Century Star Milling CNC System

Programming Guide

V3.3
December, 2007

Wuhan Huazhong Numerical Control Co., Ltd

©2007 Wuhan Huazhong Numerical Control Co., Ltd
Preface

Organization of documentation
1. General
2. Preparatory Function
3. Interpolation Function
4. Feed Function
5. Coordinate System
6. Spindle Speed Function
7. Tool Function
8. Miscellaneous Function
9. Functions to Simplify Programming
10. Comprehensive Programming Example
11. Custom Macro

Applicability
This Programming Guide is applicable to the following CNC system:
HNC-18iM/19iM v4.0
HNC-18xp/M
HNC-19xp/M
HNC-21MD/22MD v05.62.07.10

Internet Address
http://www.huazhongcnc.com/
# Table of Contents

Preface.................................................................................................................. i
1  General................................................................................................................. 1
   1.1  CNC Programming....................................................................................... 2
   1.2  Interpolation................................................................................................. 4
      1.2.1  Linear Interpolation............................................................................... 4
      1.2.2  Circular Interpolation........................................................................... 4
      1.2.3  Helical Interpolation............................................................................. 5
   1.3  Feed Function............................................................................................... 6
   1.4  Coordinate System....................................................................................... 7
      1.4.1  Reference Point.................................................................................... 7
      1.4.2  Machine Coordinate System................................................................. 8
      1.4.3  Workpiece Coordinate System.............................................................. 9
      1.4.4  Setting Two Coordinate Systems at the Same Position.................... 10
      1.4.5  Absolute Commands.......................................................................... 11
      1.4.6  Incremental Commands....................................................................... 12
      1.4.7  Polar Coordinates............................................................................... 13
   1.5  Spindle Speed Function............................................................................... 14
   1.6  Tool Function.............................................................................................. 15
      1.6.1  Tool Selection....................................................................................... 15
      1.6.2  Tool Offset............................................................................................ 15
   1.7  Miscellaneous Function............................................................................... 17
   1.8  Program Configuration.............................................................................. 18
      1.8.1  Structure of an NC Program................................................................. 18
      1.8.2  Main Program and Subprogram......................................................... 19
2  Preparatory Function (G code)......................................................................... 20
   2.1  G code List.................................................................................................. 21
3  Interpolation Functions..................................................................................... 24
   3.1  Positioning (G00)....................................................................................... 25
   3.2  Single Direction Positioning (G60)........................................................... 26
   3.3  Linear Interpolation (G01)......................................................................... 27
   3.4  Circulation Interpolation (G02, G03)........................................................ 29
   3.5  Helical Interpolation (G02, G03)................................................................. 35
   3.6  Virtual Axis (G07) and Sine Interpolation................................................. 38
   3.7  Tapping (G34)............................................................................................ 40
4  Feed Function.................................................................................................... 43
   4.1  Rapid Traverse (G00)................................................................................ 44
   4.2  Cutting Feed (G94, G95).......................................................................... 45
   4.3  Dwell (G04)............................................................................................... 46
   4.4  Exact Stop (G09, G61).............................................................................. 47
   4.5  Cutting Mode (G64)................................................................................... 49
5  Coordinate System............................................................................................ 51
   5.1  Reference Position Return (G28).............................................................. 52
   5.2  Auto Return from Reference Position (G29).......................................... 53
   5.3  Setting a Workpiece Coordinate System (G92)....................................... 55
   5.4  Selecting a Machine Coordinate System (G53)........................................ 56
   5.5  Selecting a Workpiece Coordinate System (G54–G59)............................. 57
   5.6  Plane Selection (G17, G18, G19)............................................................... 59
   5.7  Absolute and Incremental Programming (G90, G91)................................. 60
   5.8  Dimension Selection (G20, G21, G22)...................................................... 62
5.9  Polar Coordinates...................................................................................................... 63
6  Spindle Speed Function.................................................................................................. 66
7  Tool Function................................................................................................................ 67
    7.1  Tool Selection and Tool Offset (T code).................................................................. 68
    7.2  Tool Radius Compensation (G40, G41, G42)....................................................... 69
    7.3  Tool Length Compensation (G43, G44, G49)...................................................... 74
    7.4  RTCP (Rotation Tool Center Point Programming)............................................... 76
8  Miscellaneous Function............................................................................................... 77
    8.1  M code List............................................................................................................. 78
    8.2  CNC M-Function.................................................................................................... 79
          8.2.1  Program Stop (M00)....................................................................................... 79
          8.2.2  Optional Stop (M01)...................................................................................... 79
          8.2.3  End of Program (M02)................................................................................... 79
          8.2.4  End of Program with return to the beginning of program (M30).............. 79
          8.2.5  Subprogram Control (M98, M99).................................................................. 80
    8.3  PLC M Function..................................................................................................... 81
          8.3.1  Spindle Control (M03, M04, M05).................................................................. 81
          8.3.2  Tool Selection (M06)....................................................................................... 81
          8.3.3  Coolant Control (M07, M08, M09).................................................................. 81
9  Functions to Simplify Programming............................................................................. 82
    9.1  Mirror Image (G24, G25)...................................................................................... 83
    9.2  Scaling (G50, G51)............................................................................................... 85
    9.3  Coordinate System Rotation (G68, G69)............................................................. 87
    9.4  Canned Cycles....................................................................................................... 89
          9.4.1  Return to the Initial Point/R point Level (G98, G99)....................................... 90
          9.4.2  High-speed Peck Drilling Cycle (G73)............................................................ 91
          9.4.3  Left-hand Tapping Cycle (G74)...................................................................... 93
          9.4.4  Fine Boring Cycle (G76).................................................................................. 95
          9.4.5  Drilling Cycle, Spot Drilling (G81).................................................................. 97
          9.4.6  Drilling Cycle, Counter Boring Cycle (G82).................................................... 99
          9.4.7  Peck Drilling Cycle (G83).............................................................................. 101
          9.4.8  Tapping Cycle (G84)...................................................................................... 103
          9.4.9  Boring Cycle (G85)....................................................................................... 105
          9.4.10 Boring Cycle (G86)....................................................................................... 107
          9.4.11 Back Boring Cycle (G87).............................................................................. 109
          9.4.12 Manual Boring Cycle (G88)........................................................................... 111
          9.4.13 Boring Cycle (G89)...................................................................................... 113
          9.4.14 Canned Cycle Cancel (G80).......................................................................... 114
    9.5  Summary................................................................................................................ 115
10  Custom Macro.............................................................................................................. 121
    10.1  Variables.............................................................................................................. 122
          10.1.1 Type of Variables........................................................................................... 122
          10.1.2 System Variables........................................................................................... 123
    10.2  Constant................................................................................................................. 130
    10.3  Operators and Expression..................................................................................... 131
    10.4  Assignment............................................................................................................ 132
    10.5  Selection statement IF, ELSE,ENDIF................................................................. 133
    10.6  Repetition Statement WHILE, ENDW............................................................... 134
    10.7  Macro Call.............................................................................................................. 135
    10.8  Example................................................................................................................ 137
1 General

This chapter is to introduce the basic concepts in Computerized Numerical Control (CNC) system: HNC-21M/22M, HNC-18iM/19iM, HNC-18xp/M, HNC-19xp/M.
1.1 CNC Programming

To operate CNC machine tool, the first step is to understand the part drawing and produce a program manual script. The procedure for machining a part is as follows (Figure 1.1):

1) Read drawing
2) Produce the program manual script
3) Input the program manual script by using the machine control panel
4) Manufacture a part
1. Reading drawing

2. Programming

\[
\begin{align*}
\text{%3308 (the origin is on A)} \\
N1 & \text{ G92 X0 Y0 Z50} \\
N2 & \text{ M03 S500} \\
N3 & \text{ G00 X-31 Y-26} \\
N4 & \text{ Z5} \\
N5 & \text{ G01 Z-3 F40} \\
\end{align*}
\]

3. Inputting program

4. Manufacturing

Figure 1.1 The workflow of operation of CNC machine tool
1.2 Interpolation

Interpolation refers to an operation in which the machine tool moves along the workpiece parts. There are five methods of interpolation: linear, circular, helical, parabolic, and cubic. Most CNC machine can provide linear interpolation and circular interpolation. The other three methods of interpolation (helical, parabolic, and cubic interpolation) are usually used to manufacture the complex shapes, such as aerospace parts.

1.2.1 Linear Interpolation

Linear interpolation refers to the tool movement along a straight line.

![Linear Interpolation Diagram](image)

Figure 1.2 Linear Interpolation

1.2.2 Circular Interpolation

Figure 1.3 shows a tool movement along an arc.

![Circular Interpolation Diagram](image)

Figure 1.3 Circular Interpolation

Note:

In this manual, it is assumed that tools are moved against workpieces.
1.2.3 Helical Interpolation

Helical interpolation can be used to manufacture threads on a workpiece.

Figure 1.4 Helical Interpolation
1.3 Feed Function

- Feed refers to an operation in which the tool moves at a specified speed to cut a workpiece.
- Feedrate refers to a specified speed, and numeric is used to specified the feedrate.
- Feed function refers to an operation to control the feedrate.

![](image)

Figure 1.5 Feed Function

For example:

F150.0 //feed the tool at 150mm/min, while the workpiece makes one turn
1.4 Coordinate System

1.4.1 Reference Point

Reference point is a fixed position on CNC machine tool, which is determined by cams and measuring system. Generally, it is used when the tool is required to exchange or the coordinate system is required to set.

Reference Position

![Reference Position Diagram](image)

Figure 1.6 Reference Point

There are two ways to move to the reference point:

- Manual reference position return: The tool is moved to the reference point by operating the button on the machine control panel. It is only used when the machine is turned on.

- Automatic reference position return: It is used after the manual reference position return has been used. In this manual, this would be introduced.
1.4.2 Machine Coordinate System

The coordinate system is set on a CNC machine tool. Figure 1.7 is a machine coordinate system of milling machine, and shows the direction of axes:

![Figure 1.7 Machine Coordinate System](image)

In general, three basic linear coordinate axes of motion are X, Y, Z. Moreover, X, Y, Z axis of rotation is named as A, B, C correspondently. Due to different types of milling machine, the axis direction can be decided by following the rule – “three finger rule” of the right hand.

![Figure 1.8 “three finger rule”](image)

- The thumb points the X axis. X axis controls the cross motion of the cutting tool. “+X” means that the tool is away from the spindle centerline.
- The index points the Y axis. Y axis is usually a virtual axis.
- The middle finger points the Z axis. Z axis controls the motion of the cutting tool. “+Z” means that the tool is away from the spindle.
1.4.3 Workpiece Coordinate System

The coordinate system is set on a workpiece. The data in the NC program is from the workpiece coordinate system.

![Diagram of workpiece coordinate system](image)

**Figure 1.9 Workpiece Coordinate System**

Example: Those three points can be defined on workpiece coordinate system:

- P1 corresponds to $X20 \ Y35$
- P2 corresponds to $X50 \ Y60$
- P3 corresponds to $X70 \ Y20$

![Diagram of points on workpiece coordinate system](image)

**Figure 1.10 Example of defining points on workpiece coordinate system**
1.4.4 Setting Two Coordinate Systems at the Same Position

When a workpiece is set on the table, the positional relation between machine coordinate system and workpiece coordinate system are set.

![Diagram of setting two coordinate systems at the same position]

Figure 1.11 Setting two coordinate systems at the same position

According to the command program based on the workpiece coordinate system, the tool moves on the coordinate system specified by CNC, and cuts a workpiece.
1.4.5 Absolute Commands

The absolute dimension describes a point at “the distance from zero point of the coordinate system”.

Example: These three point in absolute dimensions are the following:

- P1 corresponds to X20 Y35
- P2 corresponds to X20 Y60
- P3 corresponds to X70 Y20

![Figure 1.12 Absolute Dimension]
1.4.6 Incremental Commands

The incremental dimension describes a distance from the previous tool position to the next tool position.

Example: These three point in incremental dimensions are the following:

- P1 corresponds to X20 Y35 //with reference to the zero point
- P2 corresponds to X30 Y20 //with reference to P1
- P3 corresponds to X20 Y-35 //with reference to P2

![Figure 1.13 Incremental Dimension](image)
1.4.7 Polar Coordinates

Beside the “Cartesian coordinate system”, another way to specify coordinates is “polar coordinates”. The polar coordinate method is useful only if there is radius and angle measurements on a workpiece.

