Roland DIGITAL GROUP

CAMM-3 by ROLAND DIGITAL GROUP

Model PNC-300G

# NC code

# PROGRAMMER'S MANUAL

ROLAND DG CORPORATION

# Table of Contents


**Introduction** ..... 2

**Part 1 Programming Basics**

Programming ..... 3
Coordinate Systems ..... 7
Setting Coordinate Values (Amount of Movement) ..... 10
Setting the Measurement Unit ..... 11
Real-number Entry and Integer Entry ..... 11
Program Number ..... 12
Sequence Numbers ..... 12
Optional Block Skip ..... 12
Positioning (G00) ..... 13
Linear Interpolation (G01) ..... 13
Circular Interpolation (G02 and G03) ..... 13
Cutter Compensation (G40, G41 and G42) ..... 15
Tool-length Compensation (G43, G44 and G49) ..... 15
Feed Rate ..... 16
Spindle Motor Control (M03 and M05) ..... 16
Spindle Motor Speed ..... 16
Fixed Cycle ..... 16
Program-related Errors ..... 16
Sample Program ..... 18

**Part 2 Reference**

How to Read Part 2 ..... 19
Preparatory Functions (G Functions)

| | | |
|---|---|---|
| G 00 | Positioning | 20 |
| G 01 | Linear Interpolation | 21 |
| G 02 and G 03 | Circular Interpolation | 22 |
| G 04 | Dwell | 25 |
| G10 | Data Setting | 26 |
| G 17, G 18 and G 19 | Plane | 28 |
| G20 and G21 | Setting the Measurement Unit | 28 |
| G39 | Corner-offset Circular Interpolation | 29 |
| G40, G41 and G42 | Cutter Compensation | 30 |
| G43, 44 and G49 | Tool-length Compensation | 38 |
| G 50 and G 51 | Scaling | 41 |
| G 54, G55, G56, G57, G58 and G 59 | Selects Coordinate System | 42 |
| G 80, G 81, G 82, G 85, G86 and G 89 | Fixed Cycle (Canned Cycle) | 43 |
| G 90 and G 91 | Absolute and Incremental | 47 |
| G 92 | Coordinate System | 48 |
| G 98 | Initial Level Return | 49 |
| G 99 | Point R Level Return | 49 |

Miscellaneous Functions (M Functions)
M 00 Program Stop ..... 50
M 01 Optional Stop ..... 50
M 02 End of Program ..... 50
M 03 and M 05 Spindle Motor Start/Stop ..... 50
M 06 Tool Change ..... 51
M 30 End of Program ..... 51
M 98 Subprogram Call ..... 52
M 99 End of Subprogram ..... 53
Spindle Speed Function (S Function) ..... 54
Feed Function (F Function) ..... 56
Other Functions ..... 57
N Sequence Number ..... 57
O Program Number ..... 57
/ Optional Block Skip ..... 58
% or ER Program Start ..... 59
EOB End of Block ..... 59
( ) Comment ..... 59

## Appendices

Words Table
Preparatory Functions (G Functions) ..... 60
Miscellaneous Functions (M Functions) ..... 61
Character Code Table (ISO, EIA, and ASCII) ..... 62
Index ..... 63

# Introduction

## What Is in This Manual

This manual explains those codes interpreted by numerical control machine tools (NC codes) which are supported by PNC-300G. Codes and functions which are not supported are not described.

## This Manual Has Two Parts.

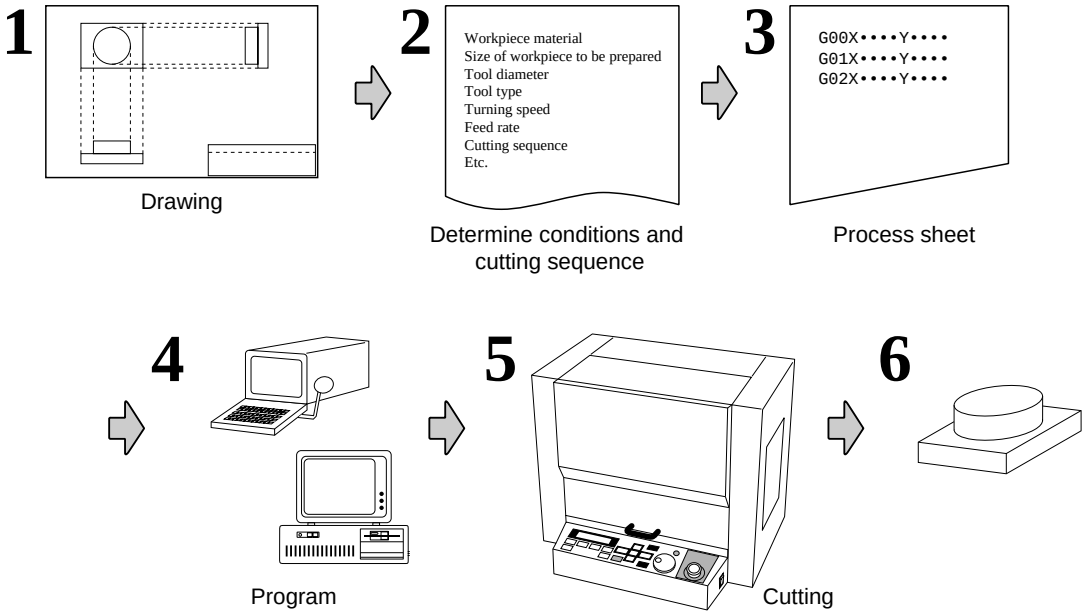
"**Part 1** — Programming Basics" describes such basic but necessary programming-related matters as programming, coordinate systems, and how to set coordinate values (amount of movement).
"**Part 2** — Reference" explains each of the codes in an encyclopedic form. Once a certain familiarity with programming has been achieved, programming can be accomplished simply by glancing through this part.

# Part 1 Programming Basics

## Programming

### The Process of Programming

First, examine the drawing and determine such conditions as the workpiece material, the size of the workpiece to be prepared, the tool diameter, the tool type, the proper turning speed, and the proper feed rate. Then determine the sequence in which to cut. The cutting sequence is in extremely important point for carrying out cutting efficiently and safely.
In actual cutting, the moving portion may be either the workpiece or the spindle. Programming is carried out with consideration given to how the tool moves, with no distinction made for the actual movement.




Once the appropriate conditions and cutting sequence for the job have been determined, then before cutting is begun, the programming steps are written down on a piece of paper like the one shown below. (This form is called a “process sheet.”) This sheet is handy for confirming the cutting sequence and numerical values.

| N | G | X | Y | Z | S | F | M | |
|---|---|---|---|---|---|---|---|---|
| N 01 | | | | | | | | % |
| N 02 | G 90 | | | | | | | |
| N 03 | G 00 | | | Z •••• | | | | |
| N 04 | | X •••• | Y •••• | | | | | |
| N 05 | | | | | S 5000 | F 300 | M 03 | |
| N 06 | G 01 | | | Z •••• | | | | |
| N 07 | | X •••• | | | | | | |
| N 08 | | | Y •••• | | | | | |
| N 09 | | X •••• | Y •••• | | | | | |
| N 10 | G 02 | X •••• | Y •••• | | | F 150 | | |
| • | | | | | | | | |

# Program Structure

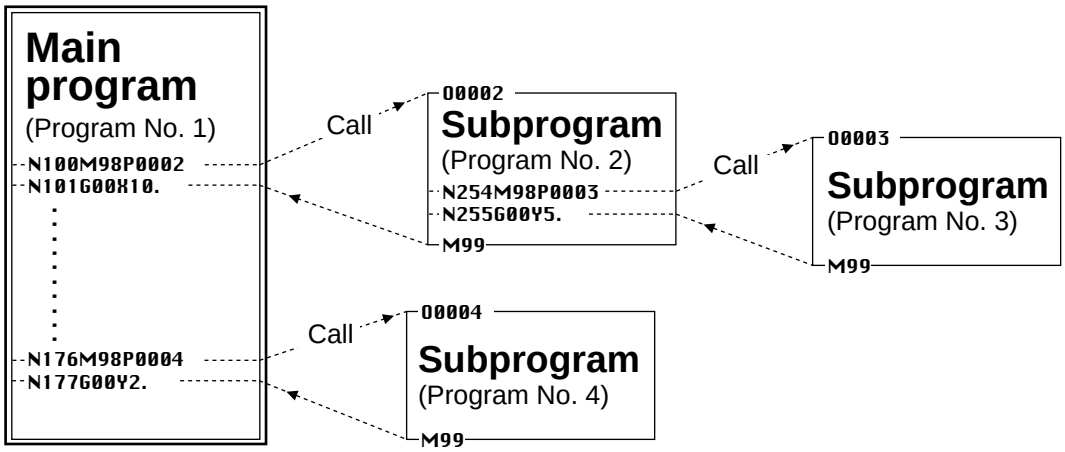
A program can be classified as a “main program” or a “subprogram.”

## Main Program

Ordinarily, the machine operates according to instructions from a main program. A subprogram, which is described next, is basically specified within a main program.

## Subprogram

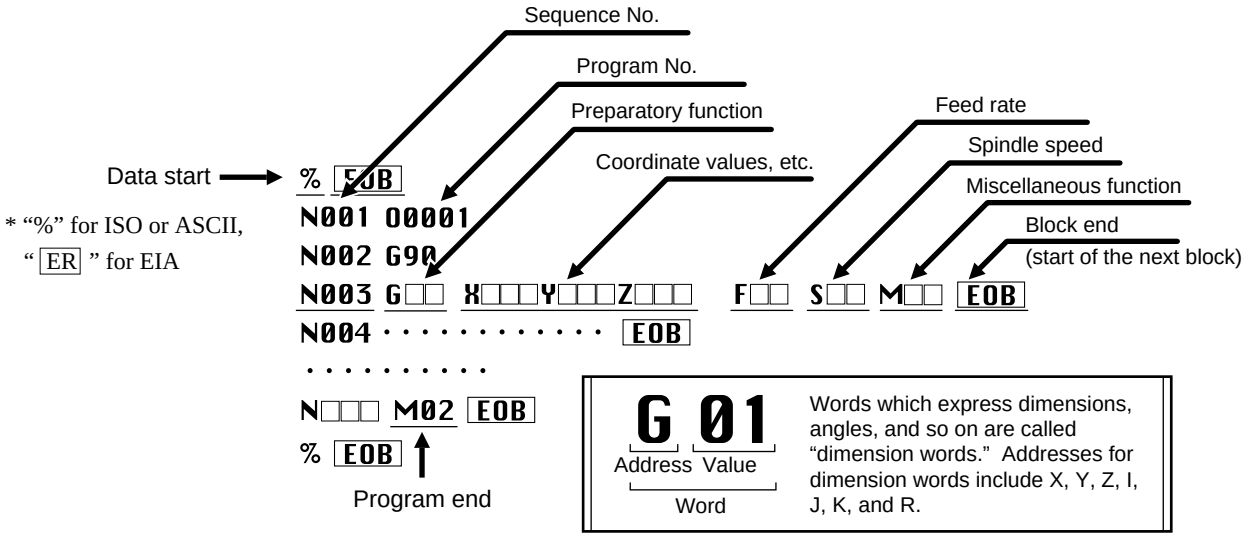
If a main program can be likened to the trunk of a tree, then subprograms are branches. When a main program specifies execution of a subprogram of a certain number, the subprogram of that number is called up and executed.
Subprograms can be convenient when writing a program where the same task is to be performed a number of times. Program modularity can also be promoted by creating subprograms which have good general utility.
A subprogram can call another subprogram.




Conceptual view of a main program and subprograms

# Program Structure and Format

Programming proceeds in accordance with the process sheet that has been prepared. The program is input using a dedicated input device for NC codes, or with a general-purpose computer.
Input is made as shown in the figure below. Addresses such as G, M, X, I, and J are combined with numbers to specify the cutting steps and tool movements.




(* For details on blocks, see “About Blocks,” which is the following section.)

**%** : A block containing only a "%" must appear at the start of the data. This notifies the machine of the start (or end) of the data. Such a block may optionally be present at the end of the data. When it appears at the end of the data, the data is specified automatically.
The character indicating the program start is "%" in the case of ISO or ASCII code, or " ER " in the case of EIA.

**EOB** : The EOB (end of block) indicates the conclusion of the block. In ASCII, this is the LF (line feed) code.

**O** : This inputs the program number. A program starts with a program number, and ends with either M02, M30, or M99. M02 or M30 signifies the end of a main program, and M99 signals the end of a subprogram.
An integer from 0001 to 9999 can be input as the program number. Care should be taken to ensure that multiple program numbers do not appear in a single program.
Program numbers are used to call up subprograms.

**N** : Sequence numbers are reference numbers for the program. They have absolutely no effect on the program (machine operation).

**G** : This is the preparatory function. The special functions are specified by the two-digit code following the "G."

**X(I)**
**Y(J)**
**Z(K)** : This indicates a coordinate value or movement distance. I, J, and K are used to specify the center coordinate of a circle (or arc).
To indicate a negative coordinate or movement in a negative direction, a "-" is prefixed to the numerical value (example: X-100). For positive coordinates or movement directions, it is not necessary to prefix a "+" to the value.

**F** : This determines the feed rate.

**S** : This determines the speed of the spindle motor.

**M** : This is a miscellaneous function. It is used for such operations as starting and stopping the spindle motor.

**M02** : This signals the end of the program. The spindle stops.

## About Blocks

A program is a series of instructions (written commands) for the machine, expressed as symbols and numbers. The instructions are separated by EOB markers, with the information between two EOB markers making up one instruction. This single instruction between two EOB markers is called a "block." Each block, in turn, is composed of "words."
The types of words include those valid only within a block, those also valid outside of blocks until a word of the same group is specified, those which activate a function immediately after being specified, those which activate a function at the end of the block in which they are specified, and those which activate a function at the start of the block in which they are specified. Programming requires knowledge of the characteristics of each word. See pages 35 and 36 for a chart of word characteristics.

### Examples of "words also valid outside of blocks until a word of the same group is specified"

```
%
N01 G90          ←— Indicates that the specified coordinate value is absolute    (G90: pages 10 and 47)
N02 G01X100Y100  ←— Linear interpolation from the current tool position to X = 100, Y = 100    (G01: pages 13 and 21)
N03 Z-20         ←— Linear interpolation from the tool position moved to by N02 to Z = -20
N04 G00Z20       ←— Movement (positioning) from the tool position moved to by N03 to Z = 20   (G00: pages 13 and 20)
N05 X0Y0         ←— Movement (positioning) from the tool position moved to by N03 to X = 0, Y = 0.
```

## Data Output from the Computer

Before outputting data, make the appropriate cable connections to link the computer and the machine. Refer to the user's manuals for each model for explanations on making connections and connection specifications.

The character code systems supported by the machine are ASCII, ISO, and EIA. ASCII is an abbreviation for the "American Standard Code for Information Interchange." ASCII is the most widely used code standard established for computers to handle text, and is used by virtually all computers. Text is output as normal ASCII unless you use software which can output ISO or EIA codes to the machine, or a program which converts ASCII to ISO or EIA.
Numerically controlled machine tools (NC machines) generally use ISO (International Organization for Standardization) or EIA (Electronic Industry Association) as their character code system. Both of these use a data length of 8 bits, which means that ASCII, with a length of 7 bits, cannot be used without modification. In an ISO code, the 8th bit is for parity checking, with bits 1 through 7 the same as the corresponding ASCII code. (Refer to the chart on page 62.)

### Example 1: Data output using MS-DOS commands

Let's say that a program has been input and stored with "cogwheel.nc" as the filename. When you input the following commands at the MS-DOS command line, the file is sent from the computer's output port. The format of these commands may vary from one computer to another, so consult the operation manual for your computer for details.

Parallel connection
```
C>type cogwheel.nc > prn
C>copy cogwheel.nc prn
```

Serial connection
```
C>type cogwheel.nc > aux
C>copy cogwheel.nc aux
```

* If you are using a serial connection, make sure the computer and machine are set to use the same communication parameters.

### Example 2: Data output using BASIC

```
10 OPEN "LPT1:" AS #1
20 PRINT #1,"%";CHR$(10)
30 PRINT #1,"N01G00X10.0Y10.0";CHR$(10)
40 PRINT #1,................
50 PRINT #1,................
................
.. PRINT #1,"M02";CHR$(10)
```

* Make any necessary modifications for your hardware or version of BASIC.
* The example program shown above is for a parallel connection. If you are using a serial connection, change line 10 to read as follows:

```
10 OPEN "COM1:9600,N,8,1" AS #1
```

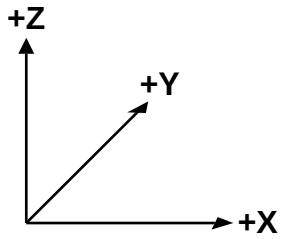
### Example 3: Data output using C

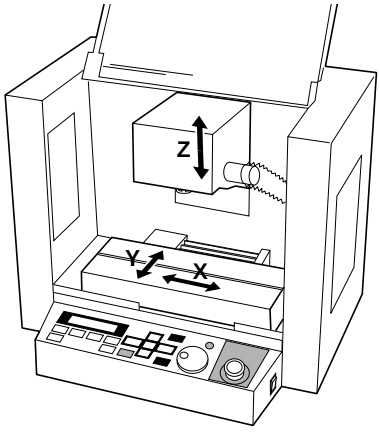
```
#include <stdio.h>
main()
  {
      fprintf(stdprn,"% \n");
      fprintf(stdprn,"N01G00X10.0Y10.0 \n");
      fprintf .......
      ...............
      fprintf(stdprn,"M02 \n");
  }
```

* The example shown above is written in Microsoft® C from Microsoft Corporation.
* The example program shown above is for a parallel connection. If you are using a serial connection, change "**stdprn**" to read "**stdaux**" in every location that it occurs.

# Coordinate Systems

The machine uses the Cartesian coordinate system, which has three axes — the X axis, the Y axis, and the Z axis — each of which is perpendicular to the other two.




## Machine Coordinate Systems

A machine coordinate system is a coordinate system determined mechanically with reference to the PNC-300G. The origin point in a machine coordinate system is a point specific to the PNC-300G, and cannot be moved. The origin point in a machine coordinate system is at the forward-left corner of the machine's maximum range of operation, and is the point to which the machine moves at powerup. Coordinate values (or amounts of movement) specified in an actual program are coordinates in a workpiece coordinate system.

## Workpiece Coordinate Systems

A workpiece coordinate system is a coordinate system for workpiece machining. The origin point of a workpiece coordinate system is the program's origin point according to an absolute specification.

There are two methods that can be used to set a workpiece coordinate system.

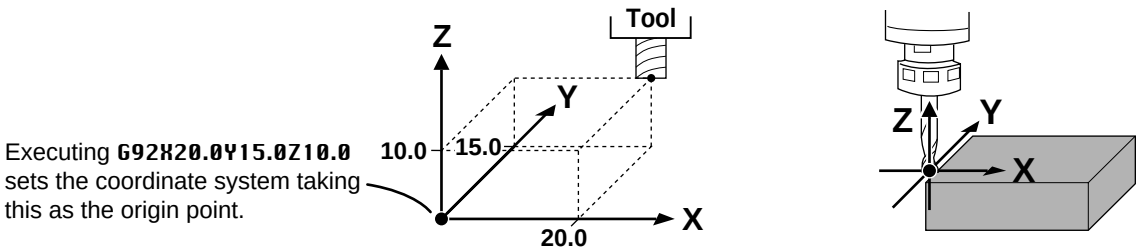
1. Setting using G92
2. Setting using G54 to G59

### 1. Setting a workpiece coordinate system using G92

A workpiece coordinate is defined by specifying the workpiece coordinate location of the current tool location.
For example, to set a point which is 20 mm (X), 15 mm (Y), 10 mm (Z) from the point to be taken as the origin of the present tool location, the following should be set:

```
G92X20.0Y15.0Z10.0
```

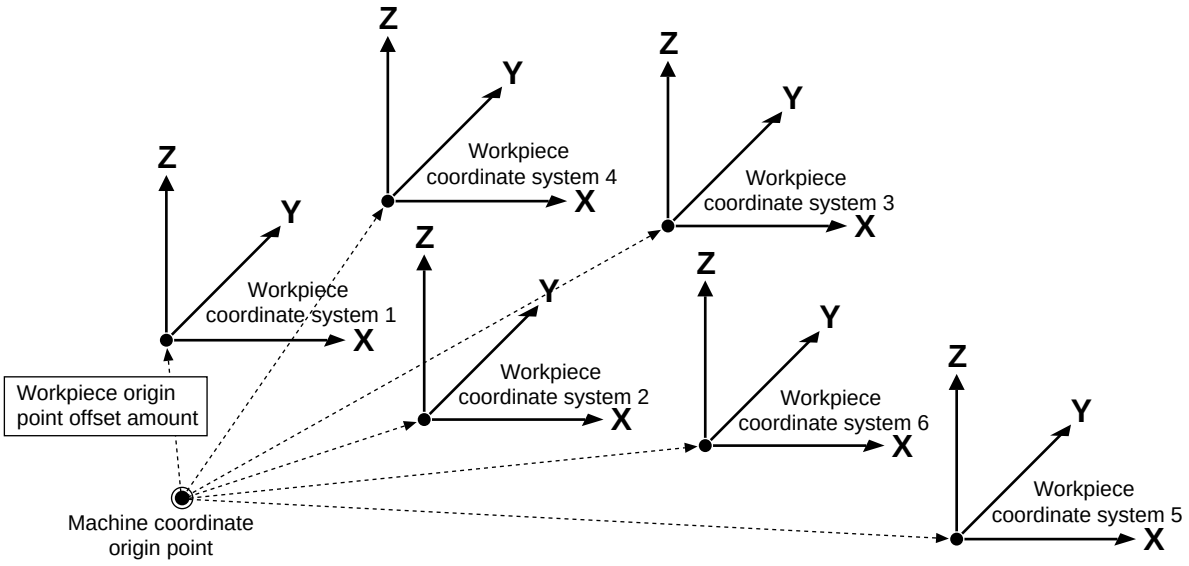
Normally, the tool is moved to a point on the loaded workpiece and G92X0Y0Z0 (the workpiece coordinate origin point) is set.



