This Manual is the Property of Productivity Inc

The document may not be reproduced without the express written permission of Productivity Inc.

The content must not be altered, nor may the Productivity Inc name be removed from the materials.

This material is to be used as a guide to operation of the machine tool. The Operator is responsible for following Safety Procedures as outlined by their instructor or manufacturer’s specifications.

NOTE: Downloading and/or other use of this manual does not certify completion of the Training Course. This manual is for reference only.

To obtain permission, please contact trainingmn@productivity.com.
# Haas Lathe Programming Training Manual

## Table of Contents

- **HAAS LATHE PROGRAMMING TRAINING MANUAL TABLE OF CONTENTS** ................................................................. 1
- **INTRODUCTION** ..................................................................................................................................................... 5
- **THE COORDINATE SYSTEM** .................................................................................................................................... 6
- **MACHINE HOME** .................................................................................................................................................... 7
- **ABSOLUTE AND INCREMENTAL POSITIONING** ....................................................................................................... 9
- **PROGRAMMING** .................................................................................................................................................. 15
  - **PROGRAM FORMAT** ............................................................................................................................................... 17
  - **DEFINITIONS WITHIN THE FORMAT** ............................................................................................................................. 18
- **ALPHABETICAL ADDRESS CODES** .......................................................................................................................... 19
- **PREPARATORY FUNCTIONS (G CODES)** ................................................................................................................ 24
- **MISCELLANEOUS FUNCTIONS (M CODES)** ............................................................................................................ 26
  - **MACHINE DEFAULTS** .............................................................................................................................................. 27
- **LATHE PROGRAMMING** ....................................................................................................................................... 28
- **MACHINING CYCLES FOR THE LATHE** ................................................................................................................ 31
  - **DRILLING, TAPPING AND BORING CANNED CYCLES** ........................................................................................ 31
  - **MACHINE CYCLES FOR TURNING AND GROOVING** .......................................................................................... 32
- **LINEAR/CIRCULAR MOVEMENT-CREATING TOOL PATHS** .................................................................................. 33
- **INTERPOLATION COMMANDS** ............................................................................................................................. 35
  - **LINEAR INTERPOLATION EXERCISE** .......................................................................................................................... 37
  - **CIRCULAR INTERPOLATION COMMANDS** ................................................................................................................ 38
  - **CIRCULAR INTERPOLATION MOTION EXERCISES** .................................................................................................. 43
  - **CIRCULAR INTERPOLATION EXERCISE** .................................................................................................................. 45
- **MANUALLY PROGRAMMING TOOL NOSE COMPENSATION** .............................................................................. 46
  - **CALCULATING COMPENSATION FOR A RADIUS ON YOUR PART** ................................................................. 46
  - **RADIUS CALCULATION** ........................................................................................................................................... 47
  - **EXTERNAL RADIUS CALCULATION** .......................................................................................................................... 48
  - **INTERNAL RADIUS CALCULATION** ............................................................................................................................ 48
  - **CALCULATING COMPENSATION FOR “AN ANGLE” ON YOUR PART** .............................................................. 52
- **TOOL NOSE RADIUS CALCULATION DIAGRAM** .................................................................................................. 53
  - **TOOL NOSE COMPENSATION – TAPER CALCULATION** ......................................................................................... 54
- **MISCELLANEOUS G CODES** ................................................................................................................................... 65
  - **REFERENCE POINT DEFINITION AND RETURN** .................................................................................................... 65
  - **SPINDLE SPEED COMMANDS** ................................................................................................................................. 66
WORK COORDINATE SYSTEM SELECTION .......................................................................................................................... 67
FEED COMMAND FUNCTIONS .................................................................................................................................................. 67
TOOL NOSE COMPENSATION G CODES ............................................................................................................................... 68
TOOL NOSE COMPENSATION PROGRAMMING .................................................................................................................. 70
TOOL NOSE COMPENSATION CONCEPTS ........................................................................................................................... 72
IMAGINARY TOOL TIP AND DIRECTION .............................................................................................................................. 73
USING TOOL NOSE COMPENSATION ............................................................................................................................... 74
CANNED CYCLES AND ADDITIONAL G CODES .................................................................................................................. 82
TOOL NOSE COMPENSATION IN CANNED CYCLES ........................................................................................................... 88
TYPE 1 DETAILS ................................................................................................................................................................. 96
TYPE II DETAILS ................................................................................................................................................................. 96
TYPE I DETAILS ............................................................................................................................................................... 103
DRILLING, BORING AND TAPPING CANNED CYCLES ....................................................................................................... 123
CANNED CYCLES FOR DRILLING AND TAPPING .................................................................................................................. 124
M CODE DETAILED DESCRIPTION .................................................................................................................................... 139
For more information on Additional Training Opportunities or our Classroom Schedule, Contact the Productivity Inc Applications Department in Minneapolis: ☏ 763.476.8600
Visit us on the Web: www.productivity.com
Click on the Training Registration Button ⌐ trainingmn@productivity.com
Introduction

Welcome to Productivity, Inc., your local Haas Factory Outlet (H.F.O.) for the Haas Lathe Programming Class. This class is intended to give a basic understanding of the set-up and operation of a Haas Turning Center.

In an "NC" (Numerically Controlled) machine, the tool is controlled by a code system that enables it to be operated with minimal supervision and with a great deal of repeatability. "CNC" (Computerized Numerical Control) is the same type of operating system, with the exception that a computer monitors the machine tool.

The same principles used in operating a manual machine are used in programming an NC or 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.

The operation of the SL-Series Turning Center requires that a part program be designed, written, and entered into the memory of the control. There are several options for getting these programs to the control. RS-232 (serial port with a computer), 3.5” Floppy Disk, Ethernet / Networking/ and USB are all viable ways to transmit and receive programs.

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.

At Productivity, we have two classes that pertain to Haas Turning Centers: Lathe Operator and Lathe Programming.

We have two classes to fill the different needs of our customers as not all people that require training require programming training.

This manual can be used as both an operator's manual and as a programmer’s manual. It is intended to give a basic understanding of CNC programming and its applications. It is not intended as an in-depth study of all ranges of 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.

Revised CK 5/9/14; Rev3 - 1/2015
The Coordinate System

All CNC machines move tools to specific locations described by coordinate systems. With lathes the coordinate system can be simply described as two number lines that intersect.

The illustration below shows two number lines that intersect at a location described as reference zero or Absolute Zero.

With lathes the vertical number line is called the X-axis. The horizontal number line is called the Z-axis.

The movements of the lathe are described by the illustration below. The up and down motion or X-axis corresponds to the vertical number line. The Z-axis or side to side motion corresponds to the horizontal number line. The intersection of the two lines is Absolute Zero.

When programming lathes X0 is always the centerline of the part you are working on. It is the X position on the Z axis that the part rotates around. Z0 normally is the front finished face of the part.
**Machine Home**

At start up all Haas machine tools must be returned to zero or taken to what is called a Home Reference Position. A zero return (POWER UP/RESTART) is required when you power on a Haas machine. During a zero return all axes are moved to extreme positive locations until limit switches are reached. The reason the machine does this is to double check its position with the “Home” switches of the machine.

This is crucial to the operation and function of a CNC machine as all of our programs, locations of fixtures and tooling are based off of **machine home**. At home position the **machine coordinates** are X0, Z0. It would not be easy or convenient to write a program using machine coordinates. Instead programs are written with values that would correspond to dimensions found on prints. To do this a secondary floating zero point is established using offsets. This floating zero is referred to as the **PART ZERO or PART ORIGIN**.

As shown above the centerline of the part becomes (X) zero. Normally the front face of the part is designated as Z (zero). The reason being it is easy to access and easier to program. With this scheme the centerline of the lathe is always the “X” zero. The “Z” zero location will float with the face of different parts.

When we setup a machine, we need to tell the machine the distance different tools at home position are from the part origin. Each tool is manually touched off the face and diameter and thru keystrokes the distances from machine zero to the part zero are saved in X and Z register of the **Tool Offsets Page**. Alternately tools may be set using the tool pre-setter. If the tools are set with a tool pre-setter or other Z reference point not at the Part Zero a Z Work Shift must be done.

A work shift is stored in the **Work Offsets Page** in the Z value. Several different work offsets may be used. The Z value in the Work Offset page is the distance and direction the Part Zero is from the Z reference or tool pre-setter touch off position. Normally X values are always zero. If the face of the part is used to determine the Tool Offsets then the value for Z in the Work Offset page will be zero.
The diagram above shows the operator’s view of X and Z grid standing in front of the lathe. At the intersection of the X and Z axis is the Part Zero or Reference Point. Note there is four different quadrants with different positive and values for X and Z.

An ST-20 lathe has travel distance of 9.3” in the X and 21” in the Z. Haas lathes have a negative one-inch (-1”) travel beyond the centerline of the Z Axis spindle. Note that most of the X values in a part program are going to be positive. Normally the only negative X value is going to be when you face the part off past centerline. As the Z zero is normally the face of the part when you are cutting into the part the Z values will be negative.

Note that the X values on the Haas lathe are in diameters. Therefore an X move from centerline or X0 to X1.0 will only be moving the machine along the X axis 0.5 inch in the positive direction.
**Absolute and Incremental Positioning**

By using WORK and TOOL OFFSETS a common point on the part is designated as “**PART ZERO**”. This is some point on our part that we can physically find. The programmer uses this point as a base to write the intended movement of the tooling.

Programmers normally use the front end of our finish machined part as (Z Zero) and the centerline of part as (X Zero).

There are two methods used by the programmer to “Steer” our machine.

The first is “**ABSOLUTE POSITIONING**”. Absolute means that X and Z code values are based on the ZERO POINT on the part. If a diameter of 1.0000 inches is needed, it is input as X1.0000. If the print requires facing a shoulder that is 3 inches back from the front of the part, Z-3.0000 in input in the code.

The programmer has another tool available to him called “**INCREMENTAL POSITIONING**”. This is movement based on where the machine is currently sitting. It is also called point to point programming. If a change of half inch smaller diameter is required of the machine from where it is currently sitting U-.5000 is put in the code. If a grooving tool is making a groove that is located ¾” behind a groove that is already finished, W-.7500 is input.

**The letters X&Z represent ABSOLUTE POSITIONING**

**The letters U&W represent INCREMENTAL POSITIONING**

If you are familiar with the mill programming language, absolute and incremental are handled differently for Mills and Lathes. A mill uses G codes (G90 and G91) to go back and forth between the two. Where as a lathe uses the different letters to differentiate them.

**QUESTION:**

*Why doesn’t a lathe take G90 and G91 like a mill?*

**ANSWER:**

*A lathe has the unique possibility to do Absolute AND Incremental moves AT THE SAME TIME.*

The programmer can place an ABS. letter and an INC. letter on a line of code together, and is most commonly used in making tapers and radius moves.

**G01 X2.000 W-.25 (Move X in Absolute, Z in Incremental)**

or

**G01 U.5000 Z-.5000 (Move X in Incremental, Z in Absolute)**
Cartesian Coordinate X, Z Grid

Typical Lathe Part
Give the X and Z coordinates for the part below. Note the X values are **diameters** on Haas Lathes not radii.

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Point 1</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Point 2</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Point 3</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Point 4</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Point 5</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Point 6</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Point 7</strong></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Typical Lathe Part

Below is a typical lathe part sitting in a chuck. Note the symbol. It denotes the floating zero determined by the programmer.
Above is the print for the part on previous page. Knowing X and Z coordinates for key positions below is the first step in programming.

Give the X and Z coordinates for the part below. Note the X values are diameters on Haas Lathes not radii.

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point 1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Point 2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Point 3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Point 4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Point 5</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
For the different points below calculate X diameters and Z lengths. 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.

\[
\begin{array}{ll}
\text{Pt 1} & \text{__________} \\
\text{Pt 2} & \text{__________} \quad (\text{__________}) \\
\text{Pt 3} & \text{__________} \\
\text{Pt 4} & (\text{__________}) \\
\text{Pt 5} & \text{__________} \\
\text{Pt 6} & \text{__________} \\
\text{Pt 7} & (\text{__________}) \\
\text{Pt 8} & \text{__________} \quad (\text{__________}) \\
\text{Pt 9} & (\text{__________}) \\
\text{Pt 10} & (\text{__________}) \\
\text{Pt 11} & (\text{__________}) \\
\text{Pt 12} & \text{__________} \\
\end{array}
\]

Programming

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 codes commands.

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

The Haas lathe has two (2) linear axes named X and Z. 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 right corner of the work cell farthest away from the spindle axis. Motion in the X-axis will move the table toward the spindle centerline for negative numbers and away from spindle center for positive numbers. Motion in the z-axis will move the tool toward the spindle chuck for negative numbers and away from the chuck for positive numbers.

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 THE SPINDLE ON AND SELECT THE RPM.
LINE #3 = TURN THE COOLANT ON.
LINE #4 = RAPID TO THE STARTING POSITION OF THE PART.
LINE #5 = CHOOSE THE PROPER FEED RATE AND MAKE THE CUT(S)
LINE #6 = TURN OFF THE SPINDLE AND THE COOLANT.
LINE #7 = RETURN TOOL TO HOLDING POSITION AND SELECT NEXT TOOL

and so on. But our machine control understands only these messages when given in machine code.
Program Format

There are no positional requirements for a line CNC code. That means the different codes in a program may be in any order on a line of code. However some standard rules are followed so the code is easier to view. It will also make it easier for the machinist to understand when he runs the part if the program has a standard level of organization. Some standard rules are followed:

X and Z values are positioned in alphabetical order and grouped together

G and M codes may be placed anywhere on a line but convention is that the G codes come first and the M codes come at the end of the block. This makes sense as the last thing to happen on a line is the M-function. The G codes are completed first then the M code is performed on any given line.

On Haas machines only one M code is allowed on a block of code.

Command codes are first given by a letter then a number. Some codes like X,Z and F require decimal points. Others like S and G require an integer (a number with no fractional part).
Definitions Within the Format

**Character:** A single alphanumeric character value or the “+” and “-” sign.

**Word:** A series of characters defining a single command such as “X” displacement or “F” Feed rate. Unique letters are assigned as the first character of a word, and each letter has either a plus (+) or minus (-) sign value of numbers.

**Block:** Series of words defining a single instruction. An instruction may consist of a single linear or circular motion, plus additional information such as a feed rate or stop command. A block of code is on one single line.

**Positive Signs:** If a number value following the address command letters, such as I, K, R, U, W, X, Z, is positive, the plus sign need not be programmed. If the number value is negative, it must be programmed with a minus (-) sign.

**Leading Zeros:** There is no need to program zeros proceeding a number.

**Example:** G00 (G0) and M01 (M1).

Trailing zeros however must be programmed.

**Example:** M30 not M3, G70 or G7.

**Modal Commands:** Once a particular word, such as G, X, Z, F, S, T, and M is programmed, it is not necessary to repeat the word in the following blocks until a different word or change of value is required in subsequent blocks of information.

**Non-Modal Commands:** A non-modal command is one that is active only in the program block in which it is issued. M00 program stop is an example of a non-modal command.

**Preparatory Functions:** “G” codes use the information contained on the line to make the machine tool do specific operations, such as:

- Move the tool at rapid traverse.
- Move the tool at feed rate along a straight line.
- Move the tool along and arc at a feed rate in a clockwise direction.
- Move the tool along an arc at a feed rate in a counterclockwise direction.
- Move the tool through a series of repetitive operations controlled by “fixed cycles” such as, spot drilling, boring, and tapping.

**Miscellaneous Functions:** “M” codes cause an action to occur at the end of the block. Only one M-Code is allowed in each block.

**Sequence Numbers:** Sequence numbers are codes N1 through N9999 and are only used to locate a certain block or line within a CNC program. A program may be input without sequence numbers.
Alphabetical Address Codes

The following is a list of the Address Codes used in programming the lathe:

A  FOURTH AXIS ROTARY MOTION

The A address character is normally used to specify motion for the optional fourth A, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, the most negative value is -8380.000, and the largest value is 8380.000 degrees.

The A axis on the Haas lathe is currently reserved for the tool turret and is hidden to the programmer. The units of rotation indicate tool positions such that 1.000 represents tool #1.

B  LINEAR B-AXIS MOTION

The B address character is used to specify absolute motion for the B axis. It specifies a position or distance along the B axis. It is either in inches with four fractional positions or millimeters with three fractional positions. It is followed by a signed number between -8380.00 and 8380.00. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 millimeters. On the Haas lathes the B axis is either travel of the tail stock or sub-spindle.

C  FIFTH AXIS ROTARY MOTION

The C address character is used to specify motion for the optional external fifth, C, axis. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions. If no decimal point is entered, the last digit is assumed to be 1/1000 degrees. The smallest magnitude is 0.001 degrees, the most negative value is -8380.00, and the largest value is 8380.00 degrees.

D  CANNED CYCLE DATA

The D address character is used within several canned macro cycles:
In G71 and G72 stock removal cycles D prescribes the depth of cut used for each pass.
In G73 D indicates the number of passes to remove a set amount of stock.
In G76 multiple pass threading cycle D sets the depth of cut on the first pass.