Example: Two points P1 and P2 with reference to the pole are described as follows.

P1 corresponds to radius=100 plus angle=30°
P2 corresponds to radius=60 plus angle=75°
1.5 Spindle Speed Function

The cutting speed (v) refers to the speed of the tool with respect to the workpiece when the workpiece is cut. The unit of the cutting speed is m/min. As for the CNC, the cutting speed can be specified by the spindle speed (N) in min⁻¹.

\[ N = \frac{1000 \cdot v}{\pi D} \]

N: the spindle speed
v: cutting speed
D: diameter value of the workpiece

Example: When the diameter of workpiece is 100mm, and the cutting speed is 80m/min, then the spindle speed: \[ N = \frac{1000 \cdot 80}{\pi \cdot 100} = \frac{1000 \cdot 80}{\pi \cdot 100} \approx 250 \text{rpm} \]

The constant surface speed refers to the speed even when the workpiece diameter is changed, and the CNC changes the spindle speed. At this time, the spindle speed is the cutting speed.
1.6 Tool Function

1.6.1 Tool Selection

It is necessary to select a suitable tool when drilling, tapping, boring or the like is performed. As it is shown in Figure 1.16, a number is assigned to each tool. Then this number is used in the program to specify that the corresponding tool is selected.

![Figure 1.16 Tool Selection](image)

1.6.2 Tool Offset

When writing a program, the operator just use the workpiece dimensions according to the dimensions in the part drawing. The tool nose radius center and the tool length are not taken into account. However, when machining a workpiece, the tool path is affected by the tool geometry. There are two kinds of tool offset: tool length compensation and tool radius compensation.

![Figure 1.17 Length compensation and Radius compensation](image)
• Tool Length Compensation

There are two kind of ways to specify the value of tool length compensation.
  - Absolute value of tool compensation (the distance between tool tip and machine reference point)
  - Incremental value of tool compensation (the distance between tool tip and the standard tool)

• Tool Radius Compensation

Figure 1.18 shows the difference between the programmed contour and the corrected tool path.

![Diagram showing programmed contour and corrected tool path](image)

Figure 1.18 Difference between programmed contour and corrected tool path
1.7 Miscellaneous Function

Miscellaneous function refers to the operation to control the spindle, feed, and coolant. In general, it is specified by an M code.

When a move command and M code are specified in the same block, there are two ways to execute these commands:

1) Pre-M function
   M command is executed before the completion of move command

2) Post-M function
   M command is executed after the completion of move command.

The sequence of the execution depends on the specification of the machine tool builder.
1.8 Program Configuration

1.8.1 Structure of an NC Program

As it is shown in Figure 1.19, an NC program consists of a sequence of NC blocks. Each block is one of machining steps. Commands in each block are the instruction.

![Figure 1.19 Structure of an NC Program](image)

- Format of program name

  The program name must be specified in the format OXXXX (X could be letters or numbers).

- Format of program number

  The program number should be started with %XXXX or OXXXX (X could be numbers only).

- Format of blocks

  A block starts with the program block number.

![Figure 1.20 Structure of Block](image)
- Format of **end of program**
  The last block should contain M02 or M03 to indicate the end of program.
- Format of **Comments**
  All information after the “;” is regarded as comments.
  All information between “( )” is regarded as comments.

### 1.8.2 Main Program and Subprogram

There are two type of program: main program and subprogram. The CNC operates according to the main program. When a execution command of subprogram is at the execution line of the main program, the subprogram is called. When the execution of subprogram is finished, the system returns control to the main program.

![Diagram of Main Program and Subprogram](image)

**Figure 1.21 Main program and subprogram**

**Note:**
Main program and its subprogram must be written in a same file with a different program codes.
2 Preparatory Function (G code)

There are two types of G code: one-shot G code, and modal G code.

<table>
<thead>
<tr>
<th>Type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>One-shot G code</td>
<td>The G code is only effective in the block in which it is specified</td>
</tr>
<tr>
<td>Modal G code</td>
<td>The G code is effective until another G code is specified.</td>
</tr>
</tbody>
</table>

Example: G01 and G00 are modal G codes.

\[
\begin{align*}
N10 & \ G01 \ X100; \\
N20 & \ Y200 \ X200; \\
N30 & \ X300; \\
N40 & \ G00 \ Y100;
\end{align*}
\]

G01 is effective from N10 to N30
## 2.1 G code List

The following table is the list of G code in HNC system.

<table>
<thead>
<tr>
<th>G code</th>
<th>Group</th>
<th>function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td></td>
<td>Rapid positioning</td>
</tr>
<tr>
<td>G01</td>
<td>01</td>
<td>Linear interpolation</td>
</tr>
<tr>
<td>G02</td>
<td></td>
<td>Circular interpolation/Helical interpolation CW</td>
</tr>
<tr>
<td>G03</td>
<td></td>
<td>Circular interpolation/Helical interpolation CCW</td>
</tr>
<tr>
<td>G04</td>
<td>00</td>
<td>Dwell</td>
</tr>
<tr>
<td>G07</td>
<td>00</td>
<td>Virtual axis</td>
</tr>
<tr>
<td>G09</td>
<td>00</td>
<td>Exact stop</td>
</tr>
<tr>
<td>G17</td>
<td></td>
<td>XY plane selection</td>
</tr>
<tr>
<td>G18</td>
<td>02</td>
<td>ZX plane selection</td>
</tr>
<tr>
<td>G19</td>
<td></td>
<td>YZ plane selection</td>
</tr>
<tr>
<td>G20</td>
<td></td>
<td>Input in inches</td>
</tr>
<tr>
<td>G21</td>
<td>08</td>
<td>Input in metrics</td>
</tr>
<tr>
<td>G22</td>
<td></td>
<td>Input in impulses equivalent weight</td>
</tr>
<tr>
<td>G24</td>
<td>03</td>
<td>Programmable mirror image</td>
</tr>
<tr>
<td>G25</td>
<td></td>
<td>Programmable mirror image cancel</td>
</tr>
<tr>
<td>G28</td>
<td>00</td>
<td>Return to reference point</td>
</tr>
<tr>
<td>G29</td>
<td></td>
<td>Return from reference point</td>
</tr>
<tr>
<td>G34</td>
<td>00</td>
<td>Thread tapping</td>
</tr>
<tr>
<td>G38</td>
<td>00</td>
<td>Polar Coordinates</td>
</tr>
<tr>
<td>G40</td>
<td></td>
<td>Cutter compensation cancel</td>
</tr>
<tr>
<td>G41</td>
<td>09</td>
<td>Cutter compensation left</td>
</tr>
<tr>
<td>G42</td>
<td></td>
<td>Cutter compensation right</td>
</tr>
<tr>
<td>G43</td>
<td></td>
<td>Tool length compensation +direction</td>
</tr>
<tr>
<td>G44</td>
<td>10</td>
<td>Tool length compensation - direction</td>
</tr>
<tr>
<td>G49</td>
<td></td>
<td>Tool length compensation cancel</td>
</tr>
<tr>
<td>G50</td>
<td>04</td>
<td>Scaling cancel</td>
</tr>
<tr>
<td>G51</td>
<td></td>
<td>Scaling</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Machine coordinate system selection</td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>-------------------------------------</td>
</tr>
<tr>
<td>G53</td>
<td>00</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>---</td>
<td>---</td>
<td>---</td>
</tr>
<tr>
<td>G54</td>
<td></td>
<td>Workpiece coordinate system 1</td>
</tr>
<tr>
<td>G55</td>
<td>11</td>
<td>Workpiece coordinate system 2</td>
</tr>
<tr>
<td>G56</td>
<td></td>
<td>Workpiece coordinate system 3</td>
</tr>
<tr>
<td>G57</td>
<td></td>
<td>Workpiece coordinate system 4</td>
</tr>
<tr>
<td>G58</td>
<td></td>
<td>Workpiece coordinate system 5</td>
</tr>
<tr>
<td>G59</td>
<td></td>
<td>Workpiece coordinate system 6</td>
</tr>
<tr>
<td>G60</td>
<td>00</td>
<td>Single direction positioning</td>
</tr>
<tr>
<td>G61</td>
<td>12</td>
<td>Exact stop mode</td>
</tr>
<tr>
<td>G64</td>
<td></td>
<td>Cutting mode</td>
</tr>
<tr>
<td>G68</td>
<td>05</td>
<td>Coordinate rotation</td>
</tr>
<tr>
<td>G69</td>
<td></td>
<td>Coordinate rotation cancel</td>
</tr>
<tr>
<td>G73</td>
<td></td>
<td>High-speed drilling cycle</td>
</tr>
<tr>
<td>G74</td>
<td></td>
<td>Left-hand tapping cycle</td>
</tr>
<tr>
<td>G76</td>
<td></td>
<td>Fine boring cycle</td>
</tr>
<tr>
<td>G80</td>
<td></td>
<td>Canned cycle cancel</td>
</tr>
<tr>
<td>G81</td>
<td></td>
<td>Drilling cycle, Spot drilling</td>
</tr>
<tr>
<td>G82</td>
<td></td>
<td>Drilling cycle, Counter boring cycle</td>
</tr>
<tr>
<td>G83</td>
<td>06</td>
<td>Peck drilling cycle</td>
</tr>
<tr>
<td>G84</td>
<td></td>
<td>Tapping cycle</td>
</tr>
<tr>
<td>G85</td>
<td></td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G86</td>
<td></td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G87</td>
<td></td>
<td>Back boring cycle</td>
</tr>
<tr>
<td>G88</td>
<td></td>
<td>Manual Boring cycle</td>
</tr>
<tr>
<td>G89</td>
<td></td>
<td>Boring cycle</td>
</tr>
<tr>
<td>G90</td>
<td>13</td>
<td>Absolute command</td>
</tr>
<tr>
<td>G91</td>
<td></td>
<td>Increment command</td>
</tr>
<tr>
<td>G92</td>
<td>00</td>
<td>Setting for work coordinate system</td>
</tr>
<tr>
<td>G94</td>
<td>14</td>
<td>Feed per minute</td>
</tr>
<tr>
<td>G95</td>
<td></td>
<td>Feed per rotation</td>
</tr>
<tr>
<td>G98</td>
<td>15</td>
<td>Return to initial point in canned cycle</td>
</tr>
<tr>
<td>G99</td>
<td></td>
<td>Return to R point in canned cycle</td>
</tr>
</tbody>
</table>
Explanation:

1) G codes in 00 group are one-shot G code, while the other groups are modal G code.

2) △ means that it is default setting.

3) Multiple G codes from different groups can be specified in the same block. If multiple G codes from the same group are specified in the same block, only the last G code specified is valid.
3 Interpolation Functions

This chapter will introduce:

1) Positioning Command (G00)
2) Single Direction Positioning (G60)
3) Linear Interpolation (G01)
4) Circular Interpolation (G02, G03)
5) Helical Interpolation (G02, G03)
6) Thread Tapping (G34)
3.1 Positioning (G00)

**Programming**

G00 X_Y_Z_A_

**Explanation of the parameters**

X, Y, Z, A  Coordinate value of the end point in the absolute command or incremental command

**Function**

The tool is moved at the highest possible speed (rapid traverse). If the rapid traverse movement is required to execute simultaneously on several axes, the rapid traverse speed is decided by the axis which takes the most time. Thus, the tool path is nonlinear. The operator can use this function to position the tool rapidly, to travel around the workpiece, or to approach the tool change position.

**Example**

Move tool from A (20, 15) to B (90, 45) at the rapid traverse speed.

![Non linear interpolation positioning](image)

Figure 3.1 Positioning (Rapid Traverse)

Absolute programming:

G00 X90 Z45

Incremental programming:

G00 X70 Y30
3.2 Single Direction Positioning (G60)

**Programming**

G60 X_Y_Z_A_

**Explanation of the parameters**

X, Y, Z, A  Coordinate value of the end point in the absolute command or incremental command

**Function**

At first, move the tool from the start point to the intermediate point at the rapid traverse speed. Then, tool is moved from the intermediate point to the end point at the specified feedrate.

![Diagram of Single Direction Positioning (G60)](image)

**Note:**

The direction and distance from the intermediate point to the end point are set by machine parameter – single direction positioning offset. When the value of the parameter is less than 0, the direction is negative. When the value of the parameter is more than 0, the direction is positive.
3.3 Linear Interpolation (G01)

Programming
G01 X_Y_Z_A_F_

Explanation of the parameters
X, Y, Z, A   Coordinate value of the end point in the absolute command or incremental command
F           Feedrate. It is effective until a new value is specified.

