14. COMPENSATION FUNCTION

FORMAT

- **Start up**
  (Tool compensation start)

<table>
<thead>
<tr>
<th>Command for axis movement</th>
<th>Code for specifying as the cutter compensation value (1–3 digits)</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00 (or G01) G41 (or G42)</td>
<td>IP_ _ D_ _ ;</td>
</tr>
<tr>
<td>G41 : Cutter compensation left (Group 07)</td>
<td></td>
</tr>
<tr>
<td>G42 : Cutter compensation right (Group 07)</td>
<td></td>
</tr>
<tr>
<td>IP_ _ : Command for axis movement</td>
<td></td>
</tr>
<tr>
<td>D_ _ : Code for specifying as the cutter compensation value (1–3 digits)</td>
<td></td>
</tr>
</tbody>
</table>

- **Cutter compensation cancel**
  (offset mode cancel)

<table>
<thead>
<tr>
<th>Command for axis movement</th>
</tr>
</thead>
<tbody>
<tr>
<td>G40 ;</td>
</tr>
<tr>
<td>G40 : Cutter compensation cancel (Group 07)</td>
</tr>
<tr>
<td>Offset mode cancel</td>
</tr>
<tr>
<td>IP_ _ : Command for axis movement</td>
</tr>
</tbody>
</table>

- **Selection of the offset plane**

<table>
<thead>
<tr>
<th>Offset plane</th>
<th>Command for plane selection</th>
<th>IP_ _</th>
</tr>
</thead>
<tbody>
<tr>
<td>XpYp</td>
<td>G17 ;</td>
<td>Xp_Yp_</td>
</tr>
<tr>
<td>ZpXp</td>
<td>G18 ;</td>
<td>Xp_Zp_</td>
</tr>
<tr>
<td>YpZp</td>
<td>G19 ;</td>
<td>Yp_Zp_</td>
</tr>
</tbody>
</table>

EXPLANATIONS

- **Offset cancel mode**

  At the beginning when power is applied the control is in the cancel mode. In the cancel mode, the vector is always 0, and the tool center path coincides with the programmed path.

- **Start Up**

  When a cutter compensation command (G41 or G42, nonzero dimension words in the offset plane, and D code other than D0) is specified in the offset cancel mode, the CNC enters the offset mode.

  Moving the tool with this command is called start-up.

  Specify positioning (G00) or linear interpolation (G01) for start-up. If circular interpolation (G02, G03) is specified, P/S alarm 34 occurs.

  When processing the start-up block and subsequent blocks, the CNC prereads two blocks.

- **Offset mode**

  In the offset mode, compensation is accomplished by positioning (G00), linear interpolation (G01), or circular interpolation (G02, G03). If two or more blocks that do not move the tool (miscellaneous function, dwell, etc.) are processed in the offset mode, the tool will make either an excessive or insufficient cut. If the offset plane is switched in the offset mode, P/S alarm 37 occurs and the tool is stopped.
• Offset mode cancel

In the offset mode, when a block which satisfies any one of the following conditions is executed, the CNC enters the offset cancel mode, and the action of this block is called the offset cancel.

1. G40 has been commanded.
2. 0 has been commanded as the offset number for cutter compensation.

When performing offset cancel, circular arc commands (G02 and G03) are not available. If a circular arc is commanded, an P/S alarm (No. 034) is generated and the tool stops.

In the offset cancel, the control executes the instructions in that block and the block in the cutter compensation buffer. In the meantime, in the case of a single block mode, after reading one block, the control executes it and stops. By pushing the cycle start button once more, one block is executed without reading the next block.

Then the control is in the cancel mode, and normally, the block to be executed next will be stored in the buffer register and the next block is not read into the buffer for cutter compensation.

Fig. 14.4 (b) Changing the offset mode

• Change of the Cutter compensation value

In general, the cutter compensation value shall be changed in the cancel mode, when changing tools. If the cutter compensation value is changed in offset mode, the vector at the end point of the block is calculated for the new cutter compensation value.

Fig. 14.4 (c) Changing the cutter compensation value
• **Positive/negative cutter compensation value and tool center path**

If the offset amount is negative (−), distribution is made for a figure in which G41’s and G42’s are all replaced with each other on the program. Consequently, if the tool center is passing around the outside of the workpiece, it will pass around the inside, and vice versa. The figure below shows one example. Generally, the offset amount is programmed to be positive (+).

When a tool path is programmed as in ((1)), if the offset amount is made negative (−), the tool center moves as in ((2)), and vice versa. Consequently, the same tape permits cutting both male and female shapes, and any gap between them can be adjusted by the selection of the offset amount. Applicable if start-up and cancel is A type. (See II—14.5.2 and 14.5.4)

![Tool center paths when positive and negative cutter compensation values are specified](image)

• **Cutter compensation value setting**

Assign a cutter compensation values to the D codes on the MDI panel. The table below shows the range in which cutter compensation values can be specified.