E  FEED RATE, 6 PLACE PRECISION (SAME AS F)

The E address character is used to select feed rate applied to any interpolating G codes or canned cycles. The unit is in inches per revolution or mm per revolution. Up to six fractional positions can be specified. The default of units/revolution (G99) can be changed to units/minute with G98. For YASNAC and FANUC control compatibility, use the E code when 5 or 6 place precision is desired.
F FEED RATE

The F address character is used to select feed rate applied to any interpolating G codes or canned cycles. The unit is in inches per revolution or mm per revolution. The default of units/revolution (G99) 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 FUNCTIONS (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 255. 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 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. More than one G code can be placed in a block in order to specify all of the setup conditions for an operation. See the following section (Preparatory Functions (G Codes)) for a detailed list of G codes.

H NOT USED, OPTIONAL MACRO PARAMETER

I CANNED CYCLE AND CIRCULAR OPTIONAL 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. It is followed by a signed number in inches between -8380.00 and 8380.00 for inches or between -8380.00 and 8380.00 for metric.

J CANNED CYCLE AND CIRCULAR OPTIONAL DATA

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

K CANNED CYCLE AND CIRCULAR OPTIONAL 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 FOR REPEATED CYCLES

The L address character is used to specify a repetition count for some canned cycles and auxiliary functions. It is followed by an unsigned number between 0 and 32767.
M  M CODE MISCELLANEOUS FUNCTIONS

The M address character is used to specify an M code for a block. These codes are used to control miscellaneous machine functions. Not 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  NUMBER OF BLOCK

The N address character is entirely optional. It can be used to identify or number each block of a program. It is followed by a number between 0 and 99999. The M97, M98 and M99 functions may reference an N line number.

O  PROGRAM NUMBER PROGRAM NAME 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; it cannot be deleted. Altering the O in the first block causes the program to be renamed. A program may have only one Onnnnn command a colon (:) may be used in the place of O, but is always displayed as “O”. “O” Programs may be deleted by highlighting in the List Program page by using the DEL key or ERASE PROG key. Up to 500 programs may be stored in the List Program page.

P  DELAY TIME OR PROGRAM NUMBER

The P address character is used to enter either a time in seconds or a program number for a subroutine call. If it is used as a time (for a G04 dwell) or a program name (for a M97), the value may be either a positive number without decimal point up to 9999. If it is used as a time, it may be a positive decimal with fraction between 0.001 and 1000.00.

Q  CANNED CYCLE OPTIONAL DATA

The Q address character is used in canned cycles and is always a positive number in inches/mm between 0 and 100.0.

R  CANNED CYCLE AND CIRCULAR OPTIONAL DATA

The R address character is used in canned cycles and circular interpolation. It is either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches between -8380.00 and 8380.00 for inches or between -8380.00 and 8380.00 for metric. It is usually used to define the reference plane for canned cycles.

S  SPINDLE SPEED COMMAND

The S address character is used to specify the spindle speed. The S is followed by an unsigned number between 1 - 99999. The S command does not turn the spindle on or off; it only sets the desired speed. If a gear change is required in order to set the commanded speed, this command will cause a gear change to occur even if the spindle is stopped. If the spindle is running, a gear change operation will occur and the spindle will continue running at the new speed. A decimal is not allowed with a S command.
**T TOOL SELECTION CODE**

The T address selects a tool and offset while initiating the tool change process. The format is Txxyy. The xx selects the tool turret position. The number depends on the location of the tool with respect to the turret position. It is a positive number between 1 and the total turret tools (i.e. Parameter 65). yy selects the tool offset and is a positive number between 1 and 50.

**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 either in inches with four fractional positions or mm with three fractional positions. It is followed by a signed number in inches between -838.0000 and 838.0000 for inches or between -8380.000 and 8380.000 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

**V OPTIONAL MACRO PARAMETER**

**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 formatted the same as address U.

**X LINEAR X-AXIS MOTION**

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. It is followed by a signed number in inches between -8380.00 and 8380.00 for inches or -8380.00 and 8380.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.

**Y LINEAR Y-AXIS MOTION**

The Y address character is used to specify absolute motion along the Y axis on Haas ST10Y, ST20Y, ST30Y and DS30Y lathes. It specifies the plus or minus Y position from Y0 to Y±2.0000”. If no decimal point is entered, the last digit is assumed to be 1/10000 of an inch or 1/100 of a millimeter.

**Z LINEAR Z-AXIS MOTION**

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 or mm with three fractional positions. It is followed by a signed number in inches between -8380.00 and 8380.00 for inches or between -8380.00 and 8380.00 for metric. If no decimal point is entered, the last digit is assumed to be 1/10000 inches or 1/1000 mm.
Preparatory Functions (G Codes)

1) G codes come in groups with each group having a specific group number.
2) A G code from the same group can be replaced by another G code in the same group. This 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 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 machine code used for programming the Haas control. The concept of grouping codes and the rules that apply will have to be remembered if we are to effectively program the machine tool. The following is a brief description on some of the codes most basic to the operation of the machine.

(*) denotes a default code.

The following is a summary of G codes.

<table>
<thead>
<tr>
<th>Code</th>
<th>Group</th>
<th>Function:</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>01*</td>
<td>Rapid Motion (ST20 945 in/min, ST30SS 1200 in/min in Z)</td>
</tr>
<tr>
<td>G01</td>
<td>01</td>
<td>Linear Interpolation Motion</td>
</tr>
<tr>
<td>G02</td>
<td>01</td>
<td>CW Interpolation Motion</td>
</tr>
<tr>
<td>G03</td>
<td>01</td>
<td>CCW Interpolation Motion</td>
</tr>
<tr>
<td>G04</td>
<td>00</td>
<td>Dwell (non-modal)</td>
</tr>
<tr>
<td>G09</td>
<td>00</td>
<td>Exact Stop (non-modal)</td>
</tr>
<tr>
<td>G10</td>
<td>00</td>
<td>Programmable Offset Setting (non-modal)</td>
</tr>
<tr>
<td>G17</td>
<td>02</td>
<td>XY Plane Selection (used with live tooling)</td>
</tr>
<tr>
<td>G18</td>
<td>02*</td>
<td>ZX Plane Selection (Plane used on lathes)</td>
</tr>
<tr>
<td>G19</td>
<td>02</td>
<td>YZ Plane Selection (used with live tooling)</td>
</tr>
<tr>
<td>G20</td>
<td>06</td>
<td>Inch Programming Selection (setting #9)</td>
</tr>
<tr>
<td>G21</td>
<td>06</td>
<td>Metric Programming Selection</td>
</tr>
<tr>
<td>G28</td>
<td>00</td>
<td>Machine Home in Rapid Traverse</td>
</tr>
<tr>
<td>G29</td>
<td>00</td>
<td>Set Return to Reference Point</td>
</tr>
<tr>
<td>G31</td>
<td>00</td>
<td>Skip Function</td>
</tr>
<tr>
<td>G40</td>
<td>07*</td>
<td>Tool Nose Compensation Cancel</td>
</tr>
<tr>
<td>G41</td>
<td>07</td>
<td>Tool Nose Compensation Left</td>
</tr>
<tr>
<td>G42</td>
<td>07</td>
<td>Tool Nose Compensation Right</td>
</tr>
<tr>
<td>G50</td>
<td>11</td>
<td>Spindle Speed Clamp, Global Coordinate</td>
</tr>
<tr>
<td>G51</td>
<td>11</td>
<td>Cancel G50 Offset (Yasnac)</td>
</tr>
<tr>
<td>G52</td>
<td>00</td>
<td>Child Coordinate System, M30, Reset Cancels</td>
</tr>
<tr>
<td>G53</td>
<td>00</td>
<td>Non-Modal Machine Coordinate Selection</td>
</tr>
<tr>
<td>G Code</td>
<td>Value</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------</td>
<td>-------------</td>
</tr>
<tr>
<td>G54</td>
<td>12*</td>
<td>Select Work Coordinate System 1</td>
</tr>
<tr>
<td>G55</td>
<td>12</td>
<td>Select Work Coordinate System 2</td>
</tr>
<tr>
<td>G56</td>
<td>12</td>
<td>Select Work Coordinate System 3</td>
</tr>
<tr>
<td>G57</td>
<td>12</td>
<td>Select Work Coordinate System 4</td>
</tr>
<tr>
<td>G58</td>
<td>12</td>
<td>Select Work Coordinate System 5</td>
</tr>
<tr>
<td>G59</td>
<td>12</td>
<td>Select Work Coordinate System 6</td>
</tr>
<tr>
<td>G61</td>
<td>13</td>
<td>Exact Stop Modal</td>
</tr>
<tr>
<td>G64</td>
<td>13*</td>
<td>G61 Cancel</td>
</tr>
<tr>
<td>G65</td>
<td>00</td>
<td>Macro Subroutine Call</td>
</tr>
<tr>
<td>G70</td>
<td>00</td>
<td>Finishing Cycle</td>
</tr>
<tr>
<td>G71</td>
<td>00</td>
<td>O.D./I.D. Stock Removal Cycle</td>
</tr>
<tr>
<td>G72</td>
<td>00</td>
<td>End Face Stock Removal Cycle</td>
</tr>
<tr>
<td>G73</td>
<td>00</td>
<td>Irregular Path Stock Removal Cycle</td>
</tr>
<tr>
<td>G74</td>
<td>00</td>
<td>End Face Grooving Cycle, Peck Drilling</td>
</tr>
<tr>
<td>G75</td>
<td>00</td>
<td>O.D./I.D. Grooving cycle, Peck Drilling</td>
</tr>
<tr>
<td>G76</td>
<td>00</td>
<td>Thread Cutting Cycle, Multiple Pass</td>
</tr>
<tr>
<td>G80</td>
<td>09*</td>
<td>Canned Cycle Cancel</td>
</tr>
<tr>
<td>G81</td>
<td>09</td>
<td>Drill Canned Cycle</td>
</tr>
<tr>
<td>G82</td>
<td>09</td>
<td>Spot Drill Canned Cycle</td>
</tr>
<tr>
<td>G83</td>
<td>09</td>
<td>Peck Drill Canned Cycle</td>
</tr>
<tr>
<td>G84</td>
<td>09</td>
<td>Tapping Canned Cycle</td>
</tr>
<tr>
<td>G85</td>
<td>09</td>
<td>Boring Canned Cycle</td>
</tr>
<tr>
<td>G86</td>
<td>09</td>
<td>Bore/Stop Canned Cycle</td>
</tr>
<tr>
<td>G87</td>
<td>09</td>
<td>Bore/Manual Retract Canned Cycle</td>
</tr>
<tr>
<td>G88</td>
<td>09</td>
<td>Bore/Dwell Canned Cycle</td>
</tr>
<tr>
<td>G89</td>
<td>09</td>
<td>Bore Canned Cycle</td>
</tr>
<tr>
<td>G90</td>
<td>01</td>
<td>O.D.I.D. Turning cycle, Modal</td>
</tr>
<tr>
<td>G92</td>
<td>01</td>
<td>Thread Cutting Cycle, Modal</td>
</tr>
<tr>
<td>G94</td>
<td>01</td>
<td>End Face Cutting cycle, Modal</td>
</tr>
<tr>
<td>G96</td>
<td>12</td>
<td>Constant Surface Speed On</td>
</tr>
<tr>
<td>G97</td>
<td>12*</td>
<td>Constant Surface Speed Cancel</td>
</tr>
<tr>
<td>G98</td>
<td>05</td>
<td>Feed per Minute</td>
</tr>
<tr>
<td>G99</td>
<td>05*</td>
<td>Feed per Revolution</td>
</tr>
<tr>
<td>G102</td>
<td>00</td>
<td>Programmable Output to RS-232</td>
</tr>
<tr>
<td>G103</td>
<td>00</td>
<td>Block Look Ahead Limit (P0-P15)</td>
</tr>
<tr>
<td>G110-G129</td>
<td>12</td>
<td>Extra Work Coordinate System 7 through 26 older Machines</td>
</tr>
<tr>
<td>G154P1 to G154P99</td>
<td>12</td>
<td>Extra Work offsets newer machines</td>
</tr>
<tr>
<td>G184</td>
<td>09</td>
<td>Reverse Tap Canned Cycle</td>
</tr>
<tr>
<td>G187</td>
<td>00</td>
<td>Accuracy Control for machining corners. G187 Ennnn will modify setting 85 – maximum corner rounding</td>
</tr>
</tbody>
</table>
**Miscellaneous Functions (M Codes)**

M-codes are non-axes moving commands; also called *machine functions*.

The format for M-codes is the letter “M” followed by two numbers, for example M01.

Only one M-code is allowed on a single block of code.

M-codes are the last thing to take effect on a line of code.

M-codes are usually place at the end of a block of code.

- **M00** Program Stop
  - Stops spindle, axes movement, turns off coolant
  - On pressing CYCLE START button program will continue on from where it stopped

- **M01** Optional program stop
  - Works exactly the same as a M00
  - Is only active when optional stop key activates it on control panel

- **M02** End of program – cannot continue
  - Most common way is using M30

- **M03** Start spindle forward (Clockwise)
  - Must be accompanied by a spindle speed

- **M04** Start spindle reverse (Counterclockwise)
  - Must have a spindle speed (S value)

- **M05** Spindle Stop

- **M08** Coolant on Command

- **M09** Coolant off Command

- **M10** Clamp Spindle Chuck

- **M11** Unclamp Spindle Chuck

- **M21** Tailstock Forward

- **M22** Tailstock Reverse

- **M23** Thread Chamfer On during a G76 or G92 cycle
M24  Thread Chamfer Off
M30  Program end and rewind to beginning of program
M41  Low Gear
M42  High Gear
M85  Automatic Door Open
M86  Automatic Door Close
M88  High Pressure Coolant On
M89  High Pressure Coolant Off
M97  Local Sub-Program Call (P, L)
M98  Sub Program Call (P, L)
M99  Sub Program Return or Loop

**NOTE: Only one "M" code can be used per line.** The "M" code will be the last item of code to be performed, regardless of where it is located in the line.

**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 entered for G54 if no other work coordinate code was specified in the actual program. The machine automatically recognizes the G54 column upon start-up. This is known as a default.

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

- G00  Rapid Traverse
- G18  X, Z 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), but once an F code is programmed, it will apply until another feed rate is entered or the machine is turned off.
**Lathe Programming**

Program format, or program style, is an important part of CNC machining. Each individual will format their programs differently. The point is that a programmer needs to be consistent and efficient. For example:

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. Write X first, Z second.

The **first line** or block in a program should be a return to machine zero (using G28). Any tool change should be after a return to machine zero. Although this is not necessary, it is a good safety measure. With extended live tools, tail stocks or sub-spindles significant clearance issues appear. As a result, programmers will take the machine home first in X with either G28 U0 or G53 X0 commands. Then take the machine home in Z with a G28 W0 or G53 Z0 command. Another option is to take the machine to a safe Z index position. This is done by using machine coordinates, G53 Z-5.0 as an example.

**No safety line?** Examples in this book do not have safety lines. A possible safety line could be:

```
G18 G20 G40 G54 G80 G97 G99
```

CAD/CAM systems will by default output a safety line in the beginning of a program or before a tool change. This is done to make sure the machine is in a safe start condition. Some programmers however don’t employ a safety line in lathes the reasons are given below:

**G18:** G18 is the default command for designating which plane (X,Z plane) a radius is thrown. If an arc is attempted in a G17 (XY plane) or a G19 (YZ plane) the Haas lathe will alarm out.

**G20:** G20(inch) and G21(mm) make sure setting #9 is set to inch or metric. Normally most shops are either inch or metric. If the wrong setting is active it will be clearly seen in graphics or the machine will give an over travel alarm.

**G40:** G40 cancels cutter compensation codes G41 or G42. Cutter compensation should be cancelled after it is used. If it is not cancelled and the program is run in graphics alarm 349 PROG STOP W/O CANCEL CUTTER COMP. As a result the machine will pick up the mistake anyway.

**G54:** Work offsets are not used as in mills. If no pre-setter is used many times there are no values in the work offsets register. All the Z offsets are reflected in the Tool Geometry offsets. If no work offsets are used some programs will not have G54 in them. G54 is however the default value and if not changed will always be active.

**G80:** Cancels canned cycles. If it is missing in a program a G00, G01, M30 or RESET will automatically cancel any can cycle.
G97: Constant spindle speed. A G97 should be programmed at the beginning and end of every tool in a program with a spindle speed.

G99: Feed per revolution: This is the default. Normally lathes are always in G99 mode, they are never changed to G98 (Feed per minute). The only time G98 would be used if live tooling is being used. If you are in G98 mode when G99 is required the feed rates are so small (.009) the machine would just move very slow taking forever to complete the command. If you are in G99 mode and a typical Feed/Minute say 10 inches/minute the machine at a speed of 200 RPM the machine would interpret this as 2000 inch per minute way above the feed limit on the machine and alarm out.

So it is up to the individual programmer to decide is a safety line is used.

