OKUMA BASIC OSP MACHINING CENTER PROGRAMMING

REFERENCE MATERIALS
IN THE ORDER WHICH THEY APPEAR AND ARE REFERENCED

BASIC G CODES

BASIC M CODES

STM FUNCTION

G00 & G01 (RAPID TRAVERSE AND FEED MODE)

G94 & G95 (IPM AND IPR)

G04 (DWELL)

G15 (FIXTURE OFFSET)

G40-G42 (CUTTER RADIUS COMPENSATION)

FIXED CYCLES

DRAWINGS

SURVEY
## SECTION 15 APPENDIX

### 15-1. G Code Table (Including Optional Functions)

<table>
<thead>
<tr>
<th>G Code</th>
<th>G Group</th>
<th>Functions</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00 ***</td>
<td>1</td>
<td>Positioning</td>
</tr>
<tr>
<td>G01 ***</td>
<td>1</td>
<td>Linear interpolation</td>
</tr>
<tr>
<td>G02</td>
<td></td>
<td>Circular interpolation - Helical cutting (CW)</td>
</tr>
<tr>
<td>G03</td>
<td></td>
<td>Circular interpolation - Helical Cutting (CCW)</td>
</tr>
<tr>
<td>G04 **</td>
<td>2</td>
<td>Dwell</td>
</tr>
<tr>
<td>G09 **</td>
<td>18</td>
<td>Exact stop</td>
</tr>
<tr>
<td>G10 *</td>
<td>3</td>
<td>Cancel of G11</td>
</tr>
<tr>
<td>G11</td>
<td></td>
<td>Parallel and rotational shift of coordinate system</td>
</tr>
<tr>
<td>G14 **</td>
<td>64</td>
<td>Designation of axis name</td>
</tr>
<tr>
<td>G15</td>
<td></td>
<td>Selection of work coordinate system (Modal)</td>
</tr>
<tr>
<td>G16 **</td>
<td>4</td>
<td>Selection of work coordinate system (One-shot)</td>
</tr>
<tr>
<td>G17 ***</td>
<td>5</td>
<td>Plane selection: XY</td>
</tr>
<tr>
<td>G18 ***</td>
<td>5</td>
<td>Plane selection: ZX</td>
</tr>
<tr>
<td>G19 ***</td>
<td>5</td>
<td>Plane selection: YZ</td>
</tr>
<tr>
<td>G20 **</td>
<td>15</td>
<td>Inch input confirmation</td>
</tr>
<tr>
<td>G21 **</td>
<td></td>
<td>Metric input confirmation</td>
</tr>
<tr>
<td>G22 ***</td>
<td>6</td>
<td>Programmable stroke limit ON</td>
</tr>
<tr>
<td>G23 ***</td>
<td></td>
<td>Programmable stroke limit cancel</td>
</tr>
<tr>
<td>G30 **</td>
<td>16</td>
<td>Positioning to home position</td>
</tr>
<tr>
<td>G31 **</td>
<td>17</td>
<td>Skip function</td>
</tr>
<tr>
<td>G33</td>
<td>1</td>
<td>Unfixed cycle for thread cutting</td>
</tr>
<tr>
<td>G37</td>
<td>22</td>
<td>Cancel of G38</td>
</tr>
<tr>
<td>G38</td>
<td></td>
<td>Air cut reduction mode ON</td>
</tr>
<tr>
<td>G39 **</td>
<td>84</td>
<td>I, J, K commands and G39 ignored in the corner circular interpolation (with vector)</td>
</tr>
<tr>
<td>G40 *</td>
<td>7</td>
<td>Cutter radius compensation (G41, G42) cancel</td>
</tr>
<tr>
<td>G41</td>
<td></td>
<td>Tool side offset (G45, G46) cancel</td>
</tr>
<tr>
<td>G42</td>
<td></td>
<td>Leading edge offset (G47) cancel</td>
</tr>
<tr>
<td>G43 *</td>
<td>8</td>
<td>3D offset (G44) cancel</td>
</tr>
<tr>
<td>G44</td>
<td></td>
<td>3D offset ON</td>
</tr>
<tr>
<td>G45</td>
<td></td>
<td>Tool side offset for 5-axis machining left</td>
</tr>
<tr>
<td>G46</td>
<td></td>
<td>Tool side offset for 5-axis machining right</td>
</tr>
<tr>
<td>G47</td>
<td></td>
<td>Leading edge offset ON</td>
</tr>
<tr>
<td>G50 *</td>
<td>9</td>
<td>Enlargement and reduction of geometry cancel</td>
</tr>
<tr>
<td>G51</td>
<td></td>
<td>Enlargement and reduction of geometry ON</td>
</tr>
<tr>
<td>G Code</td>
<td>G Group</td>
<td>Functions</td>
</tr>
<tr>
<td>----------</td>
<td>---------</td>
<td>---------------------------------------------------------------------------</td>
</tr>
<tr>
<td>G53 ***</td>
<td></td>
<td>Tool length offset cancel</td>
</tr>
<tr>
<td>G54</td>
<td></td>
<td>Tool length offset, X-axis</td>
</tr>
<tr>
<td>G55</td>
<td></td>
<td>Tool length offset, Y-axis</td>
</tr>
<tr>
<td>G56 ***</td>
<td>10</td>
<td>Tool length offset, Z-axis</td>
</tr>
<tr>
<td>G57</td>
<td></td>
<td>Tool length offset, 4th-axis</td>
</tr>
<tr>
<td>G58</td>
<td></td>
<td>Tool length offset, 5th-axis</td>
</tr>
<tr>
<td>G59</td>
<td></td>
<td>Tool length offset, 6th-axis</td>
</tr>
<tr>
<td>G60</td>
<td>1</td>
<td>One-directional positioning</td>
</tr>
<tr>
<td>G61</td>
<td>14</td>
<td>Exact stop mode ON</td>
</tr>
<tr>
<td>G62 **</td>
<td>19</td>
<td>Programmable mirror image mode</td>
</tr>
<tr>
<td>G64 *</td>
<td>14</td>
<td>Cutting mode ON</td>
</tr>
<tr>
<td>G68</td>
<td>24</td>
<td>Slope coordinate OFF</td>
</tr>
<tr>
<td>G69</td>
<td></td>
<td>Slope coordinate ON</td>
</tr>
<tr>
<td>G71 **</td>
<td>21</td>
<td>Designation of return level for M53</td>
</tr>
<tr>
<td>G72 **</td>
<td>23</td>
<td>Designation of pattern reference point (start position) for the coordinate calculation function</td>
</tr>
<tr>
<td>G73</td>
<td></td>
<td>Fixed cycle, High speed drilling cycle</td>
</tr>
<tr>
<td>G74</td>
<td></td>
<td>Fixed cycle, Reverse tapping cycle</td>
</tr>
<tr>
<td>G76</td>
<td></td>
<td>Fixed cycle, Fine boring</td>
</tr>
<tr>
<td>G79</td>
<td></td>
<td>Fixed cycle; Variable pitch cycle</td>
</tr>
<tr>
<td>G80 *</td>
<td></td>
<td>Fixed cycle, Modal cancel</td>
</tr>
<tr>
<td>G81</td>
<td></td>
<td>Fixed cycle; Spot boring</td>
</tr>
<tr>
<td>G82</td>
<td>11</td>
<td>Fixed cycle; Counter boring</td>
</tr>
<tr>
<td>G83</td>
<td></td>
<td>Fixed cycle, Deep hole drilling cycle</td>
</tr>
<tr>
<td>G84</td>
<td></td>
<td>Fixed cycle, Tapping cycle</td>
</tr>
<tr>
<td>G85</td>
<td></td>
<td>Fixed cycle, Boring cycle</td>
</tr>
<tr>
<td>G86</td>
<td></td>
<td>Fixed cycle, Boring cycle</td>
</tr>
<tr>
<td>G87</td>
<td></td>
<td>Fixed cycle, Back boring cycle</td>
</tr>
<tr>
<td>G88</td>
<td></td>
<td>Fixed cycle; Multi-step boring cycle</td>
</tr>
<tr>
<td>G89</td>
<td></td>
<td>Fixed cycle, Boring cycle</td>
</tr>
<tr>
<td>G90 ***</td>
<td>12</td>
<td>Absolute dimensioning mode</td>
</tr>
<tr>
<td>G91 ***</td>
<td></td>
<td>Incremental dimensioning mode</td>
</tr>
<tr>
<td>G92 **</td>
<td>20</td>
<td>Setting of work coordinate system</td>
</tr>
<tr>
<td>G93</td>
<td></td>
<td>Inverse time feed</td>
</tr>
<tr>
<td>G94 ***</td>
<td>13</td>
<td>Feed per minute mode</td>
</tr>
<tr>
<td>G95 ***</td>
<td></td>
<td>Feed per revolution mode</td>
</tr>
<tr>
<td>G96 **</td>
<td>20</td>
<td>Local coordinate system setting</td>
</tr>
<tr>
<td>G97 **</td>
<td>74</td>
<td>Coordinate recalculation function</td>
</tr>
<tr>
<td>G100 **</td>
<td>32</td>
<td>Cancel of macro MODIN mode selected by G101 to G110</td>
</tr>
</tbody>
</table>
## 15-3. M Code Table

