

# 6

## REFERENCE POSITION



A CNC machine tool has a special position where, generally, the tool is exchanged or the coordinate system is set, as described later. This position is referred to as a reference position.

## 6.1 REFERENCE POSITION RETURN

### General

- **Reference position**

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function. For example, the reference position is used as a position at which tools are automatically changed. Up to four reference positions can be specified by setting coordinates in the machine coordinate system in parameters (No. 1240 to 1243).

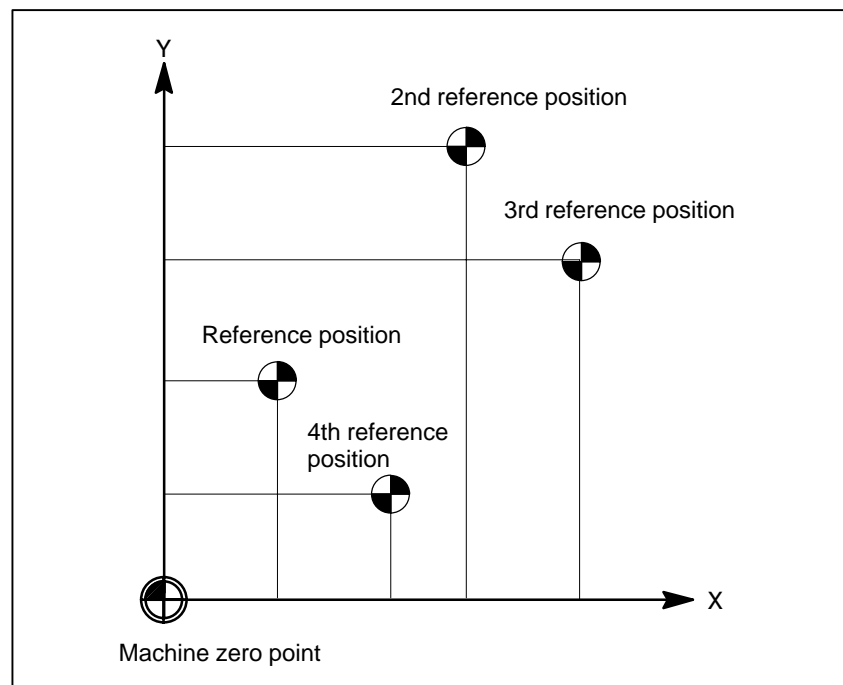
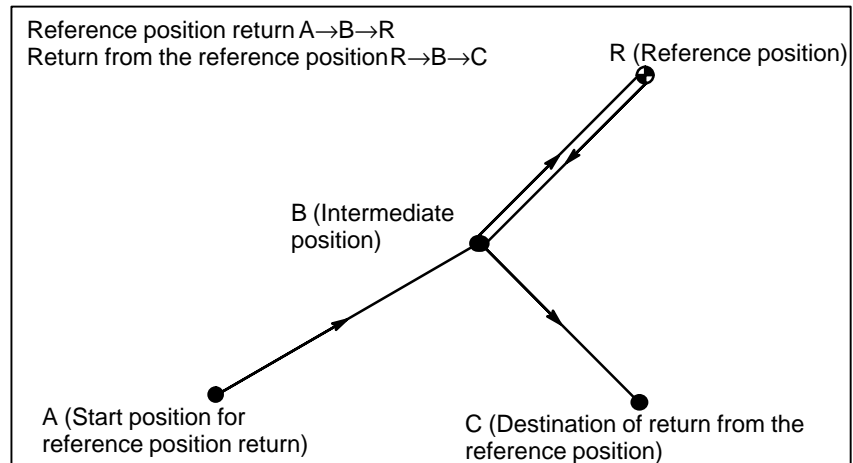


Fig. 6.1 (a) Machine zero point and reference positions

- **Reference position return and movement from the reference position**

Tools are automatically moved to the reference position via an intermediate position along a specified axis. Or, tools are automatically moved from the reference position to a specified position via an intermediate position along a specified axis. When reference position return is completed, the lamp for indicating the completion of return goes on.



**Fig. 6.1 (b) Reference position return and return from the reference position**

- **Reference position return check**

The reference position return check (G27) is the function which checks whether the tool has correctly returned to the reference position as specified in the program. If the tool has correctly returned to the reference position along a specified axis, the lamp for the axis goes on.

### Format

- **Reference position return**

**G28 IP\_ ; Reference position return**  
**G30 P2 IP\_ ; 2nd reference position return** (P2 can be omitted.)  
**G30 P3 IP\_ ; 3rd reference position return**  
**G30 P4 IP\_ ; 4th reference position return**

IP\_ : Command specifying the intermediate position  
 (Absolute/incremental command)

- **Return from reference position**

**G29 IP\_ ;**

IP\_ : Command specifying the destination of return from reference position (Absolute/incremental command)

- **Reference position return check**

**G27 IP\_ ;**

IP\_ : Command specifying the reference position  
 (Absolute/incremental command)

## Explanations

- **Reference position return (G28)**

Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis.  
 Therefore, for safety, the cutter compensation, and tool length compensation should be cancelled before executing this command.  
 The coordinates for the intermediate position are stored in the CNC only for the axes for which a value is specified in a G28 block. For the other axes, the previously specified coordinates are used.  
**Example N1 G28 X40.0 ; Intermediate position (X40.0)**  
**N2 G28 Y60.0 ; Intermediate position (X40.0, Y60.0)**
- **2nd, 3rd, and 4th reference position return (G30)**