<table>
<thead>
<tr>
<th>Cutter compensation value</th>
<th>mm input</th>
<th>inch input</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0 to ±999.999mm</td>
<td>0 to ±99.999inch</td>
</tr>
</tbody>
</table>

**NOTE**

1. The cutter compensation value corresponding to offset No. 0, that is, D0 always means 0. It is impossible to set D0 to any other offset amount.
2. Cutter compensation C can be specified by H code with parameter OFH (No. 5001 #2) set to 1.

• **Offset vector**

The offset vector is the two dimensional vector that is equal to the cutter compensation value assigned by D code. It is calculated inside the control unit, and its direction is updated in accordance with the progress of the tool in each block.

The offset vector is deleted by reset.

• **Specifying a cutter compensation value**

Specify a cutter compensation value with a number assigned to it. The number consists of 1 to 3 digits after address D (D code). The D code is valid until another D code is specified. The D code is used to specify the tool offset value as well as the cutter compensation value.
**Plane selection and vector**

Offset calculation is carried out in the plane determined by G17, G18 and G19, (G codes for plane selection). This plane is called the offset plane. Compensation is not executed for the coordinate of a position which is not in the specified plane. The programmed values are used as they are.

In simultaneous 3 axes control, the tool path projected on the offset plane is compensated.

The offset plane is changed during the offset cancel mode. If it is performed during the offset mode, a P/S alarm (No. 37) is displayed and the machine is stopped.
**Examples**

```
G92 X0 Y0 Z0 ; ......................... Specifies absolute coordinates.
N1 G90 G17 G00 G41 D07 X250.0 Y550.0 ;  
   Starts cutter compensation (start–up). The tool is shifted to the
   left of the programmed path by the distance specified in D07.
   In other words the tool path is shifted by the radius of the tool
   (offset mode) because D07 is set to 15 beforehand (the radius of
   the tool is 15 mm).

N2 G01 Y900.0 F150 ; ...................... Specifies machining from P1 to P2.
N3 X450.0 ; ............................... Specifies machining from P2 to P3.
N4 G03 X500.0 Y1150.0 R650.0 ;  .......... Specifies machining from P3 to P4.
N5 G02 X900.0 R–250.0 ; .................. Specifies machining from P4 to P5.
N6 G03 X950.0 Y900.0 R650.0 ; .......... Specifies machining from P5 to P6.
N7 G01 X1150.0 ; ......................... Specifies machining from P6 to P7.
N8 Y550.0 ; ............................... Specifies machining from P7 to P8.
N9 X700.0 Y650.0 ; ........................ Specifies machining from P8 to P9.
N10 X250.0 Y550.0 ; ........................ Specifies machining from P9 to P1.
N11 G00 G40 X0 Y0 ; ..................... Cancels the offset mode.
   The tool is returned to the start position (X0, Y0, Z0).
```
14.5 DETAILS OF CUTTER COMPENSATION C

This section provides a detailed explanation of the movement of the tool for cutter compensation C outlined in Section 14.4. This section consists of the following subsections:

14.5.1 General
14.5.2 Tool Movement in Start-up
14.5.3 Tool Movement in Offset Mode
14.5.4 Tool Movement in Offset Mode Cancel
14.5.5 Interference Check
14.5.6 Overcutting by Cutter Compensation
14.5.7 Input Command from MDI
14.5.8 G53, G28, G30 and G29 Commands in Cutter Compensation C Mode
14.5.9 Corner Circular Interpolation (G39)

14.5.1 General

When an angle of intersection created by tool paths specified with move commands for two blocks is over 180°, it is referred to as “inner side.” When the angle is between 0° and 180°, it is referred to as “outer side.”

- **Inner side and outer side**

  - **Inner side**
    - Workpiece
    - Programmed path
    - $180^\circ \leq \alpha$

  - **Outer side**
    - Workpiece
    - Programmed path
    - $0^\circ < \alpha < 180^\circ$

- **Meaning of symbols**

  The following symbols are used in subsequent figures:
  - $S$ indicates a position at which a single block is executed once.
  - $SS$ indicates a position at which a single block is executed twice.
  - $SSS$ indicates a position at which a single block is executed three times.
  - $L$ indicates that the tool moves along a straight line.
  - $C$ indicates that the tool moves along an arc.
  - $r$ indicates the cutter compensation value.
  - An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by $r$.
  - $\bullet$ indicates the center of the tool.