<table>
<thead>
<tr>
<th>M Code</th>
<th>Group</th>
<th>Function</th>
<th>Execution Timing (In Reference to Axis Movement Command)</th>
<th>Modal/One shot</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>M00</td>
<td>01</td>
<td>1 Program stop</td>
<td>After</td>
<td>One shot</td>
<td>Spindle and coolant stop (Can be selected by parameter setting)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Optional stop</td>
<td>After</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>02</td>
<td>18</td>
<td>End of program</td>
<td>After</td>
<td>One shot</td>
<td>NC reset</td>
</tr>
<tr>
<td>03</td>
<td>2</td>
<td>Work spindle start (CW)</td>
<td>At the same time</td>
<td>Modal</td>
<td>Rotates the work spindle clockwise when viewed from the workpiece.</td>
</tr>
<tr>
<td>04</td>
<td>3</td>
<td>Work spindle start (CCW)</td>
<td>At the same time</td>
<td>Modal</td>
<td>Rotates the work spindle clockwise when viewed from the workpiece.</td>
</tr>
<tr>
<td>05</td>
<td></td>
<td>Spindle stop</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>06</td>
<td>3</td>
<td>Vertical spindle tool change</td>
<td>After</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>07</td>
<td>8</td>
<td>Oil mist coolant ON</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>08</td>
<td>10</td>
<td>Coolant pump ON</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>09</td>
<td>24</td>
<td>Coolant system OFF (M07, 08, 12, 50, 51, 59 OFF)</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>30</td>
<td>A-axis clamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td></td>
<td>11</td>
<td>A-axis unclamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>22</td>
<td>Chip air blow ON</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>5</td>
<td>4th-axis rotary index table CW</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td></td>
<td>16</td>
<td>4th-axis rotary index table CCW</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td></td>
<td>Indexing head index CCW</td>
<td>At the same time</td>
<td>One shot</td>
<td>Command effective for only M73-M76</td>
</tr>
<tr>
<td>19</td>
<td>2</td>
<td>Spindle orientation (forward)</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>20</td>
<td>31</td>
<td>B-axis clamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td></td>
<td>21</td>
<td>B-axis unclamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>32</td>
<td>Y-axis clamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td></td>
<td>23</td>
<td>Y-axis unclamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>33</td>
<td>Z-axis clamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td></td>
<td>25</td>
<td>Z-axis unclamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>26</td>
<td>35</td>
<td>C-axis clamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td></td>
<td>27</td>
<td>C-axis unclamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>18</td>
<td>End of tape</td>
<td>After</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>32</td>
<td></td>
<td>Splash guard door close</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>33</td>
<td></td>
<td>Splash guard door open</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>M Code</td>
<td>Group</td>
<td>Function</td>
<td>Execution Timing (In Reference to Axis Movement Command)</td>
<td>Modal/One shot</td>
<td>Remarks</td>
</tr>
<tr>
<td>--------</td>
<td>-------</td>
<td>-----------------------------------------------</td>
<td>---------------------------------------------------------</td>
<td>----------------</td>
<td>-------------------------------------------------------------------------</td>
</tr>
<tr>
<td>40</td>
<td></td>
<td>High/middle-high/middle-low/low range</td>
<td>At the same time</td>
<td>Modal</td>
<td>Spindle gears are automatically determined by spindle speed command.</td>
</tr>
<tr>
<td>41</td>
<td></td>
<td>High/middle-high/middle-low range</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>42</td>
<td></td>
<td>High/middle-high range</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>43</td>
<td></td>
<td>High range</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>44</td>
<td>11</td>
<td>AAC (F) 1 Next attachment clear</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>45</td>
<td></td>
<td>AAC (F) 1 Preparation for attachment change preparation</td>
<td>At the same time</td>
<td>One shot</td>
<td>F: Floor type</td>
</tr>
<tr>
<td>46</td>
<td></td>
<td>AAC (F) 1 No next attachment</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>47</td>
<td></td>
<td>AAC (F) 1 No next attachment</td>
<td>At the same time</td>
<td>One shot</td>
<td>T: Table type</td>
</tr>
<tr>
<td>48</td>
<td></td>
<td>AAC (T) Next attachment clear</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>49</td>
<td></td>
<td>AAC (T) Preparation for attachment change preparation</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>50</td>
<td>23</td>
<td>Through-the-tool coolant, low pressure ON</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>51</td>
<td></td>
<td>Through-the-tool coolant, high pressure ON</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>52</td>
<td>12</td>
<td>Return level in fixed cycle, Upper limit</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>53</td>
<td>13</td>
<td>Return level in fixed cycle, Specified level</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>54</td>
<td></td>
<td>Return level in fixed cycle, Point R level</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>57</td>
<td>34</td>
<td>W-axis clamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>58</td>
<td></td>
<td>W-axis unclamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>59</td>
<td>25</td>
<td>Chip air blow ON</td>
<td>At the same time</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>60</td>
<td>4</td>
<td>Pallet change command</td>
<td>After</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>62</td>
<td></td>
<td>Vertical spindle tool change preparation</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>63</td>
<td></td>
<td>No next tool for ATC</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>64</td>
<td></td>
<td>Next tool return cycle</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>65</td>
<td></td>
<td>ATC preparation</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>66</td>
<td></td>
<td>Continuous tool change between the vertical and horizontal spindles (same tool)</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>67</td>
<td></td>
<td>Continuous tool change between the vertical and horizontal spindles (different tool)</td>
<td>At the same time</td>
<td>One shot</td>
<td></td>
</tr>
<tr>
<td>68</td>
<td></td>
<td>Vertical spindle tool clamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>69</td>
<td></td>
<td>Vertical spindle tool unclamp</td>
<td>After</td>
<td>Modal</td>
<td></td>
</tr>
<tr>
<td>70</td>
<td>3</td>
<td>Manual tool change</td>
<td>After</td>
<td>One shot</td>
<td></td>
</tr>
</tbody>
</table>
SECTION 5  S, T, AND M FUNCTIONS

This section describes the S, T, and M codes which specify necessary machine operations other than axis movement commands.