The second line of code indexes the turret to the appropriate tool giving proper geometry or tool shifts.

The third line may contain a spindle speed maximum, G50, for the lathe during any operation.

The fourth line or block should cancel any constant surface speed mode (G97) and specify a spindle speed command (S____) along with a spindle ON clockwise command (M03). This speed should correspond to a constant surface speed (G96) in line N6 at the diameter indicated in N5. If not the lathe will hesitate until it gets to the proper speed and sound strange.

The fifth line should contain a preparatory code (G00) for a rapid movement to a start X and Z position. It should specify the work coordinate being used, and to turn on the coolant (M08).

The sixth line may optionally specify a Constant Surface Speed (G96) and a surface feet per minute (SFM) defined with an (S____) command.

An example of the first five lines of a program might look like this:

N1 G28;
N2 T101;
N3 G50 S2000;
N4 G97 S1146 M03;
N5 G54 G00 X1.5 Z.02 M08;
N6 G96 S450;

This format is a good example and defines a commonly used program style.

The format defines the “language of the machine tool.” A description of the particular words a machine may accept, the order in which they must appear, the number of numeric digits associated with each word, the location of a decimal point, and the presence or absence of signs is explained in the following pages.
**Notes on Program Structure**

A CNC program consists of a one or more blocks of commands. A block of code may be considered a line of text. At the end of the block is a semicolon (;). This denotes the end of the text or block and is also called a (EOB) end of block symbol. When at the machine and manually entering a program pressing the EOB key will enter the semicolon symbol and move down to the next line in the program.

Programs must begin and end with a percentage symbol (%). These symbols will not be seen on the controls of the Haas. If however you save the program to say a USB device and look at the program using a computer they will be there. If you are trying to load a program into the CNC without percentage symbols at the beginning and end of a program the CNC machine will alarm out and not accept the program.

After the first % sign the next line of the program requires a program name using the letter O and a four digit number on older machines or 5 digit number on newer machines. This symbol identifies a particular program and is used to call up the correct program. On the program number line, text in parenthesis, example (P/N 15466 JD shaft) may further be used to give the program a title. Text inside the ( ) is not seen or used by the machine. This text will show up in list program screen of the memory.

Programs may also contain forward slashes (/). This is denotes an optional block. When BLOCK DELETE is active the program will ignore any commands on any line containing a (/).
Machining Cycles for the Lathe

A machining cycle is used to simplify the programming of a part. Machine cycles are defined for the most common machining operations. There are two types: machining cycles for turning and grooving and canned cycles for drilling, tapping, and boring.

Drilling, Tapping and Boring Canned Cycles

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 executed in the Z-axis for each positioning of X axes in a program. Some of the cycle command values can be changed after these cycles 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 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) move between each cycle.

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

- 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
**Machine Cycles for Turning and Grooving**

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

- **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 Turning with Chip Break Cycle
- **G75**  O.D./I.D. Grooving or Turning with 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 defined for the most common, repetitive operations such as turning, facing, threading and grooving. There are both modal and non-modal machine cycles. Modal cycles, such as turning cycle G90, remain in effect after they are defined. After any subsequent X or Z-axis positioning, the canned cycle is executed again. Modal canned cycles remain in effect until canceled by a G80, G00, end of program, or RESET. Non-modal machine cycles are effective only for the block that contains them, but will perform a series of machining moves to perform that command.
**Linear/Circular Movement-Creating Tool Paths**

CNC programs consist of lines and arcs. This unit will discuss the proper methods to create tool paths applied to CNC programs to create the desired results in addition to creating tool paths with and without cutter compensation.

**Objectives:**

Upon completion of this unit, the student will:

1) Understand the major differences in G01 and G00.

2) Understand and apply the five criteria needed to produce an arc.

3) Understand the principles of programming and applications with and without cutter compensation in addition to the advantages and disadvantages of each.

4) Be capable of producing a tool path program, containing lines and arcs, with and without cutter compensation.

5) Be capable of determining feeds and speeds given an SFM and cutting tool, and be capable of determining a feed rate given an RPM and chip load.

6) Be able to integrate a tool path into a part program.

7) Understand the rules governing the use of cutter compensation.

8) Have a basic understanding of the concept of arc in/arc out and some of its applications.
**G00 Rapid Motion Positioning**

- X  Absolute X-axis rapid motion command
- Z  Absolute Z-axis rapid motion command
- U  Incremental X-axis rapid motion command
- W  Incremental Z-axis rapid motion command
- B  Absolute tailstock rapid motion command

The G code is used to cause a rapid traverse of the two axes of the machine from one programmed point to the next programmed point. The auxiliary (tailstock) B-axis can also be moved with a G00. This G code is modal so that all following blocks will be in rapid motion until another group 01 is specified.

Generally, rapid motions done together in both X and Z-axes will not be in a straight line from one program location to the next program location. All of the axes specified are moved at the same speed (945 IPM), but will not necessarily complete their motions at the same time. Each axis slide is activated independently and the axis with the shortest distance to move will reach its endpoint first; thus, giving the impression of a 45-degree angle move. The control will wait until all motions are complete. These rapid moves may be made 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.

To move from point “A” to point “B”, the programmed line can be either:
Interpolation Commands

G01  Linear Interpolation Motion

<table>
<thead>
<tr>
<th>Letter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Absolute X-axis feed motion command</td>
</tr>
<tr>
<td>Z</td>
<td>Absolute Z-axis feed motion command</td>
</tr>
<tr>
<td>U</td>
<td>Incremental X-axis feed motion command</td>
</tr>
<tr>
<td>W</td>
<td>Incremental Z-axis feed motion command</td>
</tr>
<tr>
<td>B</td>
<td>Absolute tailstock feed motion command</td>
</tr>
<tr>
<td>F</td>
<td>Feed rate</td>
</tr>
</tbody>
</table>

This G code provides for straight-line (linear) motion from point-to-point. Motion can occur in one or two 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 feed rate 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 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. Only those axes specified are moved in either absolute XZ or incremental UW commands.

<table>
<thead>
<tr>
<th>Command</th>
<th>X</th>
<th>Z</th>
<th>F</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>0.9105</td>
<td>0.1</td>
<td></td>
</tr>
<tr>
<td>G01</td>
<td>1.5</td>
<td>-1.0</td>
<td>0.006</td>
</tr>
<tr>
<td>Z-3.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>X2.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>G00</td>
<td>0.9105</td>
<td>0.1</td>
<td></td>
</tr>
<tr>
<td>G01</td>
<td>U.5895</td>
<td>W-1.1</td>
<td>0.006</td>
</tr>
<tr>
<td>W-1.9</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>U.5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>G00</td>
<td>0.9105</td>
<td>0.1</td>
<td></td>
</tr>
<tr>
<td>G01</td>
<td>1.5</td>
<td>W-1.1</td>
<td>0.006</td>
</tr>
<tr>
<td>Z-3.0</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>U.5</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**Linear Interpolation Exercise**

FORMAT:  G01 (Linear interpolation [feed] Corner Chamfering for Absolute or Incremental).

With what we have learned about G01, we can move a cutter along the part profile by a series of Absolute X and Z-axis movements or Incremental U (X-axis) and W (Z-axis) movements, which can be a combination of both X and Z absolute movements and U and W incremental movements. Define the actual points around this part using both incremental and absolute.

<table>
<thead>
<tr>
<th>ABSOLUTE PROGRAMMING</th>
<th>INCREMENTAL AND ABSOLUTE PROGRAMMING</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00 X.25 Z.1</td>
<td>G00 X.25 Z.1</td>
</tr>
<tr>
<td>N1 G___ Z0. F.006</td>
<td>N1 G___ Z0. F.006</td>
</tr>
<tr>
<td>N2 X_____</td>
<td>N2 X_____</td>
</tr>
<tr>
<td>N3 X____ Z-______</td>
<td>N3 U_____ W-______</td>
</tr>
<tr>
<td>N4 Z-______</td>
<td>N4 Z-______</td>
</tr>
<tr>
<td>N5 X____ Z-______</td>
<td>N5 U_____ Z-______</td>
</tr>
<tr>
<td>N6 X_____</td>
<td>N6 X_____</td>
</tr>
<tr>
<td>N7 X____ Z-______</td>
<td>N7 X_____ W-______</td>
</tr>
<tr>
<td>N8 Z-______</td>
<td>N8 Z-______</td>
</tr>
</tbody>
</table>
Circular Interpolation Commands

G02  CW Circular Interpolation Motion

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  Distance from start point to arc center in the X-axis
K  Distance from start point to arc center in the Z-axis
R  Radius of circle (If I and K are not used)
F  Feed rate

G03  CCW Circular Interpolation Motion

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  Distance from start point to arc center in the X-axis
K  Distance from start point to arc center in the Z-axis
R  Radius of circle (If I and K are not used)
F  Feed rate

G03 will generate a counterclockwise circular motion, but is otherwise defined the same as a 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 basic command formats for defining circular interpolation depending on whether the I and K method or the R method is used to define the arc center.

R is the distance from the starting point to the center of the arc. X or Z is required to specify an endpoint different from the starting point. With a positive R, the control will generate a circular path of 180 degrees or less, but to generate a circular path of over 180 degrees, specify a negative R.
**G02 CW Circular Interpolation Motion (example)**

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

<table>
<thead>
<tr>
<th>Plane selection command</th>
<th>Plane selection command</th>
<th>Plane selection command</th>
<th>Arc start position coordinates</th>
<th>Rotation direction</th>
<th>Arc end position Absolute or Incremental</th>
<th>I and K method (arc center coordinates)</th>
<th>R method (arc radius)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G17</td>
<td>G18</td>
<td>G19</td>
<td>X,Z</td>
<td>G02</td>
<td>X,Z or U,W</td>
<td>I,K</td>
<td>R</td>
</tr>
<tr>
<td>Arc parallel to XY-plane (not available)</td>
<td>Arc parallel to ZX-plane (default)</td>
<td>Arc parallel to YZ-plane (not available)</td>
<td>Coordinates to the start position of arc</td>
<td>Clockwise interpolation direction</td>
<td>Coordinates of the end position of arc defined from the part zero or part origin</td>
<td>Incremental distance and direction from start position of arc to the arc center for X and Z-axes</td>
<td>Arc radius value of arc.</td>
</tr>
</tbody>
</table>
G03 CCW Circular Interpolation Motion (example)
To move the cutter in a clockwise direction from the START POINT to the END POINT, the programming coding would be like this:

N11
N12 G00 X0. Z.1  (tool rapids to X0 Z.1 in front of part)
N13  G01 Z0. F.012  (tool feeds to Z0. face of part)
N14  X2.  (tool feeds up in X-axis to a 2. dia.)
N15  Z-1.  (tool at 2 dia. machines to the arc start point at Z-1.)
N16 G02 X5. Z-2.5 R1.5  (cuts the 1.5 radius to the arc end point at X5. Z-2.5) (I+1.5 K0)
N17  G01 X6.0  (tool feeds up in X-axis to a 6 dia.)
N18..................
Circular Interpolation Motion Exercises

Exercise #1: Exercise for G01 and G02

From the tool’s current position, program the cutter path to turn the 1.500 diameter up to the START POINT of the 1.000 radius and then interpolate the radius. Finish the cut by feeding up to the 4.500 diameter.

```
N9   G00 X1.5 Z.100
N10 G01  Z-________  F.01
N11 G____  X________ Z________  I________  K________
N12 G____  X________
```

From the tool’s current position, feed into face, face down to the START POINT of radius, machine the radius down into the bore, and then bore through to the end of the part.

```
N19 G00 X4.0 Z.10
N20 G01  Z0.  F.01
N21 X________
N22 G____  X________ Z________  I________  K________
N23 G____  Z________
```
Exercise #2: Exercise for G01 and G03

From the tool’s current position, program the cutter path to feed up the face to the START POINT of the 1.0 radius, machine the radius and then turn the 4.000 diameter to the end of the part.

From the tool’s current position, move into the face, feed up the Feed to face of part at X0, feed up to the 45 degree angle Start going to the START POINT of the radius, machine the start point then the 1.00 radius and turn the 5.422 diameter back to the end of the part. Back to 5.0
Circular Interpolation Exercise

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, which 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 of 180 degrees or less. For a radius larger than 180 degrees, it needs to have a minus (-) sign with a radius value.

Program Example
(G02 & G03 using I and K)

<table>
<thead>
<tr>
<th>G01  Z0</th>
<th>G01  Z0</th>
</tr>
</thead>
<tbody>
<tr>
<td>X1 .01</td>
<td>X1 .01</td>
</tr>
<tr>
<td>G03  X1.25  Z-.12  I_____ K_____</td>
<td>G03  X1.25  Z-.12  R_____</td>
</tr>
<tr>
<td>G01  Z-.89</td>
<td>G01  Z-.89</td>
</tr>
<tr>
<td>G02  X1.73  Z-1.13  I_____ K_____</td>
<td>G02  X1.73  Z-1.13  R_____</td>
</tr>
<tr>
<td>G01  X2.28</td>
<td>G01  X2.28</td>
</tr>
<tr>
<td>G03  X3.  Z-1.49  I_____ K_____</td>
<td>G03  X3.  Z-1.49  R_____</td>
</tr>
<tr>
<td>G01  Z-2.375</td>
<td>G01  Z-2.375</td>
</tr>
</tbody>
</table>
**Manually Programming Tool Nose Compensation**

This section is to help those who have not had the time or patience to learn to use TNC. Our suggestion is, if you want to save time and money to learn how to use TNC.

**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 to program a 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 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 you are machining.

Refer to the illustrations on the pages while reading the text below.

For a 90-degree corner radius, 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 re-calculate the start point and end point of this larger programmed radius.
For a 90-degree fillet radius, 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 re-calculate the start point and end point of this smaller programmed radius.

**Radius Calculation**

When a radius is required TNRC has to be applied. The basic rule is as follows:

- An external radius has the Tool Nose Radius added to it.
- An internal radius has the Tool Nose Radius subtracted from it.

The example below shows this:
**External Radius Calculation**

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

\[ .25 + .031 = .281 \]

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

G1 X0 Z0  \hspace{1cm} \text{Start of program}
G1 X.438  \hspace{1cm} 1.0 -(281 \times 2)=.438
G3 X1.0 Z-.281 R.281  \hspace{1cm} \text{Move to the X and Z axis end point}
G1 Z-?  \hspace{1cm} \text{Next axis parallel move}

**Internal Radius Calculation**

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

\[ .25-.031 = .219 \]

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

G1 X1. Z0
G1 Z-.781
G2 X 1.438 Z-1. R.219
G1 X-?

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

Manually calculate compensation for a lathe tool with a .031 TNR to machine a radius on your part.

![Diagram of a part with radii and dimensions]

### Manual Programming A Radius

**G02 & G03 using I and K**

```plaintext
O0102
G28
T101
G97 S1200 M03
G54 G00 X.92 Z.05
G01 Z0. F.01
X.948 F.006
G03 X1.25 Z-.151 I0. K-.151
G01 Z-.921
G02 X1.668 Z-1.13 I.209 K0.
G01 X2.218
G03 X3. Z-1.521 I0. K-.391
G01 Z-2.375
G00 U.01 Z1.0
G28
```

### Manual Programming A Radius

**G02 & G03 using R**

```plaintext
O0103
G28
T101
G97 S1200 M03
G54 G00 X.92 Z.05
G01 Z0. F.01
X.948 F.006
G03 X1.25 Z-.151 R.151
G01 Z-.921
G02 X1.668 Z-1.13 R.209
G01 X2.218
G03 X3. Z-1.521 R.391
G01 Z-2.375
G00 U.01 Z1.0
G28
```
Circular Interpolation Exercise

O0010
(Circular Interpolation “Without” Cutter Compensation)
N1 (Finish O.D.)
G28
T101 (O.D. Turning Tool .031 TNR)
G50 S2200
G97 S682 M03
G54 G00 X4.2 Z0.1 M08
G96 S750
G00 Z0.
G01 X-0.062 F0.004
G0 X3.5 Z0.02

G01 Z0. F0.005
X_______
G03 X_______ Z_______ R_______
G01 Z_______
G02 X_______ Z_______ R_______
G01 X_______
G03 X_______ Z_______ R_______
G01 Z_______
G00 U0.02 Z1. M09
G97 S320
G28
M30

Fill in the blanks to complete one finish pass of the cutter path for the above part defining the actual line of part “without” using cutter compensation (G41 or G42) and adding in the actual radius compensation for a .031 radius O.D. turning tool.
Calculating Compensation for “An Angle” 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 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 insert you are using. If you were to try to program your part without using any compensation you would see overcutting and undercutting on your part.

The following pages contain tables and illustrations demonstrating how to calculate the compensation in order to program your part accurately. There are two tables with 1/32 and 1/64 insert radius values in both the X and Z-axes.

Along with the charts is an illustration example of compensation using offset values added to your program moves. Next is a program exercise using these values to define your part for machining an angle.

