# NC-Programming

1. **Meaning of addresses**
   - Page 5

2. **Tool Number Designation**
   - Page 2

3. **Introduction to NC-programming**
   - Page 1
   - 2.1 Program Registration
   - Page 2
   - 2.2 How to key-in a program on a FANUC control
   - Page 3
   - 2.3 Notes on program registration
   - Page 3
   - 2.4 Notes on writing a program on a PC
   - Page 3

4. **List of Commands used in NC-Lathe programming**
   - Page 4
   - 3.1 Addresses
   - Page 4
   - 3.2 Words
   - Page 5
   - 3.3 Blocks
   - Page 6
   - 3.4 End of Block Character
   - Page 6
   - 3.5 Range of input data
   - Page 6
   - 3.6 The use of decimal fractions in programming
   - Page 7
   - 3.7 Suppression of Leading or trailing Zero
   - Page 8
   - 3.8 The use of Sequence Numbers in the program
   - Page 8
   - 3.9 Optional Block Skip Character, ‘/’ ‘Slash
   - Page 9
   - 3.10 Feed Command
   - Page 9
   - 3.11 Spindle Command
   - Page 11
   - 3.12 Tool Command
   - Page 12
   - 3.13 Tool-offsets
   - Page 12
   - 3.14 Tool-Offset Command
   - Page 13

5. **Tool Nose Radius Compensation**
   - Page 14
   - 4.1 G40, G41 and G42 Tool Nose Radius Compensation Function Commands
   - Page 14
   - 4.2 Tool Nose Radius Compensation Data
   - Page 16

6. **G-Codes**
   - Page 16
   - 5.1 G-Code List (G-code system A, partial listing)
   - Page 18

7. **Miscellaneous Functions, “M”-codes**
   - Page 19
   - 6.1 M-Code List
   - Page 19

8. **Coordinate Systems**
   - Page 21
   - 7.1 Basic Coordinate System
   - Page 21
   - 7.2 NC Lathe Coordinate System
   - Page 21
   - 7.3 Machine Coordinate System
   - Page 23
## Meaning of addresses

<table>
<thead>
<tr>
<th>Function</th>
<th>Address</th>
<th>Meaning of address</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program number</td>
<td>O(EIA / ISO)</td>
<td>Program number</td>
</tr>
<tr>
<td>Block sequence number</td>
<td>N</td>
<td>Sequence number</td>
</tr>
<tr>
<td>Preparatory function</td>
<td>G</td>
<td>Specifies a motion or function</td>
</tr>
<tr>
<td>Dimension word</td>
<td>X, Z</td>
<td>Command of moving position (absolute type) of each position</td>
</tr>
<tr>
<td></td>
<td>U, W</td>
<td>Instruct moving distance and direction (incremental type)</td>
</tr>
<tr>
<td></td>
<td>I, J, K</td>
<td>Designation of circular center of Axis (I=X, J=Y, K=Z)</td>
</tr>
<tr>
<td></td>
<td>R</td>
<td>Radius of circle, corner R, edge R</td>
</tr>
<tr>
<td>Feed function</td>
<td>F</td>
<td>Designation of federate and thread lead</td>
</tr>
<tr>
<td>Auxiliary function</td>
<td>M</td>
<td>Miscellaneous function (On / Off)</td>
</tr>
<tr>
<td>Spindle speed function</td>
<td>S</td>
<td>Designation of spindle speed</td>
</tr>
<tr>
<td>Tool</td>
<td>T</td>
<td>Designation of tool number and offset number</td>
</tr>
<tr>
<td>Dwell</td>
<td>P, U, X</td>
<td>Designation of dwell time</td>
</tr>
<tr>
<td>Number of repetitions</td>
<td>L</td>
<td>Repeat of auxiliary program</td>
</tr>
<tr>
<td>Parameters</td>
<td>A, D, I, K</td>
<td>Parameter at fixed cycle</td>
</tr>
<tr>
<td>Designation of program No</td>
<td>P</td>
<td>Used for calling an auxiliary program</td>
</tr>
<tr>
<td>Designation of sequence No</td>
<td>P, Q</td>
<td>Calling of a repeat cycle and end number</td>
</tr>
</tbody>
</table>

One block is composed as follows:

```
N  G  X  Y  F  S  T  M  :  
Sequence Preparation Dimension Feed Spindle Tool Function EOB  
Auxiliary Function Word Function Speed Function Auxiliary  
No.                                                
```
**Tool Number Designation**

T Function is used for the designation of tool numbers and tool compensation. 

T Function is a tool selection code usually made of 4 digits.

\[ T \boxed{0} \boxed{2} \boxed{0} \boxed{2} \]

- Designation of tool offset compensation number
- Designation of tool number

Example: If the T number is designated as (T0202)

\[ 0 \boxed{2} \text{ Calls the tool number (turret position) and } 0 \boxed{2} \text{ is the tool offset number to use.} \]
# Introduction to NC-programming

To write a program for the NC means to translate all of the action that is required for machining a work piece into a language format that the control can understand. NC programming is done in an internationally standardized language that consists of coded text.

In NC programming code “Words” are used. A NC-word always consists of a letter and a number. A NC-word represents a command. All of the words that are needed for doing a machining process are compiled in the form of a text file called a NC-program.

A sample program is shown below.

<table>
<thead>
<tr>
<th>Program Text</th>
<th>Translation</th>
</tr>
</thead>
<tbody>
<tr>
<td>O4513(SAMPLE PROGRAM)</td>
<td>O = Program registration, 4513=program number</td>
</tr>
<tr>
<td>T0101</td>
<td>Get tool #1 and offset #1</td>
</tr>
<tr>
<td>S1200 M3 P11</td>
<td>Spindle speed 1200 RPM, turn ON the main spindle</td>
</tr>
<tr>
<td>M8</td>
<td>Turn ON the coolant</td>
</tr>
<tr>
<td>G0 X2.1 Z0</td>
<td>Tool approach, rapid move</td>
</tr>
<tr>
<td>G1 G99 X-.063 F.005</td>
<td>Cut the face of the part</td>
</tr>
<tr>
<td>Z0.05</td>
<td>Relieve the tool</td>
</tr>
<tr>
<td>G0 X4. Z4. T0100</td>
<td>Retract the tool and cancel the tool offset</td>
</tr>
<tr>
<td>M5 P11</td>
<td>Main Spindle stop</td>
</tr>
<tr>
<td>M9</td>
<td>Coolant OFF</td>
</tr>
<tr>
<td>M30</td>
<td>M30=End of program and rewind</td>
</tr>
</tbody>
</table>

The sketch below shows the tool path for the program shown above.
The program-text is input to the NC-memory by using the keypad as provided on the system. It is also possible to upload a program text file that has been created on a personal computer by connecting a PC to the RS-232 communication interface device at the NC unit.

The FANUC 18-I, 21-I and 30i Series controls are equipped with a PCMCIA-card slot, allowing uploading / downloading of NC-programs using a flash card or a compact flash card (commonly used in digital cameras) with a PC Card Adapter.

A NC-program begins with the program file number followed by several lines of coded text that instruct the machine step-by-step what to do. In “Auto Operation-Mode”, the control reads and executes the commands one line at a time, line by line. The program ends with a special code that resets or rewinds the program. After the reset and rewind command, processing stops. The automatic process is now ready to be repeated over, again.

### 2.1 Program Registration

The first line of a program must specify the letter “O”, followed by the program number. The range of this number is normally between zero and 9999. When a program number is input into the control memory it establishes a new file name which is the same as the program number.

For example: \( O4512 \) \( 4512 \) is the program file name

For purpose of detailed directory display and program identification it is helpful to include text set between parenthesis on the same line with the program number. Text in parenthesis is limited to 25 characters, maximum.

For example: \( O4513 \text{ (PROGRAM BY JIM SMITH)} \)

When a program is registered as shown above the directory display will list the program as illustrated, below. When no text is inserted the program number only is displayed.

**NC-PROGRAM DIRECTORY DISPLAY**

<table>
<thead>
<tr>
<th>Program Number</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>O0001</td>
<td></td>
</tr>
<tr>
<td>O0002</td>
<td></td>
</tr>
<tr>
<td>O4512</td>
<td></td>
</tr>
<tr>
<td>O4513 (PROGRAM BY JIM SMITH)</td>
<td></td>
</tr>
<tr>
<td>O4514</td>
<td></td>
</tr>
<tr>
<td>O7000</td>
<td></td>
</tr>
<tr>
<td>O9999</td>
<td></td>
</tr>
</tbody>
</table>

Only the program number appears when no text is included on the same line

Program number and text for identification appear in the directory when text is included.
2.2 How to key-in a program on a FANUC control

1. Switch the mode selector to “EDIT”-mode
2. Memory-protect key in OFF position
3. Press the “PROGRAM”-key. Now, either the program “TEXT” of an existing program is displayed or the program “DIRECTORY” is displayed. Pressing the “PROGRAM”-key again will toggle the display. Either one of the two displays permits program registration.
4. Key-in the letter O followed by the program number, then push the “INSERT”-key. No other characters are allowed at this time.
5. Now, all remaining text is keyed in.
6. To start a new line, press EOB, then press the “INSERT”-key.

2.3 Notes on program registration

- Upon registration of a program the new file is displayed on the CRT immediately. The text of an existing program shown on the CRT just prior to inserting a new program disappears into the background. Nothing has been deleted. The “DIRECTORY” page lists all of the files that exist in the memory by file number.
- A file number that already exists cannot be entered again. The control will not overwrite an existing file.
- Files need not necessarily be entered in numerical order. The program directory sorts the file numbers in ascending numerical order, not in the order of registration.
- When inserting a program via keypad the text in parenthesis cannot be inserted together with the program number. The program number by itself must be inserted first, then the text.
- Text that is set in parenthesis is not regarded as a NC-word or a command.
- The character “;” (semicolon) marks the end of a line. On the key-pad of the control the semicolon-key is marked as “EOB”. This stands for END OF BLOCK.

2.4 Notes on writing a program on a PC

When a NC-program is created on a PC, please note the following:

- The percent sign “%” must stand at the head of the program and on the tail end on a line by itself.
- Semicolon (EOB character) must not be added at end of a line. Pressing the “enter”-key at the end of line or end of block starts a new line.
- NC programs must be saved in Text Format. This allows uploading the file to a FANUC control directly. For details how to upload a program to the NC system, please consult the operation manual.
# List of Commands used in NC-Lathe programming

This chapter provides an overview of the basic commands and function codes that are used in NC-programming.

## 3.1 Addresses

A NC-address consists of a single letter that addresses an assigned function on the control system. The table below explains the function of addresses typically used in NC programming. A number always follows an address.

<table>
<thead>
<tr>
<th>Function</th>
<th>Address</th>
<th>Application / Use</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program number</td>
<td>O</td>
<td>O1234; Letter O followed by a 4-digit number registers a program in the NC-memory.</td>
</tr>
<tr>
<td>Sequence number</td>
<td>N</td>
<td>N1234; Line-number or sequence number. The use of line numbers allows manual and automatic line-search function.</td>
</tr>
<tr>
<td>Preparatory function</td>
<td>G</td>
<td>G1 through G99; Control function. (See the G-code list for detailed description of the G-codes)</td>
</tr>
<tr>
<td>Coordinate address</td>
<td>X,Z,U,W</td>
<td>X1.2345; Coordinate addresses for axis position command.</td>
</tr>
<tr>
<td>Coordinate address</td>
<td>R</td>
<td>R1.2345; Arc or corner radius specification.</td>
</tr>
<tr>
<td>Coordinate address</td>
<td>I,K</td>
<td>I-0.1234; Used for defining the location of an arc center point. Also used for chamfering function.</td>
</tr>
<tr>
<td>Feed function</td>
<td>F</td>
<td>F0.0005; Specifies motion distance per spindle revolution, or thread lead, or motion speed in units of inches per minute.</td>
</tr>
<tr>
<td>Spindle speed function</td>
<td>S</td>
<td>S2000; Spindle speed specifications in RPM-units or surface speed in units of feet per minute.</td>
</tr>
<tr>
<td>Tool function</td>
<td>T</td>
<td>T0101; Tool selection &amp; tool offset command</td>
</tr>
<tr>
<td>Function</td>
<td>Address</td>
<td>Application / Use</td>
</tr>
<tr>
<td>-----------------------</td>
<td>---------</td>
<td>-----------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Machine function</td>
<td>M</td>
<td>M8; Used for activating various machine functions, such as spindle, coolant, etc.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(See M-code list for details)</td>
</tr>
<tr>
<td>Search function</td>
<td>P</td>
<td>Used for sub program call M98P_ and for sequence number search command M99P_.</td>
</tr>
<tr>
<td>Dwell function</td>
<td>P, U</td>
<td>Dwell command. G4 P1= Dwell-time =1 millisecond</td>
</tr>
<tr>
<td></td>
<td></td>
<td>G4 U1. = Dwell-time =1 second.</td>
</tr>
<tr>
<td>Repetition function</td>
<td>L, P</td>
<td>Number of times to repeat a sub program. M98 P1234 L100 , or M98 P1001234</td>
</tr>
<tr>
<td>Canned cycle function</td>
<td>I, K, P, Q, R</td>
<td>These addresses when used together with canned cycles have functions other than as outlined, above.</td>
</tr>
</tbody>
</table>

### 3.2 Words

The programmer must translate an action or a task that the machine is asked to do into a command, called a NC-word. An address from the table above, combined with a number forms a word.

For example: **X2.5** - This is a word. X is the address, 2.5 representing the coordinate position on the X-axis.

A NC-word consists of an address followed by a number.

More than one word may be required at the same line in order to execute a task.

For example: **G0 X2.5 Z1 M8**. This command instructs the machine to position a tool at the coordinates X2.5, Z1 and to turn on the coolant.
3.3 **Blocks**

One or more words make up a command-line that is referred to as a block. A block may contain as many words as is required in order to specify different types of commands at the same time.

This is a block:

```
G1 G99 G96 X-.063 F.005 S500 P11;
```

When the data is processed by the control the whole block is read at once. All of the commands in a block are executed immediately. Upon completed execution of every command a completion signal is sent back from the machine to the control. Then the next block is processed, and so on.

3.4 **End of Block Character**

The character ";" (semicolon or “EOB”) is placed at the end of each block or at the end of a line. The “End of Block” needs to be inserted when the program text is manually keyed into the control. It separates one line or block from another.

When a program is created on a PC, semicolon is not required. Pressing the “Enter” key on the keyboard produces the “CR” and “LF” characters that is interpreted by the control as a semicolon or end of block (EOB).

3.5 **Range of Input Data**

Numbers or data used in programming must fit a certain range. The data range varies depending on type of address, unit of measurement and machine type. The data range in the table below is applicable for most NC lathes. Maximum data range shown for coordinates is the theoretical upper limit. Actual maximum range is lower, depending on size and type of machine.

<table>
<thead>
<tr>
<th>Address</th>
<th>Inch System</th>
<th>Metric System</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Min.</td>
<td>Max.</td>
</tr>
<tr>
<td>X,Z,U,W,R,I,K</td>
<td>+/-0.0001</td>
<td>+/-99.9999</td>
</tr>
<tr>
<td>F</td>
<td>0.000001</td>
<td>99.9999</td>
</tr>
<tr>
<td>G</td>
<td>0</td>
<td>99</td>
</tr>
<tr>
<td>S</td>
<td>1</td>
<td>32767</td>
</tr>
<tr>
<td>T</td>
<td>0</td>
<td>1232</td>
</tr>
<tr>
<td>L</td>
<td>1</td>
<td>999</td>
</tr>
<tr>
<td>M</td>
<td>0</td>
<td>99(999)</td>
</tr>
<tr>
<td>N</td>
<td>0</td>
<td>9999</td>
</tr>
<tr>
<td>O</td>
<td>0</td>
<td>9999</td>
</tr>
</tbody>
</table>
3.6 The use of decimal fractions in programming

In words with addresses G, L, M, N, O, P, Q, S, T the use of a decimal point is prohibited. Whole numbers must be commanded only.

Example: P1. Cannot be commanded. Command P1
S499.5 P11 cannot be commanded. Command S499 P11 or S500 P11

In words with addresses F, I, K, R, U, W, X, Z the use of decimal fractions is permitted:

Example: X2.5  =2.5" diameter on the X axis
        F.018  =0.018" feed rate
        Z-1.125 =1.125" in the negative direction on Z axis
        X1.    =1" diameter in X axis

Any coordinate word can be written as an integer (Integer = a number without a decimal point).

Example: The word X1.0 can be expressed as X10000
CAUTION: X1.0 cannot be expressed as X1
The number 1 is interpreted by control as 0.0001

For an address that allows the use of decimal fractions, the decimal point must not be omitted accidentally.

When the decimal point is omitted with an address that permits the use of decimal fractions, the system reverts to the least input increment as shown in the table, below.

<table>
<thead>
<tr>
<th>Program Command input</th>
<th>System output</th>
</tr>
</thead>
<tbody>
<tr>
<td>X1</td>
<td>X0.0001</td>
</tr>
<tr>
<td>X10</td>
<td>X0.0010</td>
</tr>
<tr>
<td>X100</td>
<td>X0.0100</td>
</tr>
<tr>
<td>X1000</td>
<td>X0.1000</td>
</tr>
<tr>
<td>X10000</td>
<td>X1.0000</td>
</tr>
</tbody>
</table>

Applicable for addresses I, K, P, Q, R, U, W, X, Z
3.7 Suppression of Leading or trailing Zero

Data format on older NC systems (prior to around 1982) used to require a fixed number of digits for NC-words. Systems manufactured after around 1982 do not require this type of format any longer. It is not necessary to place a zero in front or at the end of a number or in front of a decimal point.

For example instead of typing: “G01, M01, T0001, F0.1000” and so on it is acceptable to type: “G1, M1, T1, F.1” The actual value of the number must remain unchanged, of course.

For example: The accuracy of the number “1.0000” is the same as that of the number “1.”

When zero is part of a whole number or part of a decimal fraction the zero cannot be suppressed, of course.

3.8 The use of Sequence Numbers in the program

A sequence number can be placed at the beginning of each block if desired. This is the address N with a number from 1-9999.

On older, tape operated controls it is mandatory to place a sequence number on every line. On most FANUC controls in use at this time sequence numbers are required only in connection with certain types of “canned” cycles. Selective use of sequence numbers is helpful with regards to simplification of search functions.

For example:

```
N10 (DRILL HOLE)
;
N20 (ROUGH TURN OUTSIDE)
;
N30 (FINISH BORE)
;
```

The format shown above establishes good overview and organization within a program. Sequence numbers should be placed in logical order. The numbers can be spaced in increments of 10, as shown in the example. Comments that provide clues with regards to operation details, tools, etc can be placed just after the sequence number. This practice is helpful for communication between the programmer, setup and machine operator personnel. The order of processing is not influenced by the numerical order of sequence numbers.
3.9 Optional Block Skip Character, “/” “Slash

Placing a slash “/” at the beginning of a block allows optional skipping (not executing) of the commands specified on that block.

A switch located on the operation panel controls the skip function. Switch ON = skips the block, switch OFF = executes the block.

This function is used for various purposes. Here is one example:

N70 M54;
/N80 M99; - Block skip function
N90 M30

In the above example, the control will ignore the block N80 when the "Optional Block Skip" switch turned on.

The slash must be placed at the start of the block so that the whole block of information is subject to the skip function. If it is placed in random order in the block the control will read & execute the block information up to the “/”. Words on the right of the “/” will be ignored.

3.10 Feed Command

A numerical value following the address “F” sets the feed rate. There are two different feed modes available. The feed-mode is selected by these G-codes:

G98 = IPM-Mode = Inches per minute mode
G99 = IPR-Mode = Inches per revolution mode

G99 is the standard feed mode on a NC-lathe. In G99-mode the command “F 0.005”, for example, sets a feed rate of 0.005” per spindle revolution. The feed motion is dependent on the spindle rotation. Without spindle rotation no feed motion can occur. The feed rate or feed amount to be used depends on machining application. Caution must be used in selecting the feed rate.

| On control power-up, the G99-command mode is selected by the system automatically. For normal machining the feed rate is always specified in units of Inches per spindle rotation (IPR). |

For bar pulling and for broaching operations where the spindle is not allowed to rotate the “Inches per minute” feed mode must be commanded. In G98-mode the command “F 20.” for example, sets a feed rate of 20” per minute.

