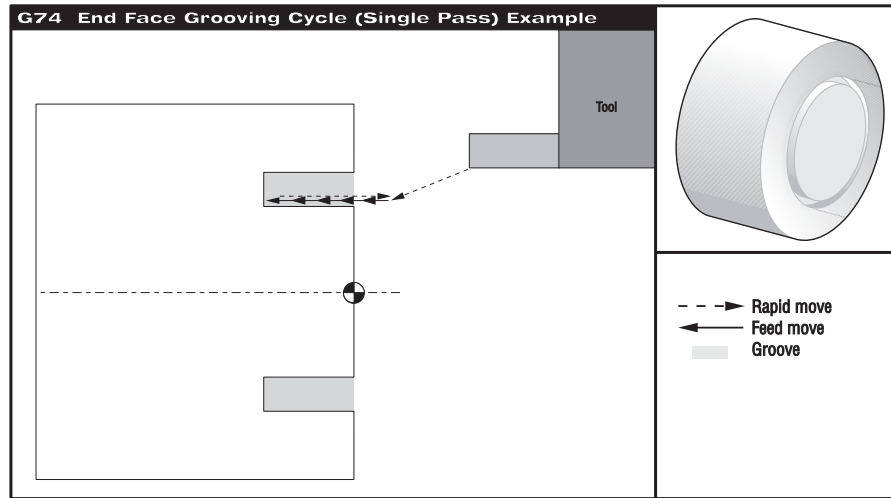
O00105 (G74 Single Pass End Face Grooving Cycle)
(Machine a .25 Wide Groove with a .25 Groove Tool)
N1 G28
N2 T404 (.25 END FACE GROOVE TOOL)
N3 G97 S1150 M03
N4 G54 G00 X1.5 Z0.1 M08   (Rapid to a start point)
N5 G74 Z-0.375 K0.1 F0.005   (Feed to a Z-.375 depth with a .1 peck)
N6 M09
N7 G28
N8 M30

SETTING 22 (CAN CYCLE DELTA Z) - *As the groove tool pecks deeper into the part, with each peck value of K, it pulls back a constant specified distance above the bottom of the groove created by the previous peck to break the chip. That specified distance it pulls back is defined in Setting 22.*

G74 MULTIPLE PASS FACE GROOVING CYCLE

G74 End Face Grooving Cycle (Multiple Pass) Example

- - → Rapid move
← Feed move
Groove
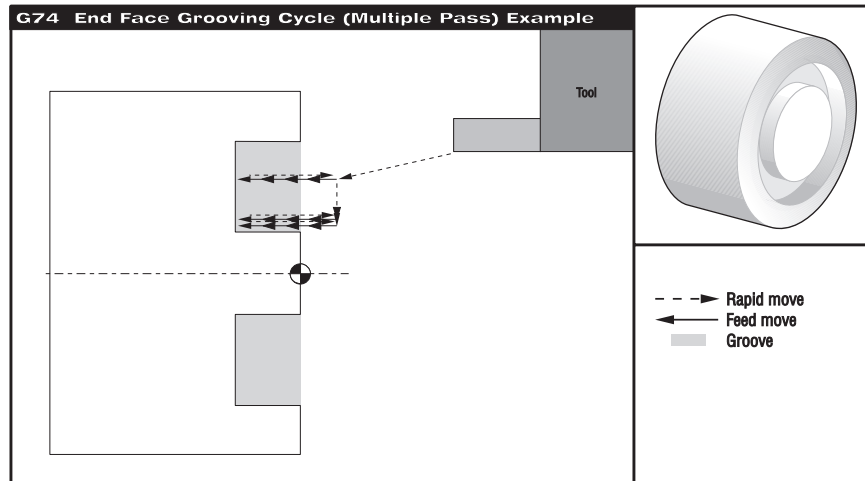
O00106 (G74 Multiple Pass End Face Grooving Cycle)
(Machine a .500 Wide Groove with a .25 Groove Tool)
N1 G28
N2 T404 (.25 WIDE GROOVE TOOL)
N3 G97 S1150 M03
N4 G54 G00 X1.75 Z0.1 M08   (Rapid to Start Point)
N5 G74 X1.25 Z-.375 I0.2 K0.05 F.005 (G74 Multiple pass groove .05 peck Z-.375 depth)
N6 M09
N7 G28
N8 M30

SETTING 22 (CAN CYCLE DELTA Z) - *As the groove tool pecks deeper into the part, with each peck value of K, it pulls back a constant specified distance above the bottom of the groove created by the previous peck to break the chip. That specified distance it pulls back is defined in Setting 22.*

HIGH SPEED PECK DRILLING USING A G74

G74 High Speed Peck Drill Cycle

Rapid move
Feed move
0   Begin or End of stroke

Z Part Zero
Initial Starting Plane

X

Z
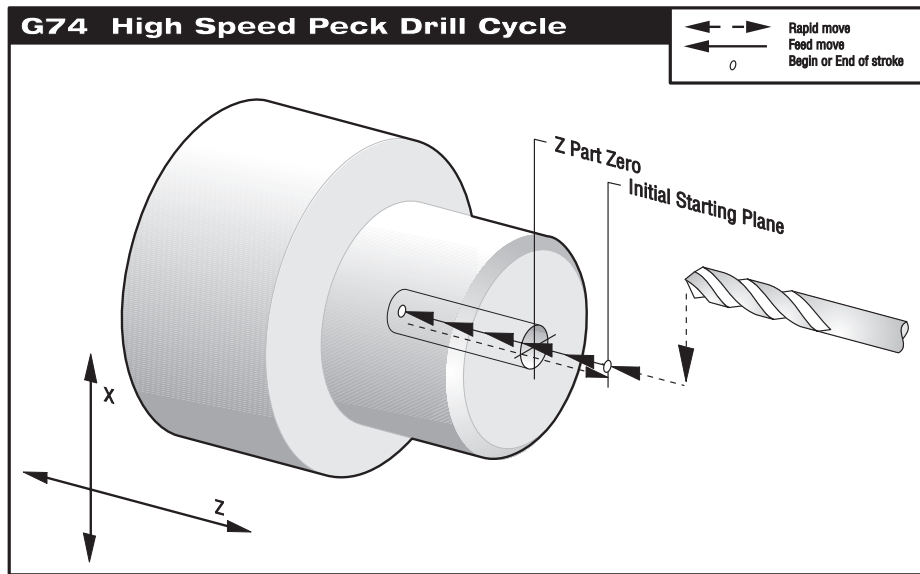
O00107 (G74 High Speed Peck Drilling example)
(Drill a .500 Diameter to a .525 Depth)
N1 G28
N2 T404 (1/2 DIA. DRILL)
N3 G97 S2445 M03
N4 G54 G00 X0. Z0.1M08   (Rapid to X0 and Z start point)
N5 G74 Z-.525 K0.1 F0.006   (Drills to Z-.525 depth, pecking every .1 to pull)
N6 G00 Z1.0 M09   (back after each peck the amount in Setting 22.)
N7 G28
N8 M30

SETTING 22 (CAN CYCLE DELTA Z) - *As the drill pecks deeper into the part, with each peck value of K, it pulls back a constant specified distance above the bottom of the hole created by the previous peck to break the chip. That specified distance it pulls back is defined in Setting 22.*

G75   O.D./I.D. GROOVING CYCLE

X   X-axis absolute pecking depth, **diameter** value
Z*   Z-axis absolute location to the furthest peck
U*   X-axis incremental pecking depth, **diameter** value
W*   Z-axis incremental distance and direction (+or -) to the furthest peck
I *   X-axis pecking depth increment, **radius** value
K*   Z-axis shift increment between pecking cycles
D*   Tool shift amount when returning to clearance plane (Caution see **NOTE**)
F   Feed rate
   * Indicates optional

G75 O.D. / I.D. Grooving Cycle
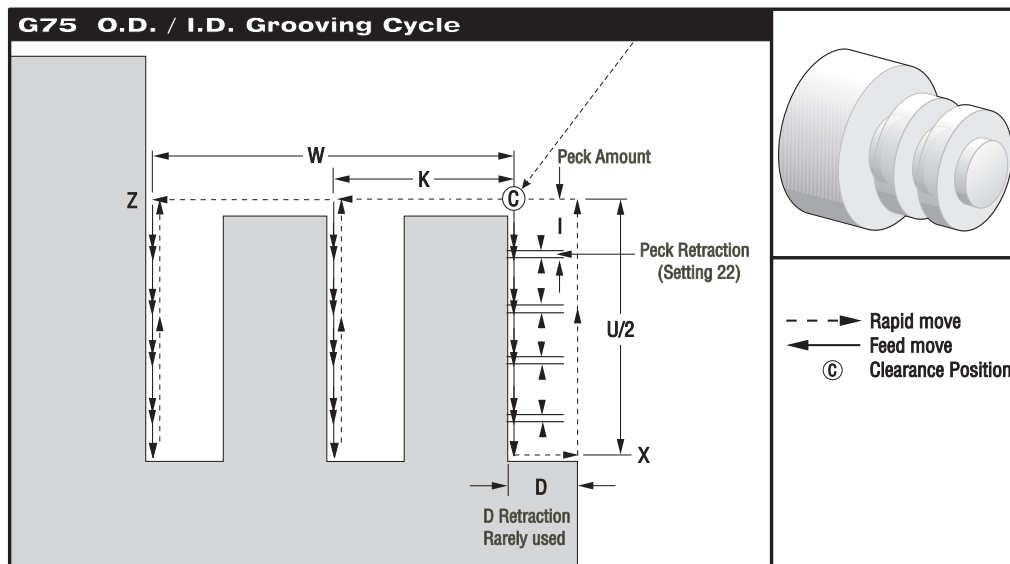
The G75 canned cycle can be used for grooving an outside diameter with a chip break. With this canned cycle either a single pecking cycle can be executed, as for a single groove, or a series of pecking cycles can be performed, for multiple grooves.

When an Z, or W, code is added to a G75 block and Z is not the current position, then a minimum of two pecking cycles will occur. One at the current location and another at the Z location. The K code is the incremental distance between Z axis pecking cycles. Adding a K will perform multiple, evenly spaced, pecking cycles between the starting position S and Z. If the distance between S and Z is not evenly divisible by K then the last interval along Z will be less than K.

When I is added to a G75 block, then pecking will be performed at each interval specified by I, the peck is a rapid move opposite the direction of feed and the peck distance is obtained from Setting 22.

***Warning:** The D code command is rarely used and should only be used if the wall on the outside of groove does not exist like the diagram above shows.*

The D code can be used in grooving and turning to provide a tool clearance shift, in the Z axis, before returning in the X axis to the "C" clearance point. But, if both sides to the groove exist during the shift, then the groove tool would break. So you wouldn't want to use the D command.

G75 SINGLE PASS O.D. GROOVING CYCLE

G75 O.D. / I.D. Grooving Cycle (Single Pass) Example

Tool

Rapid move
Feed move
Groove
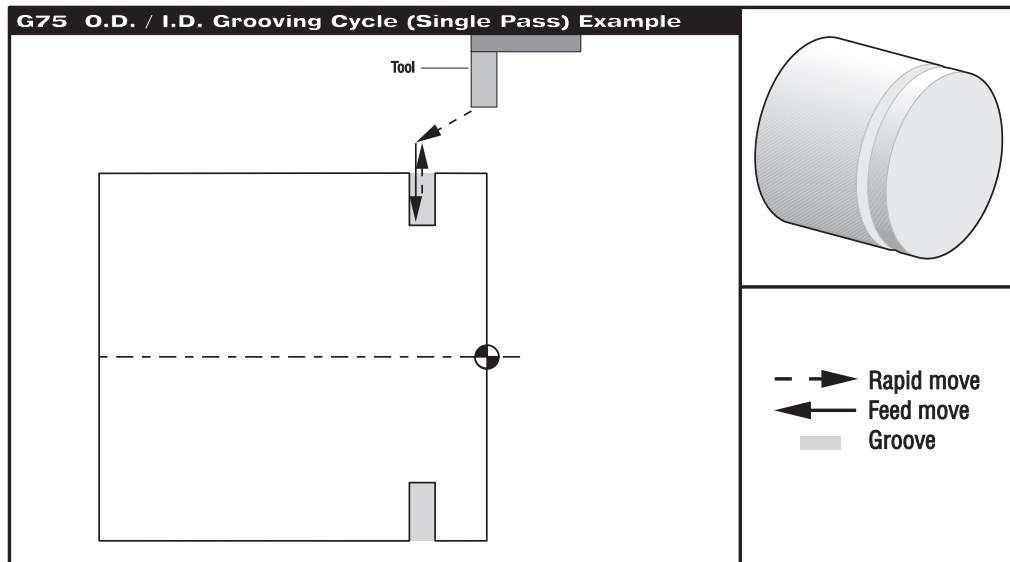
O00109 (G75 O.D./I.D. SINGLE PASS GROOVE CYCLE)
(Machine a .25 wide O.D. Groove with .25 Groove Tool)
N1 G28
N2 T505 (.25 WIDE O.D. GROOVE TOOL)
N3 G97 S960 M03
N4 G54 G00 X2.1 Z0.1 M08   (Rapid to clearance point)
N5 Z-0.75   (Rapid to a start point of groove)
N6 G75 X1.75 I0.05 F0.005   (G75 Single pass O.D. grooving cycle)
N7 M09
N8 G28
N9 M30

SETTING 22 (CAN CYCLE DELTA Z) - *As the groove tool pecks deeper into the part, with each peck value of I, it pulls back a constant specified distance above the bottom of the groove created by the previous peck to break the chip. That specified distance it pulls back is defined in Setting 22.*