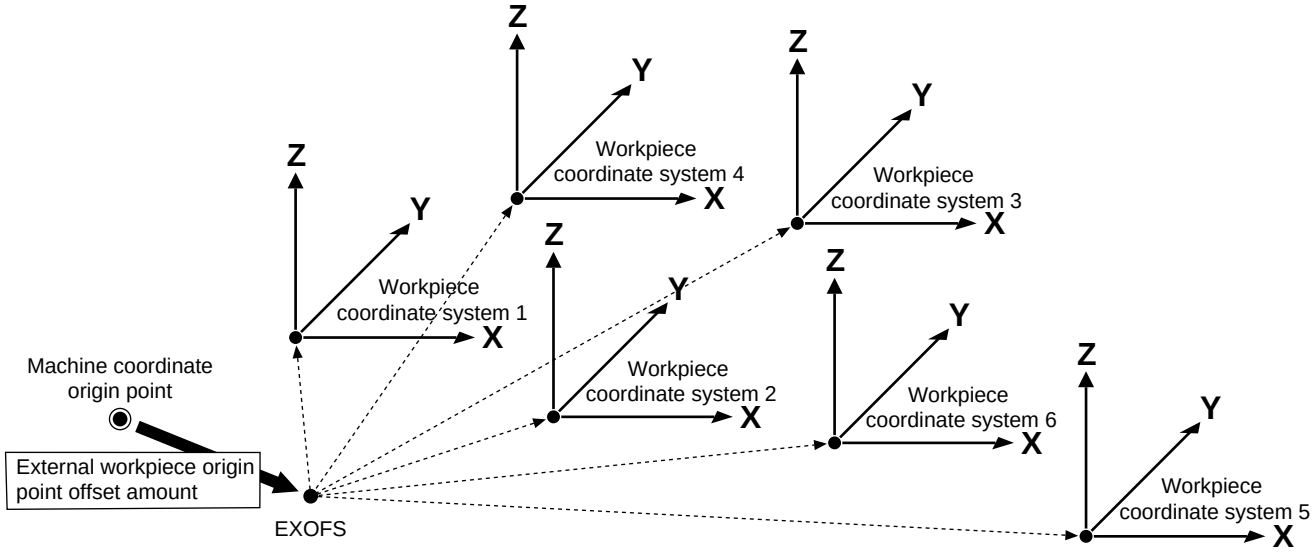
## 2. Setting a workpiece coordinate system using G54 to G59

This method is used to set up to six workpiece coordinate origin points and select a coordinate system from among these by means of the program.

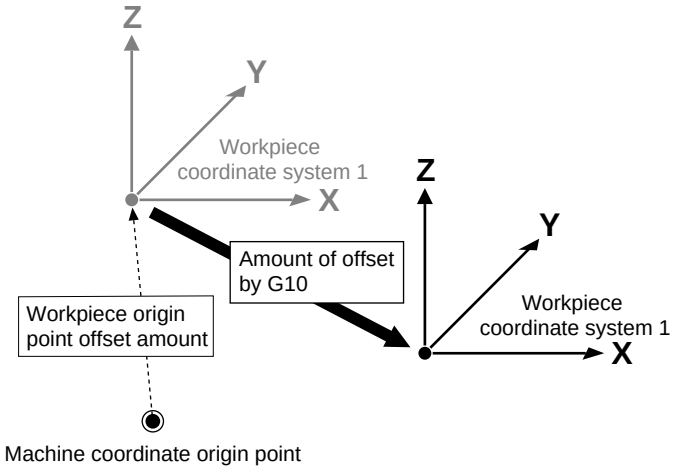
The workpiece coordinate systems 1 through 6 are set by specifying the amount of shift (the amount of workpiece origin point offset) from the machine coordinate origin point to the workpiece coordinate origin point. Each workpiece coordinate system is set using the PNC-300G's display. (Refer to the User's Manual for information on making this setting.)



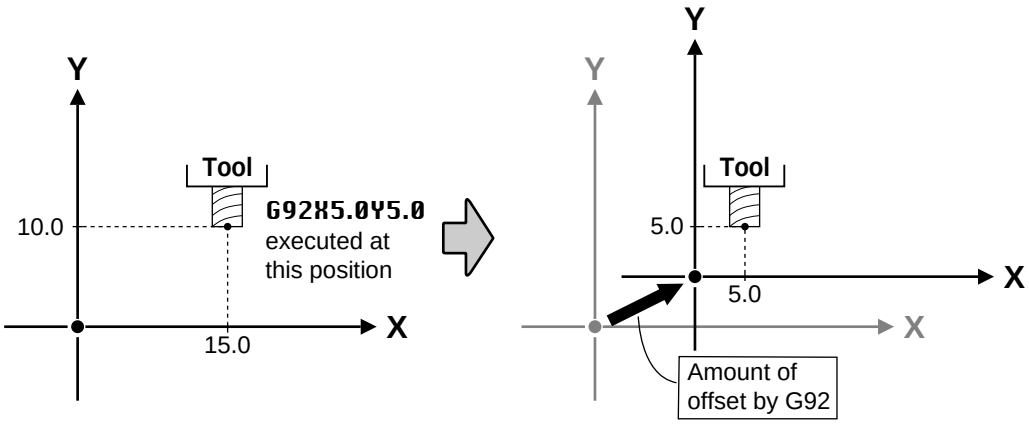

It is also possible to shift the six set workpiece coordinates by a desired distance at one time. Because the workpiece origin point is offset, this is called EXOFS (external workpiece origin point offset amount). EXOFS is set with G10 or by using the PNC-300G's display. (Refer to the User's Manual for information on how to make the setting on the PNC-300G.)



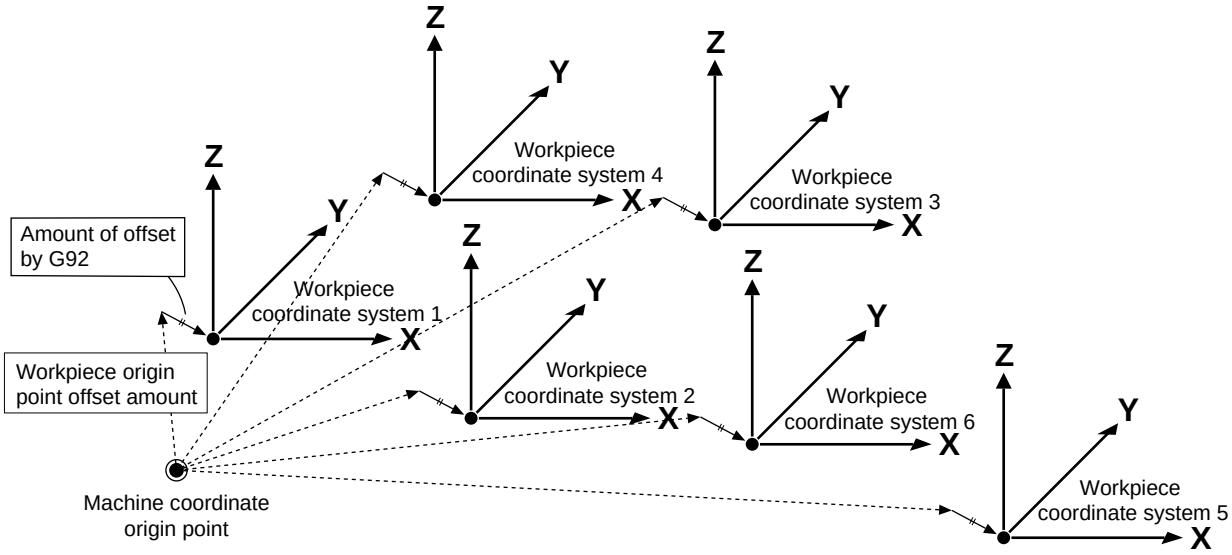
G10 can be used to set not only the amount of shift for all workpiece coordinate systems, but also amounts of offset for each individual workpiece coordinate system.




Selecting any of the workpiece coordinate systems from G54 to G59 and executing G92 causes the following to occur.




Because G92 sets the current tool position to a desired coordinate value, the workpiece coordinate system selected at that time comes to have a new origin point. The distance between the origin coordinate points before and after G92 is specified (that is, the amount of shift of the coordinate system) is added to each workpiece origin point offset amount. This causes the workpiece coordinate systems from G54 to G59 to be shifted by the same distance.



# Setting Coordinate Values (Amount of Movement)

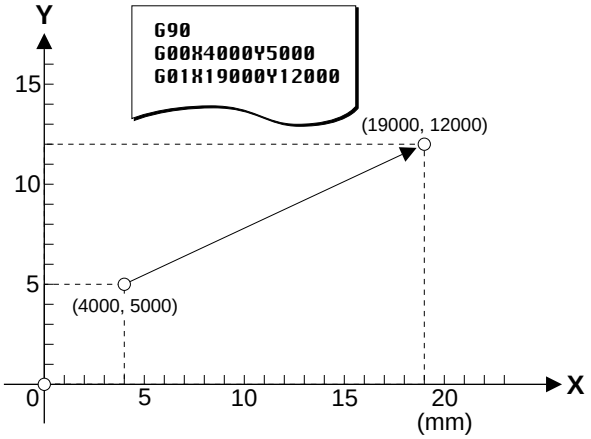
The addresses “X, Y, and Z” or “I, J, and K” are used, followed by the coordinate specification.

X, Y, and Z: These specify coordinate values for positioning (G00), linear interpolation (G01), and the like. X, Y, and Z represent the coordinates for the X, Y, and Z axes, respectively. It is not necessary to specify all three. As an example, if you want to leave the Y and Z axes as they are, and shift only the X axis by 10 mm, “G00X10.0” should be input.

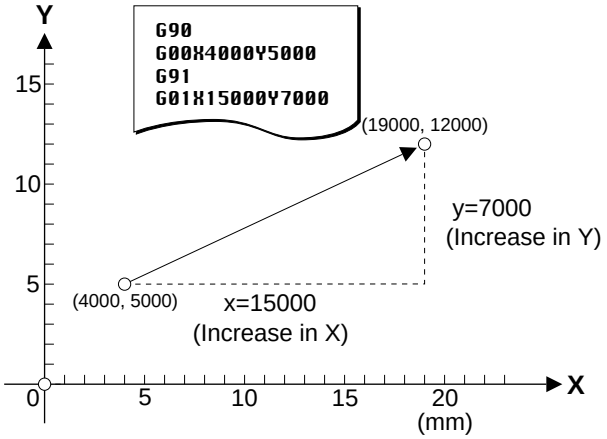
I, J, and K: These specify the center point for circular interpolation (G02 and G03). I, J, and K represent the movement for the X, Y, and Z axes from the current tool position, respectively. Another method of circular interpolation involves specifying the radius. Refer to page 18 for details.

## Absolute and Incremental

There are two types of coordinate specifications: absolute and incremental. These are toggled by G90 and G91 (see page 47).
The figure below shows the difference between absolute and incremental specifications on an X-Y plane. Absolute specifications indicate the position as the distance from the workpiece coordinate origin, whereas incremental specifications indicate the amount of movement from the current position.
Programming that specifies absolute coordinates is called “absolute programming,” and programming which specifies incremental coordinates is termed “incremental programming.”




**Absolute**




**Incremental**

The settings for G90 or G91 made on the PNC-300G remain in effect unless changed by programming.
There are no special rules for deciding when to use an absolute or incremental program. Examine the drawing and choose the one which makes for the simplest program.

# Setting the Measurement Unit

G20 and G21 can be used to set the measurement unit used for movement, feed rate, and offset amounts.

G20: Inch input
G21: Millimeter input

Either G20 or G21 is set at the start of the program, before setting the coordinate system. G20 and G21 should not be changed during the course of a program. If the unit is not set by programming, the setting made on the PNC-300G is used.
The minimum units differ for inch input and millimeter input.

| | Minimum units |
|---|---|
| Inch input | 0.0001" |
| Millimeter input | 0.001 mm |

| | Command | G20 | G21 |
|---|---|---|---|
| Movement | G00X10.0 | 10" | 10 mm |
| | G00X10000 | 1" | 10 mm |
| Feed rate | F60.0 | 60 inch/min. | 60 mm/min. |
| | F45000 | 4.5 inch/min. | 45 mm/min. |
| Offset amounts | G10P01R10.0 | 10" | 10 mm |
| | G10P01R10000 | 1" | 10 mm |

# Real-number Entry and Integer Entry

Movement amount (length), feed rate, and time may be input using either real-number entry or integer entry.
Input of a number which contains a decimal point (for example “10.0” or “10.”) is called real-number entry, and input of a number without a decimal point is called integer entry. A value such as “10.0” where the portion to the right of the decimal point is zero may be abbreviated to “10.” with no change in value.
When real-number entry is used, the numerical value is interpreted as being in the measurement unit that has been set. When integer entry is used, the numerical value is interpreted as being the minimum unit of the measurement unit that has been set. Some examples of this are as follows.

| | | |
|---|---|---|
| Amount of movement (length) | X10.0 | 10 mm or 10"* |
| | X1000 | 1 mm or 0.1"* |
| Feed rate | F60.0 | 60 mm/min. or 60 inch/min.* |
| | F120000 | 120 mm/min. or 12 inch/min.* |
| Time | G04X10.0 | 10 sec. dwell |
| | G04X10000 | 1 sec. dwell |

* The measurement unit is set by G20 (inch input) or G21 (millimeter input).
* If the unit is not set by programming, the setting made on the PNC-300G is used.

There are two types of methods for setting the speed of the spindle motor: setting as an rpm value, and setting as a numerical code. See page 54 for details.

# Program Number

A main program or subprogram calls and executes another program by specifying a program number. The program number appears at the start of the program.
A program number is specified by appending an integer of up to four digits after the letter “O.” The range for program numbers is from 0001 to 9999 -- “0” (zero) may not be used as a program number.
The section that extends from the place where the program number is input to an M02, M30, or M99 is recognized as one program (or subprogram). M02 or M30 is used to indicate the end of a main program, and M99 is used to signal the end of a subprogram.

# Sequence Numbers

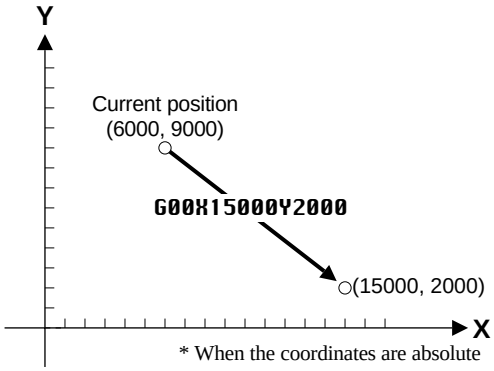
A sequence number is an integer number for a block. It is specified at the start of the block.
A sequence number may either be present or absent from any or all blocks. There is also no need for sequence numbers to be consecutive, or to be arranged in order from smaller to larger numbers. However, consecutive sequence numbers are customarily used to mark critical places within a program.
A sequence number is specified by appending an integer of up to four digits after the letter “N.” The range for program numbers is from 0001 to 9999. A sequence number cannot be substituted for a program number.

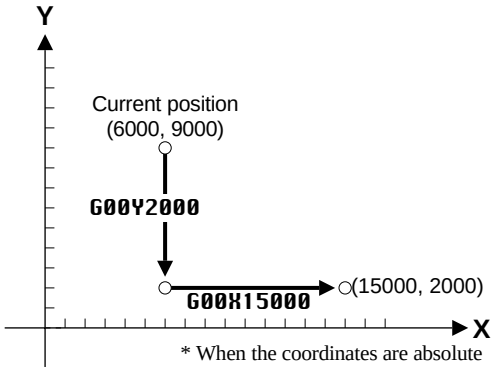
# Optional Block Skip

This function makes it possible to skip over a desired block within a program. Optional block skip is specified at the start of the program.
At the start of a block, a “/” (slash) is followed by a skip number of from 1 to 10. An absent value is interpreted as “1.” When entering two or more optional block skips in a single block, however, the numerical values should be specified.
A skip number can be used to cause only a particular block among blocks which have a “/” to be skipped. The number to be skipped is set on the PNC-300G (see the User's Manual for information on how to make this setting).

# Positioning (G00)

This moves in a straight line at maximum speed from the current tool position to the specified point. The word for positioning is “G00.” The addresses X, Y, and Z are used to specify the destination point. When X, Y, and Z are all specified, the three axes move simultaneously.
If the tool path is blocked by the workpiece or another object during movement, it is necessary to take steps to prevent the tool from striking the object, and one way to do this is to move each axis one at a time. An example of this would be to use the absolute specification “G00Z5000” to raise the tool, followed by “G00X1000Y1000” for horizontal movement.



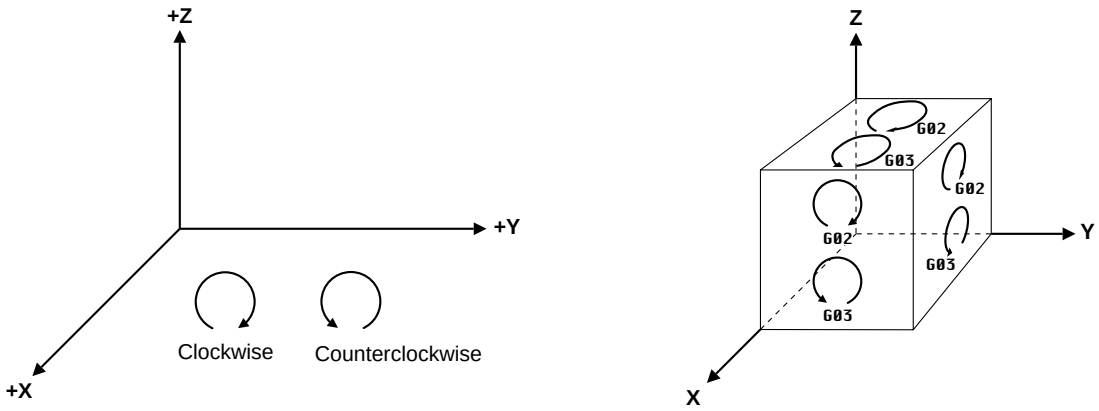




# Linear Interpolation (G01)

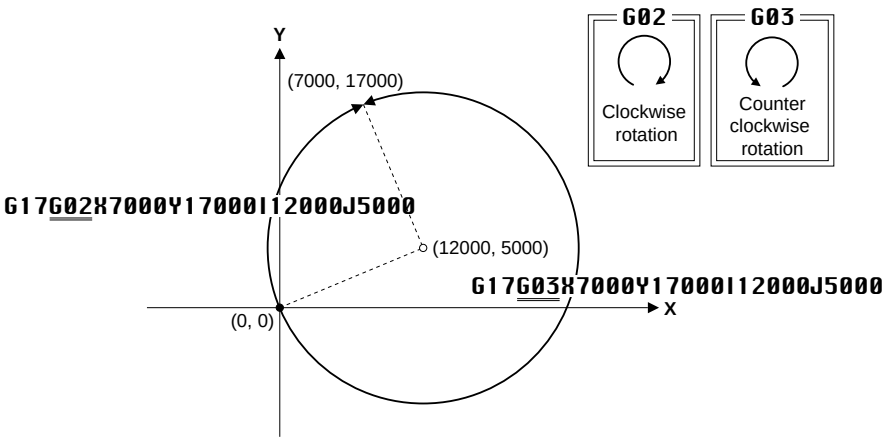
This cuts in a straight line from the current position to the specified point. The word for linear interpolation is “G01.” The addresses X, Y, and Z are used to specify the destination point. When X, Y, and Z are all specified, the three axes move simultaneously.
*G01 does not include the function for starting the spindle motor.* This means that if the spindle motor is not already turning, the M03 word must be given beforehand to start it.
In actual cutting, compensation for the tool diameter is required. Refer to page 30 for more information on compensation of the tool diameter.

# Circular Interpolation (G02 and G03)

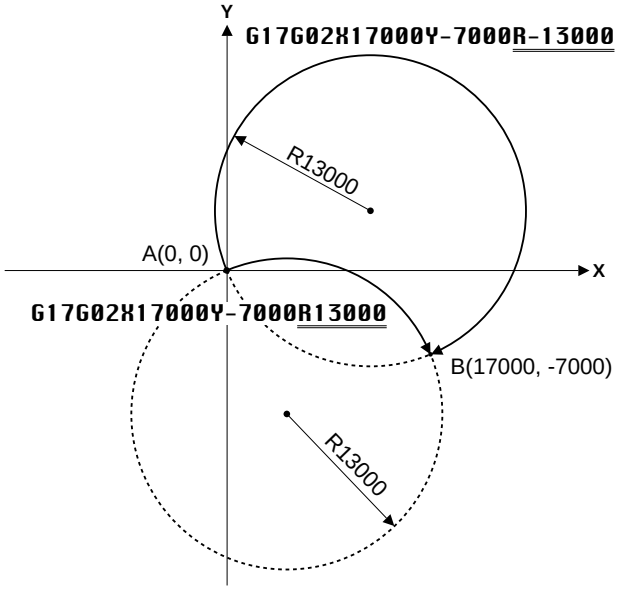
This cuts a circular shape from the current position to the specified point. The words for circular interpolation are “G02” and “G03.” Any address of the set X, Y, and Z is used to specify the destination coordinates, and any address of the set I, J, and K is used to specify the center of the circle. I, J, and K always specify the movement distance (incremental value) to the centerpoint of the circle or arc, with no regard for G90 or G91 (see pages 10 and 47).
*G02 and G03 do not include the function for starting the spindle motor*. This means that if the spindle motor is not already turning, the M03 word must be given beforehand to start it.
G02 and G03 interpolate in different directions — clockwise for G02 and counterclockwise for G03.
Circular interpolation can be carried out on any of the two-dimensional planes — the X-Y plane, the Z-X plane, or the Y-Z plane. The desired plane is specified with G17 (X-Y plane), G18 (Z-X plane), or G19 (Y-Z plane). See page 21 for the details of plane specification.