In a system without an absolute-position detector, the first, third, and fourth reference position return functions can be used only after the reference position return (G28) or manual reference position return (see III-3.1) is made. The G30 command is generally used when the automatic tool changer (ATC) position differs from the reference position.
- **Return from the reference position (G29)**

In general, it is commanded immediately following the G28 command or G30. For incremental programming, the command value specifies the incremental value from the intermediate point.  
 Positioning to the intermediate or reference points are performed at the rapid traverse rate of each axis.  
 When the workpiece coordinate system is changed after the tool reaches the reference position through the intermediate point by the G28 command, the intermediate point also shifts to a new coordinate system. If G29 is then commanded, the tool moves to the commanded position through the intermediate point which has been shifted to the new coordinate system.  
 The same operations are performed also for G30 commands.
- **Reference position return check (G27)**

G27 command positions the tool at rapid traverse rate. If the tool reaches the reference position, the reference position return lamp lights up. However, if the position reached by the tool is not the reference position, an alarm (No. 092) is displayed.
- **Setting of the reference position return feedrate**

Before a machine coordinate system is established with the first reference position return after power-on, the manual and automatic reference position return feedrates and automatic rapid traverse rate conform to the setting of parameter No. 1428 for each axis. Even after a machine coordinate system is established upon the completion of reference position return, the manual reference position return feedrate conforms to the setting of the parameter.

**NOTE**

- 1 To this feedrate, a rapid traverse override (F0,25,50,100%) is applied, for which the setting is 100%.
- 2 After a machine coordinate system has been established upon the completion of reference position return, the automatic reference position return feedrate will conform to the ordinary rapid traverse rate.
- 3 For the manual rapid traverse rate used before a machine coordinate system is established upon the completion of reference position return a jog feedrate or manual rapid traverse rate can be selected using RPD (bit 0 of parameter No. 1401).

	Before a coordinate system is established	After a coordinate system is established
Automatic reference position return (G28)	No. 1428	No.1420
Automatic rapid traverse (G00)	No.1428	No.1420
Manual reference position return	No.1428	No.1428
Manual rapid traverse rate	No.1423 *1	No.1424

**NOTE**

When parameter No. 1428 is set to 0, the feedrates conform to the parameter settings shown below.

	Before a coordinate system is established	After a coordinate system is established
Automatic reference position return (G28)	No. 1420	No.1420
Automatic rapid traverse (G00)	No.1420	No.1420
Manual reference position return	No.1424	No.1424
Manual rapid traverse rate	No.1423 *1	No.1424

1420 : Rapid traverse rate

1423 : Jog feedrate

1424 : Manual rapid traverse rate

\*1 Setting of parameter No.1424 when RPD (bit 0 of parameter No.1401) is set to 1.

### Restrictions

- **Status the machine lock being turned on**
- **First return to the reference position after the power has been turned on (without an absolute position detector)**
- **Reference position return check in an offset mode**
- **Lighting the lamp when the programmed position does not coincide with the reference position**

The lamp for indicating the completion of return does not go on when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a G27 command is specified.

When the G28 command is specified when manual return to the reference position has not been performed after the power has been turned on, the movement from the intermediate point is the same as in manual return to the reference position.

In this case, the tool moves in the direction for reference position return specified in parameter ZMIx (bit 5 of No. 1006). Therefore the specified intermediate position must be a position to which reference position return is possible.

In an offset mode, the position to be reached by the tool with the G27 command is the position obtained by adding the offset value. Therefore, if the position with the offset value added is not the reference position, the lamp does not light up, but an alarm is displayed instead. Usually, cancel offsets before G27 is commanded.

When the machine tool system is an inch system with metric input, the reference position return lamp may also light up even if the programmed position is shifted from the reference position by the least setting increment. This is because the least setting increment of the machine tool system is smaller than its least command increment.

### Reference

- **Manual reference position return**

See III-3.1.