G75 MULTIPLE PASS O.D. GROOVING CYCLE

G75 O.D. / I.D. Grooving Cycle (Multiple Pass) Example

Tool

- - - ➤ Rapid move
←——— Feed move
▨ Groove
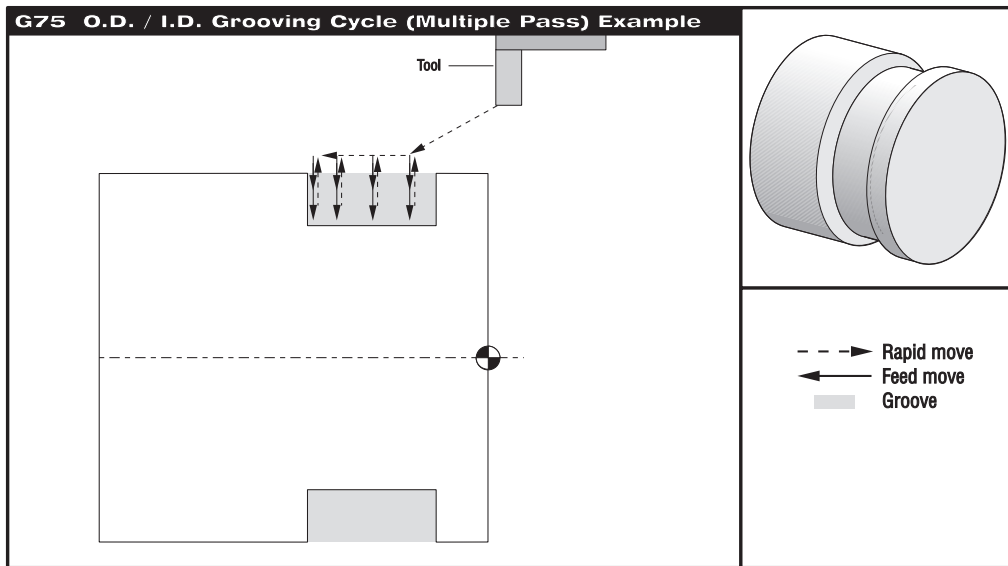
(Machine a 1. wide O.D. Groove with .25 Groove Tool)
O00110 (G75 O.D./I.D. MULTIPLE PASS GROOVING CYCLE)
N1 G28
N2 T505 (.25 WIDE O.D. GROOVE TOOL)
N3 G97 S960 M03
N4 G54 G00 X2.1 Z0.1 M08   (Rapid to front of part)
N5 Z-0.75   (Rapid to start point of groove)
N6 G75 X1.75 Z-1.5 I0.05 K0.2 F0.005   (G75 Multiple pass O.D. grooving cycle)
N7 M09
N8 G28
N9 M30

SETTING 22 (CAN CYCLE DELTA Z) - *As the groove tool pecks deeper into the part, with each peck value of I, it pulls back a constant specified distance above the bottom of the groove created by the previous peck to break the chip. That specified distance it pulls back is defined in Setting 22.*

G76   O.D./I.D. THREAD CUTTING CYCLE, MULTIPLE PASS

X* X-axis absolute thread finish point, **diameter** value

Z* Z-axis absolute distance, thread end point location

U* X-axis incremental total distance to finish point, **diameter**

W* Z-axis incremental thread length finish point

K   Thread height, **radius** value

I * Thread taper amount, **radius** value

D   First pass cutting depth

P   Thread Cutting Method P1-P4 (added in software ver. 6.05)

A* Tool nose angle, no decimal with A command

    (0 to 120 degrees, If not used then 0 degrees is assumed)

F   Feed rate (Threading feed rate, is the thread distance per revolution)

  * Indicates optional

G76 Thread Cutting Cycle, Multiple Pass

THREAD ANGLE OUT SIZE SETTING 95
THREAD ANGLE OUT ANGLE SETTING 96
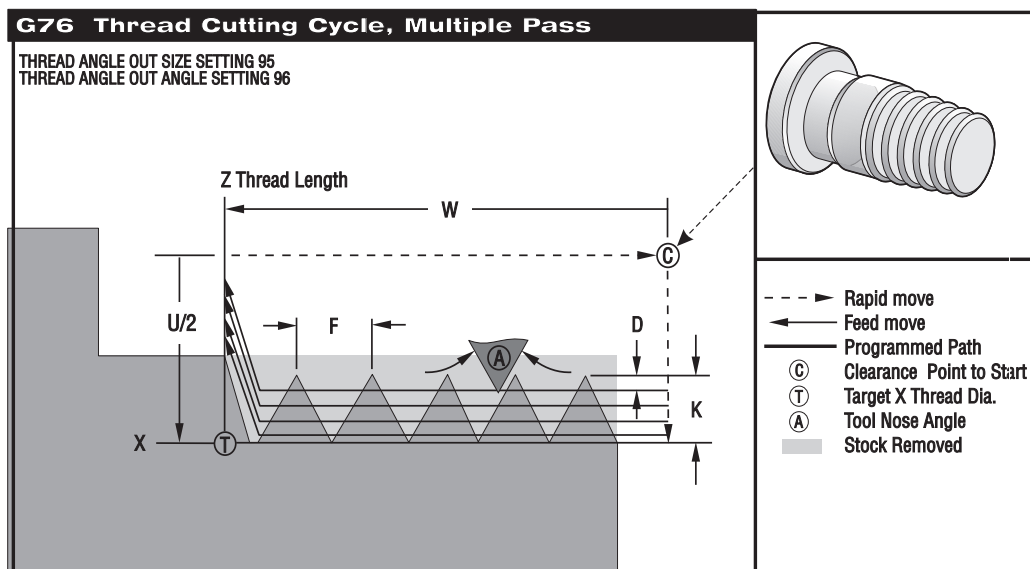
The G76 canned cycle can be used for threading both straight or tapered (pipe) threads. With G76 a programmer can easily command multiple cutting passes along the length of a thread.

The height of the thread is specified in K. The height of the thread is defined as the distance from the crest of the thread to the root. The calculated depth of the thread will be K less the finish allowance. Setting 86 (THREAD FINISH ALLOWANCE) is this stock allowance for a finish pass allowance, if needed.

The depth of the first cut of the thread is specified in D. This also determines the number of passes over the thread based on the value of K and the cutting method used.

The depth of the last cut on the thread can be controlled with Setting 99 (THREAD MINIMUM CUT). The last cut will never be less than this value. The default value is .001 inches/.01 mm.

The feed rate is the Lead of thread. The F feed rate in a G76 threading cycle is 1.0 divided by the number of threads per inch = F. (1.0 ÷ 12 tpi = F.083333)

G76 Cutting Methods

P1: FANUC SINGLE EDGE CUTTING IN, CONSTANT ONLY

THREAD MINIMUM CUT
SETTING 99

CUTTING TIP ANGLE

A

D

1
2
3
N

K

The D will
determine
the number
of passes
with K. (See
note below)

MATERIAL

FINISH ALLOWANCE
SETTING 86

Note: For the number of passes in a G76 cycle
$$K \div \text{square root of } N = D$$
(K is the thread height, N is number of passes D depth of first pass)

G76 Thread Cutting Cycle, Multiple Pass

Z

W

THREAD ANGLE OUT SIZE SETTING 95
THREAD ANGLE OUT ANGLE SETTING 96

U/2

D

A

T

I

K

C

X

- - - ▶ Rapid move
◀──── Feed move
──── Programmed Path
Ⓒ Clearance Point
Ⓧ Thread Form X Dia.
Ⓣ X & Z Target
A Tool Angle
▨ Machining stock

F feed rate is the Lead of thread
The F feed rate in a G76 cycle is 1.0 divided by the pitch# = F

The thread taper distance amount is specified with the I command. It is measured from the target end position in X and Z axis down to the point in X axis where this cycle begins and is a radius amount. A conventional O.D. taper thread will have a negative I value and a conventional I.D. taper thread will have a positive I value.
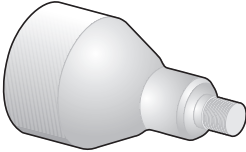
G76 MULTIPLE PASS O.D./I.D. THREADING CYCLE

G76 Multiple Pass Thread Cutting Cycle

Setup information
Setting 33: FANUC

Tool 6 - Threading tool

Tool	Offsets	Radius	Tip
6	06	0.	0

O00113 (G76 Multiple threading cycle to machine a 3/4-16 O.D. thread)
N10 G28
N20 T606 (O.D. THREADING TOOL)
N30 G97 S720 M03
N40 G54 G00 X0.85 Z1. M08   (Rapid to start point above diameter of the part)
N50 Z0.2 M23   (Z start point, chamfer at end of thread ON)
N60 G76 X0.674 Z-1.25 K0.0383 D0.0122 F0.0625 (G76 Multiple pass O.D. thread)
N70 M09
N80 G28
N90 M30

M23 Chamfer (angle out of thread) at End of Thread is ON
An angle out of thread move can improve the appearance and functionality of a thread. This
M23 commands the control to exit the thread with angle out move on a thread executed by
a G76 or G92. This M code is modal and is also the default. It remains in effect until
changed by M24. Refer to Settings 95 and 96 to control the move distance and angle. M23
will again be active, with an M30, RESET, or a POWER ON condition.

SETTING 95 (THREAD CHAMFER SIZE) - The distance of angling out of the thread.
          The distance is designated thread pitch, so that if 1.0 is in Setting 95 and
          the threading feedrate is .05, then the angle out distance will be .0500.
          The default in Setting 95 is 1.000.

SETTING 96 (THREAD CHAMFER ANGLE) - Angle out of thread chamfer. The default
          angle of 45 degrees is in Setting 96.

M24 Chamfer (angle out of thread) at End of Thread is OFF
An M24 commands the control to perform no angle out departure move at the end of a G76
or G92 threading cycle. This M code is modal. M24 is cancelled with an M23 (Chamfer at
End of Thread ON), RESET, M30 or a POWER ON condition.

G76 MACHINING AN I.D. THREAD

(This example will be in the next revision)

THREAD CHART

THREADS PER INCH (T.P.I.)	THREAD LEAD For Feedrate in Inches Per Rev.	O.D. THREAD Single Depth "K" Thread Height	I.D. THREAD Single Depth "K" Thread Height	SPINDLE SPEED MAXIMUM FOR THREADING First is machine max. RPM, then it is threading at a max. 150 I.P.M. though 125 I.P.M. is recommended.
T.P.I.	1.0 divided by the Threads Per Inch			A CALCULATED SPINDLE SPEED RPM for 125 I.P.M. is T.P.I. x 125 I.P.M. = RPM
7	.142857	.0876	.0773	875 RPM
8	.125	.0767	.0677	1000 RPM
9	.111111	.0682	.0601	1125 RPM
10	.1	.0613	.0541	1250 RPM
11	.090909	.0558	.0492	1375 RPM
12	.083333	.0511	.0451	1500 RPM
13	.076923	.0472	.0416	1625 RPM
14	.071429	.0438	.0387	1750 RPM
16	.0625	.0383	.0338	2000 RPM
18	.055556	.0341	.0301	2250 RPM
20	.05	.0307	.0271	2500 RPM
24	.041667	.0256	.0225	3000 RPM
28	.035714	.0219	.0193	3500 RPM
32	.03125	.0192	.0169	4000 RPM
36	.027778	.0170	.0150	4500 RPM
40	.025	.0153	.0135	5000 RPM

"D" VALUE CHART for O.D. THREADS
CALCULATED 1ST. PASS DEPTH ("D") FOR A DESIRED NUMBER OF PASSES

Threads Per Inch

PASSES	7TPI	8TPI	9TPI	10TPI	11TPI	12TPI	13TPI	14TPI
6	.0358	.0313	.0278	.0250	.0228	.0209	.0193	.0179
7	.0331	.0290	.0258	.0232	.0211	.0195	.0178	.0166
8	.0310	.0271	.0241	.0217	.0197	.0181	.0167	.0155
9	.0292	.0256	.0227	.0204	.0186	.0170	.0157	.0146
10	.0277	.0243	.0216	.0194	.0176	.0162	.0149	.0139
11	.0264	.0231	.0206	.0185	.0168	.0154	.0142	.0132
12	.0253	.0221	.0197	.0177	.0161	.0148	.0136	.0127
13	.0243	.0213	.0189	.0170	.0155	.0142	.0131	.0122
14	.0234	.0205	.0182	.0164	.0149	.0137	.0126	.0117
15	.0226	.0198	.0176	.0158	.0144	.0132	.0129	.0113
16	.0219	.0192	.0171	.0153	.0140	.0128	.0118	.0110

Threads Per Inch

PASSES	16TPI	18TPI	20TPI	24TPI	28TPI	32TPI	36TPI	40TPI
3	.0157	.0139	.0177	.0147	.0126	.0111	.0098	.0089
4	.0145	.0129	.0153	.0128	.0110	.0096	.0085	.0077
5	.0136	.0121	.0137	.0114	.0098	.0086	.0076	.0069
6	.0128	.0114	.0125	.0101	.0089	.0078	.0070	.0063
7	.0122	.0108	.0116	.0096	.0083	.0073	.0064	.0058
8	.0116	.0103	.0108	.0090	.0077	.0068	.0060	.0054
9	.0111	.0098	.0102	.0085	.0073	.0064	.0057	.0051
10	.0106	.0095	.0097	.0081	.0069	.0061	.0054	.0049
11	.0103	.0091	.0093	.0077	.0066	.0058	.0051	.0046
12	.0099	.0088	.0089	.0074	.0063	.0055	.0049	.0044
13	.0096	.0085	.0085	.0071	.0061	.0053	.0047	.0043

G76 O.D. THREADING EXERCISE
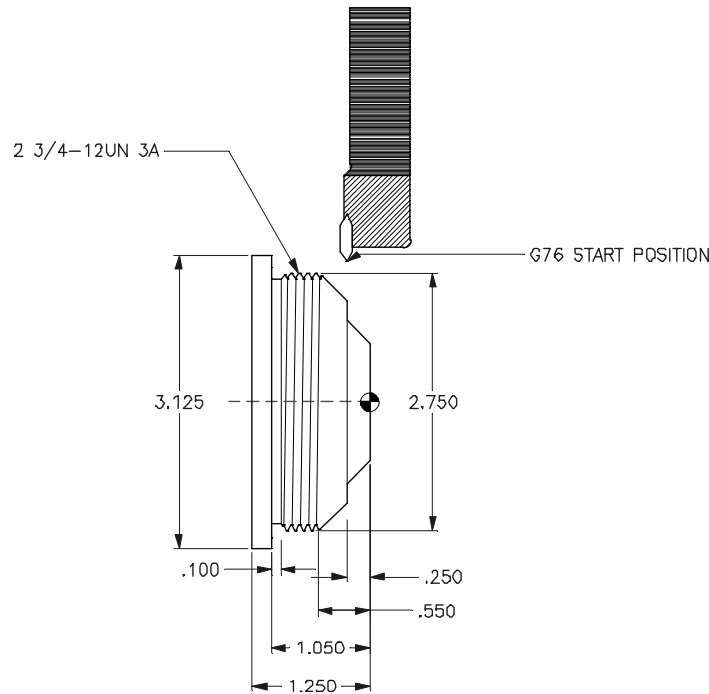
Fill in the blank lines for Tool #1 using a G76 Theading cycle command to define a multiple pass thread cycle for a 2 3/4-12UN 3A Thread.