Note that the G-codes G98 and G99 as well as the feed rate command “F” is modal. The meaning of “modal” is that a command remains active until it has been cancelled or replaced by another command related to the same family or group of commands.
For example:

- Once G98 has been commanded the inches per minute feed mode remains active in every block. It’s not necessary to repeat the command. The G99 command cancels the G98-mode.
- Once a specific feed rate has been commanded it remains active until replaced with a different feed rate.

Upon completion of an operation that uses the G98-feed mode the G99 command together with an applicable feed rate command must restore the normal feed mode, without fail.

For example:

When the command \texttt{G1 G98 Z-1.25 F20.0} has been used for bar pulling. Upon completion of this operation, the instructions for the machining operation to follow just after the bar pulling operation must include the commands G99 with an applicable feed rate, without fail. Suppose the spindle runs at 3000 RPM at the time. If the feed rate of 20” is still active and the G99-mode has been command the feed is now 20” per revolution or about 83 feet per second – just under 60 miles per hour.
3.11 Spindle Command

A numerical value following the address “S” sets the spindle speed. There are two different types of spindle control modes available. The spindle control mode is set by these G-codes:

- G96 = Constant surface speed control mode
- G97 = RPM control mode

The command: G97 S2000 M3 P11 runs the spindle at 2000 RPM. The M3 or M4 command the direction of spindle rotation, forward or reverse. The M5 command stops the spindle.

The command: G96 S500 M3 P11 runs the spindle at a constant surface speed (circumference speed) of 500 feet per minute. In this mode the control automatically calculates the spindle RPM, based on the diameter that is machined, at any given time. The following formula is applied for calculation of the RPM:

\[
\text{RPM} = \frac{12 \times V}{D \times 3.14}
\]

Where:
- \( V \) = Cutting Speed (feet/minute)
- \( D \) = Work piece diameter (inch)

OR:

\[
\text{RPM} = \frac{3.82 \times V}{D}
\]

For example:

According to this formula, when a cutting speed (\( V \)) of 500 feet per minute G96 S500 P11 has been commanded the spindle will run 1910 RPM when machining is done on a 1” diameter part.

When machining is done on 0.1-inch diameter the control will command the spindle to run at 19100 RPM. The maximum RPM available on a CNC Turning Center is usually less than 6000 RPM. On larger machines a spindle speed over 1500 RPM may cause hazardous operating conditions.

For safety the maximum spindle RPM must be clamped at a safe speed. Suppose the maximum permissible spindle speed is 1200 RPM, the command G50 S1200 must be included in the program.

When constant surface speed control is used the command G50 S-(maximum) must stand at the beginning of a program, prior to any other spindle command.
The G50 S-command is valid only during G96-mode. “G50 S1200 P11” does not override a “G97 S2000 P11” -command. The G50 S-command is modal.

3.12 Tool Command

A tool command consists of the Address “T” followed by a number ranging from 0 to around 1232. The upper limit of the range depends upon the number of turret positions and the number of offsets available on a system. Two different functions are combined in this number. Separate functions are assigned to the lower two digits and to the upper two digits of the tool command number, as shown, below.

```
T □ □ □ □ □ □ □ □ □
```

TOOL OFFSET NUMBER

TURRET POSITION OR TOOL NUMBER

The upper two digits represent the turret position or the tool number command. The lower two digits represent the tool-offset number command.

When the lathe is equipped with a turret a tool is indexed or rotated to the cutting position by commanding the turret position number of that tool using the first two digits of the T-command.

For example:  
T0100 = Turret position #1 (or tool #1).

This command takes care of tool selection only. The tool-offset must be commanded, additionally.

3.13 Tool-offsets

The need for tool-offsets is due to the fact that it is very difficult to physically attach a tool to the machine so that the tool-tip lines up precisely with a desired point in the coordinate system.

A tool offset is applied to make up for the difference that exists between the actual location of the tool tip and the theoretical point where the tip should be located in relation to the coordinate system. Once a tool has been firmly attached to the machine the difference between tool tip and coordinate system is measured and recorded on a data table called the “Tool-Offset Register”.

At the time when a tool is commanded into cutting position the compensation data for that tool must be activated. This is done by the tool-offset command.

The tool-offset command instructs the control to get the data from the offset register and compound it to the coordinate system.

Tool Offset Register
Tool offsets are important for size control of the part to be machined. After a part has been machined and inspected for dimensional accuracy, any size correction that might be needed is manually keyed into the tool-offset register. The next machined part will reflect the size correction that has been input.

### 3.14 Tool-Offset Command

Every tool must use tool-offset compensation in order to cut a specified dimension correctly on size. A specific tool offset number in the tool-offset register is assigned to every tool number. For example, the offset compensation data for tool #1 is recorded in tool-offset register #1, for tool #2 the offset data is in offset register #2, and so forth.

In a standard tool command the tool number and tool-offset number is always commanded at the same time.

The complete tool command for tool #1 is “T0101” = Tool #1 and offset #1

- **Alternate tool-offsets.**

A tool can be commanded using different offsets.

For example:

- T0101 = Tool #1 and offset #1 -primary offset
- Or: T0121 = Tool #1 and offset #21 -alternate offset

Tool offsets are not stacked or compounded on top of one another. When T0121 has been commanded after T0101, then tool-offset #21 only is effective.

- **Offset command, only**

It is possible to command a tool offset number only, without a tool number. Once a tool has been placed into the cutting position on the turret the tool-offset command can be used just by itself.

For example: T0021 = Tool-offset #21

Typically, on a gang-type lathe where all the tools are arranged in a position ready for cutting, no tool call is needed. The tool is already in place. The upper two digits of the tool command are never used in this case. Only the lower two digits are used for calling the tool offset.
4 Tool Nose Radius Compensation

Most carbide turning tool inserts employ a tool nose radius at the tool tip for the purpose of increasing tool life. Standard radii range from 1/64" up to 3/32" in 1/64th increments, typically. In some cases, radius-grooving tools are used for machining.

In writing a NC-program the tool nose radius must be considered. When machining is done parallel to "X" or parallel to "Z" tool nose radius compensation is not required, regardless of tool nose radius size. Machining of tapers or arcs requires tool nose radius compensation.

Tool nose radius compensation can be solved in two different ways:

1. Compensating for the tool nose radius in the tool path geometry in the NC-program. This method produces the best results. However, without the use of a PC equipped with CAM software, the calculation of the tool path geometry can become very cumbersome and time consuming.

2. Utilizing the tool nose radius compensation function that is available on the NC control, by including the TNR-COMP commands (G41, G42 and G40) in the program.

4.1 G40, G41 and G42 Tool Nose Radius Compensation Function Commands

G40 = Cancel the tool nose radius compensation function
G41 = Tool nose radius compensation - left
G42 = Tool nose radius compensation - right

Geometry of a tool path that has been programmed using the actual part dimensions can be produced correctly when the automatic tool nose radius compensation function is applied.

Rules to apply in the use of the tool nose radius compensation function

1. Tool nose radius and tool nose vector must be registered in the tool-offset tables under the offset number to be used for a given tool. (See next page for more details)
2. The TNR compensation code (G41 or G42) must be commanded together with an axis move command, (G0 or G1) one block before actual machining on the part begins. This is called a “RAMP-ON”-move. The axis move must not be smaller than twice the TNR in “X” or not smaller than the actual TNR when ramping ON in “Z” direction.
3. RAMP-ON moves should be done to a point in the Z direction that is at least “1 x R” clear of the first surface to be machined and “X” on the diameter to be machined whenever possible.
4. The TNR compensation command is modal. When a program is interrupted during TNR COMP, “G40” must be commanded at the program start up.
5. When G40 is commanded together with an axis move ("RAMP-OFF"), the move distance must be at least twice the TNR in X, or at least one TNR in Z.
6. G40 must not be commanded during actual machining on the part. It will produce errors on the part geometry.
7. Arc command (G2 or G3) for inside radii that are smaller than the tool nose radius is not allowed.
8. Small steps at an inside corner or undercuts, smaller than the TNR are not allowed.
9. In TNR-COMP mode, the cutting direction must never be reversed 180°.
10. In TNR-COMP mode, not more than one block without an “X” or “Z” command is allowed.
4.2 Tool Nose Radius Compensation Data

Tool nose radius compensation is stored in the tool-offset tables under the columns “R” and “T”. The tool offset that stores the TNR COMP data for the tool in use must be activated by the tool command, otherwise no compensation is possible.

Tool Offset Register

<table>
<thead>
<tr>
<th>NO. offset #</th>
<th>X offset</th>
<th>Z offset</th>
<th>R Tool nose radius</th>
<th>T Tool nose vector</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>1.75</td>
<td>-.025</td>
<td>.0312</td>
<td>3</td>
</tr>
<tr>
<td>02</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

1. The “R” column must contain the actual tool nose radius.
2. The “T” column must contain the tool vector, which is selected from the sketch as shown below.

NOTES:
1. Turning tools are touched off the normal way. TNR-Compensation only covers the 90°-sector in which the TNR is located.
2. Compensation for radius grooving tools covers the 180°-sector in which the TNR is located. When TNR-COMP is used with radius grooving tools, they must be touched of as follows:
   - Vector 5 and 7: touch off “Z” the normal way, touch off “X” at the center of the radius
   - Vector 6 and 8: touch off “X” the normal way, touch off “Z” at the center of the radius
   Vector 0: touch off “X” the and “Z” at center of the radius

5 G-Codes
G-codes activate various types of control functions or set modes of operation. G-codes are subdivided into groups or families, by function-type as shown in the table, below.

**Special Notes on G-Codes**

- The following G-codes are active upon initial power-up of the control: G0, G18, G22, G40, G54, G80, G97, and G99.

- G-codes from different groups can be commanded at the same time or in the same block.

- One G-code only from the same group can be commanded in a block. When more than one G-code from the same group is commanded the last G-code from within a group specified is activated.

- A G-code that is currently active is replaced by commanding another G-code from within the same group or family.

- G-codes of group 00 are classified as “single-shot” G-codes, meaning that the G-command remains active only on the block in which it is specified. They are also called “Non-modal G-codes”.

- G-codes of all other groups are “Modal”, meaning that once the G-code has been commanded it remains active until replaced by another G-code from within the same group or family.

- Commanding a G-code alone may or may not be sufficient for executing a function. Programming examples shown later in this manual explain detailed application of G-codes.

- The standard G-codes used for NC-lathe programming are somewhat different from G-codes used in Machining Center programming. G-code System “A” is mostly used for NC-lathe programming. G-code systems “B” and “C” are more closely related to G-codes used in machining Center programming. System B and C are optional equipment on certain types of controls.
### 5.1 G-Code List (G-code system A, partial listing)

<table>
<thead>
<tr>
<th>G Code</th>
<th>Group</th>
<th>Modal</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>01</td>
<td>Yes</td>
<td>Rapid traverse</td>
</tr>
<tr>
<td>G01</td>
<td>01</td>
<td>Yes</td>
<td>Linear interpolation</td>
</tr>
<tr>
<td>G02</td>
<td>01</td>
<td>Yes</td>
<td>CW Circular interpolation</td>
</tr>
<tr>
<td>G03</td>
<td>01</td>
<td>Yes</td>
<td>CCW Circular Interpolation</td>
</tr>
<tr>
<td>G04</td>
<td>00</td>
<td>No</td>
<td>Dwell</td>
</tr>
<tr>
<td>G10</td>
<td>00</td>
<td>Yes</td>
<td>Data Setting</td>
</tr>
<tr>
<td>G11</td>
<td>00</td>
<td>Yes</td>
<td>Data Setting cancel</td>
</tr>
<tr>
<td>G17</td>
<td>16</td>
<td>Yes</td>
<td>X-Y Plane select</td>
</tr>
<tr>
<td>G18</td>
<td>16</td>
<td>Yes</td>
<td>Z-X Plane select</td>
</tr>
<tr>
<td>G19</td>
<td>16</td>
<td>Yes</td>
<td>Y-Z Plane select</td>
</tr>
<tr>
<td>G20</td>
<td>06</td>
<td>Yes</td>
<td>Inch data input</td>
</tr>
<tr>
<td>G21</td>
<td>06</td>
<td>Yes</td>
<td>Metric data input</td>
</tr>
<tr>
<td>G28</td>
<td>00</td>
<td>No</td>
<td>Return to home position</td>
</tr>
<tr>
<td>G30</td>
<td>00</td>
<td>No</td>
<td>Return to 2ⁿ reference point</td>
</tr>
<tr>
<td>G31</td>
<td>00</td>
<td>No</td>
<td>Skip function</td>
</tr>
<tr>
<td>G32</td>
<td>01</td>
<td>Yes</td>
<td>Thread cutting</td>
</tr>
<tr>
<td>G40</td>
<td>07</td>
<td>Yes</td>
<td>Cancel tool nose radius compensation</td>
</tr>
<tr>
<td>G41</td>
<td>07</td>
<td>Yes</td>
<td>Tool nose radius compensation left</td>
</tr>
<tr>
<td>G42</td>
<td>07</td>
<td>Yes</td>
<td>Tool nose radius compensation right</td>
</tr>
<tr>
<td>G50</td>
<td>00</td>
<td>No</td>
<td>Spindle speed limit in G96 mode</td>
</tr>
<tr>
<td>G50</td>
<td>00</td>
<td>No</td>
<td>Coordinate system setting</td>
</tr>
<tr>
<td>G53</td>
<td>00</td>
<td>No</td>
<td>Machine coordinate system</td>
</tr>
<tr>
<td>G54</td>
<td>14</td>
<td>Yes</td>
<td>Work coordinate system 1</td>
</tr>
<tr>
<td>G55</td>
<td>14</td>
<td>Yes</td>
<td>Work coordinate system 2</td>
</tr>
<tr>
<td>G56</td>
<td>14</td>
<td>Yes</td>
<td>Work coordinate system 3</td>
</tr>
<tr>
<td>G57</td>
<td>14</td>
<td>Yes</td>
<td>Work coordinate system 4</td>
</tr>
<tr>
<td>G58</td>
<td>14</td>
<td>Yes</td>
<td>Work coordinate system 5</td>
</tr>
<tr>
<td>G59</td>
<td>14</td>
<td>Yes</td>
<td>Work coordinate system 6</td>
</tr>
<tr>
<td>G70</td>
<td>00</td>
<td>No</td>
<td>Finishing cycle</td>
</tr>
<tr>
<td>G71</td>
<td>00</td>
<td>No</td>
<td>Roughing cycle, multiple repetitive</td>
</tr>
<tr>
<td>G72</td>
<td>00</td>
<td>No</td>
<td>Face roughing, multiple repetitive</td>
</tr>
<tr>
<td>G73</td>
<td>00</td>
<td>No</td>
<td>Pattern repeating, for castings</td>
</tr>
<tr>
<td>G74</td>
<td>00</td>
<td>No</td>
<td>Z axis chip breaking (drilling)</td>
</tr>
<tr>
<td>G75</td>
<td>00</td>
<td>No</td>
<td>Grooving, X axis</td>
</tr>
<tr>
<td>G76</td>
<td>00</td>
<td>No</td>
<td>Thread cutting, multiple repetitive</td>
</tr>
<tr>
<td>G90</td>
<td>01</td>
<td>Yes</td>
<td>ID/ OD box cycle</td>
</tr>
<tr>
<td>G92</td>
<td>01</td>
<td>Yes</td>
<td>Thread cutting cycle</td>
</tr>
<tr>
<td>G96</td>
<td>02</td>
<td>Yes</td>
<td>Constant surface footage programming</td>
</tr>
<tr>
<td>G97</td>
<td>02</td>
<td>Yes</td>
<td>Continuous RPM programming</td>
</tr>
<tr>
<td>G98</td>
<td>05</td>
<td>Yes</td>
<td>Feed rate per minute</td>
</tr>
<tr>
<td>G99</td>
<td>05</td>
<td>Yes</td>
<td>Feed rate per revolution</td>
</tr>
</tbody>
</table>
## 6 Miscellaneous Functions, “M”-codes

Please note that M-codes may vary from one machine tool builder to another. Most of the M-codes shown on the list shown below are generally valid for PUMA Turning Centers only. M0 through M9 may apply for other brands of turning centers, as well.

One M-code only is allowed in a block. M-codes can be executed in the same block with other NC-commands, such as G-codes, spindle, tool and axis commands.

### 6.1 M-Code List

<table>
<thead>
<tr>
<th>M-Code</th>
<th>Description</th>
<th>Spec.</th>
</tr>
</thead>
<tbody>
<tr>
<td>M0</td>
<td>Program Stop</td>
<td></td>
</tr>
<tr>
<td>M1</td>
<td>Optional Stop</td>
<td></td>
</tr>
<tr>
<td>M2</td>
<td>Program Reset or Rewind and Reset</td>
<td></td>
</tr>
<tr>
<td>M3</td>
<td>Spindle Forward</td>
<td></td>
</tr>
<tr>
<td>M4</td>
<td>Spindle Reverse</td>
<td></td>
</tr>
<tr>
<td>M5</td>
<td>Spindle Stop</td>
<td></td>
</tr>
<tr>
<td>M7</td>
<td>High Pressure Coolant</td>
<td>Option</td>
</tr>
<tr>
<td>M8</td>
<td>Coolant On</td>
<td></td>
</tr>
<tr>
<td>M9</td>
<td>Coolant Off</td>
<td></td>
</tr>
<tr>
<td>M10</td>
<td>Parts Catcher Advance</td>
<td>Option</td>
</tr>
<tr>
<td>M11</td>
<td>Parts Catcher Retract</td>
<td>Option</td>
</tr>
<tr>
<td>M14</td>
<td>Main Spindle Air Blow</td>
<td>Option</td>
</tr>
<tr>
<td>M15</td>
<td>Main Spindle Air Blow Off</td>
<td>Option</td>
</tr>
<tr>
<td>M17</td>
<td>Machine Lock ON</td>
<td></td>
</tr>
<tr>
<td>M18</td>
<td>Machine Lock OFF</td>
<td></td>
</tr>
<tr>
<td>M19</td>
<td>Main Spindle Orientation</td>
<td>Option</td>
</tr>
<tr>
<td>M24</td>
<td>Chip Conveyor Run</td>
<td>Option</td>
</tr>
<tr>
<td>M25</td>
<td>Chip Conveyor Stop</td>
<td>Option</td>
</tr>
<tr>
<td>M30</td>
<td>Program End With Rewind and reset</td>
<td></td>
</tr>
<tr>
<td>M31</td>
<td>Interlock by-pass (for Spindle &amp; Tailstock)</td>
<td></td>
</tr>
<tr>
<td>M34</td>
<td>C1-AXIS SELECT OFF</td>
<td></td>
</tr>
<tr>
<td>M35</td>
<td>C1-AXIS SELECT ON</td>
<td></td>
</tr>
<tr>
<td>M38</td>
<td>Steady Rest Right Clamp</td>
<td>Option</td>
</tr>
<tr>
<td>M39</td>
<td>Steady Rest Right Unclamp</td>
<td>Option</td>
</tr>
<tr>
<td>M40</td>
<td>Gear Change Neutral</td>
<td></td>
</tr>
<tr>
<td>M41</td>
<td>Gear Change Low</td>
<td></td>
</tr>
<tr>
<td>M42</td>
<td>Gear Change Middle</td>
<td></td>
</tr>
<tr>
<td>M43</td>
<td>Gear Change Middle High</td>
<td></td>
</tr>
<tr>
<td>M44</td>
<td>Gear Change High</td>
<td></td>
</tr>
<tr>
<td>M46</td>
<td>Tailstock Body Unclamp. &amp; Traction-Bar engage.</td>
<td>Option</td>
</tr>
<tr>
<td>M47</td>
<td>Tailstock Body Clamp. &amp; Tract-Bar Retract.</td>
<td></td>
</tr>
<tr>
<td>M48</td>
<td>Override Invalid</td>
<td></td>
</tr>
<tr>
<td>M49</td>
<td>Override Valid</td>
<td></td>
</tr>
<tr>
<td>M50</td>
<td>Bar Feeder Command 1</td>
<td>Option</td>
</tr>
<tr>
<td>M-Code</td>
<td>Description</td>
<td>Spec.</td>
</tr>
<tr>
<td>--------</td>
<td>-------------------------------------------------</td>
<td>---------</td>
</tr>
<tr>
<td>M51</td>
<td>Bar Feeder Command 2</td>
<td>Option</td>
</tr>
<tr>
<td>M52</td>
<td>Splash Guard Door Open</td>
<td>Option</td>
</tr>
<tr>
<td>M53</td>
<td>Splash Guard Door Close</td>
<td>Option</td>
</tr>
<tr>
<td>M54</td>
<td>Parts Count</td>
<td>Option</td>
</tr>
<tr>
<td>M58</td>
<td>Steady Rest Clamp</td>
<td>Option</td>
</tr>
<tr>
<td>M59</td>
<td>Steady Rest Unclamp</td>
<td>Option</td>
</tr>
<tr>
<td>M61</td>
<td>Switching Low Speed</td>
<td></td>
</tr>
<tr>
<td>M62</td>
<td>Switching High Speed</td>
<td></td>
</tr>
<tr>
<td>M66</td>
<td>Main CHUCKING LOW PRESSURE</td>
<td>Option</td>
</tr>
<tr>
<td>M67</td>
<td>Main CHUCKING HIGH PRESSURE</td>
<td>Option</td>
</tr>
<tr>
<td>M68</td>
<td>Main-Chuck Clamp</td>
<td></td>
</tr>
<tr>
<td>M69</td>
<td>Main-Chuck Unclamp</td>
<td></td>
</tr>
<tr>
<td>M70</td>
<td>Dual Tailstock Low Advance</td>
<td>Option</td>
</tr>
<tr>
<td>M73</td>
<td>TOUCH PROBE OFF</td>
<td></td>
</tr>
<tr>
<td>M74</td>
<td>TOUCH PROBE ON</td>
<td></td>
</tr>
<tr>
<td>M76</td>
<td>Q SETTER SWING ARM UP</td>
<td></td>
</tr>
<tr>
<td>M77</td>
<td>Q SETTER SWING ARM DOWN</td>
<td></td>
</tr>
<tr>
<td>M78</td>
<td>Tailstock Quill Advance</td>
<td></td>
</tr>
<tr>
<td>M79</td>
<td>Tailstock Quill Retract</td>
<td></td>
</tr>
<tr>
<td>M86</td>
<td>A AXIS Torque Skip Active</td>
<td></td>
</tr>
<tr>
<td>M87</td>
<td>A AXIS Torque Skip Cancel</td>
<td></td>
</tr>
<tr>
<td>M89</td>
<td>C-AXIS clamp or, Spindle Clamp for non C-axis machines</td>
<td></td>
</tr>
<tr>
<td>M90</td>
<td>C-AXIS un-clamp or, Spindle Un-Clamp for non C-axis machines</td>
<td></td>
</tr>
<tr>
<td>M91</td>
<td>External M91 Command</td>
<td>Option</td>
</tr>
<tr>
<td>M92</td>
<td>External M92 Command</td>
<td>Option</td>
</tr>
<tr>
<td>M93</td>
<td>External M93 Command</td>
<td>Option</td>
</tr>
<tr>
<td>M94</td>
<td>External M94 Command</td>
<td>Option</td>
</tr>
<tr>
<td>M98</td>
<td>Sub-Program Call</td>
<td></td>
</tr>
<tr>
<td>M99</td>
<td>End of Sub-Program</td>
<td></td>
</tr>
</tbody>
</table>
7 Coordinate Systems