Taper Calculation

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

<table>
<thead>
<tr>
<th>ANGLE</th>
<th>X-OFFSET .015 N/R</th>
<th>Z-OFFSET .015 N/R</th>
<th>X-OFFSET .031 N/R</th>
<th>Z-OFFSET .031 N/R</th>
</tr>
</thead>
<tbody>
<tr>
<td>15</td>
<td>.0072</td>
<td>.0136</td>
<td>.0146</td>
<td>.0271</td>
</tr>
<tr>
<td>30</td>
<td>.0132</td>
<td>.0114</td>
<td>.0264</td>
<td>.0229</td>
</tr>
<tr>
<td>45</td>
<td>.0184</td>
<td>.0092</td>
<td>.0366</td>
<td>.0183</td>
</tr>
<tr>
<td>60</td>
<td>.0228</td>
<td>.0066</td>
<td>.0458</td>
<td>.0132</td>
</tr>
</tbody>
</table>

X-OFFSET numbers above are diametric.
**Tool Nose Compensation – Taper Calculation**

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

\[
\text{TNRC for } X = R - (\tan \theta \times R)
\]

If the angle is 30° and the radius is 0.03125, the X-axis calculation is:

\[
X = 0.03125 - (\tan 30 \times 0.03125) = 0.03125 - (0.57735 \times 0.03125) = 0.03125 - 0.01804 = 0.013207
\]

(Note this is a radial number)

\[
\text{TNRC for } Z = \frac{\text{Compensation in } X}{\tan \theta}
\]

The Z-axis calculation is:

\[
Z = \frac{0.013207}{0.57735} = 0.02287
\]

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

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

The following tables are taken from SL-Series Programming and Operation Manual.

Note that Xc values do not have to be multiplied by 2 as they are already calculated as diametric numbers.
## TOOL RADIUS AND ANGLE CHART

**1/32 RADIUS**

NOTE: THE MEASUREMENT CALCULATED IS BASED ON PART DIAMETER

<table>
<thead>
<tr>
<th>ANGLE</th>
<th>(X_c) CROSS</th>
<th>(Z_c) LONGITUDINAL</th>
<th>ANGLE</th>
<th>(X_c) CROSS</th>
<th>(Z_c) LONGITUDINAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>.0010</td>
<td>.0310</td>
<td>46.</td>
<td>.0372</td>
<td>.0180</td>
</tr>
<tr>
<td>2.</td>
<td>.0022</td>
<td>.0307</td>
<td>47.</td>
<td>.0378</td>
<td>.0177</td>
</tr>
<tr>
<td>3.</td>
<td>.0032</td>
<td>.0304</td>
<td>48.</td>
<td>.0386</td>
<td>.0173</td>
</tr>
<tr>
<td>4.</td>
<td>.0042</td>
<td>.0302</td>
<td>49.</td>
<td>.0392</td>
<td>.0170</td>
</tr>
<tr>
<td>5.</td>
<td>.0052</td>
<td>.0299</td>
<td>50.</td>
<td>.0398</td>
<td>.0167</td>
</tr>
<tr>
<td>6.</td>
<td>.0062</td>
<td>.0296</td>
<td>51.</td>
<td>.0404</td>
<td>.0163</td>
</tr>
<tr>
<td>7.</td>
<td>.0072</td>
<td>.0293</td>
<td>52.</td>
<td>.0410</td>
<td>.0160</td>
</tr>
<tr>
<td>8.</td>
<td>.0082</td>
<td>.0291</td>
<td>53.</td>
<td>.0416</td>
<td>.0157</td>
</tr>
<tr>
<td>9.</td>
<td>.0092</td>
<td>.0288</td>
<td>54.</td>
<td>.0422</td>
<td>.0153</td>
</tr>
<tr>
<td>10.</td>
<td>.01</td>
<td>.0285</td>
<td>55.</td>
<td>.0428</td>
<td>.0150</td>
</tr>
<tr>
<td>11.</td>
<td>.0111</td>
<td>.0282</td>
<td>56.</td>
<td>.0434</td>
<td>.0146</td>
</tr>
<tr>
<td>12.</td>
<td>.0118</td>
<td>.0280</td>
<td>57.</td>
<td>.0440</td>
<td>.0143</td>
</tr>
<tr>
<td>13.</td>
<td>.0128</td>
<td>.0277</td>
<td>58.</td>
<td>.0446</td>
<td>.0139</td>
</tr>
<tr>
<td>14.</td>
<td>.0136</td>
<td>.0274</td>
<td>59.</td>
<td>.0452</td>
<td>.0136</td>
</tr>
<tr>
<td>15.</td>
<td>.0146</td>
<td>.0271</td>
<td>60.</td>
<td>.0458</td>
<td>.0132</td>
</tr>
<tr>
<td>16.</td>
<td>.0154</td>
<td>.0269</td>
<td>61.</td>
<td>.0464</td>
<td>.0128</td>
</tr>
<tr>
<td>17.</td>
<td>.0162</td>
<td>.0266</td>
<td>62.</td>
<td>.047</td>
<td>.0125</td>
</tr>
<tr>
<td>18.</td>
<td>.017</td>
<td>.0263</td>
<td>63.</td>
<td>.0474</td>
<td>.0121</td>
</tr>
<tr>
<td>19.</td>
<td>.018</td>
<td>.0260</td>
<td>64.</td>
<td>.0480</td>
<td>.0117</td>
</tr>
<tr>
<td>20.</td>
<td>.0188</td>
<td>.0257</td>
<td>65.</td>
<td>.0486</td>
<td>.0113</td>
</tr>
<tr>
<td>21.</td>
<td>.0196</td>
<td>.0255</td>
<td>66.</td>
<td>.0492</td>
<td>.0110</td>
</tr>
<tr>
<td>22.</td>
<td>.0204</td>
<td>.0252</td>
<td>67.</td>
<td>.0498</td>
<td>.0106</td>
</tr>
<tr>
<td>23.</td>
<td>.0212</td>
<td>.0249</td>
<td>68.</td>
<td>.0504</td>
<td>.0102</td>
</tr>
<tr>
<td>24.</td>
<td>.022</td>
<td>.0246</td>
<td>69.</td>
<td>.051</td>
<td>.0098</td>
</tr>
<tr>
<td>25.</td>
<td>.0226</td>
<td>.0243</td>
<td>70.</td>
<td>.0514</td>
<td>.0094</td>
</tr>
<tr>
<td>26.</td>
<td>.0234</td>
<td>.0240</td>
<td>71.</td>
<td>.052</td>
<td>.0090</td>
</tr>
<tr>
<td>27.</td>
<td>.0242</td>
<td>.0237</td>
<td>72.</td>
<td>.0526</td>
<td>.0085</td>
</tr>
<tr>
<td>28.</td>
<td>.025</td>
<td>.0235</td>
<td>73.</td>
<td>.0532</td>
<td>.0081</td>
</tr>
<tr>
<td>29.</td>
<td>.0256</td>
<td>.0232</td>
<td>74.</td>
<td>.0538</td>
<td>.0077</td>
</tr>
<tr>
<td>30.</td>
<td>.0264</td>
<td>.0229</td>
<td>75.</td>
<td>.0542</td>
<td>.0073</td>
</tr>
<tr>
<td>31.</td>
<td>.0272</td>
<td>.0226</td>
<td>76.</td>
<td>.0548</td>
<td>.0068</td>
</tr>
<tr>
<td>32.</td>
<td>.0278</td>
<td>.0223</td>
<td>77.</td>
<td>.0554</td>
<td>.0064</td>
</tr>
<tr>
<td>33.</td>
<td>.0286</td>
<td>.0220</td>
<td>78.</td>
<td>.056</td>
<td>.0059</td>
</tr>
<tr>
<td>34.</td>
<td>.0252</td>
<td>.0217</td>
<td>79.</td>
<td>.0564</td>
<td>.0055</td>
</tr>
<tr>
<td>35.</td>
<td>.03</td>
<td>.0214</td>
<td>80.</td>
<td>.057</td>
<td>.0050</td>
</tr>
<tr>
<td>36.</td>
<td>.0306</td>
<td>.0211</td>
<td>81.</td>
<td>.0576</td>
<td>.0046</td>
</tr>
<tr>
<td>37.</td>
<td>.0314</td>
<td>.0208</td>
<td>82.</td>
<td>.0582</td>
<td>.0041</td>
</tr>
<tr>
<td>38.</td>
<td>.032</td>
<td>.0205</td>
<td>83.</td>
<td>.0586</td>
<td>.0036</td>
</tr>
<tr>
<td>39.</td>
<td>.0326</td>
<td>.0202</td>
<td>84.</td>
<td>.0592</td>
<td>.0031</td>
</tr>
<tr>
<td>40.</td>
<td>.0334</td>
<td>.0199</td>
<td>85.</td>
<td>.0598</td>
<td>.0026</td>
</tr>
<tr>
<td>41.</td>
<td>.034</td>
<td>.0196</td>
<td>86.</td>
<td>.0604</td>
<td>.0021</td>
</tr>
<tr>
<td>42.</td>
<td>.0346</td>
<td>.0193</td>
<td>87.</td>
<td>.0608</td>
<td>.0016</td>
</tr>
<tr>
<td>43.</td>
<td>.0354</td>
<td>.0189</td>
<td>88.</td>
<td>.0614</td>
<td>.0011</td>
</tr>
<tr>
<td>44.</td>
<td>.036</td>
<td>.0186</td>
<td>89.</td>
<td>.062</td>
<td>.0005</td>
</tr>
<tr>
<td>45.</td>
<td>.0366</td>
<td>.0183</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
## TOOL RADIUS AND ANGLE CHART

1/64 RADIUS

**NOTE:** THE X MEASUREMENT CALCULATED IS BASED ON PART DIAMETER

<table>
<thead>
<tr>
<th>ANGLE</th>
<th>Xc CROSS</th>
<th>Zc LONGITUDINAL</th>
<th>ANGLE</th>
<th>Xc CROSS</th>
<th>Zc LONGITUDINAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>.00062</td>
<td>.0155</td>
<td>46.</td>
<td>.00186</td>
<td>.0090</td>
</tr>
<tr>
<td>2.</td>
<td>.00011</td>
<td>.0154</td>
<td>47.</td>
<td>.00191</td>
<td>.0088</td>
</tr>
<tr>
<td>3.</td>
<td>.00162</td>
<td>.0152</td>
<td>48.</td>
<td>.01921</td>
<td>.0087</td>
</tr>
<tr>
<td>4.</td>
<td>.00222</td>
<td>.0151</td>
<td>49.</td>
<td>.01961</td>
<td>.0085</td>
</tr>
<tr>
<td>5.</td>
<td>.00262</td>
<td>.0149</td>
<td>50.</td>
<td>.01981</td>
<td>.0083</td>
</tr>
<tr>
<td>6.</td>
<td>.00316</td>
<td>.0148</td>
<td>51.</td>
<td>.02021</td>
<td>.0082</td>
</tr>
<tr>
<td>7.</td>
<td>.00362</td>
<td>.0147</td>
<td>52.</td>
<td>.02041</td>
<td>.0080</td>
</tr>
<tr>
<td>8.</td>
<td>.00402</td>
<td>.0145</td>
<td>53.</td>
<td>.02081</td>
<td>.0078</td>
</tr>
<tr>
<td>9.</td>
<td>.00446</td>
<td>.0144</td>
<td>54.</td>
<td>.0211</td>
<td>.0077</td>
</tr>
<tr>
<td>10.</td>
<td>.00502</td>
<td>.0143</td>
<td>55.</td>
<td>.0214</td>
<td>.0075</td>
</tr>
<tr>
<td>11.</td>
<td>.00542</td>
<td>.0141</td>
<td>56.</td>
<td>.0216</td>
<td>.0073</td>
</tr>
<tr>
<td>12.</td>
<td>.00602</td>
<td>.0140</td>
<td>57.</td>
<td>.0221</td>
<td>.0071</td>
</tr>
<tr>
<td>13.</td>
<td>.00642</td>
<td>.0138</td>
<td>58.</td>
<td>.0222</td>
<td>.0070</td>
</tr>
<tr>
<td>14.</td>
<td>.00682</td>
<td>.0137</td>
<td>59.</td>
<td>.0226</td>
<td>.0068</td>
</tr>
<tr>
<td>15.</td>
<td>.00722</td>
<td>.0136</td>
<td>60.</td>
<td>.0228</td>
<td>.0066</td>
</tr>
<tr>
<td>16.</td>
<td>.00782</td>
<td>.0134</td>
<td>61.</td>
<td>.0232</td>
<td>.0064</td>
</tr>
<tr>
<td>17.</td>
<td>.00822</td>
<td>.0133</td>
<td>62.</td>
<td>.0234</td>
<td>.0062</td>
</tr>
<tr>
<td>18.</td>
<td>.00862</td>
<td>.0132</td>
<td>63.</td>
<td>.0238</td>
<td>.0060</td>
</tr>
<tr>
<td>19.</td>
<td>.00902</td>
<td>.0130</td>
<td>64.</td>
<td>.0241</td>
<td>.0059</td>
</tr>
<tr>
<td>20.</td>
<td>.00942</td>
<td>.0129</td>
<td>65.</td>
<td>.0244</td>
<td>.0057</td>
</tr>
<tr>
<td>21.</td>
<td>.00982</td>
<td>.0127</td>
<td>66.</td>
<td>.0246</td>
<td>.0055</td>
</tr>
<tr>
<td>22.</td>
<td>.01022</td>
<td>.0126</td>
<td>67.</td>
<td>.0248</td>
<td>.0053</td>
</tr>
<tr>
<td>23.</td>
<td>.01062</td>
<td>.0124</td>
<td>68.</td>
<td>.0252</td>
<td>.0051</td>
</tr>
<tr>
<td>24.</td>
<td>.01112</td>
<td>.0123</td>
<td>69.</td>
<td>.0254</td>
<td>.0049</td>
</tr>
<tr>
<td>25.</td>
<td>.01142</td>
<td>.0122</td>
<td>70.</td>
<td>.0258</td>
<td>.0047</td>
</tr>
<tr>
<td>26.</td>
<td>.01182</td>
<td>.0120</td>
<td>71.</td>
<td>.0260</td>
<td>.0045</td>
</tr>
<tr>
<td>27.</td>
<td>.01222</td>
<td>.0119</td>
<td>72.</td>
<td>.0264</td>
<td>.0043</td>
</tr>
<tr>
<td>28.</td>
<td>.01242</td>
<td>.0117</td>
<td>73.</td>
<td>.0266</td>
<td>.0041</td>
</tr>
<tr>
<td>29.</td>
<td>.01282</td>
<td>.0116</td>
<td>74.</td>
<td>.0268</td>
<td>.0039</td>
</tr>
<tr>
<td>30.</td>
<td>.01322</td>
<td>.0114</td>
<td>75.</td>
<td>.0272</td>
<td>.0036</td>
</tr>
<tr>
<td>31.</td>
<td>.01362</td>
<td>.0113</td>
<td>76.</td>
<td>.0274</td>
<td>.0034</td>
</tr>
<tr>
<td>32.</td>
<td>.01412</td>
<td>.0111</td>
<td>77.</td>
<td>.0276</td>
<td>.0032</td>
</tr>
<tr>
<td>33.</td>
<td>.01422</td>
<td>.0110</td>
<td>78.</td>
<td>.0280</td>
<td>.0030</td>
</tr>
<tr>
<td>34.</td>
<td>.01462</td>
<td>.0108</td>
<td>79.</td>
<td>.0282</td>
<td>.0027</td>
</tr>
<tr>
<td>35.</td>
<td>.01512</td>
<td>.0107</td>
<td>80.</td>
<td>.0286</td>
<td>.0025</td>
</tr>
<tr>
<td>36.</td>
<td>.01542</td>
<td>.0103</td>
<td>81.</td>
<td>.0288</td>
<td>.0023</td>
</tr>
<tr>
<td>37.</td>
<td>.01562</td>
<td>.0104</td>
<td>82.</td>
<td>.0291</td>
<td>.0020</td>
</tr>
<tr>
<td>38.</td>
<td>.01612</td>
<td>.0102</td>
<td>83.</td>
<td>.0294</td>
<td>.0018</td>
</tr>
<tr>
<td>39.</td>
<td>.01642</td>
<td>.0101</td>
<td>84.</td>
<td>.0297</td>
<td>.0016</td>
</tr>
<tr>
<td>40.</td>
<td>.01662</td>
<td>.0099</td>
<td>85.</td>
<td>.0299</td>
<td>.0013</td>
</tr>
<tr>
<td>41.</td>
<td>.01712</td>
<td>.0098</td>
<td>86.</td>
<td>.0302</td>
<td>.0011</td>
</tr>
<tr>
<td>42.</td>
<td>.01742</td>
<td>.0096</td>
<td>87.</td>
<td>.0305</td>
<td>.0008</td>
</tr>
<tr>
<td>43.</td>
<td>.01762</td>
<td>.0095</td>
<td>88.</td>
<td>.0308</td>
<td>.0005</td>
</tr>
<tr>
<td>44.</td>
<td>.01812</td>
<td>.0093</td>
<td>89.</td>
<td>.0311</td>
<td>.0003</td>
</tr>
<tr>
<td>45.</td>
<td>.01842</td>
<td>.0092</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Angular Straight Line Interpolation Exercise
- Manually Adding Radius Compensation

Manual Programming a Finish Pass
With an Angle using a .031 TNR:

```
O0020
T101
G97 S1200 M03
G54 G00 X.85 Z.1 M08
G01 Z0. F.03
X-_____ F.01 (Face)
G00 X_____ Z.02
G___ Z0. F.006
X_____  
X_____ Z-______
Z-______
X_____ Z-______
X_____  
X_____ Z-______
Z-______
G00 U.01 Z1.0 M09
G28
M30
```
AUTOMATIC G01 CORNER ROUNDED

AUTOMATIC CORNER ROUNDED 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

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

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

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

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.
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

Interpolation with G01 using I, K, R & A

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 G42 G01 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

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 G___ G42 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 (0.01 seconds); otherwise, the delay is in seconds.