2 3/4-12UN 3A THREAD

Major Dia. is 2.7500 / 2.7386   Minor Dia. is 2.6478
Pitch Dia. is 2.6959 / 2.6914   Thread Pitch is .083333

O00100   (O.D. Multiple Pass Threading with G76)
N1 G28
N2 T101 (O.D. THREADING TOOL)   (Tool #1 Offset #1)
N3 G97 S590 M03   (Cancel CSS 590 spindle speed)
N4 G54 G00 X_____ Z_____ M08   (Rapid X and Z above part, coolant ON)
N5 Z_____M____ (Z-.250 is start of thread, M code for NO chamfer end of thread)
N6 G____ X_____ Z_____ K_____ D_____ F_____ (Use G76 Thread cycle 10 passes)
N7 M_____   (Turn coolant off)
N8 G_____   (Return to reference point)
N9 M_____   (End of program rewind)

DRILLING, BORING, AND TAPPING CANNED CYCLES

The canned cycles used to define and simplify programming for the most common Z-axis repetitive operations such as drilling, tapping, and boring. Once selected, a canned cycle is active until canceled with a G80 code. There are six operations involved in every canned cycle:

1) Positioning of **X** and **Z** axes.
2) Rapid traverse to the reference R-plane.
3) Drilling, boring, or tapping action.
4) Operation at the bottom of the hole.
5) Retraction to the reference R-plane.
6) Rapid traverse to the initial start point.

Modal canned cycles remain in effect after they are defined and are executed in the Z axis for each positioning of X axes in a the program.   Positioning moves in X axis during a canned cycle is performed as a rapid motion. Some of the canned cycle command values can also be changed after the canned cycle is defined. The command most often changed during a Lathe canned cycle is the Z depth value. Modal canned cycles will be canceled with the G80, G01 or G00 command.

The operation of a canned cycle will vary according to whether incremental (U,W) or absolute (X,Z) is specified. Incremental motion is often useful in a canned cycle. If a loop count (Lnn code number) is defined within the block, the canned cycle will repeat that many times with an incremental U or W move between each cycle.

Canned cycle will only be executed in the Z-axis when positioning to a new X axis position during a canned cycle command.

G Code	Z Drilling motion	Operation at the bottom of hole	Retraction Z axis motion	Application
G81	feed	none	rapid	drilling
G82	feed	dwell	rapid	counter boring
G83	intermittent feed	dwell	rapid	peck drilling
G84	feed	spindle reverse	feed	tapping cycle
G85	feed	none	feed	boring cycle
G86	feed	spindle stop	rapid	boring cycle
G87	feed	spindle stop	manual retract	boring cycle
G88	feed	dwell and spindle stop	manual retract	boring cycle
G89	feed	dwell	feed	boring cycle

A canned cycle is presently limited to operations in the Z-axis. That is, only the G18 plane is allowed. This means that the canned cycle will be executed in the Z-axis whenever a new position is selected in the X axis. The following is a summary of the canned cycles that can be used on HAAS lathe controls.

G80   CANNED CYCLE CANCEL

This **G** code is modal in that it deactivates all canned cycles until a new one is selected.
Note: The use of a G00 or G01 will also cancel a canned cycle.

G81   DRILLING CANNED CYCLE

X* Absolute X-axis rapid location
Z* Absolute Z-depth (feeding to Z-depth starting from R-plane)
W* Incremental Z-depth (feeding to Z-depth starting from R-plane)
R Rapid to R-plane (where you rapid, to start feeding)
F Feed rate
   * Indicates optional

G81 Drill In/Rapid Out Canned Cycle
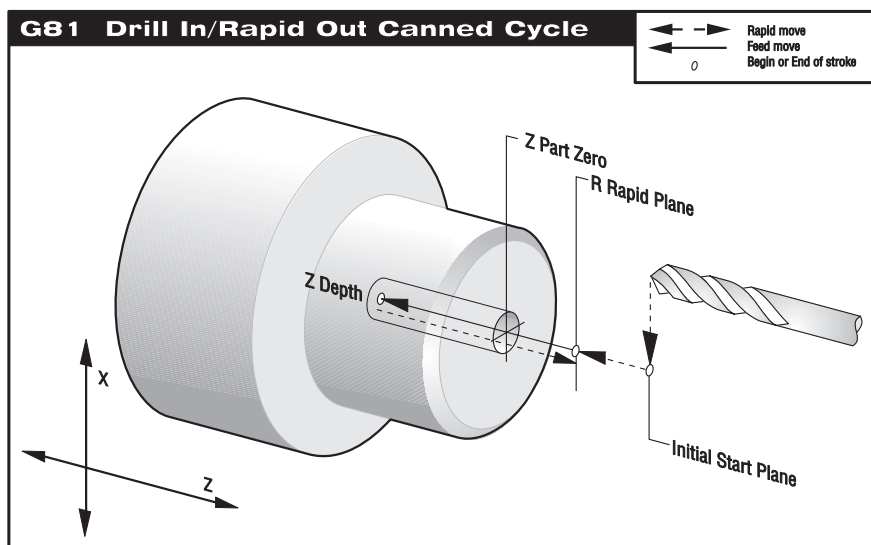
TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
1/2 DIA. DRILL	1	01	0	0

```
O00119  (G81 Drilling)
N1 G28
N2 T101 (1/2 DIA. DRILL)        (Tool 1 Offset 1)
N3 G97 S1450 M03
N4 G54 G00 X0. Z1. M08          (Rapid to Initial Start Point)
N5 G81 Z-0.625 R0.1 F0.005      (G81 Drilling Cycle)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

G82   SPOT DRILL/COUNTERBORE CANNED CYCLE

X*   Absolute X-axis rapid location
Z*   Absolute Z-depth (feeding to Z-depth starting from R-plane)
W*   Incremental Z-depth (feeding to Z-depth starting from R-plane)
P   Dwell time at Z-depth
R   Rapid to R-plane (where you rapid, to start feeding)
F   Feed rate
     * Indicates optional

G82 Drill In/Dwell/Rapid Out Canned Cycle
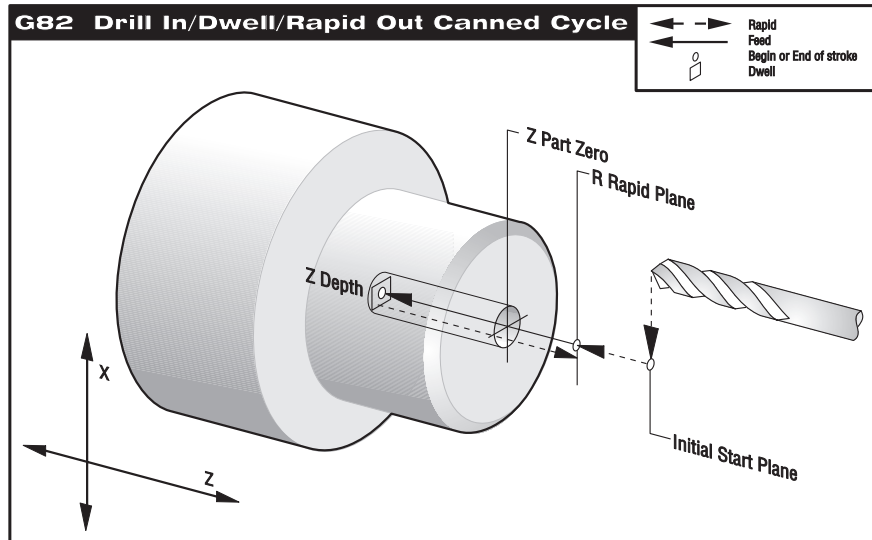
TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
1/2 DIA. FLAT BOTTOM DRILL	2	02	0	0

```
O00120 (G82 Drilling with a Dwell)
N1 G28
N2 T202 (1/2 DIA. FLAT BOTTOM DRILL)   (Tool 2 Offset 2)
N3 G97 S1450 M03
N4 G54 G00 X0. Z1. M08            (Rapid to Initial Start Point)
N5 G82 Z-0.625 P0.5 R0.1 F0.005   (G82 Drill with a Dwell at Z Depth Cycle)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

G83 DEEP HOLE PECK DRILLING CANNED CYCLE

X* Absolute X-axis rapid location
Z* Absolute Z-depth (feeding to Z-depth starting from R-plane)
W* Incremental Z-depth (feeding to Z-depth starting from R-plane)
Q* Pecking depth amount, always incremental (if I, J and K are not used)
 I* Size of first peck depth (if Q is not used)
J* Amount reducing each peck after first peck depth (if Q is not used)
K* Minimum peck depth (if Q is not used)
P Dwell time at Z-depth
R Rapid to R-plane (where you rapid, to start feeding)
F Feed rate
  * Indicates optional

G83 Peck Drilling Canned Cycle
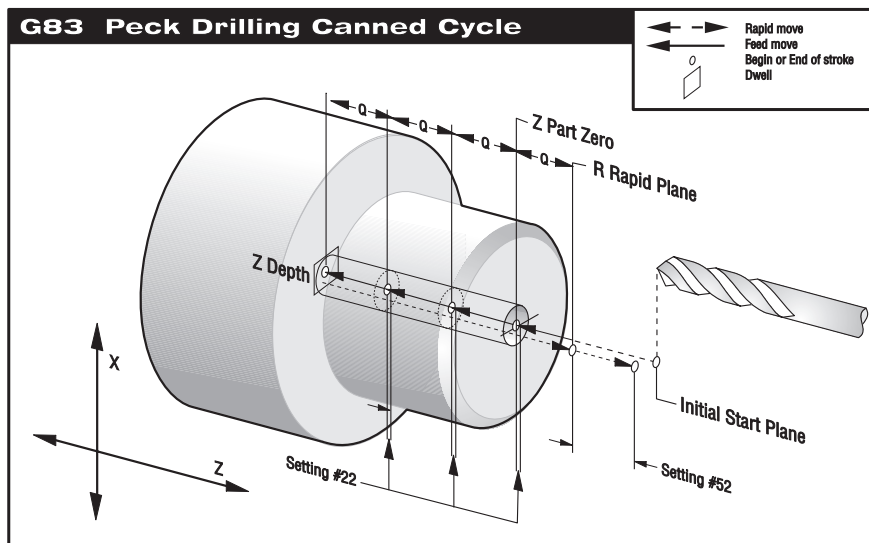
G83 USING Q TO DEFINE PECK AMOUNT

If Q is specified with the G83 command. Every pass will cut in by the Q amount, then rapid out to the R plane to clear chips and flush with coolant, than rapid into hole for the next Q peck amount until Z depth is reached.

TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
1/2 DIA. DRILL	3	03	0	0

```
O00121 (G83 Peck Drilling)
N1 G28
N2 T303 (1/2 DIA. DRILL)          (Tool 3 Offset 3)
N3 G97 S1820 M03
N4 G54 G00 X0. Z1. M08            (Rapid to Initial Start Point)
N5 G83 Z-1.5 Q0.2 R0.1 F0.005    (G83 Peck Drilling Cycle with Q)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

G83 USING I, J AND K TO DEFINE A MORE VARIABLE PECK AMOUNT

If I, J, and K are specified, a different operating mode is selected. The first pass will cut in by I depth, each succeeding peck will be reduced by the J amount, with the minimum peck cutting depth amount to be K.

After each peck the tool will then rapid out to the R plane to clear chips and flush with coolant, than rapid into hole for the next peck amount until the Z depth is reached.

    I   Optional size of first cutting depth (If Q is not used)
    J   Optional amount to reduce cutting depth each pass (If Q is not used)
    K   Optional minimum depth of cut (If Q is not used)

DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
1/2 DIA. DRILL	3	03	0	0

```
O00122 (G83 Peck Drilling)
N1 G28
N2 T303 (1/2 DIA. DRILL)           (Tool 3 Offset 3)
N3 G97 S1820 M03
N4 G54 G00 X0. Z1. M08             (Rapid to Initial Start Point)
N5 G83 Z-1.5 I0.5 J0.1 K0.2 R0.1 F0.005   (G83 Peck Drill Cycle with I,J,K)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

SETTING 22 - *As the tool pecks deeper into the hole with a G83. After each peck it rapids out to the R-plane, and then back into a constant specified distance above the bottom of the hole that was created by the previous peck. That specified distance is defined in Setting 22.*

SETTING 52 - *Changes the way G83 works when it returns to the R-plane. Most programmers set the R-plane well above the cut to insure that the chip clear motion actually allows the chips to get out of the hole but this causes a wasted motion when first drilling through this "empty" space. Or you may need to define a clearance move above the part, in which the R-plan emay be down inside a part or pocket. If Setting 52 is set to the distance required to clear chips, the R-plane can be put much closer to the part being drilled. The Z axis will be moved above the R-plane by this amount ins Setting 52.*

G84   TAPPING CANNED CYCLE

X*   Absolute X-axis rapid location
Z*   Absolute Z-depth (feeding to Z-depth starting from R-plane)
W*   Incremental Z-depth (feeding to Z-depth starting from R-plane)
R   Rapid to R-plane (where you rapid, to start feeding)
F   Feed rate
   *   Indicates optional

G84   Tapping Canned Cycle
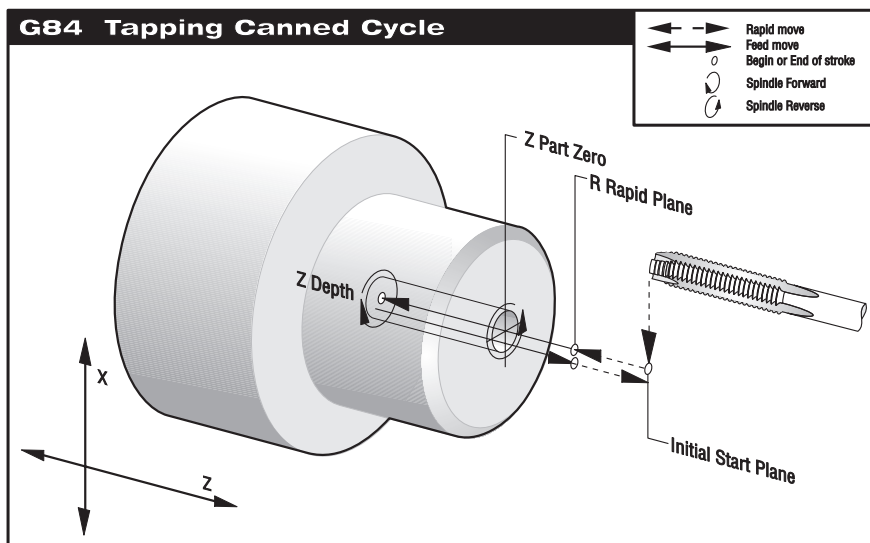
Note: This cycle will perform different if the rigid tapping option is active. With rigid tapping, the feedrate for the spindle speed must be precisely the thread pitch being cut.