7.1 Basic Coordinate System

Shown below is a standard two-dimensional coordinate system where the X-axis runs in horizontal direction and the Y-axis in vertical direction. X is the first and Y is the second axis in the basic coordinate system. In NC-lathe programming a different coordinate system is used, as shown on the sketch, below.

7.2 NC Lathe Coordinate System

In two-axis NC-lathe programming a coordinate system is applied that uses the first axis (X) and the third axis (Z) of the three-dimensional coordinate system. **X-axis coordinates are specified in diameter.** The Y-axis shown here is used when the lathe is equipped with a Y-axis.
The sketch below shows the standard two-axis NC-lathe-coordinate system in a two-dimensional view in which the X-axis runs vertically and the Z-axis horizontally. The point X0, Z0 is called the ORIGIN or ZERO-POINT of the coordinate system.

X-axis coordinates represent diameters on a part. Z-axis coordinates represent length dimensions.

“DIAMETER PROGRAMMING” is applied so that a diameter dimension from a part drawing can be entered directly into the program as an X-axis coordinate. Blue print dimensions are easily identified when looking at the text of a NC-program. This simplifies programming and program editing.

The size of a cylindrical object is normally specified as a diameter not as a radius. Using calipers or a micrometer for measuring a circular shaped object on the diameter is much easier than on the radius.

“Radius programming” (Using radius dimensions for X-axis coordinates) on a NC-lathe is possible but it is not recommended.
7.3 Machine Coordinate System

The origin or zero point of the machine coordinate system is normally located at the intersection of the main-spindle center axis (X0) and the spindle flange face (Z0). This point serves as a “hard” reference for calibration of the turret “HOME-position”. The machine coordinate system origin is the base point for all other coordinate systems. However, machine coordinates are normally not used for programming of a part.

The sketch below shows the machine coordinate system-grid with the location of MACHINE ORIGIN and MACHINE REFERENCE POINT. The coordinates as shown in this example: X10, Z10 are for illustration purpose only. Actual machine coordinates are different depending on size of machine.


7.4 **Work Coordinate System**

The ORIGIN of the coordinate system used for programming is established at a specific point on the part to be machined. This is called the WORK ZERO POINT. The coordinate system used in a NC-program is called the WORK COORDINATE SYSTEM.

The X-axis work zero point on a NC-lathe is always set at the center axis of the spindle. This is also the center axis of the work piece. The origin of the X-axis work coordinates is always the same as the origin of the X-axis machine coordinates.

Before programming a new part the programmer must decide the location of the work zero point along the Z-axis. Placing the Z-axis zero point at the right end face of the part to be machined is recommended. However, this is at the programmer’s discretion.

Suppose the work zero point has been decided on the right face of the part as shown above. During machine setup the distance between machine zero point and the work zero point along the Z-axis is measured and recorded in the work offset register. (See sketch, below)
Older NC-lathes are equipped with a single work offset register, known as the “WORK SHIFT”. The work-offset distance is entered into the work shift register. This will set the origin of the “Work Coordinate System” or the program zero point. Setting the work zero point on the machine is the responsibility of the setup person.

At the instant when the work zero point is set the “ABSOLUTE POSITION”-display on the machine is updated automatically.

Modern turning centers are equipped with six work-offset registers that make up six different work coordinate systems.

Work coordinate systems are selected or commanded by G-codes, as follows:

- G54 - Work coordinate 1
- G55 - Work coordinate 2
- G56 - Work coordinate 3
- G57 - Work coordinate 4
- G58 - Work coordinate 5
- G59 - Work coordinate 6

G54 serves as the default coordinate system on power-up. When no work coordinate system has been commanded Work coordinate 1 (G54) is selected automatically.
The sketch below shows half a cross-section of the part to be machined. For programming purposes the part drawing is placed onto the coordinate system grid with the right face and the center aligned with the work coordinate zero point. The intersecting points on the contour of the part represent the actual coordinates used in the NC-program.
7.7 Absolute Coordinate Command

A distance measured from the zero point to any point in the coordinate system is called an **ABSOLUTE Dimension**. Once the work zero point has been established the coordinates used for programming are referenced to that point.

For programming of the tool path that cuts the shape of the part as shown in the sketch below, the X and Z coordinates at the intersecting points P1 through P5 of the contour must be known.

The sketch shows **ABSOLUTE dimensions**. All of the dimensions are referenced to the origin of the coordinate system X0, Z0.

**Addresses X and Z specify absolute dimensions. Address X specifies diameter.**

The table below shows the points on the tool path, using absolute dimensions.

<table>
<thead>
<tr>
<th>POINT</th>
<th>diameter</th>
<th>length</th>
</tr>
</thead>
<tbody>
<tr>
<td>P1</td>
<td>X.6</td>
<td>Z0</td>
</tr>
<tr>
<td>P2</td>
<td>X.6</td>
<td>Z-.3</td>
</tr>
<tr>
<td>P3</td>
<td>X.8</td>
<td>Z-.35</td>
</tr>
<tr>
<td>P4</td>
<td>X.8</td>
<td>Z-.7</td>
</tr>
<tr>
<td>P5</td>
<td>X1.6</td>
<td>Z-.8</td>
</tr>
</tbody>
</table>

7.8 Incremental Coordinate Command
An incremental dimension is a distance measured from a point in a coordinate system to another point.

Dimensioning found on a shop drawing is not always convenient for use in NC-programming. When absolute coordinate commands \((X, Z,)_r\) are used all dimensions need to be referenced to the origin of the part. For the programmer’s convenience, NC-systems allow programming using both absolute and incremental dimensions.

**Address U specifies an incremental coordinate command along the X-axis. It represents an increment on diameter – (not on the radius).**

**Address W specifies an incremental coordinate command along the Z-axis.**

The table below shows incremental coordinate commands for the tool path that cuts the part shown above. Absolute coordinates are used only for the start point (P1).

<table>
<thead>
<tr>
<th>POINT</th>
<th>diameter</th>
<th>length</th>
</tr>
</thead>
<tbody>
<tr>
<td>P1</td>
<td>X.6</td>
<td>20</td>
</tr>
<tr>
<td>P2</td>
<td>U0</td>
<td>W-.3</td>
</tr>
<tr>
<td>P3</td>
<td>U.2</td>
<td>W-.1</td>
</tr>
<tr>
<td>P4</td>
<td>U0</td>
<td>W-.3</td>
</tr>
<tr>
<td>P5</td>
<td>U.8</td>
<td>W-.2</td>
</tr>
</tbody>
</table>
### 7.9 Absolute & Incremental Command in same Block

Absolute & incremental coordinates can be specified together in the same block.

For example: \( X3.395 \ W-3.0 \) Or: \( U1.625 \ Z-3.459 \)

### 8 Positioning

#### 8.1 G0 – Positioning in the Work Coordinate System

**Format G0 X (U) Z (W) Rapid traverse-move. (Modal)**

This command moves the turret or tools from the current position to a point specified by \( X, Z, U \) or \( W \) in the work coordinate system.

Positioning is used for moving a tool near to the part where machining starts or for retracting the tool away from the machining area. The positioning speed is up to 1000 inches per minute.

For example: Suppose a tool is located at the position \( X10" \), \( Z5" \) The tool needs to be positioned at \( X6" \), \( Z1" \)

Absolute command: \( G0 \ X6.0 \ Z1.0 \)

Incremental command: \( G0 \ U-4.0 \ W-4.0 \)

Either one of these commands will position the tool as shown, below.
As illustrated in the sketch, positioning is not necessarily done in a straight line from point A to point B. Positioning speed of both axis servos is about the same. In this case the travel distance along the X-axis is shorter than along Z. X arrives at the destination before Z.

In order to avoid collision between turret and Tailstock it is best to command the Z-axis move first then X-axis on the next block.

8.2 Positioning in the Machine Coordinate System

Format G53 X_ Z_ - Rapid traverse-move. Non modal

This command moves the turret or tools from the current position to a point specified by X, Z, in the machine coordinate system.

In the example as shown on the sketch, above:

A tool is located at the position X5", Z0" in the work coordinate system. The tool is to be positioned at X 7", Z 8" in the machine coordinate system.

Command: G53 X7.0 Z8.0 (Tool moves from A to B)

Notes for G53 Command:

- Incremental commands U and W are not valid with the G53-command
- G53 is a “one-shot” G-code, non-modal.
- Tool offset is not compounded to the coordinates
- The G53 command cannot be used for any other purpose other than as outlined, above.
- G53 can be used as a tool exchange point
9 Interpolation Function

9.1 G1 - Linear Interpolation

Format = G1 X (U) Z (W) F - This commands a linear move at a feed rate.

Linear interpolation means that both axes, X and Z will arrive at the commanded point at the same time.

The tool path as shown above is accomplished by linear interpolation commands as follows:

<table>
<thead>
<tr>
<th>Absolute dimensions</th>
<th>Incremental dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Start point is X0, Z0).</td>
<td></td>
</tr>
<tr>
<td>G1 X3.0 F 0.005</td>
<td>G1 U 3.0 F0.005</td>
</tr>
<tr>
<td>X5.0 Z-2.5</td>
<td>U2.0 W-2.5</td>
</tr>
<tr>
<td>Z-3.5</td>
<td>W-1.0</td>
</tr>
</tbody>
</table>

For linear interpolation, please note the following:

- When G1 is commanded a feed rate must be commanded as well or a feed rate must be active (modal).
- Linear interpolation starts from the current position of the tool. The commanded position in the G1-block represents the end position.
- When the end point is specified for one axis only a move parallel to that axis is produced.
- When a 2-axis move is commanded by G1 a straight line at an angle is produced.
- G1 remains modal. G0, G2, G3,cancels G1
9.2  G2 - Circular Interpolation Clockwise

Format = G2 X (U) Z (W) R_ F_ - Circular move (CW) at a commanded feed rate.

Start-point of arc: X1.0 Z0

Circular interpolation command: G2 X5.0 Z-2.0 R2.0 F0.005
Or: G2 U4.0 W-2.0 R2.0 F0.005

9.3  G3 - Circular interpolation Counter Clockwise

Format = G3 X (U) Z (W) R_ F_ - Circular move (CCW) at a commanded feed rate.

Start-point of arc: X1.0 Z0

Circular interpolation command: G3 X5.0 Z-2.0 R2.0 F0.005
Or: G3 U4.0 W-2.0 R2.0 F0.005
For circular interpolation in general, please note the following:

- Feed rate must be commanded or a feed rate must be active (modal) when G2 or G3 is commanded.
- G2 and G3 are modal.
- Circular interpolation starts from the current position of the tool. The commanded position in the G2 or G3-block represents the end-point of the arc.
- The start-point and end-point of an arc must be located geometrically accurate on the arc within 0.001".
- The "R"-command can be applied for any arc when observing the following rules:
  Positive "R command" is used for arc of 180 degrees or less.
  Negative "R command" is used for arc of more than 180 degrees.

For Example:

When an arc is larger than 180 degrees the "R"-command must be negative:
G3 X__ Z__ R (negative) - produces the correct tool path as shown in Figure 1.

When a positive "R" command is used for an arc larger than 180 degrees:
G3 X__ Z__ R (positive) - an incorrect tool path as shown in Figure 2 is produced.

![Figure 1](image1.png)
![Figure 2](image2.png)
### 9.4 Circular Interpolation using arc center point specification

**Format:** G3 X (U) Z (W) I_ K_ F_

![Diagram showing circular interpolation using arc center point specification](image)

Arc shown in the sketch:  
Start-point of arc: X5.0 Z0  
G3 X6.0 Z-4.0 I-1.75 K-2.25 F0.005

**Notes:**

- "I" and "K" specify the location of the arc-center relative to the start point of the arc.

- "I" represents a radial distance (not diameter) measured parallel to "X" (positive or negative) from the start point of the arc to the arc center point. When the distance is equal zero it can be specified as "I0" or it can be omitted.

- "K" represents a distance measured parallel to "Z" (positive or negative) from the start point of the arc to the arc center point. When the distance is equal zero it can be specified as "K0" or it can be omitted.

- "I" and or "K" replace the radius command "R". When "I" and or "K" is specified, "R" must not be specified.

- For an arc less than 360 degrees "R" can replace "I" and or "K". R is calculated by the following Formula:  
  \[ R = \sqrt{I^2 + K^2} \]

- "I" and or "K" specify a full circle, when start and end point coordinates X, Z both are the same. (This is normally used on lathes with milling capability only)
9.5 Chamfering & Corner Rounding Function (using Addresses “C”, “R”)

A 45-degree chamfer or a 90-degree arc can be produced when a surface parallel to X and an adjacent surface parallel to Z is machined in two consecutive blocks during G1-mode. The chamfering or corner rounding function is not available on surfaces at an angle other than 90 degrees to either X or Z.

In the examples shown below, either “C” for chamfer or “R” for corner rounding can be inserted in the block containing the X coordinates.

**Outside Chamfering Example**

G1 X0 Z0 (start point)  
G1 X1.0 **C-0.1** F.005  
G1 Z-0.5

**Outside Corner Rounding Example**

G1 X0 Z0 (start point)  
G1 X1.0 **R-0.1** F.005  
G1 Z-0.5
9.6 Chamfering Function (using Addresses “I”, “K”)

A chamfering function similar to the function described in the previous chapter is available, using the addresses “I” or “K”.

A 45-degree chamfer can be produced when a surface parallel to X and an adjacent surface parallel to Z is machined in two consecutive blocks during G1-mode. The chamfering function is not available on surfaces at an angle other than 90 degrees to either X or Z.

For this chamfering function the following rules apply:

- In the block that commands an X-axis move, use address “K” for defining the size of the chamfer.
- In the block that commands an Z-axis move, use address “I” for defining the size of the chamfer.

External Chamfer Example

G0 X1.0 Z0
G1 Z-.4 F.008
X2.4 K-.2
Z-.13 I.2
X3.2

Internal Chamfer Example

G0 X3.7 Z.1
G1 Z-.9 I-.2 F.007
X1.8 K-.2
Z-.16
X1.5

The chamfering function described above is dependent upon setting of parameter #3405, bit 4 setting = 0.
9.7 Thread Cutting Function (G32)

The G32-command is used for various types of thread cutting applications. This thread cutting function works similar to the linear interpolation function, G1, except that in G32-mode the rotation angle of the main spindle and the starting of the feed motion are synchronized. Feed rate and spindle override in thread cutting mode is disabled.

“Single-Point Threading” is normally applied for cutting a thread on a lathe. A form-tool that matches the shape of the thread is used. Due to the relatively weak structure of the thread-cutting tool and the high chip-load that is encountered, a thread cannot be cut in a single pass. Several cutting-passes at different depths along the entire length of the thread are usually required.

Principle of Single Point Threading

A rectangular shaped pattern as shown in the sketch below is used for the tool path that cuts the thread. The pattern is repeated at different cutting depths until the full depth of the thread is established.

G32 Format:

G32 X (U) Z (W) F (Thread Lead)

The G32-command is modal. (The commands: G0, G1, G2, G3, and some others cancel G32)
In the block with the G32-command the feed rate “F” specifies the Lead of the thread in inches per revolution.

“Inch-Standard” threads are normally specified by thread size and by the Pitch of the thread. Pitch, meaning the number of threads per inch, abbreviated “TPI”.
For a thread specified by threads per inch (TPI) the feed rate “F” is calculated as follows:

\[ F = \frac{1}{\text{TPI}} \]

For best LEAD-accuracy, “F” can be specified by up to 6-digits after the decimal point.

In G32-mode the synchronization between spindle rotation angle and starting of the feed- motion is automatic. This allows the tool to follow the path of an already existing thread lead on every subsequent cutting pass.

The sketch above shows (4) thread cutting passes. When the tool is to follow the lead of the first pass, each of the subsequent passes must start from the same Z-axis position as the first pass. The program would look something like this:

```
G0 X1.1 Z.1 (START POSITION)
X.980
G32 Z-.75 F.083333 (1ST PASS)
G0 X1.1
Z.1
X.960
G32 Z-.75 (2ND PASS)
G0 X1.1
Z.1
X.940
G32 Z-.75 (3RD PASS)
G0 X1.1
Z.1
X.920
G32 Z-.75 (4TH PASS)
G0 X1.1
Z.1
```

The program shown above represents a simplified example, plunge cutting the thread.

**Comments**

The program shown on the left shows four-passes on a 1”-12 UN OD-thread. The thread length = 0.75”.

The feed rate: 1/12=0.083333” per rev.
The feed rate is modal.

The Z axis start-position in the block prior to the G32-command always starts from Z.1
Programming of a thread requires some machining skills and experience. The following important factors must be considered when programming a thread:

- Type of material to be cut
- Thread shape or form
- Thread lead and thread height
- Mechanical strength of the work-piece

Selection of the cutting tool, spindle speed and thread cutting method is made based on the above factors.

The sketch below shows three different thread cutting methods that can be applied for V-shaped threads such as common screw and pipe threads.

- **Plunge cutting**

In plunge cutting the cutting tool is fed into the material perpendicular to the Z-axis. When a V-shaped thread is plunge cut, the tool is contact with the material at the tool tip and on both flanks. This cutting method works OK for plastic material, brass, bronze or cast iron. For V-shaped threads the chip forming action obtained by plunge cutting is not suitable when tough materials are cut. For square shaped thread forms, plunge cutting is the only cutting method available. In plunge cutting the G32-command is repeated always from the same starting position on the Z-axis. When V-shaped threads are plunge cut, the depth of cut should be reduced progressively with each pass. This will provide for constant chip-load.
Leading edge cutting

Leading edge cutting means that after the first pass only the left edge of the tool does the cutting. This is accomplished by shifting the Z-axis start position of the tool toward the thread with every pass. For V-shaped thread forms the leading edge cutting method works best, especially for materials with tough chip forming characteristics.