**EXAMPLE:**
- G04 P1.0 (for a delay of 1.0 second)
- G04 P25 (for a delay of 25 milliseconds)

### INCH / METRIC SELECTION (G20, G21)

Selecting between inch and metric programming can only be done from the Setting page, Setting 9. When changing the setting from inches to metric all work offsets and tool offsets will be converted to mm. The machine will not convert the programs however to metric. This must be done by the programmer.

The G codes G20 and G21 are sometimes 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.

### Reference Point Definition and Return

#### G28 RETURN TO MACHINE ZERO AND CANCEL OFFSET (FANUC)

The G28 code is used to 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. If you do not want to position through an intermediate point while specifying an X or Z-axis to position to machine home, then use an incremental U0 and/or W0 command for the specific axis you want to send to machine zero. G28 U0 or G28 W0 will command those axes specified to position incrementally to a zero distance as an intermediate point, and then those axes will go directly to machine zero. G28 also cancels tool offsets.

#### G51 RETURN TO MACHINE ZERO AND CANCEL OFFSET (YASNAC)

G51, used on a YASNAC control, cancels out any existing tool wear and work coordinate shift offsets and then returns to the machine zero position.
**Spindle Speed Commands**

**G50  SPINDLE SPEED CLAMP**

G50 can be used to clamp 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  S650  M03;  (Enter constant surface speed mode, spindle on)
```

The effect of centrifugal force on its gripping force, unbalanced condition of the work piece, etc., can be constrained by programming a G50 code.

The MAXIMUM spindle speed is designated by the “S” word along with the G50 preparatory command and must be in the same block.

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) exceeding 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 not having work zero and geometry offset capabilities. This will not be covered because it is not desirable or as easy to use.

**G96  CONSTANT SURFACE SPEED ON**

G96 commands the control to maintain a constant surface speed of the part relative to the tool tip. Surface speed is based on the distance of the tool tip to the spindle center (radius of the cut). Constant surface speed is maintained by adjusting the spindle speed based on the radius of the cut. The current S code is used to determine the surface speed. G96 is modal.

**G97  CONSTANT SURFACE SPEED OFF**

G97 commands the control to NOT adjust the spindle speed based on the radius of cut. It is used to cancel any current G96 command. When G97 is in effect, any S command is in revolution per minute (RPM). G97 is modal.
Work Coordinate System Selection

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  CHILD OR LOCAL COORDINATE SYSTEM

G52 selects the child coordinate system. It is non-modal and FANUC compatible. M30, Reset or power down will remove any values in G52.

G53  NON-MODAL MACHINE COORDINATE SELECTION

G53 temporarily cancels work coordinates offset and uses the machine coordinate system. It is non-modal, therefore, the next block will revert to the work coordinate that was active.

G54-59  SELECT COORDINATE SYSTEM #1 THRU #6

These codes select one of the six user coordinate systems stored within the offset memory. All subsequent references to positions of axes will be interpreted in the new coordinate system. Work coordinate system offsets are entered from the Offsets display page.

G110-G129  COORDINATE SYSTEM #7 THRU #26 OR
G154P1 to G154 P99

Feed Command Functions

G98  FEED PER MINUTE

G98 affects how the F address code is interpreted. The value of F indicates inches per minute when Setting 9 is set to INCH and millimeters-per-minute when Setting 9 is set to METRIC. This code is modal.

\[ \text{IPM} = \text{Current RPM} \times \text{IPR} \]

G99  FEED PER REVOLUTION

G99 affects how the F address is interpreted. The value of F indicates inches-per-revolution of the spindle when Setting 9 is set to INCH and millimeters-per-revolution of the spindle when Setting 9 is set to METRIC. This code is modal and is the default feed mode.

\[ \text{IPR} = \frac{\text{IPM}}{\text{Current RPM}} \]
**Tool Nose Compensation G Codes**

**G40  TOOL NOSE COMPENSATION CANCEL**

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>X-axis absolute departure location</td>
</tr>
<tr>
<td>Z</td>
<td>Z-axis absolute departure location</td>
</tr>
<tr>
<td>U</td>
<td>X-axis incremental departure distance</td>
</tr>
<tr>
<td>W</td>
<td>Z-axis incremental departure distance</td>
</tr>
<tr>
<td>I</td>
<td>X-axis intersection vector direction, (radius)</td>
</tr>
<tr>
<td>K</td>
<td>Z-axis intersection vector direction</td>
</tr>
</tbody>
</table>

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

The departure target location usually does not correspond with a point on the part. In many cases, overcutting or undercutting can occur. Figure 3 illustrates this.

When address codes I and K are used in a G40 departure block, the control will use these values as an intersection vector for the end point of the last, completely compensated, motion stroke. Figure 4 illustrates where I and K lie in relation to the departure stroke. Usually I and K lie along a face of the machined part.
G41  TOOL NOSE COMPENSATION – LEFT

G41 will select tool nose compensation left; that is, the tool is moved to the left of the programmed path to compensate for the size of a tool. A tool offset must be selected with a Tnxx code, where xx corresponds to the offsets to be used with the tool. If a negative radius is specified, tool nose compensation will operate as though a G42 was programmed.

G42  TOOL NOSE COMPENSATION – RIGHT

G42 will select tool nose compensation right; that is, the tool is moved to the right of the programmed path to compensate for the size of a tool. A tool offset must be selected with a Tnxx code, where xx corresponds to the offsets to be used with the tool.
Tool Nose Compensation Programming

Overview

Tool nose compensation is a feature that allows the user to adjust a programmed tool path for normal cutter wear or in response to differing cutter tool nose radius sizes. The user can do this by entering minimal offset data at runtime without any additional programming effort.

When to Use Tool Nose Compensation

Tool nose compensation is used when the tool nose radius changes and cutter wear is to be accounted for with curved surfaces or tapered cuts. Tool nose compensation generally does not need to be used when programmed cuts are solely along the X-axis (diameters) or Z-axis (faces). For angled and circular cuts, as the tool nose radius changes, undercutting or overcutting can occur. In the figure below, suppose that immediately after setup, C1 is the radius of the cutter that cuts the programmed tool path. As the cutter wears to C2, the operator might adjust the tool geometry offset to bring the part length and diameter to dimension. If this were done, as shown in Figure 8, a smaller radius would occur. If tool nose compensation is used, an incorrect cut does not occur. The control will automatically adjust the programmed path based on the offset for tool nose radius as set up in the control. The control will alter or generate code to cut the proper part geometry.
Note that in the above figure, the programmed path coincides with the final part dimension. Although parts do not have to be programmed using tool nose compensation, it is the preferred method because it makes program problems easier to detect and resolve.
**Tool Nose Compensation Concepts**

Tool nose compensation works by shifting the PROGRAMMED TOOL PATH to the right or to the left. The programmer will usually program the tool path to the current manufacturing operation finish size. When tool nose compensation is used, the control will compensate for the diameter of a tool based on special instructions written into the program. Two G-code commands are used to do this for compensation within a two-dimensional plane. G41 commands the control to shift to the left of the programmed tool path, and G42 commands the control to shift to the right of the programmed tool path. Another command, G40, is provided to cancel any shift made by tool nose compensation. G40, G41, and G42 are described in detail later in the “Using Tool Nose Compensation G Codes” section.

![Shift direction diagram](image)

Shift direction

The shift direction is based on the direction of the tool movement relative to the tool and the side of the part it is located. When thinking about which direction the compensated shift will occur in tool nose compensation, imagine yourself standing on the line of the part to be cut, looking in the direction the cutter is traveling. Which side of the line is the cutter going to pass you on, left or right? G41 will shift the tool tip on the left and G42 will shift 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 for the correct tool compensation, while normal I.D. turning will require a G41.
Imaginary Tool Tip and Direction

Tool nose compensation assumes that a compensated tool has a radius at the tool tip that it must compensate for. This is known as the TOOL NOSE RADIUS (TNR). For a lathe, it is not easy to determine the center of a tool nose radius. The cutting edges are set when a tool is touched off to record tool geometry. The control can calculate where the center of the tool radius is by using the tool geometry offset information, the radius of the tool, the shift direction the cutter is expected to cut, and the direction the cutter is pointing. The X and Z-axis' geometry offsets intersect at a point, called the IMAGINARY TOOL TIP, which determines 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 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.
Using Tool Nose Compensation

Tool nose compensation accomplishes its task by reading ahead one or two blocks to determine how to modify the current block of code. This is referred to as BLOCK LOOKAHEAD or LOOKAHEAD 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 the G41 or G42 command. When this command is executed, the control will look ahead to determine where the first compensated move will be. The first compensated move is generally a move from a non-compensated position to a compensated position and is therefore unusual. This first move is called the APPROACH move and is required when using tool nose compensation. Similarly, a DEPART move is also required. In a DEPART move, the control will move from a compensated position to a non-compensated position. A depart move occurs when tool nose compensation is cancelled with a G40 command or Txx00 command. Although approach and depart moves can be precisely planned, they are generally uncontrolled moves and the tool should not be in contact with the part when they occur.

Follow the steps below when using tool nose compensation.

PROGRAM the part to finished dimensions.

1. **Approach and Departure Moves:** Ensure that there is an approach move for each compensated path that is executed and determine if G41 or G42 is to be used. Ensure that there is also a departure move for each compensated path. Approach and departure moves should be equal to or greater than the tool nose radius of the tool.

2. **Tool Geometry Offsets:** Set the tool length geometry offsets and clear the length wear offsets of each tool.

3. **Tool Nose Radius and Wear Offsets:** Select a standard insert (with a defined radius) that will be used for each tool. Set the tool nose radius offset for each compensated tool. Clear the corresponding tool nose wear offset to start at zero for each tool.

4. **Tool Tip Direction:** Input the tool tip direction for each tool that is using compensation, G41 or G42.

5. **Check 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: an alarm will be generated indicating compensation interference or the incorrect geometry will be seen generated in graphics mode.

6. **Run and Inspect First Article:** Adjust wear offsets for the setup part.

Each of the above steps is described in detail in the following sections.
1. Approach And Departure 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, 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. This is shown in the following figure.

Approach and Departure moves

Any line containing a G40 will cancel tool nose compensation. This is called the DEPARTURE move. The last move must be a linear move, 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: Figure 13 (below) shows the normal condition just prior to cancellation of tool nose compensation. Some geometries will result in overcutting 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 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. Figure 13 shows how I and J can correct undesired cutting in a departure move. Refer to the G40 command description for instructions on calculating values of I and K.

Figure 13: Use of I and K in a G40 block
2. **Tool Geometry Offsets**

The length geometries offsets of tools that use tool nose compensation are set up in the same manner as tools not using compensation. Refer to the “Setup Procedures” section of your operation manual for details on touching off tools and recording tool-length geometries. When a new tool is set up, the geometry WEAR should be cleared to zero.

Often a tool will exhibit uneven wear. This occurs when particularly heavy cuts occur on one edge of the tool. In this case, it may be desirable to adjust the X or Z GEOMETRY WEAR OFFSETS rather than the RADIUS WEAR OFFSETS. By adjusting X or Z length geometry wear, the operator can often compensate for uneven tool nose wear. Length geometry wear will shift ALL dimensions for a single axis. The part design “may not allow” the operator to compensate for wear by using the length geometry shift. One can determine which wear to adjust by checking several X and Z dimensions on a finished part. Wear that is even will result in similar dimensional changes on the X and Z-axes and suggests that the radius wear offset should be increased. Wear that affects the dimensions on one axis only suggests length geometry wear.

3. **Tool Nose Radius 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.
4. **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. The following figure 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.
<table>
<thead>
<tr>
<th>TOOL TIP CODE #</th>
<th>TOOL TIP DIRECTION ORIENTATION</th>
<th>COMMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td></td>
<td>Zero (0) indicates no Specified direction. It is usually not used when tool nose compensation is desired.</td>
</tr>
<tr>
<td>1</td>
<td></td>
<td>Tool tip off in the X+, Z+ direction.</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>Tool tip off in the X+, Z- direction.</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>Tool tip off in the X-, Z- direction.</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>Tool tip off in the X-, Z+ direction.</td>
</tr>
<tr>
<td>TOOL TIP CODE #</td>
<td>TOOL TIP DIRECTION ORIENTATION</td>
<td>COMMENT</td>
</tr>
<tr>
<td>----------------</td>
<td>--------------------------------</td>
<td>---------</td>
</tr>
<tr>
<td>5</td>
<td>Tool Edge Z+ direction</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Tool Edge X+ direction</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>Tool Edge Z- direction</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>Tool Edge X- direction</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>Same as tip 0</td>
<td></td>
</tr>
</tbody>
</table>
NOTE: It is important to remember that tool nose compensation values are in terms of radii rather than diameter. This is important in blocks where tool nose compensation is cancelled or turned on. **If the incremental distance of an approach or departure move for a compensated tool path is not greater than the radius of the cutting tool, alarm (369 TOOL TOO BIG) will occur.** On older machines over cutting may occur. If the movement is in the X direction allow for twice the tool nose radius on approach and departure moves as the machine is programmed in diameters. The following example illustrates how incorrect programming will result in alarms.

**Example**

**Setting 33 is FANUC:**

<table>
<thead>
<tr>
<th>Tool Geometry Offset 8:</th>
<th>X</th>
<th>Z</th>
<th>Radius</th>
<th>Tip</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-6.0000</td>
<td>-8.000</td>
<td>0.0156</td>
<td>2</td>
</tr>
</tbody>
</table>

O0104;
G28;
T808; (Boring Bar);
G97 S2400 M03;
G54 G00 X.49 Z.05;
G41 G01 X.5156 F.004;  (Alarm 369 occurs, move needs to be greater than 2X the radius of the tool .0312 starting X.49 needs to be smaller example X .4836)
Z-.05;
X.3438 Z-.25;
Z-.5;
X.2814;

**Note:** Move is greater than .0312, which is the value required to avoid alarms or cut-in with a departure move when TNC is cancelled with G40.

G40 G00 X.25;
Z.05;
G28;
M30;
Circular Interpolation Exercise with Tool Compensation