The feed rate is the Lead of thread. To calculate the "F" feedrate in a G84 threading cycle divided 1.0 by the number of threads per inch = the "F" feedrate in a G84.
(Example: 1.0 ÷ 12 tpi = F.08333)

You don't need to start the spindle CW before this G84 canned cycle. The control turns it on automatically for you.

TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
3/8-16 TAP	4	04	0	0

```
O00123 (G84 Tapping)
N1 G28
N2 T404 (3/8-16 TAP)          (Tool 4 Offset 4)
N3 G97 S650 M05               (G84 will turn on the spindle for you)
N4 G54 G00 X0. Z1. M08     (Rapid to Initial Start Point)
N5 G84 Z-0.75 R0.2 F0.0625    (G84 Tapping Cycle)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

G184   REVERSE TAPPING CANNED CYCLE

X*   Absolute X-axis rapid location
Z*   Absolute Z-depth (feeding to Z-depth starting from R-plane)
W*   Incremental Z-depth (feeding to Z-depth starting from R-plane)
R   Rapid to R-plane (where you rapid, to start feeding)
F   Feed rate

   *   Indicates optional

G184   Reverse Tapping Canned Cycle
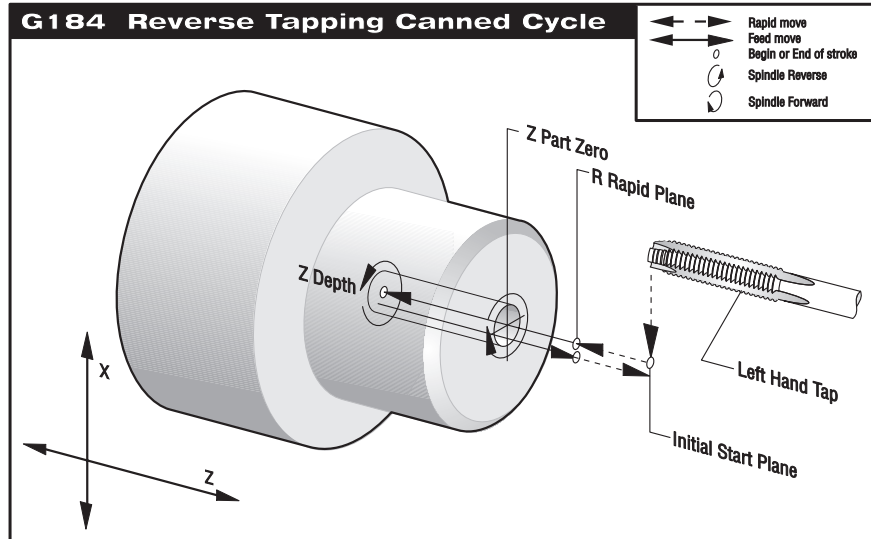
Note: This cycle will perform different if the rigid tapping option is active. With rigid tapping, the feedrate for the spindle speed must be precisely the thread pitch being cut.

The feed rate is the Lead of thread. To calculate the "F" feedrate in a G184 threading cycle divided 1.0 by the number of threads per inch = the "F" feedrate in a G184 .
(Example: 1.0 divided by 12 tpi = F.08333)

You do not need to start the spindle CW before this G184 canned cycle. The control turns it on automatically for you.

TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
9/16-12 LEFT HANDED TAP	4	04	0	0

```
O00124 (G184 Left Handed Tapping)
N1 G28
N2 T404 (9/16-12 LEFT HANDED TAP)    (Tool 4 Offset 4)
N3 G97 S650 M05                      (G184 will turn on the spindle for you)
N4 G54 G00 X0. Z1. M08               (Rapid to Initial Start Point)
N5 G184 Z-0.625 R0.2 F0.08333        (G184 Tapping Cycle)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

G85 BORE IN~BORE OUT, CANNED CYCLE

X*   Absolute X-axis rapid location
Z*   Absolute Z-depth (feeding to Z-depth starting from R-plane)
U*   Incremental X-axis rapid location
W*   Incremental Z-depth (feeding to Z-depth starting from R-plane)
R   Rapid to R-plane (where you rapid, to start feeding)
**F**   Feed rate

   * Indicates optional

G85 Bore In/Bore Out Canned Cycle
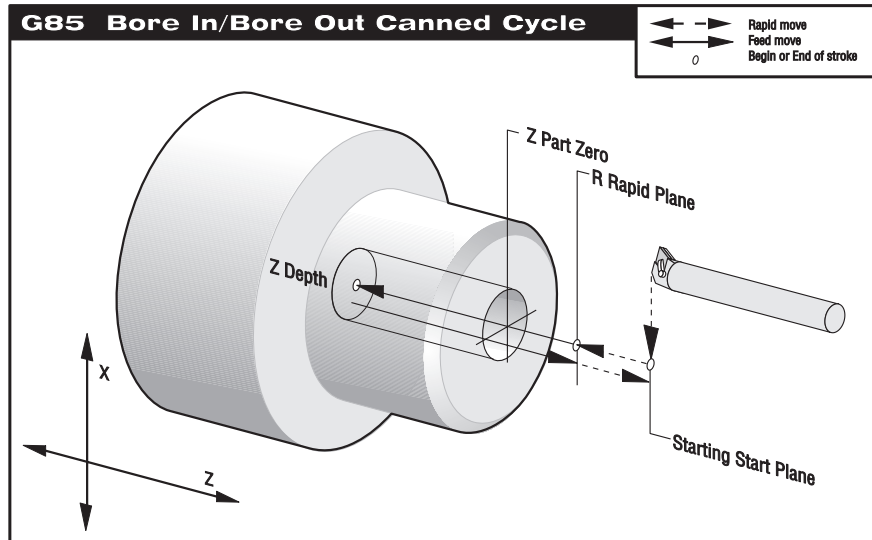
TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
BORING BAR	5	05	0	0

```
O00125 (G85 Bore In-Bore Out)
N1 G28
N2 T505 (BORING BAR)            (Tool 5 Offset 5)
N3 G97 S1820 M03
N4 G54 G00 X0.625 Z1. M08       (Rapid to Initial Start Point)
N5 G85 Z-0.5 R0.1 F0.005        (G85 Bore In, Bore Out Cycle)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

G86   BORE IN~STOP~RAPID OUT CANNED CYCLE

X*   Absolute X-axis rapid location
Z*   Absolute Z-depth (feeding to Z-depth starting from R-plane)
U*   Incremental X-axis rapid location
W*   Incremental Z-depth (feeding to Z-depth starting from R-plane)
R   Rapid to R-plane (where you rapid, to start feeding)
F   Feed rate
    * Indicates optional

G86 Bore/Stop/Rapid Out Canned Cycle
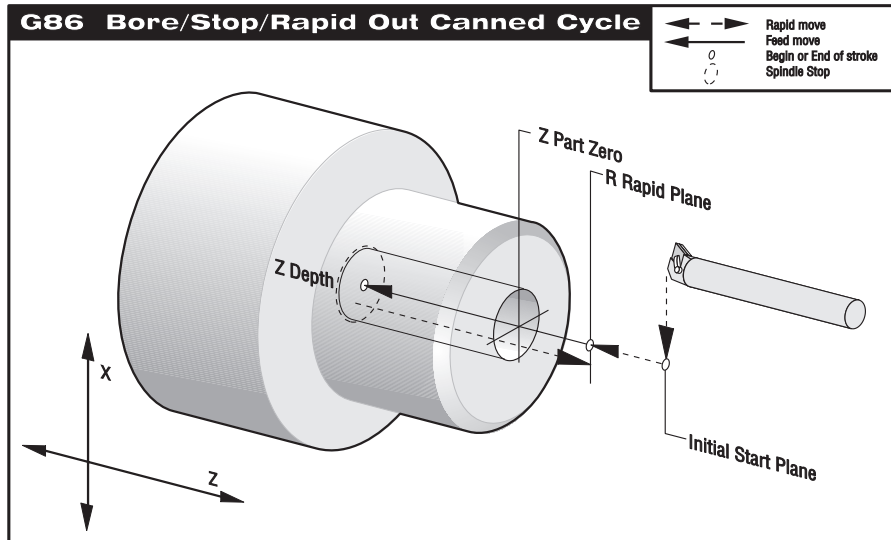
TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
BORING BAR	6	06	0	0

```
O00126  (G86 Bore In-Stop-Rapid Out)
N1 G28
N2 T606 (BORING BAR)              (Tool 6 Offset 6)
N3 G97 S1820 M03
N4 G54 G00 X0.325 Z1. M08        (Rapid to Initial Start Point)
N5 G86 Z-0.55 R0.1 F0.005        (G86 Bore In, Stop, Rapid Out Cycle)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

G87 BORE IN~MANUAL RETRACT~CANNED CYCLE

X* Absolute X-axis rapid location
Z* Absolute Z-depth (feeding to Z-depth starting from R-plane)
U* Incremental X-axis rapid location
W* Incremental Z-depth (feeding to Z-depth starting from R-plane)
R Rapid to R-plane (where you rapid, to start feeding)
F Feed rate