The G32-command can be applied for cutting of straight threads and tapered threads, internal or external. It can be used for scroll-threads or spirals that are located on the font face of a part.

The G32-command provides great flexibility in programming of the cutting pattern. Typically, the G32-command is utilized by most CAM systems. Using the G32-command when a thread is to be programmed manually can be labor-intensive and cumbersome.

Here is a sample program showing several threading passes for a leading edge-cutting cycle.

The thread is a 1-12. Normal thread angels are 29 degrees, and a safe approach distance is 4 threads. With this information we can calculate the Z start at about .4 and the incremental Z shift to .0047 by using the formula below.

\[ \text{TAN } 29 \times \text{depth of pass} = \text{Z distance} \]

Calculate the difference between the X diameters block N110 and N150 (0.9812 - 0.9461) / 2 = 0.0085. Multiply this radius value by the tangent of 29 degrees (0.0085 x TAN 29 = 0.0047). This is the incremental distance in Z between block N100 and N140 (0.400 - 0.0047 = 0.3953).

By using G32 you can control the exact depth and distance of each machining pass. Even if a CAD/CAM system provides the code, with a few simple calculations you can better understand the program and the machining process.

FANUC-controls offer “canned threading cycles” that simplify programming of threads. For example, the use of threading cycles G92 or G76 substantially reduces programming time and simplifies program editing on the shop floor. Please refer to the section covering the canned cycles in this manual.
9.8 Tapping

In theory, the G32 thread-cutting function can be applied for tapping when the optional canned cycles for tapping (G84 and M29-rigid tapping option covered in a different section in this manual) are not available.

When using the G32 function for tapping a floating tap holder is required that provides freedom of movement to the tap in axial direction. Spindle speed must be kept below 300 RPM to prevent excessive coasting.

**Tapping Example:** 3/8-16 UN, 0.625 deep. Material: Steel 1018

<table>
<thead>
<tr>
<th>Program Text</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>O4513(TAPPING 3/8-16 UN)</td>
<td>Get tool #5 and offset #5</td>
</tr>
<tr>
<td>T0505</td>
<td>Spindle speed 250 RPM, CW</td>
</tr>
<tr>
<td>S250 M3 P11</td>
<td>Tool approach &amp; Coolant ON</td>
</tr>
<tr>
<td>G0 Z0.15 M8</td>
<td>Tool at center of spindle</td>
</tr>
<tr>
<td>X0</td>
<td>Tapping 0.625 deep</td>
</tr>
<tr>
<td>G32 Z-.625 F0.0625</td>
<td>Reverse the spindle</td>
</tr>
<tr>
<td>M4 P11</td>
<td>Retract tap clear from hole</td>
</tr>
<tr>
<td>G0 X5.0</td>
<td>Retract tool</td>
</tr>
<tr>
<td>Z6.0 M9</td>
<td>M30=End of program</td>
</tr>
</tbody>
</table>

NOTE: This program must not be run in single block mode – tap will break at the bottom of the hole when spindle keeps running forward.

10 Reference point return functions

10.1 G28- Reference Point Return (Rapid traverse)

The reference point, also called “Home Position” represents a fixed position in the machine coordinate system that is located near the travel limits of X and Z in the positive direction.

During manual zero return-mode the reference point is established by the system electronically. Switches attached to the X and Z-axis slides send signals to the control when the slides enter the area near the “plus limit” of travel. When the signal is received the Reference point is established. At this point all of the coordinate systems are preset by the control with dependable accuracy of within 0.0001” or 0.001mm.

The reference point can be used as a “Safe-position” or as a tool-exchange point.

- The program command: G28 U0 W0 returns the X and Z-axis from any point in the work coordinate system directly to the reference point, in rapid traverse mode, as shown in the sketch, below.
- The program command: G28 X7.0 Z1.0 returns the X and Z-axis from any point in the work coordinate system to a specified intermediate point.
See point “B” shown in the sketch below. Subsequently the move to the reference point is done in rapid traverse mode.

- Caution must be used with the G28 X__ Z__ (absolute command). The point X, Z, as specified must be clear of the work piece.

10.2 G30 - 2nd Reference Point Return (Rapid traverse)

A second reference point can be set by system parameter #1241. A metric distance measured from the machine origin that specifies the location of the second reference point for X and Z is entered at this parameter. Please refer to section 12.4 for setting the second reference point before using the G30 command in your programs.

The 2nd reference point offers an advantage in that it can be set at any desired point in the machine coordinate system. Once set the position is always at the same location. It is not influenced by tool offsets or by changes in the work coordinate system.

The 2nd reference point can be used as a “Safe-position” or as a tool-exchange point in same way as the machine reference point.

- The program command: G30 U0 W0 returns the X and Z-axis from any point in the work coordinate system directly to the 2nd reference point, in rapid traverse mode, as shown in the sketch, below.

- The program command: G30 X6.0 Z0 returns the X and Z-axis from any point in the work coordinate system to a specified intermediate point in the work coordinate system at first. (See point “B” shown in the sketch below). Subsequently, the turret is moved from that point to the 2nd reference point. Both moves are done in rapid traverse mode.
Caution must be used with the G28 X__ Z__ (absolute command). The specified point X, Z, must be clear of the work piece, without fail.

11 Standard Program Format

O1234;

LETTER O FOLLOWED BY A 4 DIGIT PROGRAM NUMBER

G50 S-----;

SETS A MAXIMUM ALLOWABLE CHUCK RPM IN G96 MODE

N100 T0101 M8;

N100 = FIRST CUTTING SEQUENCE
N200 = SECOND CUTTING SEQUENCE
N300 = THIRD CUTTING SEQUENCE

G40 M42;

G40 = CUTTER COMP CANCEL
M41 = FIRST GEAR
M42 = SECOND GEAR
M43 = THIRD GEAR
M44 = FOURTH GEAR

NOTE: M-CODES M41-M44 NOT TO BE USED WHEN MACHINE HAS NO GEARBOX

G96/97 S_____ M3/M4 P11;

G96 = CONSTANT SURFACE FEET
G97 = CONSTANT R.P.M.
S_____ = VALUE OF G96/97
M3 P11= SPINDLE FORWARD
M4 P11= SPINDLE REVERSE

G00 X_____ Z_____ G41/42;
RAPID UP TO PART AND ADD CUTTER COMP.

MACHINING INSTRUCTIONS

G00 G40 X_____ Z_____ ;
RAPID BACK TO THE TOOL CHANGE POSITION CANCELS THE CUTTER COMP.

M1;
OPTIONAL STOP

M30;
END OF PROGRAM,REWIND TO BEGINNING
12 Sub Programs

Programs can be created for various types of operations or routines that can be used repetitively. For example: Sub programs for operations such as bar pulling or bar feeding, repetitive grooving, contouring or hole drilling routines, etc. can be stored in the NC-memory. Whenever the need arises, a sub program can be conveniently called for execution.

The format of a sub program is no different from a normal NC program, except in that it ends with an **M99-command** instead of the M30-command at the bottom of the program.

12.1 Sub Program Call

A sub program can be called or activated from any active program or from MDI mode by the following command:

**M98 P** (Call any program number stored in memory that ends with M99)

*M98 = Call or get a subprogram, P = Program number.*

At the M98 P__ command, processing of the current program is halted and the sub program is processed, immediately. Upon completed execution of the sub program the **M99** command returns processing back to the main program, resuming operation just at the line below the M98 P2 command, in the case as shown below.

When a program number is called that does not exist in the memory, the alarm: “NUMBER NOT FOUND” occurs.
12.2 Sub program Repetition

When a routine needs to be repeated several times consecutively, the letter “L” specifies the repetitive count. When L is omitted, the sub program is executed once only.

For example:

**M98 P1234 L5**  (L5= Repeat program # 1234, 5 times)

Some older controls such as FANUC 0T use the following repetition format:

**M98 P0051234**  (P0051234 = repeat program # 1234, 5 times)

Please note that the first three digits specify the repetitive count, while the last four digits specify the program number.

12.3 Nesting of sub programs

A sub program can be called or activated from other sub programs, up to four levels deep. This is called “Nesting”.

Please review the table shown below:

<table>
<thead>
<tr>
<th>0100 (MAIN PROGRAM)</th>
<th>O1 (SUB-1)</th>
<th>O2 (SUB-2)</th>
<th>O3 (SUB-3)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>M98 P2</td>
<td>M98 P3</td>
<td>M98 P4</td>
</tr>
<tr>
<td></td>
<td>M30</td>
<td>M99</td>
<td>M99</td>
</tr>
</tbody>
</table>

In the case shown above, program 100 calls sub program 1 at first. Next, program 1 calls program 2, then 2 calls 3.

The M99 command on each sub program returns processing back to the program that called the sub program. Processing of the program that called the sub resumes at the line just below the M98 P__ command.
13 Simple Canned Cycles for turning (G90)

The G90 canned cycles perform a box pattern consisting of in-feed, retract and returning the tool back to the initial start position by specifying one block of information only.

13.1 G90 Canned Turning and Boring

G90 is a straight box turning cycle that will permit cutting along the Z axis, the syntax is as shown:

G90 X(U) Z(W) F

![Diagram showing the G90 cycle]

X = X axis endpoint (order point) coordinate (absolute)
(U) = X axis incremental distance, start point to order point
Z = Z axis endpoint (order point) coordinate (absolute)
(W) = Z axis incremental distance, start point to order point
F = feed rate

The tool must first be positioned to the start point of the cycle, after that the G90 command will instruct the correct tool path & feed rate.

Tool path of G90 cycle:

1) Rapid traverse-move to the finish diameter (X-axis)
2) Feed to finish point on Z-axis
3) Feed out to start point diameter
4) Rapid to Z-axis start point
Notes on using G90 canned cycles for turning and boring

<table>
<thead>
<tr>
<th></th>
<th>O. D. Turning</th>
<th>I. D. Turning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Positioning, G00 to</td>
<td>X axis .2&quot; larger than stock diameter</td>
<td>X axis .2&quot; smaller than bore diameter</td>
</tr>
<tr>
<td>start point</td>
<td>Z axis in front of Work piece by .1000&quot;</td>
<td></td>
</tr>
<tr>
<td>G90 turning/boring</td>
<td>Cutting diameter is smaller than starting</td>
<td>Cutting diameter is larger than starting</td>
</tr>
<tr>
<td>cycle</td>
<td>diameter</td>
<td>diameter</td>
</tr>
</tbody>
</table>

**OD Turning With G90**

- G0 X4.1 Z.1
- G90 X3.6 Z-2. F.015
- X3.2
- X2.8
- X2.4
- G0 G40 X10. Z3. T100
- M1

**ID Boring With G90**

- G0 X2.2 Z.1
- G90 X2.6 Z-8 F.015
- X3.
- X3.4
- X3.6
- X4.
- G0 G40 X10. Z4. T200
- M1
It is possible to use the G90 command and vary your endpoint of the Z-axis.

The cycle time can be reduced by commanding a G0 with the X axis 0.05 larger than the cutting diameter. This will let the tool rapid back to its starting position in Z.

G0 X4.1 Z1
G90 X3.8 Z-1.4 F.015
X3.2
X2.8 Z-0.8
X2.4
G0 G40 X1.0 Z3 T100
M1
G0 X3.85
G90 X3.6 Z-1.4 F.015
G0 X3.65
G90 X3.4 Z-1.4 F.015
G0 X3.45
G90 X3.2 Z-1.4 F.015
G0 X3.25
G90 X3. Z-1.4 F.008
G0 G40 X12. Z8.
M1

Here the X axis is commanded to a point .050 larger than it's start diameter and will rapid back it's starting position.

Please note that the G0 will cancel the G90 making it necessary to command G90 each time as shown
13.2 G90 Canned Cycles for Taper Turning and Boring

Taper cutting can be specified using the G90 cycle by the following syntax:

G90 X__ (or U) Z__ (or W) R__ F__;

In the above example the new variable is R, this is used to specify the direction and amount of taper, the taper is specified radial as the difference in diameter from front of the taper to back of the taper. This value is signed + or - depending on if the taper increases or decreases in diameter.
13.3 G94 Canned Facing

G94 is a box turning cycle that will permit the programmer to execute facing cuts on the part. The syntax is as follows:

G94 X(U) Z(W) F

X       X coordinate of order point relative to X0
(U) Incremental dimension of order point relative to the start position on X axis
Z       Z coordinate of order point relative to Z0
(W) Incremental dimension of order point relative to the start position on Z axis

The tool must first be positioned to the start point of the cutting cycle then G94 should be programmed.

Tool path of G94 cycle

1) Rapid to order point on X & Z axis
2) Feed to order point diameter
3) Feed out to order point of Z-axis
4) Rapid back to start point diameter
### Notes on using G94 facing

<table>
<thead>
<tr>
<th>Positioning - G0 To Start Point</th>
<th>X axis, approximately .2&quot; larger than the work piece</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Z axis, approximately .1&quot; in front of the work piece</td>
</tr>
<tr>
<td>Facing cycle, G94</td>
<td>Order point at smaller diameter than at start point</td>
</tr>
</tbody>
</table>

![Diagram of G94 Example](image)

G94 Example

```plaintext
G0 X3.9 Z1
G94 X6.0 Z -1.67 F0.15
Z -3.34
Z -500
G0 G40 X10. Z5. T100
M1
```
13.4 G94 Canned Cycles for Facing on a Taper

The G94 can be programmed to execute a taper cutting action, the syntax for doing this is as follows:

G94 X(U) Z(W) R F

In the above example the new variable is R, this is used to specify the direction and amount of taper, the taper is specified as the difference in Z-axis position from the top of the taper to the bottom of the taper. This value is signed + or - depending on if the taper increases or decreases the its depth in the Z axis.
14 Multiple Repetitive Cycles

Multiple repetitive cycles allow the programmer to write programs for complex shapes while keeping the number of program lines down to the absolute minimum. The programmer will typically write a line that contains various cutting parameters and after that will write the machine code that specifies the finished shape of the part. The control will automatically guide the machine in its various movements from the outside of the material to the finished shape performing repetitive cutting until the finished shape is complete.

14.1 G70 Finishing

This cycle is used after using one of the multiple repetitive roughing cycles (G71, G72 and G73). The G70 command will allow the contour to be finish turned with the stock allowance of the X & Z-axis being machined. The F & S functions specified in the contour description are active and will be used. The syntax is as follows:

G70 P(NS) Q(NF)

P is the start of the contour
Q is the end of the contour

Note that the finishing tool must be positioned to start cutting at the same start point as the roughing tool.
14.2 G71 Turning – Boring Roughing Cycle

G71 permits the rough machining of a contour along the Z-axis from a solid blank of material leaving a allowance of stock on the X & Z-axis to be finish machined afterwards. The syntax is as follows:

\[ \text{G71 } U \ R \]
\[ \text{G71 } P \ Q \ U \ W \ F \]

- \( U \) = the depth of cut for each roughing pass, this is to be designated radial and without the use of a decimal point. \( .125" = 1250, .250" = 2500. \)
- \( R \) = is the size of the 45 degree pullout during each roughing pass.
- \( P \) = the sequence number for the start of the program contour.
- \( Q \) = the sequence number for the end of the program contour.
- \( U \) = will allow the programmer to specify the amount of stock left on the X-axis for finishing this is to be specified radial. This value must be signed negative (\(-\)) when doing ID work.
- \( W \) = will allow the programmer to specify the amount of stock left on the Z-axis for finishing.
- \( F \) = roughing feed rate

Note that the shape of the part must increase or decrease in diameter and move from right to left. If the part goes from a larger to smaller and back to a larger diameter you must have the Type 2 option. As of June 98 standard.

The finishing feed rates should be inserted in the finish part description. This will allow you to change feed rates according to the surface finish requirements.
Tool path of G71 Cycle

1) Rapid from start point of tool towards the diameter specified in P by the stated depth of cut.
2) Feed parallel to the spindle axis to a point in the programmed contour minus the value W.
3) Retract the tool at a 45-degree angle to clear the tool out of the cut, the amount of retraction is specified by parameter 5133.
4) Rapid back to the start point position in the Z-axis.
5) Positioning at new depth of cut.
6) Feed parallel to the spindle center axis to a point in the programmed contour minus the value W.
7) Retract the tool at a 45-degree angle to clear the tool out of the cut, the amount of retraction is specified by parameter 5133.
8) The above process is repeated until such time that the entire programmed contour has been rough turned.
9) Feed the tool over the contour profile leaving the material for finishing.
10) Return back to the start point of the cycle.

Finishing allowance U & W

The finishing allowance U will be signed as a negative (-) integer when ID work is performed.
Notes on using G71 turning - boring

<table>
<thead>
<tr>
<th>Rapid Positioning G00</th>
<th>OD Turning</th>
<th>ID Boring</th>
</tr>
</thead>
<tbody>
<tr>
<td>X axis to the largest diameter to be turned</td>
<td>X axis to the smallest diameter to be turned</td>
<td></td>
</tr>
<tr>
<td>Z axis .1&quot; in front of Work piece Z0</td>
<td>Z axis .1&quot; in front of Work piece Z0</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>First Line Of Contour Description</th>
<th>Rapid move in X to the smallest diameter of the contour</th>
<th>Rapid move in X to the largest diameter of the contour</th>
</tr>
</thead>
<tbody>
<tr>
<td>Command finishing SFM</td>
<td>Command finishing SFM</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Second Line Of Contour Description</th>
<th>Feed towards chuck, G1, G2, G3</th>
<th>Feed towards chuck, G1, G2, G3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Command finishing FPR</td>
<td>Command finishing FPR</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>During Contour Description</th>
<th>X axis must not decrease in diameter</th>
<th>X axis must not increase in diameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z axis motion must be towards the chuck</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>All F &amp; S functions in are ignored during G71</th>
<th>the blocks from P to Q execution</th>
</tr>
</thead>
</table>

| All linear & circular rounding & chamfering describe the contour | interpolation, corner may be used to |

| 58 |
G73 DD Example

N100 T101 M8
G40
G96 S900 M3 P11
G0 G42 X4.5 Z1
G73 U.5 W.2 R.05
G73 P101 Q102 U.02 W.005 F.012
N101 G0 X2. S1200 P11
G1 Z-5 F.008
X3. Z-1.
Z-2.
N102 X4.

Finishing With Different Tool

N200 T303 M8
M3 G40 P11
G0 G42 X4.5 Z1
G70 P101 Q102
G0 G40 X10. Z5. T100
M1

Finishing With Same Tool

G70 P101 Q102
G0 G40 X10 Z5. T100
M1
14.3 G72 Facing

G72 permits the rough machining of a contour along the X-axis from a solid blank of material leaving a allowance of stock on the X & Z-axis to be finish machined afterwards. The syntax is as follows:

**G72 W R**

**G72 P Q U W F**

W = the depth of cut for each roughing pass, this is to be designated radial and without the use of a decimal point. .125" = 1250, .250" = 2500.

R = is the size of the 45 degree pull out.

P = the sequence number for the start of the program contour.

Q = the sequence number for the end of the program contour.

U = will allow the programmer to specify the amount of stock left on the X-axis for finishing this is to be specified radial. This value must be signed negative (-) when doing ID work.

W = will allow the programmer to specify the amount of stock left on the Z-axis for finishing.

F = roughing feed rate

Note that the shape of the part must decrease or increase in diameter, the control will not pocket.

The finishing feed rates should be inserted in the finish part description. This will allow you to change feed rates according to the surface finish requirements.
Tool path of G72 Cycle

1) Rapid from start point of tool towards the diameter specified by “R” stated depth of cut.
2) Feed perpendicular to the spindle centerline to a point in the programmed contour minus the value W.
3) Retract the tool at a 45-degree angle to clear the tool out of the cut.
4) Rapid back to the start point position in the X-axis.
5) Rapid to the next Z-axis position by the depth of cut.
6) Feed perpendicular to the spindle axis to a point in the programmed contour minus the value W.
7) Retract the tool at a 45-degree angle to clear the tool out of the cut.
8) The above process will be repeated until such time that the entire programmed contour has been rough turned.
9) Feed the tool over the contour profile leaving the material for finishing.
10) Return to the cycle-start point.