S: Spindle speed
T: Tool number for tool change cycle
M: Turning solenoids and other similar devices on and off

Only one of each of these types of code may be specified in one block. If two or more commands of the same code type are issued to one block, the rightmost command of each code type will be executed.

<Example> M6 T1 T2------M6 T2 will be executed.

5-1. S Code Function

[Function]
The spindle function specifies a spindle speed with a numeric value (up to five digits) entered following address S.

[Details]
• The desired spindle speed (min⁻¹) is directly specified by a numeric value following the address S.
  Programmable range: 0 to 65535
• If an S command is specified with axis movement commands in the same block, the S command becomes valid at the same time axis movement commands are executed.
• Although an S command is not canceled when the NC is reset, it is cleared when the power supply is turned off.
• To execute a spindle rotation command (M03, M04), an S command must be specified in the same or a previous block.

5-2. T Code Function

[Function]
The tool function selects a tool in the machine with a numeric value (up to eight digits) entered following address T.

[Details]
• The programmable range of a T command is indicated below.
  Programmable range: 0 to 99999999
• When a T code is executed, the next tool is prepared (indexing the next tool in the magazine, or taking the next tool out of the magazine and setting it in the ready station position).
• The actual tool change cycle is executed by M06.
• If a T command is specified with axis movement commands in the same block, the execution timing of the T code can be selected from the following two timings:
  Executed simultaneously with axis movement commands
  Executed after the completion of axis movement commands
5-3. M Code Function

[Function]
The M code function outputs an M code number, consisting of a four-digit number and address M, and the strobe to the PLC. The programmable range of M codes is from 0 to 1023.

5-3-1. Examples of M Codes

The followings are examples of M codes.

(1) M02, M30 (End of Program)
These M codes indicate the end of a program. When M02 or M30 is executed, the main program ends and reset processing is executed. The program is rewound to its start. (In the case of a schedule program, execution of M02 or M30 in the main program does not reset the NC.)

(2) M03, M04, M05 (Spindle CW/CCW and Stop)
These M codes control spindle rotation and stop; spindle CW (M03), spindle CCW (M04), and spindle stop (M05).

(3) M19 (Spindle Orientation)
The M19 command is used with machines equipped with the spindle orientation mechanism. The spindle orientation function stops the spindle at a specified angular position.

- Multi-point spindle indexing
  By specifying “RS=angle” following M19, it is possible to index the spindle at the specified angular position. Although the following explanation uses M19 as an example, the same applies to M118 and M119.
  M19 RS = \( \theta \)
  - \( \theta \) represents the desired index angle and it is specified in units of 1°. If a value smaller than 1° is specified, it is truncated.
  - Programmable range of \( \theta \): 0 to 360°
  - \( \theta \) specifies the desired index angle of the spindle, measured in the CW rotation angle in reference to the spindle orientation position.

[Supplement]

- If M19 (M118, M119) is specified without argument RS, ordinary spindle orientation is performed. That is, the called operation is the same as that called by “M19 RS=0”.
- RS must always be specified in the same block as M19 (M118, M119).

(4) M52 (Fixed Cycle - Return to the Retract End)
In various fixed cycles, this command sets the return position of the cycle axis 0.1 mm away from the travel limit of the Z-axis in the positive direction. For details, refer to SECTION 7, “7-2. Fixed Cycle Operations”.

(5) M53 (Fixed Cycle - Return to the Specified Point)
In various fixed cycles, this command sets the return position of the cycle axis at the position specified by G71. For details, refer to SECTION 7, “7-2. Fixed Cycle Operations”.
(6) M54 (Fixed Cycle - Return to Point R Level)
In various fixed cycles, this command sets the return position of the cycle axis at the position
specified by R command.
For details, refer to SECTION 7, “7-2. Fixed Cycle Operations”.

(7) M132, M133 (Single Block Valid/Invalid)
These M codes set whether the single block function is made invalid (M132) or valid (M133)
independently of the setting of the single block switch on the machine operation panel.

(8) M201 to M210 (M Code Macro)
By setting the program names which correspond to M201 to M210 in the parameters, the sub
programs can be executed by specifying the M codes.
For details of M code macro, refer to SECTION 10, “10-4. G and M Code Macro Functions”.

(9) M238, M239 (Soft-override Valid/Invalid)
These commands set whether or not the soft-override value (%) set for system variables <VFSOV> is valid (M238) or invalid (M239) for the cutting feedrate (F command × override value).

(10) M00 (Program Stop)
After the execution of M00, the program stops. If the NC is started in this program stop state, the program restarts.

(11) M01 (Optional Stop)
When M01 is executed while the optional stop switch on the machine operation panel is ON, the program stops. If the NC is started in this optional stop state, the program restarts.

(12) M06 (Tool Change)
This M code is used with machines equipped with the tool change mechanism as the tool change cycle start command.

(13) M15, M16 (Fourth Axis - Rotary Table CW, CCW)
These M codes are used with machines equipped with the rotary table as the fourth axis to specify the direction of rotary table rotation; CW (M15), CCW (M16).
For details of the rotary table control, refer to SECTION 17, “Additional Axis (Rotary Axis) Function” in SPECIAL FUNCTIONS Manual No.1.

(14) M115, M116 (Fifth Axis - Rotary Table CW, CCW)
These M codes are used with machines equipped with the rotary table as the fifth axis to specify the direction of rotary table rotation; CW (M115), CCW (M116)
For details of rotary table control, refer to SECTION 17, “Additional Axis (Rotary Axis) Function” in SPECIAL FUNCTIONS Manual No.1.

(15) M118, M119 (Spindle Index - CCW, Shorter Path)
These M codes are used with machines equipped with the spindle index mechanism as the spindle orientation direction specifying command.
[Programming format]
- M118 Spindle index (CCW)
- M119 Spindle index (shorter path)

(16) M130, M131 (For Cutting Feed, Spindle Rotation Condition Valid / Invalid)
Usually, in the G01, G02, and G03 modes, the spindle must be rotating to execute axis feed.
These M codes are set to ignore this condition (M130) or validate it (M131).
(17) M134, M135 (Spindle Speed Override Valid / Invalid)
Even in the status in which spindle speed override control from the PLC is valid, the spindle speed override function can be made invalid (M134) or valid (M135) with these commands.

(18) M136, M137 (Axis Feed Override Valid / Invalid)
These M codes set whether the axis feed override function is made invalid (M136) or valid (M137) independently of the ON status of the axis feed override signal from the PLC.

(19) M138, M139 (Dry Run Valid / Invalid)
These M codes set whether the dry run function is made invalid (M138) or valid (M139) independently of the setting of the dry run switch on the machine operation panel.

(20) M140, M141 (Slide Hold Valid / Invalid)
These M codes set whether the slide hold function is made invalid (M140) or valid (M141) independently of the setting of the slide hold switch on the machine operation panel.