  * Indicates optional

G87 Bore In/Stop/Manual Retract Cycle
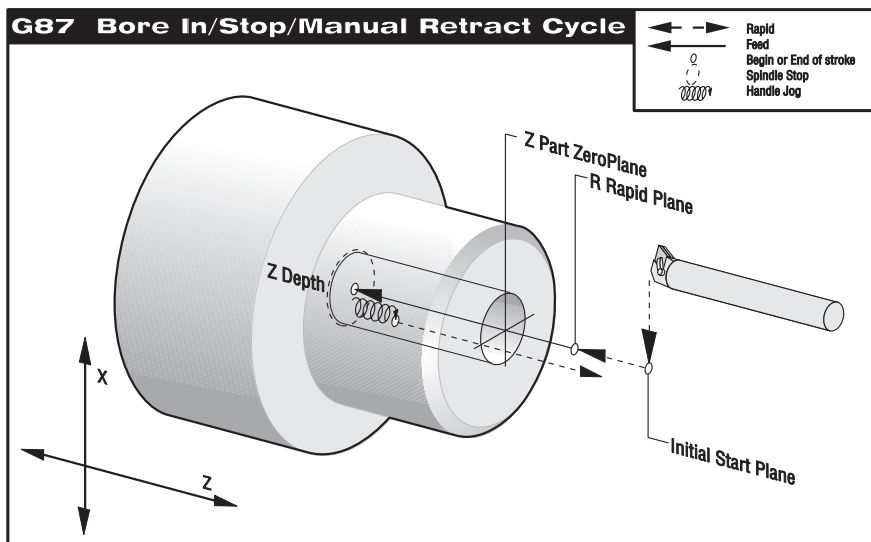
TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
BORING BAR	7	07	0	0

```
O00127 (G87 Bore In-Stop-Manual Handle Out)
N1 G28
N2 T707 (BORING BAR)              (Tool 7 Offset 7)
N3 G97 S1820 M03
N4 G54 G00 X0.25 Z1. M08          (Rapid to Initial Start Point)
N5 G87 Z-0.625 R0.1 F0.005        (G87 Bore In, Stop, Manual Retract Cycle)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

G88   BORE IN~DWELL~MANUAL RETRACT CANNED CYCLE

X*   Absolute X-axis rapid location
Z*   Absolute Z-depth (feeding to Z-depth starting from R-plane)
U*   Incremental X-axis rapid location
W*   Incremental Z-depth (feeding to Z-depth starting from R-plane)
P*   Dwell time at Z-depth
R   Rapid to R-plane (where you rapid, to start feeding)
F   Feed rate
    * Indicates optional

G88 Bore In/Dwell/Manual Retract Out Canned Cycle
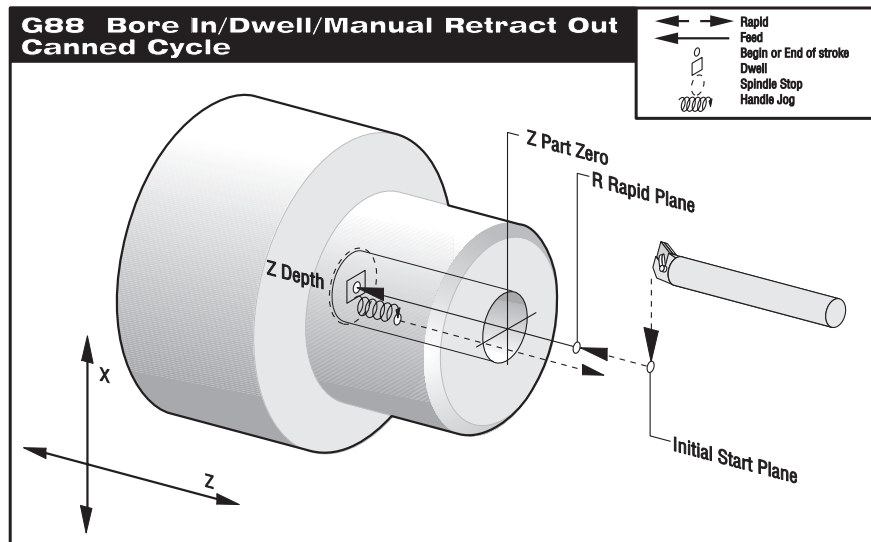
TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
BORING BAR	8	08	0	0

O00128 (G88 Bore In-Dwell-Manual Handle Out)
N1 G28
N2 T808 (BORING BAR)   (Tool 8 Offset 8)
N3 G97 S1820 M03
N4 G54 G00 X0.875 Z1. M08   (Rapid to Initial Start Point)
N5 G88 Z-0.5 P0.5 R0.1 F0.005   (G88 Bore In, Dwell, Bore Out, Manual Retract)
N6 G80 G00 Z1. M09
N7 G28
N8 M30

G89   BORE IN-DWELL-BORE OUT CANNED CYCLE

X*   Absolute X-axis rapid location
Z*   Absolute Z-depth (feeding to Z-depth starting from R-plane)
U*   Incremental X-axis rapid location
W*   Incremental Z-depth (feeding to Z-depth starting from R-plane)
P*   Dwell time at Z-depth
R   Rapid to R-plane (where you rapid, to start feeding)
F   Feed rate
     * Indicates optional
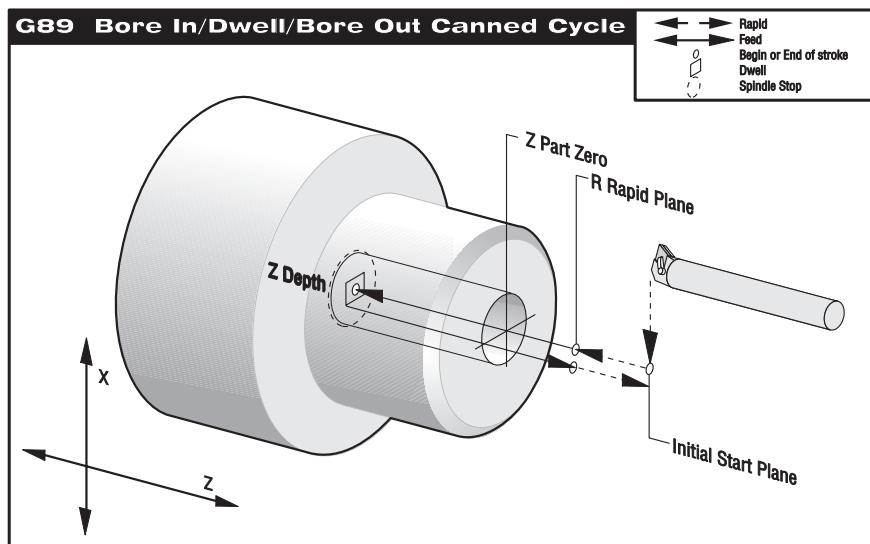
TOOL DESCRIPTION	TOOL	OFFSET	RADIUS	TIP
BORING BAR	9	09	0	0

```
O00129 (G89 Bore In-Dwell-Bore Out)
N1 G28
N2 T909 (BORING BAR)                (Tool 9 Offset 9)
N3 G97 S1820 M03
N4 G54 G00 X0.25 Z1. M08            (Rapid to Initial Start Point)
N5 G89 Z-0.625 P0.5 R0.1 F0.005     (G89 bore In, Dwell, Bore Out Cycle)
N6 G80 G00 Z1. M09
N7 G28
N8 M30
```

G90   O.D./I.D. TURNING CYCLE

X *   Absolute X-axis target location
Z *   Absolute Z-axis target location
U * Incremental X-axis target distance, diameter
W * Incremental Z-axis target distance
I *   Distance and direction of X axis taper, radius value
F   Feed rate
  *   Indicates optional
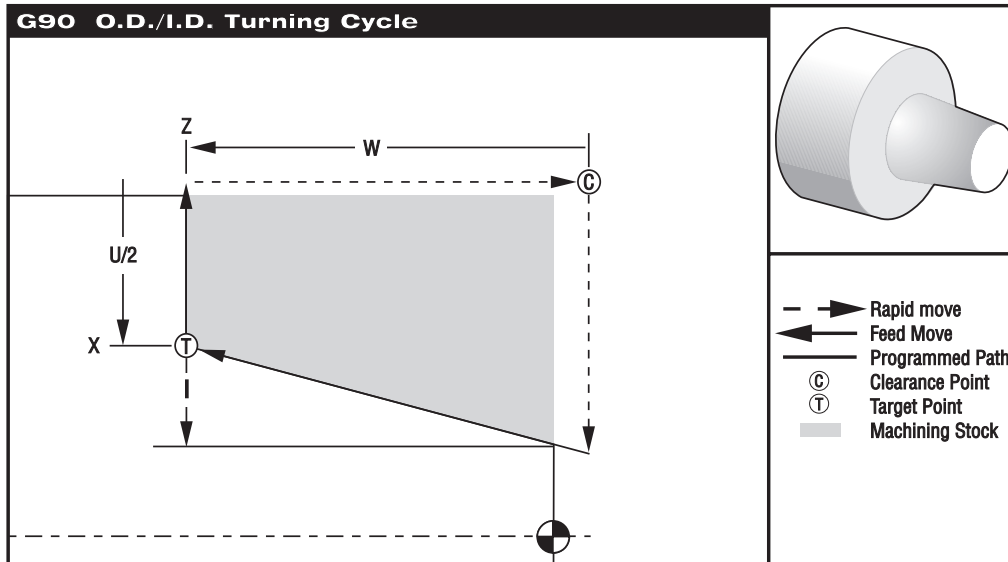
G90 is a modal canned cycle. It can be used for simple turning. Since it is modal, you can do multiple passes for turning by just specifying a new X location for successive passes.

Straight turning cuts can be made by just specifying X, Z and F. By adding I a taper cut can be made. The amount of taper is defined with the I value added to the X value target point.

Any of the four ZX quadrants can be programmed by varying U, W or X, and Z. The taper can be positive or negative. Selecting the sign direction is not intuitive.

EXAMPLE: G90 MODAL TURNING CYCLE WITH TNC
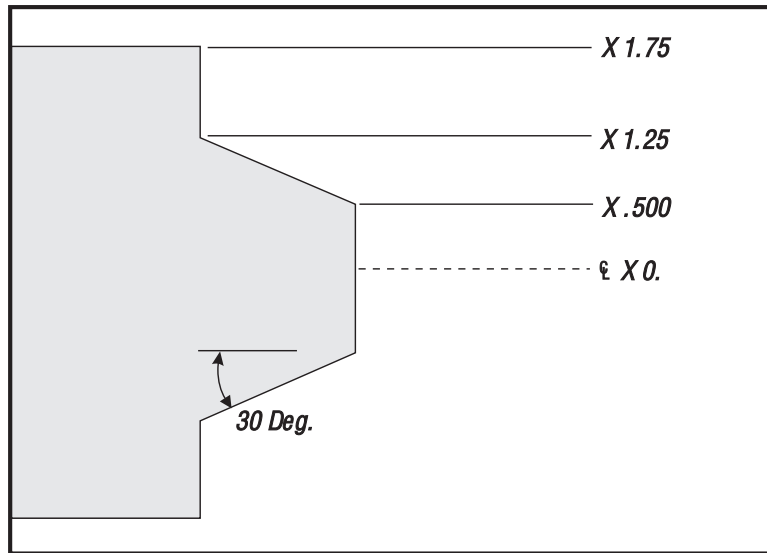
This example uses tool nose compensation with a G90 modal rough turning cycle.

O00131 (G90 Modal Turning with TNC)
N11 G28
N12 T101 (O.D. TURNING TOOL)
N13 G50 S3000
N14 G97 S480 M03
N15 G54 G00 X1.85 Z1. M08   (Rapid to Start Point)
N16 G96 S390
N17 Z0.1
N18 G90 G42 X1.65 Z-0.6495 I-0.375 F0.006 (Rough 30 Deg. angle to X2.3476
N19 X1.55   (Additional Pass)   Dia. using G90 and TNC)
N20 X1.45   (Additional Pass)
N21 X1.35   (Additional Pass)
N22 X1.25   (Additional Pass)
N23 G00 G40 X3.1 Z1. M09   (TNC Departure)
N24 M05
N25 G28
N26 M30

G92 THREAD CUTTING CYCLE

X* Absolute X-axis target location
Z* Absolute Z-axis target location
U* Incremental X-axis target distance, diameter
W* Incremental Z-axis target distance
I * Distance and direction of X axis taper, radius value
F Feed rate

* Indicates optional
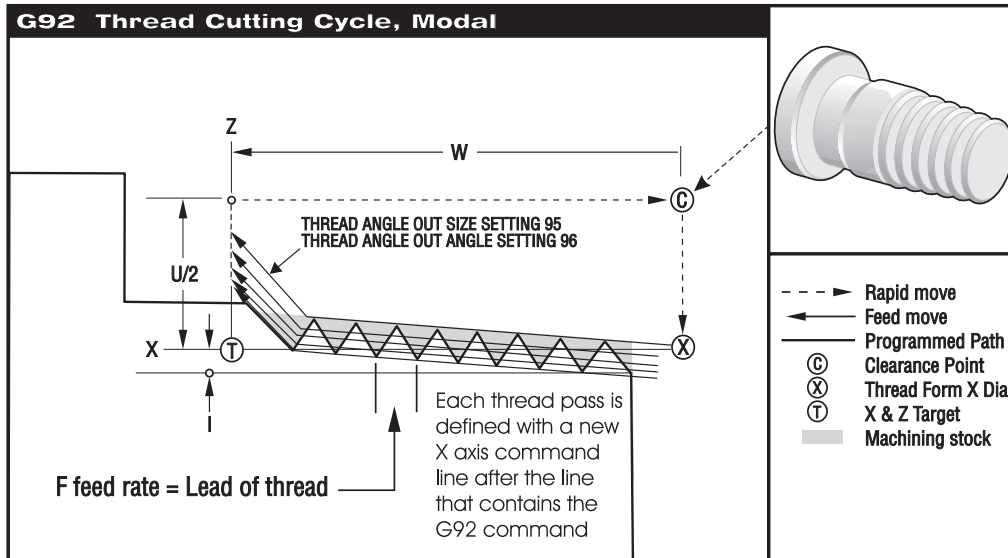
G92 is a modal canned cycle. It can be used for simple threading. Since it is modal, you can do multiple passes for threading by just specifying a new X location for successive passes.

Straight threads can be made by just specifying X, Z and F. By adding I a pipe or taper thread can be cut. The amount of taper is defined with the I value added to the X value target point.

At the end of the thread, an automatic chamfer is executed before reaching the target default for this chamfer is one thread at 45 degrees. These values can be changed with Setting 95 and Setting 96.

Any of the four ZX quadrants can be programmed by varying U, W, X, and Z. The taper can be positive or negative. Selecting the sign direction is not intuitive. The figure shows a few examples of the values required for machining in each of the four quadrants.

EXAMPLE: G92 MODAL THREADING CYCLE

G92 Thread Cutting Example - Thread a 1"-12
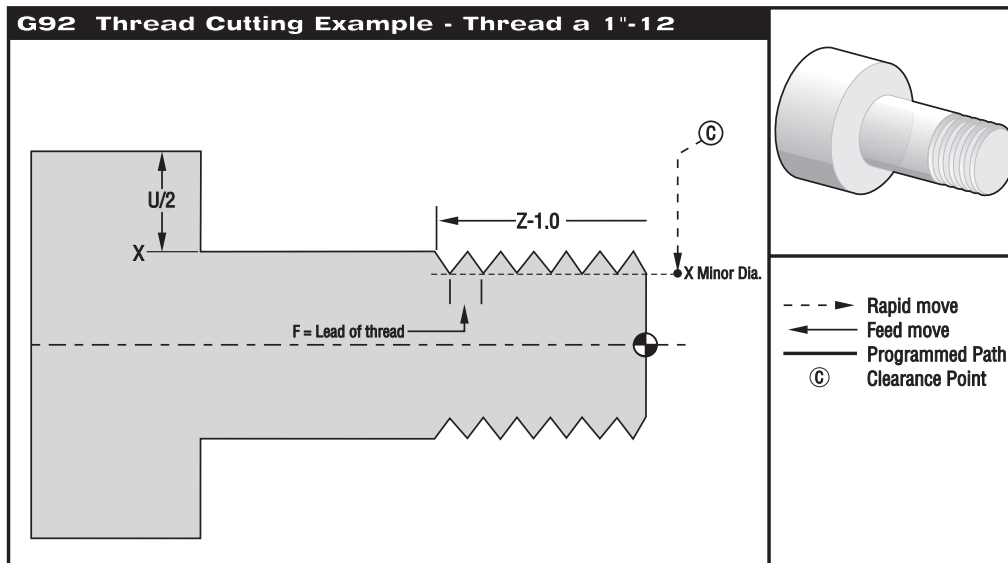
O00133 (G92 Modal Threading)
N10 (1.0-12UN Thread)
N11 G28
N12 T404 (O.D. THREADING TOOL)
N13 G97 S825 M03
N14 G54 G00 X1.1 Z1. M08 (Rapid to Start Point)
N15 Z0.25
N16 G92 X.98 Z-1.05 F0.08333 M23   (First Pass of a G92 O.D. Thread Cycle)
N17 X.96   (Additional Pass)
N18 X.94   (Additional Pass)
N19 X.935   (Additional Pass)
N20 X.93   (Additional Pass)
N21 X.925   (Additional Pass)
N22 X.9225   (Additional Pass)
N23 X.92   (Additional Pass)
N24 X.9175   (Additional Pass)
N25 X.9155   (Additional Pass)
N26 X.915   (Additional Pass)
N27 X.9148   (Additional Pass)
N28 G00 X1.1 Z1. M09
N29 M05
N30 G28
N31 M30

G94   END FACE CUTTING CYCLE

X* Absolute X-axis target location
Z* Absolute Z-axis target location
U* Incremental X-axis target distance, diameter
W* Incremental Z-axis target distance
K* Distance and direction of Z axis coning
F Feed rate
  * Indicates optional
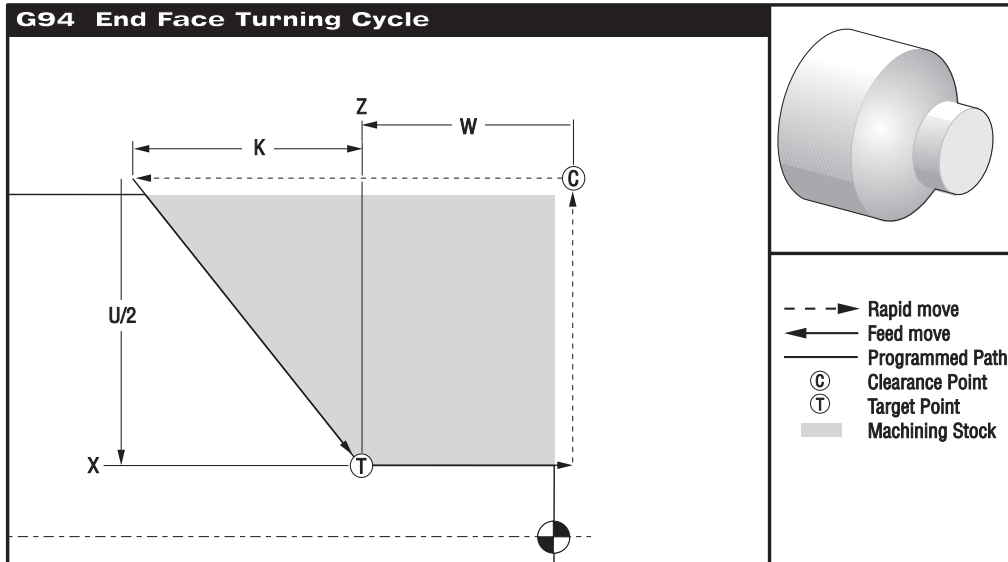
G94 is a modal canned cycle. It can be used for simple end facing. Since it is modal, you can do multiple passes for facing by just specifying a new Z location for successive passes.

Straight end facing cuts can be made by just specifying X, Z and F. By adding K a conical face can be cut. The coning amount is defined with the K value that is added to the Z value target point.

Any of the four ZX quadrants can be programmed by varying U, W, X, and Z. The coning can be positive or negative. Selecting the sign direction is not intuitive.

EXAMPLE: G94 MODAL END FACING WITH TNC
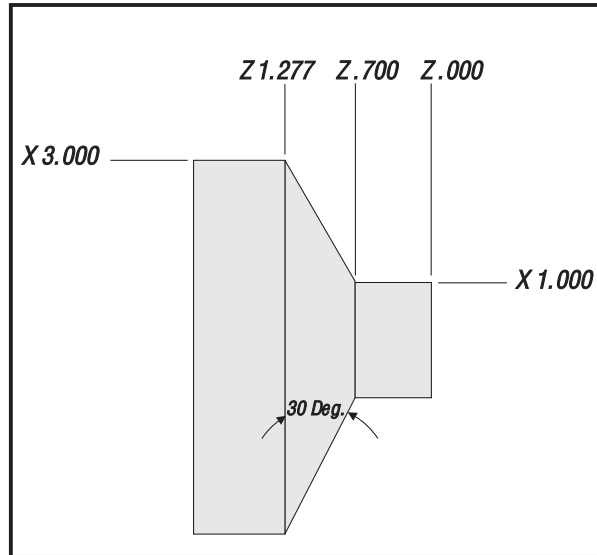
This example uses tool nose compensation with a G94 modal rough facing cycle.