Finishing Allowance U & W

The finishing allowance U will be signed as a negative (-) integer when ID work is performed.
## Notes on using G72 turning - boring

<table>
<thead>
<tr>
<th></th>
<th><strong>OD Turning</strong></th>
<th><strong>ID Boring</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Rapid Positioning G00</strong></td>
<td>X axis approximately .2&quot; larger than the stock diameter</td>
<td>X axis approximately .2&quot; smaller than the stock diameter</td>
</tr>
<tr>
<td></td>
<td>Z axis .1&quot; in front of Work piece Z0</td>
<td>Z axis .1&quot; in front of Work piece Z0</td>
</tr>
<tr>
<td><strong>First Line Of Contour Description</strong></td>
<td>Rapid move in Z to the furthest point on the Z axis</td>
<td>Rapid move in Z to the furthest point on the Z axis</td>
</tr>
<tr>
<td></td>
<td>Command finishing SFM</td>
<td>Command finishing SFM</td>
</tr>
<tr>
<td><strong>Second Line Of Contour Description</strong></td>
<td>Feed towards spindle centerline</td>
<td>Feed towards spindle centerline</td>
</tr>
<tr>
<td></td>
<td>Command finishing FPR</td>
<td>Command finishing FPR</td>
</tr>
<tr>
<td><strong>During Contour Description</strong></td>
<td>X axis must not increase in diameter</td>
<td>X axis must not decrease in diameter</td>
</tr>
<tr>
<td></td>
<td>Z axis motion must be away from the chuck</td>
<td></td>
</tr>
<tr>
<td></td>
<td>All F &amp; S functions in are ignored during G71 execution</td>
<td>The blocks from P to Q execution</td>
</tr>
<tr>
<td></td>
<td>All linear &amp; circular rounding &amp; chamfering describe the contour</td>
<td>Interpolation, corner may be used to</td>
</tr>
</tbody>
</table>
G72 OD Example

N100 T010 M0
G96 S450 M3 P11
G0 G41 X37 Z.1
G72 W1000
G72 P101 Q102 U02 W005 F012
N101 G0 Z-7 S900 P11
G1 X3.5 F007
X2.2 Z-55
Z-4
X.9
Z-2
X.5
N102 Z0

Finishing With Different Tool
N200 T303 M0
M3 P11
G40
G0 G41 X37 Z.1
G70 P101 Q102
G0 G40 X10. Z10. T300
M1

Finishing With The Same Tool
G70 P101 Q102
G0 G40 X10. Z10. T100
M1
14.4 G73 Turning - Boring, Pattern Repeating

The G73 cycle permits the removal of stock in a fixed pattern cycle leaving a specified amount of stock for a finish pass. This is most often used with a casting or forging. The contour will be generated in a number of passes determined by the programmer. The syntax for this command is as follows:

**G73 U W R**
**G73 P Q U W F**

- **U** (first line) = the thickness of stock to be machined on the X-axis.
- **W** (first line) = the thickness of stock to be machined on the Z-axis.
- **R** = (first line) = the number of roughing passes.
- **P** = sequence number for the beginning of the contour.
- **Q** = sequence number for the end of the contour.
- **U** = direction and radial amount of finish allowance on the X-axis.
- **W** = direction and amount of finish allowance on Z-axis.
- **F** = roughing feed rate.
# Tool path of G73 Cycle

<table>
<thead>
<tr>
<th></th>
<th>OD Turning</th>
<th>ID Boring</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Rapid Positioning G00</strong></td>
<td>X-axis to the largest diameter to be turned</td>
<td>X-axis to the smallest diameter to be turned</td>
</tr>
<tr>
<td></td>
<td>Z-axis .1&quot; in front of Work piece Z0</td>
<td>Z-axis .1&quot; in front of Work piece Z0</td>
</tr>
<tr>
<td><strong>First Line Of Contour Description</strong></td>
<td>Rapid move in X to the smallest diameter to be turned Command finishing SFM</td>
<td>Rapid move in X to the largest diameter to be turned Command finishing SFM</td>
</tr>
<tr>
<td><strong>Second Line Of Contour Description</strong></td>
<td>Feed towards spindle centerline Command finishing FPR</td>
<td>Feed towards spindle centerline Command finishing FPR</td>
</tr>
<tr>
<td><strong>During Contour Description</strong></td>
<td>X-axis must not decrease in diameter</td>
<td>X axis must not increase in diameter</td>
</tr>
<tr>
<td></td>
<td>Z-axis motion must be towards the chuck.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>All F &amp; S functions in the blocks from P to Q are ignored during G73 execution</td>
<td></td>
</tr>
<tr>
<td></td>
<td>All linear &amp; circular interpolation, corner rounding &amp; chamfering may be used to describe the contour</td>
<td></td>
</tr>
</tbody>
</table>
G73 OD Example

N100 T101 M8
G40
G96 S900 M3 P11
G0 G42 X4.5 Z.1
G73 U.5 W.2 R.05
G73 P101 Q102 U.02 W.005 F.012
N101 G0 X2. S1200 P11
G1 Z-5 F.008
X3. Z-1.
Z-2.
N102 X4.

Finishing With Different Tool

N200 T303 M8
M3 G40 P11
G0 G42 X4.5 Z.1
G70 P101 Q102
G0 G40 X10. Z5. T100
M1

Finishing With Same Tool

G70 P101 Q102
G0 G40 X10 Z5. T100
M1
G74 Peck Drilling & Face Grooving (trepanning) On The Z Axis

The G74 command can be used both as a peck drilling cycle (to break chips) and as a face grooving cycle (to "pocket out" a groove area larger than the groove tool).

The syntax for peck drilling is as follows:

**G74 R**  
**G74 Z Q F**

R (first line) = retraction amount after each peck, no decimal, this setting will over ride parameter #5139  
Z = the total depth to be drilled on the Z axis  
Q = the length of each peck, no decimal point  
F = the feed rate of the drill

Tool path of G74 peck drilling cycle:

1 - Rapid to X & Z axis order point, typically X0, Z.2  
2 - Feed into work piece at rate "F", to the depth specified by "Q"  
3 - Rapid back by amount "R"  
4 - Feed back into work piece  
5 - Continue until dimension "Z" is achieved

Example of G74 peck drilling:

N100 T202 M8  
M42 (as needed)  
G97 S1400 M3  
G0 X0 Z.2  
G74 R500 (this first line is optional)  
G74 Z-2. Q2000 F.007  
G0 G40 X10. Z10. T200  
M30
The syntax for G74 trepanning is as follows:

**G74 R**

**G74 X Z P**

R (first line) = retraction amount after each peck, no decimal, this setting will over ride parameter #5139

X = final diameter (note 1)

Z = depth of groove

P = step over amount, no decimal point

Q = depth of each peck, no decimal point

Example:

G00 X5.1 Z.1

G74 X3.95 Z-.2 P1100

G0 G40 X9. Z5. T900

M1

Note 1: in order to obtain the X axis coordinate of this point you must add the desired diameter to the tool thickness multiplied by two.
Example #2

G0 X3.5 Z.1
G74 2.3 Z-.5 P3000 Q1000
G0 G40 X(. Z5. T900
M1
14.6 *G75 Peck Grooving on the X Axis*

The G75 command can be used both as a peck drilling cycle (to break chips) and as a face grooving cycle (to "pocket out" a groove area larger than the groove tool).

The syntax for *peck grooving* is as follows:

**G75 X Z P Q F**

X = bottom of groove dimension (diameter)  
Z = length of groove in Z axis from Z0  
P = depth of each peck  
Q = step over amount on the Z axis  
F = feed rate

**Tool-path of G75 Cycle**

1- rapid tool over material to be grooved, clear part by .1 radial in X.  
   (before G75)  
2- feed tool into material down to its programmed diameter in a pecking  
   motion described by P.  
3- rapid out to X axis start point.  
4- shift tool by value “Q”.  
5- feed tool into material down to its programmed diameter in a pecking  
   motion described by P.  
6- repeat above procedures until Z length of groove is obtained.  
7- rapid back to start point.

**Example #1**

G00 X2.9 Z-.525 (note 1)  
G75 X2.1 Z-.9 P.1 Q.110 F.002  
G0 G40 X8. Z6. T900  
M1

Note 1: Z-.525 = .4 + .125
Example #2

G00 X2.9 Z-.625
G75 X2.2 Z-2.175 P.1 Q.45 F.002 (note 1)
G0 G40 X7. Z6. T900
M1

Note 1: Z-.625 = .5 + .125
15 Thread Cutting Cycles

By programming a single point tool to feed axially over the same point again and again a thread will be cut. Three thread cutting cycles are provided: G32, G92 and G76. When these are used, each tool path will start out at the same point. Threading must be done in G97 mode only.

G32- Each axial pass requires the input of four blocks of data.

G92- Each axial pass requires the input of one block of data.

G76- One block of data is required to cut the whole thread, automatic in-feed and compound cutting are provided.

15.1 Thread Cutting Limitations

Due to the response delay in the servo system there is a limit to how fast the threading tool can be programmed to move. This limit is on the maximum allowable RPM with respect to the pitch of the thread.

The following formula will apply:

\[ \text{RPM} \times \text{PITCH} = <160 \]

The above means that the feed rate must not exceed 160 inches per minute.

The maximum allowable RPM in threading can be calculated as follows:

\[ \frac{160 \text{ IPM}}{\text{PITCH}} \]
15.2 Imperfect Thread Calculation

When threading it is important to take into consideration the distance needed for the acceleration and deceleration of the cutting tool while it is in the work piece.

The cutting tool should be positioned far enough in front of the start of the thread to allow enough distance for acceleration before it enters the material (d₂). At the end of the thread, there should be enough distance allowed for deceleration of the tool (d₁).

\[
\begin{align*}
  d₂ &= \frac{\text{RPM} \times \text{Pitch}}{1800} \\
  d₁ &= d₂ \times 3.605
\end{align*}
\]

Notes on threading:
1) Do not use G96, use G97
2) Do not exceed the maximum allowable RPM in threading
3) Calculate the Imperfect Thread Portion and position tool accordingly.
4) Feed rate override is held at 100% during the threading operation.
15.3 G76 Thread Cutting, Multiple Repetitive

The G76 command is a two-line call out the same as all multiple repetitive cycles. However by setting certain parameters you can eliminate the first line. However the program will then not dictate these values the parameters will. If you need to change these values for each part you are better served to put them in the program. The program overrides the parameters.

The first line is as follows:

G76 P (m) (r) (a) Q (d min) R (d)

P is a six digit character, two digits each for m, r, and a
(m) = number of finishing passes
(r) = chamfering amount
(a) = included angle of the tool tip
(d min) = sets the minimum cutting depth
(d) = finishing allowance.

Standard values for each modifier m, r, a, dmin, and d are normally preset, their parameters and initial values are as follows:

<table>
<thead>
<tr>
<th>Modifier</th>
<th>Parameter #</th>
<th>Standard Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>M</td>
<td>5142</td>
<td>1</td>
</tr>
<tr>
<td>R</td>
<td>5130</td>
<td>10</td>
</tr>
<tr>
<td>A</td>
<td>5143</td>
<td>60</td>
</tr>
<tr>
<td>Dmin</td>
<td>5140</td>
<td>20</td>
</tr>
<tr>
<td>D</td>
<td>5141</td>
<td>10</td>
</tr>
</tbody>
</table>

The above parameters are for the 16/18/21 control and are to be set as desired for the thread/machining conditions on hand.
The correct syntax for the second line is as follows:

\[ \text{G76 X Z P Q F} \]

\( X \) = for OD threads, the minor diameter, for ID threads, the major diameter.

\( Z \) = the endpoint of the thread in the Z-axis.

\( P \) = height of a complete thread, radial, no decimal point (note 1)

\( Q \) = depth of first cutting pass, no decimal point (note 1)

\( F \) = feed rate of tool, this is the same as the thread pitch

Calculation of either the number of passes or the depth of the first pass is possible by applying one of the formulas as shown below:

Where \( D \) = the depth of the first pass.

\( P \) = the radial height of a single thread.

\( N \) = Number of passes (minus spring passes).

\[ N = \left( \frac{P}{D} \right)^2 \]

\[ D = P \sqrt{N} \]

Note 1: \( .001 = 10, .01 = 100, .015 = 150 \)
Tool-path of G76 cycle, OD thread:

1- Rapid to major diameter, minus 2*D
2- Thread to Z axis dimension, first pass
3- Rapid out to start point diameter
4- Rapid to start point Z
5- Rapid to diameter for second pass
6- Thread to Z-axis dimension.
7- Sequence is repeated until thread reaches it's programmed depth (minor diameter)
8- Repeat one pass without any additional in feed, this is a spring pass
9- Return to start point of cycle in X & Z.
Example of two line G76

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thread = 2 - 18</td>
<td>Pitch = .0555</td>
</tr>
<tr>
<td>Major Diameter = 2.0</td>
<td>Number of Finish Passes = 3</td>
</tr>
<tr>
<td>Minor Diameter = 1.9302</td>
<td>Chamfer Amount = 0 (pull straight out)</td>
</tr>
<tr>
<td>Radial Height of Thread = .0349</td>
<td></td>
</tr>
</tbody>
</table>

G0 X2.1 Z.2 rapid to clear stock
G76 P030060
G76 X1.903 Z-.8 P349 Q120 F.0555
G0 X9. X5. T500

Notes on using G76 threading cycle:

<table>
<thead>
<tr>
<th>Rapid move to start point</th>
<th>OD Threading</th>
<th>ID Threading</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X axis approx. .1&quot; larger than major thread diameter</td>
<td>X axis approx. .1&quot; smaller than minor thread diameter</td>
</tr>
<tr>
<td></td>
<td>Z axis, in front of thread by at least 2 threads</td>
<td></td>
</tr>
<tr>
<td>G76 Threading Cycle</td>
<td>Order point is smaller than start point</td>
<td>Order point is larger than start point</td>
</tr>
</tbody>
</table>

Example: 2 1/2 – 18TPI
Major Diameter: 2.0"  Radial Thread Depth: .0349"

G0 X2.1 Z.2
G76 X1.9303 Z-.825 P349 Q120 F.0555
G0 G40 X10. Z5. T500


15.4 G76 Thread Cutting, Multiple Repetitive, Taper

By adding address R to the standard G76 line along with a numerical value taper threads can be generated.

G76 X Z R P Q F

**R** = is the difference in diameter from the Start point of the tool at the beginning of the tapered thread to the end point of the tool at the end of the tapered thread. It is preferred to start at least .2 off the beginning of the thread and where possible to continue passed the end of the thread. The distance at the beginning and at the end must be part of the calculation. This value is expressed **radial**. The value is signed depending on the direction of the taper.

All other addresses relative to G76 are in effect.
15.5 **G76 – THREADING CYCLE – TWO LINE FORMAT** *(FS 0,16,18,21T,31i,32i, -FORMAT)*

(Applicable with Fanuc Controls, T series, systems 0,16,18,21,31i,32i. Also: Mitsubishi 500L, 50, 64)

**FIRST COMMAND LINE:** G76 P021060 Q05 R10 (See detailed explanation, below)

P 02 10 60 Specify “P”, followed by a six digit number. Translation is shown, here:

- **02** = Number of finish passes at the bottom of the thread (02 means 2 Passes) (Sets PAR 5142– see note 1)
- **10** = Chamfer-width or pullout-width at the “Z” end position of the thread. Chamfer size is expressed in 1/10th fractions of the lead. 10: means the chamfer-width equals one lead. 05: means the chamfer width equals ½ of lead. 00: means no chamfer. See note 3, below. (Sets PAR 5130)
- **60** = The included angle between the thread flanks. ½ of the angle as specified is applied for the in-feed angle. Normally, 60° is used for standard threads. Other angles, such as: 80°, 60°, 55°, 30°, 29° or 0° can be specified (Sets PAR 5143)

Q05 = Minimum cutting depth (depth of cut becomes progressively smaller, after each pass. Remainder, smaller than “Q” is discarded (Sets PAR 5140).

R10 = Material allowance for finishing passes at bottom of thread (Sets PAR 5141)

**SECOND COMMAND LINE:** G76 X__ Z__ P100 Q200 R300 F0.038461 (See detailed explanation, below)

X = Diameter of the thread. For an external Thread – specify the minor diameter.
   For an internal Thread – specify the major diameter. In case of taper threads, specify the diameter at the opposite end from the start point.

Z = End position of the thread.

P = Height of the thread. Calculation: Major diameter minus minor diameter, divided by 2 (Radius value, without decimal point) See note 2, below.

Q = Depth of the first cut. If “P” and “Q” are the same, then the thread is `cut in a single pass. (Radius value without decimal point) See note 2, below

R = Taper. (Radial height difference of taper slope, per side)
   Specify a negative value for OD taper thread.
   Specify a positive value for ID taper thread).

F = Lead: distance between two threads. (1 divided by the pitch), six digits allowed after the decimal point.

**NOTES:**
1.) Upon execution of the G76-cycle all data contained on the first G76-command line is automatically stored in the parameter tables.

2.) Values for “P” and “Q” to be specified without decimal point for all Fanuc Controls. For example:

\[
0.0001"=1, \ 0.001"=10 \quad 0.01"=100 \quad 0.1"=1000 \quad 1.0"=10000
\]

3.) Specifying a chamfer (pullout distance) reduces possible damage to the last thread lead near the Z-end position.
G76 – THREADING CYCLE – TWO LINE FORMAT (FS 0.16, 18, 21T-FORMAT)
(Applicable with Fanuc Controls, T series, systems 0, 16, 18, 21. Also: Mitsubishi 500L, 50, 64)

FIRST LINE:
G76 P02 10 60 Q05 R10

Material allowance for finishing passes at bottom of thread. (PAR 5141)

Minimum cutting depth (depth of cut becomes progressively smaller, after each pass. Fractions smaller than “Q” is discarded (PAR 5140).

Included angle of thread in degrees (60 for std. thread). (PAR 5143)

Chamfer amount at pull out end of thread, expressed by lead times 10. (10 = chamfer size equal to one lead, 05= chamfer size equal to ½ of lead). (No chamfer = 00). (PAR 5130)

Number of finish passes at bottom of thread (02 = 2 Passes). (PAR 5142)

SECOND LINE:
G76 X-- Z-- P100 Q200 R300 F0.038461

Lead (1 divided by pitch), six digits allowed after the decimal point.

Taper (Radial height difference per side) Specify a negative value for OD taper thread, Specify a positive value for ID taper thread.

Depth of the first cut. (Radius value).

Height of the thread, (Radius value).

Z - end position of the threaded (W) In case of incremental command

External Thread – specify the minor diameter
Internal Thread - specify the major diameter

NOTES:
1.) Upon execution the data of the first G76-command line, data is automatically stored in the parameter tables as indicated.

2.) Values for “P” and “Q” to be specified without decimal point for all Fanuc Controls. (For example, 0.0001=1, 0.001=10, 0.01=100, 0.1=1000)
G76 – THREADING CYCLE - SINGLE LINE FORMAT - (FS 15T-FORMAT)
(Applicable with Fanuc Controls, T series, systems 10, 11, 12 AND 15T)

This format can also be used with Fanuc Controls, T series, systems 0, 16,18,21 and 30 series, when the tape format setting option is available. In this case, please display the “SETTING PAGE”, then check the “TAPE-F” -setting. When “TAPE-F” is set = 0, the two-line format is valid (see previous page). When it is set = 1, the single-line format is valid. This setting will affect all G70-series canned cycles, not just the threading.

Cycle Format: G76 X__ Z__ I__ K__ D__ F__ A__ P__ Q__

X = Diameter of the thread. For an external Thread - specify the minor diameter.
   For an internal Thread - specify the major diameter.
   In case of taper threads, specify the diameter at the opposite end from the cutting start point.

Z = End position of the thread.

I = Taper: Radial height difference of taper slope. Calculate the height difference of the slope using the entire Z-axis moving distance, including the Z-clearance at the start of the thread. Specify a negative value for OD taper thread. Specify a positive value for ID taper thread.

K = Height of the thread, radius value. Calculation: Major diameter minus minor diameter, divided by 2.

D = Depth of the first cut (Radius value). If “K” and “D” are the same, then the thread is `cut in a single pass.

F = Lead: distance between two threads. (1 divided by the pitch), six digits allowed after the decimal point.