Function
The tool is moved along the straight line at the specified feedrate.

Example 1
Move tool from A (20, 15) to B (90, 45) at the rapid traverse speed.

![Diagram of linear interpolation between points A and B]

Figure 3.3 Linear Interpolation – Example 1

Absolute programming
G01 X90 Y45 F800

Incremental programming
G01 X70 Y30 F800
**Example 2**

Use the tool (Φ8) to machine a groove (3mm) on a workpiece.

![Figure 3.4 Linear Interpolation – Example 2](image)

%3308 (the origin is on A)  %3309 (the origin is on B)
N1 G92 X0 Y0 Z50        N1 G92 X0 Y0 Z50
N2 M03 S500              N2 M03 S500
N3 G00 X-31 Y-26         N3 G00 X19 Y14
N4 Z5                    N4 Z5
N5 G01 Z-3 F40           N5 G01 Z-3 F40
N6 Y26 F100              N6 Y66 F100
N7 X31                   N7 X81
N8 Y-26                  N8 Y14
N9 X-31                  N9 X19
N10 G00 Z50              N10 G00 Z50
N11 X0 Y0                N11 X0 Y0
N12 M05                  N12 M05
N13 M30                  N13 M30
3.4 Circulation Interpolation (G02, G03)

Programming

\[
\begin{align*}
G17 & \quad \begin{bmatrix} G02 \\ G03 \end{bmatrix} \begin{bmatrix} X \\ Y \end{bmatrix} = \begin{bmatrix} I \\ J \end{bmatrix}, F_-
\end{align*}
\]

\[
\begin{align*}
G18 & \quad \begin{bmatrix} G02 \\ G03 \end{bmatrix} \begin{bmatrix} X \\ Z \end{bmatrix} = \begin{bmatrix} I \\ K \end{bmatrix}, F_-
\end{align*}
\]

\[
\begin{align*}
G19 & \quad \begin{bmatrix} G02 \\ G03 \end{bmatrix} \begin{bmatrix} Y \\ Z \end{bmatrix} = \begin{bmatrix} J \\ K \end{bmatrix}, F_-
\end{align*}
\]

Explanation of the parameters

G17 The working plane is XY, and the infeed direction is Z
G18 The working plane is XZ, and the infeed direction is Y
G19 The working plane is YZ, and the infeed direction is X
G02 a circular path in clockwise direction (CW) (Figure 3.5)
G03 a circular path in counterclockwise direction (CCW)

G02 and G03 are defined when the working plane is specified. Figure 3.5 shows the direction of circular interpolation.

![Figure 3.5 Direction of circular interpolation](image)

X, Y/X, Z/Y, Z For an absolute command, the coordinate values of the circle end point in the specific working plane. For an incremental command, the coordinate values of the circle end point with reference to the circle starting point in the specific working plane.
I, J/I, K/J, K  Coordinate values of the circle center point with reference to the circle starting point in incremental command. (Figure 3.6)

![Figure 3.6 Distance from the start point to the circle centre point](image)

R  Circle radius. When the arc is less than 180° (minor arc), R is positive. If the arc is more than 180° (major arc), R is negative.

F  Feedrate along the circle

**Function**

The tool is moved along a full circle or arcs.

**Note:**

1) When it is full circle programming, R can not be used in the program. I, J, K can only be used in this case.

2) When it is not full circle programming, the operator can select R or I, J, K to program. If I, J, K, and R addresses are all specified in the program, R takes precedence and the other are ignored.
Example 1

Use G02 to program the minor arc a and the major arc b.

(i) Arc a
G91 G02 X30 Y30 R30 F300
G91 G02 X30 Y30 I30 J0 F300
G90 G02 X0 Y30 R30 F300
G90 G02 X0 Y30 I30 J0 F300

(ii) Arc b
G91 G02 X30 Y30 R-30 F300
G91 G02 X30 Y30 I0 J30 F300
G90 G02 X0 Y30 R-30 F300
G90 G02 X0 Y30 I0 J30 F300
Example 2

Use G02/G03 to program the full circle.

![Diagram of circular interpolation]

Figure 3.8 Circular Interpolation – Example 2

i) Clockwise circle from A to A

G90 G02 X30 Y0 I-30 J0 F300  
G91 G02 X0 Y0 I-30 J0 F300

(ii) Counterclockwise circle from B to B

G90 G03 X0 Y-30 I0 J30 F300  
G91 G03 X0 Y0 I0 J30 F300
Example 3

Use the tool (Φ8) to machine a groove (3mm) on a workpiece.

```
%3314
N1 G92 X0 Y0 Z50
N2 M03 S500
N3 G00 X10 Y30
N4 Z5
N5 G01 Z-3 F40
N6 X30
N7 G02 X38.66 Y25 R10
  (N7 G02 X38.66 Y25 J-10)
N8 G01 X47.32 Y10
N9 G02 X30 Y-20 R20
  (N9 G02 X30 Y-20 J-10 I-17.32)
N10 G01 X0
N11 G02 X0 Y20 R20
  (N11 G02 X0 Y20 J20)
N12 G03 X10 Y30 R10
  (N13 G03 X10 Y30 J10)
N14 G00 Z50
N15 X0 Y0
N16 M30
```
**Example 4**

Use the tool (Φ8) to machine a groove (3mm) on a workpiece.

![Figure 3.10 Circular Interpolation – Example 4](image)

```latex
\begin{verbatim}
\%3315
N1 G92 X0 Y0 Z50
N2 M03 S500
N3 G00 X-25 Y-8.66
N4 Z5
N5 G01 Z-3 F40
N6 G02 X-25 Y8.66 R10
N7 G01 X-10 Y17.32
N8 G02 X-10 Y-17.32 R-20
N9 G01 X-25 Y-8.66
N10 G00 Z50
N11 X0 Y0
N12 M05
N13 M30
\end{verbatim}
```

### 3.5 Helical Interpolation (G02, G03)
Programming

G17 \[ \begin{align*}
    &G02 \\
    &G03
\end{align*} \]
X Y I J Z F L

G18 \[ \begin{align*}
    &G02 \\
    &G03
\end{align*} \]
X Z I K R Y F L

G19 \[ \begin{align*}
    &G02 \\
    &G03
\end{align*} \]
Y Z J K R X F L

Explanation of the parameters

G17 The working plane is XY, and the infeed direction is Z
G18 The working plane is XZ, and the infeed direction is Y
G19 The working plane is YZ, and the infeed direction is X
G02 a circular path in clockwise direction (CW) (Figure 3.5)
G03 a circular path in counterclockwise direction (CCW)

X, Y/X, Z/Y, Z For an absolute command, the coordinate values of the circle end point in the specific working plane. For an incremental command, the coordinate values of the circle end point with reference to the circle starting point in the specific working plane.
I, J/I, K/J, K Coordinate values of the circle center point with reference to the circle starting point in incremental command.
R Circle radius. When the arc is less than 180° (minor arc), R is positive. If the arc is more than 180° (major arc), R is negative.
Z, Y, X The coordinate value of the end point with reference to the starting point on the third axis in the incremental command.
F Feedrate along the circle
L Number of circles on a workpiece

Figure 3.11 Helical Interpolation (G02, G03)

Function

Helical interpolation can be used to manufacture threads on the workpiece.
**Example 1**

Use G03 to program.

![Helical Interpolation – Example 1](image)

**Absolute programming**

G90 G17 F300

G03 X0 Y30 R30 Z10

**Incremental programming**

G91 G17 F300

G03 X-30 Y30 R30 Z10
Example 2

Use the tool (Φ10mm) to machine a hole (the diameter is 50mm, and the height is 10mm) on a workpiece.

![Figure 3.13 Helical Interpolation – Example 2]

%3317
N1 G92 X0 Y0 Z30
N2 G01 Z11 X20 F200
N3 G91 G03 I-20 Z-1 L11
N4 G03 I-20
N5 G90 G01 X0
N6 G00 Z30
N7 X30 Y-50
N8 M30
3.6 Virtual Axis (G07) and Sine Interpolation

Programming
G07 X_Y_Z_A

Explanation of the parameters
X, Y, Z, A One of axes is set as the virtual axis.
If it is set to 0, then that axis is the virtual axis. If it is set to 1, then that axis is the actual axis.

Function
G07 command can be used with helical interpolation command (G02, G03). The operation combined G07 and G02/G03 is called sine interpolation.

Note
The tool would not be moved along the virtual axis.

Example 1
Use G03 to program

G90 G00 X-50 Y0 Z0
G07 X0 G91
G03 X0 Y0 I10 J50 Z60 F800
Example 2

To implement the sine interpolation on the working plane XY.

\[ Z \times Z + Y \times Y = R \times R \] (R: radius)

\[ Y = R \sin(2\pi \times X/L) \] (L: the distance on Z axis for each cycle)

\[ X \]
\[ Y \]
\[ Z \]

\[ R \]

\[ 0 \]
\[ 5 \]

Figure 3.15 Sine Interpolation – Example 2

%3319
N01 G92 X0 Y0 Z0
N02 G07 Z0
N03 G19 G90 G03 Y.0 Z0 J5 K0 X20.0 F100
N04 G07 Z1
N05 M30
3.7 Tapping (G34)

Programming

G34 K F P

Explanation of the parameters

K The distance from the starting point to the bottom of the hole
F Thread lead. If it is positive, the spindle turns clockwise during tapping. If it is negative, the spindle turns counterclockwise during tapping.
P Dwell time at the bottom of a hole. (The unit is seconds.)

Function

With this command, the operator can rigid tap a thread.

Note

1) When the spindle turns clockwise during tapping, the spindle would turn counterclockwise during retraction.
2) When the spindle turns counterclockwise during tapping, the spindle would turn clockwise during retraction.

In general, there is overshoot of the tap at the bottom of the thread during the spindle-braking portion of the tapping cycle. It can be set by PMC parameters (Table 3-1) to eliminate the overshoot errors.
Table 3  1 PMC parameters

<table>
<thead>
<tr>
<th>CNC system</th>
<th>PMC parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>HNC 18/19i</td>
<td>#0062  Maximum spindle speed during tapping</td>
</tr>
<tr>
<td></td>
<td>#0063  Minimum spindle speed during tapping</td>
</tr>
<tr>
<td></td>
<td>#0064  Dwell unit for tapping</td>
</tr>
<tr>
<td></td>
<td>#0065  Optional dwell unit for tapping</td>
</tr>
<tr>
<td>HNC 21/22</td>
<td>#0017  Maximum spindle speed during tapping</td>
</tr>
<tr>
<td></td>
<td>#0018  Minimum spindle speed during tapping</td>
</tr>
<tr>
<td></td>
<td>#0019  Dwell unit for tapping</td>
</tr>
<tr>
<td></td>
<td>#0030  Optional dwell unit for tapping</td>
</tr>
</tbody>
</table>

Optional dwell unit for tapping is only effective when “dwell unit for tapping” is assigned to “0”. Moreover, it is not necessary to restart the system.

The following formular is to calculate the dwelled unit (X):

\[ D = \left( S \times S / C \right) \times X / 10000 = L \times 360 / F \]

- D  dwell amount
- S  spindle speed
- C  Transmission gear ratio
- X  dwell unit
- L  overshoot error
- F  thread lead
Example

Use G34 to program.

%0002
G92 X-20 Y-20 Z50
M03 S200
G00 X20 Y12
Z5
G34 K-27 F1.5
G00 X100
G34 K-27 F1.5
G00 Z50
X-20 Y-20
M05
M30
4 Feed Function

This chapter would introduce:

1) Rapid Traverse
   The tool is moved at the rapid traverse speed set in CNC.

2) Cutting Feed
   The tool is moved at the programmed cutting feedrate.

3) Dwell

4) Exact Stop

5) Cutting Mode
4. Feed Function

4.1 Rapid Traverse (G00)

Positioning command (G00) is to move the tool at the rapid traverse speed (the highest possible speed).

This rapid traverse speed can be controlled by the machine control panel. For more detailed information, please refer to turning operation manual.
4.2 Cutting Feed (G94, G95)

**Programming**

G94 [F_ ]
G95 [F_ ]

**Explanation of the parameters**

G94 feedrate per minute.

On linear axis, the unit of feedrate is mm/min, or in/min.

On rational axis, the unit of feedrate is degree/min.