Fill in the blanks to complete one finish pass of the cutter path for the above part. Assume the part has already been roughed out. Define the tool path of the part using cutter compensation (G41 or G42) where the tool nose radius is .031 radius for an O.D. turning tool.

O0057
(Circular Interpolation With Cutter Compensation)
N1 (Finish O.D.)
G28
T101 (O.D. Turning Tool .031 TNR)
G50 S2200
G97 S____ M03
G54 G00 X4.2 Z0.1 M08
G96 S750
G00 Z0.
G01 X-0.062 F0.004
G0 X3.5 Z0.10

G01 G____ Z0. F0.005 (TURN ON COMP)
X_______
G03 X_______ Z-_______ R_______
G01 Z_______
G02 X_______ Z-_______ R_______
G01 X_______
G03 X_______ Z-_______ R_______
G01 Z-_______
G00 G40 X____ Z1. M09 (COMP OFF)
G97 S320
G28
M30
Canned Cycles and additional G Codes

One of the most common types of codes in CNC programming are “canned cycles.” This unit will define and discuss canned cycles and state some uses and applications.

Objectives:

Upon completion of this unit, the student will:

1) Be able to define “canned cycle.”

2) Be familiar with common canned cycles, their variables, the proper applications, and the correct use of them.

3) Complete final exercise.

Machining Cycles for the Lathe

A machining cycle 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 machine cycles for turning and grooving. There are cycles for drilling and tapping and can be either single block canned cycles or modal canned cycles.

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

- G70 Finishing Cycles
- 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 Turning with Chip Break Cycle
- G75 O.D./I.D. Grooving or Turning with 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
G71  O.D./I.D. STOCK REMOVAL CYCLE

P  Starting Block number of path to rough
Q  Ending Block number of path to rough
U  X-axis value and direction of G71 rough stock allowance, diameter
W  Z-axis value and direction of G71 rough stock allowance
D  Depth of cut for each pass of stock removal, positive radius
*I  X-axis stock value and direction for a G71 finish pass, radius
*K  Z-axis stock value and direction for a G71 finish pass
*S  Spindle speed to use throughout G71 PQ block
*T  Tool and offset to use throughout G71 PQ block
*F  Feedrate to use throughout G71 PQ block
*R1  YASNAC select Type II roughing
*  Indicate optional

This canned cycle will rough out material on a part given 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 path definition to the G71 call by means of a PQ block designation. Any feeds, spindle speeds or tools within the block defining the path are ignored by the G71 call. Any F, S or T commands on the G71 line or in effect at the time of the G71, are used throughout the G71 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 defined within the PQ block definition.

Two types of machining paths are addressed with a G71 command. The first type of path, a Type I, is when the X-axis of the programmed path does not change direction. This type of path is called a monotonic path. The second type of path, a Type II, allows the X-axis to change direction. For both the first and second type of programmed path, the Z-axis must be monotonous; it cannot change direction.
For FANUC, **Type I** is selected by having only an **X-axis motion** in the block specified by the **P block** in the G71 call. For FANUC, **Type II** is selected by having both an **X-axis** and **Z-axis** motion in the **P block**. When in YASNAC mode, Type II roughing is selected by including R1 on the G71 command block.

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 G70 block using the programmed feeds, speeds, tools and offsets defined within the PQ block definition. **The final motion** in either types is a **return to the starting position S**.

The **start position S** is the position of the tool at the time of the G71 call. The Z clearance plane is derived from the Z-axis start position and the sum of W and optional K finish.

**Program Example using G70 and G71**
Tool Description:

Tool 1  80 Degree CNMG 432 Roughing insert .0312 Radius  
Tool 2  55 Degree TPG 432 Finishing insert .0312 Radius  
Tool 3  .250 Wide Grooving Tool with .0156R

<table>
<thead>
<tr>
<th>Tools</th>
<th>Offsets X</th>
<th>Z</th>
<th>Radius</th>
<th>Tip</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>01</td>
<td>-8.9650</td>
<td>-12.8470</td>
<td>.0312</td>
</tr>
<tr>
<td>T2</td>
<td>02</td>
<td>-8.9010</td>
<td>-12.8450</td>
<td>.0312</td>
</tr>
<tr>
<td>T3</td>
<td>03</td>
<td>-8.8400</td>
<td>-12.8380</td>
<td>.016</td>
</tr>
<tr>
<td>T3</td>
<td>13</td>
<td>-8.8400</td>
<td>-12.5880</td>
<td>.016</td>
</tr>
</tbody>
</table>
Program Example

%  
O0105 (General TNC Example)

N1

G28  (Return to machine zero for tool change)
T0101  (Select Tool 1 Offset 1)
G50  S1000
G97  S500  M03
G54  G00  X2.1  Z0.1  (Rapid to Point S, Defines Stock)
G96  S300
G71  P10  Q20  U0.02  W0.01  D.1  F0.01  (Rough P to Q using G71 and TNC)

N10  G42  G00  X0  Z0.1  F.01  (P Block) (G71 type II, TNC approach)
  G01  Z0  F.005
  X0.65
  X0.75  Z-0.05
  Z-0.75
  G02  X1.25  Z-1.  R0.25
  G01  Z-1.5
  G02  X1.  Z-1.625  R0.125
  G01  Z-2.5
  G02  X1.25  Z-2.625  R0.125
  G01  Z-3.5
  X2.  Z-3.75

N20  G00  G40  X2.1  (TNC departure) (Q block)
  G97  S500
  G28
  M01

N2

G28  (Return to machine zero for tool change)
T202  (Select Tool 2 Offset 2)
G50  S1000
G97  S750  M03
G00  X2.1  Z0.1  (Rapid to point S)
G96  S400
G70  P10  Q20  (Finish P to Q using G70 and TNC)
G97  S750
G28  (Return to machine zero for tool change)
M01
N3
G28   (Return to machine zero for tool change)
T303   (Select Tool 3 Offset 3)
G50  S1000   (Groove to Point B using Offset 3)
G97  S200
G00  G42 X1.5  Z-2.0   (Rapid to Point C)(TNC approach)
G96  S200
G01  X1.  F0.003
G01  Z-2.5
G02  X1.25 Z-2.625 R0.125
G40  G01  X1.5   (TNC Depart)

(Move to right side of groove, to
Point A, using Offset 13)
(Change offset to other side of tool)
(Move to Point C)(TNC approach)

G313
G00  G41  X1.5  Z-2.125
G01  X1.  F0.003
G01  Z-1.625
G03  X1.25 Z-1.5  R0.125   (A)
G40  G01  X1.6   (TNC Depart)
G97  S200
G28   (Zero for tool change clearance)
M30
%

Note: Two offsets must be used when finishing the groove using tool nose compensation. Here, the
same tool is used, but the tool cuts on different sides along the Z-axis. Offset 3 Z length geometry is set
on the left side of the tool whereas offset 13 Z length geometry is set on the right side of the tool.
Tool Nose Compensation in Canned Cycles

This section describes how tool nose compensation works when a canned cycle is used. Refer to the “Canned Cycles” section of this manual for a detailed description of canned cycles. Some canned cycles ignore tool nose compensation; some canned cycles expect a specific coding structure, while other canned cycles perform their own specific canned cycle activity.

The following canned cycles will ignore tool nose radius 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, peck drilling  
G76 Thread cutting cycle, multiple pass  
G92 Thread cutting cycle, modal

The following canned cycles work well when a specific programming sequence is used. This programming sequence is called a TEMPLATE. By using the suggested template, the programmer should have no problem using these canned cycles with tool nose compensation. These canned cycles make use of P and Q to identify a path the canned cycle is to work with.

G70 FINISHING CYCLE

Usually, G70 is used following the use of a G71, G72 or G73, but it can be used alone. Below is the template for using tool nose compensation with G70 alone. Note that TNC approach is part of the PQ path definition sequence, whereas TNC departure is after the execution of G70.

G71 and G72 are similar canned cycles with regard to tool nose compensation. The finishing and rough finishing passes of G71 and G72 recognize tool nose compensation; however, the roughing pass of these two G codes does not. The template below can be applied to either G71 or G72.

G73 is similar to G71 and G72. G73 recognizes TNC on all passes.

    P   Starting Block number of routine to execute  
    Q   Ending Block number of routine to execute

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 a beginning block number (P code) and an ending block number (Q code) be specified for the machine code defining 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 containing an N code matching the Q code in the G70 calling block is found and executed.

After execution of the Q block, a rapid (G00) is executed returning the machine to the start position saved earlier during G70 initialization. The program then returns to the block following the G70 call.

EXAMPLE G70 WITH TNC

A traditional calling sequence using G70 appears below.

Rough cycle feedrate to rough out part between N10 and N50 for the G71 roughing cycle will be F.012, which is defined in the G71 block.

Finish Feedrate for the G70 finishing cycle will be whatever is defined between N10 and N50.

G71 P10 Q70 U.01 W.005 D.08 F.012  (Roughing cycle feedrate will be F.012 which is defined this block)
N10 G00  (machine code that defines part path geometry)
....
....
....
....
N70
G70 P10 Q70  (Finish cycle feedrate is defined between N10 to N50).

This example demonstrates the use of a G70 finishing canned cycle. The basic shape should be roughed out using a G71, G72, or G73 roughing cycle.

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

![Part drawing for TNC with G70 example Type 1](image)

<table>
<thead>
<tr>
<th>Program Example</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>%</td>
<td>(Select Tool 2)</td>
</tr>
<tr>
<td>O0106</td>
<td>(G71 Define part path lines P thru Q)</td>
</tr>
<tr>
<td>G28</td>
<td>(P Block)(Approach move, turn on tool nose compensation)</td>
</tr>
<tr>
<td>T0202</td>
<td></td>
</tr>
<tr>
<td>G50 S1750</td>
<td></td>
</tr>
<tr>
<td>G97 S320 M03</td>
<td></td>
</tr>
<tr>
<td>G54 G00 X3. Z0.1 M08</td>
<td></td>
</tr>
<tr>
<td>G96 S300</td>
<td></td>
</tr>
<tr>
<td>G71 P10 Q20 U.02 W.01 D.1 F.012</td>
<td></td>
</tr>
<tr>
<td>N10 G42 G00 X0.5</td>
<td></td>
</tr>
<tr>
<td>G01 Z0. F.012</td>
<td></td>
</tr>
<tr>
<td>X.6</td>
<td></td>
</tr>
<tr>
<td>X0.8 Z-0.1 F.008</td>
<td></td>
</tr>
<tr>
<td>Z-0.5</td>
<td></td>
</tr>
<tr>
<td>G02 X1.0 Z-0.6 I0.1</td>
<td></td>
</tr>
<tr>
<td>G01 X1.5</td>
<td></td>
</tr>
<tr>
<td>X2.0 Z-0.85</td>
<td></td>
</tr>
<tr>
<td>Z-1.6</td>
<td></td>
</tr>
<tr>
<td>X2.3</td>
<td></td>
</tr>
<tr>
<td>G03 X2.8 Z-1.85 K-0.25</td>
<td></td>
</tr>
<tr>
<td>G01 Z-2.1</td>
<td></td>
</tr>
<tr>
<td>N20 G00 G40 X3.0</td>
<td>(Q)(Departure move turning off tool nose compensation)</td>
</tr>
<tr>
<td>G70 P10 Q20</td>
<td>(G70 Finishing Cycle)</td>
</tr>
<tr>
<td>G28</td>
<td>(Return to machine zero for tool change clearance)</td>
</tr>
<tr>
<td>M30</td>
<td></td>
</tr>
<tr>
<td>%</td>
<td></td>
</tr>
</tbody>
</table>
Note that the suggested template of the previous section for G70 is used. Also note that compensation is enabled in the PQ sequence, but is cancelled after G70 is completed.

O0107 (O.D. Roughing and Finishing)

G28
T101 (CNMG 432)
G50 S2000
G97 S400 M03
G54 G00 X6.6 Z0.1 M08
G96 S630
G71 P10 Q20 U0.01 W0.005 D0.15 F0.012
N10 G00 X0.6634 (P Block)(Begin definition)
G01 X1. Z-0.1683 F0.004
Z-1.
X1.9376
G03 X2.5 Z-1.2812 R0.2812
G01 Z-40312
G02 X2.9376 Z-4.25 R0.2188
G01 X3.9634
X4.5 Z-4.5183
Z-10.75
N20 X6.5 (Q Block)(End definition)
G97 S400 M09
G28
T202
G50 S2500
G97 S400 M03
G54 G00 X6.6 Z0.1 M08
G96 S730
G70 P10 Q20 (G70 Finishing Cycle)
G97 S400 M09
G28
M30
%

(FANUC G71 – TYPE I example)
(Rapid to Home Position)
(Tool change & apply offsets)
(Set Max RPM 2000)
(Spindle On)
(Rapid to Start Position, Coolant On)
(Constant surface speed on)
(G71 Roughing Cycle)
(P Block)(Begin definition)

(Rapid to tool change position)
(Finish tool)
**G71 and G70 Exercise with Tool Nose Compensation**

Fill in the blank line 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 finish pass using 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. Take .120 depth of cut at .012 feed.
O0030
N1
T101 (O.D. Tool X.031 TNR)
G50 S2500
G97 S_____ M03

G54 G_____ X_____ Z_____ M08
G96 S_____ 
G71 P_____ Q_____ U_____ W_____ D_____ F_____ 
N_____ G_____ G_____ X_______ Z________

G_____ Z____ F.006
X________
G_____ X_____ Z_____ R_____
G_____ Z_____ 
G_____ X_____ Z_____ R_____ 
G_____ X_____ 
G_____ X_____ Z_____ R_____
G_____ Z_______ F.003
X_____ F.015
N_____ G_____ X_____

G_____ S____ M____
G____
M_____ 

N2
T_____
G_____ S_____
G_____ S____ M____

G_____ X_______ Z_______ M08
G_____ S_____

G_____ P_______ Q_____
G_____ S____ M____
G____
M_____

(Program number)
(First Operation)
(Tool #1 and Offset #1)
(Spindle speed clamp at 2500 RPM)
(Cancel CCS, define a 400 RPM, spindle
ON forward)
(Rapid X and Z to starting location, coolant ON)
(Turn on CSS to 375)
(Roughing O.D. G71 cycle command)
(Pnn starting number, Rapid X and Z axis, cutter comp ON)
(G71 Part Geometry)
(G71 Part Geometry)
(G71 Part Geometry)
(G71 Part Geometry)
(G71 Part Geometry)
(G71 Part Geometry)
(G71 Part Geometry)
(G71 Part Geometry)
(G71 Part Geometry)
(Qnnn ending number, Cancel Cutter Comp.)
(Cancel CSS, define 400 RPM, Coolant Off)
(Return to reference point)
(Optional stop command)
(Second operation)
(Tool #2 and Offset #2)
(Spindle speed clamp at 2500 RPM)
(Cancel CSS, turn on spindle 1200 RPM)
(Rapid to start position above part)
(Turn on CSS to 650)
(Define finish pass using part geometry)
(Cancel CSS, define 400 RPM, Coolant Off)
(Return to reference point)
(End of program rewind)
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.

<table>
<thead>
<tr>
<th>TOOL</th>
<th>OFFSET</th>
<th>RADIUS</th>
<th>TIP</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>04</td>
<td>.032</td>
<td>2</td>
</tr>
</tbody>
</table>

O00088 (ID G71 EXTRA)
N1 G53 G00 X0 Z0 T0
N2 T404
N3 G50 S3000
N4 G97 S_____ M03
N5 G54 G00 X_____ Z0.1 M08 (ST PT BELOW STK OF ID)
N6 G96 S420
N7 G71 P8 Q18 U-_____ W_____ D0.08 F0.010 (G71 is MINUS for G71ID)
N8 G41 G00 X_____ (N8 start position of part path geometry)
N9 G01 Z0 F0.008
N10 X_____ F0.005
N11 G02 X2.63 Z-____ R0.05
N12 G01 Z-____
N13 G0_ X_____ Z-0.825 R0.1
N14 G0_ X2.25
N15 G0_ X1.25 Z-1._____ R0.5
N16 G01 Z-3.25
N17 G03 X0._____ Z-_____ R0.25
N18 G01 G40 X____
N19 G70 P8 Q18
N20 G97 S1780 M09
N21 G53 G00 X0 Z0 T0
N22 M30

G71 and G70 may be used on ID work. With G71 and G70 ID work the U value must be negative reflecting the stock left on will be smaller than with OD work. The part needs a hole drilled before using G71. G71 also works good for boring jaws. The above exercise has a 1” hole drilled 3.5” deep.
**Type 1 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.

**Type II 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 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, placing a reference to both the X and Z-axis in the block specified by P specifies Type II.

Roughing is similar to Type I except that 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 included in a PQ path definition. A trough can be defined as a change in direction creating 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.
Example G71 Type II Roughing

%  O0135      (Fanuc G71 Type II roughing)
G28
T101      (Roughing Tool)
G97 S1200  M03  
G00  G54 X2.  Z0.05  (Start position)
G71 P1  Q6  D.035  U.03  W.01  F.01
N1  G1  X1.5  Z-5 F.004    (PQ path definition)
N2  X1.0  Z-1.
N3  X1.5  Z-1.5
N4  Z-2.
N5  G02  X.5  Z-2.5  R.5
N6  G1 X2.
G28
T202      (Finishing tool)
G97 S1500  M03
G70  P1  Q6 (Finish pass)
G28
M30
%

Productivity Inc – Haas Lathe Programming Manual
Page 97
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 a .120 depth of cut at .012 feed.
O0040
N1
T101 (O.D. Tool X .031 TNR)
G50 S2500
G97 S400 M03
G54 G____ X_____ Z_____ M08
G____ S______
Z______
G____ X_____ F.005
G____ X_____ Z_____
G71 P____ Q____ U____ W____ D____ F____
N____ G____ X_____
G____ Z____ F.006
X_____
G____ X____ Z____ R____
G____ Z____
X____ Z____ F.012
Z____
X____ Z____
Z____
G____ X____ Z____ R____
G____ X____
G____ X____ Z____ R____
G____ Z____ F.003
X____ F.015
N____ G____ X____
G____ S____ M____
G____ M____
N2
T____
G____ S____
G____ S____ M____
G____ X____ Z____ M08
G____ S____
G____ P____ Q____
G____ S____ M____
G____
M____

(Program number)
(First operation)
(Tool #1 and Offset)
(Spindle speed clamp at 2500 RPM)
(Cancel CSS, define a 400 RPM, spindle ON forward)
(Rapid X & Z to starting location, coolant ON)
(Turn on CSS to 375)
(Position .005 from end of part)
(Feed down to rough face end of part)
(Rapid to start position above part)
(Roughing O.D. G71 cycle command)
(Pnn starting number, Rapid X axis, Cutter comp ON)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 Type II part geometry)
(G71 Type II part geometry)
(G71 Type II part geometry)
(G71 Type II part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(Qnnnn ending number, Cancel Cutter Comp)
(Cancel CSS, define 400 RPM, Coolant Off)
(Return to reference point)
(Optional stop command)

(Second operation)
(Tool #2 and Offset #2)
(Spindle speed clamp at 2500 RPM)
(Cancel CSS, turn on spindle 1200)
(Rapid to start position above part)
(Turn on CSS to 650)
(Define finish pass using part geometry)
(Cancel CSS, define 400 RPM, Coolant off)