(21) M234 to M237 (Gear Selection Range for Synchronized Tapping)
These M codes set the gear selection range for synchronized tapping. For details, refer to “Torque Monitoring Function” in Synchronized Tapping of SPECIAL FUNCTIONS Manual.

(22) M326, M327 (Torque Monitor ON / OFF for Synchronized Tapping)
These M codes turn ON (M326) and OFF (M327) the torque monitor mode for synchronized tapping.

(23) M331, M332 (Sixth Axis - Rotary Table CW / CCW)
These M codes are used with machines equipped with the rotary table as the sixth axis to specify the direction of rotary table rotation; CW (M331), CCW (M332). For details of rotary table control, refer to SECTION 17, “Additional Axis (Rotary Axis) Function” in SPECIAL FUNCTIONS Manual No.1.

(24) M396 to M399 (Gear Position Selection for Synchronized Tapping)
These are gear position commands, specially for synchronized tapping, introduced by the gear selection range specifying M codes (M234 to M237) and the S command. They are automatically generated by the NC.

* M396: 1st gear command for synchronized tapping
* M397: 2nd gear command for synchronized tapping
* M398: 3rd gear command for synchronized tapping
* M399: 4th gear command for synchronized tapping
6-1. Tool Length Offset Function (G53 - G59)

[Function]
The tool length offset function compensates for the position of a cutting tool so that the tip of the cutting tool is located at the programmed position.

Available G Codes

<table>
<thead>
<tr>
<th>G Code</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>G53</td>
<td>Cancel tool length offset</td>
</tr>
<tr>
<td>G54</td>
<td>Tool length offset, X-axis</td>
</tr>
<tr>
<td>G55</td>
<td>Tool length offset, Y-axis</td>
</tr>
<tr>
<td>G56</td>
<td>Tool length offset, Z-axis</td>
</tr>
<tr>
<td>G57</td>
<td>Tool length offset, 4th-axis</td>
</tr>
<tr>
<td>G58</td>
<td>Tool length offset, 5th-axis</td>
</tr>
<tr>
<td>G59</td>
<td>Tool length offset, 6th-axis</td>
</tr>
</tbody>
</table>

[Programming format]

{G54 - G59}  IP__ H__

| IP:   | Current position of tool tip after compensation |
| H:    | Tool length offset number                     |

The standard tool length offset numbers are HA to HC and H00 to H100, and this can be expanded to H200, H300 or H999. When H00 is specified, the offset amount always becomes "0".

Tool length offset data are set in the tool data setting mode. Setting range: 0 to ±999.999 mm (0 to ±39.3700 inches)

[Details]

- The displayed actual tool position value always includes the tool length offset amount.
- The tool length offset cannot be applied to two or more axes at the same time or to the rotary axis.
- Tool length offset numbers may be changed directly without having to cancel the previous command with G53.
- When the NC is reset, H00 is automatically set.
3-6. Positioning

3-6-1. Positioning (G00)

[Function]
The axes move from the present position to the target position at rapid feedrate. During this movement, axes are automatically accelerated and decelerated.

[Programming format]
G00 IP

In the positioning operation executed in the G00 mode, in-position check is executed. The commands in the next block are executed only after the in-position state is confirmed (in-position width is set for a system parameter).

[Details]

- Whether positioning is executed in the linear pattern or a non-linear pattern is determined by the setting for NC optional parameter (bit) No. 46, bit 0.

  a. Linear interpolation pattern
  The tool path is generated along a straight line from the actual position to the target position. In this movement, the feedrates of the individual axes are determined within the individual rapid feedrates so that positioning time can be minimized.

  ![](image1.png)

  b. Non-linear interpolation pattern
  The individual axes move independently of each other at the individual rapid feedrates. Therefore, the resultant tool path is not always a straight line.

  ![](image2.png)

- The rapid feedrate of the individual axes is set by the machine tool builder and cannot be changed.

- The in-position range is set for each axis using system parameters.
3-7. Linear Interpolation (G01)

[Function]
In the G01 linear interpolation mode, axes move directly from the actual position to the specified target point at the specified feedrate.

[Programming format]
G01 IP__F__
IP: Target point (end point)
F: Feedrate. The specified feedrate remains valid until updated by another value.

[Details]
- A feedrate value specified with address “F” is cleared to zero when the NC is reset. Note that the F command value is saved when the NC is reset if a feedrate is specified in an F1-digit command.
- The feedrate for each axis is as indicated below. (For values X, Y, and Z, convert them into an incremental value.)

\[ \text{G01 } XxYyZzFf \]
Calculation of feedrates:
- X-axis feedrate: \( FX = \frac{x}{\sqrt{x^2+y^2+z^2}}f \)
- Y-axis feedrate: \( FY = \frac{y}{\sqrt{x^2+y^2+z^2}}f \)
- Z-axis feedrate: \( FZ = \frac{z}{\sqrt{x^2+y^2+z^2}}f \)

Where, \( L = \sqrt{x^2+y^2+z^2} \)

For the rotary axis, the unit of feedrate is regarded as indicated below:
- 1 mm/min = 1 deg/min
- 1 inch/min = 1 deg/min

In linear interpolation including a rotary axis, the feedrates are determined according to the formulas given above for the individual axes.

Example:
G91 G01 X10 C20 F30.0

<“mm” input>

\[ X\text{-axis feedrate } FX = \frac{10}{\sqrt{10^2+20^2}} \times 30 = 13.41 \text{ mm/min} \]
\[ C\text{-axis feedrate } FC = \frac{20}{\sqrt{10^2+20^2}} \times 30 = 26.83 \text{ deg/min} \]
SECTION 3 FEED FUNCTIONS

3-1. Rapid Feed

In the rapid feed mode, each of the axes moves at the specified rapid feedrate independently of other axes that are moved at the same time. Note that rapid feedrate differs depending on the machine specification. Consequently, the individual axes arrive at the target point at different times. Override is possible.

3-2. Cutting Feed

3-2-1. Feed per Minute (G94)

[Function]
This function sets the feedrate per minute of a cutting tool with a numeric value following address “F”.

[Programming format]
G94

Setting unit:
Selection is possible from among 1 mm/min, 0.1 mm/min, 1 inch/min, 0.1 inch/min and 0.01 inch/min by setting NC optional parameter (INPUT UNIT SYSTEM).

Setting range: 0.1 ~ 24000.0 mm/min, 0.01 ~ 2400.00 inch/min

[Details]

- The allowable maximum feedrate that is called the “clamp feedrate” is set with NC optional parameter (long word) No. 10. If an axis is going to move beyond this limit, its feedrate is clamped at this clamp feedrate and the following alarm message is displayed at the alarm display line on the screen.
  4204 ALARM-D Feedrate command limit over (replacing)

- The programmed feedrate can be overridden. The clamp feedrate is applied to the actual feedrate, or the overridden feedrate.

3-2-2. Feed per Revolution (G95)

[Function]
This function sets the feedrate per revolution of a cutting tool with a numeric value following address “F”.

[Programming format]
G95

Setting unit:
Selection is possible from among 1 mm/rev, 0.01 mm/rev, 0.001 mm/rev, 1 inch/rev, 0.001 inch/rev or 0.0001 inch/rev by setting NC optional parameter (INPUT UNIT SYSTEM).