G95 feedrate per revolution

The unit of feedrate is mm/rev, or in/rev.

**Note:**

1) G94 is the default setting

2) G95 is only used when there is spindle encoder.

**Function**

The feedrate can be set by G94 or G95.
4.3 Dwell (G04)

**Programming**

G04 P_

**Explanation of the parameters**

P dwell time (specified in seconds)

**Function**

It can be used to interrupt machining to get the smooth surface. It can be used to control the groove cutting, drilling, and turning path.

**Example**

Use G04 to get the smooth surface.

```plaintext
%0004
G92 X0 Y0 Z0
G91 F200 M03 S500
G43 G01 Z-6 H01
G04 P5
G49 G00 Z6 M05 M30
```

![Figure 4.1 Dwell – Example](image)
4.4 Exact Stop (G09, G61)

Programming
G09
G61

Explanations of the parameters
The tool is moved to the end point of a block, then the position of the end point is checked. Then, the next block is proceeded.

![Position check diagram](image)

Figure 4.2 Exact Stop (G09/G61) – tool path from block (1) to block (2)

The difference between G09 and G61 is that G09 is one-shot G code. And G61 is modal G code.

Function
G09 or G61 can be used to machine a sharp edge.
Example

Use G61 to program.

Figure 4.3 Exact Stop - Example

%0061
G92 X0 Y0 Z0
G91 G00 G43 Z-10 H01
G41 X50 Y20 D01
G01 G61 Y80 F300
X100
...

49
4.5 Cutting Mode (G64)

**Programming**

G64

**Explanation of the parameters**

The tool is moved to the end point of a block. Then, the next block is proceeded. The tool path is shown in the following figure.

![Figure 4.4 Cutting Mode (G64) – tool path from block (1) to block (2)]

**Function**

G64 command can make the tool move smoothly between two blocks.
**Example**

Use G64 to program.

![Graph showing cutting mode example](image)

Figure 4.5 Cutting Mode – Example

%0064
G92 X0 Y0 Z0
G91 G00 G43 Z-10 H01
G41 X50 Y20 D01
G01 G64 Y80 F300
X100
...

...
5 Coordinate System

This chapter would introduce:

1) Reference Position Return (G28)
2) Auto Return from Reference Position (G29)
3) Setting a Workpiece Coordinate System (G92)
4) Selecting a Machine Coordinate System (G53)
5) Selecting a Workpiece Coordinate System (G54–G59)
6) Plane Selection (G17, G18, G19)
7) Absolute and Incremental Programming (G90, G91)
8) Dimension Selection (G20, G21, G22)
9) Polar Coordinates (G38)
5.1 Reference Position Return (G28)

Programming
G28 X_Y_Z_A_

Explanation of the parameters
X, Y, Z, A  Coordinate values of the intermediate point in absolute command/incremental command

Function
The tool is moved to the intermediate point rapidly, and then returned to the reference point.

![Diagram of Reference Position Return (G28)](image)

Figure 5.1 Reference Position Return (G28)

Note:
1) In general, G28 is used to change tools or cancel the mechanical error. Tool radius compensation and tool length compensation should be cancelled when G28 is executed.
2) G28 can not only make the tool move to the reference point, but also can save the intermediate position to be used in G29.
3) When the power is on and manual reference position return is not available, G28 is same as the manual reference position return. The direction of this reference position return (G28) is set by the axis parameter – reference approach direction.
4) G28 is one-shot G code.
5.2 Auto Return from Reference Position (G29)

Programming
G29 X_ Y_ Z_ A_

Explanation of the parameters
X, Y, Z, A Coordinate value of the end point in absolute command/incremental command

Function
The tool is moved rapidly from the intermediate point defined in G28 to the end point. Thus, G29 is generally used after G28 is defined.

![Diagram of Auto Return from Reference Position (G29)]

Figure 5.2 Auto Return from Reference Position (G29)

Note:
G29 is one-shot G code.
**Example**

Use G28, G29 command to program the track shown in. It moves from the starting point A to the intermediate point B, and then returns to the reference point R. At last, it moves from the reference point R to the end point C through the intermediate point B.

![Diagram of Reference Position – Example](image)

```
G91 G28 X100 Y20 ;A→B→R
M06 T02 ;Changing the tool
G29 X50 Y-40 ;R→B→C
```

...
5.3 Setting a Workpiece Coordinate System (G92)

Programming
G92 X_ Y_ Z_ A_

Explanation of the parameters
X, Y, Z, A  Coordinate values of the tool position in the workpiece coordinate system.

Functions
G92 can set a workpiece coordinate system based on the current tool position (X_ Y_ Z_ A_).

Example
Use G92 to set a workpiece coordinate system.

![Figure 5.4 Setting a Workpiece Coordinate System – Example](image)

G92 X30.0 Y30.0 Z20.0
5.4 Selecting a Machine Coordinate System (G53)

**Programming**

G53 X_ Y_ Z_ A_

**Explanation of the parameters**

X, Y, Z, A  Absoulte coordinate values of a point in the machine coordinate system.

**Function**

A machine coordinate system is selected, and the tool moves to the position at the rapid traverse speed.

**Note:**

1) Absolute values must be specified in G53. The incremental values would be ignored by G53.

2) G53 is one-shot G code.
5.5 Selecting a Workpiece Coordinate System (G54~G59)

Programming

\[
\begin{align*}
G54 \\
G55 \\
G56 \\
G57 \\
G58 \\
G59 \\
X\_Y\_Z\_A_ \\
\end{align*}
\]

Explanation of the parameters

X, Y, Z, A  Coordinate values of the point with reference to the origin of machine in absolute command

Function

There are six workpiece coordinate system to be selected. If one coordinate system is selected, the tool is moved to a specified point.

Note:

1) The workpiece coordinate system must be set before using these commands (G54~G59). The workpiece coordinate system can be set by using the MDI panel. For detailed information, please refer to the milling operation manual.

2) Reference position must be returned before these commands (G54~G59) are executed.

3) G54 is the default setting.
Example

Select one of workpiece coordinate system, and the tool path is Current point→A→B.

Figure 5.5 Workpiece Coordinate System – Example

%1000
N01 G54 G00 G90 X30 Y40
N02 G59
N03 G00 X30 Y30
N04 G54
N05 X0 Y0
N06 M30
5.6 Plane Selection (G17, G18, G19)

**Programming**

G17
G18
G19

**Explanation of the parameters**

G17  working plane is XY, infeed direction is Z
G18  working plane is ZX, infeed direction is Y
G19  working plane is YZ, infeed direction is X

**Function**

The working plane is specified and used for tool radius compensation and circular interpolation.

**Note:**

Move command is not related with the plane selection. For example, in the command G17 G01 Z10, Z axis does still move.
5.7 Absolute and Incremental Programming (G90, G91)

**Programming**

G90 X_ Y_ Z_ A_

G91 X_ Y_ Z_ A_

**Explanation of the parameters**

G90 Absolute programming

X, Y, Z, A  Coordinate values of the point with reference to the origin of programming

G91 Incremental programming

X, Y, Z, A  Coordinate values of the point with reference to the previous position

**Function**

The tool is moved to the specified position.
5. Coordinate System

**Example**

Move the tool from point 1 to point 2 through point 3, and then return to the current point.

![Figure 5.6 Absolute and Incremental Programming – Example](image)

G90 programming

`%0001
M03 S500
N01 G92 X0 Y0 Z10
N02 G01 X20 Y15
N03 X40 Y45
N04 X60 Y25
N05 X0 Y0 Z10
N06 M30`

G91 programming

`%0001
M03 S500
N01 G92 X0 Y0 Z10
N02 G91 G01 X20 Y15
N03 X20 Y30
N04 X20 Y-20
N05 G90 X0 Y0
N06 M30`
5.8 Dimension Selection (G20, G21, G22)

Programming

G20
G21
G22

Explanation of the parameters

G20: Inch input
G21: Metric input
G22: Impulses equivalent weight input

The units of linear axis and circular axis are shown in the following table

<table>
<thead>
<tr>
<th></th>
<th>Linear axis</th>
<th>Circular axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inch system (G20)</td>
<td>Inch</td>
<td>Degree</td>
</tr>
<tr>
<td>Metric system (G21)</td>
<td>Mm</td>
<td>Degree</td>
</tr>
<tr>
<td>Pulse system (G22)</td>
<td>Impulses equivalent weight</td>
<td>Impulses equivalent weight</td>
</tr>
</tbody>
</table>

Function

Depending on the part drawing, the workpiece geometries can be programmed in metric measures, inches, or impulses equivalent weight.
5.9 Polar Coordinates

Programming

G38 X_ Y_
G01 AP=_ RP=_
G02/G03 AP=_ RP=_ R_

Explanation of the parameters

G38 Setting a polar coordinate system
X, Y Coordinate value of the pole in the workpiece coordinate system
AP Polar angle
RP Polar radius
R Circle radius

Function

The polar coordinate method is useful only if there is radius and angle measurements on a workpiece.

Note

These commands can be used with commands of workpiece coordinate system.
Example 1

Use polar coordinates command to program.

```
%3326
G92 X0 Y0 Z10
G00 X-50 Y-60
G00 Z-3
G01 G41 X-42 D01 F1000
Y0
G38 X0 Y0
G02 AP=0 RP=42 R42
G01 Y-50
X-50
G00 G40 Y-60
Z10
G00 X0 Y0
M30
```

Figure 5.7 Polar Coordinates – Example 1
Example 2

When the tool is turning clockwise, the polar radius increases 2mm as the polar angle increases 10°.

![Figure 5.8 Polar Coordinate – Example 2](image)

%0001
G54 G00 X-15 Y-15 Z10
G00 Z-3
G01 G41 X0 D01 F1000
Y50
G38 X42 Y50
#0=180
#1=42
while #0 gt 0
G01 AP=[#0] RP=[#1]
#0=#0-10
#1=#1+2
Endw
G01 AP=0 RP=78
Y0
X-15
G00 G40 Y-15
Z10
M30
6 Spindle Speed Function

Spindle function controls the spindle speed (S), the unit of spindle speed is r/min. S is modal G code command; it is only available when the spindle is adjustable. Spindle speed programmed by S code can be adjusted by overrides on the machine control panel.
7 Tool Function

This chapter would introduce:

1) Too selection and Tool offset (T code)

2) Tool radius compensation (G40, G41, G42)
7.1 Tool Selection and Tool Offset (T code)

**Programming**

T  XX  XX

**Explanation of the parameters**

XX  Tool number (two digits). The number of tool depends on manufacture’s configuration.
XX  Tool offset number (two digits). It corresponds to the specific compensation value.

**Functions**

To select the desired tool, T command makes the turret turn, selects a cutter, and calls the compensation value.

**Note:**

1) T command is only effective when it is used with tool move command, such as G00.
2) When T command and tool move command are in the same program block, T command is executed at first.
3) The same tool can have different compensation values. For example, T0101, T0102, T0103 are possible.
4) Different tool can have same compensation values. For example, T0101, T0201, and T0301 are possible.
7.2 Tool Radius Compensation (G40, G41, G42)

Programming

\[
\begin{cases}
G17 \\
G18 \\
G19
\end{cases}
\begin{cases}
G40 \\
G41 \\
G42
\end{cases}
\begin{cases}
G00 \\
G01
\end{cases}
X \_\_Y\_Z\_D\_
\]

Explanation of the parameters

G17 Tool radius compensation on plane XY

G18 Tool radius compensation on plane ZX

G19 Tool radius compensation on plane YZ

G40 Deactivate tool radius compensation

G41 Activate tool radius compensation, tool operates in machining operation to the left of the contour.

G42 Activate tool radius compensation, tool operates in machining operation to the right of the contour.

(a) Cutter compensation left

(b) Cutter compensation right

Figure 7.1 Tool Radius Compensation

X, Y, Z Coordinate values of the end point. It is the point where the tool radius compensation is activated or deactivated.

D There are two ways to specify the value of D.

- D01–D99 Each code corresponds to the different values of the tool radius compensation.
- #100–#199 Variable of radius compensation
Function

These commands can control the tool radius compensation to get the equidistant tool paths for different tools.

Note:

1) G40, G41, and G42 must be used with G00 or G01.
2) Changing the plane of tool radius compensation can only be done when there is no compensation.
**Example 1**

Use the tool radius compensation, and program for the part shown in Figure 7.2. The dashed line stands for the actual tool path.