```
O00135  (G94 Modal End Facing with TNC example)
N11 G28
N12 T101  (O.D. FACING TOOL)
N13 G50 S3000
N14 G97 S480 M03
N15 G54 G00 X3.1 Z1. M08        (Rapid above part)
N16 G96 S390
N17 Z.1  (rapid to start point)
N18 G94 G41 X1.0 Z-0.3 K-0.5774 F0.01    (Rough 30 Deg. angle to  X1. and
N19 Z-0.4               (Additional Pass)        Z-0.7 using G94 and TNC)
N20 Z-0.5               (Additional Pass)
N21 Z-0.6               (Additional Pass)
N22 Z-0.69              (Additional Pass)
N23 Z-0.7               (Additional Pass)
N24 G40 G00 X3.1 Z1. M09        (Cancel TNC)
N25 M05
N26 G28
N27 M30
```

M CODES (MISCELLANEOUS FUNCTIONS)

M CODE SUMMARY

Only one **M** code may be programmed per block of a program. All **M** codes are effective or cause an action to occur at the end of the block. However, when Parameter 278 bit "CNCR SPINDLE" is set to 1, spindle start occurs at the beginning of a block and will be concurrent with axis motion. The settings listed next to the M code in some way effects that M-code. The M-codes with double asterisk (**) are machine options.

M00	Stop Program (Setting 42,101)
M01	Optional Program Stop (Setting 17)
M02	Program End
M03	Spindle Forward (S) (Setting 144)
M04	Spindle Reverse (S) (Setting 144)
M05	Spindle Stop
M08	Coolant On (Setting 32)
M09	Coolant Off
M10	Chuck Clamp (Setting 92)
M11	Chuck Unclamp (Setting 92)
M12**	Auto Air Jet On (P)
M13**	Auto Air Jet Off
M14**	Main Spindle Clamp
M15**	Main Spindle Unclamp
M17	Turret Rotation Always Forward (T) (Setting 97)
M18	Turret Rotation Always Reverse (T) (Setting 97)
M19**	Orient Spindle (P, R)
M21**	Tailstock Advance (Setting 93, 94, 106, 107, 121, 145)
M22**	Tailstock Retract (Setting 105)
M23	Angle Out of Thread ON (Setting 95, 96)
M24	Angle Out of Thread OFF
M30	Program End and Reset (Setting 2, 39, 56, 83)
M31	Chip Auger Forward (Setting 114, 115)
M33	Chip Auger Stop
M36**	Parts Catcher On
M37**	Parts Catcher Off
M41	Spindle Low Gear Override
M42	Spindle High Gear Override
M43	Turret Unlock (Service Use Only)
M44	Turret Lock (Service Use Only)
M51-M58	Optional User M-Code Set
M59	Output Relay Set (N)
M61-M68	Optional User M-Code Clear
M69	Output Relay Set (N)
M76	Program Displays Inactive

M77   Program Displays Active
M78   Alarm if Skip Signal Found
M79   Alarm if Skip Signal Not Found
M85** Automatic Door Open (Setting 51, 131)
M86** Automatic Door Close (Setting 51, 131)
M88** High Pressure Coolant On (Setting 32)
M89**   High Pressure Coolant Off
M93**   Axis Position Capture Start (P, Q)
M94**   Axis Position Capture Stop
M95   Sleep Mode (hh:mm)
M96   Jump if no Input (P,Q)
M97   Local Sub-Routine Call (P, L)
M98   Sub Program Call (P, L)
M99   Sub Program/Sub-Routine Return Or Loop (P) (Setting 118)
M109** Interactive User Input (P)
M110** Tailstock Chuck Clamp (Setting 122)
M111** Tailstock Chuck Unclamp (Setting 122)
M119** Sub-Spindle Orient (P, R)
M121-128   Optional User M Code Interface with M-Fin Signal
M133**   Live Tool Drive Forward (P)
M134**   Live Tool Drive Reverse (P)
M135**   Live Tool Drive Stop
M143**   Sub-Spindle Forward (P)
M144**   Sub-Spindle Reverse (P)
M145**   Sub-Spindle Stop
M154**   C-Axis Engage (Setting 102)
M155**   C-Axis Disengage
M164**   Rotate APL Grippers to "n" Position (P)
M165**   APL Gripper 1 Open (Raw Material)
M166**   APL Gripper 1 Close (Raw Material)
M167**   APL Gripper 2 Open (Finished Material)
M168**   APL Gripper 2 Close (Finished Material)
        **Options.

M CODE DETAILED DESCRIPTION

M00 STOP PROGRAM (SETTING 42)
The M00 code is used to stop a program. It also stops the spindle and turns off the coolant and stops interpretation look-ahead processing. The program pointer will advance to the next block and stop. A cycle start will continue program operation from the next block.

M01 OPTIONAL PROGRAM STOP (SETTING 17)
The M01 code is identical to M00 except that it only stops if OPTIONAL STOP is turned on from the front panel. A cycle start will continue program operation from the next block.

M02 PROGRAM END
The M02 code will stop program operation the same as M00 but does not advance the program pointer to the next block. It will not reset the program pointer to the beginning of the program as an M30 does.

M03 SPINDLE FORWARD (S) (SETTING 144)
The M03 code will start the spindle in a clockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed. If bit 31 of parameter 209 (CNCR SPINDLE) is set to 1, then this command is executed at the beginning of the block execution rather than the end as most M codes are.

M04 SPINDLE REVERSE (S) (SETTING 144)
The M04 code will start the spindle in a counterclockwise direction at whatever speed was previously set. The block will delay until the spindle reaches about 90% of commanded speed. If bit 31 of parameter 209 (CNCR SPINDLE) is set to 1, then this command is executed at the beginning of the block execution rather than the end as most M codes are.

M05 SPINDLE STOP
The M05 code stops the spindle. The program is delayed until spindle slows below 10 RPM.

M08 COOLANT ON (SETTING 32)
The M08 code will turn the coolant on. Note that M codes are performed at the end of a block; so that if a motion command is in the same block, the coolant is turned on after the motion. The low coolant status is only checked at the start of a program so a low coolant condition will not stop a program that is already running.

M09 COOLANT OFF
The M09 code will turn off the coolant supply.

M10 CHUCK CLAMP (SETTING 92)
The M10 code is used to Clamp the chuck. It is only used when M11 is used to unclamp the chuck. A delay is provided to allow the chuck time to clamp prior to the next block executing. This is parameter 249 CHUCK CLAMP DELAY and is specified in milliseconds. The default is 500 for .5 seconds.

M11 CHUCK UNCLAMP (SETTING 92)

The M11 code will unclamp the chuck. A delay is provided to allow the chuck time to unclamp prior to the next block executing. This is parameter 250 CHUCK UNCLAMP DELAY and is specified in milliseconds. The default is 500 for .5 seconds. If the spindle is turning, it will be stopped before the chuck is unclamped.

M12 AUTO AIR JET ON (P) (OPTION)

M13 AUTO AIR JET OFF

M14 MAIN SPINDLE CLAMP (OPTION) (SETTING 92)

This M code will clamp the main spindle.

M15 MAIN SPINDLE UNCLAMP (SETTING 92)

This M code will unclamp the main spindle.

M17 TURRET ROTATION ALWAYS FORWARD (SETTING 97)

The M17 code is a modal M code that forces the turret to rotate in the forward direction when a tool change is made. Whereas most M codes are invoked as the last action in a block, M17 works concurrently with other commands in the same block. This means that the following command will cause the tool turret to advance in the forward direction to tool #1.

        N1 T101 M17;

Because M17 is modal, any subsequent T command will cause the turret to rotate in the forward direction to the commanded tool. Select M17/M18 in Setting 97 to activate this feature.

M18 TURRET ROTATION ALWAYS REVERSE (SETTING 97)

The M18 code is a modal M code that forces the tool turret to rotate in the reverse direction when a tool change is made. Whereas most M codes are invoked as the last action in a block, M18 works concurrently with other commands in the same block. This means that the following command will cause the tool turret to advance in the reverse direction to tool #10.

        N1 T1010 M18;

Because M18 is modal, any subsequent T command will cause the turret to rotate in the forward direction to the commanded tool. Select M17/M18 in Setting 97 to activate this feature.

M19 SPINDLE ORIENT (P,R) (OPTION)

This feature will only work on a vector drive lathe. The M19 command orients the spindle to the zero position. A P value can be added that will cause the spindle to be oriented to a particular position (in degrees.) For example, M19 P270 will orient the spindle to 270 degrees. Note that P270.001 (or any other fraction) will be truncated to 270. Also a P365 will be treated as P5.

M21 TAILSTOCK ADVANCE (OPTION) (SETTING 106, 107, 121)
The M21 code uses Settings 105, 106 and 107 to advance to the tailstock HOLD POINT.

M22 TAILSTOCK RETRACT (SETTING 105)
The M22 code uses Setting 107 to withdraw the tailstock to the RETRACT POINT.

M23 ANGLE OUT OF THREAD, ON (SETTING 95, 96)
The M23 code commands the control to execute an angle out chamfer at the end of a thread executed by G76 or G92. This M code is modal. It remains in effect until changed by M24. Refer to Settings 95 and 96 to control the angle out chamfer size and angle.

M24 ANGLE OUT OF THREAD, OFF
The M24 code command the control to perform no angle out at the end of a G76 or G92 threading cycle. This M code is modal. M24 may be cancelled with an M23, Reset, or a POWER ON condition.

M30 PROGRAM END AND RESET (SETTING 2, 39, 56, 83)
The M30 code is used to stop a program. It also stops the spindle and turns off the coolant. The program pointer will be reset to the first block of the program and stop. The parts counters displayed on the current commands display are also added to. M30 will also cancel tool length offsets.

M31 CHIP AUGER FORWARD (SETTING 114, 115)
M31 starts the chip auger motor in the forward direction. The forward direction is defined as the direction that the conveyor must move to transport chips out of the work cell. If the auger motor is on, then the conveyor will be stopped and restarted in the forward direction. If setting 114 has a number of minutes entered in, it will cycle after that many minutes. And it will stay on for the number of minutes in setting 115 before it turns back off until it cycles again. The conveyor will not turn if the door is open. This may be overridden by setting bit 17 of parameter 209 (CNVY DR OVRD).

M33 CHIP AUGER STOP
M33 stops chip conveyor motion.

M36 PARTS CATCHER ON (OPTION)
This code is used to activate the optional Parts Catcher. It will rotate the Parts Catcher counterclockwise, into position to catch a part. A G04 code might need to be programmed between an M36 and M37, to pause the catcher pan in the open position long enough to catch the part.

M37 PARTS CATCHER OFF
This code is used to deactivate the optional Parts Catcher. It will rotate the Parts Catcher clockwise, out of the work envelope.

M41 SPINDLE LOW GEAR OVERRIDE

The M41 code is used to select low gear. The spindle will come to a stop when changing gears. M41 is ignored if there is no gear box. The machine will remain in it's current gear even after the machine is powered off. When the machine is powered up, it will be in the same gear as when it was powered off.

M42 SPINDLE HIGH GEAR OVERRIDE

The M42 code is used to select high gear. The spindle will come to a stop when changing gears. M42 is ignored if there is no gear box. The machine will remain in it's current gear even after the machine is powered off. When the machine is powered up, it will be in the same gear as when it was powered off.

M43 TURRET UNLOCK

For Service use only.

M44 TURRET LOCK

For Service use only.

M51-M58 OPTIONAL USER M-CODE SET

The M51 through M54 codes are optional for user interfaces. They will activate one of relays 25 through 28 and leave it active. These are the same relays used for M121-M124. Use M61-M64 to turn these off. The RESET key will turn off all of these relays.

M59 OUTPUT RELAY SET(N)

This M-code directly enables a discrete output relay. The syntax for its usage is M59 Pnn where "nn" specifies the number of the relay being turned ON. An M59 command can be used to turn ON any of the discrete output relays in the range of 1100 and 1155. For example, M59 P1103 does the same thing as #1103=1 except that it is processed as an M code (processed when coming out of the queue instead of going into the queue.) To turn off a relay, use M69. WARNING: This mcode is not a general-purpose M-code. It is a special function, and its use should be strictly monitored. Any misuse of this function will cause a serious damage to the machine and will void any warranty.

M61-M68 OPTIONAL USER M OFF

The M61 through M68 codes are optional for user interfaces. They will deactivate one of relays 25 through 28. These are the same relays used for M121-M128.

M69 OUTPUT RELAY CLEAR (N)

This M-code directly disables a discrete output relay. The syntax for its usage is M69 Pnn where "nn" specifies the number of the relay being turned OFF. An M69 command can be used to turn OFF any of the discrete output relays in the range of 1100 and 1155. For example, M69 P1103 does the same thing as #1103=0 except that it is processed as an M code (processed when coming out of the queue instead of going into the queue.) To turn on a relay, use M59. WARNING: This mcode is not a general-purpose M-code. It is a special function, and its use should be strictly monitored. Any misuse of this function will cause a serious damage to the machine and will void any warranty.

M76   PROGRAM DISPLAYS INACTIVE

This code is used to disable the updating of the screen displays. It is not necessary for machine performance.

M77   PROGRAM DISPLAYS ACTIVE

This code is used to enable the updating of the screen displays. It is only used when M76 is in use.

M78   ALARM IF SKIP SIGNAL FOUND

This code is used to generate an alarm if the previous skip function actually got the skip signal. This is usually used when a skip signal is not expected and may indicate a probe crash. This code can be placed in a block with the skip function or in any subsequent block. The skip functions are G31, G36, and G37.

M79   ALARM IF SKIP SIGNAL NOT FOUND

This code is used to generate an alarm if the previous skip function did not actually get the skip signal. This is usually done when the absence of the skip signal means a positioning error of a probe. This code can be placed in a block with the skip function or in any subsequent block. Skip functions are G31, G36, and G37.

M85   AUTOMATIC DOOR OPEN (OPTION) (SETTING 131)

This code opens the optional automatic door. The automatic door can be used to load parts with a parts loader. Lathes equipped with an automatic door use discrete output and input labeled A DOOR to open, to close and to detect if the door is open. When M85 is invoked, the A DOOR output is set high. If the input A DOOR is not received with the time allotted by Parameter 251 (A DOOR OPEN ERRTIME), Alarm 127 (DOOR FAULT) is generated. The units of Parameter 251 are milliseconds.

M86   AUTOMATIC DOOR CLOSE (OPTION) (SETTING 131)

This code closes the optional automatic door. When closing the automatic door, the A DOOR output is set low and control execution continues with delay.

M88   HIGH PRESSURE COOLANT ON (OPTION) (SETTING 32)

M89   HIGH PRESSURE COOLANT OFF

M93   AXIS POSITION CAPTURE START (OPTION)

M94   AXIS POSITION CAPTURE STOP

M95   SLEEP MODE (hh:mm)

Sleep mode is essentially a long dwell but with power to the servo motors turned off. Sleep mode can be used when the user wishes his machine to begin warming itself up early in the morning and be ready for use upon his arrival. The format of the M95 command is:

M95 (hh:mm)

The comment immediately following the M95 must contain the hours and minutes that the machine is to sleep for. For example, if the current time were 6pm and the user wanted the machine to sleep until 6:30am the next morning, the following command would be used:

M95 (12:30)

Up to 99 hours can be specified thus allowing the machine to sleep for over 4 days. If the time is specified using an incorrect format, alarm 324 DELAY TIME RANGE ERROR will be generated. When the machine enters sleep mode, it will turn off power to the servos motors and display the following message:

        HAAS
      SLEEP MODE
   REMAINING TIME nnn MIN.

The message will be re-displayed in a different position on the screen each second so that the user can see at a glance that the machine is sleeping. This has the advantage of preventing the message from being "burned" into one spot on the screen.
When less than one minute of sleep time remains, the message will change to:

    REMAINING TIME nn SEC.

If the user presses any key or opens the door, sleep mode will be cancelled, power will be restored to the servo motors and the active program will wait at the block following the M95 until the user presses the Cycle Start key.
For the last 30 seconds of the sleep time, the machine will beep and display an additional message:

    WAKE UP IN nn SECONDS

When the sleep time has elapsed, power will be restored to the servo motors and the active program will continue at the block following M95.

M96 JUMP IF NO SIGNAL (P, Q)
   P   Block to branch to when conditional test succeeds
   Q   Discrete input to test, 0..31

This code is used to test a discrete input for 0 status. When this block is executed and the input signal specified by Q is 0, a branch to the block specified by P is performed. A Pnnnn code is required and must match a line number within the same program. The Q value must be in the range of 0 to 31. These correspond to the discrete inputs found on the diagnostic display page with the upper left being input 0 and the lower right being 31. Q is not required within the M96 block. The last specified Q will be used. This command stops the look-ahead queue until the test is made at run-time. Since the look-ahead queue is exhausted, M96 cannot be executed when cutter compensation is invoked. M96 cannot be executed from a main DNC program. If you wish to use M96 in DNC, it must be in a resident subroutine called from the DNC program.
The following is an M96 example:

```
N05 M96 P5 Q8    (TEST INPUT DOOR S, UNTIL CLOSED);
N10    (START OF SOME PROGRAM LOOP);
.
.   (PROGRAM THAT MACHINES PART);
.
N85 M21    (EXECUTE AN EXTERNAL USER FUNCTION)
N90 M96 P10 Q27    (LOOP TO N10 IF SPARE INPUT IS 0);
M95 M30    (IF SPARE INPUT IS 1 THEN END PROGRAM);
```

M97 LOCAL SUB-ROUTINE CALL

This code is used to call a sub-routine, referenced by a line number **N** within the same program. A **Pnnnnn** code is required and must match the **N** line number. This is used for simple sub-routines within a program and does not require the complication of having a separate program. A local sub-routine must still end with an M99. If there is an **L** count on the M97 line, the sub-routine will be repeated that number of times.

    Main program:
    O04321 *(Start of main program)*
    ...
    ...   *(Part program)*
    ...
    M97 P123 *(Jumps to line N123, after the M30, to execute a local sub-routine.)*
    ...   *(The M99 at the end of the sub-routine will cause it to jump back here.)*
    ...
    ...   *(Finish part program)*
    ...
    M30   *(End of main program)*
    N123 *(Identifies the start of the Local Sub-Routine called up by M97 P123)*
    ...
    ...   *(Local sub-routine portion of part)*
    ...
    M99   *(Jumps back to the line after the local sub-routine call in the main program)*

M98 SUB-PROGRAM CALL

This M98 code is used to call a sub-program. The **Pnnnn** code is the sub-program number being called; it must be in the same block as the M98. The sub-program number being called must already be loaded into the control, and it must contain an M99 at the end in order to return to the next line in the main program. An **L** count can also be included on the line containing the M98, which will cause the subroutine to be repeated **L** times before continuing to the next block.

Main program:
O05432 *(Start of main program)*

...
... *(Part program)*

...
M98 P234 *(Jumps to program O00234 to execute sub-program)*
... *(The M99 at the end of the sub-program will jump back here)*

...
... *(Finish part program)*

...
M30 *(End of main program)*

Sub-program:
O00234 *(Identifies the start of a separate sub-program)*

...
… *(Sub-program portion of part)*

...
M99 *(Jumps back to the line after the sub-program call in the main program)*

M99 SUB-PROGRAM RETURN OR LOOP (SETTING 118)

In the main program, an M99 will cause the program to loop back to the beginning and repeat over and over again without stopping.

```
Main program:
O06543

...
...     (Complete part program)
...

...
M99    (This will cause the program to jump back to the beginning and repeat itself)
```

An M99 without a **P** code at the end of a sub-program will return to the main program after executing the sub-routine or macro called.

```
Main program:
O07654

...
...     (Part program)
...
M98 P345   (Jumps to program O00345 to run)
...     (The M99 at the end of the sub-program will jump back here)

...
...     (Finish part)

...
M30    (End of main program)


Sub-program:
O00345  (Identifies sub-program)

...
...     (Sub-program portion of part)