A = Tool nose angle or angle between thread flanks (Range: 0 to 120 degrees, in 1-degree increments) If “A” is omitted it is regarded as 0, straight in-feed is applied (In case of a V-shaped tool, both edges will cut at the same time)

P = Cutting method:
P1=constant chip load, single edge cutting
P2=constant chip load, zigzag in-feed, alternating cutting edges
P3=constant cut depth, single edge cutting
P4=constant cut depth, zigzag in-feed

Q = Spindle rotation shift angle. Data range is from 0 to plus or minus 360000 (360 degrees = 360000, without decimal point) This function is used for cutting of multiple-Lead threads. For example: in case of a 3-start thread the shift angle is 120 degrees between each thread. Hence, the first thread lead is cut, using Q=0, the second at Q=120000 and the third at Q=240000, where the Z-axis start position remains the same for each thread.
### Example 1: Cutting a 1”-10 UNS -external thread:

<table>
<thead>
<tr>
<th>Action</th>
<th>Program</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Enter modal commands</td>
<td>G0 G18 G40 G97 G99</td>
</tr>
<tr>
<td>2. Enter the tool and tool offset command</td>
<td>T0101</td>
</tr>
</tbody>
</table>
| 3. Enter the Spindle command  
(Always use G97, NEVER G96) | G97 S100 M3 (M4)P11 |
| 4. Turn ON the coolant | M8 |
| 5. Move the tool to the start position of the thread  
For “Z”, allow 125 % of the Lead for start-up clearance away from the thread  
Move “Z” fist, then “X”.  
For “X”, allow 0.05” ~ 0.1” diametrical clearance above the major diameter (OD) | G0 Z0.125  
X1.075 |
| 6. Enter the thread cutting cycle | G76 P020560 Q05 R0  
G76 X0.875 Z-1.0 P625 Q250  
F0.1 |
| 7. Return the tool to the tool exchange point  
Move the “X”-axis first, then “Z”  
Optional stop | G0 X  
G0 Z___  
M1 |
Example 2: Cutting a 1”-10 UNS.internal thread:

<table>
<thead>
<tr>
<th>Action</th>
<th>Program</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Enter modal commands</td>
<td>G0 G18 G40 G97 G99</td>
</tr>
<tr>
<td>2. Enter the tool and tool offset command</td>
<td>T0101</td>
</tr>
<tr>
<td>3. Enter the Spindle command (Always use G97, NEVER G96)</td>
<td>G97 S100 M3 (M4)P11</td>
</tr>
<tr>
<td>4. Turn ON the coolant</td>
<td>M8</td>
</tr>
<tr>
<td>5. Move the tool to the start position of</td>
<td>G0 Z0.125</td>
</tr>
<tr>
<td>the thread</td>
<td>X0.800</td>
</tr>
<tr>
<td>For “Z”, allow 125% of the Lead for</td>
<td></td>
</tr>
<tr>
<td>start-up clearance away from the thread</td>
<td></td>
</tr>
<tr>
<td>Move “Z” fist, then “X”.</td>
<td></td>
</tr>
<tr>
<td>For “X”, allow 0.05” ~ 0.1”</td>
<td></td>
</tr>
<tr>
<td>diametrical clearance below the minor</td>
<td></td>
</tr>
<tr>
<td>diameter (I.D.)</td>
<td></td>
</tr>
<tr>
<td>6. Enter the thread cutting cycle</td>
<td>G76 P020560 Q05 R0</td>
</tr>
<tr>
<td></td>
<td>G76 X1.0 Z-1.0 P500 Q150 F0.1</td>
</tr>
<tr>
<td>7. Move the tool out of the bore, clearing</td>
<td>G0 Z___</td>
</tr>
<tr>
<td>the face</td>
<td></td>
</tr>
<tr>
<td>8. Return the tool to the tool exchange</td>
<td>G0 X___</td>
</tr>
<tr>
<td>point</td>
<td>G0 Z___</td>
</tr>
<tr>
<td>Move the “X”-axis first, then “Z”</td>
<td>M1</td>
</tr>
<tr>
<td>Optional stop</td>
<td></td>
</tr>
</tbody>
</table>

Note: Source for thread dimensions used in the thread cutting cycles shown above: “Machinery’s Handbook” (Twentieth edition).
Multiple start threads are possible in the G76 mode, you have to shift the starting point for the extra threads by 1/n of the pitch.

Example

Cut 5" - 4TPI, 3 start

G0 X5.1 Z.15 rapid to start of thread
G76 X4.9633 Z-1.4 P1534 Q250 F.25 cut first thread
G0 W.0833 shift by 1/n of pitch
G76 X4.9633 Z-1.4 P1534 Q250 F.25 cut second thread
G0 W.0833 shift by 1/n of pitch
G76 X4.9633 Z-1.4 P1534 Q250 F.25 cut third thread
G0 X9. Z5. T900

Procedure:

1- Cut first thread starting in front of thread by at least d₁
2- Shift starting point by 1/n of pitch (this is the starting point of the second thread)
3- Cut second thread
4- Shift starting point by 1/n of pitch (this is the starting point of the third thread)

The shift amount (W) is determined as follows:

\[ W = \left( \frac{1 \text{ inch}}{\# \text{ of starts}} \right) \times \text{Pitch} \]

In this example \( W = (1/3) \times 0.25 \)
\[ W = 0.0833 \]
15.8 G92 Thread Cutting

The G92 command will drive the cutting tool in a "box" pattern.

Straight threads can be cut using the following command:

\[
\text{G92 } X \ Z \ F \ Q
\]

- \(X\) = the diameter that you are cutting the pass at
- \(Z\) = the endpoint of the threading pass
- \(F\) = the feed rate (pitch) of the thread
- \(Q\) = the

The tool must first be positioned to the start point of the cutting cycle. The G92 command will then specify the diameter (order point) to thread at and the pitch.

Tool path of G92 cycle, OD thread:

1 - Tool rapids to cutting diameter (order point)
2 - Thread to Z axis dimension (order point)
3 - Rapid out to start point diameter
4 - **Rapid to start point Z**
Notes on using G92 threading cycle

<table>
<thead>
<tr>
<th>Rapid move to start point</th>
<th>OD Threading</th>
<th>ID Threading</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>X axis approx. .1&quot; larger than major thread diameter</td>
<td>X axis approx. .1&quot; smaller than minor thread diameter</td>
</tr>
<tr>
<td></td>
<td>Z axis, in front of thread by at least 2 threads</td>
<td>Order point is smaller than start point</td>
</tr>
<tr>
<td>G92 Threading Cycle</td>
<td>Order point is smaller than start point</td>
<td>Order point is larger than start point</td>
</tr>
</tbody>
</table>

Example of G92

<table>
<thead>
<tr>
<th>Thread = 2 - 18</th>
<th>Pitch = .0555</th>
</tr>
</thead>
<tbody>
<tr>
<td>Major Diameter = 2.0</td>
<td>Minor Diameter = 1.9302</td>
</tr>
<tr>
<td>Minor Diameter = 1.9302</td>
<td>Radial Thread Height = .0349</td>
</tr>
</tbody>
</table>

G00 X2.1 Z.2          rapid to stock
G92 X1.976 Z-.8 F.0555 1st pass
X1.966 2nd pass
X1.9584 3rd pass
X1.952 4th pass
X1.9464 5th pass
X1.9412 6th pass
X1.9366 7th pass
X1.9322 8th pass
X1.9303 9th pass
X1.9303 spring pass
G0 X9. Z5. T500 tool change position, cancel G92 & offset

G92 is modal, it is not necessary to repeat the command for each block, just change the X-axis order point. When you are finished with this cycle cancel it out with the G0 command.

If the thread does not run into a relief groove it is suggested that chamfering pullout be applied. This will be done at a 45-degree angle and is controlled by parameter #5130. The chamfering distance is set from .1 - 12.7 of pitch. 1 = .1 pitch of chamfer, 10 = 1 pitch of chamfer.
15.9 G92 Thread Cutting, Taper

By adding address R to the standard G92 line along with a numerical value taper threads can be generated.

G92 X Z R F

R = is the deference in diameter from the beginning of the tapered thread to the end of the tapered thread. This is expressed radial. The value is signed depending on the direction of the taper. All other addresses relative to G92 are in effect.
16 Canned Cycles for hole machining (G80 Series)

Canned cycles for hole machining are optional equipment. Most of the older FANUC controls are not equipped with this option.

On a 2-axis turning center, the following canned cycles can be used:
1. G83 (Z-axis Peck Drilling Cycle)
2. G84 (Z-axis Tapping Cycle)

On a turning center equipped with live tools, the following additional canned cycles can be used:
3. G87 (X-axis Peck Drilling Cycle)
4. G88 (X-axis Tapping Cycle)

16.1 G83 Z-axis Peck Drilling Cycle

FORMAT: G83 Z___ Q___ F___ R___ P___

Z = End position at depth of hole
Q = Peck distance, number without decimal point
F = Feed rate
R = Distance from initial point (Z) to R-level
P = Dwell time at bottom of hole, in milliseconds, without decimal point

NOTES:
1. Position the tool at X0 and Z at the desired clearance point first. Then specify the G83-command.
2. When “R” is used, it is to be specified as an increment (positive or negative) from the initial level, not as an absolute position.
3. It is recommended not to use “R” at all.
**EXAMPLE 1: G83 (Z-axis Peck Drilling Cycle)**

<table>
<thead>
<tr>
<th>Program Text</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>O4513 (DRILL DIA .3125 x .75 DEEP)</td>
<td></td>
</tr>
<tr>
<td>T0303</td>
<td>Get tool #3 and offset #3</td>
</tr>
<tr>
<td>G97S750 M3 P11</td>
<td>Spindle speed 750 RPM, CW</td>
</tr>
<tr>
<td>G0 Z0.15 M8</td>
<td>Tool approach &amp; Coolant ON</td>
</tr>
<tr>
<td>X0</td>
<td>Tool at center of spindle</td>
</tr>
<tr>
<td>G99G83 Z-.750 Q2500 F0.005</td>
<td>Peck drilling 0.75 deep, 0.005 IPR Pecking depth 0.25”</td>
</tr>
<tr>
<td>G80 G0 X5.0</td>
<td>Cancel canned cycle, Retract tool</td>
</tr>
<tr>
<td>Z6.0 M9</td>
<td></td>
</tr>
<tr>
<td>M30</td>
<td>M30=End of program</td>
</tr>
</tbody>
</table>

**16.2 G84 Z-axis Tapping Cycle**

**FORMAT:** \( \text{G84 } \text{Z } \text{Q } \text{F } \text{R} \text{ P} \)

- \( \text{Z} \): End position at depth of hole
- \( \text{Q} \): Peck distance, number without decimal point
- \( \text{F} \): Feed rate
- \( \text{R} \): Distance from initial point (Z) to R-level
- \( \text{P} \): Dwell time at bottom of hole, in milliseconds, without decimal point

**EXAMPLE 2: G84 (Z-axis Tapping Cycle)**

<table>
<thead>
<tr>
<th>Program Text</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>O4513 (TAPPING 3/8-16 UN)</td>
<td></td>
</tr>
<tr>
<td>T0404</td>
<td>Get tool #4 and offset #4</td>
</tr>
<tr>
<td>G97S250 M3 P11</td>
<td>Spindle speed 250 RPM, CW</td>
</tr>
<tr>
<td>G0 Z0.15 M8</td>
<td>Tool approach &amp; Coolant ON</td>
</tr>
<tr>
<td>X0</td>
<td>Tool at center of spindle</td>
</tr>
<tr>
<td>G99G842-0.625 F0.0625</td>
<td>Tapping 0.625 deep, 16 TPI</td>
</tr>
<tr>
<td>G80 G0 X5.0</td>
<td>Cancel canned cycle, Retract tool</td>
</tr>
<tr>
<td>Z6.0 M9</td>
<td></td>
</tr>
<tr>
<td>M30</td>
<td>M30=End of program</td>
</tr>
</tbody>
</table>

**EXAMPLE 3: G84 (Z-axis Rigid Tapping)**

*Note: Never pre-start spindle in rigid tapping mode. (M03, M04 etc.)*

<table>
<thead>
<tr>
<th>Program Text</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>O4513 (TAPPING 3/8-16 UN)</td>
<td></td>
</tr>
<tr>
<td>T0404</td>
<td>Get tool #4 and offset #4</td>
</tr>
<tr>
<td>G0 Z0.15 M8</td>
<td>Tool approach &amp; Coolant ON</td>
</tr>
<tr>
<td>X0</td>
<td>Tool at center of spindle</td>
</tr>
<tr>
<td>G97S250 M29 P11</td>
<td>Spindle speed 250 RPM, Rigid Tap</td>
</tr>
<tr>
<td>G99G842-0.625 F0.0625</td>
<td>Tapping 0.625 deep, 16 TPI</td>
</tr>
<tr>
<td>G80 G0 X5.0</td>
<td>Cancel canned cycle, Retract tool</td>
</tr>
<tr>
<td>Z6.0 M9</td>
<td></td>
</tr>
<tr>
<td>M30</td>
<td>M30=End of program</td>
</tr>
</tbody>
</table>

90
G87 X-axis Peck Drilling Cycle with Live Tools

FORMAT: \texttt{G87 X\_\_\_ Q\_\_\_ F\_\_\_ P\_\_\_}

\texttt{X} = End position at depth of hole (X-Diameter)
\texttt{Q} = Peck distance, number without decimal point
\texttt{F} = Feed rate
\texttt{P} = Dwell time at bottom of hole, in milliseconds, without decimal point

NOTES:
1. Position the tool in X and Z at the desired clearance point above the part where the hole is to be drilled or tapped. Note that the working direction is parallel to the X-axis. Use diameter dimensions for X. Then specify the G87 or G88-command.
2. It is recommended not to use “R” at all.

EXAMPLE 1: G87 (X-axis Peck Drilling Cycle)

<table>
<thead>
<tr>
<th>Program Text</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>O4513(DRILL DIA .3125 x .75 DEEP)</td>
<td></td>
</tr>
<tr>
<td>T0303</td>
<td>Get tool #3 and offset #3</td>
</tr>
<tr>
<td>G97 S750 M03 P12</td>
<td>Spindle speed 750 RPM, CW</td>
</tr>
<tr>
<td>G0 Z-1.0 C0 M8</td>
<td>Tool approach &amp; Coolant ON</td>
</tr>
<tr>
<td>X2.1</td>
<td>Tool at clearance point at OD</td>
</tr>
<tr>
<td>G99 G87 X1.0 Q2500 F0.005</td>
<td>Peck drilling 0.75 deep, 0.005 IPR Pecking depth 0.25”</td>
</tr>
<tr>
<td>G80 G0 X5.0</td>
<td>Cancel canned cycle, Retract tool</td>
</tr>
<tr>
<td>Z6.0 M9</td>
<td></td>
</tr>
<tr>
<td>M30</td>
<td>M30=End of program</td>
</tr>
</tbody>
</table>
17 Miscellaneous Settings

17.1 Instructions for Setting the Work-Zero Point on Lathes with Fanuc 18T 21T, Or 30 Series Controls.

Every CNC-lathe has one basic coordinate system that remains fixed and cannot be changed. This is known as the “Machine Coordinate System”. The origin or zero point of the machine coordinate system is normally located at the intersection of the main-spindle center axis (x0) and the spindle flange face (z0).

For NC-programming purposes a “Work-Zero Point” is utilized whose origin is located on the actual work-piece. Modern CNC-lathes are equipped with six different data registers known as “Work Coordinate Systems”. NC-program commands “G54” through “G59” activate one of the six work coordinate systems available. G54 serves as the default coordinate system.

Before executing a NC program the zero point of the work coordinate system to be used must be established. The distance between machine coordinate zero point and work coordinate zero point is called the “Work-Offset”. The actual distance along the z-axis only, between machine zero point and work zero point must be entered at one of the work-offset registers (G54 through G59) as in the picture, below. The x-axis on all work-offset registers must remain zero.
Depending upon programming method applied the work zero point “Z0” may be located at an arbitrary point along the z-axis of a work-piece, while “X0” is always located at the center axis of the spindle. Thus, the X-register for all work offsets must always remain zero.
17.2 Work-offset setting procedure for lathes equipped with Q-setter.

1. Every tool attached to the turret must be touched-off at first, using the automatic tool setter (Q-setter), before setting the work zero point.
2. The raw material for the work-piece to be machined is placed into the chucking device and clamped.
3. The turret is moved to a safe position for indexing. Select a tool that is capable of cutting the face on the right-end of the part to be machined by manually indexing the tool around.
4. Set the mode selector switch to MDI-mode. Press the “Program”-key then press the “MDI software-key”.
5. When the G54 coordinate system is selected and tool #1 is the tool to be used for “Touching-off” key-in “G54 T0101 EOB” (G55 T0707 EOB, if G55 and tool #7) then push “INSERT”. Push “CYCLE START” several times, until the command line disappears from the screen.
6. Move the tool near the face of the part. You may cut the face of the part, if you wish. Or you may position the tool as close as possible to the raw face of the part. When positioning of the tool has been completed press the “OFFSET”-key, then the software-key “WORK”. Now the screen as shown on page 1 above will appear.
7. Move the yellow cursor onto the Z-data field on the “G54” work offset. Press the “Tool Measure-Key” on the operation panel for two seconds.
8. Now, key-in the z-coordinate value at which the tool is positioned at this time. For example, when the tool is positioned at the finished face, key-in “Z 0” then press the “MEASURE”-software key. For example, when the tool is positioned at the raw face, key-in “Z 0.025” if you want to remove 0.025” from the raw face, then press the “MEASURE”-software key.
9. After pressing the “MEASURE”-key the work-offset is entered, automatically. If you are not sure that the setting is right repeat step 7 and 8 again. If the work offset value remains the same the procedure has been executed correctly.

17.3 Work-offset setting procedure for lathes without Q-setter

1. Cutting tools attached to the turret that already have a valid tool offset in the tool offset registers from a job done previously need not to be touched-off anew, when setting the work-offset.
2. The raw material for the work-piece to be machined is placed into the chucking device and clamped.
3. The turret is moved to a safe position for indexing. Select a tool that is capable of cutting the face on the right-end of the part to be machined by manually indexing the tool around.
4. Set the mode selector switch to MDI-mode. Press the “PROGRAM”-key then press the “MDI software-key”.
5. When the G54 coordinate system is selected and tool #1 is the tool to be used for “touching-off” key-in “G54 T0101 EOB” (or G55 T0707 EOB, if G55 and tool #7 is used) then push “INSERT”. Push “CYCLE START” several times, until the command line disappears from the screen.
6. Move the tool near the face of the part. You may cut the face of the part, if you wish. Or you may position the tool as close as possible to the raw face of the part. When positioning of the tool has been
completed press the “OFFSET”-key, then the software-key “WORK”. Now the screen as shown on page 1 above will appear.

7. Move the yellow cursor onto the Z-data field on the “G54”-work offset. Press the “Tool Measure-Key” on the operation panel for two seconds.

8. Now, key-in the Z-coordinate value at which the tool is positioned at this time. For example, when the tool is positioned at the finished face, key-in “Z 0” then press the “MEASURE”-software key. For example, when the tool is positioned at the raw face, key-in “Z 0.025” if you want to remove 0.025” from the raw face, then press the “MEASURE”-software key.

9. After pressing the “MEASURE-key” the work-offset is entered, automatically. If you are not sure that the setting is right repeat step 7 and 8 again. If the work offset value remains the same the procedure has been executed correctly.

17.4 How To Set the 2nd reference point (G30) on the Fanuc 16/18/21-T/ and 30 Series controls

Machine coordinates for the G30 position are stored in parameter #1241. The easiest way to calculate the G30 position is by physically moving the turret to the desired tool exchange point. Please make sure that all tools are clear of the part and the chuck. Then display the position screen and write down the machine coordinates, (not “absolute” or “relative”).

To set the data, proceed as follows:

- Select MDI mode.
- Press the Offset/Setting Function key.
- Choose the “Setting Software-key”.
- Change the Parameter enable key to “1” and press input.
- Press the “System” function key.
- Choose the “Parameter” Software-key.
- Key in “1241”. (Do not press “input”)
- Press the “Number Search” Software-key.

Parameter 1241 will look like this:

<table>
<thead>
<tr>
<th>1241</th>
<th>Ref point #2</th>
<th>X</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Z</td>
</tr>
</tbody>
</table>

Note 1: The parameter data must be set in metric units. If the machine coordinate display is in inch units, then you must calculate the metric equivalent for the G30 position.