![Figure 7.2 Tool Radius Compensation – Example 1](image)

```
%3322
G92 X–10 Y–10 Z50
G90 G17
G42 G00 X4 Y10 D01
Z2 M03 S900
G01 Z–10 F800
X30
G03 X40 Y20 I10 J10
G02 X30 Y30 I10 J10
G01 X10 Y20
Y5
G00 Z50 M05
G40 X–10 Y–10
M02
```
Example 2
Use the tool (diameter is $\Phi 8$). The depth of cutting is 3mm.

![Figure 7.3 Tool Radius Compensation – Example 2](image)

%3323
N1 G92 X-40 Y50 Z50
N2 M03 S500
N4 G01 Z-3 F400
N5 G01 G41 X5 Y30 D01 F40
N6 X30
N7 G02 X38.66 Y25 R10
  (N7 G02 X38.66 Y25 J-10)
N8 G01 X47.32 Y10
N9 G02 X30 Y-20 R20
  (N9 G02 X30 Y-20 I-17.32 J-10)
N10 G01 X0
N11 G02 X0 Y20 R20
  (N11 G02 X0 Y20 J20)
N12 G03 Y40 R10
  (N12 G03 Y40 J10)
N13 G00 G90 G40 X-40 Y50
N14 G00 Z50
N15 M30
Example 3

Use the tool (diameter is $\Phi 8$). The depth of cutting is 3mm.

Figure 7.4 Tool Radius Compensation – Example 3

%3322 (female die)  %3323 (male die)
N1 G92 X-10 Y-10 Z50  N1 #101=4
N2 M03 S500  N2 G92 X-10 Y-10 Z50
N3 Z5  N3 M03 S500
N4 G00 X25 Y20  N4 Z5
N5 G01 Z-3 F40  N5 G01 Z-3 F40
N6 G41 Y30 D01 f100  N6 G41 X15 D101 f100
N7 G03 Y10 R10  N7 Y60
N8 G01 X75  N8 G02 X25 Y70 R10
N9 G03 X85 Y20 R10  N9 G01 X75
N10 G01 Y60  N10 G02 X85 Y60 R10
N11 G03 X75 Y70 R10  N11 G01 Y20
N12 G01 X25  N12 G02 X75 Y10 R10
N13 G03 X15 Y60 R10  N13 G01 X25
N14 G01 Y20  N14 G02 X15 Y20 R10
N15 G03 X23 Y12 R8  N15 G01 Z10
N16 G01 Z10  N16 G00 G40 X0 Y0
N17 G00 G40 X25 Y20  N17 G0 Z50
N18 G0 Z50  N18 M30
N19 M30
7.3 Tool Length Compensation (G43, G44, G49)

**Programming**
\[
\begin{align*}
  \{ & G17 \} \quad \{ & G43 \} \quad \{ & G00 \} \quad X \_ Y \_ Z \_ H \\
  \{ & G18 \} \quad \{ & G44 \} \quad \{ & G01 \} \\
  \{ & G19 \} \quad \{ & G49 \} \\
\end{align*}
\]

**Explanation of the parameters**
- G17  XY plane selection (compensate for the difference in tool length along Z axis)
- G18  ZX plane selection (compensate for the difference in tool length along Y axis)
- G19  YZ plane selection (compensate for the difference in tool length along X axis)
- G43  Positive offset
- G44  Negative offset
- G49  Deactivate the tool length compensation
- X, Y, Z  Coordinate value of the end point
- H  H00–H99: Each code corresponds to the different values of the tool length compensation.

**Function**
These command can compensate the difference between the assumed tool length in the programming and the actual tool length.

![Diagram of Tool Length Compensation](image)

Figure 7.5 Tool Length Compensation (G43, G44, G49)
Example

Use the tool length compensation function to program.

Figure 7.6 Tool Length Compensation - Example

%1050
G92 X0 Y0 Z0
G91 G00 X120 Y80 M03 S600
G43 Z-32 H01
G01 Z-21 F300
G04 P2
G00 Z21
X30 Y-50
G01 Z-41
G00 Z41
X50 Y30
G01 Z-25
G04 P2
G00 G49 Z57
X-200 Y-60
M05 M30
7.4 RTCP (Rotation Tool Center Point Programming)

RTCP (Rotation Tool Center Point Programming) refers to the auto tool length compensation when the spatial orientation of the tool changes.

G01 (linear interpolation), G00 (rapid positioning), and G02/G03 (circular interpolation) can be used in the rotation tool center point programming.

G43, G44, G49 can also be used for the tool length compensation.
8 Miscellaneous Function

As it is mentioned in Chapter 1.8, there are two ways of execution when a move command and M code are specified in the same block.

1) Pre-M function
   M command is executed before the completion of move command.

2) Post-M function
   M command is executed after the completion of move command

There are two types of M code: one-shot M code, and modal M code.

<table>
<thead>
<tr>
<th>Type</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>One-shot M code</td>
<td>The M code is only effective in the block in which it is specified</td>
</tr>
<tr>
<td>Modal M code</td>
<td>The M code is effective until another M code is specified.</td>
</tr>
</tbody>
</table>
8.1 M code List

The following is a list of M command.

<table>
<thead>
<tr>
<th>CNC M-function</th>
<th>Type of Mode</th>
<th>Function</th>
<th>Pre/Post-M function</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>One-shot</td>
<td>Program stop</td>
<td>Post-M function</td>
</tr>
<tr>
<td>M01</td>
<td>One-shot</td>
<td>Optional stop</td>
<td>Post-M function</td>
</tr>
<tr>
<td>M02</td>
<td>One-shot</td>
<td>End of program</td>
<td>Post-M function</td>
</tr>
<tr>
<td>M30</td>
<td>One-shot</td>
<td>End of program with return to the beginning of program</td>
<td>Post-M function</td>
</tr>
<tr>
<td>M98</td>
<td>One-shot</td>
<td>Calling of subprogram</td>
<td>Post-M function</td>
</tr>
<tr>
<td>M99</td>
<td>One-shot</td>
<td>End of subprogram</td>
<td>Post-M function</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>PLC M-function</th>
<th>Type of Mode</th>
<th>Function</th>
<th>Pre/Post-M function</th>
</tr>
</thead>
<tbody>
<tr>
<td>M03</td>
<td>Modal</td>
<td>Spindle forward rotation</td>
<td>Pre-M function</td>
</tr>
<tr>
<td>M04</td>
<td>Modal</td>
<td>Spindle reverse rotation</td>
<td>Pre-M function</td>
</tr>
<tr>
<td>M05</td>
<td>Modal</td>
<td>▲ Spindle stop</td>
<td>Post-M function</td>
</tr>
<tr>
<td>M06</td>
<td>One-shot</td>
<td>Tool Selection</td>
<td>Post-M function</td>
</tr>
<tr>
<td>M07</td>
<td>Modal</td>
<td>Number1 Coolant on</td>
<td>Pre-M function</td>
</tr>
<tr>
<td>M08</td>
<td>Modal</td>
<td>Number2 Coolant on</td>
<td>Pre-M function</td>
</tr>
<tr>
<td>M09</td>
<td>Modal</td>
<td>▲ Coolant off</td>
<td>Post-M function</td>
</tr>
</tbody>
</table>

▲: default setting
8.2 CNC M-Function

8.2.1 Program Stop (M00)

M00 is one-shot M function, and it is post-M function. The program can be stopped, so that the operator could measure the tool and the part, adjust part and change speed manually, and so on. When the program is stopped, the spindle is stopped and the coolant is off. All of the current modal information remains unchanged. Resuming program could be executed by pushing “Cycle Run” button on the machine control panel.

8.2.2 Optional Stop (M01)

M01 is one-shot M function, and it is post-M function. Similarly to M00, M01 can also stop the program. All of the modal information is maintained. The difference between M00 and M01 is that the operator must press M01 button ( ) on the machine control panel. Otherwise, the program would not be stopped even if there is M01 code in the program.

8.2.3 End of Program (M02)

M02 is one-shot M function, and it is post-M function. When M02 is executed, spindle, feed and coolant are all stopped. It is usually at the end of the last program block. To restart the program, press “Cycle Run” button on the operational panel.

8.2.4 End of Program with return to the beginning of program (M30)

M30 is one-shot M function, and it is post-M function. Similarly to M02, M30 can also stop the program. The difference is that M30 returns control to the beginning of program. To restart the program, press “Cycle Run” button on the operational panel.
8.2.5 Subprogram Control (M98, M99)

- End of Subprogram (M99)
M99 indicates the end of subprogram and returns control to the main program. It is one-shot M function, and it is post-M function.

- Calling a Subprogram (M98)
  
  M98 P L
  
  P  program number of the subprogram
  L  repeated times of subprogram

M98 is used to call a subprogram. It is one-shot M function. Moreover, it is post-M function.
8.3 PLC M Function

8.3.1 Spindle Control (M03, M04, M05)

M03 starts spindle to rotate CW at the set speed set in the program.
M04 starts spindle to rotate CCW at the set speed in the program.
M05 stops spindle.
M03, M04 are modal M code, and they are pre-M function. M05 is modal M code, and it is post-M function. M05 is the default setting.

8.3.2 Tool Selection (M06)

M06 can select a desired tool to set on the spindle.
For example, M06 T01; the tool No.01 is selected.
M06 is one-shot M code, and it is post-M function.

8.3.3 Coolant Control (M07, M08, M09)

M07, M08 can turn on the coolant.
M09 can turn off the coolant.
M07 and M08 are modal M code, and they are pre-M function. M09 is one-shot M code, and it is post-M function. Moreover, M09 is the default setting.
9 Functions to Simplify Programming

This chapter would introduce:

1) Mirror Image (G24, G25)
2) Scaling (G50, G51)
3) Coordinate System Rotation (G68, G69)
4) Canned Cycle
9.1 Mirror Image (G24, G25)

Programming
G24 X_Y_Z_A_
M98 P_
G25 X_Y_Z_A_

Explanation the parameters
G24 Activate a mirror image
X, Y, Z, A Position and axis of symmetry for producing a mirror image
M98 P_ The sequence number of subprogram for producing a image on a part
G25 Deactivate a mirror image
X, Y, Z, A Axis of symmetry for producing a mirror image

Function
These commands can be used to mirror workpiece shapes on coordinate axes.

![Figure 9.1 Mirror Image](image)

Note
When there is only one axis of symmetry, the movement of tool for the mirror image is opposite to the tool movement of origin image.
Example

Use the mirror image function to machine a workpiece. The distance from the tool tip to the workpiece is 100mm. The depth of cutting is 5mm.

![Figure 9.2 Mirror Image - Example](image)

%0024 ; Main program
G92 X0 Y0 Z0
G91 G17 M03 S600
M98 P100 ; Machining part ①
G24 X0 ; The symmetry axis is Y-axis.
 ; The position of symmetry is X=0
M98 P100 ; Machining part ②
G24 Y0 ; The symmetry axes are X-axis and Y-axis.
 ; The position of symmetry is (0, 0)
M98 P100 ; Machining part ③
G25 X0 ; The symmetry Y-axis is cancelled.
 ; X-axis is still valid
M98 P100 ; Machining part ④
G25 X0 Y0 ; Mirror image is cancelled
M30
%100
N100 G41 G00 X10 Y4 D01
N120 G43 Z=98 H01
N130 G01 Z=7 F300
N140 Y26
N150 X10
N160 G03 X10 Y=10 I10 J0
N170 G01 Y=10
N180 X=25
N185 G49 G00 Z105
N200 G40 X=5 Y=10
N210 M99
9.2 Scaling (G50, G51)

Programming

G51 X_Y_ Z_ P_
M98 P_
G50

Explanation of the parameters

G51    Activate the scaling
X, Y, Z, Coordinate value of scaling center point in absolute command
P      Scaling magnification
M98    P_    The sequence number of subprogram for produce a shape on a part.
G50    Deactivate the scaling

Function

This command enables the size of a shape to be changed.