### Examples

**G28G90X1000.0Y500.0** ; (Programs movement from A to B)

**T1111** ; (Changing the tool at the reference position)

**G29X1300.0Y200.0** ; (Programs movement from B to C)

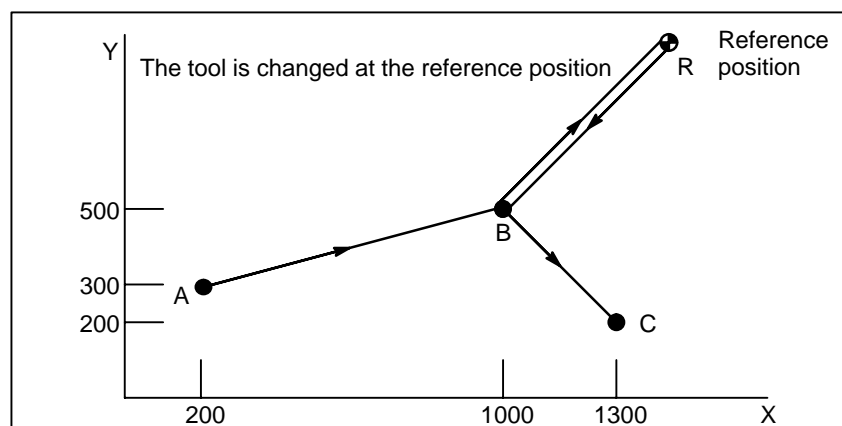


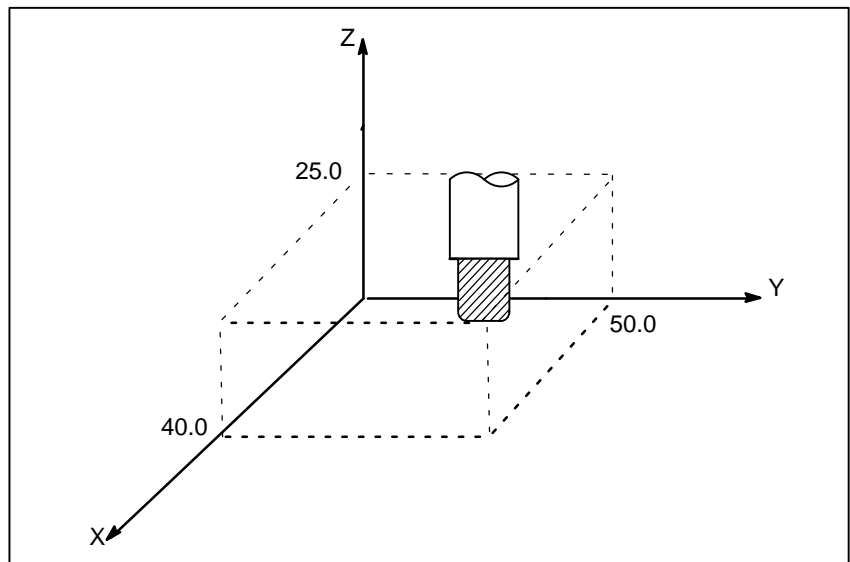
Fig. 6.1 (c) Reference position return and return from the reference position

# 7 COORDINATE SYSTEM

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes. When three program axes, the X-axis, Y-axis, and Z-axis, are used, coordinates are specified as follows:

X Y Z

This command is referred to as a dimension word.



**Fig. 7 Tool position specified by X40.0Y50.0Z25.0**

Coordinates are specified in one of following three coordinate systems:

- (1) Machine coordinate system
- (2) Workpiece coordinate system
- (3) Local coordinate system

The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP\_.

## 7.1 MACHINE COORDINATE SYSTEM

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine.

A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.

A machine coordinate system is set by performing manual reference position return after power-on (see III-3.1). A machine coordinate system, once set, remains unchanged until the power is turned off.

### Format

**(G90)G53 IP\_ ;**  
**IP\_ ; Absolute dimension word**

### Explanations

- **Selecting a machine coordinate system (G53)**

When a command is specified the position on a machine coordinate system, the tool moves to the position by rapid traverse. G53, which is used to select a machine coordinate system, is a one-shot G code; that is, it is valid only in the block in which it is specified on a machine coordinate system. Specify an absolute command (G90) for G53. When an incremental command (G91) is specified, the G53 command is ignored. When the tool is to be moved to a machine-specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

### Restrictions

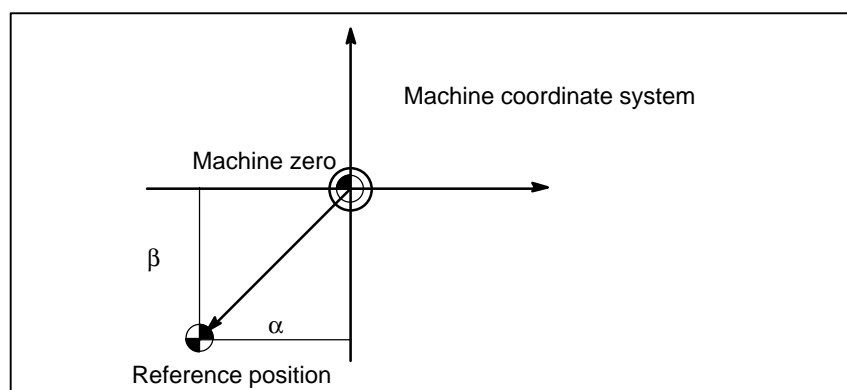
- **Cancel of the compensation function**
- **G53 specification immediately after power-on**

When the G53 command is specified, cancel the cutter compensation, tool length offset, and tool offset.

Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute-position detector is attached.

### Reference

When manual reference position return is performed after power-on, a machine coordinate system is set so that the reference position is at the coordinate values of ( $\alpha$ ,  $\beta$ ) set using parameter No.1240.





## 7.2 WORKPIECE COORDINATE SYSTEM

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the CNC beforehand (setting a workpiece coordinate system).

A machining program sets a workpiece coordinate system (selecting a workpiece coordinate system).

A set workpiece coordinate system can be changed by shifting its origin (changing a workpiece coordinate system).

### 7.2.1 Setting a Workpiece Coordinate System

A workpiece coordinate system can be set using one of three methods:

#### (1) Method using G92

A workpiece coordinate system is set by specifying a value after G92 in the program.