Even when the point for the destination and the center of the circle are identical, circular interpolation is carried out as shown below according to the direction of interpolation.



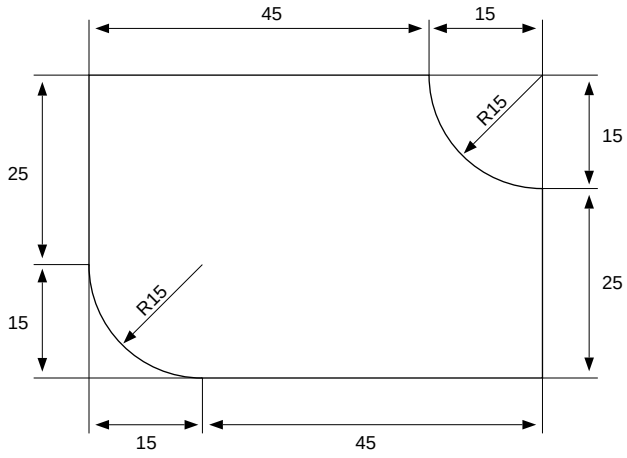

There is another method, which involves specifying the radius of the circle instead of specifying the circle's center point. This method is convenient because numerical values read from the drawing can be used directly.
Two circles with identical radii and passing through two points exist. This means that if the interpolation direction, radius of the circle, and point for the destination of interpolation have been specified, there are two circles. These two circles can be differentiated by specifying a positive value for the radius if the center angle is 180 degrees or less, and a negative radius if the center angle exceeds 180 degrees.

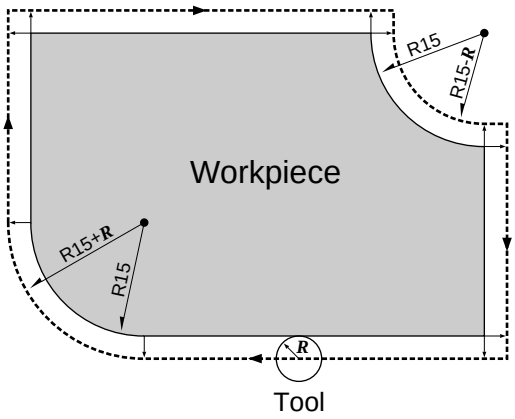


# Cutter Compensation (G40, G41 and G42)

The movement of the tool specified by the program is the path taken by the center of the tool. Because the tool has a certain thickness (i.e., a certain diameter), it will over-cut by an amount equal to its radius if the coordinates on the drawing are input just as they are.
To cut a shape as specified by the drawing, the tool must be made to move at a place which shifted away by a distance equal to the tool radius. This is called the "tool-diameter offset."
Using this function makes it possible to input the values from the drawing as coordinate values (or amounts of movement) with no need for modification, thus facilitating programming. Also, if cutting is to be performed with a tool that has a different tool diameter, it is only necessary to change the amount of offset.
The words for cutter compensation are "G40," "G41," and "G42."

**G40**: Cancel cutter compensation
**G41**: Cutter compensation -- left
**G42**: Cutter compensation -- right

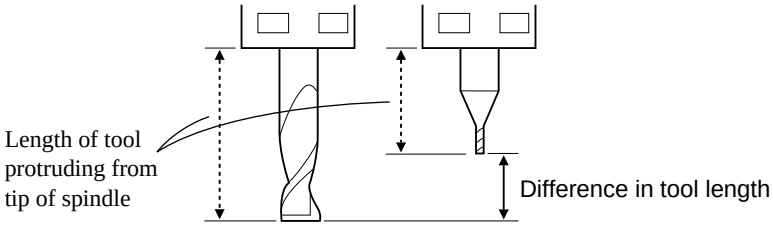



: Tool path

# Tool-length Compensation (G43, G44, and G49)

Tool-length compensation offsets the difference between the tool length assumed by the program and the actual tool length. When compensation for tool length is performed using program coordinate values, the entire program must be changed. Using the tool-length compensation function makes it possible to absorb differences in tool length by changing only the amount of compensation (i.e., the amount of offset).
With NC machines that can change tools automatically, tool-length compensation can also be used to offset the difference in tool length before and after the tool-change. Because such NC machines can determine in advance the length of the tool protruding from the tip of the spindle, compensation of tool length by programming is simple. The PNC-300G has no function for automatic tool-changing, so it cannot determine the in advance the length of the tool protruding from the tip of the spindle.




The words for tool-length compensation are "G43," "G44," and "G49."

**G43**: Positive tool-length compensation (addition of amount of offset in Z-axis direction)
**G44**: Negative tool-length compensation (subtraction of amount of offset in Z-axis direction)
**G49**: Cancel tool-length compensation

# Feed Rate

This determines the feed rate for the workpiece and the spindle. The **F** function is used to make the setting.
The feed rate generally varies according to the cutting parameters (such as the spindle speed, tool diameter, and workpiece material).
The **F** function is activated at the start of the block in which it is specified.
The feed rate is specified as a real or integer value following the "F."

| | |
|---|---|
| `F100.0` | Feed rate set at 100 mm/min. |
| `F100000` | Feed rate set at 100 mm/min. (when millimeter input is used) |

### Feed Rate Override

Feed rate override" refers to manually changing the feed rate specified by the program. This is mainly used to adjust the feed rate during cutting. This function may or may not be supported, depending on the machine. Refer to the User's Manual for information on how to make this setting.

# Spindle Motor Control (M03 and M05)

These turn the spindle motor on or off. The **M** function is used for this.
M03 and M05 are used to control the spindle motor. M03 starts rotation of the spindle, and M05 stops it.
These functions are activated at the end of the block in which they are specified together.

# Spindle Motor Speed

The **S** function is used to set the speed for the spindle motor.
The **S** function does not include the function for starting the spindle motor. It functions only when the spindle has been started with M03, or when the spindle is already turning.
This function is activated at the end of the block in which it is specified.

# Fixed Cycle

The fixed cycle (or canned cycle) is a command for executing a series of pre-established operations for cutting, such as drilling.
This command can execute several blocks of cutting operations in a single block, thereby simplifying programming. This also reduces the amount of data.
See page 23 for details of the specifications for the fixed cycle.

# Program-related Errors

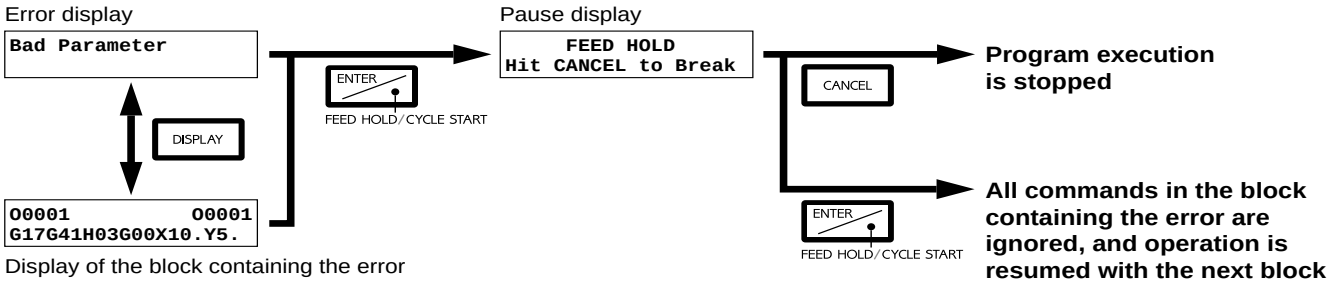
An error is generated when an unsuitable value has been set for a parameter, or when the PNC-300G cannot interpret the program. Only error messages related to programming are described in this manual.

## Errors Occurring During Program Execution

When an error occurs, operation pauses and an error message is displayed. When this happens, press the **[DISPLAY]** key to display the block which contains the error.
The cutting operation can be continued as-is after an error has been displayed, but this is not recommended because operation following the occurrence of an error may not be correct. Instead, stop program execution and correct the place where the error was generated.

| Error message | Description |
|---|---|
| `Bad Parameter` | The value of a parameter exceeds the allowable range, or the value of the radius for circular interpolation or the amount of offset is not correct. |
| `Address Undefined` | Only a parameter has been set. The code which specifies the parameter has not been set. |
| `Parameter Undefined` | A parameter has not been set. |
| `Code Cannot Execute` | This is displayed when an attempt was made to execute an unrecognizable command, when cutter compensation was started while in the circular interpolation mode, or when an attempt was made to execute a command which cannot be used during tool-diameter or tool-length compensation. |
| `Program Number Not found` | The program number specified by M98 or M99 could not be found. |
| `Sub-Program Nest Over` | An attempt was made to call a fifth-level subprogram from a fourth-level subprogram of a main program. |

Display Operations When an Error Occurs During Program Execution

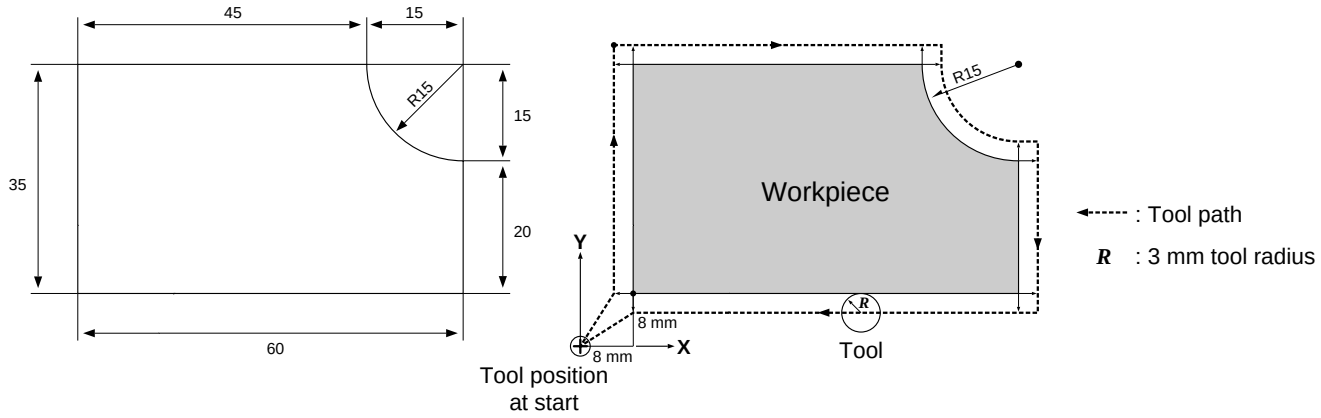



## Errors When Execution of Cutting Data Is Attempted

When program execution is attempted, subprograms are checked before cutting is performed. One of the following error messages may be displayed as a result of this check.

| Error message | Description |
|---|---|
| `Sub-Program Table Over` | There are more than ten subprograms. The maximum number of subprograms that can be specified in one set of data is ten.<br>Press the **[ENTER]** key to call up the coordinate display. After correcting the program, resend the data. |
| `Duplicate Sub-Program Number` | The program contains multiple subprograms with the same program number. The same program number may not be set for more than one subprogram within a single set of data.<br>Press the **[ENTER]** key to call up the coordinate display. After correcting the program, resend the data. |

# Sample Program




| | |
|---|---|
| % | Data start |
| O0001 | Program number |
| N01 G91 | Incremental programming |
| N02 G21 | Set millimeter input |
| N03 G92X0Y0Z0 | Set workpiece coordinate origin point |
| N04 G10P01R3.0 | Set 3 mm of offset for offset number 1 |
| N05 G00Z5.0 | Move tool to position of 5 mm on Z axis (X0Y0 unchanged) |
| N07 F300.0S6000M03 | Set motor speed to 6,000 rpm and feed rate to 300 mm/min., and rotate spindle |
| N06 G17G41D01G00X8.0Y8.0 | Start cutter compensation and move tool to X-axis 8 mm and Y-axis 8 mm |
| N08 G01Z-7.0 | Linear interpolation to -7 mm on Z axis |
| N09 G01Y35.0 | Linear interpolation to 35 mm on Y axis |
| N10 X45.0 | Linear interpolation to 45 mm on X axis |
| N11 G03X15.0Y-15.0I15.0 | Circular interpolation (counterclockwise) to position X 15 mm Y -15 mm from current tool position |
| N12 G01Y-20.0 | Linear interpolation to -20 mm on Y axis |
| N13 X-60.0 | Linear interpolation to -60 mm on X axis |
| N14 Z7.0 | Linear interpolation to 7 mm on Z axis |
| N15 G40G00X-8.0Y-8.0 | Cancel cutter compensation and return to start point |
| N16 M05 | Main spindle rotation halt |
| M02 | Program end |
| % | Data end |

* The example program shown above is for when using ISO or ASCII as the code.

# Part 2 Reference

## How to Read Part 2

**Preparatory Functions (G Functions)**

Word — ***G00*** | ***Positioning*** — Word function

Words in square brackets ("**[]**") may be omitted. Parameters are given in italics (such as "*x*," "*y*," and "*feed rate*").
*The words enclosed in curly brackets ("{}") are a range of available selections. Any one may be chosen.*

**Format**

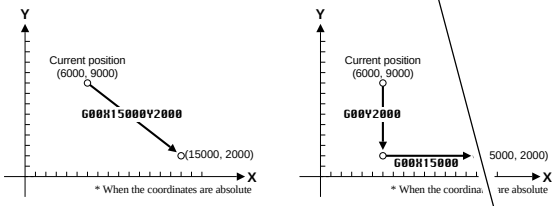
**G00[X** *x***][Y** *y***][Z** *z***]**

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |
| *z* | Coordinate or movement distance (Z axis) | Range 1 | Maximum cutting range |

**Description**

This effects movement in a straight line and at maximum speed from the current tool p ion to the specified coordinate point. When incremental programming is used, the tool moves the specified movement distance. G0 nores any set feed rate, and always effects movement at maximum speed.
The destination's coordinates (or movement distances) are specified with addresses X, Y, l Z. It is not necessary to specify values for every one of these addresses. As an example, if only the X axis is specified (e.g., G00X10 the tool moves only along the X axis, with no movement on the Y or Z axes. This is also the case when only the Y axis, Z axis, X and xes, Y and Z axes, or Z and Y axes are specified. When addresses X, Y, and Z are all specified, the tool moves along all three axes ultaneously.
G00 is also effective outside the block until a word of the same type is encountered. If XxYy. is specified in the block after specifying G00, with no G01, G02, or G03, linear movement to the specified coordinate is effected.
If the tool path is blocked by the workpiece or another object during movement, it is necessary t ke steps to prevent the tool from striking the object, and one way to do this is to move each axis one at a time.

This is an explanation of the functions of the word and its parameters, along with cautions or other important points to observe when using.




For example,

**G17** { **G02** / **G03** } **[X** *x* **][Y** *y* **]** { **[I** *cx* **][J** *cy* **]** / **R** *radius* }

uses this shorthand form to indicate the following four expressions:

**G17G02[X** *x* **][Y** *y* **][I** *cx* **][J** *cy* **]**
**G17G03[X** *x* **][Y** *y* **][I** *cx* **][J** *cy* **]**
**G17G02[X** *x* **][Y** *y* **] R** *radius*
**G17G03[X** *x* **][Y** *y* **] R** *radius*

The range and functions of a parameter are shown in table form.
"Range 1" and "Range 2" are shorthand expressions for ranges in which an error does not occur. These correspond to the following ranges.

Range 1: Integer entry -- -67,108,863 to 67,108,863
Real-number entry -- -67,108.863 to 67,108.863 (when millimeter input is used)
-6,710.8863 to 6,710.8863 (when inch input is used)

Range 2: -67,108,863 to 67,108,863

# Preparatory Functions (G Functions)

## G00 Positioning

### Format

**G00[X** *x* **][Y** *y* **][Z** *z* **]**

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |
| *z* | Coordinate or movement distance (Z axis) | Range 1 | Maximum cutting range |

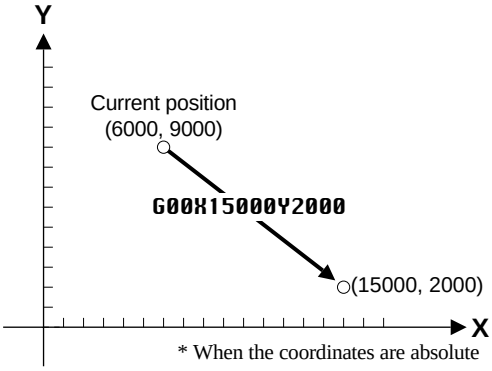
### Description

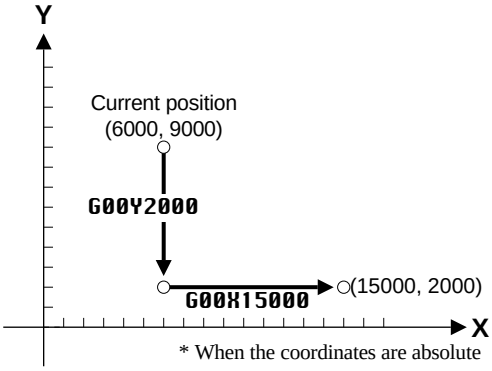
This effects movement in a straight line and at maximum speed from the current tool position to the specified coordinate point. When incremental programming is used, the tool moves the specified movement distance. G00 ignores any set feed rate, and always effects movement at maximum speed. When "**RAPID OVERRIDE**" has been set on the PNC-300G, however, operation is according to this setting.

The destination's coordinates (or movement distances) are specified with addresses X, Y, and Z. It is not necessary to specify values for every one of these addresses. As an example, if only the X axis is specified (e.g., G00X100), the tool moves only along the X axis, with no movement on the Y or Z axes. This is also the case when only the Y axis, Z axis, X and Y axes, Y and Z axes, or Z and Y axes are specified. When addresses X, Y, and Z are all specified, the tool moves along all three axes simultaneously.

G00 is also effective outside the block until a word of the same group is encountered. If **X** *x* **Y** *y* **Z** *z* is specified in the block after specifying G00, with no G01, G02, or G03, linear movement to the specified point is effected.

If the tool path is blocked by the workpiece or another object during movement, it is necessary to take steps to prevent the tool from striking the object, and one way to do this is to move each axis one at a time.



# G01 Linear Interpolation

## Format

**G01[X** *x* **][Y** *y* **][Z** *z* **]**

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |
| *z* | Coordinate or movement distance (Z axis) | Range 1 | Maximum cutting range |

## Description

This effects linear cutting from the current tool position to the specified coordinate. When incremental programming is used, cutting for the specified movement distance is performed.

Cutting is performed at the spindle speed and feed rate that have been specified. Refer to page 56 for an explanation of the feed rate and to page 54 for a description of spindle speed.

The destination's coordinates (or movement distances) are specified with addresses X, Y, and Z. It is not necessary to specify values for every one of these addresses. As an example, if only the X axis is specified (e.g., G01X100), the tool moves only along the X axis, with no movement on the Y or Z axes. This is also the case when only the Y axis, Z axis, X and Y axes, Y and Z axes, or Z and Y axes are specified. When addresses X, Y, and Z are all specified, the tool moves along all three axes simultaneously.

G01 is also effective outside the block until a word of the same group is encountered. If **X** *x* **Y** *y* **Z** *z* is specified in the block after specifying G01, with no G00, G02, or G03, linear interpolation to the specified point is effected. This makes it possible to carry out continuous linear interpolation.

G01 does not include the function for starting the spindle motor. If the spindle motor is not already turning, the M03 word should be given beforehand to start it.

The specified tool movement is the path followed by the center of the tool. Programming should be done so that the tool passes over a path which is offset by a distance equal to the radius of the tool.