Setting range:0.001 ~ 500.000 mm/rev, 0.0001 ~ 50.000 inch/rev

[Details]
3-9. Circular Interpolation (G02, G03)

[Function]
The circular interpolation function moves a tool from the actual position to the specified position along an arc at the specified feedrate.

[Programming format]

Arc on Xp-Yp plane: G17 \( \{ \frac{Xp}{Yp} \_ \frac{G02}{G03} \_ \frac{R_\_}{I_\_J_\_} \_ F_\_ \)\n
Arc on Zp-Xp plane: G18 \( \{ \frac{Zp}{Xp} \_ \frac{G02}{G03} \_ \frac{R_\_}{K_\_I_\_} \_ F_\_ \)\n
Arc on Yp-Zp plane: G19 \( \{ \frac{Yp}{Zp} \_ \frac{G02}{G03} \_ \frac{R_\_}{J_\_K_\_} \_ F_\_ \)\n
\( Xp = X\text{-axis or U-axis} \)
\( Yp = Y\text{-axis or V-axis} \)
\( Zp = Z\text{-axis or W-axis} \)

- G codes used for the circular interpolation function are indicated below.

  G17 : Plane selection : Sets the circular arc in the Xp-Yp plane.
  G18 : Plane selection : Sets the circular arc in the Zp-Xp plane.
  G19 : Plane selection : Sets the circular arc in the Yp-Zp plane.
  G02 : Direction of rotation : Sets the clockwise direction.
  G03 : Direction of rotation : Sets the counterclockwise direction.

Two axes among Xp, Yp, and Zp, G90 mode:
Sets the end point in the work coordinate system
Two axes among Xp, Yp, and Zp, G91 mode:
Sets the position in reference to the start point with signed values.
Two axes among I, J, and K:
Sets the distance from the start point to the center of an arc with signed values.
R: Sets the radius of an arc.
F: Sets the feedrate.

[Details]

- Direction of rotation, clockwise or counterclockwise, is defined when viewing the plane from the positive direction of the Zp-axis (Yp-axis, Xp-axis) on the Xp-Yp (Zp-Xp, Yp-Zp) plane, as shown in the illustrations below.
• The end point is defined by either an absolute value or an incremental value according to G90 or G91. The center point of an arc is determined by the I, J, and K values which correspond to Xp, Yp, and Zp, respectively. Their coordinate values are always specified as incremental values, regardless of G90 or G91.

A minus sign should be used for the I, J, and K values when necessary.

• The end point of an arc can be designated by specifying the coordinate value on one of the two axes. If only one axis is specified, the processing may be selected from the following two methods.

a. For axes with no command, the previous command value is used as the end point of the arc. (For this processing, set the value for axis not programmed (circular single-axis) of NC optional parameter (circular interpolation) as the current value.)

When programming an arc as illustrated to the left, the end point of the arc can be designated with only the coordinate value of the horizontal axis, since the coordinate value of the vertical axis is the same at the start and end points. An alarm occurs if the end point does not lie on an arc.

The left program defines a clockwise arc:

Radius: 100
Center: (0, 0)
Start point: (-70.711, -70.711)
End point: (70.711, -70.711)

Ex: X -70.711, Y -70.711
G02 X70.711 I70.711 J70.711

The left program will cause an alarm, since the end point (10, -70.711) is not on the arc.

b. For the omitted axis, the coordinate value is calculated using the coordinate value of the specified axis. For this processing, choose point on arc at command value for the axis not programmed (single-axis) of NC optional parameter (circular interpolation).

When programming an arc as illustrated in the left, the end point can be designated with only the horizontal axis coordinate value. The vertical axis coordinate value is calculated from the horizontal axis coordinate value.
If more than one cross point is possible, the one which is reached first in the designated arc direction is selected.

Example:

X-70.711 Y-70.711 F200 The program to the left defines a clockwise arc:
G02 X10 I70.711 J70.711 Radius: 100
Center: (0, 0)
Start point: (-70.711, -70.711)
End point: (10, 99.499)

The operations explained above also apply when designation of a vertical axis is omitted.

The center of an arc can be defined by specifying the radius (R) of the arc instead of specifying I, J, and K. If an arc is specified by the radius, four arcs that pass the same start and end points are defined. To define a specific arc from among these four arcs, an R value is used in the manner indicated below.

Clockwise arc (G02)
An arc whose central angle is smaller than or equal to 180 degrees: Radius R > 0
An arc whose central angle is greater than 180 degrees: Radius R < 0

Counterclockwise arc (G03)
An arc whose central angle is smaller than or equal to 180 degrees: Radius R > 0
An arc whose central angle is greater than 180 degrees: Radius R < 0

The feedrate in circular interpolation is the feedrate component tangential to the arc.

[Supplement]

- If I, J, or K is omitted, it is regarded that "0" is specified.
- An arc with radius 0 (R = 0) cannot be specified.
- If the values for Xp, Yp, and Zp are omitted, an arc having the start and end points on the same point is defined in the following manner:
  a) If the center is specified by I, J, and/or K, a 360-degree arc
  b) If the radius is specified by R, a 0-degree arc
- It is not possible to specify R, and I, J, and K at the same time.
- It is not possible to specify any axis parallel to the axes which make up the selected plane. For example, designation of the W-axis is not allowed when the Z-X plane is selected.
- An alarm will occur if the difference in radius between the start point and the end point of an arc is greater than or equal to the value set at arc check data (difference in radius between start and end) of the NC optional parameter (circular interpolation).
3-10. Helical Cutting (G02, G03) (Optional)

[Function]
Helical cutting or helical interpolation may be executed by synchronizing circular interpolation with linear interpolation of the axis which intersects at right angles the plane in which the arc is defined.

[Programming format]

\[ \text{XpYp plane : G17 } \begin{cases} \text{G02} \\ \text{G03} \end{cases} \text{ Xp__Yp__ } \begin{cases} \text{R__} \\ \text{I__J__} \end{cases} \text{ } \alpha \text{ } F \]

\( \alpha \): An axis not parallel to the axes comprising the arc plane

[Details]

- Helical cutting may also be programmed on the Zp-Xp (G18) and Yp-Zp (G19) planes, using a format similar to that above.

- To program helical cutting, simply add the command of the axis which intersects the arc plane to the circular interpolation.

- Helical cutting is possible for an arc having a center angle of smaller than 360 degrees.

- The feedrate specified by an F command is valid for circular interpolation. Therefore, the feedrate in the direction of the linear axis is calculated by the following formula:

\[
\text{Feedrate in the linear axis direction} = \text{Motion distance of the linear axis} \times F \\
\text{Arc length}
\]

- Tool length offset is valid for the axis at right angles to the arc plane.

- Cutter radius compensation is valid only for circular interpolation commands.
SECTION 4 PREPARATORY FUNCTIONS

G codes consisting of address character G and a three-digit number (00 to 399) set the mode that specifies how the commands are executed. Instead of using address character G, some G codes are expressed by mnemonics. A mnemonic code consists of up to eight alphabetic characters (A to Z).

- Valid range of G codes
  - One-shot G codes:
    Valid only in the specified block. Such G codes are automatically canceled when a program advances to the next block.
  - Modal G codes:
    Once specified, such G codes remain valid until another G code in the same group is specified.