#### (2) Automatic setting

If bit 0 of parameter SPR No. 1201 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed (see Part III-3.1.).

#### (3) Input using the CRT/MDI panel

Six workpiece coordinate systems can be set beforehand using the MDI panel (see Part III-11.4.6.).

When using an absolute command, establish the workpiece coordinate system in any of the above ways.

### Format

- Setting a workpiece coordinate system by G92

**(G90) G92 P\_**

### Explanations

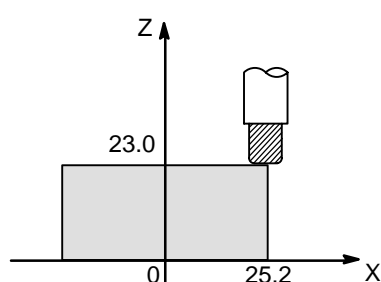
A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates. If a coordinate system is set using G92 during tool length offset, a coordinate system in which the position before offset matches the position specified in G92 is set.

Cutter compensation is cancelled temporarily with G92.

### Examples

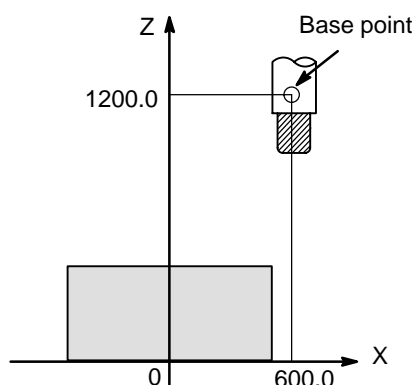
#### Example 1

Setting the coordinate system by the G92X25.2Z23.0; command  
(The tool tip is the start point for the program.)



#### Example 2

Setting the coordinate system by the G92X600.0Z1200.0; command  
(The base point on the tool holder is the start point for the program.)



If an absolute command is issued, the base point moves to the commanded position. In order to move the tool tip to the commanded position, the difference from the tool tip to the base point is compensated by tool length offset.

## 7.2.2 Selecting a Workpiece Coordinate System

The user can choose from set workpiece coordinate systems as described below. (For information about the methods of setting, see II- 7.2.1.)

- (1) Once a workpiece coordinate system is selected by G92 or automatic workpiece coordinate system setting, absolute commands work with the workpiece coordinate system.
- (2) Choosing from six workpiece coordinate systems set using the CRT/MDI panel  
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

G54 Workpiece coordinate system 1

G55 Workpiece coordinate system 2

G56 Workpiece coordinate system 3

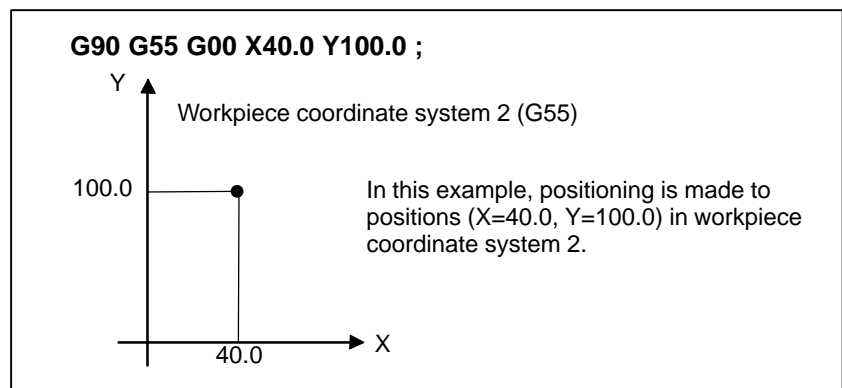
G57 Workpiece coordinate system 4

G58 Workpiece coordinate system 5

G59 Workpiece coordinate system 6

Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected.

## Examples



**Fig. 7.2.2**

### 7.2.3 Changing Workpiece Coordinate System

The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.

Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.

(1) Inputting from the MDI panel (see III-11.4.6)

(2) Programming by G10 or G92

(3) Using the external data input function

An external workpiece zero point offset value can be changed by input signal to CNC. Refer to machine tool builder's manual for details