![Figure 9.3 Scaling](image)

Note

Scaling is not applicable to the tool compensation. When the operator use the tool compensation function in the program, the machine would execute the scaling before calculating the tool offset value.
Example

Use the scaling function to scale down a triangular $\triangle ABC$ to $\triangle A'B'C'$. The center point of scaling is D point (50, 50). The scaling magnification is 0.5. The starting point of tool is 50mm away from the workpiece.

```
%0051 ;Main program
G92 X0 Y0 Z60
G91 G17 M03 S600 F300
G43 G00 X50 Y50 Z-46 H01
#51=14
M98 P100 ;Machining $\triangle ABC$
#51=8
G51 X50 Y50 P0.5 ;Scaling center(50,50)
;Scaling magnification 0.5
M98 P100 ;machining $\triangle A'B'C'$
G50
G49 Z46
M05 M30

%100 ;Subprogram($\triangle ABC$)
N100 G42 G00 X-44 Y-20 D01
N120 Z[-51]
N150 G01 X84
N160 X-40 Y80
N170 X-44 Y-88
N180 Z[#51]
N200 G40 G00 X44 Y28
N210 M99
```
9.3 Coordinate System Rotation (G68, G69)

**Programming**

G17 G68 X_ Y_ P_
G18 G68 X_ Z_ P_
G19 G68 Y_ Z_ P_
M98 P_
G69

**Explanation of the parameters**

G17, G18, G19  One of planes (XY/ZX/YZ) is selected to be rotated.
G68  Activate the coordinate system rotation.
X, Y/X, Z/Y, Z  Coordinate value of the center point of rotation
P  Angle of rotation. The unit is °. The range is 0≤P≤360°.
M98 P_  The sequence number of subprogram for producing a shape on a part
G69  Deactivate the coordinate system rotation.

**Function**

These command can rotate a programmed shape at a specified angle.

**Note**

1) Coordinate system rotation command is not applicable to the tool compensation function. The system would rotate the coordinate system before executing the tool offset function.

2) When coordinate system rotation command is used with scaling function, scaling is proceeded before the rotation command.
Example

Use the rotation command to machine a part. The depth of cutting is 5mm. The starting point is 50mm away from the workpiece.

![Coordinate System Rotation - Example](image)

Figure 9.5 Coordinate System Rotation – Example

%0068 ; Main program
N10 G92 X0 Y0 Z50
N15 G90 G17 M03 S600
N20 G43 Z-5 H02
N25 M98 P200 ; Machining ①
N30 G68 X0 Y0 P45 ; Rotation degree 45°
N40 M98 P200 ; Machining ②
N60 G68 X0 Y0 P90 ; Rotation degree 90°
N70 M98 P200 ; Machining ③
N20 G49 Z50
N80 G69 M05 M30 ; Rotation cancel
%200 ; Subprogram (①)
G41 G01 X20 Y-5 D02 F300
N105 Y0
N110 G02 X40 I10
N120 X30 I-5
N130 G03 X20 I-5
N140 G00 Y-6
N145 G40 X0 Y0
9. Functions to Simplify Programming

9.4 Canned Cycles

To simplify programming, the canned cycle command can execute the specific operation using one G code, instead of several separated G commands in the program.

In general, a canned cycle consists of six operations.

![Diagram of canned cycle operation]

**Figure 9.6 Sequence of canned cycle operation**

1) Positioning to the initial point
2) Rapid traverse to point R
3) Hole machining
4) Operation at the bottom of hole
5) Retraction to point R
6) Retraction to the initial point
9.4.1 Return to the Initial Point/R point Level (G98, G99)

Programming

\[
\begin{align*}
\{ \text{G98} \} & \quad \text{G X Y Z R Q P I J K F L} \\
\{ \text{G99} \} & \quad \text{G X Y Z R Q P I J K F L}
\end{align*}
\]

Explanation of the parameters

G98 Return to the initial point
G99 Return to point R level

The other parameters would be explain in the specific canned cycle.

Function

Generally, G99 is used for the first drilling operation. G98 is used for the last drilling operation.

![Diagram](https://via.placeholder.com/150)

Figure 9.7 Return to the initial point/R point level
9.4.2 High-speed Peck Drilling Cycle (G73)

**Programming**

\[
\begin{align*}
\text{G98} & \quad \text{G73 X Y Z R Q P K F L}\end{align*}
\]

**Explanation of the parameters**

X, Y  Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z  Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R  Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

Q  Depth of cutting for each cutting feed in the incremental command

P  Dwell time at the bottom of a hole

K  Retraction amount at each time in the incremental command

F  Cutting feedrate

L  Number of repeats

![Diagram of High-speed Peck Drilling Cycle](image)

*Figure 9.8 High-speed Peck Drilling Cycle*
9. Functions to Simplify Programming

**Function**

This command can be used to drill a hole intermittently, so that the operator can remove the chips during machining.

**Note**

1) If the value of Z/K/Q is zero, G73 would not be performed.

2) |Q|>|K|

**Example**

Use a tool (Φ 10) to drill a hole.

![Diagram of High-speed Peck Drilling]

Figure 9.9 High-speed Peck Drilling – Example

%3337

N10 G92 X0 Y0 Z80

N15 M03 S700

N20 G00 Y25

N30 G98G73G91X20G90R40P2Q-10K2Z-3L2F80

N40 G00 X0 Y0 Z80

N45 M30

94
9.4.3 Left-hand Tapping Cycle (G74)

Programming

\[
\begin{align*}
G98 & : G74 \_Y\_Z\_R\_P\_F\_L
\end{align*}
\]

Explanation of the parameters

X, Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P Dwell time at the bottom of a hole

F Thread lead

L Number of repeats

Function

G74 command can create a reverse thread. Tapping is performed by turning the spindle counterclockwise. Then, the spindle turns clockwise for retraction when the tool reaches the bottom of the hole.

![Figure 9.10 Left-hand Tapping Cycle (G74)]
Note
If the value of Z is zero, G74 would not be performed.

Rigid Tapping Mode
There are two ways for the rigid tapping:
1) C-axis tapping: the tapping is performed along C-axis.
2) Z-axis tapping: the tapping is performed along Z-axis.

The default setting is Z-axis tapping. To set the C-axis tapping, M29 is used (M29 is modal M code). The format of setting the C-axis tapping is as follows:

M29 ; C-axis tapping is set as rigid tapping mode
G74 xx xxxxx ; C-axis tapping is performed

Example
Use the tool (M10×1) for the left-hand tapping.

Figure 9.11 Left-hand Tapping – Example

%3339
N10 G92 X0 Y0 Z80 F200
N15 M04 S300
N20 G98G74X50Y40R40P10G90Z-5F1
N30 G00 X0 Y0 Z80
N40 M30
9.4.4 Fine Boring Cycle (G76)

Programming

\[
\begin{align*}
\{G98\} & \quad \text{G76 X Y Z R P I J F L} \\
\{G99\} & \quad \text{G76 X Y Z R P I J F L}
\end{align*}
\]

Explanation of the parameters

X,Y  Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z  Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R  Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P  Dwell time at the bottom of a hole

I  Shift amount along X-axis at the bottom of a hole

J  Shift amount along Y-axis at the bottom of a hole

F  Cutting feedrate

L  Number of repeats

Function

G76 command would bore a hole precisely. When the tool reaches the bottom of the hole, the spindle stops, and the tool is retracted to the direction opposite to the tool tip.

Figure 9.12 Fine Boring Cycle (G76)
Note

If the value of Z, G76 would not be performed.

Example

Use the tool to bore a hole on a workpiece.

![Diagram of fine boring cycle example]

Figure 9.13 Fine Boring Cycle – Example

%3341
N10 G54
N12 M03 S600
N15 G00 X0 Y0 Z80
N20 G98 G76 X20 Y15 R40 P21 I-5 Z-4 F100
N25 X40 Y30
N30 G00 G90 X0 Y0 Z80
N40 M30
9.4.5 Drilling Cycle, Spot Drilling (G81)

**Programming**

\[
\{ \begin{array}{c}
G98 \\
G99
\end{array} \right. G81 \_X\_Y\_Z\_R\_F\_L_
\]

**Explanation of the parameters**

- **X,Y** Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- **Z** Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- **R** Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- **F** Cutting feedrate
- **L** Number of repeats

**Function**

G81 command can be used to drill a hole. The tool is moved at the cutting feedrate when it is drilling a hole. Then, the tool is moved at the rapid traverse speed when it is retracting from the bottom of the hole.

![Drilling Cycle, Spot Drilling (G81)](image)

**Figure 9.14 Drilling Cycle, Spot Drilling (G81)**

**Note**

If the value of Z is zero, G81 would not be performed.
Example

Use the tool (Φ10) to drill a hole.

![Diagram of drilling cycle]

Figure 9.15 Drilling Cycle, Spot Drilling - Example

%3343
N10 G92 X0 Y0 Z80
N15 M03 S600
N20 G98 G81 G91 X20 Y15 G90 R20 Z-3 P2 L2 F200
N30 G00 X0 Y0 Z80
N40 M30
9.4.6 Drilling Cycle, Counter Boring Cycle (G82)

Programming

\[
\begin{cases}
G98 \\
G99
\end{cases}
\]

G82 X_ Y_ Z_ R_ P_ F_ L_

Explanation of the parameters

X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P Dwell time at the bottom of the hole

F Cutting feedrate

L Number of repeats

Function

G82 command can be used to drill a counter bore or a blind bore.

![Diagram of Drilling Cycle, Counter Boring Cycle (G82)](image)

Figure 9.16 Drilling Cycle, Counter Boring Cycle (G82)

Note

If the value of Z is zero, G82 would not be performed.
Example

Use the tool to drill a counter bore.

Figure 9.17 Drilling Cycle, Counter Boring Cycle – Example

%3345
N10 G92 X0 Y0 Z80
N15 M03 S600
N20 G98G82G90X25Y30R40P2Z25F200
N30 G00 X0 Y0 Z80
N40 M30
9.4.7 Peck Drilling Cycle (G83)

Programming

\[
\begin{align*}
\{ & G98 \} \\
G99 \ X \ Y \ Z \ R \ Q \ P \ K \ F \ L 
\end{align*}
\]

Explanation of the parameters

X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

Q Depth of cutting for each cutting feed in the incremental command

P Dwell time at the bottom of the hole

K Retraction amount at each time in the incremental command

F Cutting feedrate

L Number of repeats
**Function**

G83 command can perform a peck drilling. The intermittent cutting feed allows the operator to remove the chips from the hole.

![Peck Drilling Cycle (G83)](image)

**Note**

If the value of Z/Q/K is zero, G83 would not be performed.

**Example**

Use the tool (Φ10) to drill a hole.

![Peck Drilling Cycle – Example](image)

```
%3347
N10 G55 G00 X0 Y0 Z80
N15 Y25
N20 G98G83G91X20G90R40P2Q-10K5G91Z-43F100L2
N30 G90 G00 X0 Y0 Z80
N40 M30
```
9.4.8 Tapping Cycle (G84)

Programming

\[
\{ \text{G98} \} \text{G84 X_ Y_ Z_ R_ P_ F_ L_} \\
\{ \text{G99} \}
\]

Explanation of the parameters

X,Y  Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z   Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R   Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P   Dwell time at the bottom of the hole

F   Thread lead

L   Number of repeats

Function

G84 command is used for tapping. When the tool has reached the bottom of the hole, it is retracted in the reverse direction.

![Figure 9.20 Tapping Cycle (G84)](image)
Note
If the value of Z is zero, G84 would not be performed.

Example
Use the tool (M10×1) to drill a hole.

Figure 9.21 Peck Drilling Cycle – Example

%3349
N10 G92 X0 Y0 Z80
N15 M03 S300
N20 G98G74G91X50Y40G90R38P3G91Z-40F1
N30 G90 G0 X0 Y0 Z80
N40 M30
9.4.9 Boring Cycle (G85)

**Programming**

\[ \begin{align*}
  \{ &G98\} & G85 & X & Y & Z & R & F & L \\
  \{ &G99\} & & & & & & & 
\end{align*} \]

**Explanation of the parameters**

- **X,Y**  Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command
- **Z**  Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command
- **R**  Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command
- **P**  Dwell time at the bottom of the hole
- **F**  Cutting feedrate
- **L**  Number of repeats

![Diagram of Boring Cycle (G85)]

**Function**

G85 command is used to bore a hole, which is not required the precise boring.

**Note**

If the value of Z is zero, G85 would not be performed.
Example

Use the boring tool to bore a hole.

![Figure 9.23 Boring Cycle – Example](image)

%3351
N10 G92 X0 Y0 Z80
N15 M03 S600
N20 G98G85G91X20Y15R-42P2Z-40L2F100
N30 G90 G00 X0 Y0 Z80
N40 M30
9.4.10 Boring Cycle (G86)

Programming

\[
\begin{cases}
  \text{G98} \\
  \text{G99}
\end{cases}
\quad \text{G86 } X \_ Y \_ Z \_ R \_ F \_ L \\
\]

Explanation of the parameters

X, Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

F Cutting feedrate

L Number of repeats

Figure 9.24 Boring Cycle (G86)

Function

G86 command is used to bore a hole, which is not required the precise boring.