- Special G codes
  Mnemonic codes used for subprogram call and those used as branch instructions are called special G codes. Special G codes must be specified at the beginning of a block and entry of such codes at a middle of a block is not allowed. Note, however, that a slash “/” code (optional block skip code) or a sequence name may be placed before a special G code.

- For the tables of G codes and mnemonic codes, refer to “15-1. G Code Table” and “15-2. Table of Mnemonic Codes” in APPENDIX.

4-1. Dwell Command (G04)

[Function]
At the end of the specified block, the dwell function suspends the execution of a program for the specified length of time before proceeding to the next block.

[Programming format]
The following two programming formats may be used to specify the dwell function.

- G04 F__
  F: Sets the length of dwell time
  The unit of dwell time can be selected from 1, 0.1, 0.01 and 0.001 seconds by the NC optional parameter (INPUT UNIT SYSTEM).
  The maximum programmable dwell time is 99999.999 seconds.

- G04 P__
  P: Sets the length of dwell time
  The unit of dwell time is selected in the same manner as when specified by F.
4-3. **Work Coordinate System Selection (G15, G16)**

[Function]
20 sets of work coordinate systems are supplied as a standard feature and this can be expanded to 50, 100 or 200 sets optionally.

[Programming format]
Modal G code: G15 Hn (0 ≤ n ≤ 200)
Once a new work coordinate system "n" is set using the modal G code, the coordinate values specified in the same and later blocks are interpreted as coordinate values in the selected work coordinate system "n".
One-shot G code: G16 Hn (0 ≤ n ≤ 200)
If a new work coordinate system "n" is set using the one-shot G code, only the coordinate values specified in the same block are interpreted as coordinate values in the selected work coordinate system "n".

[Details]
- For G15 and G16, the work coordinate system number between 1 and 200 is specified by "n" (1 to 200). If "0" is specified for "n", the machine coordinate system is selected.
- When the power supply is turned ON, and after the NC is reset, the work coordinate system previously selected by G15 is automatically selected.
- G15 and G16 may not be specified in the following modes:
  - Cutter radius compensation mode
  - Three-dimensional offset mode
  - Geometry enlargement/reduction mode
  - Coordinate system parallel shift/rotation mode

[Supplement]
Axis feed commands specified immediately after G15 must be specified in the absolute mode.
6-2. Cutter Radius Compensation (G40, G41, G42)

6-2-1. Cutter Radius Compensation Function

[Function]
The cutter radius compensation function automatically compensates for the cutter radius. Programming the geometry of a workpiece as it is will not result in a correct final product because the size (diameter) of the tool is not taken into consideration. It would, however, be extremely complicated and difficult to develop a program which takes the tool diameter into account. This problem may be solved by a function called cutter radius compensation which automatically compensates for the tool diameter. If the cutter radius compensation function is used for programming, the correctly offset tool center path is automatically generated by programming the tool path along the geometry of workpiece to be machined.

[Programming format]

G17 G41 (G42) Xp__ Yp__ D__
G18 G41 (G42) Zp__Xp__ D__
G19 G41 (G42) Yp__Zp__ D__

G40: Cancel cutter radius compensation (The mode automatically selected when the power is turned ON.)
For details, refer to “6-2-4. Tool Movement when Cutter Radius Compensation is Canceled”.
G41: Cutting at left (Offset to the left side as seen from the tool moving direction; downward cutting)
For details, refer to “6-2-5. Changing Compensation Direction in Cutter Radius Compensation Mode”.
G42: Cutting at right (Offset to the right side as seen from the direction of tool motion; upward cutting)
The cutter radius compensation mode is set when either G41 or G42 is specified and this mode is canceled by G40. For details, refer to “6-2-5. Changing Compensation Direction in Cutter Radius Compensation Mode”.
G17: Xp-Yp plane selection Select the plane in the same manner as in the G02 or G03 mode.
G18: Zp-Xp plane selection Select the plane in the same manner as in the G02 or G03 mode.
G19: Yp-Zp plane selection Select the plane in the same manner as in the G02 or G03 mode.
D**: Cutter radius compensation number. (For details, refer to “6-2-7. Notes on Cutter Radius Compensation”.)

[Supplement]

- The explanation below assumes G17 (Xp-Yp plane), which is automatically set when power is turned ON. For the Zp-Yp plane and the Yp-Zp plane, the same explanation applies.
- Entry to the cutter radius compensation mode is allowed only in the G00 or G01 mode. An alarm occurs if the cutter radius compensation mode is called in other modes.
- The mode is changed to the cutter radius compensation mode in the first block that contains a command that actually causes axis movement after the designation of the cutter radius compensation command.
# 7-1. Table of Fixed Cycle Functions

<table>
<thead>
<tr>
<th>G Code</th>
<th>Function</th>
<th>Spindle Rotation at Positioning Point</th>
<th>Hole Machining Operation</th>
<th>Operation at Hole Bottom Level</th>
<th>Retraction Operation</th>
<th>Spindle Rotation at Return Level</th>
</tr>
</thead>
<tbody>
<tr>
<td>G71</td>
<td>Specifies the return level</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>G73</td>
<td>High speed deep hole drilling</td>
<td>CW</td>
<td>Pecking feed</td>
<td>Dwell</td>
<td>Rapid feed</td>
<td>CW</td>
</tr>
<tr>
<td>G74</td>
<td>Reverse tapping</td>
<td>CCW</td>
<td>Cutting feed</td>
<td>CW after dwell</td>
<td>Cutting feed</td>
<td>CW after dwell</td>
</tr>
<tr>
<td>G76</td>
<td>Fine boring</td>
<td>CW</td>
<td>Cutting feed</td>
<td>After the dwell, the tool bit is moved away from the bored surface. Then, the spindle stops at a specified position and shifts to the direction opposite that of the tool bit</td>
<td>Rapid feed</td>
<td>CCW after shifting to tool bit direction</td>
</tr>
<tr>
<td>G80</td>
<td>Cancel fixed cycle mode</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>G81</td>
<td>Drilling</td>
<td>CW</td>
<td>Cutting feed</td>
<td>Dwell</td>
<td>Rapid feed</td>
<td>CW</td>
</tr>
<tr>
<td>G82</td>
<td>Drilling</td>
<td>CW</td>
<td>Cutting feed</td>
<td>Dwell</td>
<td>Rapid feed</td>
<td>CW</td>
</tr>
<tr>
<td>G83</td>
<td>Deep hole drilling</td>
<td>CW</td>
<td>Pecking feed</td>
<td>Dwell</td>
<td>Rapid feed</td>
<td>CW</td>
</tr>
<tr>
<td>G84</td>
<td>Tapping</td>
<td>CW</td>
<td>Cutting feed</td>
<td>CCW after dwell</td>
<td>Cutting feed</td>
<td>CW after dwell</td>
</tr>
<tr>
<td>G85</td>
<td>Boring</td>
<td>CW</td>
<td>Cutting feed</td>
<td>Dwell</td>
<td>Cutting feed</td>
<td>CW</td>
</tr>
<tr>
<td>G86</td>
<td>Boring</td>
<td>CW</td>
<td>Cutting feed</td>
<td>Stop after dwell</td>
<td>Rapid feed</td>
<td>CW</td>
</tr>
<tr>
<td>G87</td>
<td>Back boring</td>
<td>(*)</td>
<td>Cutting feed</td>
<td>After the dwell, the tool bit is moved away from the bored surface. Then, the spindle stops at a specified position and shifts to the direction opposite that of the tool bit</td>
<td>Rapid feed</td>
<td>CW after shifting to tool bit direction</td>
</tr>
</tbody>
</table>

(*) After orientation, the spindle shifts to the direction opposite that of the tool bit and moves up to level R at a rapid feedrate. After shifting to the tool bit direction, the spindle rotates forward.