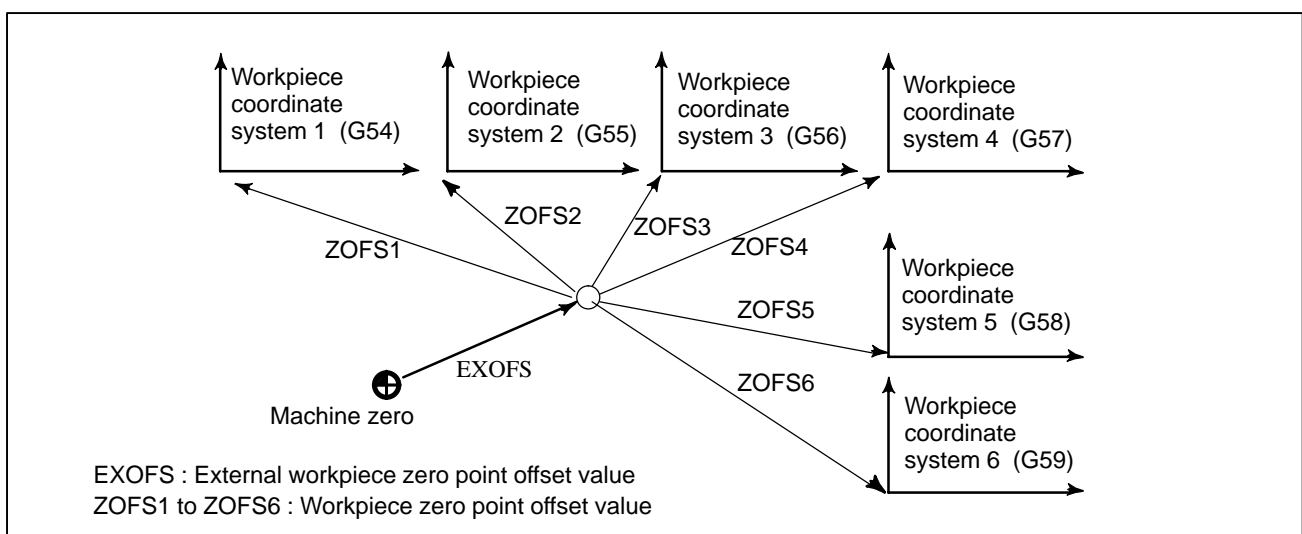


Fig. 7.2.3 Changing an external workpiece zero point offset value or workpiece zero point offset value

#### Format

##### • Changing by G10

**G10 L2 Pp IP\_;**

p=0 : External workpiece zero point offset value

p=1 to 6 : Workpiece zero point offset value correspond to workpiece coordinate system 1 to 6

IP\_ : For an absolute command (G90), workpiece zero point offset for each axis.

For an incremental command (G91), value to be added to the set workpiece zero point offset for each axis (the result of addition becomes the new workpiece zero point offset).

##### • Changing by G92

**G92 IP\_;**

## Explanations

- **Changing by G10**

With the G10 command, each workpiece coordinate system can be changed separately.

- **Changing by G92**

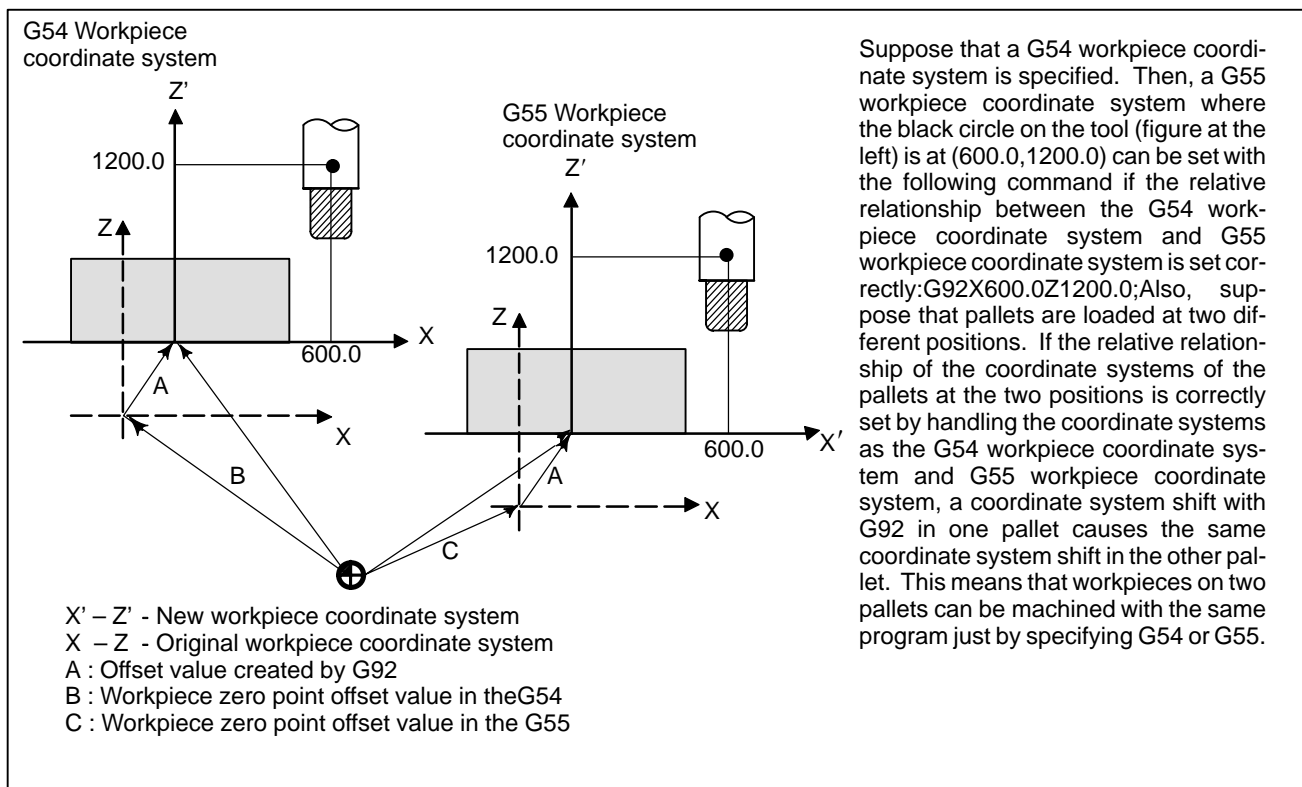
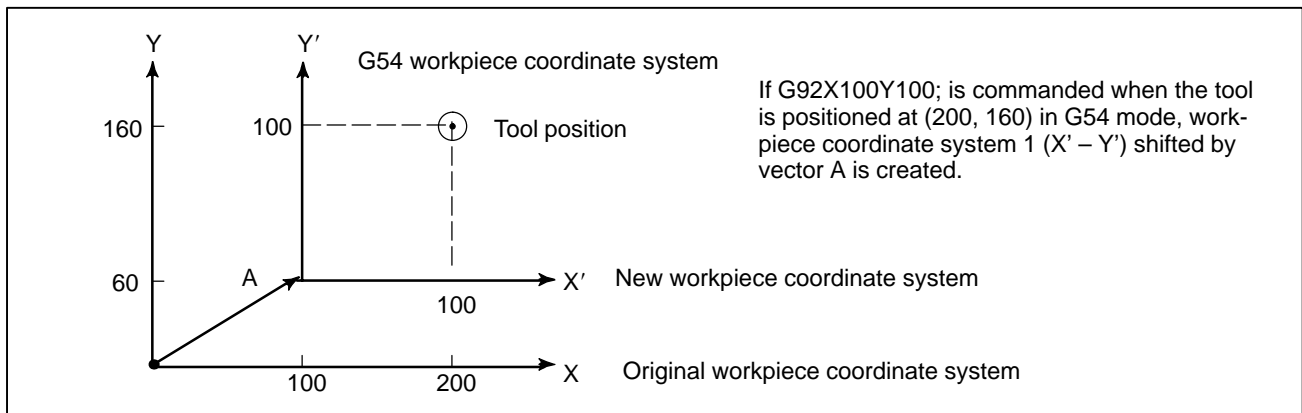
By specifying G92IP\_;, a workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates ( IP\_).