(Return to reference point)
(End of program rewind)
PROGRAM THE SECOND SIDE OF BALL JOINT FOR ABOVE PART OUT 2 1/4" ROUND STOCK. THE FIRST SIDE 1.040 Ø AND 1.314 Ø HAVE BEEN TURNED. ROUGH TURN .05 DEPTH OF CUT, LEAVE .005 ON FACE.01 ON DIAMETER FOR G70.

O00175 (BALL JOINT)
N1 (35 DEG TURN)
T101 (OD TOOL .0312R)
N2 G___ S_____ (CLAMP SPINDLE SPEED AT 2500)
N3 G97 S_____ M03
N4 G54 G00 X____ Z0.1 M08 (START POSITION FOR G71)
N5 G96 S375
N6 G71 P10 Q20 U____ W_____ D______ F0.012
N10 G00 G___ X-0.07 Z0.1 (TURN ON TOOL COMP)
N11 G01 Z__ F0.006
N12 X0
N13 G03 X_____ Z-_____ R____
N14 G0__ X_____ Z-_____ R0.125
N15 G01 Z-_____ 
N16 G01 X______
N18 G01 X_____ Z-_____
N20 G00 G40 X2.50
N21 G97 S400 M09
N22 G28
M01
N2
T202 (55 DEG FIN)
N23 G50 S2500
N24 G97 S_____ M03
N25 G00 X_____ Z_____ M08
N26 G96 S650
N27 G70 P___ Q____
N28 G97 S400 M09
N29 G28
M30
**G72 End Face Stock Removal Cycle**

- **P** Starting block number of path to rough
- **Q** Ending block number of path to rough
- **U** X-axis size and direction of G72 rough allowance, diameter
- **W** Z-axis size and direction of G72 rough allowance
- **D** Depth of cut for each pass of stock removal, positive number
- **I** X-axis size and direction of G72 finish pass allowance, radius
- **K** Z-axis size and direction of G72 finish pass allowance
- **S** Spindle speed to use throughout G72 PQ block
- **T** Tool and offset to use throughout G72 PQ block
- **F** Feed rate to use throughout G72 PQ block

* Indicates optional

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 define the shape of a part by programming the finished tool path and 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 are 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 is depicted in Figure 3-12. 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 and second type of programmed path, the X-axis must be monotonic; 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, 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.
Type I 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 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.
Example: TNC with G72

This example uses tool nose compensation with a G72 roughing canned cycle.

<table>
<thead>
<tr>
<th>Program Example</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>% O0814 G50 S1000 T0101 G0 X3.5 Z.1 G96 S100 M03</td>
<td>(Select Tool 1) (Move to point S)</td>
</tr>
<tr>
<td>G72 P80 Q180 U.005 W.01 D.05 F.010</td>
<td>(Rough P to Q with T1 using G72 and TNC)</td>
</tr>
<tr>
<td>N80 G41 G0 Z-1.6</td>
<td>(P)(G72 Type I, TNC Approach)</td>
</tr>
<tr>
<td>G1 X2.0 F0.005 X1.4 Z-0.9 X1.0 Z-.60</td>
<td></td>
</tr>
<tr>
<td>G3 X0.8 Z-0.5 K0.1</td>
<td></td>
</tr>
</tbody>
</table>

Machining a chamfer, when cutter compensation is NOT used on the control, requires that calculations must be made for the tool tip dimensions.
G1 Z-0.1
X0.6 Z0.0
N180 X0.0
G40 G0 X3.1 Z0.1 M5     (TNC departure)
(*****OPTIONAL FINISHING SEQUENCE*****)
G28     (Zero for tool change clearance)
M01
N2 G50 S1000
T0202     (Select tool 2)
G0 X3.5 Z.1     (Move to point S)
G96 S100 M03
(Finish P to Q with T2 using G70 and TNC)
G70 P80 Q180
G0 G40 Z.5 M5     (TNC departure)
G28     (Zero for tool change clearance)
M30
%

G72 is used instead of G71 because the roughing strokes in X are longer than the Z roughing strokes of a G71. It is therefore more efficient to use G72. Caution should be used however because of the direction of the force used in machining with face cutting. This is especially true with large parts held on with small jaw engagement and low jaw pressures because of part deformation. These parts may be thrown out of the chuck using face cutting canned cycle.
TYPE I – G72 AND G70 Exercise with Tool Nose Compensation

Fill in the blank lines for Tool #1 using a G72 roughing cycle command to define multiple rough facing passes for the part geometry defined between N10 and N20. Define tool #2 to do a finish pass using a G70 finishing cycle command. On the G72 command line, leave .010 stock on diameters and .005 on the faces. Take .12 depth of cut at .012 feed.
O0050

G28

T101 (O.D. Turning Tool .031R)

G50 S2500

G97 S4000 M03

G54 G00 X_____ Z_____ M08

G____ S____

Z_____

G72 P___ Q___ U____ W____ D___ F___

N___ G____ G____ Z_______

G____ X______ F.006

X______ Z______

X______

X______ Z______

X______

N____ G____ Z_____

G____ S____

G____

M____

T_____ (Tool #2 and Offset #2)

G____ S____

G____ S____ M____

G____ G____ X_____ Z____ M____

G____ S____

Z_____

G____ P____ Q____

G____ S____

G____ Z______ M____

G____

M____
**G73 Irregular Tool Path Stock Removal Cycle**

- **P**: Starting block number of path to rough
- **Q**: Ending block number of path to rough
- **U**: X-axis size and direction of G73 finish allowance, diameter
- **W**: Z-axis size and direction of G73 finish allowance
- **D**: Number of cutting passes, positive number, integer
- **I**: X-axis distance and direction from finish cut to first, radius
- **K**: Z-axis distance and direction from finish cut to first
- **S**: Spindle speed to use throughout G73 PQ block
- **T**: Tool and offset to use throughout G73 PQ block
- **F**: Feed rate to use throughout G73 PQ block

* Indicates optional

The G73 canned cycle can be used for rough cutting of castings or forgings. The canned cycle assumes that the extra material will follow the profile that is programmed between the tool path P through Q.

Machining either starts at the start position S and 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 roughing pass is completed, the tool returns to the 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 P through Q does not have to be monotonic in X or Z, but care has to be taken to ensure 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.
EXAMPLE TNC WITH G73

This example uses tool nose compensation with a G73 roughing canned cycle.

Preparation: Setting 33 FANUC

Tools:
- T1 Insert with .064 radius, roughing
- T2 Insert with .016 radius, finishing

Offsets

<table>
<thead>
<tr>
<th></th>
<th>Tool</th>
<th>Radius</th>
<th>Tip</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>T1</td>
<td>.064</td>
<td>3</td>
</tr>
<tr>
<td>2</td>
<td>T2</td>
<td>.016</td>
<td>3</td>
</tr>
</tbody>
</table>

Program Example

% 000815
G50 S1000  (Example section 8.10 example 5)
T101  (Select Tool 1)
G00 X3.5 Z.1  (Move to point S)
G96 S250 M03  (Rough P to Q with T1 using G73 and TNC)
G73 P80 Q180 U.01 W.005 I0.3 K0.15 D4 F.012
(Define part path PQ sequence)
N80 G42 G00 X0.6  (P)(G72 Type I, TNC Approach)
G01 Z0. F0.1
X0.8 Z-0.1 F.005
Z-0.5
G02 X1.0 Z-0.6 I0.1
G01 X1.4
X2.0 Z-0.9
Z-1.6
X2.3
G03 X2.8 Z-1.85 R.25
N180 G40 G01 Z-2.1  (Q)
G00 X3.0 Z0.1 M05  (TNC departure)

(*****OPTIONAL FINISHING SEQUENCE*****)
G28  (Zero for tool change clearance)
M01
N2 G50 S1000
T202  (Select tool 2)
G00 X3.0 Z.1  (Move to point S)
G96 S250 M03
(Finish P to Q with T2 using G70 and TNC)
G70 P80 Q180
G00 G40 Z.5 M05  (TNC departure)
G28  (Zero for tool change clearance)
M30
%

Productivity Inc – Haas Lathe Programming Manual
Fill in the blank line for tool #1 using a G73 roughing cycle command defining roughing passes for the part geometry N10 and N20. Then define a finish pass using a G70 finishing cycle command, with tool #2, to define a finish pass for the part geometry defined between N10 and N20. On the G73 command line leave .010 stock on diameters and .005 on the faces. Take three passes at .012 feed.
O0060
N1
T101 (O.D. Tool X .031 TNR)
G50 S2500
G97 S400 M03

G54 G___ X_____ Z____ M08
G____ S______
Z______
G____ X____ F.005
G____ X____ Z____
G73 P_____ Q____ W____ D____ F____ I____ K______
N___ G____ G____ X_____

G____ Z____ F.006
X_____
G____ X____ Z_____ R____
G____ Z_____
G____ X____ Z____ R____
G____ X_____
G____ X____ Z_____ R____
G____ Z____ F.003
X____ F.015
N____ G____ X_____

G____ S____ M____
G____
M____

N2
T_____
G____ S_____
G____ S____ M____
G54 G____ X____ Z____ M____
G____ S_____
G____ P____ Q____
G____ S____ M____
G____
M____

(Program number)
(First operation)
(Tool #1 and Offset #1)
(Spindle speed clamp at 2500 RPM)
(Cancel CSS, define a 400 RPM spindle on forward)
(Rapid X & Z to starting location, coolant ON)
(Turn on CSS to 375)
(Position .005 from end of part)
(Feed down to rough face End of part)
(Rapid to start position above Part)
(Roughing O.D. G73 cycle command)
(Pnn starting number, Rapid X-axis, cutter comp ON)
(G73 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(G71 part geometry)
(Qnnnn ending number, cancel Cutter comp)
(Cancel CSS, define 400 RPM, spindle ON forward, Coolant Off)
(Return to reference point)
(Optional stop command)
(Second operation)
(Tool #2 and offset #2)
(Spindle speed clamp at 2500 rpm)
(Cancel CSS, turn on spindle 1200 RPM)
(Rapid, X, Z location, Turn coolant on)
(Turn on CSS to 650)
(Define finish pass using part geometry)
(Cancel CSS, define 400 RPM, spindle ON forward, Coolant Off)
(Return to reference point)
(End of program rewind)
G74  *End Face Grooving Cycle, Peck Drilling*

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>Z-axis absolute location total pecking depth, signed</td>
</tr>
<tr>
<td>W</td>
<td>Z-axis incremental distance to total pecking depth, signed</td>
</tr>
<tr>
<td>*X</td>
<td>X-axis absolute location of furthest peck cycle, signed diameter</td>
</tr>
<tr>
<td>*U</td>
<td>X-axis incremental distance to furthest peck, signed diameter</td>
</tr>
<tr>
<td>*I</td>
<td>X-axis size of increment between peck cycles, positive radius</td>
</tr>
<tr>
<td>*K</td>
<td>Z-axis size of increment between pecks in a cycle</td>
</tr>
<tr>
<td>*D</td>
<td>Tool clearance when returning to starting plane, positive</td>
</tr>
<tr>
<td>*F</td>
<td>Feedrate</td>
</tr>
</tbody>
</table>

* Indicates Optional

The G74 canned cycle can be used for grooving on the face of a part for peck drilling or for turning with a chip break. With this canned cycle, either a single pecking cycle can be executed, as for drilling on the spindle centerline, or a series of pecking cycles can be performed.

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 specified by K. The peck is a rapid move opposite the direction of feed and the peck distance is obtained from Setting 22. The D code can be used for grooving and turning to provide material clearance when returning to starting plane S.
Program Example

% 
O0074
T101 M03 S750
G00 X3. Z0.05
G74 I0.2 K0.1 X1.75 Z-0.5 F0.01
G28
M30
%

(Rapid to start position)
(Face grooving cycle multiple pass)
**G75 O.D./I.D. Grooving Cycle, Peck Drilling**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>Z-axis absolute location to furthest peck cycle, signed</td>
</tr>
<tr>
<td>W</td>
<td>Z-axis incremental distance to furthest peck cycle, signed</td>
</tr>
<tr>
<td>*X</td>
<td>X-axis absolute location total pecking depth, signed diameter</td>
</tr>
<tr>
<td>*U</td>
<td>X-axis incremental distance to total pecking depth</td>
</tr>
<tr>
<td>*D</td>
<td>Tool clearance when returning to starting plane, positive</td>
</tr>
<tr>
<td>*I</td>
<td>X-axis size of increment when pecking (radial)</td>
</tr>
<tr>
<td>*K</td>
<td>Z incremental distance between pecking cycles</td>
</tr>
<tr>
<td>*F</td>
<td>Feed Rate</td>
</tr>
<tr>
<td>*</td>
<td>Indicates optional</td>
</tr>
</tbody>
</table>

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 (as for multiple grooves).

When adding a Z or W code 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**.

The D code can be used for grooving to provide material clearance when returning to starting plane S.
G75 Single Pass

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 I.05 F0.005  (G75 Single pass O.D. grooving cycle)  
N7 M09  
N8 G28  
N9 M30
(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.
Groove Exercise

Program above groove with Top Notch insert NG3125R. This insert is .125 wide with a radius of .0075". Program with a G75 cycle to rough out the material using a peck depth of .020. Leave .002 on the bottom and both sides. Finish cut the .03 chamfers and wall on each side. Program along the bottom so both ends to finish and meet in the middle. Run at 300 ft/min feed rate of .003”/rev.

O00109 (G75 W BRK)
G28
T505
G97 S_____ M03
G54 G00 X_____ Z0.1 M08
Z-0.____
G75 X_____ I0.____ F0.00__ Z-______ K0.____
G00 X____
Z-1.______
G01 X____ Z-1._____ F0.00__
X1._____ Z-1.____
X_____ Z-____
G00 X____
Z-0._____ G01 X____ Z-0.____
X1._____ Z-0.____
X____ Z-_____ G00 X____ M09
G28
M30
G76 Thread Cutting Cycle, Multiple Pass

G00 X1.1 Z0.3  
G76 X0.913 Z-.85 K.0451 D.0136 A58 F0.0714 (1-14 THREAD Minor .9132)

- D First pass cutting depth  
- K Thread height, defines limit of multiple passes, radius measure  
- *X X-axis absolute location, maximum thread I.D.  
- *Z Z-axis absolute location, maximum thread length  
- *U X-axis incremental distance, start to maximum thread I.D.  
- *W Z-axis incremental distance, start to maximum thread length  
- *I Thread taper amount, radius measure  
- *P Subsequent pass positioning algorithm, cutting method  
- *F(E) Feedrate (specifies feedrate for threading – also indicates the pitch or lead)  
- *A Tool nose angle (0 to 120 degree)  Defaults to 0. Do not use decimal point.  
- * Indicates Optional

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 the thread. The nature of tool load and wear can be controlled by using the P code. The P code can specify which side the tool cuts on and it can specify how much material will be cut.

The **height of the thread** is specified in **K**. The height of the thread is defined as the distance from the crest to the root of the thread. K must agree with the direction that the X-axis is being cut. The actual depth of cut will be K less the finish allowance. **Setting 86 (THREAD FINISH ALLOWANCE)** is this amount and is defaulted to 0.
The **thread taper** amount is specified in I. It is measured from the target position X, Z at **point T** to the beginning **position C**. A conventional O.D. taper thread will have a **negative I value**.

The **depth of the first cut** through the thread is specified in D, which 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 through the thread can be controlled with **Setting 99 (THREAD MINIMUM CUT)**. For any of the methods specified in P, the last cut will never be less than this value. The default value is .001 inches/.01 mm. On most carbon and alloy steels the value of minimum pass of .005 will produce acceptable finishes. For some materials .001 minimum pass will improve the surface finish of the thread.

At the end of the thread, an optional chamfer is performed. The size and angle of the chamfer is controlled with **Setting 95 (THREAD CHAMFER SIZE)** and **Setting 96 (THREAD CHAMFER ANGLE)**. The chamfer size is designated in number of threads, so that if 1.000 is recorded in Setting 95 and the feedrate is .05, then the chamfer will be .05. A chamfer can improve the appearance and functionality of threads that must be machined up to a shoulder. If relief is provided for at the end of the thread, then the chamfer can be eliminated by specifying 0.000 for the chamfer size in Setting 95. The default value for Setting 95 is 1.000 and the default angle for the thread (Setting 96) is 45 degrees. **M23 commands chamfer on. M24 commands chamfer off. M23 is the default value.**

The **tool nose angle** for the thread is A. The value can run from 0 to 120. If no A is given 0 degrees is assumed. If no A value is given then the thread is created with a radial cut. With a radial cut both sides of the threading tool are cutting. This creates a v shaped chip which may be hard to evacuate. This may cause the insert to chip. Also chattering may occur as the entire tool is engaged at the end of the thread cycle.
If \( A \) has a 60 degree value, the same as the thread cutting tool, the Haas machine will make a flank cut. In a flank cut just the leading edge of the tool will do the cutting. Each successive pass will in feed the tool at 30 degrees as shown above. The advantage of using a flank cut is that the chip will flow out of thread form area easier than a radial cut. This also reduces a burr from forming on the trailing edge of the tool. To avoid rubbing on the trailing edge of the tool a modified flank cut is recommended. With a modified flank cut the value of \( A \) is 58 (in feed angle 29 degrees). A channel chip is formed but because of uneven thickness on the chip flows out similar to a flank cut. The disadvantage to flank or modified flank cutting is torn or poor finish may result when soft or gummy materials are cut such as low carbon steels, aluminum and stainless steels.

**P Options:**
- **P1:** Single cutting, cutting amount constant (default)
- **P2:** Double edge cutting, cutting amount constant
- **P3:** Single edge cutting, cutting depth constant
- **P4:** Double edge cutting, cutting depth constant

Alternating flank cutting is possible with P2 and P4. Alternating flank cutting will increase tool life as both sides of the inserts are being used equally. Option **P2** may give superior results according to Haas.

Example of G76 code for a 1-8 Thread:

```
T404
G50 S2000
G97 S1200 M03
G00 X1.2 Z.3 M08
G76 X.8512 Z-1.50 K.0789 D.0197 A58 P2 F.125
G00 G28 U0
G28 W0.
M30
```
Thread Programming Exercise

2 ¾-12UN 3A Thread
Major Diameter 2.7500/2.7386 Minor Diameter 2.6478

O0010
N1 G28
N2 T101 (Thread Tool)
N3 G97 S590 M03
N4 G54 G00 X_______Z_______M08 (Rapid X and Z above part)
N5 Z_______ M____ (Z-.25 is start of thread, M code for no chamfer)
N6 G76 X_______ Z_______ K_____D_____ A____ P__ F____(12 passes, alternating flank, in feed cut 29 degrees)
N7M_____ (Turn Coolant off)
N8 G____ U_______ (Take Home in X)
N9 G____ W_______ (Take Home in Z)
N10 M____ (End of Program)
Drilling, Boring and Tapping Canned Cycles

These canned cycles are used to define and simplify programming for the most common Z-axis repetitive operations. 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.

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

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.

A canned cycle will only be executed in the Z-axis when positioning to a new X-axis position during a canned cycle command.
**Canned Cycles for Drilling and Tapping**

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

### Canned Cycles

<table>
<thead>
<tr>
<th>G Code</th>
<th>Z Drilling Direction</th>
<th>Operation at Bottom of Hole</th>
<th>Retraction Z Direction</th>
<th>Application</th>
</tr>
</thead>
<tbody>
<tr>
<td>G80</td>
<td>Canned Cycle Cancel</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>G81</td>
<td>Drill Canned Cycle</td>
<td>None</td>
<td>Rapid</td>
<td>Spot drilling</td>
</tr>
<tr>
<td>G82</td>
<td>Spot Drill Canned Cycle</td>
<td>Dwell</td>
<td>Rapid</td>
<td>Counter boring</td>
</tr>
<tr>
<td>G83</td>
<td>Peck Drill Canned Cycle</td>
<td>None</td>
<td>Rapid</td>
<td>Peck drilling</td>
</tr>
<tr>
<td>G84</td>
<td>Tapping Canned Cycle</td>
<td>Spindle CCW</td>
<td>Feed</td>
<td>Tapping cycle</td>
</tr>
<tr>
<td>G85</td>
<td>Bore in Bore out Canned Cycle</td>
<td>None</td>
<td>Feed</td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G86</td>
<td>Bore in Rapid out Canned Cycle</td>
<td>Spindle stop</td>
<td>Rapid</td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G87</td>
<td>Bore with Manual Retract Canned Cycle</td>
<td>Spindle stop</td>
<td>Manual/rapid</td>
<td>Back cycle</td>
</tr>
<tr>
<td>G88</td>
<td>Bore in Dwell with Manual Retract Canned Cycle</td>
<td>Dwell, then spindle stop</td>
<td>Manual/rapid</td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G89</td>
<td>Bore in Dwell Bore out Canned Cycle</td>
<td>Dwell</td>
<td>Feed</td>
<td>Boring cycle</td>
</tr>
</tbody>
</table>

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 used on HAAS lathe controls.

### G80 CANNED CYCLE CANCEL

The G code is modal; deactivating all canned cycles until a new one is selected. Note that use of G00 or G01 will also cancel a canned cycle.
G81  DRILL CANNED CYCLE

X  Optional X-axis motion command
Z  Position of bottom of hole
R  Position of the R plane
F  Feed Rate

![Diagram of G81 Drill In/Rapid Out Canned Cycle]

<table>
<thead>
<tr>
<th>TOOL DESCRIPTION</th>
<th>TOOL</th>
<th>OFFSET</th>
<th>RADIUS</th>
<th>TIP</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/2 DIA. DRILL</td>
<td>1</td>
<td>01</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>O00119 (G81 Drilling)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N1 G28</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N2 T101 (1/2 DIA. DRILL)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N3 G97 S1450 M03</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N4 G54 G00 X0. Z1. M08</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N5 G81 Z-0.625 R0.1 F0.005</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N6 G80 G00 Z1.0 M09</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N7 G28</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>N8 M30</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

(N1 N2 N3 N4 N5 N6 N7 N8)
G82 SPOT DRILL COUNTER BORE CANNED CYCLE

X  Optional X-axis motion command
Z  Position of bottom of hole
P  The dwell time at the bottom of the hole
R  Position of the R plane
F  Feed Rate

<table>
<thead>
<tr>
<th>TOOL DESCRIPTION</th>
<th>TOOL</th>
<th>OFFSET</th>
<th>RADIUS</th>
<th>TIP</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/2 DIA. FLAT BOTTOM DRILL</td>
<td>2</td>
<td>02</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

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 Z1.0 M09
N7 G28
N8 M30
G83  DEEP HOLE PECK DRILLING CANNED CYCLE

<table>
<thead>
<tr>
<th>X</th>
<th>Optional X-axis motion command</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>Position of bottom of hole</td>
</tr>
<tr>
<td>I</td>
<td>Optional size of first cutting depth</td>
</tr>
<tr>
<td>J</td>
<td>Optional amount to reduce cutting depth each pass</td>
</tr>
<tr>
<td>K</td>
<td>Optional minimum depth of cut</td>
</tr>
<tr>
<td>Q</td>
<td>The cut-in value, always incremental</td>
</tr>
<tr>
<td>R</td>
<td>Position of the R plane</td>
</tr>
<tr>
<td>F</td>
<td>Feedrate</td>
</tr>
</tbody>
</table>

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

Setting 52 also changes the way G83 works when it returns to the R plane. Most programmers set the R plane well above the cut to ensure that the chip clear motion actually allows the chips to get out of the hole. This causes a wasted motion when first drilling through this “empty” space. 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. When the clear move to R occurs, the Z will be moved above R by this setting.

![G83 Peck Drilling Canned Cycle](image)

The operation of this cycle is different if the rigid tapping option is installed. When rigid tapping is used, the ratio between the feed rate and spindle speed must be precisely the thread pitch being cut.

You do not need to start the spindle CW before this canned cycle. The control does this automatically.
G84  **TAPPING CANNED CYCLE**

- **X**: Optional X-axis motion command
- **Z**: Position of bottom of hole
- **R**: Position of the R plane
- **F**: Feed rate

Feed rate is the **Lead** of the thread. Feed Rate = 1/ Threads per inch

**G184  Reverse Tapping Canned Cycle** (the same as a G84, but is used for left handed taps)
Turn Threads, Drill and Tap Exercise

T0101 Rough Turn G71 S700 ft/min F .011 in/rev
T0202 Finish Turn G70 S700 ft/min F .007 in/rev
T0303 .125” Groove  (Use G75) S200ft/min, F .002/rev
T0404 Thread G76 1-3/4 -16 UN 2A 1.0” back from face at a 300 ft/min
  Major diameter 1.748, Minor diameter 1.6739
T1111 5/16 Drill  G83 1.3” deep S 80 ft/min F .006”/rev peck diameter of drill
T1010 Tap 3/8-16 x 1.0” deep, 200 rev/min

O0096
G28 (80 DEG TURN)
T101 (OD TOOL .0312R)
G___ S_____  Clamp spindle speed 2000 rpm
G___ S___ M_____  Turn on Spindle
G54 G00 X____ Z_____ M08  Rapid to start position
G___ S_____  Turn on constant surface feed at 375 ft/min
G01 Z0 F.011
G01 X____ F0.007  Face off front of part
G00 X3.6 Z0.1
G54 G00 X3.6 Z0.1 M08  Start Position
G71 P___ Q___ U___ W___ D____ F____  G71 Line
N___ G00 G42 X____ Z____  P line
G01 Z____ F0.006  Part geometry
X____
X____ Z-____
G01 Z-____
G02 X____ Z____ R____
G01 X____
G03 X____ Z____ R0____
G01 Z-4.125 F0.003
X_____ F0.0____
X____ Z-______
N__ G40 X____
G97 S400 M09
G28
M01
T202 (60 DEG FIN)
G50 S2500
G97 S1200 M03
G00 X____ Z____
G96 S650
G70 P____ Q____
G97 S400 M09
G28
M01
T303 (.125 GR)
G97 S______ M03
G54 G00 X____ Z____ M08
Z____
G75 X____ Z-____ I____ K____ F0.002
G97 S400 M09
G28
M01
T404 (OD THREAD)
G____ S____ M____
G____ G____ X____ Z____ M____
G____ X____ Z-____ K____ D____ A____ F____
G97 S____ M09
G28
M01
T1111 (5/16 DRILL)
G____ S____ M____
G____ G____ X____ Z____ M____
G____ X____ Z____ R____ Q____ F____
G____
G00 Z0.2
G28
M09
M01
T1010 (3/8-16 TAP)
G97 S____ M____
G54 G____ X____ Z____ M08
G____ X____ Z____ R0.____ F0.0____
G80
G28
M30
G90 O.D./I.D. Turning Cycle

X  X-axis absolute location of target
Z  Z-axis absolute location of target
*I  Optional distance and direction of X-axis taper, radius
*U  X-axis incremental distance to target, diameter
*W  Z-axis incremental distance to target
F (E)  Feedrate

* 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 specifying the X locations of successive passes.

Straight turning cuts can be made by specifying X, Z, and F. By adding I, a taper cut can be made. The amount of taper is referenced from the target; I is added to the value of X at the target.

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.
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 X3.1 Z1. M08 (Rapid to Start Point)
N16 G96 S390
N17 Z0.1
N18 G90 G42 X2.8 Z-1.6001 I-0.9238 F0.01 (Rough 30 Deg. angle to X2.3476)
N19 X2.65 (Additional Pass) Dia. using G90 and TNC
N20 X2.55 (Additional Pass)
N21 X2.45 (Additional Pass)
N22 X2.355 (Additional Pass)
N23 X2.3476 (Additional Pass)
N24 G00 G40 X3.1 Z1. M09 (TNC Departure)
N25 M05
N26 G28
N27 M30
G92 Thread Cutting Cycle

X  X-axis absolute location of target
Z  Z-axis absolute location of target
*I  Optional distance and direction of X-axis taper, radius
*U  X-axis incremental distance to target, diameter
*W  Z-axis incremental distance to target
F (E)  Feedrate, the lead of the thread

* 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 specifying the X locations of successive passes.

Straight threads can be made by specifying X, Z, and F. By adding I, a pipe or taper thread can be cut. The amount of taper is referenced from the target; I is added to the value of X at the target.

At the end of the thread, an automatic chamfer is executed before reaching the target default. 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

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  X-axis absolute location of target
Z  Z-axis absolute location of target
*U  X-axis incremental distance to target, diameter
*W  Z-axis incremental distance to target
*K  Optional distance and direction of Z-axis coning
F (E)  Feedrate

* 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 specifying the Z locations of successive passes.

Straight end facing cuts can be made by specifying X, Z, and F. By adding K, a conical face can be cut. The amount of coning is referenced from the target. K is added to the value of X at the target.

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 7  TNC WITH G94

This example uses tool nose compensation with a G94 modal, rough turning cycle.
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 Code Detailed Description**

**M00  STOP PROGRAM**
The M00 code is used to stop a program. It also stops the spindle, 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**
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 the 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**
The M03 code will start the spindle moving 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 block execution rather than the end as most M codes are.

**M04  SPINDLE REVERSE**
The M04 code will start the spindle moving 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 a parameter 209 (CNCR SPINDLE) is set to 1, then this command is executed at the beginning of block execution rather than the end as most M codes are.

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

**M08  COOLANT ON**
The M08 code will turn on the coolant. Note that the M code is performed at the end of a block, so that if a motion is commanded in the same block, the coolant is turned on after the motion.

**M09  COOLANT OFF**
The M09 code will turn off the coolant.
M10  CLAMP CHUCK
The M10 code is used to clamp the chuck. It is only used when M11 is used to unclamp the chuck.

M11  UNCLAMP CHUCK
The M11 code will unclamp the chuck.

M17  TURRET ROTATION ALWAYS FORWARD
The M17 code is modal and forces the turret to rotate in the forward direction when a tool change is made. Setting 97 TOOL CHANGE DIRECTION needs to be switched from SHORTEST to M17/M18.

T1010  M17:
This command will advance the tool turret in a FORWARD direction to position #10. Because M17 is modal, any subsequent T command will cause the turret to rotate in the forward direction to the commanded tool.

M18  TURRET ROTATION ALWAYS REVERSE
The M18 code is modal and forces the turret to rotate in the forward direction when a tool change is made. Setting 97 TOOL CHANGE DIRECTION needs to be switched from SHORTEST to M17/M18.

T1010  M18:
This command will advance the tool turret in a REVERSE direction to position #10. Because M18 is modal, any subsequent T command will cause the turret to rotate in the reverse direction to the commanded tool.

M21-M22  OPTIONAL USER M
The M21 through M22 codes are optional for user interfaces.

M25-M28  OPTIONAL USER M
The M25 through M28 codes are optional for user interfaces.

M30  PROG END AND REWIND
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 is also incremented. M30 will also cancel tool length offsets.

M31  CHIP CONVEYOR FORWARD
M31 starts the chip conveyor motor in the forward direction.

M32  CHIP CONVEYOR REVERSE
M32 starts the chip conveyor motor in the reverse direction.

M33  CHIP CONVEYOR STOP
M33 stops chip conveyor motion.
M51- M58  OPTIONAL USER M ON
The M51 through M58 codes are optional for user interfaces.

M61- M68  OPTIONAL USER M OFF
The M61 through M68 codes are optional for user interfaces.

M76  DISABLE DISPLAYS

M77  ENABLE DISPLAYS

M78  ALARM IF SKIP SIGNAL FOUND

M79  ALARM IF SKIP SIGNAL NOT FOUND

M85  OPEN AUTOMATIC DOOR

M86  CLOSE AUTOMATIC DOOR

M88  HIGH PRESSURE COOLANT ON (OPTION)

M89  HIGH PRESSURE COOLANT OFF (OPTION)

M97  LOCAL SUB-PROGRAM CALL
This code is used to call a subroutine referenced by a line N number within the same program. A Pnnnn code is required and must match a line number within the same program. This is useful for simple subroutines within a program and does not require the complication of a separate program. The subroutine must end with an M99. An L count on the M97 block will repeat the subroutine call that number of times.

M98  SUB PROGRAM CALL
This code is used to call a subroutine. The Pnnnn code is the number of the program being called. The Pnnnn code must be in the same block. The program by the same number must already be loaded into memory and it must contain an M99 to return to the main program. An L count can be put on the line containing the M98 and will cause the subroutine to be called L times before continuing to the next block.

M99  SUB PROGRAM RETURN OR LOOP
This code is used to return to the main program from a subroutine or macro. It will also cause the main program to loop back to the beginning without stopping if it is used in other than a subprogram without a P code. If an M99 Pnnnn is used, it will cause a jump to the line containing Nnnnn of the same number.
(Solution to BALL JOINT PROGRAM on page 99)

O00175 (BALL JOINT)
N1 (35 DEG TURN)
T101 (OD TOOL .0312R)
N2 G50 S2500
N3 G97 S636 M03
N4 G54 G00 X2.25 Z0.1 M08
N5 G96 S375
N6 G71 / P10 Q20 U0.01 W0.005 D0.05 F0.012
N10 G00 G42 X-0.070 Z0.1
N11 G01 Z0 F0.006
N12 X0
N13 G03 X0.871 Z-1.9 R1.
N14 G02 X0.73 Z-2.013 R0.125
N15 G01 Z-2.388
N16 G01 X1.285
N18 G01 X1.385 Z-2.438
N20 G00 G40 X2.25
N21 G97 S400 M09
N22 G28
M01
N2
T202 (60 DEG FIN)
N23 G50 S2500
N24 G97 S1200 M03
N25 G00 X2.25 Z0.1
N26 G96 S650
N27 G70 P10 Q20
N28 G97 S400 M09
N29 G28
M30