M codes used to select the return level:

- M52  Return to the upper limit level
- M53  Return to the specified point level set by G71
- M54  Return to the point R level
7-5. High Speed Deep Hole Drilling Cycle (G73)

[Programming format]
G73 X__Y__Z__R__P__Q__F__

Machining Sequence

(1) Positioning along the X- and Y-axis at a rapid feedrate

(2) Positioning to the point R level at a rapid feedrate

(3) Drilling by the pecking amount specified by Q at a cutting feedrate and with the spindle rotating in the forward direction

(4) Cutting tool retraction by “d” at a rapid feedrate.
   Set the retraction amount “d” at retraction in G73 CYCLE (HIGH-SPEED DEEP HOLE) OR G83 CYCLE (DEEP HOLE) WITH I, J COMMAND of the NC optional parameter (fixed cycle).

(5) Drilling to the point Z level by repeating steps (3) and (4)

(6) Dwelling at the point Z level for P seconds

(7) Returning to the return-point level at a rapid feedrate
7-6. Reverse Tapping Cycle (G74)

[Programming format]
G74 X__Y__Z__R__P__Q__F__

Machining Sequence

(1) Positioning along the X- and Y-axis at a rapid feedrate

(2) Positioning to the point R level at a rapid feedrate

(3) Tapping to the point Z level at the specified cutting feedrate with the spindle rotating in the CCW direction.

(4) Dwelling at the point Z level for P seconds, then reversal of the spindle rotating direction to the CW direction.

(5) Returning to the point R level at a cutting feedrate

(6) Dwelling at the point R level for Q seconds, then reversal of the spindle rotating direction back to the CCW direction.

(7) Returning to the return-point level at a rapid feedrate.

[Details]

• Dwell is not executed if a P and/or Q value is not specified.
The units of P and Q values are the same as used for the G04 mode dwell command.

• A feed override is disregarded during reverse tapping operation.

• If the SLIDE HOLD button is pressed during the return from the point Z level to the point R level, the cycle stops after the point R level is reached.

• If positioning to the next tapping point is executed at the point R level after the start of the spindle counterclockwise rotation but before the tapping tool is completely disengaged from the workpiece, enter a dwell at this level by specifying Q.

• Both the cutting feedrate override and the spindle speed override value are fixed at 100%. A rapid feed override can be set.
7-7. Fine Boring (G76)

[Programming format]
G76 X__Y__Z__R__Q__(I__J__) P__F__

Machining Sequence

(1) Positioning along the X- and Y-axis at a rapid feedrate

(2) Positioning to the point R level at a rapid feedrate

(3) Boring to the point Z level at the specified cutting feedrate with the spindle rotating in the forward direction

(4) Dwelling at the point Z level for P seconds, retracting by the amount set at SHIFT DIRECTION AND AXIS in G76, G87 of NC optional parameter (FIXED CYCLE), then spindle stop in the orientation position. After that, the tool shifts by the shift amount, Q, to the direction the tool bit moves away from the machined workpiece inner surface.

(5) Returning to the return point level at a rapid feedrate

(6) Tool shifts back in the bit direction by the shift amount, Q, then the spindle starts rotating in the clockwise direction.
[Details]

- Retraction amount at the point Z level
  The amount the Z-axis retracts upward from the point Z level is set at SHIFT DIRECTION AND AXIS IN G76, G87 of the NC optional parameter (FIXED CYCLE).

- Shift amount
  a. Q is used to specify the shift amount if the cycle axis is fixed as the Z-axis by the setting at SHIFT DIRECTION AND AXIS IN G76, G87 of the NC optional parameter (FIXED CYCLE). The value set must always be positive. The direction for shift motion, +X, -X, +Y, or -Y, should be set using a parameter beforehand. Note that a Q value is modal data and address Q is also used in the G73 and G83 cycles. A Q value is given priority over I and J values.
  b. I and J are used to specify the shift amount when the plane is selected using G17, G18, or G19. The relationship between the plane selecting G code and the addresses to be used is shown below.
     
     | G17   | I, J |
     | G18   | K, I |
     | G19   | J, K |

     For addresses I, J, and K, all values are set as incremental values. The shift direction is always defined in the machine coordinate system.
  c. If the shift amount is not specified by Q, or I and J, an alarm occurs.

7-8. Fixed Cycle Cancel (G80)

[Function]
G80 cancels a fixed cycle mode (G73, G74, G76, G81 to G87, and G89). When G80 is executed, all hole machining defining commands including point R and point Z are canceled and the interpolation mode (G00 to G03, G60) valid before the fixed cycle mode was called is restored. At the same time, the M05 code (spindle stop command) is generated. If G00 or G01 is specified preceding the G80 block, however, M05 is not generated.

- Example program that does not generate M05
  
  G81 X__Y__Z__R__F__
  G00 X__Y__
  G80

- Example program that generates M05
  
  G81 X__Y__Z__R__F__
  G80
  G00 X__Y__

[Details]

- Fixed cycle modes are also canceled if an interpolation mode call G code (G00, G01, G02, G03), with the exception of G60, is specified.

- If axis movement commands are specified with G80 in the same block, the fixed cycle mode is canceled first and the axis movement commands are executed after that.
7-9. Drilling Cycle (G81, G82)

[Programming format]

G81  X__Y__Z__R__P__F__
G82  X__Y__Z__R__P__F__

G81 and G82 are used in the same manner.

**Machining Sequence**

1. Positioning along the X- and Y-axis at a rapid feedrate
2. Positioning to the point R level at a rapid feedrate
3. Drilling to the point Z level at the specified cutting feedrate with the spindle rotating in the clockwise direction
4. Dwelling at the point Z level for P seconds.
5. Returning to the return point level at a rapid feedrate
7-10. Deep Hole Drilling Cycle (G83)

[Programming format]
G83 X__Y__Z__R__Q__(I__J__) P__F__

- Programming using Q

- Programming using I and J

If a Q value is programmed in the same block as I and J values, the Q value will be given priority.
[Setting values]
Retraction amount d1:
Set at RETRACTION POSITIONING FROM LEVEL ‘R’ TO WORK IN G83 CYCLE (DEEP HOLE) of the NC optional parameter (fixed cycle).
Retraction amount d2:
Set at RETRACTION IN G73 CYCLE (HIGH-SPEED DEEP HOLE) OR G83 CYCLE (DEEP HOLE) WITH I, J COMMAND of the NC optional parameter (fixed cycle).

Machining Sequence

(1) Programming using Q

a. Positioning along the X- and Y-axis at a rapid feedrate
b. Positioning to the point R level at a rapid feedrate
c. Drilling by pecking amount specified by Q at the specified cutting feedrate with the spindle rotating in the forward direction
d. Returning to the point R level at a rapid feedrate
e. In the second and later in-feed operations: Positioning at a level “d1” above the previously machined depth at a rapid feedrate and drilling by “Q + d1”.
f. Returning to the point R level at a rapid feedrate
g. Drilling to the point Z level by repeating steps e). and f).