Then, the amount of coordinate system shift is added to all the workpiece zero point offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

### **WARNING**

When a coordinate system is set with G92 after an external workpiece zero point offset value is set, the coordinate system is not affected by the external workpiece zero point offset value. When G92X100.0Z80.0; is specified, for example, the coordinate system having its current tool reference position at X = 100.0 and Z = 80.0 is set.

## Examples



## 7.2.4 Workpiece coordinate system preset (G92.1)

The workpiece coordinate system preset function presets a workpiece coordinate system shifted by manual intervention to the pre-shift workpiece coordinate system. The latter system is displaced from the machine zero point by a workpiece zero point offset value.

There are two methods for using the workpiece coordinate system preset function. One method uses a programmed command (G92.1). The other uses MDI operations on the absolute position display screen, relative position display screen, and overall position display screen (III-11.1.4).

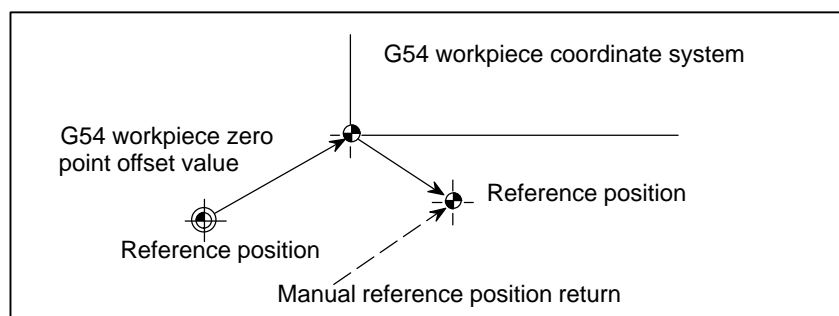
### Format

#### **G92.1 IP 0 ;**

IP 0 ; Specifies axis addresses subject to the workpiece coordinate system preset operation. Axes that are not specified are not subject to the preset operation.

### Explanations

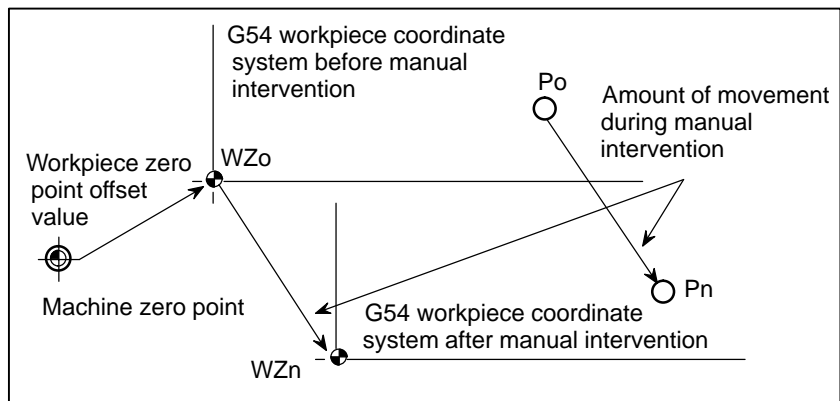
When manual reference position return operation is performed in the reset state, a workpiece coordinate system is shifted by the workpiece zero point offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate system is selected with G54. In this case, a workpiece coordinate system is automatically set which has its zero point displaced from the machine zero point by the G54 workpiece zero point offset value; the distance from the zero point of the workpiece coordinate system to the reference position represents the current position in the workpiece coordinate system.