# G02 and G03 Circular Interpolation

## Format

G17 { G02 / G03 } [X *x* ][Y *y* ] { [I *cx* ][J *cy* ] / R *radius* }

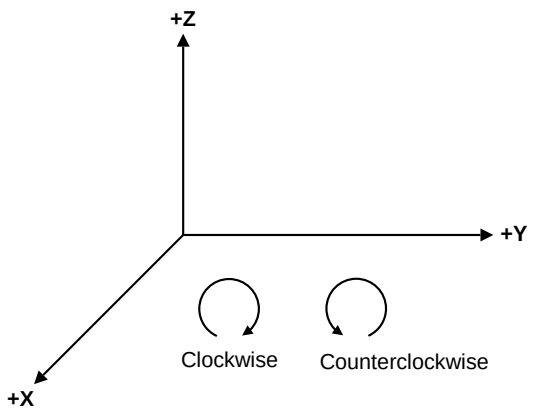
G18 { G02 / G03 } [X *x* ][Z *z* ] { [I *cx* ][K *cz* ] / R *radius* }

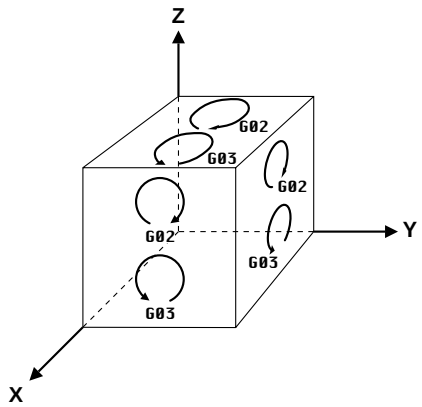
G19 { G02 / G03 } [Y *y* ][Z *z* ] { [J *cy* ][K *cz* ] / R *radius* }

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |
| *z* | Coordinate or movement distance (Z axis) | Range 1 | Maximum cutting range |
| *cx* | Movement distance to circle (arc) center (X axis) | Range 1 | Maximum cutting range |
| *cy* | Movement distance to circle (arc) center (Y axis) | Range 1 | Maximum cutting range |
| *cz* | Movement distance to circle (arc) center (Z axis) | Range 1 | Maximum cutting range |
| *radius* | radius | Range 1 | Maximum cutting range |

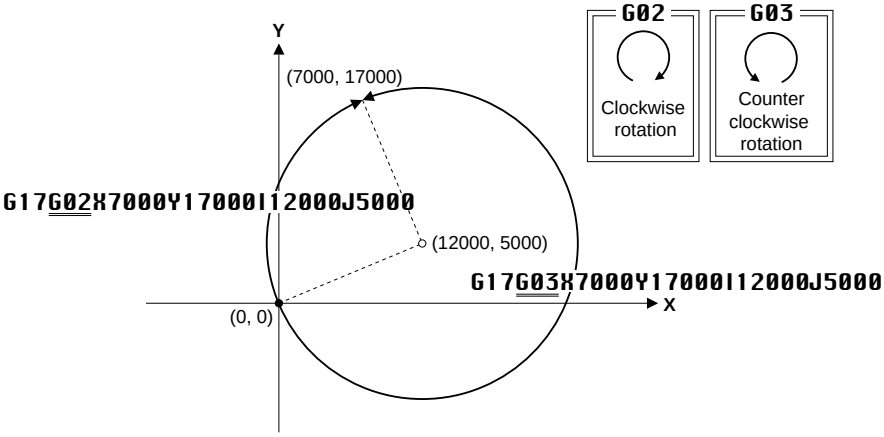
## Description

This cuts an arc at the specified feed rate and spindle speed from the current tool position to the specified point. Circular interpolation can be carried out only on the X-Y plane, Z-X plane, or Y-Z plane.
The specification for the two-dimensional plane is made with G17, G18, or G19. These specify the X-Y plane, Z-X plane, and Y-Z plane, respectively. The specification for the X-Y plane is enabled when the machine's power is switched on.
To specify the destination point for interpolation, use is made of addresses X and Y for G17, X and Z for G18, and Y and Z for G19.
The center point for the arc is specified with addresses I and J for G17, I and K for G18, and J and K for G19. I, J, and K always specify the movement distance (incremental value) to the centerpoint of the circle or arc, with no regard for G90 or G91 (see page 26). It is also possible to specify the radius R for the arc instead of using I, J, or K.
When the point of the current tool position is specified as the destination for interpolation, a circle with a center angle of 360° is cut.
G02 and G03 differ in the direction of interpolation for the arc (i.e., the direction of tool movement). G02 performs clockwise circular interpolation, whereas G03 performs counterclockwise interpolation.

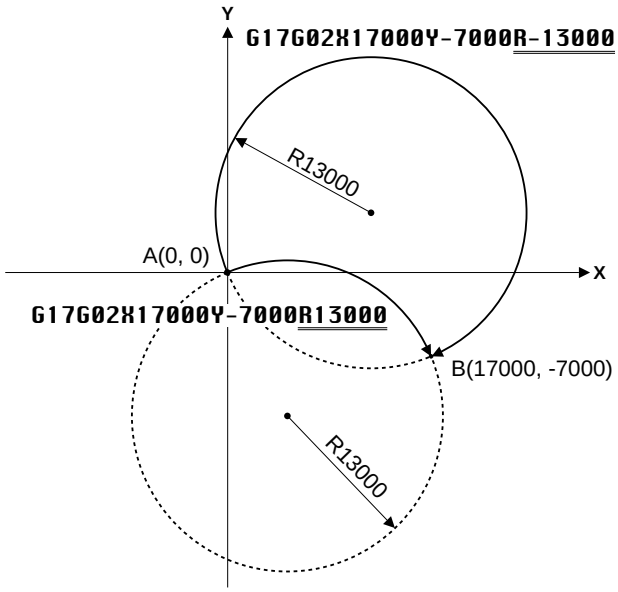


Even when the point for the destination and the center of the circle are identical, circular interpolation is carried out as shown below according to the direction of interpolation.



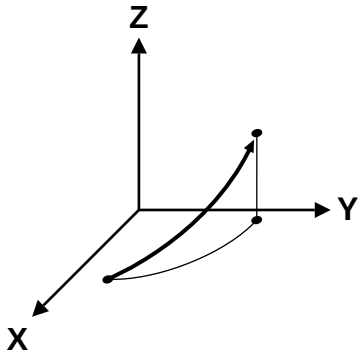

Two circles with identical radii and passing through two points exist. This means that if the interpolation direction, radius of the circle, and point for the destination of interpolation have been specified, there are two circles. These circles can be differentiated by specifying a positive value for the radius if the center angle is 180 degrees or less, and a negative radius if the center angle exceeds 180 degrees.




G02 and G03 are also effective outside the block until a word of the same group (G00, G01, G02, or G03) is encountered. G17, G18, G19 also remain effective even outside the block until a word of the same group (G17, G18, or G19) is specified.
G02 and G03 do not include the function for starting the spindle motor. This means that if the spindle motor is not already turning, the M03 word must be given beforehand to start it.
The specified tool movement is the path followed by the center of the tool. Please make calculations to have the tool pass through a location that is offset by a distance equal to its radius, and then carry out the programming accordingly. An error is generated when an attempt is made to execute a code for starting cutter compensation (G41 or G42) while in the circular interpolation mode.

# Helical Interpolation

When an axis is added to the coordinate point for the destination of interpolation, movement in the form of a helix is carried out, as shown below. This is called helical interpolation. A three-dimensional curve is cut by performing a synchronized linear operation along the added axis while carrying out circular interpolation.




## Format

```
G17 { G02 } [X x ][Y y ] { [I cx ][J cy ] } [Z z ]
    { G03 }              { [R radius ]  }

G18 { G02 } [X x ][Z z ] { [I cx ][K cz ] } [Y y ]
    { G03 }              { [R radius ]  }

G19 { G02 } [Y y ][Z z ] { [J cy ][K cz ] } [X x ]
    { G03 }              { [R radius ]  }
```

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |
| *z* | Coordinate or movement distance (Z axis) | Range 1 | Maximum cutting range |
| *cx* | Movement distance to circle (arc) center (X axis) | Range 1 | Maximum cutting range |
| *cy* | Movement distance to circle (arc) center (Y axis) | Range 1 | Maximum cutting range |
| *cz* | Movement distance to circle (arc) center (Z axis) | Range 1 | Maximum cutting range |
| *radius* | radius | Range 1 | Maximum cutting range |

# G04 Dwell

## Format

**G04[X** *time(X)* **]**
**G04[P** *time(P)* **]**

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *time(X)* | Dwell time | Range 1 | — |
| *time(P)* | Dwell time | Range 2 | — |

## Description

G04 specifies the time interval for moving from the previous block to the next block. G04 is normally specified as a single block all by itself.
G04 is used with the aim of cutting a precise angle, ensuring precision when cutting the bottom of a drilled hole, or the like.
The desired dwell time is specified after X or P. X and P are functionally equivalent, and may be used interchangeably. A numerical value (real or integer) is used to specify the dwell time. The specified time is in seconds when a real number is used, and in milliseconds when an integer is used.

| | |
|---|---|
| **G04X10.0** | 10-second dwell |
| **G04X10000** | 10-second dwell (in millisecond units) |

# G10 Data Setting

## Format

**G10L2[X** *coordinate* **][X** *x***][Z** *z***]**
**G10[P** *number* **][R** *offset* **]**

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *coordinate* | Work coordinate | Range 2 | 0—6 |
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |
| *z* | Coordinate or movement distance (Z axis) | Range 1 | Maximum cutting range |
| *number* | Offset number | Range 2 | 1—10 |
| *offset* | Amount of offset | -300—300 [mm]<br>or -11.81"—11.81" | -300—300 [mm]<br>or -11.81"—11.81" |

## Description

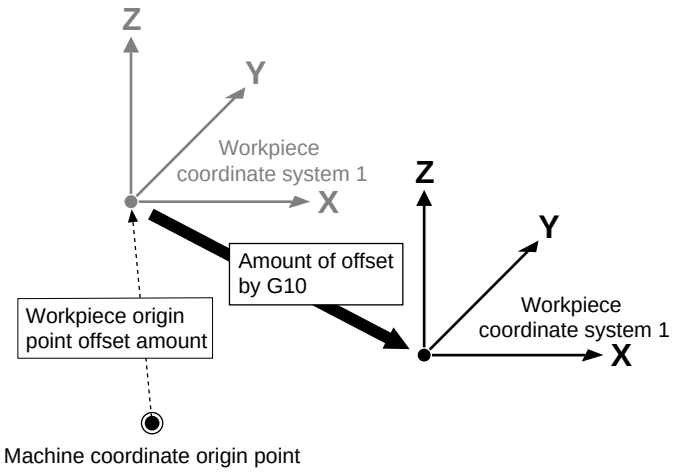
This sets the amount of shift for workpiece coordinate systems as well as the amount of offset used by the cutter compensation and tool-length compensation.

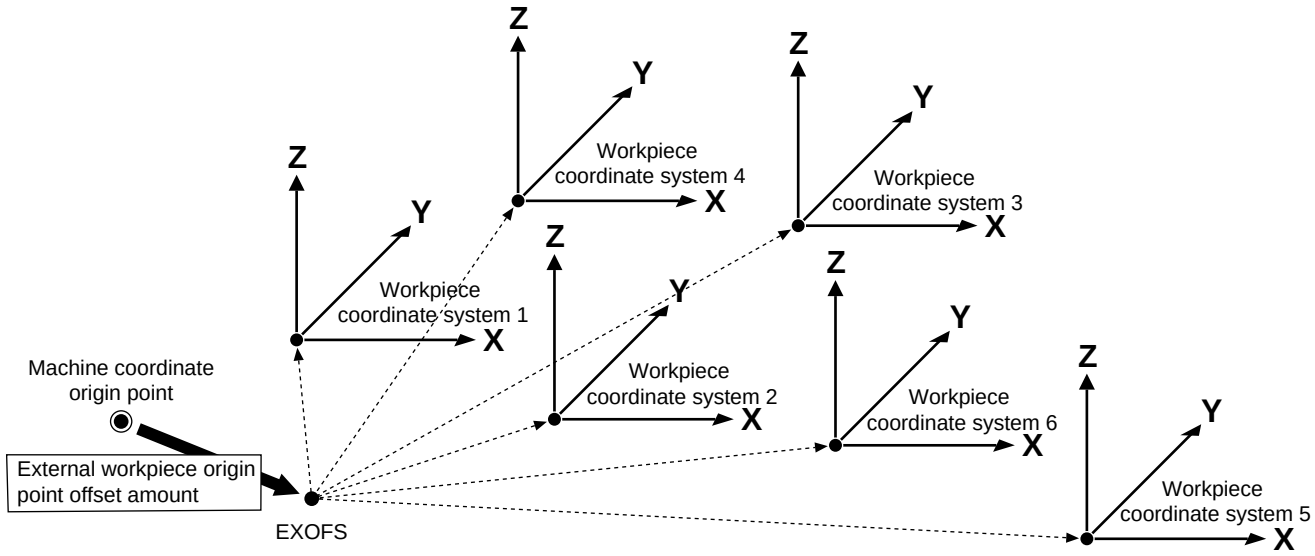
### Setting the Amount of Shift for Workpiece Coordinate Systems

This sets the amount of shift for workpiece coordinate systems 1 to 6 of G54 and G59. The format for setting the amount of shift is as follows.

**G10L2[X** *coordinate* **][X** *x***][Z** *z***]**

The number of the workpiece coordinate system (1 to 6) is specified by *coordinate*. Specifying “0” causes the amount of shift (EXOFS) to be set for all workpiece coordinate systems.
The amounts of shift for the coordinate system are specified by *x*, *y*, and *z*. When “0” has been specified for *coordinate*, the value is set with the machine coordinate origin taken as 0. When *coordinate* specifies the number of a workpiece coordinate system (1 to 6), the value is set with a point shifted from the machine coordinate origin by a distance equal to EXOFS taken as 0.
Refer to page 7 for detailed information about coordinate systems.






## Setting the Amount of Offset

This sets the amount of offset used by the cutter compensation (G41 and G42) and tool-length compensation (G43 and G44). The format for setting the amount of offset is as follows.

**G10[P** *number* **][R** *offset* **]**

The offset number for which an amount of offset is to be specified is indicated by *number*. An integer from 1 to 10 may be specified. The amount of offset is indicated by *offset*. A setting within the range of -300.00 to 300.00 mm (or within the range of -11.81" to 11.81" for inch input) may be made. However, the range for the amount of offset used during cutter compensation is -10.00 to 10.00 mm (-0.39" to 0.39" for inch input). An error is generated if a value for an outside the range of -10.00 to 10.00 mm is set for the specified offset number when cutter compensation is started.
Setting a negative value for the amount of offset causes the direction of offset to be reversed.

Example: When an amount of offset of -3 mm is specified for offset number 1

**G00G41H01X100.0**

Offset of -3 mm to the left-hand side relative to the direction of forward movement = 3 mm offset to the right-hand side

* Reverse-direction offsets are also applied to G42, G43, and G44.

The amount of offset can be specified using the LCD panel on the PNC-300G. (Refer to the User's Manual for a description of the procedure.)

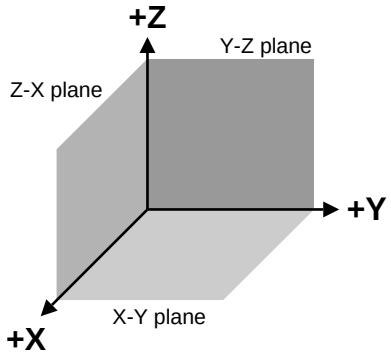
# G17, G18 and G19 Plane

## Format

```
G17
G18
G19
```

## Description

This specifies a two-dimensional plane for circular interpolation (G02 or G03).
G17 specifies the X-Y plane, G18 specifies the Z-X plane, and G19 specifies the Y-Z plane. Each of these is normally used in combination with G02 or G03 in the same block. (Refer to page 22).




# G20 and G21 Setting the Measurement Unit

## Format

```
G20
G21
```

## Description

This sets the measurement unit used for movement, feed rate, and offset amounts. G20 sets inch input, and G21 sets millimeter input. Either G20 or G21 is set at the start of the program, before setting the coordinate system. G20 and G21 should not be changed during the course of a program.
The minimum units differ for inch input and millimeter input.

| | Minimum units |
|---|---|
| Inch input | 0.0001" |
| Millimeter input | 0.001 mm |

Changing the measurement unit results in interpretation as shown below.

| | Command | G20 | G21 |
|---|---|---|---|
| Movement | G00X1.0 | 1" | 1 mm |
| | G00X1000 | 0.1" | 1 mm |
| Feed rate | F60.0 | 60 inch/min. | 60 mm/min. |
| | F45000 | 4.5 inch/min. | 45 mm/min. |
| Offset amounts | G10P01R10.0 | 10" | 10 mm |
| | G10P01R10000 | 1" | 10 mm |

# G39 Corner-offset Circular Interpolation

## Format

G39[X *x*][Y *y*]

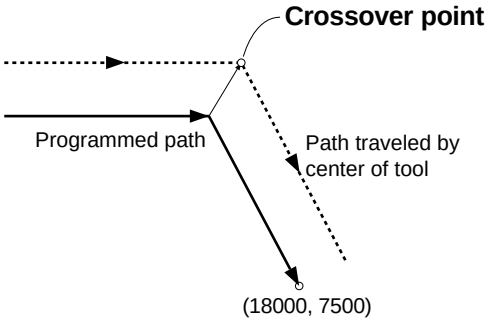
| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |

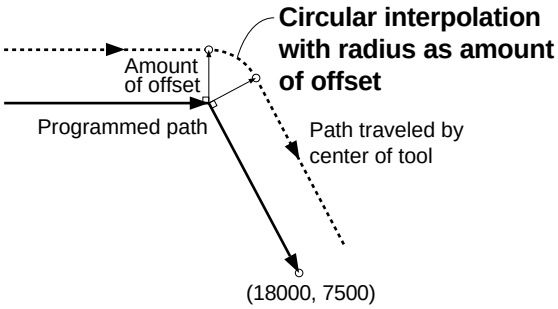
## Description

Corner-offset circular interpolation is a function which performs tool movement at crossover points during cutter compensation by means of circular interpolation. The radius of circular interpolation is the amount of offset (tool radius).
Corner-offset circular interpolation can be executed during cutter compensation, when G41 or G42 has already been executed.
The destination point (or amount of offset) after performing corner-offset circular interpolation are indicated by *x* and *y*.

```
••••••••
N0132 G17G00G41D01X1000Y1000
••••••••
N0163 G01X17000
N0164 X18000Y7500
••••••••
```

```
••••••••
N0132 G17G00G41D01X1000Y1000
••••••••
N0163 G01X17000
N0164 G39X18000Y7500
••••••••
```




G39 is a word which is effective only within a block. Arc interpolation is used only for corners specified as G39. G39 does not affect G00, G01, G02, or G03.

# G40, G41 and G42 Cutter Compensation

## Format

{ G00 / G01 } G40 [X *x* Y *y* ]

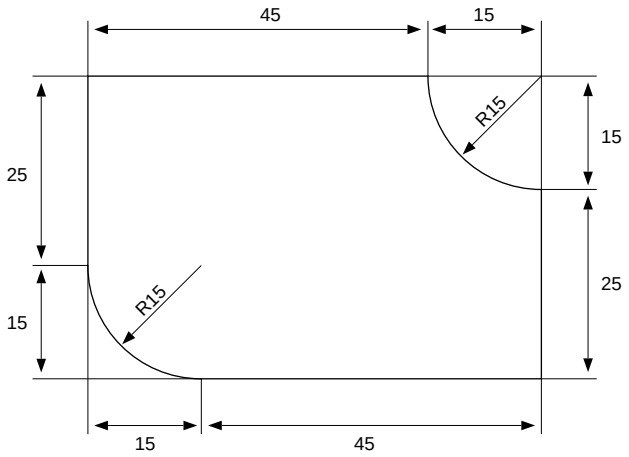
G17 { G00 / G01 } { G41 / G42 } D *number* [X *x* Y *y* ]

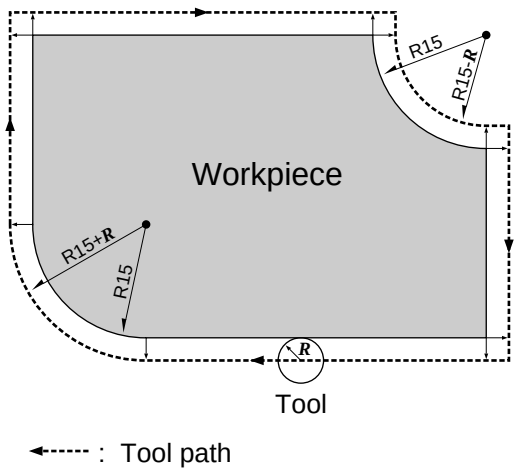
| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |
| *number* | Offset number | 0—10 | 0—10 |

## Description

The movement of the tool specified by the program is the path taken by the center of the tool. Because the tool has a certain thickness (i.e., a certain diameter), it will over-cut by an amount equal to its radius if the coordinates on the drawing are input just as they are.
To cut a shape as specified by the drawing, the tool must be made to move at a place which shifted away by a distance equal to the tool radius. This is called the "tool-diameter offset."
Using this function makes it possible to input the values from the drawing as coordinate values (or amounts of movement) with no need for modification, thus facilitating programming. Also, if cutting is to be performed with a tool that has a different tool diameter, it is only necessary to change the amount of offset.
The words for cutter compensation are "G40," "G41," and "G42."

**G40**: Cancel cutter compensation
**G41**: Cutter compensation -- left
**G42**: Cutter compensation -- right




### Restrictions on Cutter Compensation

Cutter compensation is subject to the following restrictions.

1. Cutter compensation can be performed only in the XY plane.
2. The amount of offset that can be used with cutter compensation is 10 mm or less. An error is generated if an offset number set with an amount of offset exceeding 10 mm is specified.
3. An error is generated if there is not at least one command code for movement between the start code (G41 or G42) and cancel code (G40) for cutter compensation.

4. Do not position two or more blocks without X- and Y-axis motion commands next to each other during tool diameter compensation. It may cause excessive or insufficient cutting depth.
5. No interference check for cutter compensation is performed. However, an error is generated if an attempt is made to machine the inner side of a circle or arc with an amount of offset that is larger than the radius for circular interpolation.
6. When circular interpolation has been specified, an error is generated if cutter compensation is started or canceled. When positioning (G00) or linear interpolation (G01) has been specified, cutter compensation should be started or canceled.
7. When cutter compensation for circular interpolation is performed, parameters cannot be changed using the display with operation paused.
8. When fixed-cycle operation has been specified, executing or canceling cutter compensation causes an error to be generated.
9. Performing any of the following operations or settings during cutter compensation causes an error to be generated.

| | |
|---|---|
| Changing the offset number | : To change the offset number, first cancel cutter compensation. Then execute G41 or G42 again, and change the offset number. |
| Switching the direction of compensation | : When G41 (cutter compensation -- left) has been used, executing G42 causes an error to be generated. Similarly, when G42 (cutter compensation -- right) has been used, executing G41 causes an error to be generated. |
| Specifying a plane | : Executing G17, G18, or G19 causes an error to be generated. |
| Scaling | : Executing G50 or G51 causes an error to be generated. |
| Fixed cycle | : Executing G81, G81, G82, G85, G86, or G89 causes an error to be generated. |
| Specifying a coordinate system | : Executing G10, G54 to G59, or G92 causes an error to be generated. |
| Tool-length compensation | : Tool-length compensation should be executed before executing cutter compensation. |
| 3D interpolation | : Executing 3D interpolation (simultaneous interpolation of the X, Y, and Z axes) causes an error to be generated. |
| Calling a subprogram or returning to the main program | : Executing M98 or M99 causes an error to be generated. |
| Tool change | : Executing M06 causes an error to be generated. |

## Setting the Amount of Offset

The PNC-300G allows amounts of offset to be set individually for offset numbers 1 to 10. An amount of offset can be set using either of two methods.