Note 2: When the G30-command is used, neither X or Z data on parameter 1241 must be set zero. This would set the 2nd reference point at the spindle centerline and the spindle face. (CRASH!). If in doubt, use the same settings as shown in Parameter 1240. This will set the G30 at the home position.
17.5 *Changing Parameters on 16/18TC and 30 Series Controls*

1- Select mode push button for MDI  
2- Set program protect key to OFF  
3- Press <OFFSET /SETTING> key on panel until <SETTING HANDY> screen is displayed  
4- At Parameter Write Enable, key in <1> and press <input>. Alarm #100 will occur, this is normal  
5- Press the <SYSTEM> key on panel  
6- Press the <Parameter> soft key  
7- Press <OPRT> soft key  
8- Key in the parameter number to be changed and press the <NO SRCH> soft key  
9- Use the arrow right or left keys on the keypad to move to the correct bit or data set to be changed.  
10- Key in value then press <INPUT>  
11- Press the <OFFSET / SETTING> key on panel until the <SETTING HANDY> screen is displayed  
12- At Parameter Write Enable, key in <0> then press <INPUT>  
13- Press the <RESET> key on the panel, clearing the alarm  

**NOTE:**  
Be sure to know the exact consequences of changing a specific parameter setting. After changing the setting of certain types of parameters, rebooting of the system may be required. Normally, the system displays a message when power OFF and rebooting is required.
18 Operator's Control Panel

The machine is controlled by the various keys and switches located on the operation panel.

**MODE PUSHBUTTON SWITCHES** - Use this to select the desired operating mode

- **Edit Mode** - Edit mode is used to enter a new program into memory whether by keyboard or downloading process, edit an existing program or searching for data.
- **Auto Mode** - Auto mode is used to run a program from memory.
- **Tape Mode** - Tape mode is used to run a program from a tape reader.
- **MDI** - MDI mode is used in order to execute commands to the machine without putting them in program form.

**Reference Point Return** - Also known as the machine "home" position. This is a spot the machine must be made to travel to upon power up in order for the machine to know its position. Until this operation is performed you cannot run a program or perform MDI commands.

**Jog** - Permits the turret to be moved in the X & Z-axis using one of the jog-feed direction buttons along with the rapid button.

**Handle X & Z** - Allows you to move the turret in just the X or Z-axis using the manual pulse generator.

**<CYCLE START>** - Starts program execution when mode selector is in AUTO mode or with an active command in MDI mode.

**<FEED HOLD>** - Halts execution of a part program until <CYCLE START> is pressed again.

**<MACHINE READY>** - Must be pushed as last part of power up procedure.

**<EMG. RELEASE>** - If you overrun the stroke limit of either axis, push this and one of the Jog Feed Direction buttons at the same time to bring the turret head back to within its limit.

**<EMERGENCY STOP>** - This button should be pressed to immediately halt the operation of the machine. This button has to be unlocked (by rotating clockwise) prior to pressing <MACHINE READY>

**Handle Feed Dial** - There are three positions. When used in conjunction with the manual pulse generator and in handle mode, you may move the turret in the following increments:

- X1 = .0001
- X10 = .001
- X1 = .01

**Manual Pulse Generator** - Rotate this dial in order to move the turret.

**Rapid Override Dial** - You may select the rate at which the machine travels in rapid traverse. AT the F0 setting the rapid traverse moves at
a relatively slow speed as set by parameter #1421. F25 will move at 25% speed, F50 will move at 50% speed and F100 will enable 100% of the rapid feed rate as set by parameter #1420.

**Feed rate Override Dial** - While the machine is in operation, you may alter the current programmed feed rate. You can stop the feed entirely with 0% or override up to 150% of the programmed feed rate.

**Jog Feed Direction Buttons & <RAPID>** - There are 4 button switches arranged around a <RAPID>, while you have the mode selector switch set to "JOG" you may press any of these keys and move the turret in that key's direction. Each button corresponds with one of the machine's axial movements. By pressing <RAPID> at the same time that you press one of these buttons you make the machine move at the rapid traverse rate.

**Spindle Override Dial** - While the machine is in operation you may elect to alter the spindle speed taking it from 50% to 120%.

**<START>** - When the mode selector switch is in "JOG" or "HANDLE" mode you may manually start the spindle by pressing this button. When no spindle speed command is active the spindle will not rotate.

**<STOP>** - This button will manually stop the spindle.

**Low, High, FWD, RVS, JOG Dial (Spindle Mode Selector)** - This dial controls the direction of the spindle when operating it manually, the Low/High LED's indicate the gearbox position (not all machines have this indicator as not all machines have a gearbox).

**Spindle Load** - This meter shows the percentage of spindle horsepower used during a cut.

**Alarm** - This display shows the alarm codes of the machine as different situations arise.

**M02/30 LED** - This LED will blink upon the control reading M02 or M30 in the program.

**Chuck LED** - This LED will light up when the chuck is clamped. The machine will not normally start a cutting cycle unless this is lit up (See: chucking key switch).

**Single Block Pushbutton** - When turned on will allow the control to execute one block of information at a time.

**Optional Stop Pushbutton** - When turned on will permit the execution of M01.

**Optional Block Pushbutton** - When the "/" character is inserted at the start of a block you may turn this switch on and the control will ignore that block of code.

**Dry Run** - When this Pushbutton is turned ON the rapid traverse speed and feed rate move at jog speed. Dry run must not be used during actual machining because the feed rate will be too high and tools may break.
The feed override switch controls feed velocity in dry run mode. Use DRY RUN for tool path verification only, without actually machining a part.

**<Tool Measure>** - Press this first when setting Tool Offsets and the Work shift Zero (see sections 14 & 15).

**Coolant Pushbutton** - Allows you to control the coolant, in the Manual position the coolant is always on, in Auto the coolant is waiting for M08 & M09, Off turns the coolant off.

**Machine Lock Lamp (LED)** - Machine lock is used for tool path verification, using the graphics display. When M17 has been commanded the Machine Lock status lamp (LED) will light up. When a program is executed in machine lock mode the spindle and coolant will run normally. The turret will index at the tool command. Only the (X, Y, Z, B) axes are locked. The M18 command followed by manual Zero Return of all axes will restore normal operation.

**Program Protect Switch** - When turned to the left and the key is removed editing of programs, changing of geometry offsets, work offsets and other data entry is disabled.

**Chucking Switch** - Selects inward / outward clamping confirmation of the chuck. For OD chucking turn the key to the left. For ID chucking turn the key to the right.

**Door Interlock Switch** - This switch is a safety feature and should not be tampered with (altered). When the switch is set to the left and the door is opened during machining the coolant and turret feed rate will stop, the spindle will continue to rotate.
NC Programming for Turning Centers
For 30 Series Control single path machines
Equipped with
Live Tools & C- Axis
ROTARY AXIS FUNCTIONS

When machining with live tools a rotary-axis allows angular positioning of the work piece between zero and 360 degrees. The CNC system converts one of the lathe spindles into a rotary axis.

**C - Axis**
PUMA Turing centers equipped with a turret and driven tools normally employ a rotary axis, called the C-axis. The main spindle motor drives the rotary axis. A position-encoding device attached to the spindle provides for positioning of the rotary axis at 0.001-degree resolution. Linear interpolation with the rotary axis, together with any other axis is possible. For circular interpolation between a rotary axis and a linear axis, special control functions such as polar coordinate interpolation or cylindrical interpolation is applied. The rotary axis is switched ON or OFF by M-codes, alternating between normal spindle operation and C-axis operation.

**Rotary Axis Mode**
M-code M35 switches the C¹-axis (rotary-axis).ON

**Main Spindle Mode:** M-code M34 switches C¹-axis, OFF

**Reference Return Command:**
- G28 H0, (or G30 H0)

**C-axis positioning Command:**
- G0 C180.000  – Absolute command, degrees
- G0 H180.000  - Incremental command, degrees

Work offsets G54 through G59 or the coordinate system setting command G50 sets the work coordinates for the rotary axis. System parameter 1240 & 1250 sets the reference point (Home position) for the C-axis.

**Linear Interpolation command:**
- G98 G1 C___(H___) F___ (F = degrees of rotation per minute)
- G99 G1 C___(H___) F___ (F = degrees of rotation per tool revolution)

**C-axis locking function**
During machining with live tools, locking of the C-axis can provide improved stability.

**High-pressure clamp M89** (fixed at maximum hydraulic system pressure)
Both, rapid positioning axis interpolation are disabled while M89 is active.

**Unlock command M90**
**Feed Rate Calculation for the Rotary Axis**

The feed rate for a rotary-axis is specified in units of angular velocity, either in degrees per minute or in degrees per tool revolution.

To convert the tangential feed rate on the circumference of a circle that is defined by the radius $R$ from inches per minute (IPM) into degrees per minute ($^\circ$PM), the following formula is applied:

$$F^\circ \text{ per minute} = F \text{ (IPM)} \times \frac{57.296}{R}$$

To convert a feed rate from inches per revolution (IPR) into degrees per tool rotation ($^\circ$/ REV) the formula is the same:

$$F^\circ \text{ per revolution} = F \text{ (IPR)} \times \frac{57.296}{R}$$

The above formulas calculate the feed velocity for moving the rotary axis alone, not together with another axis.

**For example:** Suppose that machining is done on the OD of a 1.5” diameter part, rotating the C-axis only. The tangential feed rate desired is 5” per minute. What is the required feed rate in degrees per minute?

**Answer:** Feed rate required=5 x 57.296 / 0.75=382 degrees per minute
Feed Rate Calculation for Linear Interpolation with Rotary Axis

Caution concerning the feed rate must be applied when linear interpolation between the rotary axis and the Z-axis is done. The tangential feed rate along the tool path becomes high when the arc length of the rotary axis move is relatively short in comparison to the travel distance along the Z-axis. The feed rate must be reduced, accordingly. It can be calculated as shown in the example, below.

Example: Machining is done on the OD of a 1.5” diameter part, rotating the C-axis Angle = 30° while moving the Z-axis minus 1”, at the same time. The desired feed rate along the tool path F = 5”/minute. Calculate the feed rate to be used for the interpolation command: G1G98 H60. W-1.0 F___?

Steps for calculation of the tangential feed rate:

1. Calculate the length of the 30° arc segment on the periphery of a 1.5” diameter circle: Arc length=2Rxπ/360x60=2x0.75x3.14/360x30=0.392”

2. Calculate the length of the tool path: L= Square root of (0.392²+1²)=1.07”

3. Calculate the time it should take for the 1.07” long cut, applying the feed rate of 5” per minute. Time = 60/5x1.07=12.84 seconds.

4. Calculate the feed rate in degrees per minute that is required for a rotation of 30 degrees in12.8 seconds: F=30/12.8*60=141 degrees per minute.

Or apply the following formula, where:

\[ F \text{ ° per minute} = F \text{ (IPM)} \times \frac{A}{L} \]

Feed rate in degrees per minute =5 x 30 / 1.07=141 degrees per minute

Command line for above example: G1G98 H60.0 W-1.0 F141.
SPINDLE MODE AND ROTARY AXIS MODE COMMANDS

For PUMA Lathes, equipped with a C-axis, the program commands as shown below apply. Commands are shown for turning mode and for live tool mode, separately.

Main Spindle Mode (C-Axis Disconnected)
For turning operations on the main spindle, the commands as shown in the table, below are applicable. These commands may be used at the initial program start-up in Turning-Mode or when switching from Live Tool-Mode to Turning-Mode.

<table>
<thead>
<tr>
<th>Command</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>M5, M3 or M4</td>
<td>These commands are normally used for starting or stopping the spindles. The designation of P__ Selects which spindle P11 = Main Spindle, P12 = Live Tool, P13 = Right Spindle</td>
</tr>
<tr>
<td>G0 G18 G40 G80</td>
<td>Use these G-codes at the beginning of any program segment where “Canned cycles” G81 through G88 or cutter compensation G41, G42 is used. G18 (X-Z Plane select, default on power up)</td>
</tr>
<tr>
<td>G99</td>
<td>IPR-feed mode should always be used for turning. (G99-mode is set as default on power up)</td>
</tr>
<tr>
<td>G96 S__</td>
<td>Constant surface speed control command is used for turning only. Not to be used for drilling, tapping, milling or thread cutting.</td>
</tr>
<tr>
<td>G97 S__</td>
<td>Constant (RPM) control command. Use G97 for drilling, tapping, milling or thread cutting. (G97-mode is set as default on power-up).</td>
</tr>
</tbody>
</table>

Rotary axis mode (C-Axis connected)
For Live Tool operations, the commands as shown in the table, below are applicable. These commands may be used at the initial program start-up in Live Tool-Mode or when switching from Turning-Mode to Live Tool Mode.
<table>
<thead>
<tr>
<th><strong>Command</strong></th>
<th><strong>Remarks</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>M05 P12</td>
<td>Live tool spindle rotation-stop command</td>
</tr>
<tr>
<td>G0 G40 G80</td>
<td>Use these G-codes at the beginning of any program segment where “Canned cycles” G81 through G88 or cutter compensation G41, G42 is used.</td>
</tr>
<tr>
<td>M90</td>
<td>C-axis unclamp-command. Use at the beginning of any program segment where C-axis clamp function (M88 or M89) is used.</td>
</tr>
<tr>
<td>G28 H0</td>
<td>C-axis Reference-point-return command. This command should be used always after the C-axis has been newly activated.</td>
</tr>
<tr>
<td>G50 C__</td>
<td>G50 &quot;C “only! No other axis. This may be used to pre-set the C-axis coordinates, at the reference point, if desired.</td>
</tr>
<tr>
<td>G97 S__M3 P12,</td>
<td>Constant (RPM) control command must be used always when C-axis is active. (G97-mode is set as default on power-up). <strong>Note:</strong> The G96 command must never be used in Live Tooling Mode.</td>
</tr>
<tr>
<td>M206</td>
<td>Allows simultaneous spindle rotation of more than one spindle at a time. It will keep the live tool spindle running.</td>
</tr>
<tr>
<td>M3 P12</td>
<td>Live tool spindle-forward rotation command.</td>
</tr>
<tr>
<td>M4 P12</td>
<td>Live tool spindle-reverse rotation command.</td>
</tr>
<tr>
<td>M89</td>
<td>C-axis high pressure clamp. Use only when necessary.</td>
</tr>
<tr>
<td>G99</td>
<td>IPR-feed mode may be used for any live tool operation, except on machines built before 1998. (G99 set default on power-up).</td>
</tr>
<tr>
<td>G98</td>
<td>IPM-feed mode may be used for any live tool operation. Preferably, the IPR (G99) feed mode should be used, if possible. For machines built before 1998, the IPM-feed mode must be applied for Live-Tooling operations.</td>
</tr>
</tbody>
</table>
ANGULAR POSITIONING FUNCTION FOR SPINDLES
Angular positioning function for spindles can be utilized for machining with live tools. Angular positioning is applied typically on the sub spindle for the PUMA MS-series turning centers.

**Spindle orientation**
When the spindle orientation option is provided the command **M19 S0 P11** is used for positioning the main spindle at a preset rotation angle. Spindle orientation is used for applications such as bar pulling of polygon shaped stock, in-feeding of polygon shaped bar material from a bar feeding device, positioning of the chuck for loading of work pieces, etc.

**Parameter Settings related to Spindle Orientation**
Entering data at system parameter 4077 does setting of the orientation reference angle.

**Main Spindle:** #4077 S1
Data range for parameter setting: zero ~ 4096, positive or negative value.
One full rotation (360 degrees)=4096 units. One unit equals 0.088 degrees. (360/4096=0.088 degrees) One degree equals 11.3636 units.
(4096=1000 Hexadecimal value, or 4096=Bit 12 =1 Binary value (1'0000'0000'0000)

**Caution:** Parameter 4077 S2 must not be changed. This parameter sets the live tool spindle orientation position that is critical about alignment of the drive coupling.

**Angular spindle positioning**
On machines where the spindle positioning option is available, positioning at a spindle rotation angle is possible in angular increments of 0.1 degrees. This function cannot do interpolation with another axis.

**Angular positioning of the main spindle**
The command for spindle positioning is as follows:

- Zero-degree angle: **G97 S0 M19 P11**
- 180-degree angle: **G97 S1800 M19 P11** (multiply positioning angle by 10)
- Any angle: **G97 S3599 M19 P11** (not to exceed 3600 units)
Once commanded, the spindle is held in position under power by the spindle motor. The M3, M4 or M5-command cancels spindle positioning.

**System parameter 4077 S-1** sets the reference angle for the main spindle.
DRILLING AND TAPPING WITH LIVE TOOLS ON THE C-AXIS

Canned cycles for hole machining with the C and Z-axis

Z-axis peck drilling, C-axis positioning

G83 C___Z___Q___ P___F___

Z-axis tapping

G84 C___Z___F___

Notes: C = C-axis position, X = X-end position, (diameter), Q = peck distance (No decimal point allowed with the Q. Repeat Q on each subsequent line), P = Dwell, F = Feed Rate.

C-axis clamping command M89 is optional. It can be added to the cycle, as shown in the example, below.

Example: Drilling and Tapping on the Front Face of a part
Drill (4) Holes, diameter 0.201 on the front face equally spaced on a 1.5" Diameter circle, 0.45" deep. Peck depth is 0.125". Clamp the C-axis during drilling. Tap the 4 holes, ¼-20-UN, and 0.35 deep.

<table>
<thead>
<tr>
<th>Peck Drilling Program</th>
<th>Tapping Program (Rigid Mode)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G0G40G80G99</td>
<td>G0G40G80G99</td>
</tr>
<tr>
<td>M90</td>
<td>M90</td>
</tr>
<tr>
<td>M35</td>
<td>M35</td>
</tr>
<tr>
<td>G28 H0</td>
<td>G28 H0</td>
</tr>
<tr>
<td>T0707</td>
<td>T0808</td>
</tr>
<tr>
<td>G97S2500M3 P12</td>
<td>G0C0Z.5</td>
</tr>
<tr>
<td>G0C0Z.5</td>
<td>X1.5 M8</td>
</tr>
<tr>
<td>--------------</td>
<td>--------------</td>
</tr>
<tr>
<td>X1.5 M8</td>
<td>Z.1</td>
</tr>
<tr>
<td>Z.1</td>
<td>G97S1000M29P12</td>
</tr>
<tr>
<td>G83C0Z-.45.Q1250F.005M89</td>
<td>G84C0Z-.35F.05M89</td>
</tr>
<tr>
<td>C90.Q1250M89</td>
<td>C90. M89</td>
</tr>
<tr>
<td>C180.Q1250M89</td>
<td>C180. M89</td>
</tr>
<tr>
<td>C270.Q1250M89</td>
<td>C270. M89</td>
</tr>
<tr>
<td>G0G80Z.5M90</td>
<td>G0G80Z.5M90</td>
</tr>
<tr>
<td>M1</td>
<td>M1</td>
</tr>
</tbody>
</table>
Canned cycles for hole machining with the C and X-axis

*X-axis peck drilling, C-axis positioning*

\[ G87 \text{ C____X____Q____ P____F____} \]

*X-axis tapping*

\[ G88 \text{ C____X____F____} \]

**Notes:** C = C-axis position, Z = Z-end position, Q = peck distance (No decimal point allowed with the Q. Repeat Q on each subsequent line), P = Dwell, F = Feed Rate.

C-axis clamping command M89 is optional. It can be added to the cycle, as shown in the example, below.

**Example: Drilling and Tapping on the OD of a part**

Drill (4) Holes, diameter 0.201, located at Z (minus)-0.5". Holes equally spaced around a 2" OD. Drill through into the 1.5" diameter bore. Peck depth is 0.125". Clamp the C-axis during drilling. Tap (4) holes ¼-20-UN, 0.35 deep from the OD.