...
M99   (Jumps back to the line after the sub-program call)
```

Note: If M99 **Pnnnn** is used at the end of the sub-program, it will cause a jump to line number **Nnnnn (**containing the same number as **Pnnnn**) in the main program.

A condition of using an M99 **Pnnnn** in the Haas control varies from that seen in Fanuc-compatible controls. In Fanuc-compatible controls, M99 **Pnnnn** will return to the main program and resume execution at block N specified by M99 **Pnnnn**. For the Haas control, M99 will NOT return to block N specified in the M99 return call, but (like always) will jump to the line after the sub-program call in the main program.

You can simulate Fanuc behavior by using the following code-calling program:

Fanuc	**Haas**
Main program:	Main program:
O00432	O00543
...	…
N50 M98 P9876	N50 M98 P9876
...	N51 M99 P101
...	…
N100 *(to continue here)*	…
...	N101 *(to continue here)*
...	…
...	…
M30	M30
Sub-program:	Sub-program:
O09876	O09876
M99 P101	M99

If you have macros, you can use a global variable and specify a block to jump to by adding #nnn=dddd in the subroutine and then using M99 P#nnn after the subroutine call. There are many ways to jump conditionally after an M99 return when using macros.

M109   INTERACTIVE USER INPUT (P) (OPTION)

This M code allows a G-code program to place a short prompt on the screen, get a single character input from the user and store it in a macro variable. The first 15 characters from the comment following the M109 will be displayed as a prompt in the lower left corner of the screen. A macro variable in the range 500 through 599 must be specified by a P code. Note also that due to the look-ahead feature, it is necessary to include a loop in the program following the M109 to check for a non-zero response before continuing. The program can check for any character that can be entered from the keyboard by comparing with the decimal equivalent of the ASCII character.

M110   TAILSTOCK CHUCK CLAMP (OPTION) (SETTING 122)

M111   TAILSTOCK CHUCK UNCLAMP (SETTING 122)

M119   SUB-SPINDLE ORIENT (P,R) (OPTION)

M121-M128   OPTIONAL USER M

The M121 through M124 codes are optional for user interfaces. They will activate one of relays 25 through 28, wait for the M-fin signal, release the relay, and wait for the M-fin signal to cease. The RESET button will terminate any operation that is hung-up waiting for M-fin.

M133   LIVE TOOL DRIVE FORWARD (P) (OPTION)

This M code commands the Live Tool Drive to turn forward direction and requires a P code specifying the Live Tooling Drive RPM, for example M133 P1000.

M134   LIVE TOOL DRIVE REVERSE (P) (OPTION)

This M code commands the Live Tool Drive to turn reverse direction and requires a P code specifying the Live Tooling Drive RPM, for example M134 P1000.

M135   LIVE TOOL DRIVE STOP

This M code commands the Live Tool Drive to stop.

M143   SUB-SPINDLE FORWARD (P) (OPTION)

This M code commands the sub-spindle to turn forward direction and requires a P code specifying the RPM, for example M143 P1000.

M144   SUB-SPINDLE REVERSE (P) (OPTION)

This M code commands the sub-spindle to turn reverse direction and requires a P code specifying the RPM, for example M144 P1000.

M145   SUB-SPINDLE STOP

This M code commands the Live Tool Drive to stop.

M154   **C AXIS ENGAGE (OPTION) (SETTING 102)**

M155   **C AXIS DISENGAGE**

M164   **ROTATE APL GRIPPERS**

M165   **OPEN APL GRIPPER 1 (RAW MATERIAL)**

M166   **CLOSE APL GRIPPER 1 (RAW MATERIAL)**

M167   **OPEN APL GRIPPER 2 (RAW MATERIAL)**

M168   **CLOSE APL GRIPPER 2 (RAW MATERIAL)**

Conversion Formulas

inch x 25.4 = mm   mm x 0.03937 = inch
foot x 304.8 = mm   meter x 39.37 = inch
mile x 1.609 = km   km x 0.6214 = mile

Fahrenheit to Celsius: $(°F - 32) ÷ 1.8 = °C$
Celsius to Fahrenheit: $(°C × 1.8) + 32 = °F$
Degrees, Minutes, Seconds to Decimal Degrees: $Degrees + (Minutes/60) + (Seconds/3600) = Decimal\ Degrees$

Mill and Lathe Conversions

To Find:			Formula:
Revolutions Per Minute	**RPM**	**=**	(SFM **x** 3.82) ÷ D
Surface Feet Per Minute	**SFM**	**=**	RPM **x** D **x** 0.262
Surface Meters Per Minute (Metric)	**SMPM**	**=**	SFM **x** 0.3048
Feed Per Minute Milling Feedrate	**FPM**	**=**	FPT **x** T **x** RPM
Feed Per Revolution	**FPR**	**=**	FPM ÷ RPM
Feed Per Tooth - Mill	**FPT**	**=**	FPM ÷ (T **x** RPM)
Feed Per Minute - *Lathe*	**FPM**	**=**	FPR **x** RPM
Metal Removal Rate	**MMR**	**=**	W **x** d **x** F
Advance Per Revolution	**ADV/R**	**=**	F ÷ RPM

Threads

Mill Tapping Feedrate	**FPM**	**=**	1 ÷ TPI **x** RPM
Lathe Threading Feedrate (Thread Lead)	**FPR**	**=**	1 ÷ TPI

Tap Drill Size $=$ Major Dia. of Tap $-$ $\dfrac{\%\ of\ Thread\ Height\ \textbf{x}\ .01299}{TPI}$

Percent of Full Thread $=$ TPI **x** $\dfrac{Major\ Dia.\ of\ Tap\ -\ Drill\ Dia.}{.01299}$

Mill Tapping Feedrate (Metric) $=$ RPM **x** Metric Pitch

Tap Drill Size (Metric) $=$ Tap Major Dia. (mm) $-$ $\dfrac{\%\ of\ Thread\ Height\ \textbf{x}\ Metric\ Pitch}{76.980}$

Percent of Full Thread (Metric) $=$ $\dfrac{76.980}{Metric\ Pitch}$ **x** Basic Major Diameter (mm) $-$ Drilled Hole (mm)

Miscellaneous

Radius of Circle	**=**	Circumference **x** 0.159155
Diameter of Circle	**=**	Circumference **x** 0.31831
Circumference of Circle	**=**	D **x** 3.1416
Area of Circle	**=**	R2 **x** 3.1416
Cutting Time in Minutes (Mill)	**=**	L ÷ FPM
Cutting Time in Seconds (Lathe)	**=**	$\dfrac{Distance\ to\ go\ \textbf{x}\ 60\ sec}{FPR\ \textbf{x}\ RPM}$

Abbreviations and Measurement Units

D	=	Diameter of Milling Cutter or Lathe Part	**RPM**	=	Revolutions per Minute (Spindle Speed)
F	=	Feed (Inch or metric)	**SFM**	=	surface feet per minute
d	=	depth of cut	**SMPM**	=	Surface Meters per Minute
FPR	=	Feed per Revolution	**T**	=	Number of Teeth in the Cutter
FPT	=	Feed per Tooth	**TPI**	=	Threads per Inch
FPM	=	Feed per Minute (Table Travel Feedrate)	**W**	=	Width of Cut
FPR	=	Feed per Revolution	**°C**	=	Degrees Celsius
L	=	Length of Cut (Inches)	**°F**	=	Degrees Fahrenheit

01-14-04

DECIMAL EQUIVALENTS WITH TAP DRILLS CHART

Decimal Equiv.	Drill Size	mm	Tap Sizes	Decimal Equiv.	Drill Size	mm	Tap Sizes	Decimal Equiv.	Drill Size	mm	Tap Sizes	Decimal Equiv.	Drill Size	mm	Tap Sizes
.0019685		0.05		.0781	5/64	1.984		.2008		5.1		.3750	3/8	9.525	
.0039370		0.1		.0785	47	1.994	#3-48	.2010	7	5.105	1/4-20	.3770	V	9.576	
.0059	97	0.150		.0787		2.0		.2031	13/64	5.159		.3780		9.6	
.0063	96	0.160		.0807		2.05	M2.5 x 0.45	.2040	6	5.182		.3819		9.7	
.0067	95	0.170		.0810	46	2.057		.2047		5.2		.3858		9.8	
.0071	94	0.180		.0820	45	2.083	#3-56	.2055	5	5.220		.3860	W	9.804	
.0075	93	0.191		.0827		2.1		.2087		5.3		.3898		9.9	
.0079	92	0.2		.0846		2.15		.2090	4	5.309		.3906	25/64	9.921	7/16-20
.0083	91	0.211		.0860	44	2.184		.2126		5.4		.3937		10.0	
.0087	90	0.221		.0866		2.2		.2130	3	5.410	1/4-28	.3970	X	10.084	
.0091	89	0.231		.0886		2.25		.2165		5.5		.4016		10.2	M12 x 1.75
.0095	88	0.241		.0890	43	2.261	#4-40	.2188	7/32	5.558	1/4-32	.4040	Y	10.262	7/16-28
.0098		0.25		.0906		2.3		.2205		5.6		.4062	13/32	10.318	
.0100	87	0.254		.0925		2.35		.2210	2	5.613		.4130	Z	10.490	
.0105	86	0.267		.0935	42	2.375	#4-48	.2244		5.7		.4134		10.5	
.0110	85	0.279		.0938	3/32	2.383		.2280	1	5.791		.4219	27/64	10.716	1/2-13
.0115	84	0.292		.0945		2.4		.2283		5.8		.4331		11.0	
.0118	-	0.3		.0960	41	2.438		.2323		5.9		.4375	7/16	11.113	
.0120	83	0.305		.0965		2.45		.2340	A	5.944		.4528		11.5	
.0125	82	0.318		.0980	40	2.489		.2344	15/64	5.954		.4531	29/64	11.509	1/2-20
.0130	81	0.330		.0984		2.5	M3 x 0.5	.2362		6.0	M7 x 1	.4688	15/32	11.908	1/2-28
.0135	80	0.343		.0995	39	2.527		.2380	B	6.045		.4724		12.0	M14 x 2
.0138	-	0.35		.1015	38	2.578	#5-40	.2402		6.1		.4844	31/64	12.304	9/16-12
.0145	79	0.368		.1024		2.6		.2420	C	6.147		.4921		12.5	
.0156	1/64	0.396		.1040	37	2.642	#5-44	.2441		6.2		.5000	1/2	12.700	
.0158	-	0.4		.1063		2.7		.2460	D	6.248		.5118		13.0	
.0160	78	0.406		.1065	36	2.705	#6-32	.2480		6.3		.5156	33/64	13.096	9/16-18☐ 9/16-24
.0177	-	0.45		.1094	7/64	2.779		.2500	1/4 & E	6.350		.5312	17/32	13.493	5/8-11
.0180	77	0.457		.1100	35	2.794		.2520		6.4		.5315		13.5	
.0197	-	0.5		.1102		2.8		.2559		6.5		.5469	35/64	13.891	
.0200	76	0.508		.1110	34	2.819		.2570	F	6.528	5/16-18	.5512		14.0	M16 x 2
.0210	75	0.533		.1130	33	2.870	#6-40	.2598		6.6		.5625	9/16	14.288	5/8-16
.0217	-	0.55		.1142		2.9	M3.5 x 0.6	.2610	G	6.629		.5709		14.5	
.0225	74	0.572		.1160	32	2.946		.2638		6.7		.5781	37/64	14.684	5/8-18☐ 5/8-20
.0236	-	0.6		.1181		3.0		.2656	17/64	6.746		.5906		15.0	
.0240	73	0.610		.1200	31	3.048		.2660	H	6.756		.5938	19/32	15.083	5/8-24☐ 5/8-28☐ 5/8-32
.0250	72	0.635		.1220		3.1		.2677		6.8	M8 x 1.25	.6094	39/64	15.479	11/16-12
.0256	-	0.65		.1250	1/8	3.175		.2717		6.9		.6102		15.5	M18 x 2.5
.0260	71	0.660		.1260		3.2		.2720	I	6.909	5/16-24	.6250	5/8	15.875	11/16-16
.0276	-	0.7		.1285	30	3.264		.2756		7.0		.6299		16.0	
.0280	70	0.711		.1299		3.3	M4 x 0.7	.2770	J	7.036		.6406	41/64	16.271	11/16-20
.0292	69	0.742		.1339		3.4		.2795		7.1		.6496		16.5	
.0295	-	0.75	M1 x 0.25	.1360	29	3.454	#8-32☐ #8-36	.2810	K	7.137		.6562	21/32	16.668	11/16-24☐ 11/16-28
.0310	68	0.787		.1378		3.5		.2812	9/32	7.143	5/16-32	-	-		11/16-32☐ 3/4-10
.0313	1/32	0.793		.1405	28	3.569		.2835		7.2		.6693		17.0	
.0315	-	0.8		.1406	9/64	3.571		.2874		7.3		.6719	43/64	17.066	3/4-12
.0320	67	0.813		.1417		3.6		.2900	L	7.366		.6875	11/16	17.463	3/4-16
.0330	66	0.838		.1440	27	3.658		.2913		7.4		.6890		17.5	M20 x 2.5
.0335	-	0.85	M1.1 x 0.25	.1457		3.7		.2950	M	7.493		.7031	45/64	17.859	3/4-20
.0350	65	0.889		.1470	26	3.734		.2953		7.5		.7087		18.0	
.0354	-	0.9		.1476		3.75	M4.5 x 0.75	.2969	19/64	7.541		.7188	23/32	18.258	3/4-28☐ 3/4-32
.0360	64	0.914		.1495	25	3.797	#10-24	.2992		7.6		.7283		18.5	
.0370	63	0.940		.1496		3.8		.3020	N	7.671		.7344	47/64	18.654	13/16-12
.0374	-	0.95	M1.2 x 0.25	.1520	24	3.861		.3031		7.7		.7480		19.0	
.0380	62	0.965		.1535		3.9		.3071		7.8	M9 x 1.25	.7500	3/4	19.050	13/16-16
.0390	61	0.991		.1540	23	3.912		.3110		7.9		.7656	49/64	19.446	13/16-20☐ 7/8-9
.0394	-	1.0		.1562	5/32	3.968		.3125	5/16	7.938	3/8-16	.7677		19.5	M22 x 2.5
.0400	60	1.016		.1570	22	3.988		.3150		8.0		.7812	25/32	19.843	13/16-32
.0410	59	1.041		.1575		4.0		.3160	O	8.026		.7874		20.0	
.0420	58	1.067		.1590	21	4.039	#10-32	.3189		8.1		.7969	51/64	20.241	7/8-12
.0430	57	1.092		.1610	20	4.089		.3228		8.2		.8071		20.5	
.0433	-	1.1	M1.4 x 0.3	.1614		4.1		.3230	P	8.204		.8125	13/16	20.638	7/8-14☐ 7/8-16
.0465	56	1.181		.1654		4.2	M5 x 0.8	.3268		8.3		.8268		21.0	M24 x 3
.0469	3/64	1.191	#0-80	.1660	19	4.216		.3281	21/64	8.334	3/8-20	.8281	53/64	21.034	7/8-20
.0472	-	1.2		.1693		4.3		.3307		8.4		.8438	27/32	21.433	7/8-28☐ 7/8-32
.0492	-	1.25	M1.6 x 0.35	.1695	18	4.305		.3320	Q	8.433	3/8-24	.8465		21.5	
.0512	-	1.3		.1719	11/64	4.366		.3346		8.5	M10 x 1.5	.8594	55/64	21.829	15/16-12
.0520	55	1.321		.1730	17	4.394		.3386		22.0		.8661		22.0	
.0550	54	1.397		.1732		4.4		.3390	R	8.611		.8750	7/8	22.225	15/16-16☐ 1.0-8
.0551	-	1.4		.1770	16	4.496	#12-24	.3425		8.7		.8858		22.5	
.0571	-	1.45	M1.8 x 0.35	.1772		4.5		.3438	11/32	8.733	3/8-32	.8906	57/64	22.621	15/16-20
.0591	-	1.5		.1800	15	4.572		.3465		8.8		.9055		23.0	
.0595	53	1.511	#1-64☐ #1-72	.1811		4.6		.3480	S	8.839		.9062	29/32	23.018	15/16-28☐ 15/16-32
.0625	1/16	1.588		.1820	14	4.623	#12-28	.3504		8.9		.9219	59/64	23.416	1.0-12
.0630	-	1.6	M2 x 0.4	.1850	13	4.699	#12-32	.3543		9.0		.9252		23.5	
.0635	52	1.613		.1875	3/16	4.763		.3580	T	9.093		.9375	15/16	23.813	1.0-16☐ 11/16-8
.0669	-	1.7		.1890	12	4.801		.3583		9.1		.9449		24.0	M27 x 3
.0670	51	1.702		.1910	11	4.851		.3594	23/64	9.129		.9531	61/64	24.209	1.0-20
.0689	-	1.75	M2.2 x 0.45	.1929		4.9		.3622		9.2		.9646		24.5	
.0700	50	1.778	#2-56☐ #2-64	.1935	10	4.915		.3661		9.3		.9688	31/32	24.608	1.0-28☐ 1.0-32
.0709	-	1.8		.1960	9	4.978		.3680	U	9.347	7/16-14	.9843		25.0	
.0730	49	1.854		.1969		5.0	M6 x 1	.3701		9.4		.9844	63/64	25.004	11/16-12☐ 11/8-7
.0748	-	1.9		.1990	8	5.055		.3740		9.5		1.000	1	25.400	11/16-16☐ 11/8-8
.0760	48	1.930													

07/9/03