1. Using the display on the PNC-300G
   The PNC-300G's LCD screen and control keys are used to set the amount of offset. See the “User's Manual” for a description of the procedure.
2. Using code (G10)
   **G10[P** *number***][R** *offset***]**

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *number* | Offset number | Range 2 | 1—10 |
| *offset* | Offset value | -10—10 [mm]<br>or -0.39"—0.39" | -10—10 [mm]<br>or -0.39"—0.39" |

Note: The ranges shown above are the ranges where the amount of offset for cutter compensation can be used.

* If G41 or G42 is used to specify an offset number for which no amount of offset has been set with G10, the value that has been set on the PNC-300G is used.
* Setting a negative value for an amount of offset causes the offset direction of G41 and G42 to be reversed as shown below.

**G00G41D01X100.0** Offset of -3 mm to the left-hand side relative to the direction of forward movement = 3 mm offset to the right-hand side

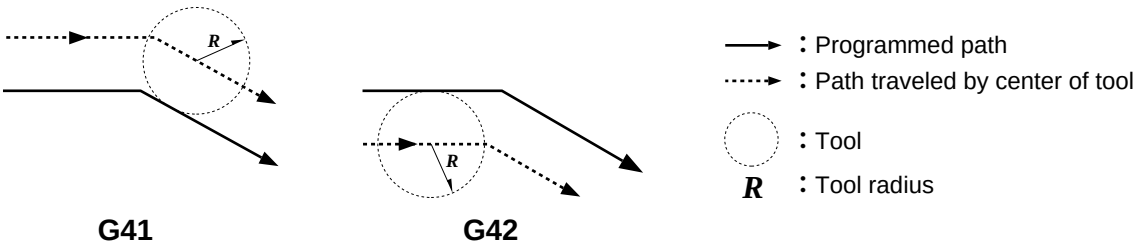
**G00G42D01X100.0** Offset of -3 mm to the right-hand side relative to the direction of forward movement = 3 mm offset to the left-hand side

* An amount of offset of zero is set for offset number 0. The amount of offset for offset number 0 cannot be changed.
* The amounts of offset set for offset numbers 1 to 10 are used by both cutter compensation and tool-length compensation. For example, if an amount of offset of 10 mm has been set for offset number 2, operation is as follows.

| | | |
|---|---|---|
| Cutter compensation | : **G00G41D02X100.0** | 10 mm offset to the left-hand side relative to the direction of forward movement |
| Tool-length compensation | : **G00G43H02Z7.0** | 10 mm offset in the positive direction of the Z axis |

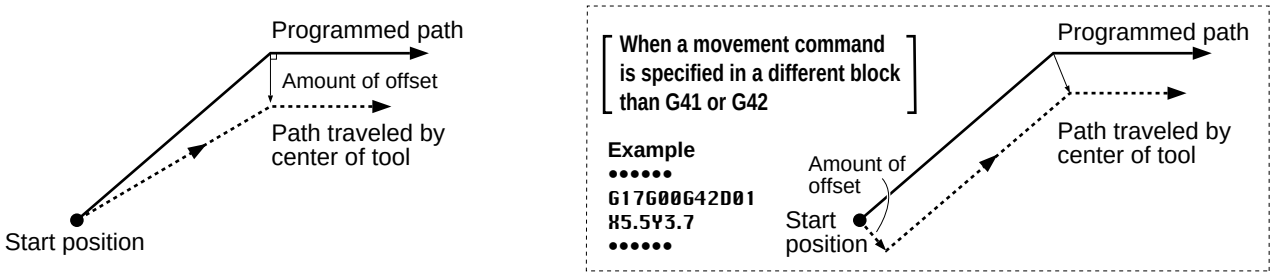
## Starting Cutter Compensation

Cutter compensation is started with G41 or G42. G41 performs offset to the left-hand side relative to the direction of forward movement. Similarly, G42 performs offset to the right-hand side relative to the direction of forward movement. The direction of offset cannot be changed while cutter compensation is in progress.




G41 or G42 is specified immediately after positioning (G00) or linear interpolation (G01). Cutter compensation cannot be started with circular interpolation (G02 or G03). Also, compensation on the PNC-300G is performed only for the XY plane, and so G17 (setting of the XY plane) is specified immediately after G00 or G01.

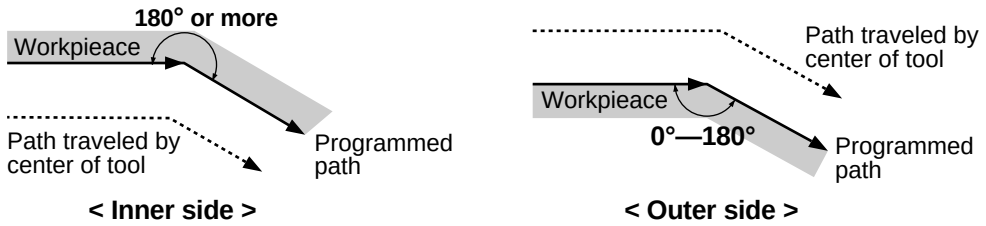
As shown in the figure below (on the left-hand side), the tool is shifted to the left or the right by the amount of offset as it moves forward from the starting point. If no movement command is specified in the block when cutter compensation is started, operation proceeds as shown in the dotted box.




Now let's take a look at tool movement when cutter compensation is started in actual use.

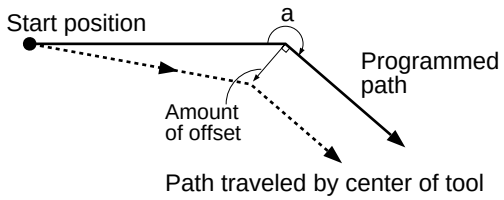
As the following figures show, the shift from the start of offset to the next operation can be classified as travel on the inner side of the program path, travel on the outer side as an obtuse angle, and travel on the outer side as an acute angle. Outer-side travel includes “Type A” and “Type B” paths. The settings for Type A or Type B are made using the PNC-300G's display. (Refer to the “User's Manual.”)

### Definitions of “Inner Side” and “Outer Side”

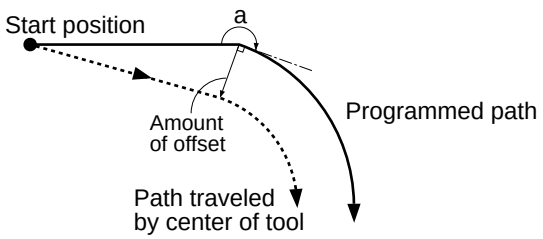



**Inner Side** **(180° ≦ a)**

**From a line to a line**

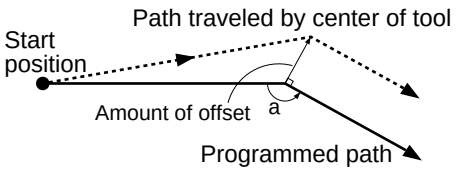



**From a line to an arc**

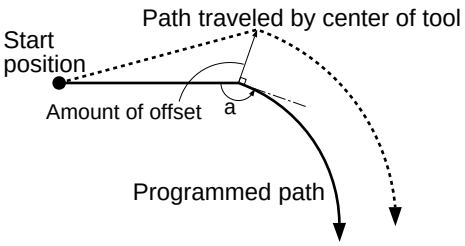


## Outer-side Obtuse Angle (90° ≦ a < 180°)

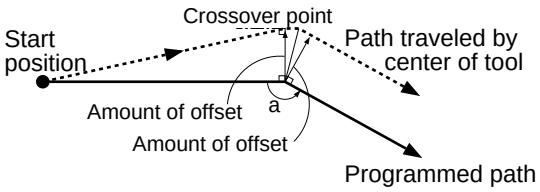
**From a line to a line -- Type A**



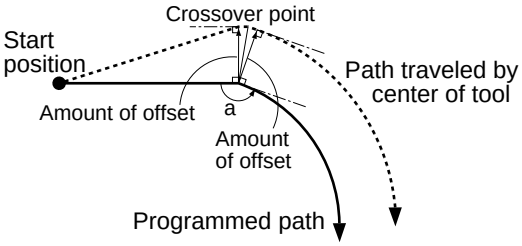

**From a line to an arc -- Type A**




**From a line to a line -- Type B**

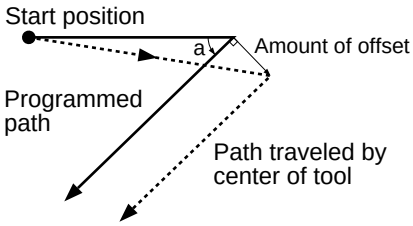



**From a line to an arc -- Type B**

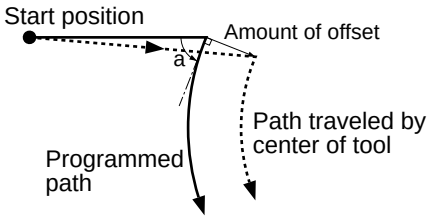



## Outer-side Acute Angle (a < 90°)

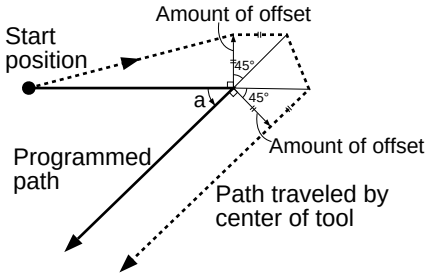
**From a line to a line -- Type A**



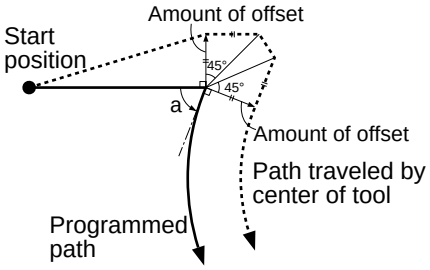

**From a line to an arc -- Type A**



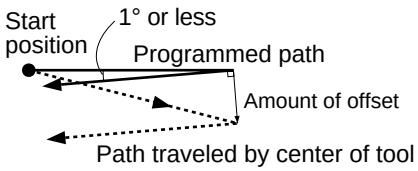

**From a line to a line -- Type B**




**From a line to an arc -- Type B**




**Exceptions: Acute angles of 1° or less**

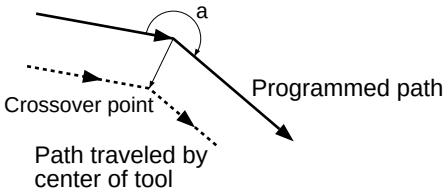


## Operation at Crossover Points During Cutter Compensation

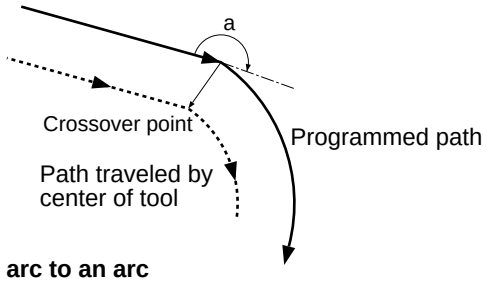
During offset, the tool moves at a position that is always shifted away from the program path by a distance equal to the amount of offset. The figures below show the operation that takes place at a crossover point for a line and another line, a crossover point for a curve and another curve, and a crossover point for a line and a curve.

**Inner Side** **(180° ≦ a)**

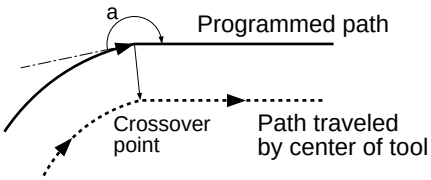
**From a line to a line**



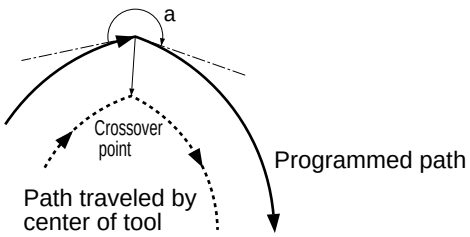

**From a line to an arc**



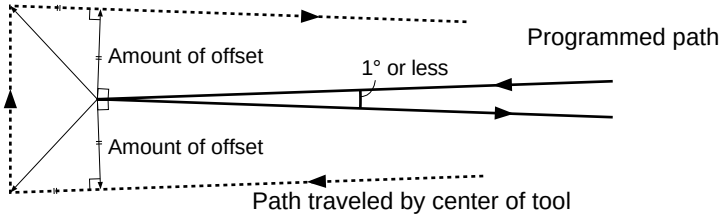

**From an arc to a line**




**From an arc to an arc**

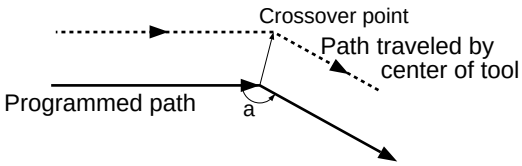



**Exceptions: Inner-side passage of 1° or less (obtuse angle of 359° or more and less than 360°)**

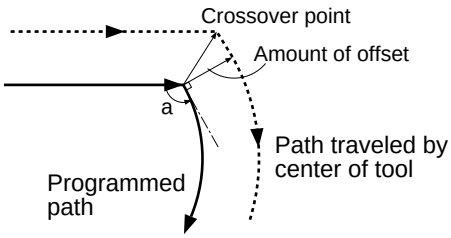



**Outer-side Obtuse Angle** **(90° ≦ a < 180°)**

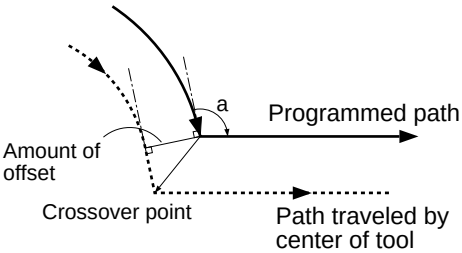
**From a line to a line**



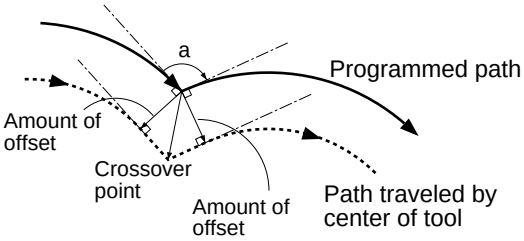

**From a line to an arc**




**From an arc to a line**

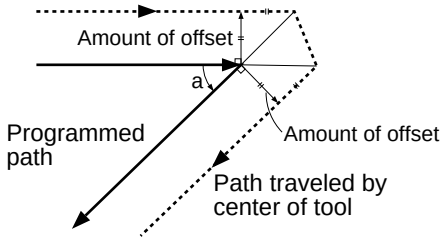



**From an arc to an arc**

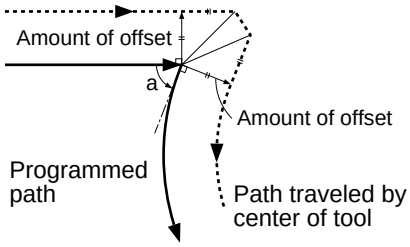


## Outer-side Acute Angle (a < 90°)

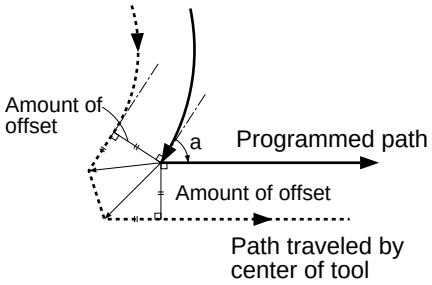
**From a line to a line**



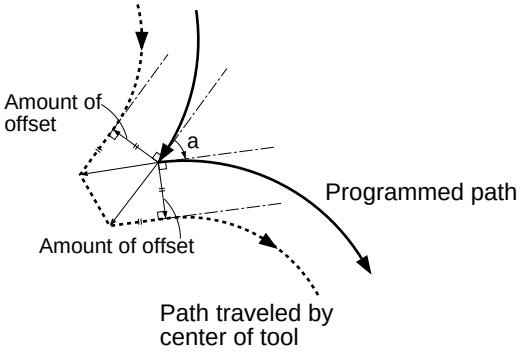

**From a line to an arc**




**From an arc to a line**



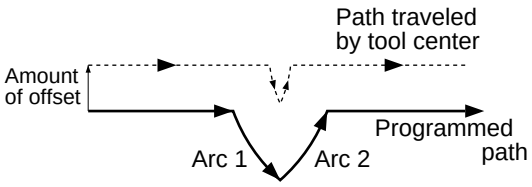

**From an arc to an arc**



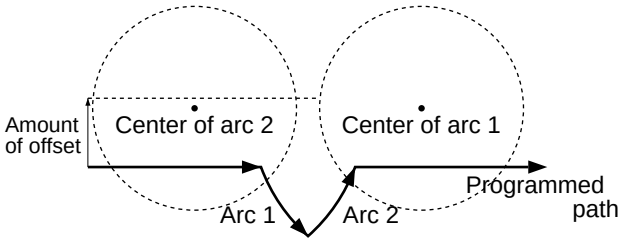

A case such as the following is an exception. In the figure at left, a crossover point exists on the path traveled by the tool center, and the tool path is created normally. When the amount of offset becomes larger, however, no crossover point exists on the tool-center path, as shown in the figure at right, and an error occurs.

## Exception

**No crossover point on the tool path**




* In this case, a crossover point exists on the tool-center path for arcs 1 and 2.

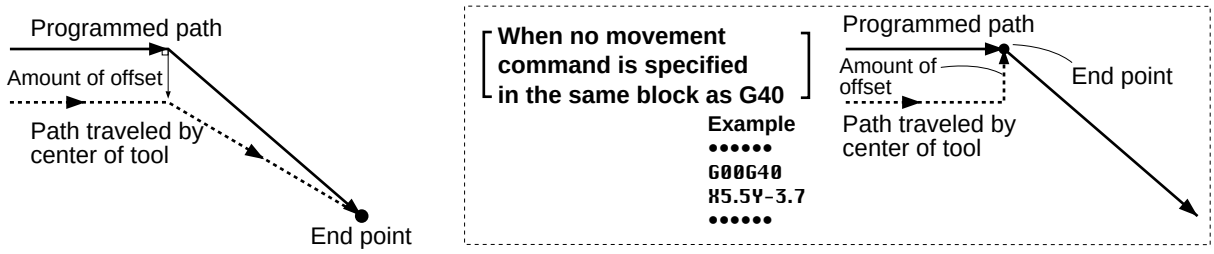



* When the amount of offset becomes larger, the crossover point disappears from the tool-center path for arcs 1 and 2.

## Ending Cutter Compensation

Cutter compensation is ended with G40. A positioning (G00) specification is followed by G40. Cutter compensation cannot be ended by circular interpolation (G02 or G03).

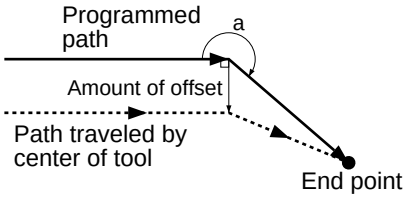
As shown in the figure below (on the left-hand side), the tool is shifted to the left or the right by the amount of offset as it returns to the end point. If no movement command is specified in the block when cutter compensation is ended, operation proceeds as shown in the dotted box.



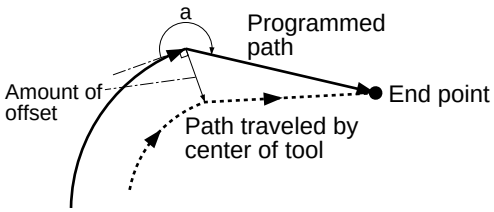

In the same way as when starting cutter compensation, outer-side travel includes Type A and Type B paths.