<table>
<thead>
<tr>
<th>Peck Drilling Program</th>
<th>Tapping Program (Rigid Mode)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G0G40G80G99</td>
<td>G0G40G80G99</td>
</tr>
<tr>
<td>M90</td>
<td>M90</td>
</tr>
<tr>
<td>M35</td>
<td>M35</td>
</tr>
<tr>
<td>G28 H0</td>
<td>G28 H0</td>
</tr>
<tr>
<td>T0909</td>
<td>T1010</td>
</tr>
<tr>
<td>G97S2500M3P12</td>
<td>G0C0Z.5</td>
</tr>
<tr>
<td>G0C0Z.5</td>
<td>X2.25 M8</td>
</tr>
<tr>
<td>X2.15 M8</td>
<td>Z-.5</td>
</tr>
<tr>
<td>Z-.5</td>
<td>G97S1000M29P12</td>
</tr>
<tr>
<td>G87X1.3C0Q1250F.005M89</td>
<td>G88X1.3C0F.05M89</td>
</tr>
<tr>
<td>C90.Q1250M89</td>
<td>C90. M89</td>
</tr>
<tr>
<td>C180.Q1250M89</td>
<td>C180. M89</td>
</tr>
<tr>
<td>C270.Q1250M89</td>
<td>C270.M89</td>
</tr>
<tr>
<td>G0G80X2.15</td>
<td>G0G80X2.2</td>
</tr>
<tr>
<td>Z.5</td>
<td>Z.5</td>
</tr>
<tr>
<td>M1</td>
<td>M1</td>
</tr>
</tbody>
</table>
POLAR COORDINATE INTERPOLATION FUNCTION G12.1

On a Turning Center that is equipped with a C-axis (rotary axis), interpolation between the linear axis “X” the rotary axis “C” is possible by use of the G12.1-function. This function simplifies programming of shapes machined on the front face of a part, such as the rectangular shape with rounded corners as shown here.

Machining of such shapes is accomplished by use of an end mill that is attached to a “Z-axis live tool attachment”, with the end mill pointing toward the front face of the part.

Programming with the G12.1-function is done on the X-C coordinate system plane. In this coordinate system plane the C–axis is regarded as a linear axis instead of a virtual rotary axis. Programming is done similar to the way it is done on a basic X-Y plane. Linear or circular interpolation can be done. Cartesian coordinates are used for defining either part shape or the tool path geometry. In G12.1-mode the control converts Cartesian coordinates to Polar coordinates, automatically.
Layout of the X-C coordinate system plane

- The diagram above shows the X-C coordinate system plane as viewed when looking at the front face of the main spindle.
- The address “X” defines a point by the distance from origin horizontally on diameter (Positive or negative value). “On diameter” means: twice the actual distance from origin.
- The address “C” defines a point by the actual distance vertically from origin (Positive or negative value). “C” is defined in linear units of measurement, not in angular units.

Notes on programming with the G12.1 function

- **Plane selection**
  The G18-plane select command must be active in G12.1-mode. On turning centers the X-Z coordinate system plane (G18) is set as the default plane. System parameter settings related to the G12.1-function are normally set to allow polar coordinate interpolation in the G18-plane.

- **Coordinate system origin**
  The Origin of the X-C coordinate system is fixed at the center of the revolving work spindle. The origin (X0, C0) must not be shifted.

- **Angular orientation of the X-C coordinate system plane**
  Angular orientation is set by the absolute C-axis (angle) that exists at the time when entering the G12.1 – mode. For example: when G0 C60.0 has been commanded before entering the G12.1-mode, the X-C coordinate system plane is set on a 60° angle relative to both the horizontal and vertical axis.
• **Positioning command**
  “G0” cannot be used in G12.1-mode. Positioning is done in G1- mode, using a feed rate of around 30" to 60" per minute, depending on application.

• **Feed command**
  In the G12.1-mode the feed velocity can be specified either by units of linear distance per minute (G98-mode) or by units of linear distance per spindle revolution (G99-mode). Use of excessive feed rate can adversely influence the accuracy of a machined shape. Recommend range of feed rates for polar coordinate interpolation is from 1" to around 10" per minute, depending on application. Feed rate must be reduced in case when circular interpolation is done near the X-C zero point. Velocity of the rotary axis may become excessive and as a result, servo errors or servo overload may occur.

• **Incremental axis move command**
  Address “U” can be used for incremental move command along the X-axis. U= horizontal distance from a current point to the next point – on diameter.

  Address “H” can be used for incremental move command along the C-axis. H= vertical distance from a current point to the next point.

• **Linear interpolation command**
  G1 X__C__F__ (absolute) or: U__H__ F__ (incremental)

  Interpolation between X and Z or between C and Z cannot be done. Z-axis move command must be specified in a separate block, not together with X-axis or C-axis commands.

• **Circular Interpolation command**
  (G2 or G3) X__C__R__F__ (absolute) or: U__H__R__F__ (incremental)

  Addresses X and C define the end point of an arc. The address “R” defines the radius of an arc when the included angle of the arc segment is 180° or less.

  Addresses “I” and “J” can be used for defining the arc center.

  Address “I” specifies the actual distance and direction (+/-) from the start point of the arc to the arc center along the X-axis.

  Address “J” specifies the actual distance and direction (+/-) from the start point of the arc to the arc center along the C-axis.
Command for arc of less than 360°:  (G2 or G3) X_ C_ I_ J_

Command used for full circle:   (G2 or G3) I_ or: J_

- **Cutter compensation function**
  In polar coordinate interpolation the cutter compensation function should always be used, regardless of programming method. Size control on a machined shape is done by use of the cutter compensation function, not by changing the X-offset data.

  G40 must be active at the time when entering the G12.1-mode.
  G41 or G42 must be commanded after the G12.1- command.
  G40 should be commanded before canceling the G12.1-mode.

  Cutter compensation commands should be done together with a G1-command, moving the tool onto the part or away from it. For example:

  G1 G41 X_ or C_ F_ (“Ramp-ON”)
  G1 G40 X_ or C_ F_ (“Ramp-OFF”)

  When ramping ON or OFF along the X-axis the moving distance must be greater than or equal to twice the “R-data”. When ramping ON or OFF along the C-axis the moving distance must be greater than or equal to the “R-data”.

- **Cutter compensation data setting**
  Cutter compensation data (R-offset) must be set under column “R” located in the tool offset data tables. Data setting depends on the programming method that has been used:

  a) When the program-coordinates represent the geometry of the tool center-path, the R-data is set at zero, initially.
  b) When the program-coordinates represent the geometry of the actual part shape, the actual cutter radius must be input on the R-data.

  The “Tool nose type”- data (located at the column “T” at the tool-offset tables) is part of the cutter compensation data. On the offset number that is used for a milling cutter the “T” data must be set = 0.
Tool offset data, including the “R” and “T”-data are activated by the tool offset command. (The “D”-command, such as used in machining center programming cannot be used).

- **Adjusting the part size**
  Suppose that an external hexagon shape was machined over-size by 0.005” (measurement across the flats). Under the condition that the cutter compensation function has been properly applied in the program, size adjustment on this part can be done by reducing the “R”-data in the tool offset table by the amount of -0.005”. Size control cannot be accomplished by adjusting the X-axis offset.

- **Tool types to be used / touching-off tools**
  Machining in the G12.1-mode is normally done by use of a flat bottom end mill that is pointing toward the end-face of the part. This tool must be touched off (along the X-axis) at the cutter center, not at the periphery of the tool. Once the X-axis tool-offset for a given tool has been established accurately, it must not be modified later in attempting to control the size of the machined part. (Please refer to paragraph above). Erroneous tool offset data causes faulty part geometry.

  In polar coordinate interpolation, a flat bottom end mill that is pointing toward the OD of the part cannot produce an acceptable part shape. On some applications, a ball-nose end mill that is pointing toward the OD of the part can be used. This type of tool must be touched off along the X-axis at the center of the ball-nose.

- **Programming the Tool approach point**
  Caution must be used when positioning the milling cutter near the OD of a part. The X-axis tool offset data is based on the cutter center, not the periphery. The tool approach point for the X-axis is calculated as follows: \( X = \text{Part OD} + \text{Cutter diameter} + \text{clearance} \).
**Programming example**

The figure above left shows two flat surfaces to be machined on the front face of a 1.25" diameter part. A clearance diameter of 1.300" that intersects with both of the flats has been added to the figure on the right. The coordinates (X 1.0, C0.4153) located at the upper right corner and (X1.0, C-0.4153) located at the lower right corner will be used for preparing the machining program. The axial depth of the flats is assumed to be at Z-0.375" from the front face. A ½" diameter end mill is used for machining the two flats.

**Hints on programming and machining with the polar coordinate interpolation function**

- Before programming the part, a simple part layout should be prepared. An end-view of the part should be drawn, showing the part just the way it is held by the chuck as viewed when looking toward the face of the chuck on the main spindle. In this layout the X-coordinates run horizontally and the C-coordinates run vertically. X-plus direction runs to the right of X0, C-plus direction runs from zero to 12 o’clock. The X-coordinates are specified on diameter, C-coordinates are specified as actual distance. Radii are specified as actual distance.
- For programming purposes it is assumed that the cutter approaches the part always from the 3 o’clock position (X- axis plus side) on your layout. When deciding the cutting start point it is recommended placing it at the plus-side of the X-axis.
- During machining of the flat surfaces as shown in the above example, the cutter does not actually move in a vertical direction. In the G12.1-mode, machining of a contour shape is accomplished by moving the cutter horizontally along the X-axis and by rotating the C-axis, synchronously.
- When negative X-axis coordinates are commanded in the program the cutter center will not actually travel past the minus side of X0. Instead, the C-axis is rotated around automatically so that machining on the negative quadrant is accomplished with the cutter always remaining on the plus-side of the X-axis.
Deciding the machining method

The milling operation on this part can be programmed in various different ways. Examples for three different programming methods A, B and C are shown.

Programming Method “A”

The figure on right shows programming of the tool center-path. The cutter to be used is ½” on diameter. The X and C coordinates must be calculated, considering both the part geometry as well as the cutter radius. This programming method is used mostly by CAM software programming systems.

Both surfaces are being machined by use of a continuous tool path. No actual machining is done on the 0.650”-radius. The cutter will clear the 1.25” part diameter. Hence the cutter can be moved around the arc at a high feed rate.

The cutter compensation function should be applied, always. However the cutter radius amount entered in the tool-off data should be set either at zero or at a small (plus) amount which will make the part come out slightly oversize, initially.

Programming example, using programming method “A”

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>M5P11</td>
<td>Stop the main spindle</td>
</tr>
<tr>
<td>M35</td>
<td>C-axis select command</td>
</tr>
<tr>
<td>G40</td>
<td>Cutter comp cancel command</td>
</tr>
<tr>
<td>G13.1</td>
<td>Polar coordinate interpolation cancel command</td>
</tr>
<tr>
<td>G30 U0 W0</td>
<td>Tool exchange point</td>
</tr>
<tr>
<td>G28 H0</td>
<td>Zero return C-axis</td>
</tr>
<tr>
<td>T0808 (3/4” DIA. CUTTER)</td>
<td>When the cutter center path is programmed the R-offset for the tool is set = 0, initially. After inspecting the first part, adjust the R either plus or minus, as needed for size control. When</td>
</tr>
<tr>
<td>G97 S2000 M3P12</td>
<td>Live tool spindle ON</td>
</tr>
<tr>
<td>G0Z.1 C0 M8</td>
<td>Z-approach, C-axis at zero degrees</td>
</tr>
<tr>
<td>X2.15</td>
<td>X-approach (1.3+0.75+0.1=2.15)</td>
</tr>
</tbody>
</table>
G12.1  Polar coordinate interpolation ON
G1 G98 C.5339 F60.  C-axis position at the first point of the contour shape, use IPM - feed mode, if desired (Note: the C-command at this time represents a linear dimension – not degrees)
G1 G41 X1.75F7. (1)  Cutter comp ON  
X-axis position at the first point of the contour shape
Z-.88  Move the Z-axis to the desired depth on the part
G1 C-.5339 (2)  C-axis position at the second point of contour
G2 X-1.75 R1.025 F60. (3)  X-axis position at the third point of the contour (No cutting is done on the arc, a high feed rate is used for the arc move )
G1 C.5339 F7. (4)  Fourth point of the contour
G1 G40 X-2.15 F60.  Cutter comp OFF  
The X-axis must move at least two times the “R”-value that is used in the tool offset
G13.1  Polar coordinate interpolation OFF
G99 G0 X2.5  Retract X-axis & switch back to IPR- feed mode
Z.1 M5P12  Retract Z-axis and stop milling spindle
G30 U0 W0. M9  Second reference point return
M1  Optional stop

**Programming Method “B”**

By this method, both surfaces are being machined in one continuous path similar to example “A”, above. However, in this case the X-C coordinates used for programming are the same as the actual part geometry, as shown on the figure on right. The cutter compensation function must be applied in this case, without fail. The cutter radius amount entered in the tool-off data must be the same as the actual cutter radius to be used, or slightly larger to produce a slightly oversized part.

**Programming Method “C”**

By this method, each one of the flats is programmed separately. One of the surfaces is machined at 0-degree angle first as shown on the left figure. Next, the part is rotated 180° then the other surface is machined as shown on the figure on right. The cutter compensation function must be applied in this case, without fail.
Preparing the machining program for methods “B” and “C”

“Climb Cutting” is done in both cases. Hence the cutting start point coordinates in both cases are at X1.0 C0.4153. The automatic cutter compensation function G41 is applied. The cutting start point is located on the top right corner on each of the figures shown. The part dimensions as shown on the sketches can be ‘plugged’ directly into the program.

<table>
<thead>
<tr>
<th>Programming Method “B”</th>
<th>Programming Method “C”</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>NC Program – machining both of the flats in one continuous path, using a 1/2”-diameter cutter.</strong></td>
<td><strong>NC Program – doing each flat separately, using a 1/2”-diameter cutter.</strong></td>
</tr>
<tr>
<td>N100 (MILL TWO FLATS CONTINUOUS PATH)</td>
<td>N100 (MILL TWO FLATS)</td>
</tr>
<tr>
<td>G40</td>
<td>G40</td>
</tr>
<tr>
<td>G13.1</td>
<td>G13.1</td>
</tr>
<tr>
<td>T0101</td>
<td>T0101</td>
</tr>
<tr>
<td>M5P11</td>
<td>M5P11</td>
</tr>
<tr>
<td>M35</td>
<td>M35</td>
</tr>
<tr>
<td>G28 H0</td>
<td>G28 H0</td>
</tr>
<tr>
<td>G97 S1000 M3P12</td>
<td>G97 S1000 M3P12</td>
</tr>
<tr>
<td>G0 Z.1 C0</td>
<td>G0 Z.1 C0</td>
</tr>
<tr>
<td>X1.8</td>
<td>X1.8</td>
</tr>
<tr>
<td>G12.1</td>
<td>G12.1</td>
</tr>
<tr>
<td>G1 G98 C0.4153 F20.</td>
<td>G1 G98 C0.4153 F20.</td>
</tr>
<tr>
<td>G1 G41 X1.0 F5.</td>
<td>G1 G41 X1.0 F5.</td>
</tr>
<tr>
<td>Z - .375 (depth of cut)</td>
<td>Z - .375 (depth of cut)</td>
</tr>
<tr>
<td>C-0.4153</td>
<td>C-0.4153</td>
</tr>
<tr>
<td>G2 X-1.0 R.650 F60.</td>
<td>G2 X-1.0 R.650 F60.</td>
</tr>
<tr>
<td>G1 C0.4153 F5.</td>
<td>G1 C0.4153 F5.</td>
</tr>
<tr>
<td>G40 X-1.8</td>
<td>G40 X-1.8</td>
</tr>
<tr>
<td>G13.1</td>
<td>G13.1</td>
</tr>
<tr>
<td>G0 Z0.1</td>
<td>G0 Z0.1</td>
</tr>
<tr>
<td>G0 X—Z—M5P12</td>
<td>G0 X—Z—M5P12</td>
</tr>
<tr>
<td>M34 (C-Axis Off)</td>
<td>M34 (C-Axis Off)</td>
</tr>
<tr>
<td>M1</td>
<td>M1</td>
</tr>
</tbody>
</table>

119
**CYLINDRICAL INTERPOLATION**

*Principle of Operation*

The cylindrical interpolation function “G7.1” allows circular interpolation between the **Z-axis and a rotary axis**. Programming is done using Cartesian coordinates for the Z-axis and degrees of rotation for the rotary axis. Arc specifications are given in units of linear measurement. Typical applications for this function include engraving operation for lettering or for milling of cam shapes on the circumference of a cylinder.

*Layout of the Z-C Coordinate system*

The sketch below shows the Z-C coordinate system.

---

**Programming Notes**

- **Plane Select Command:** G18

- **G7.1H < 0** or **G7.1 C < 0** activates the cylindrical interpolation function. An H-value or a C-value greater than zero specifies the radius of the cylinder to be machined. For example: Cylindrical interpolation mode is set by this command:

  G1 G18 W0 H0 followed by G7.1 H0.75 in separate block.

- **G7.1 H0** or **G7.1 C0** cancels the cylindrical interpolation function.
• **Z-coordinates** specify absolute dimensions parallel to the length of the cylinder. The letter “W” can be used for incremental specification along the Z-axis.

• **C-axis rotation** is specified as an absolute angle in degrees. The letter “H” for incremental angle specification can be used, instead.

• **X-coordinates** specify absolute dimensions on the OD of the cylinder. The letter “U” can be used for incremental specification along the X-axis.

• **Positioning G0** cannot be done when cylindrical interpolation mode is active.

• **Linear interpolation G1** is possible with all three axes, simultaneously.

• **Circular interpolation** (G2, G3) between Z-linear coordinates and C-angular coordinates is performed automatically by the control using the G7.1-function. Circular interpolation between X and C axis cannot be done.

• **Arc radius specification.** The letter “R” must be used for arc specifications. Letters I J or K cannot specify an arc radius in cylindrical interpolation.

• **Cutter Radius Compensation Functions (G40, G41 and G42)** can be applied. The cutter radius as registered under “R” on the tool-offset tables is applied for cutter radius compensation automatically.

• **Tool path:** For programming purposes, the surface on the circumference of a cylinder is laid out in the shape of a rectangle whose length is equal to the cylinder diameter times pi. The height equals the height of the cylinder. The tool path is then projected onto this rectangle. Horizontal dimensions are to be converted from linear to angular C axis coordinates. The Vertical dimensions represent Z-axis coordinates. The zero point of the coordinate system can be decided at an arbitrary location.

---

**Formula for converting the length of an arc to degrees of rotation**

The use of RADIANS can simplify conversion from linear units to degree-units. To convert the length of an arc for a segment of a circle into degrees of rotation, the following formula is applied:

\[ C^\circ = \frac{L}{R} \times 57.29578^\circ \]

C = Degrees of rotation, L = linear distance

R = radius of the circle, 57.29578° = one radian.
When diameter “D” is used to define the circle, use this formula:

\[ C^\circ = \frac{L}{D} \times 114.59156^\circ \]

114.59156° = two radians.

**Cylindrical Interpolation Example**

The letters “J and R” to be engraved around the OD of a 2.9”-diameter part, using cylindrical interpolation-function G7.1 A 1/32-radius ball-nose end mill is used for engraving the letters. In order to define the tool path, coordinates X, C and Z for every point on the entities are required.

**Layout of tool path**

In order to simplify programming the cylindrical surface of the part to be machined is represented in form of a flat sheet that measures the equivalent of the part’s circumference vertically and the part’s length in horizontal direction. Orientation of the part is the same as viewed looking down from the operator’s side of the machine when the part is clamped in the chuck.
Converting linear coordinates to degrees of rotation

For the sample part at hand the factor for converting linear units into degrees is calculated as follows: \( \frac{1}{2.9} \times 114.59156 = 39.514331^\circ \) per 1" of linear distance

\[
C^\circ = L \times 39.5143^\circ
\]

The table below shows the start-points and end-points for the lettering

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Z</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start point of letter J</td>
<td>2.9</td>
<td>-0.7</td>
<td>0.4 * 39.5143 = 15.806º</td>
</tr>
<tr>
<td>End point of letter J</td>
<td>2.9</td>
<td>-0.45</td>
<td></td>
</tr>
<tr>
<td>Start point of letter R</td>
<td>2.9</td>
<td>-0.3</td>
<td>-0.1 * 39.5143 = -3.951º</td>
</tr>
<tr>
<td>End point of letter R</td>
<td>2.9</td>
<td>-0.4</td>
<td>-0.4 * 39.5143 = -15.806º</td>
</tr>
</tbody>
</table>

N100 (ENGRAVING LETTERS J & R )
G0G80G40G18
M35
G7.1H0
G28H0
T1111
G97S4000M3P12
G0Z-.7
G0X3.1C15.806 M8
G1G98
G18W0H0
G7.1H1.45
X2.9F5.
C3.951
Z-.45
G3Z-.45C15.805R.15
G1X3.5F200.
Z-.3C-3.951
G1X2.9F5.
Z-.7
C-10.8664
G3Z-.45C-10.866R.125
G1C-3.9514
C-10.866
G1Z-.3C-15.8057
G1X3.1F200.
G7.1H0
G30U0M5P12
G30W0
M34
M1