If an absolute position detector is provided, the workpiece coordinate system automatically set at power-up has its zero point displaced from the machine zero point by the G54 workpiece zero point offset value. The machine position at the time of power-up is read from the absolute position detector and the current position in the workpiece coordinate system is set by subtracting the G54 workpiece zero point offset value from this machine position. The workpiece coordinate system set by these operations is shifted from the machine coordinate system using the commands and operations listed next page.

- (a) Manual intervention performed when the manual absolute signal is off
- (b) Move command executed in the machine lock state
- (c) Movement by handle interrupt
- (d) Operation using the mirror image function
- (e) Setting the local coordinate system using G52, or shifting the workpiece coordinate system using G92

In the case of (a) above, the workpiece coordinate system is shifted by the amount of movement during manual intervention.



In the operation above, a workpiece coordinate system once shifted can be preset using G code specification or MDI operation to a workpiece coordinate system displaced by a workpiece zero point offset value from the machine zero point. This is the same as when manual reference position return operation is performed on a workpiece coordinate system that has been shifted. In this example, such G code specification or MDI operation has the effect of returning workpiece coordinate system zero point WZn to the original zero point WZo, and the distance from WZo to Pn is used to represent the current position in the workpiece coordinate system.

Bit 3 (PPD) of parameter No. 3104 specifies whether to preset relative coordinates (RELATIVE) as well as absolute coordinates.

## Limitations

- **Cutter compensation, tool length compensation, tool offset**
- **Program restart**
- **Prohibited modes**

When using the workpiece coordinate system preset function, cancel compensation modes: cutter compensation, tool length compensation, and tool offset. If the function is executed without cancelling these modes, compensation vectors are temporarily cancelled.

The workpiece coordinate system preset function is not executed during program restart.

Do not use the workpiece coordinate system preset function when the scaling, coordinate system rotation, programmable image, or drawing copy mode is set.

## 7.2.5 Adding Workpiece Coordinate Systems (G54.1 or G54)

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used. Alternatively, up to 300 additional workpiece coordinate systems can be used.

### Format

- **Selecting the additional workpiece coordinate systems**
- **Setting the workpiece zero point offset value in the additional workpiece coordinate systems**

#### **G54.1Pn ; or G54Pn ;**

Pn : Codes specifying the additional workpiece coordinate systems  
n : 1 to 48

#### **G10L20 Pn IP\_;**

Pn : Codes specifying the workpiece coordinate system for setting the workpiece zero point offset value  
n : 1 to 48  
IP\_ : Axis addresses and a value set as the workpiece zero point offset

### Explanations

- **Selecting the additional workpiece coordinate systems**

When a P code is specified together with G54.1 (G54), the corresponding coordinate system is selected from the additional workpiece coordinate systems (1 to 48).

A workpiece coordinate system, once selected, is valid until another workpiece coordinate system is selected. Standard workpiece coordinate system 1 (selectable with G54) is selected at power-on.

```
G54.1 P1 ... Additional workpiece coordinate system 1
G54.1 P2 ... Additional workpiece coordinate system 2
      ⋮
G54.1 P48 .. Additional workpiece coordinate system 48
```

As with the standard workpiece coordinate systems, the following operations can be performed for a workpiece zero point offset in an additional workpiece coordinate system:

- (1) The OFFSET function key can be used to display and set a workpiece zero point offset value.
- (2) The G10 function enables a workpiece zero point offset value to be set by programming (refer to II-7.2.3).
- (3) A custom macro allows a workpiece zero point offset value to be handled as a system variable.
- (4) Workpiece zero point offset data can be entered or output as external data.
- (5) The PMC window function enables workpiece zero point offset data to be read as program command modal data.



- **Setting the workpiece zero point offset value in the additional workpiece coordinate systems**

When an absolute workpiece zero point offset value is specified, the specified value becomes a new offset value. When an incremental workpiece zero point offset value is specified, the specified value is added to the current offset value to produce a new offset value.

### **Limitations**

- **Specifying P codes**

A P code must be specified after G54.1 (G54). If G54.1 is not followed by a P code in the same block, additional workpiece coordinate system 1 (G54.1P1) is assumed.

If a value not within the specifiable range is specified in a P code, an P/S alarm (No. 030) is issued.

P codes other than workpiece offset numbers cannot be specified in a G54.1 (G54) block.

Example) G54.1 (G54) G04 P1000 ;

## 7.3 LOCAL COORDINATE SYSTEM

When a program is created in a workpiece coordinate system, a child workpiece coordinate system can be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

### Format

**G52 IP \_; Setting the local coordinate system**

.....

**G52 IP 0 ; Canceling of the local coordinate system**

IP \_ : Origin of the local coordinate system

### Explanations

By specifying G52 IP \_;, a local coordinate system can be set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP \_ in the workpiece coordinate system.

When a local coordinate system is set, the move commands in absolute mode (G90), which is subsequently commanded, are the coordinate values in the local coordinate system. The local coordinate system can be changed by specifying the G52 command with the zero point of a new local coordinate system in the workpiece coordinate system.

To cancel the local coordinate system and specify the coordinate value in the workpiece coordinate system, match the zero point of the local coordinate system with that of the workpiece coordinate system.