Note

If the value of Z is zero, G86 would not be performed.
Example

Use the reamer to bore a hole.

![Diagram of boring cycle example](image)

Figure 9.25 Boring Cycle – Example

%3353
N10 G92 X0 Y0 Z80
N15 G98 G86 G90 X20 Y15 R38 Q10 K5 P2 Z2 F200
N20 X40 Y30
N30 G90 G00 X0 Y0 Z80
N40 M30
9.4.11 Back Boring Cycle (G87)

**Programming**

G98 G87 X_ Y_ Z_ R_ P_ I_ J_ F_ L_

**Explanation of the parameters**

X,Y  Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z  Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R  Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P  Dwell time at the bottom of the hole

I  Shift amount along X-axis at the bottom of a hole

J  Shift amount along Y-axis at the bottom of a hole

F  Cutting feedrate

L  Number of repeats

**Function**

G87 command is used to bore a hole.

![Diagram of Back Boring Cycle (G87)](image)

Figure 9.26 Boring Cycle (G87)
9. Functions to Simplify Programming

**Note**

1) If the value of Z is zero, G87 would not be performed.
2) G87 can not be used with G99.

**Example**

Use the reamer to bore a hole (Φ 28).

![Diagram of Back Boring Cycle – Example](image)

Figure 9.27 Back Boring Cycle – Example

```
%3355
N10G92 X0 Y0 Z80
N15M03 S600
N20G00 Y15 F200
N25G98G87G91X20I-5R-83P2Z23L2
N30G90 G00 X0 Y0 Z80 M05
N40M30
```
9.4.12 Manual Boring Cycle (G88)

Programming

\[
\begin{align*}
\{ G98 \} & \text{G88 X Y Z R P F L} \\
\{ G99 \} & \text{G98 X Y Z R P F L}
\end{align*}
\]

Explanation of the parameters

X, Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P Dwell time at the bottom of the hole

F Cutting feedrate

L Number of repeats

![Figure 9.28 Manual Boring Cycle (G88)](image)

Function

G88 command can be used to bore a hole precisely. The main difference is that the tool is returned to the initial point or the point R in the manual mode key (on the machine control panel).
Note

1) If the value of Z is zero, G88 would not be performed.
2) When manually moving the tool for retraction, the position of tool must be higher than the initial point (G98) or the point R (G99).

Example

Use the reamer to bore a hole.

%3357
N10 G54
N12 M03 S600
N15 G00 X0 Y0 Z80
N20 G98G88G91X20Y15R-42P2I-5Z-40L2F100
N30 G00 G90 X0 Y0 Z80
N40 M30
9. Functions to Simplify Programming

9.4.13 Boring Cycle (G89)

Programming

\[
\begin{align*}
&\text{G98} \\
&\text{G99}
\end{align*}
\]

G89 X Y Z R P F L

Explanation of the parameters

X,Y Coordinate value of the hole position on XY plane in the absolute command, or the coordinate value of the hole position with reference to the initial point on XY plane in the incremental command

Z Coordinate value of the hole position on Z axis in the absolute command, or the coordinate value of the hole position with reference to the point R on Z axis in the incremental command

R Coordinate value of the point R in the absolute command, or the coordinate value of the point R with reference to the initial point in the incremental command

P Dwell time at the bottom of the hole

F Cutting feedrate

L Number of repeats

![Diagram of Boring Cycle (G89)](image)

Figure 9.30 Boring Cycle (G89)

Function

G89 command is used to bore a hole.

Note

If the value of Z is zero, G89 would not be performed.
9.4.14 Canned Cycle Cancel (G80)

**Programming**

G80

**Explanation the parameters**

G80  a canned cycle is cancelled. Meanwhile, the point R and the point Z (the bottom of the hole) are all cancelled.

**Function**

G80 is used to cancel a canned cycle.
9. Functions to Simplify Programming

9.5 Summary

In summary, five points should be noted when using the canned cycle.

1) Use M03/M04 to rotate the spindle before the canned cycle starts.
2) One of the parameters (X, Y, Z, R) must be used in the canned cycle command.
3) When using G74, G84, or G86 command, G04 command can be used in the canned cycle to ensure that the spindle is reached the appropriate speed.
4) G00–G03 can also be used to cancel the canned cycle. When G00–G03 and the canned cycle command are in the same block, the execution depends on the sequence of these commands.
5) If M is specified in the canned cycle, the cycle only starts working after sending M signal is done.
Example 1

Use the canned cycle command to bore a hole. The tool is 100mm away from the workpiece, and the depth of cut is 10mm.

(i)  G81 command is used at first

%1000
G92 X0 Y0 Z30
G91 G00 M03 S600
G98 G81 X40 Y40 G90 R2 Z–10 F200
G91 X40 L3
Y50
X-40 L3
G90 G80 X0 Y0 Z0 M05
M30

(ii) Then, G84 command is used

%2000
G92 X0 Y0 Z30
G91 G00 M03 S600
G98 G84 X40 Y40 G90 R2 Z–10 F100
G91 X40 L3
Y50
X-40 L3
G90 G80 X0 Y0 Z0 M05
M30
Example 2

Use the tool (Φ20) to machine the contour. Then, use the tool (Φ16) to machine the concave. And use the tool (Φ6, Φ8) to bore a hole.

![Diagram of the machining process](image)

Figure 9.32 Canned Cycle – Example 2

%3360
G92 x-20 y-20 z100
M03 S500
N1 M06 T01
G00 G43 Z-23 H01
G01 G41 X0 Y-8 D01 F100
Y42
X7 Y56
X80
Y12
G02 X70 Y0 R10
G01 X-10
G00 G40 X-20 Y-20
G49 Z100
N2 M06 T2
G00 G43 Z-10 H02
X5 Y-10
G01 Y66 F100
X19
Y-10
X20
Y66
G00 G49 Z100
G00 X-20 Y-20
N3 M06 T03
G00 G43 Z10 H03
G98 G73 X14 Y26 Z-23 R-6 Q-5F50
G99 G73 X42 Y40 Z-23 R4 Q-5F50
G99 G73 X42 Y12 Z-23 R4 Q-5F50
G98 G73 X56 Y26 Z-23 R4 Q-5F50
G00 G49 Z100
X-20 Y-20
M05
M30
Example 3

Use the tool (Φ20) to machine the contour. Then, use the tool (Φ16) to machine the concave. And use the tool (Φ6) to drill a hole.

Figure 9.33 Canned Cycle – Example 3

%3361
G92 x-20 y-20 z100
M03 S500
N1 M06 T01
G00 G43 Z-23 H01
G01 G41 X0 Y-8 D01 F100
Y56
X80
Y0
X-10
G00 G40 X-20 Y-20
G49 Z100
N2 M06 T2
G00 G43 Z-10 H02
X5 Y-10
G01 Y70 F100
X13
Y-10
X14
Y70
G00 X75
G01 Y-10 F100
X67
Y70
X66
Y-10
G00 G49 Z100
G00 X-20 Y-20
N3 M06 T03
G00 G43 Z10 H03
G98 G73X12Y14Z-23R-6Q-5K3F50
G73G91X23G90Z-23R4Q-5K3L2F50
G73X58Y42Z-23R-6Q-5K3F50
G73G91X-23G90Z-23R4Q-5K3L2F50
G00 G49 Z100
X-20 Y-20
M05
M30
10 Custom Macro

Similarly to subprogram, the custom macro function allows operators to define their own program. The way of calling the custom macro is same as subprogram’s.

The difference is that custom macro allows use of variables, arithmetic and logic operations, selection and repetition.
10.1 Variables

Format and Explanation

#_ Variable is composed of a number sign (#) and a number.

Example

#1
#1=#2+100

10.1.1 Type of Variables

There are four types of variables.

Table 10 1 Type of Variables

<table>
<thead>
<tr>
<th>Variable number</th>
<th>Type of variables</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>#0~#49</td>
<td>Local variables</td>
<td>They are used in a macro program.</td>
</tr>
<tr>
<td>#50~#199</td>
<td>Common variables</td>
<td>They can be shared among different macro programs.</td>
</tr>
<tr>
<td>#200~#249</td>
<td>0 layers local variables</td>
<td></td>
</tr>
<tr>
<td>#250~#299</td>
<td>1 layers local variables</td>
<td></td>
</tr>
<tr>
<td>#300~#349</td>
<td>2 layers local variables</td>
<td></td>
</tr>
<tr>
<td>#350~#399</td>
<td>3 layers local variables</td>
<td></td>
</tr>
<tr>
<td>#400~#449</td>
<td>4 layers local variables</td>
<td></td>
</tr>
<tr>
<td>#450~#499</td>
<td>5 layers local variables</td>
<td></td>
</tr>
<tr>
<td>#500~#549</td>
<td>6 layers local variables</td>
<td></td>
</tr>
<tr>
<td>#550~#599</td>
<td>7 layers local variables</td>
<td></td>
</tr>
<tr>
<td>#600~</td>
<td>System variables</td>
<td>They are used to read and write NC data.</td>
</tr>
</tbody>
</table>

Note:

1) The operator can only use the #0~#599 local variables for programming.

2) Variables after #599 can only be used by the system programmer for reference.
10.1.2 System Variables

#1000 “current position X in machine coordinate system”
#1001 “current position Y in machine coordinate system”
#1002 “current position Z in machine coordinate system”
#1003 “current position A in machine coordinate system”
#1004 “current position B in machine coordinate system”
#1005 “current position X in machine coordinate system”
#1006 “current position U in machine coordinate system”
#1007 “current position V in machine coordinate system”
#1008 “current position W in machine coordinate system”
#1009 “diameter programming”
#1010 “position X – machine coordinate system in programming”
#1011 “position Y – machine coordinate system in programming”
#1012 “position Z – machine coordinate system in programming”
#1013 “position A – machine coordinate system in programming”
#1014 “position B – machine coordinate system in programming”
#1015 “position C – machine coordinate system in programming”
#1016 “position U – machine coordinate system in programming”
#1017 “position V – machine coordinate system in programming”
#1018 “position W – machine coordinate system in programming”
#1019 reserved
#1020 “position X – workpiece coordinate system in programming”
#1021 “position Y – workpiece coordinate system in programming”
#1022 “position Z – workpiece coordinate system in programming”
#1023 “position A – workpiece coordinate system in programming”
#1024 “position B – workpiece coordinate system in programming”
#1025 “position C – workpiece coordinate system in programming”
#1026 “position U – workpiece coordinate system in programming”
#1027 “position V – workpiece coordinate system in programming”
#1028 “position W – workpiece coordinate system in programming”
#1029 reserved
#1030 “origin X in workpiece coordinate system”
“origin Y in workpiece coordinate system”
“origin Z in workpiece coordinate system”
“origin A in workpiece coordinate system”
“origin B in workpiece coordinate system”
“origin C in workpiece coordinate system”
“origin U in workpiece coordinate system”
“origin V in workpiece coordinate system”
“origin W in workpiece coordinate system”
“axis of the coordinate system”
“origin X of G54”
“origin Y of G54”
“origin Z of G54”
“origin A of G54”
“origin B of G54”
“origin C of G54”
“origin U of G54”
“origin V of G54”
“origin W of G54”
reserved
“origin X of G55”
“origin Y of G55”
“origin Z of G55”
“origin A of G55”
“origin B of G55”
“origin C of G55”
“origin U of G55”
“origin V of G55”
“origin W of G55”
reserved
“origin X of G56”
“origin Y of G56”
“origin Z of G56”
“origin A of G56”

“origin B of G56”

“origin C of G56”

“origin U of G56”

“origin V of G56”

“origin W of G56”

reserved

“origin X of G57”

“origin Y of G57”

“origin Z of G57”

“origin A of G57”

“origin B of G57”

“origin C of G57”

“origin U of G57”

“origin V of G57”

“origin W of G57”

reserved

“origin X of G58”

“origin Y of G58”

“origin Z of G58”

“origin A of G58”

“origin B of G58”

“origin C of G58”

“origin U of G58”

“origin V of G58”

“origin W of G58”

reserved

“origin X of G59”

“origin Y of G59”

“origin Z of G59”

“origin A of G59”