**Inner Side** **(180° ≦ a)**

**From a line to a line**

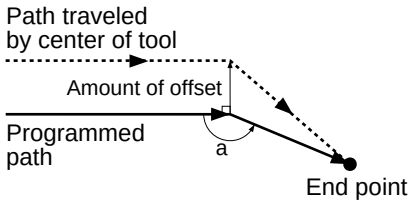



**From an arc to a line**

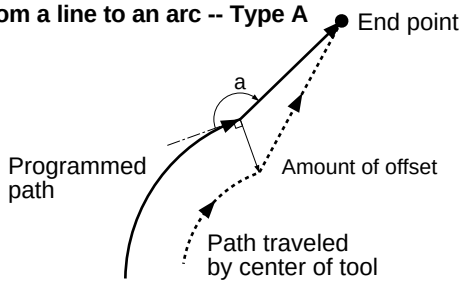



**Outer-side Obtuse Angle** **(90° ≦ a < 180°)**

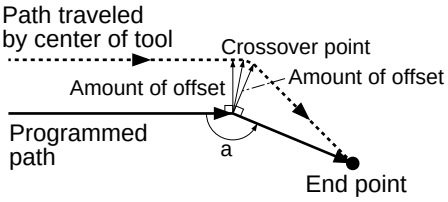
**From a line to a line -- Type A**



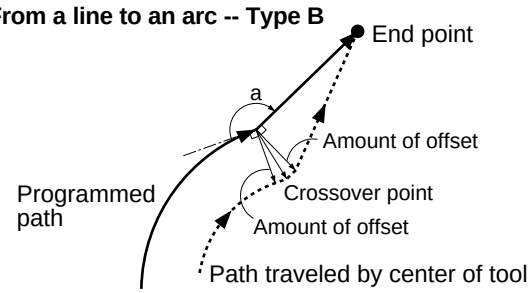

**From a line to an arc -- Type A**




**From a line to a line -- Type B**

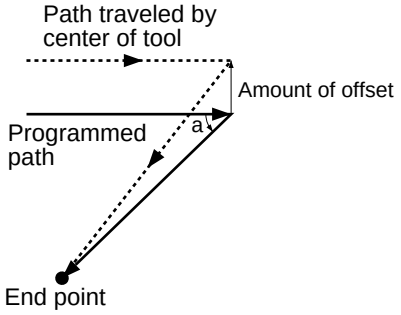



**From a line to an arc -- Type B**

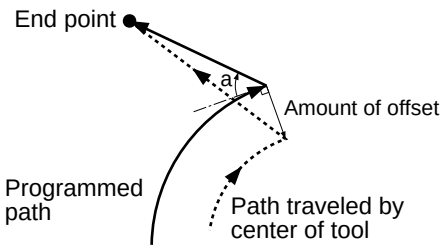


## Outer-side Acute Angle (a < 90°)

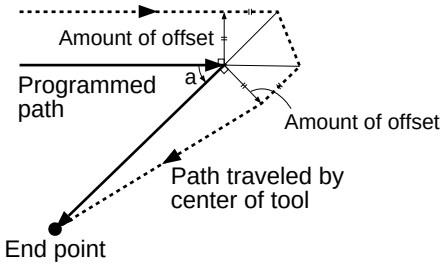
**From a line to a line -- Type A**



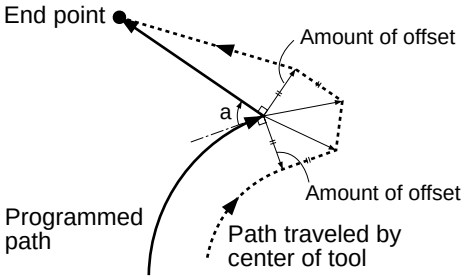

**From a line to an arc -- Type A**



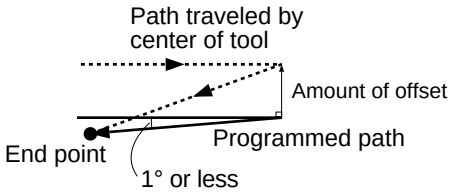

**From a line to a line -- Type B**




**From a line to an arc -- Type B**




**Exceptions: Acute angles of 1° or less**



# G43, G44 and G49 Tool-length Compensation

## Format

G17 { G00 / G01 }{ G43 / G44 }[Z *z* ] H *number*

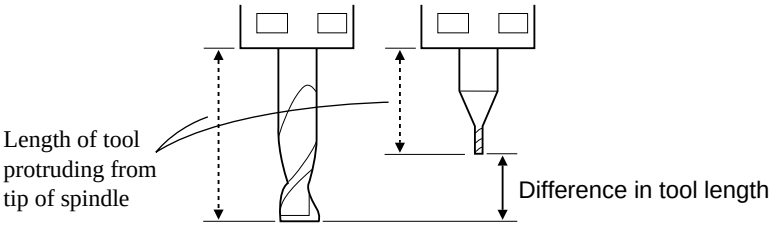
G49

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *z* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *number* | Offset number | 0—10 | 0—10 |

## Description

Tool-length compensation offsets the difference between the tool length assumed by the program and the actual tool length. When compensation for tool length is performed using program coordinate values, the entire program must be changed. Using the tool-length compensation function makes it possible to absorb differences in tool length by changing only the amount of compensation (i.e., the amount of offset).

With NC machines that can change tools automatically, tool-length compensation can also be used to offset the difference in tool length before and after the tool-change. Because such NC machines can determine in advance the length of the tool protruding from the tip of the spindle, compensation of tool length by programming is simple. The PNC-300G has no function for automatic tool-changing, so it cannot determine the in advance the length of the tool protruding from the tip of the spindle.




The words for tool-length compensation are “G43,” “G44,” and “G49.”

- G43: Positive tool-length compensation (addition of amount of offset in Z-axis direction)
- G44: Negative tool-length compensation (subtraction of amount of offset in Z-axis direction)
- G49: Cancel tool-length compensation

### Restrictions on Tool-length Compensation

Tool-length compensation is subject to the following restrictions.

1. Tool-length compensation can be performed only in the XY plane.
2. When fixed-cycle operation has been specified, executing or canceling tool-length compensation causes an error to be generated.
3. Executing M06 (tool change) during tool-length compensation causes an error to be generated.
4. The offset number cannot be changed during tool-length compensation. To change the offset number, first cancel tool-length compensation. Then execute G43 or G44 again, and change the offset number.

## Setting the Amount of Offset

The PNC-300G allows amounts of offset to be set individually for offset numbers 1 to 10. An amount of offset can be set using either of two methods.

**1.** Using the display on the PNC-300G

The PNC-300G's LCD screen and control keys are used to set the amount of offset. See the “User's Manual” for a description of the procedure.

**2.** Using code (G10)

**G10[P** *number***][R** *offset***]**

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *number* | Offset number | Range 2 | 1—10 |
| *offset* | Offset value | -300—300 [mm]<br>or -11.81"—11.81" | -300—300 [mm]<br>or -11.81"—11.81" |

* If G43 or G44 is used to specify an offset number for which no amount of offset has been set with G10, the value that has been set on the PNC-300G is used.

* Setting a negative value for the amount of offset results in operation as described below.

For example, setting an amount of offset of -24 mm for offset number 1 has the following results.

**G00G43H01Z100.0** -24 mm added (i.e., 24 mm subtracted) in the Z-axis direction

**G00G44H01Z100.0** -24 mm subtracted (i.e., 24 mm added) in the Z-axis direction

* An amount of offset of zero is set for offset number 0. The amount of offset for offset number 0 cannot be changed.

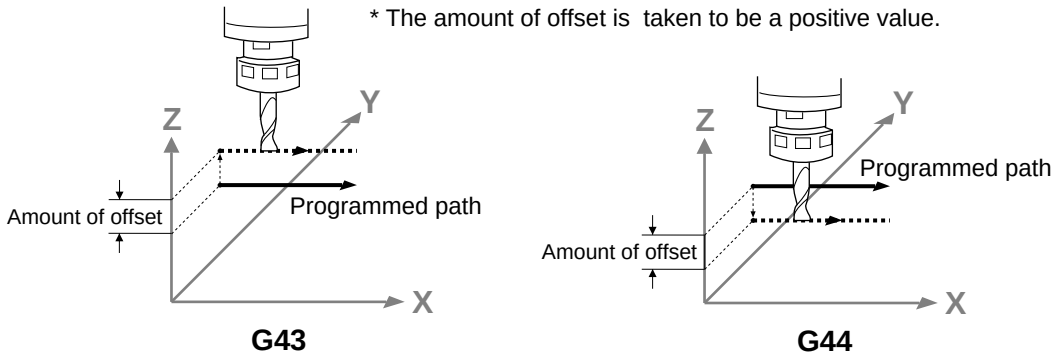
* The amounts of offset set for offset numbers 1 to 10 are used by both cutter compensation and tool-length compensation. For example, if an amount of offset of 10 mm has been set for offset number 2, operation is as follows.

Cutter compensation : **G00G41D02X100.0** 10 mm offset to the left-hand side relative to the direction of forward movement

Tool-length compensation : **G00G43H02Z7.0** 10 mm offset in the positive direction of the Z axis

## Starting Tool-length Compensation

Tool-length compensation is started with G43 or G44. G43 performs offset in the positive direction of the Z axis, and G44 performs offset in the negative direction of the Z axis. The direction of offset cannot be changed while tool-length compensation is in progress.

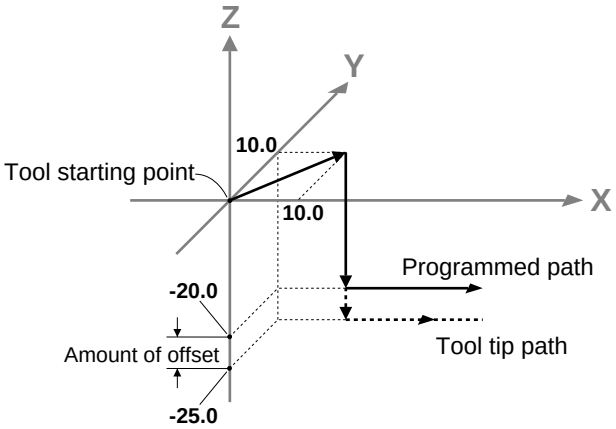


A specification for positioning (G00) or linear interpolation (G01) is followed by G43 or G44. Because tool-length compensation on the PNC-300G can be performed only in the XY plane, G17 (XY plane setting) is specified immediately before the G00 or G01 specification.

As the following figure shows, the amount of offset is applied after the amount of movement along the Z axis following the G43 or G44 specification.

```
••••••••••
G00X0Y0Z0
G01X10.0Y10.0
G17G01G44Z-20.0H01
X20.0
••••••••••
••••••••••
G49
••••••••••
••••••••••
```

H01: 5 mm
Millimeter input (G21)




## Ending Tool-length Compensation

Tool-length compensation is ended with G40 or H00. A positioning (G00) specification is followed by G40 or H00. When tool-length compensation is ended, the tool returns by a distance equal to the amount of offset.

# G50 and G51 Scaling

## Format

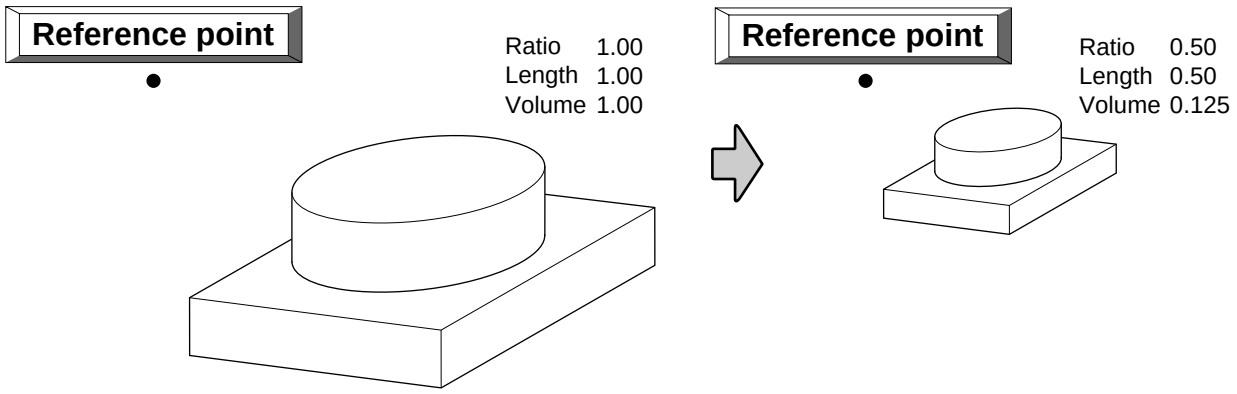
**G50**
**G51[X** *x* **][Y** *y* **][Z** *z* **][P** *scale* **]**

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |
| *z* | Coordinate or movement distance (Z axis) | Range 1 | Maximum cutting range |
| *scale* | Scaling ratio | Range 2 | 0.00001—999.999 |

## Description

G51 executes equal enlargement or reduction for each axis, referenced to the specified point. It is used for such application as the creation of reduced-scale models. Because this instruction affects the entire program, G50 is normally specified immediately after the start of the program.
G50 cancels G51.
When enlargement or reduction has been specified with G51, it remains in effect until canceled with G50 or until another program is executed.
The reference point for enlargement or reduction is specified with the addresses X, Y, and Z. When not specified, the current tool position is used as the reference point.
*scale* is a numerical value specifying the ratio. Its effective range is a ratio of 0.00001 to a ratio of 999.999. A specified ratio less than 0.00001 is treated as a ratio of 0.00001, and a specified ratio larger than 999.999 is similarly taken to be a ratio of 999.999. When P *scale* is not specified, the settings made on the PNC-300G are used.

As an example, specifying a ratio of 0.5 produces the results shown below. If the length is a ratio of 0.5, the volume ratio becomes 0.125.



# G54, G55, G56, G57, G58 and G59

# Selects Coordinate System

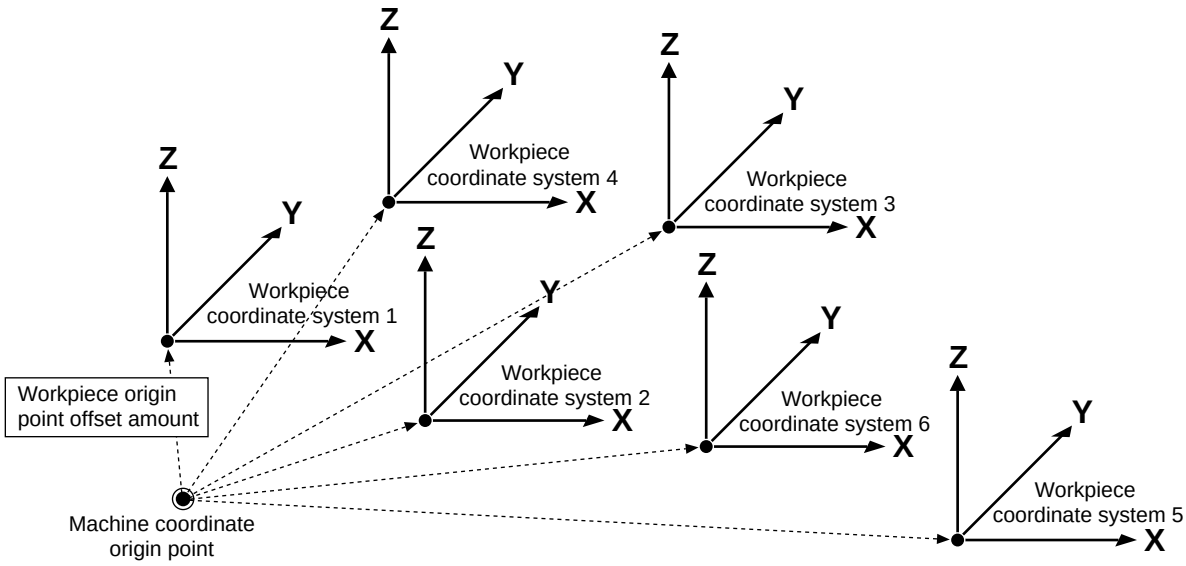
## Format

```
G54
G55
G56
G57
G58
G59
```

## Description

Up to six workpiece coordinate systems can be set, and any of the set coordinate systems can be selected by programming.

- **G54**: Selects workpiece coordinate system 1
- **G55**: Selects workpiece coordinate system 2
- **G56**: Selects workpiece coordinate system 3
- **G57**: Selects workpiece coordinate system 4
- **G58**: Selects workpiece coordinate system 5
- **G59**: Selects workpiece coordinate system 6




G54 through G59 are used to select workpiece coordinate systems which have been set in advance. Workpiece coordinate systems 1 through 6 are set using the display on the PNC-300G. (Refer to the “User's Manual” for an explanation of how to make the setting.) Coordinate systems are described on page 7.

# G80, G81, G82, G85, G86 and G89

# Fixed Cycle (Canned Cycle)

## Format

```
G80
G98G81[X x ][Y y ][Z z ][R r ][K times ]
G99G81[X x ][Y y ][Z z ][R r ][K times ]
G98G82[X x ][Y y ][Z z ][R r ][P time ][K times ]
G99G82[X x ][Y y ][Z z ][R r ][P time ][K times ]
G98G85[X x ][Y y ][Z z ][R r ][K times ]
G99G85[X x ][Y y ][Z z ][R r ][K times ]
G98G86[X x ][Y y ][Z z ][R r ][K times ]
G99G86[X x ][Y y ][Z z ][R r ][K times ]
G98G89[X x ][Y y ][Z z ][R r ][P time ][K times ]
G99G89[X x ][Y y ][Z z ][R r ][P time ][K times ]
```

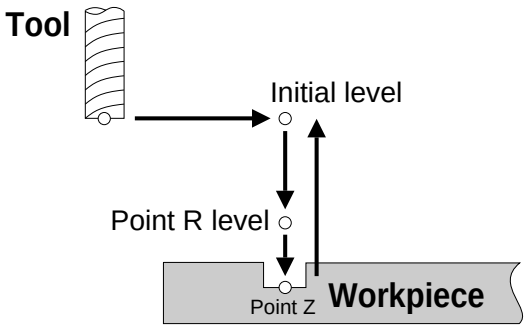
| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Coordinate or movement distance (X axis) | Range 1 | Maximum cutting range |
| *y* | Coordinate or movement distance (Y axis) | Range 1 | Maximum cutting range |
| *z* | Coordinate or movement distance (Z axis) | Range 1 | Maximum cutting range |
| *r* | Point R level (Z axis) | Range 1 | Maximum cutting range |
| *time* | Dwell time | Range 2 | — |
| *times* | Number of repetitions | Range 1 | 0—9999 |

## Description

A fixed (or canned) cycle is a command that executes a series of predetermined operations for cutting, such as for drilling a hole. This simplifies programming, because cutting operations spanning several blocks can be executed in a single block. The amount of data is also reduced.

G81, G82, G85, and G89 are fixed cycles for drilling. The functions of each of these words vary in terms of the feed rates between the specified points, and in the presence or absence of a dwell interval. G80 cancels a fixed cycle.

G98 and G99 specify the tool position (along the Z axis) after the completion of the fixed cycle. G98 specifies a return to the initial level, whereas G99 specifies return to the point R level. The initial level is the Z-axis tool position in effect before the fixed cycle was specified. The point R level is set between the Z-axis position on the surface of the workpiece and the initial level. Point R is specified in order to increase the tool movement distance at maximum speed and reduced the cutting time.

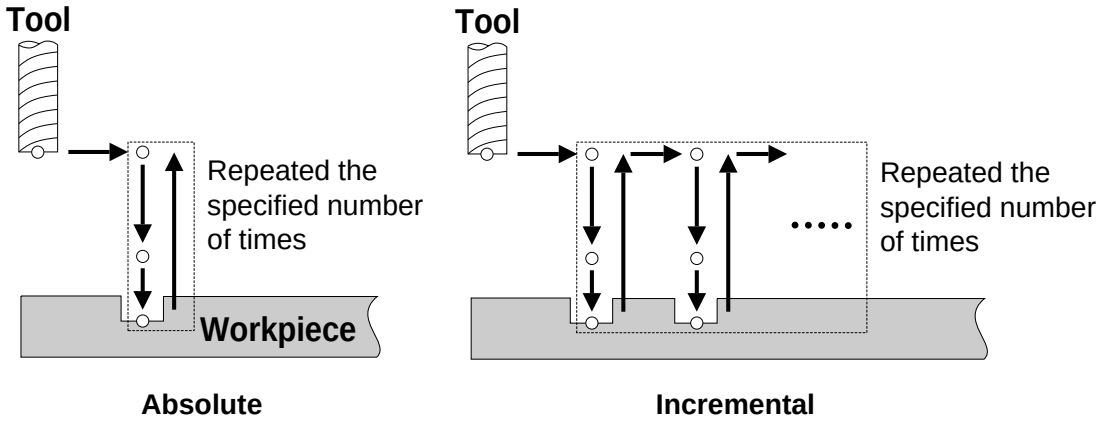



Cutting is performed at the spindle speed and feed rate that have been specified. Refer to page 56 for an explanation of the feed rate and to page 54 for a description of spindle speed.

**X** *x* and **Y** *y* move the tool to the starting point. When not specified, drilling is carried out at the current tool position.
**Z** *z* specifies the location of the bottom of the hole (along the Z axis). When not specified, no drilling is performed.
**R** *r* specifies the point R level. This specifies the Z coordinate of point R for absolute programming and the distance from the initial level along the Z axis for incremental programming. When not specified, the same point as at the initial level is used.
**P** *time* is specified for fixed cycles that include dwell (G82 and G89). A numerical value for the time interval is specified after P. The specified time is in seconds when a real number is used, and in milliseconds when an integer is used.

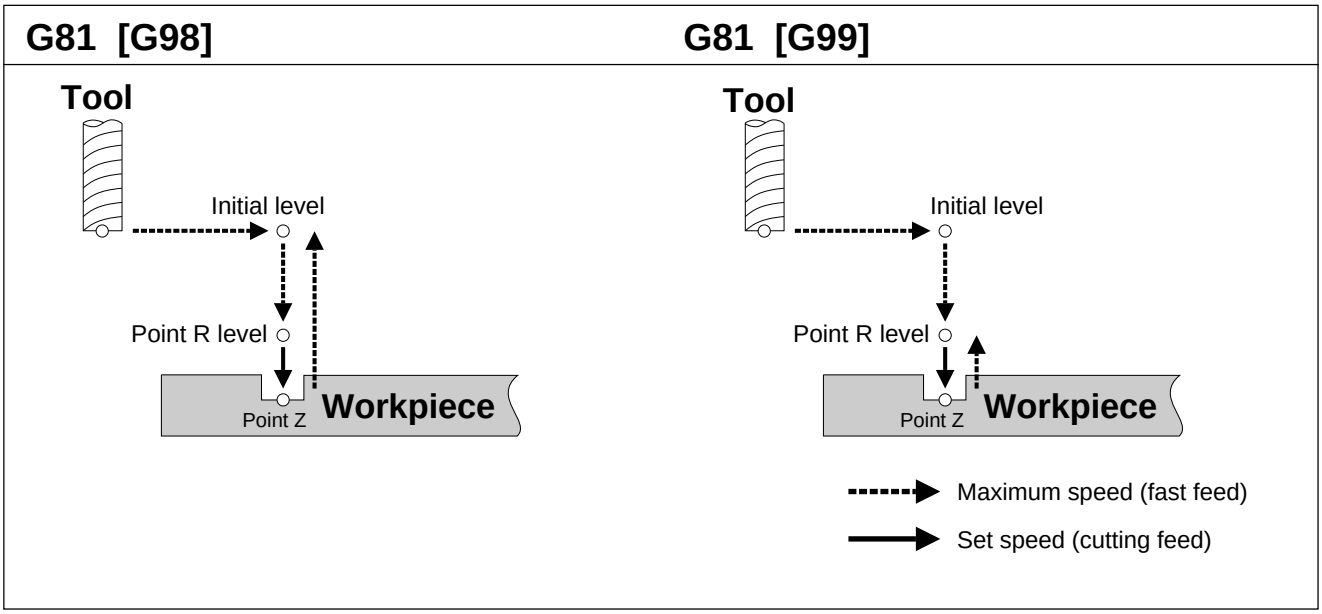
| | |
|---|---|
| **P1.0** | 1-second dwell (units in seconds) |
| **P1000** | 1-second dwell (units in milliseconds) |

When not specified, there is no dwell.
**K** *times* specifies the number of repetitions. If programming is absolute, drilling is carried out as many times as specified at the same position. In incremental programming, drilling is carried out as many times as specified at equidistant points, as shown in the figure below. If **K** *times* is not specified, drilling is performed only once. The effective range is from 0 (no drilling) to 9,999 times. The operation is executed zero times if a number less than 0 (i.e., a negative number) is specified, and is executed 9,999 times if a number larger than 9,999 is specified.