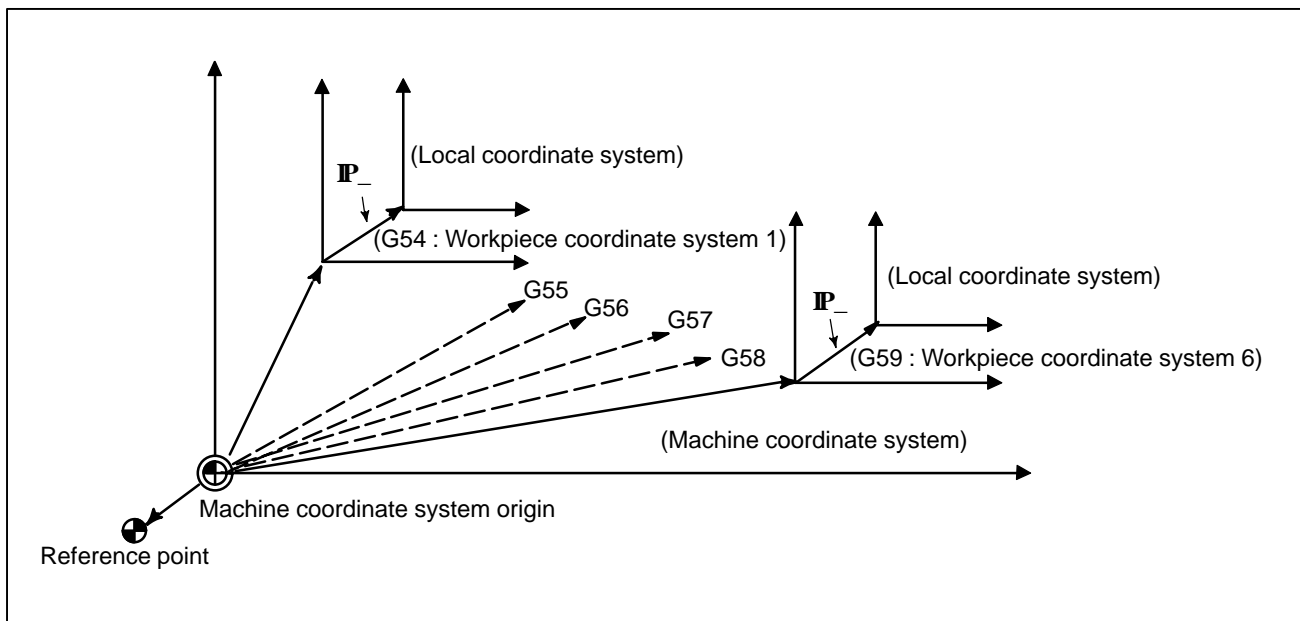


Fig. 7.3 Setting the local coordinate system

**WARNING**

- 1 When an axis returns to the reference point by the manual reference point return function, the zero point of the local coordinate system of the axis matches that of the work coordinate system. The same is true when the following command is issued:  
    G52 $\alpha$ 0;  
     $\alpha$ : Axis which returns to the reference point
- 2 The local coordinate system setting does not change the workpiece and machine coordinate systems.
- 3 Whether the local coordinate system is canceled at reset depends on the parameter setting. The local coordinate system is canceled when either CLR, bit 6 of parameter No.3402 or RLC, bit 3 of parameter No.1202 is set to 1.
- 4 If coordinate values are not specified for all axes when setting a workpiece coordinate system with the G92 command, the local coordinate systems of axes for which coordinate values were not specified are not cancelled, but remain unchanged.
- 5 G52 cancels the offset temporarily in cutter compensation.
- 6 Command a move command immediately after the G52 block in the absolute mode.

## 7.4 PLANE SELECTION

Select the planes for circular interpolation, cutter compensation, and drilling by G-code.

The following table lists G-codes and the planes selected by them.

### Explanations

**Table 7.4 Plane selected by G code**

G code	Selected plane	Xp	Yp	Zp
G17	Xp Yp plane	X-axis or an axis parallel to it	Y-axis or an axis parallel to it	Z-axis or an axis parallel to it
G18	Zp Xp plane			
G19	Yp Zp plane			

Xp, Yp, Zp are determined by the axis address appeared in the block in which G17, G18 or G19 is commanded.

When an axis address is omitted in G17, G18 or G19 block, it is assumed that the addresses of basic three axes are omitted.

Parameter No. 1022 is used to specify that an optional axis be parallel to the each axis of the X, Y-, and Z-axes as the basic three axes.

The plane is unchanged in the block in which G17, G18 or G19 is not commanded.

When the power is turned on or the CNC is reset, G17 (XY plane), G18 (ZX plane), or G19 (YZ plane) is selected by bits 1 (G18) and 2 (G19) of parameter 3402.

The movement instruction is irrelevant to the plane selection.

### Examples

Plane selection when the X-axis is parallel with the U-axis.

G17X\_Y\_ XY plane,

G17U\_Y\_ UY plane

G18X\_Z\_ ZX plane

X\_Y\_ Plane is unchanged (ZX plane)

G17 XY plane

G18 ZX plane

G17 U\_ UY plane

G18Y\_ ; ZX plane, Y axis moves regardless without any relation to the plane.