“origin B of G59”
“origin C of G59”
“origin U of G59”
“origin V of G59”
“origin W of G59”
reserved
“break point X”
“break point Y”
“break point Z”
“break point A”
“break point B”
“break point C”
“break point U”
“break point V”
“break point W”
“axis of the coordinate system”
“middle point X of G28”
“middle point Y of G28”
“middle point Z of G28”
“middle point A of G28”
“middle point B of G28”
“middle point C of G28”
“middle point U of G28”
“middle point V of G28”
“middle point W of G28”
“shield of G28”
“mirror-image position X”
“mirror-image position Y”
“mirror-image position Z”
“mirror-image position A”
“mirror-image position B”
“mirror-image position C”
“mirror-image position U”
10. Custom Macro

- “mirror-image position V”
- “mirror-image position W”
- “shield of mirror image”
- “rotational axis 1”
- “rotational axis 2”
- “rotation angle”
- “shield of rotational axis”
- reserved
- “scale axis 1”
- “scale axis 2”
- “scale axis 3”
- “scaling”
- “shield of scale axis”
- “code 1 of changing a coordinate system”
- “code 2 of changing a coordinate system”
- “code 3 of changing a coordinate system”
- reserved
- “number of tool length compensation”
- “number of tool radius compensation”
- “linear axis 1”
- “linear axis 2”
- “shield of virtual axis”
- “specified feedrate”
- “modal value of G code – 0”
- “modal value of G code – 1”
- “modal value of G code – 2”
- “modal value of G code – 3”
- “modal value of G code – 4”
- “modal value of G code – 5”
- “modal value of G code – 6”
- “modal value of G code – 7”
- “modal value of G code – 8”
#1159 "modal value of G code – 9"
#1160 "modal value of G code – 10"
#1161 "modal value of G code – 11"
#1162 "modal value of G code – 12"
#1163 "modal value of G code – 13"
#1164 "modal value of G code – 14"
#1165 "modal value of G code – 15"
#1166 "modal value of G code – 16"
#1167 "modal value of G code – 17"
#1168 "modal value of G code – 18"
#1169 "modal value of G code – 19"
#1170 "residual CACHE"
#1171 "spare CACHE"
#1172 "residual buffer storage"
#1173 "spare buffer storage"
#1174 reserved
#1175 reserved
#1176 reserved
#1177 reserved
#1178 reserved
#1179 reserved
#1180 reserved
#1181 reserved
#1182 reserved
#1183 reserved
#1184 reserved
#1185 reserved
#1186 reserved
#1187 reserved
#1188 reserved
#1189 reserved
#1190 "customized input"
#1191  “customized output”
#1192  “customized output shield”
#1193  reserved
#1194  reserved
10.2 Constant

PI  \( \pi, 3.14151926 \)
TRUE  True condition
FALSE  False condition
10.3 Operators and Expression

1) Mathematic operator

+, -, *, /

2) Conditional operator

EQ(=), NE(≠), GT(>), GE(≥), LT(<), LE(≤)

3) Logic operator

AND, OR, NOT

4) Function

SIN    Sine
COS    Cosine
TAN    Tangent
ATAN   Arctangent
ATAN2  Arctangent2
ABS    Absolute value
INT    Integer
SIGN   Sign
SQRT   Square root
EXP    Exponential function

5) Expression

The expressions are composed of constants, operators and variables.

Example:

175/SQRT[2] * COS[55 * PI/180 ];
#3*6 GT 14;
10.4 Assignment

Assignment refers to assign a variable value to a constant or expression.

Format:
Variable=constant or expression

Example

#2 = 175/SQRT[2] * COS[55 * PI/180]
#3 = 124.0
10.5 Selection statement IF, ELSE,ENDIF

**Format (i)**

IF Conditional expression

...  
ELSE  
...  
ENDIF  

**Explanation (i)**

If the specified conditional expression is satisfied, the statements between IF and ELSE are executed. If the specified conditional expression is not satisfied, the statements between ELSE and ENDIF are executed.

**Format (ii)**

IF Conditional expression

...  
ENDIF  

**Explanation (ii)**

If the specified conditional expression is satisfied, the statements between IF and ENDIF are executed. If the specified conditional expression is not satisfied, the system would proceed to the blocks after ENDIF.
10.6 Repetition Statement WHILE, EN DW

**Format**

WHILE  Conditional expression

...  

ENDW

**Explanation**

When the conditional expression is satisfied, the statements between WHILE and EN DW are executed. If the conditional expression is not satisfied, the system would proceed to the blocks after EN DW.
10.7 Macro Call

The following table shows the local variable and the corresponding system variable when it is macro call.

<table>
<thead>
<tr>
<th>Local variables</th>
<th>System variables in macro call</th>
</tr>
</thead>
<tbody>
<tr>
<td>#0</td>
<td>A</td>
</tr>
<tr>
<td>#1</td>
<td>B</td>
</tr>
<tr>
<td>#2</td>
<td>C</td>
</tr>
<tr>
<td>#3</td>
<td>D</td>
</tr>
<tr>
<td>#4</td>
<td>E</td>
</tr>
<tr>
<td>#5</td>
<td>F</td>
</tr>
<tr>
<td>#6</td>
<td>G</td>
</tr>
<tr>
<td>#7</td>
<td>H</td>
</tr>
<tr>
<td>#8</td>
<td>I</td>
</tr>
<tr>
<td>#9</td>
<td>J</td>
</tr>
<tr>
<td>#10</td>
<td>K</td>
</tr>
<tr>
<td>#11</td>
<td>L</td>
</tr>
<tr>
<td>#12</td>
<td>M</td>
</tr>
<tr>
<td>#13</td>
<td>N</td>
</tr>
<tr>
<td>#14</td>
<td>O</td>
</tr>
<tr>
<td>#15</td>
<td>P</td>
</tr>
<tr>
<td>#16</td>
<td>Q</td>
</tr>
<tr>
<td>#17</td>
<td>R</td>
</tr>
<tr>
<td>#18</td>
<td>S</td>
</tr>
<tr>
<td>#19</td>
<td>T</td>
</tr>
<tr>
<td>#20</td>
<td>U</td>
</tr>
<tr>
<td>#21</td>
<td>V</td>
</tr>
<tr>
<td>#22</td>
<td>W</td>
</tr>
<tr>
<td>#23</td>
<td>X</td>
</tr>
<tr>
<td>#24</td>
<td>Y</td>
</tr>
<tr>
<td>#25</td>
<td>Z</td>
</tr>
<tr>
<td>#26</td>
<td>Mode value of Z-plane in canned cycle</td>
</tr>
<tr>
<td>#27</td>
<td>Unavailable</td>
</tr>
<tr>
<td>#28</td>
<td>Unavailable</td>
</tr>
<tr>
<td>#29</td>
<td>Unavailable</td>
</tr>
<tr>
<td>#30</td>
<td>Absolute coordinate of 0-axis when subprogram call</td>
</tr>
<tr>
<td>#31</td>
<td>Absolute coordinate of 1-axis when subprogram call</td>
</tr>
<tr>
<td>#32</td>
<td>Absolute coordinate of 2-axis when subprogram call</td>
</tr>
<tr>
<td>#33</td>
<td>Absolute coordinate of 3-axis when subprogram call</td>
</tr>
<tr>
<td>#34</td>
<td>Absolute coordinate of 4-axis when subprogram call</td>
</tr>
<tr>
<td>#35</td>
<td>Absolute coordinate of 5-axis when subprogram call</td>
</tr>
<tr>
<td>#36</td>
<td>Absolute coordinate of 6-axis when subprogram call</td>
</tr>
<tr>
<td>#37</td>
<td>Absolute coordinate of 7-axis when subprogram call</td>
</tr>
<tr>
<td>#38</td>
<td>Absolute coordinate of 8-axis when subprogram call</td>
</tr>
</tbody>
</table>
**Explanation**

1) To check whether the variable is defined in the program, the format is as follows:

   AR [#Variable number]

   Return:

   0 – the variable is not defined
   90 – the variable is defined as absolute command G90
   91 – the variable is defined as incremental command G91

2) When it is macro call (subprogram or canned cycle) with G code, the system would copy the system variables (A~Z) to local variables #0-#25 in the macro. Meanwhile, the system can copy the axis position (machine coordinate value in absolute command) of nine channels to local variables #30-#38.

3) When calling a subprogram, the subprogram can modify the system mode.

4) When calling a canned cycle, the canned cycle does not modify the system mode.
10.8 Example

Example 1

Use the tool (Φ10) to machine a frustum of a cone and an inclined plane. The included angle between the frustum of a cone and an incline plane is 10°. The first time of finishing amount is 1mm, and the second time of finishing amount is 3mm.

![Diagram of frustum and inclined plane](image)

Figure 10.1 Custom Macro – Example 1

%8002
#10=10.0 ; Height of column
#11=10.0 ; Height of square
#12=124.0
#13=124.0
N01 G92 X0.0 Y0.0 Z0.0
N05 G00 Z10.0
#0=0
N06 G00 X[−#12] Y[−#13]
N07 Z[−#10] M03 S600
WHILE #0 LT 3 ; Column machining
N[08+#0*6] G01 G42 X[−#12/2] Y[−175/2] F280.0 D[#0+1]
N[09+#0*6] X[0] Y[−175/2]
N[10+#0*6] G03 J[175/2]
N[13+#0*6] G00 X[−#12] Y[−#13]
#0=#0+1
ENDW
N100 Z[−#10−#11]
#2=175/SQRT[2]*COS[55*PI/180]
#3=175/SQRT[2]*SIN[55*PI/180]
#4=175*COS[10*PI/180]
#5=175*SIN[10*PI/180]
#0=0
WHILE #0 LT 3 ; Square machining
N[103+#0*6] X[−#5] Y[+#4]
N[104+#0*6] X[−#4] Y[−#5]
N[105+#0*6] X[+#5] Y[−#4]
#0=#0+1
ENDW
G00 X0 Y0 M05
M30
Example 2

Use the tool to machine an ellipse. \( X = a \times \cos \alpha , \ Y = b \times \sin \alpha \)

![Figure 10.2 Custom Macro – Example 2]

%0001
#0=5 ; the radius of tool
#1=20 ; the value of a
#2=10 ; the value of b
#3=0 ; the value of \( \alpha \)
N1 G92 X0 Y0 Z10
N2 G00 X[2*#0+#1] Y[2*#0+#2]
N3 G01 Z0
N4 G41 X[#1]
N5 WHILE #3 GE [-360]
N6 G01 X[#1*COS[#3*PI/180]] Y[#2*SIN[#3*PI/180]]
N7 #3=#3-5
ENDW
G01 G91 Y[-2*#0]
G90 G00 Z10
G40 X0 Y0
M30
Example 3

Use the drilling tool (diameter is 20mm) to drill a thread (M60 × 1.5). The smaller diameter is 60-2+0.376=58.376. The height of workpiece is 10mm.

![Figure 10.3 Custom Macro – Example 3](image)

```
%0027
N1 G92 X0 Y0 Z30
N2 MO3 S500
N3 G01 Z11 X19.178 F1200
N4 #0=58.376/2-10+0.3
WHILE #0 LE 20
N5 G91 G03 I[-#0] Z-1.5 L8
N6 G90 G01 X0
N7 Z11
N8 #0=#0+0.2
N9 G01 X[#0]
N10 ENDW
N11 G01 X20
N12 G91 G03 I-20 Z-1.5 L8
N13 G90 G00 X0
N14 G00 Z30
N15 X30 Y-50
N16 M30
```
Example 4

Use ball-end milling tool to machine a fillet surface (R5).

![Diagram of the milling operation](image)

Figure 10.4 Custom Macro – Example 4

%0001
G92 X-30 Y-30 Z25 ; the tool is at the center of the circle.
#0=5 ; the radius of fillet surface
#1=4 ; the radius of the tool
#2=180 ; the value of γ, the unit is degree.
WHILE #2 GT 90
G01 Z[25+#0+#1]*SIN[#2*PI/180]] ; the height on Z axis
#101=ABS[#0+#1]**CO [2*PI/180]-#0 ; the radius offset
G01 G41 X-20 D101
Y14
G02 X-14 Y20 R6
G01 X14
G02 X20 Y14 R6
G01 Y-14
G02 X14 Y-20 R6
G01 X-14
G02 X-20 Y-14 R6
G01 X-30
G40 Y-30
#2=#2-10
ENDW
M30