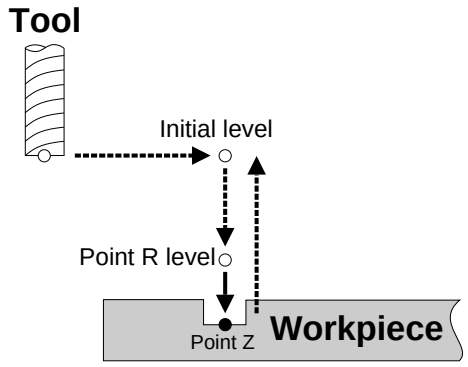



None of the fixed cycles includes the function for starting the spindle motor. If the spindle motor is not already turning, the M03 word should be given beforehand to start it. Executing a fixed cycle while the motor is not turning causes an error to be generated.

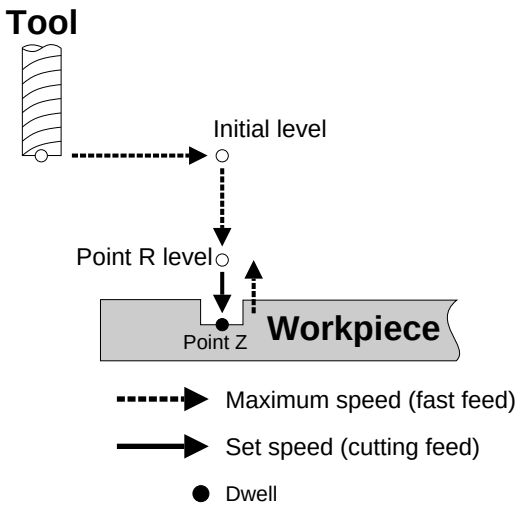
The following figures illustrate the specifications for each of the fixed cycles.



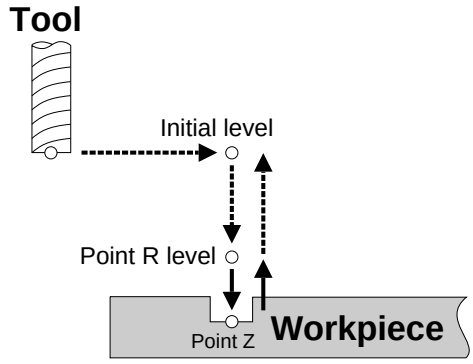
## G82 [G98]



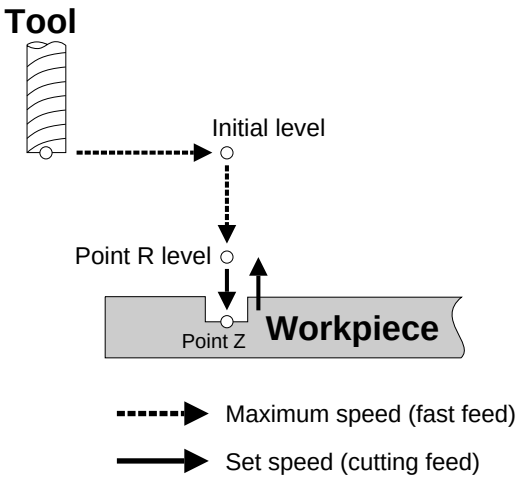

## G82 [G99]



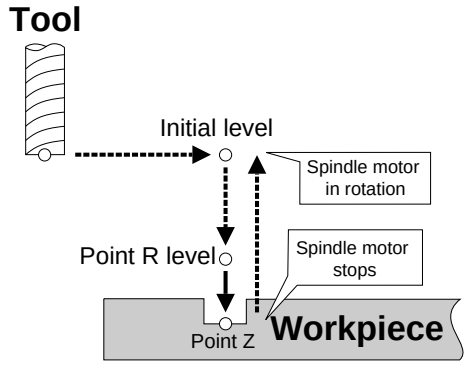

## G85 [G98]



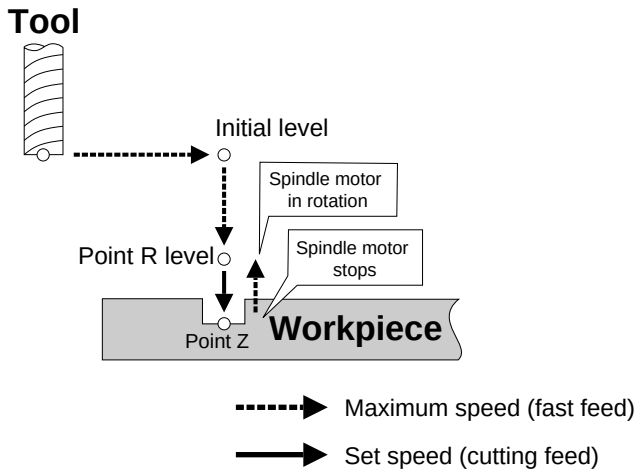

## G85 [G99]

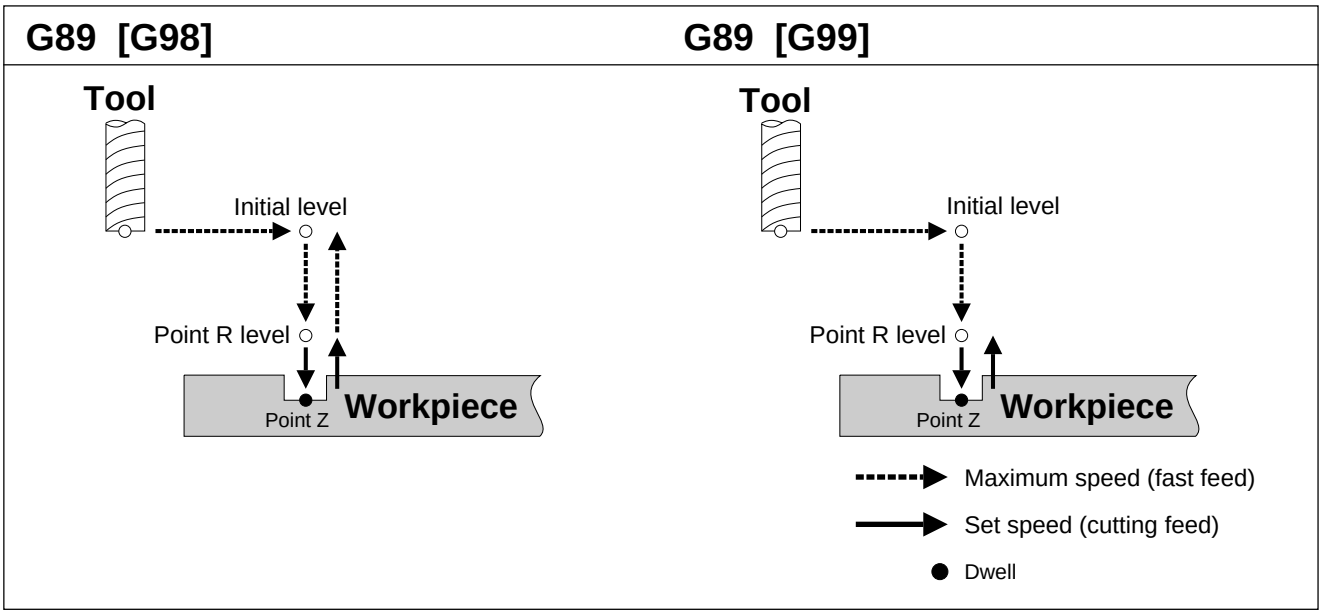



## G86 [G98]




## G86 [G99]




G89 [G98]
G89 [G99]
Tool
Initial level
Point R level
Point Z
Workpiece
Tool
Initial level
Point R level
Point Z
Workpiece
Maximum speed (fast feed)
Set speed (cutting feed)
Dwell

# G90 and G91 — Absolute and Incremental

## Format

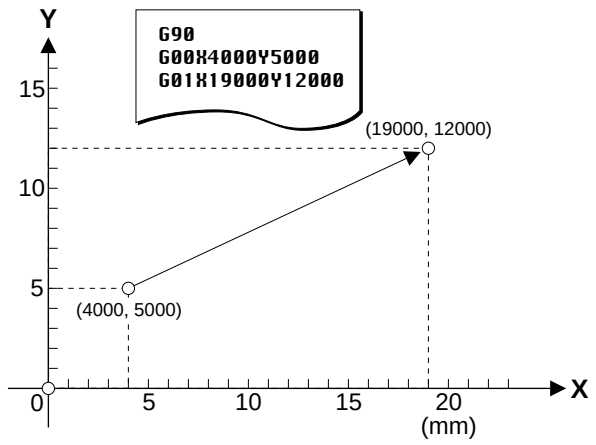
```
G90
G91
```

## Description

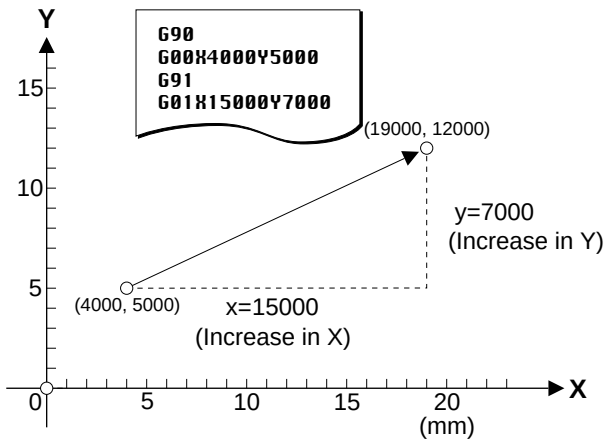
There are two types of coordinate specifications: absolute and incremental.

The figure below shows the difference between absolute and incremental specifications on an X-Y plane. Absolute specifications indicate the position as the distance from the workpiece coordinate origin, whereas incremental specifications indicate the amount of movement from the current position.

Programming that specifies absolute coordinates is called “absolute programming,” and programming which specifies incremental coordinates is termed “incremental programming.”




**Absolute**




**Incremental**

The settings for G90 or G91 made on the PNC-300G remain in effect unless changed by programming.

There are no special rules for deciding when to use an absolute or incremental program. Examine the drawing and choose the one which makes for the simplest program.

# G92 Coordinate System

## Format

G92[X *x* ][Y *y* ][Z *z* ]

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *x* | Workpiece coordinate (X axis) | Range 1 | Maximum cutting range |
| *y* | Workpiece coordinate (Y axis) | Range 1 | Maximum cutting range |
| *z* | Workpiece coordinate (Z axis) | Range 1 | Maximum cutting range |

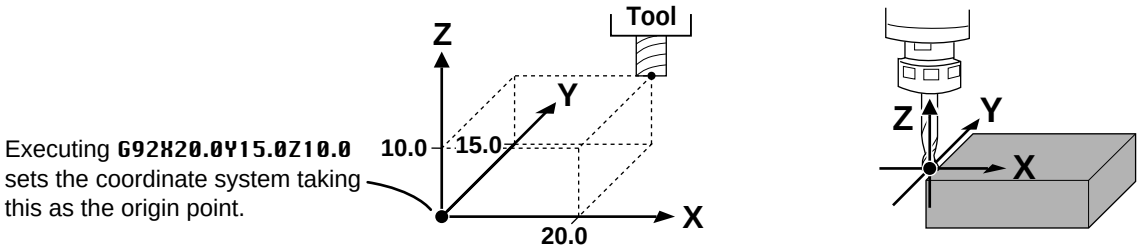
## Description

This sets the present position of the tool to the specified workpiece coordinate.

This word is used to operate the machine to move the tool to a certain point on the workpiece and set the workpiece coordinate for that point. This changes the origin point for the workpiece coordinate system. Refer to page 7 for an explanation of workpiece coordinate systems.

G92 is a code effective only within the block. For this reason, only coordinate values specified in the same block as G92 are interpreted as the set workpiece coordinates.

In general, the workpiece coordinate origin is not changed during the course of program execution. Consequently, this word is used at the start of a program.



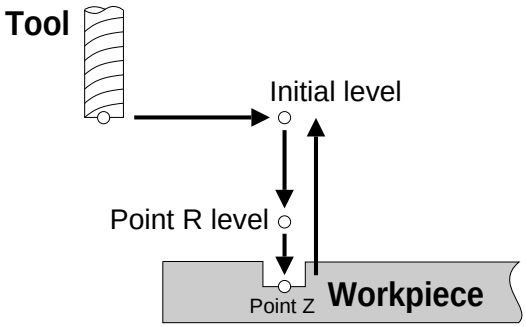
# G98 Initial Level Return

## Format

```
G98
```

## Description

This specifies the tool position (along the Z axis) after the completion of a fixed cycle. G98 specifies a return to the initial level. The initial level is the Z-axis tool position in effect before the fixed cycle was specified. See page 43 for an explanation of fixed cycles.



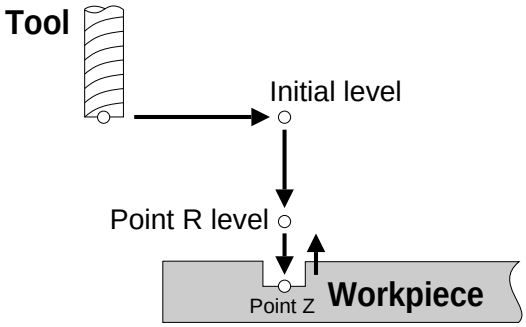

# G99 Point R Level Return

## Format

```
G99
```

## Description

This specifies the tool position (along the Z axis) after the completion of a fixed cycle. G99 specifies return to the point R level. The point R level is set between the Z-axis position on the surface of the workpiece and the initial level. Point R is specified in order to increase the tool movement distance at maximum speed and reduced the cutting time. Refer to page 43 for an explanation of fixed cycles.



# Miscellaneous Functions (M Functions)

## M00 Program Stop

### Format

```
M00
```

### Description

After the operations specified within the block have been completed, the spindle motor stops. The state of the spindle motor (rotating or stopped) does not change.

## M01 Optional Stop

### Format

```
M01
```

### Description

This is active when "**OPTIONAL STOP**" on the PNC-300G has been set to "**ON**." In the same way as for M00, a stop takes place after the operations specified within the block have been completed. The state of the spindle motor (rotating or stopped) does not change.

## M02 End of Program

### Format

```
M02
```

### Description

This indicates that the main program has ended.

## M03 and M05 Spindle Motor Start/Stop

### Format

```
M03
M05
```

### Description

M03 instructs the machine to start the spindle motor, and M05 instructs the machine to stop it.
M03 is the only instruction that is available to start the spindle motor. Cutting instructions such as G01, G02, and G03 do not include the function for starting the spindle motor, so M03 must be given to start the motor before any cutting instruction is input to the machine.
If the motor is already turning, M03 is ignored and the motor continues to turn. Similarly, M05 is ignored if the motor is already stopped, and the motor remains stopped.

# M06 Tool Change

## Format

```
M06
```

## Description

Execution is carried out up to the word just before M06, and operation stops immediately when M06 is executed. If the spindle motor is rotating, the rotation stops. M06 is active when "**TOOL CHANGE**" on the PNC-300G has been set to "**PAUSE**."

```
Tool Change, Pause
```

Display when MO6 is executed

Canceling the paused state with the **[CANCEL]** key returns the spindle motor and coordinate values to their state before stopping. Execution continues from the code which appears after M06.

# M30 End of Program

## Format

```
M30
```

## Description

This instructs the main program to end.

# M98 Subprogram Call

## Format

**M98[P** *times number***]**

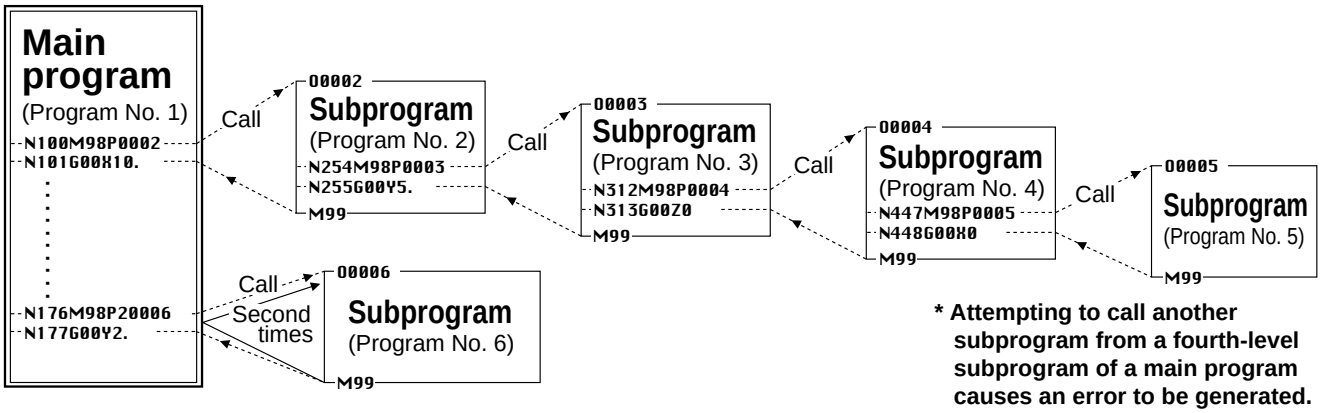
| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *times* | Number of calls | Range 2 | 1—9999 |
| *number* | Program number | Range 2 | 1—9999 |

## Description

The subprogram of the specified number is called up and executed. A subprogram call can be made not only from a main program, but from another subprogram as well. However, attempting to call another subprogram from a fourth-level subprogram of a main program causes an error to be generated.

The *times* parameter indicates the number of calls. The subprogram is called and executed the number of times specified by this parameter. When *times* is not specified, the subprogram is called once.

The *number* parameter indicates the program number of the subprogram. A four-digit number must be specified. For example, "0002" is used to specify Program number 2. If a program of the specified number does not exist, an error is generated.




*** Attempting to call another subprogram from a fourth-level subprogram of a main program causes an error to be generated.**

# M99 End of Subprogram

## Format

M99[P *times number*]

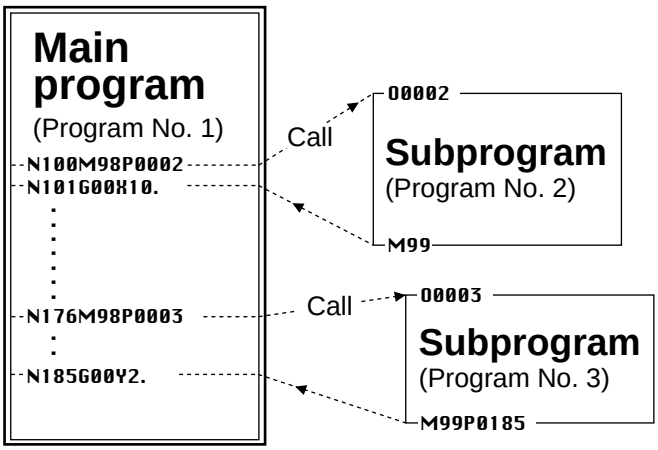
| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *number* | Program number or sequence number | Range 1 | 1—9999 |

## Description

This indicates the end of a subprogram. M99 is normally specified alone, with no *number* parameter, and execution returns to the code after the call source (M98) at that time.

The *number* parameter specifically designates a program number or sequence number as the destination for returning. The entire program is searched from its beginning, and execution returns to the first program number or sequence number. The PNC-300G does not operate while the number search is in progress. The search process may take some time when execution the return destination is a number near the end of a lengthy program.

An error is generated if the specified number does not exist.




When M99 is specified at the end of a main program, execution of the main program is repeated.

# Spindle Speed Function (S Function)

This specifies the speed of the spindle motor.
The S function does not include a function for starting the spindle motor. It is effective only when the spindle has been started with M03 or is otherwise already turning.

## Format

**S** ***revolution speed***

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *revolution speed* | Spindle speed | -65535—65535 | Specifying the speed (in rpm) : 3000—8000<br>Numerical code specification : 70—78 |

## Description

When the spindle motor has already been started, the setting made on the PNC-300G determines whether operation is performed simultaneously with specification or after completion of the block in which the specification is made. If specified when the spindle motor is stopped, the speed specified by the S function is enabled when M03 is given.
A numerical value for the speed is specified following S. The rotation speed may be specified as rpm or through the speed code specification. *The number input by the operator will be interpreted by the system depending on the setting in the PNC-300G display* (refer to the User's Manual for information on setting this).

### Specifying the speed (in rpm)

This method specifies the speed in units of rpm (revolutions per minute).
If the specified speed exceeds the maximum speed, the maximum speed is set. Similarly, the minimum speed is set if the specified speed is less than the minimum speed.

### Numerical code specification

With this method, speeds are pre-assigned to numerical codes from 01 to 99, and these numerical codes are specified to set the desired speed.
If the specified speed exceeds the maximum speed, the maximum speed is set. Similarly, the minimum speed is set if the specified speed is less than the minimum speed.
If a value of 0 (zero) is specified, the minimum speed is set.
If a real number is specified, any value to the right of the decimal point is truncated.
The speeds assigned to the numerical codes are shown in the table on the following page.

| Code | Spindle speed (rpm) | Code | Spindle speed (rpm) | Code | Spindle speed (rpm) | Code | Spindle speed (rpm) | Code | Spindle speed (rpm) |
|---|---|---|---|---|---|---|---|---|---|
| 00 | * (1) | 20 | 10.0 | 40 | 100 | 60 | 1000 | 80 | 10000 |
| 01 | 1.12 | 21 | 11.2 | 41 | 112 | 61 | 1120 | 81 | 11200 |
| 02 | 1.25 | 22 | 12.5 | 42 | 125 | 62 | 1250 | 82 | 12500 |
| 03 | 1.40 | 23 | 14.0 | 43 | 140 | 63 | 1400 | 83 | 14000 |
| 04 | 1.60 | 24 | 16.0 | 44 | 160 | 64 | 1600 | 84 | 16000 |
| 05 | 1.80 | 25 | 18.0 | 45 | 180 | 65 | 1800 | 85 | 18000 |
| 06 | 2.00 | 26 | 20.0 | 46 | 200 | 66 | 2000 | 86 | 20000 |
| 07 | 2.24 | 27 | 22.4 | 47 | 224 | 67 | 2240 | 87 | 22400 |
| 08 | 2.50 | 28 | 25.0 | 48 | 250 | 68 | 2500 | 88 | 25000 |
| 09 | 2.80 | 29 | 28.0 | 49 | 280 | 69 | 2800 | 89 | 28000 |
| 10 | 3.15 | 30 | 31.5 | 50 | 315 | **70** | **3150** | 90 | 31500 |
| 11 | 3.55 | 31 | 35.5 | 51 | 355 | **71** | **3550** | 91 | 35500 |
| 12 | 4.00 | 32 | 40.0 | 52 | 400 | **72** | **4000** | 92 | 40000 |
| 13 | 4.50 | 33 | 45.0 | 53 | 450 | **73** | **4500** | 93 | 45000 |
| 14 | 5.00 | 34 | 50.0 | 54 | 500 | **74** | **5000** | 94 | 50000 |
| 15 | 5.60 | 35 | 56.0 | 55 | 560 | **75** | **5600** | 95 | 56000 |
| 16 | 6.30 | 36 | 63.0 | 56 | 630 | **76** | **6300** | 96 | 63000 |
| 17 | 7.10 | 37 | 71.0 | 57 | 710 | **77** | **7100** | 97 | 71000 |
| 18 | 8.00 | 38 | 80.0 | 58 | 800 | **78** | **8000** | 98 | 80000 |
| 19 | 9.00 | 39 | 90.0 | 59 | 900 | 79 | 9000 | 99 | * (2) |

Effective range of operation for PNC-300G

* (1) : Minimum spindle motor speed

* (2) : Maximum spindle motor speed

# Feed Function (F Function)

This determines the feed rate for the workpiece and the spindle.
The feed rate generally varies according to the cutting parameters (such as the spindle speed, tool diameter, and workpiece material).

## Format

**F** *feed rate*

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *feed rate* | Feed rate | Range 2 | 0—1800 [mm/min.]<br>0—70.87 [inch/min.] |

## Description

The feed rate is specified as a real or integer value following the "F."
The setting made on the PNC-300G determines whether the F function is performed simultaneously with specification or after completion of the block in which the specification is made.

For millimeter input

**F100.0** Feed rate set at 100 mm/min.

For inch input

**F100000** Feed rate set at 100 mm/min. (in 1/1000 mm/min. units)

If a speed exceeding the maximum speed is specified, the maximum speed is set. In the same way, the minimum speed is set if the specified speed is less than the minimum speed.

The feed rate which is actually used is determined in a stepwise manner on the PNC-300G. The actual feed rate is either 0.5 mm/sec., or from 1 to 30 mm/sec. (in steps of 1 mm/sec.). The equivalent per-minute speeds are 30 mm/min. and from 60 to 1,800 mm/min. (in steps of 60 mm/min.).
When the *feed rate* parameter is set to 0, operation is at a speed of 30 mm/min.
The specified value is rounded down to the nearest multiple of 60.

Examples:

| Command | Operating speed (mm/min.) |
|---|---|
| **F70** | 60 |
| **F119** | 60 |
| **F120** | 120 |

Operation is similar for inch input, except that the millimeter values are converted to inches.

# Other Functions

## N — Sequence Number

### Format

**N** *number*

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *number* | Sequence number | 1—9999 | 1—9999 |

### Description

A sequence number is an integer number for a block. It is specified at the start of the block.

A sequence number may either be present or absent from any or all blocks. There is also no need for sequence numbers to be consecutive, or to be arranged in order from smaller to larger numbers. However, consecutive sequence numbers are customarily used to mark critical places within a program.

An integer of up to four digits (i.e., from 1 to 9,999) is input for the *number* parameter.

A sequence number can be used as the return destination for a called subprogram. (See the description of M99 on page 53.) However, a sequence number cannot be used to call a subprogram.

## O — Program Number

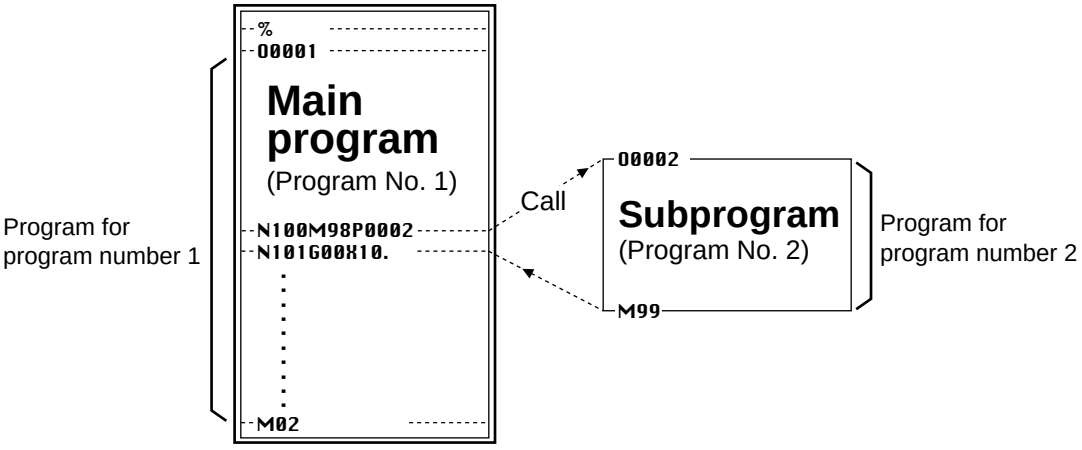
### Format

**O** *number*

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *number* | Program number | 1—9999 | 1—9999 |

### Description

Program numbers are sequential numbers for programs. A program begins with a program number and ends with either M02, M30, or M99. The program number is specified at the start of a program.




The *number* parameter is a program number, and is specified with an integer of up to four digits (i.e., from 1 to 9,999). Normally, a four-digit number is specified. (For example, "**0002**" is specified for program number 2.) Do not specify a program number of 0 (0000).

A program number can be used to call a subprogram, and can also be used as the return destination for a called subprogram. (See the description of M99 on page 53.)

# / Optional Block Skip

## Format

/ *number*

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *number* | Skip number | — | 1—10 |

## Description

This function makes it possible to skip over a desired block within a program. Optional block skip is specified at the start of the block.

```
•••••••••
•••••••••
G01Z-7.0
G01Y35.0
/M98P0002          ←— Subprogram call is skipped (not called)
G03X15.0Y-15.0I15.0
•••••••••
•••••••••
```

The *number* parameter is a skip number, and is input as an integer from 1 to 10. An absent value is interpreted as "1." When entering two or more optional block skips in a single block, however, the numerical values should be specified. Any specified value outside the range of 1 to 10 is ignored.

A skip number can be used to cause only a particular block among blocks which have a "/" to be skipped. The number to be skipped is set on the PNC-300G (see the User's Manual for information on how to make this setting).

```
•••••••••
•••••••••
/2/3G01Z-7.0       ←— Linear interpolation is not performed (number 2 is off, but number 3 is on)
G01Y35.0
/2M98P0002         ←— Subprogram call is performed (number 2 is on)
/G03X15.0Y-15.0I15.0 ←— Circular interpolation is not performed (number 1 is on)
•••••••••
•••••••••
```

* The settings on the PNC-300G are taken to be as follows.
Number 1 is on
Number 2 is off
Number 3 is on

# % or ER — Program Start

## Format

% (ISO or ASCII code)
or
ER (EIA code)

## Description

A block containing only a “%” must appear at the start of the data. No other word should be specified within a block in which “%” or “ER” is specified. This notifies the machine of the start (or end) of the data. Such a block may optionally be present at the end of the data. When it appears at the end of the data, the data is specified automatically. The data start character is “%” if ISO or ASCII is in use as the character code system, of “ER” if EIA is in use.
Be sure to specify this at the beginning of the first program input after powering up the machine. If not specified, the first several blocks of the program are ignored.

# EOB — End of Block

## Description

A program is a series of instructions (written commands) for the machine, expressed as symbols and numbers. These instructions are separated by EOB markers, with the information between two EOB markers forming one instruction. This single instruction between two EOB markers is called a block. Each block, in turn, is composed of words.
The character code systems supported by the machine are ASCII, ISO, and EIA. Each character of ASCII data has a length of 7 bits, whereas ISO and EIA characters are each 8 bits in length. The EOB marker differs according the code system. ASCII uses LF (line feed), ISO uses LF or NL (new line), and EIA uses CR (carriage return).
Refer to page 37 for a table of the character code systems. The character code system is selected from the machine. Refer to the “User's Manual” for a description of the setting procedure.

# ( ) — Comment

## Format

( *message* )

| Parameter | Function | Acceptable range | Effective range |
|---|---|---|---|
| *message* | Comment (text string) | — | — |

## Description

Comments can be included within a program.
A text string appearing between a “(” and “)” is considered to be a comment and is skipped over during program execution. Comments can be useful for noting a program's revision history, describing the content of a program, indicating cautions regarding cutting time, and so on.
There is no restriction on the number of characters which a comment may contain.

```
%
G90
O0001
(,main program start)        ←— This is a comment.
G00Z5.0
•••••••••
•••••••••
```

# APPENDICES

## Words Table

### Preparatory Functions (G Functions)

| | Functions | Group (*) | Effective only within the block in which specified |
|---|---|---|---|
| G 00 | Positioning | a | |
| G 01 | Linear interpolation | | |
| G 02 | Clockwise circular interpolation | | |
| G 03 | Counterclockwise circular interpolation | | |
| G 04 | Dwell | | ○ |
| G 10 | Data setting | | ○ |
| G 17 | Specifies the X-Y plane | b | |
| G 18 | Specifies the Z-X plane | | |
| G 19 | Specifies the Y-Z plane | | |
| G 20 | Inch input | c | |
| G 21 | Millimeter input | | |
| G 39 | Corner-offset circular interpolation | | ○ |
| G 40 | Cancel cutter compensation | d | |
| G 41 | Cutter compensation — left | | |
| G 42 | Cutter compensation — right | | |
| G 43 | Positive tool-length Compensation | e | |
| G 44 | Negative tool-length Compensation | | |
| G 49 | Cancel tool-length Compensation | | |
| G 50 | Cancels scaling | f | |
| G 51 | Scaling | | |
| G 54 | Selects workpiece coordinate system 1 | g | |
| G 55 | Selects workpiece coordinate system 2 | | |
| G 56 | Selects workpiece coordinate system 3 | | |
| G 57 | Selects workpiece coordinate system 4 | | |
| G 58 | Selects workpiece coordinate system 5 | | |
| G 59 | Selects workpiece coordinate system 6 | | |
| G 80 | Cancels fixed cycle | h | |
| G 81 | Fixed cycle | | |
| G 82 | Fixed cycle | | |
| G 85 | Fixed cycle | | |
| G 86 | Fixed cycle | | |
| G 89 | Fixed cycle | | |
| G 90 | Absolute specifications | i | |
| G 91 | Incremental specifications | | |
| G 92 | Coordinate system | | ○ |
| G 98 | Initial level return | j | |
| G 99 | Point R level return | | |

The group indications “a” through “j” signify that the respective functions belong to the same group.

* Maintained until another command is encountered in the same group.

## Miscellaneous Functions (M Functions)

| Code | Functions | Function start | | Function continue | |
|---|---|---|---|---|---|
| | | Functions when specified | Functions after completion of the block in which specified | Maintained until canceled or changed | Effective only within the block in which specified |
| M 00 | Program stop | | ○ | | ○ |
| M 01 | Optional stop | | ○ | | ○ |
| M 02 | End of program | ○ * | | | ○ |
| M 03 | Spindle start | ○ * | | ○ | |
| M 05 | Spindle stop | ○ * | | ○ | |
| M 06 | Tool Change | ○ * | | | ○ |
| M 30 | End of program | ○ * | | | ○ |
| M 98 | Subprogram Call | ○ * | | | ○ |
| M 99 | End of subprogram | ○ * | | | ○ |

○ * : Operation is according to the setting on the PNC-300G.

## Spindle speed Functions (S Functions)

The setting made on the PNC-300G determines whether the operation is performed simultaneously with specification or after completion of the block in which the specification is made.

## Feed Functions (F Functions)

The setting made on the PNC-300G determines whether the operation is performed simultaneously with specification or after completion of the block in which the specification is made.

# Character Code Table (ISO, EIA, and ASCII)

| Meaning | ISO | | | EIA | | | ASCII | | |
|---|---|---|---|---|---|---|---|---|---|
| | Character | Hexadecimal | Decimal | Character | Hexadecimal | Decimal | Character | Hexadecimal | Decimal |
| Numeral 0 | 0 | 30 | 48 | 0 | 20 | 32 | 0 | 30 | 48 |
| Numeral 1 | 1 | B1 | 177 | 1 | 01 | 1 | 1 | 31 | 49 |
| Numeral 2 | 2 | B2 | 178 | 2 | 02 | 2 | 2 | 32 | 50 |
| Numeral 3 | 3 | 33 | 51 | 3 | 13 | 19 | 3 | 33 | 51 |
| Numeral 4 | 4 | B4 | 180 | 4 | 04 | 4 | 4 | 34 | 52 |
| Numeral 5 | 5 | 35 | 53 | 5 | 15 | 21 | 5 | 35 | 53 |
| Numeral 6 | 6 | 36 | 54 | 6 | 16 | 22 | 6 | 36 | 54 |
| Numeral 7 | 7 | B7 | 183 | 7 | 07 | 7 | 7 | 37 | 55 |
| Numeral 8 | 8 | B8 | 184 | 8 | 08 | 8 | 8 | 38 | 56 |
| Numeral 9 | 9 | 39 | 57 | 9 | 19 | 25 | 9 | 39 | 57 |
| Address A | A | 41 | 65 | a | 61 | 97 | A | 41 | 65 |
| Address B | B | 42 | 66 | b | 62 | 98 | B | 42 | 66 |
| Address C | C | C3 | 195 | c | 73 | 115 | C | 43 | 67 |
| Address D | D | 44 | 68 | d | 64 | 100 | D | 44 | 68 |
| Address E | E | C5 | 197 | e | 75 | 117 | E | 45 | 69 |
| Address F | F | C6 | 198 | f | 76 | 118 | F | 46 | 70 |
| Address G | G | 47 | 71 | g | 67 | 103 | G | 47 | 71 |
| Address H | H | 48 | 72 | h | 68 | 104 | H | 48 | 72 |
| Address I | I | C9 | 201 | i | 79 | 121 | I | 49 | 73 |
| Address J | J | CA | 202 | j | 51 | 81 | J | 4A | 74 |
| Address K | K | 4B | 75 | k | 52 | 82 | K | 4B | 75 |
| Address L | L | CC | 204 | l | 43 | 67 | L | 4C | 76 |
| Address M | M | 4D | 77 | m | 54 | 84 | M | 4D | 77 |
| Address M | N | 4E | 78 | n | 45 | 69 | N | 4E | 78 |
| Address O | O | CF | 207 | o | 46 | 70 | O | 4F | 79 |
| Address P | P | 50 | 80 | p | 57 | 87 | P | 50 | 80 |
| Address Q | Q | D1 | 209 | q | 58 | 88 | Q | 51 | 81 |
| Address R | R | D2 | 210 | r | 49 | 73 | R | 52 | 82 |
| Address S | S | 53 | 83 | s | 32 | 50 | S | 53 | 83 |
| Address T | T | D4 | 212 | t | 23 | 35 | T | 54 | 84 |
| Address U | U | 55 | 85 | u | 34 | 52 | U | 55 | 85 |
| Address V | V | 56 | 86 | v | 25 | 37 | V | 56 | 86 |
| Address W | W | D7 | 215 | w | 26 | 38 | W | 57 | 87 |
| Address X | X | D8 | 216 | x | 37 | 55 | X | 58 | 88 |
| Address Y | Y | 59 | 89 | y | 38 | 56 | Y | 59 | 89 |
| Address Z | Z | 5A | 90 | z | 29 | 41 | Z | 5A | 90 |
| Delete | DEL | FF | 255 | Del | 7F | 127 | DEL | 7F | 127 |
| Back Space | BS | 88 | 136 | BS | 2A | 42 | BS | 08 | 8 |
| Tab | HT | 09 | 9 | Tab | 3E | 62 | HT | 09 | 9 |
| End of Block | LF or NL | 0A | 10 | CR | 80 | 128 | LF | 0A | 10 |
| Carriage Return | CR | 8D | 141 | | | | CR | 0D | 13 |
| Space | SP | A0 | 160 | SP | 10 | 16 | SP | 20 | 32 |
| Program Start | % | A5 | 165 | ER | 0B | 11 | % | 25 | 37 |

# Index


/ ..... 58
( ) ..... 59
% ..... 5, 59

## A

Absolute ..... 10, 47
Absolute programming ..... 10, 47
Addresses ..... 4
Amount of movement ..... 10
Amount of offset ..... 27, 31, 39
ASCII ..... 6, 62

## B

BASIC ..... 6
Block ..... 5

## C

C ..... 6
Centerpoint ..... 22
Character code ..... 6, 62
Character code table ..... 62
Circular interpolation ..... 13, 22
Clockwise ..... 22
Comment ..... 59
Coordinate systems ..... 7
Corner-offset circular interpolation ..... 29
Counterclockwise ..... 22
Cutter compensation ..... 30
Cutting feed ..... 44
Data output ..... 6

## D

Dimension words ..... 4
Drilling a hole ..... 43
Dwell ..... 25

## E

EIA ..... 6, 62
End of block ..... 5, 59
End of program ..... 50, 51
End of subprogram ..... 53
EOB ..... 5, 59
Errors ..... 16
EXOFS ..... 8
External workpiece origin point offset amount ..... 8

## F

F functions ..... 56
Fast feed ..... 44
Feed rate ..... 16, 56
Fixed cycle ..... 43

## G

G functions ..... 20
G00 ..... 20
G01 ..... 21
G02 ..... 22
G03 ..... 22
G04 ..... 25
G10 ..... 26
G17 ..... 28
G18 ..... 28
G19 ..... 28
G20 ..... 28
G21 ..... 28
G39 ..... 29
G40 ..... 30
G41 ..... 30
G42 ..... 30
G43 ..... 38
G44 ..... 38
G49 ..... 38
G50 ..... 41
G51 ..... 41
G54 ..... 42
G55 ..... 42
G56 ..... 42
G57 ..... 42
G58 ..... 42
G59 ..... 42
G80 ..... 43
G81 ..... 43
G82 ..... 43
G85 ..... 43
G86 ..... 43
G89 ..... 43
G90 ..... 47
G91 ..... 47
G92 ..... 48
G98 ..... 49
G99 ..... 49
Group of words ..... 60

## H

Helical interpolation ..... 24

## I

Incremental ..... 10, 47
Incremental programming ..... 10, 47
Initial level ..... 43, 49
Integer entry ..... 11
ISO ..... 6, 62
LF (Line Feed) ..... 5
Linear interpolation ..... 13, 21

## M

M functions ..... 50
M00 ..... 50
M01 ..... 50
M02 ..... 50
M03 ..... 50
M05 ..... 50
M06 ..... 51
M30 ..... 51
M98 ..... 52
M99 ..... 53
Machine coordinate systems ..... 7
Main program ..... 4
Miscellaneous functions ..... 50
MS-DOS ..... 6

## N

N ..... 5, 57
Numerical code ..... 54

## O

O ..... 5, 57
Offset number ..... 27, 31, 38
Optional block skip ..... 12, 58
Optional stop ..... 50

## P

Plane ..... 28
Point R level ..... 49
Positioning ..... 13, 20
Preparatory functions ..... 20
Process of programming ..... 3
Process sheet ..... 3
Program format ..... 4
Program number ..... 5, 57
Program start ..... 5, 59
Program stop ..... 50
Program structure ..... 4
Programming ..... 3

## R

Real-number entry ..... 11
Restrictions on cutter compensation ..... 30
Restrictions on tool-length compensation ..... 38

## S

S functions ..... 54
Scaling ..... 41
Selects workpiece coordinate system ..... 8, 42
Sequence number ..... 5, 57
Setting the measurement unit ..... 11, 28
Skip number ..... 58
Specify the radius ..... 22
Spindle motor control ..... 16, 50
Spindle speed ..... 16, 54
Spindle speed function ..... 54
Subprogram ..... 4, 52, 53
Subprogram call ..... 52

## T

Tool change ..... 51
Tool path ..... 15, 30
Tool-length compensation ..... 38

## W

Words ..... 5
Workpiece coordinate systems ..... 7
Workpiece origin point offset amount ..... 8

Roland DIGITAL GROUP

# NC code

## PNC-300G

ZZ100695DE R1-1