(2) Programming using I and J

a. Positioning along the X- and Y-axis at a rapid feedrate
b. Positioning to the point R level at a rapid feedrate
c. Drilling by pecking amount I then retracting by d2. After that, drilling by “I + d2”. The pecking and retraction cycle is repeated until depth J (tool extraction depth) is reached.
d. Returning to the point R level at a rapid feedrate
e. Positioning at a level “d1” above the previously machined depth at a rapid feedrate and drilling by “I + d1”. Then step c). is repeated to machine by depth J.
f. Returning to the point R level at a rapid feedrate
g. Drilling to the point Z level by repeating steps e). and f).
h. Dwelling at the point Z level for P seconds.
i. Returning to the return point level at a rapid feedrate.
[Details]
According to the value of I, J, cutting is performed as follows. If either I or J is specified, the other one is considered as "0" is specified.

- No Q designation

J = 0  G73 cycle is executed, using pecking amount I.
I ≥ J  G83 cycle is executed, using pecking amount J.
I < J  Operation steps c) to g), explained in item (2) above, are executed.
I = 0  An alarm occurs. (Alarm “Q”)

- Q designated with I and J in the same block
  Operation steps c) to g), explained in (1) above, are executed disregarding I and J values.

7-11. Tapping Cycle (G84)

[Programming format]
G84 X__Y__Z__R__ (P__) (Q__) F__

Machining Sequence

(1) Positioning along the X- and Y-axis at a rapid feedrate
(2) Positioning to the point R level at a rapid feedrate
(3) Tapping to the point Z level at the specified cutting feedrate with the spindle rotating in the clockwise direction
(4) Dwelling at the point Z level for P seconds, then reversal of the spindle rotating direction to the counterclockwise direction
(5) Returning to the point R level at a cutting feedrate
(6) Dwelling at the point R level for Q seconds, then reversal of the spindle rotating direction back to the clockwise direction
(7) Returning to the return-point level at a rapid feedrate
[Details]

- Dwell is not executed if a P and/or Q value is not specified. Units of P and Q values are the same as used for the G04 mode dwell command.
- A feed override is disregarded during reverse tapping operation.
- If the SLIDE HOLD button is pressed during the return from the point Z level to the point R level, the cycle stops after the point R level is reached.
- If positioning to the next tapping point is executed at the point R level after the start of the spindle clockwise rotation but before the tapping tool is completely disengaged from the workpiece, enter a dwell at this level by specifying Q.
- Both the cutting feedrate override and the spindle speed override value are fixed at 100%. A rapid feed override can be set.

7-12. Boring Cycle (G85, G89)

[Programming format]
G85 (G89) X__Y__Z__R__P__F__FA=__

Machining Sequence

1. Positioning along the X- and Y-axis at a rapid feedrate
2. Positioning to the point R level at a rapid feedrate
3. Boring to the point Z level at the specified cutting feedrate with the spindle rotating in the clockwise direction
4. Dwelling at the point Z level for P seconds
5. Returning to the point R level at a return speed
6. Returning to the return point level at a rapid feedrate

- If FA is not specified, F is applied for the return operation from the point R level to the return point level.
7-13. Boring Cycle (G86)

[Programming format]
G86 X__Y__Z__R__P__F__

Machining Sequence
(1) Positioning along the X- and Y-axis at a rapid feedrate
(2) Positioning to the point R level at a rapid feedrate
(3) Boring to the point Z level at the specified cutting feedrate with the spindle rotating in the clockwise direction
(4) Dwelling at the point Z level for P seconds. Then, the spindle stops rotating.
(5) Returning to the return point level at a rapid feedrate
(6) Spindle rotation start in the clockwise direction.

[Details]
The difference between the G86 boring cycle and the G81/G82 drilling cycles is that the spindle stops at the hole bottom level in the G86 cycle.

- If the distance from the return-point level where the spindle restarts rotating in the clockwise direction to the point R level of the next hole is short, the spindle may not reach the commanded speed. Therefore, it is recommended to enter a dwell command (G04) with an appropriate dwell time before the cycle for boring the next point starts.
7-14. Back Boring Cycle (G87)

Note that this cycle differs somewhat from other fixed cycles.

[Programming format]
G87 X__Y__Z__R__Q__ (I__J__) P__F__

[Setting values]
Retraction amount at the point Z level: Set at retraction for G76/G87 (fine boring/back boring) before orientation of NC optional parameter (fixed cycle).

The return point level may be specified from the following three levels using an M code.

- M52 Upper limit level
- M53 Specified point level (to be set in advance with G71)
- M54 Start point level

Machining Sequence

1. Positioning along the X- and Y-axis at a rapid feedrate. Then, the spindle stops (spindle orientation).
2. Shifting by shift amount Q in the direction opposite the tool bit direction
3. Positioning to the point R level at a rapid feedrate
4. At the point R level, the tool shifts back by the shift amount. Then, the spindle starts rotating in the clockwise direction.
5. Boring to the point Z level in the position direction of the Z-axis at the specified cutting feedrate
6. Dwelling at the point Z level for P seconds. After that the Z-axis moves back by the amount set for the parameter and the spindle stops (spindle orientation). The tool shifts by the shift amount Q in the direction opposite the tool bit direction.
7. Returning to the return point level at a rapid feedrate
8. Shifting back by shift amount Q
SPEEDS AND FEEDS

SPEEDS

Surface speed is a term used to describe the machinability of a material. It refers to the amount of material that can be cut in one minute. Material that can be machined at a rate of 500 feet per minute is more “machinable” than material that can be machined at a rate of 250 feet per minute.

The machinability rating of material can be found in tooling catalogs, the “Machinist Handbook”, and online. To calculate the RPM of a tool (for a machining center) or material to be turned (lathe), use these simple formulas.

Inch (imperial):

\[ \text{RPM} = \frac{3.82}{\text{DIA}} \times \text{the rating} \]

or

\[ \text{RPM} = \frac{3.82}{\text{DIA}} \times \text{SFM} \quad (\text{SFM} = \text{surface feet per minute}) \]

An example is a .75” drill, in material with a machinability rating of 500SFM.

\[ \text{RPM} = \frac{3.82}{.75} \times 500 \]

\[ \text{RPM} = (5.09) \times 500 \]

\[ \text{RPM} = 2546 \]

Millimeter (metric):

\[ \text{RPM} = \frac{318}{\text{DIA}} \times \text{the rating} \]

or

\[ \text{RPM} = \frac{318}{\text{DIA}} \times \text{SMM} \quad (\text{SMM} = \text{surface meters per minute}) \]

An example is a 19mm drill, in material with a machinability rating of 150SMM.

\[ \text{RPM} = \frac{318}{19} \times 150 \]

\[ \text{RPM} = (16.74) \times 150 \]

\[ \text{RPM} = 2510 \]

RPMs that are too high generally cause the tool to generate too much heat, and the cutting edge is burned or deformed by the excessive heat.
FEEDS

Most machining centers are programmed in inches per minute (IPM). Tools are designed to feed into the material at a distance per each revolution of the tool (Feed per Rev), or distance per tooth.

Inch (imperial):
An example is a four flute end mill, intended to feed at .0015” per flute, cutting at 2200rpm.

\[
\text{IPM} = \text{RPM} \times \text{IPR}
\]

\[
\text{IPM} = 2200 \times (4 \times .0015”)
\]

\[
\text{IPM} = 2200 \times .006”
\]

\[
\text{IPM} = 13.2
\]

F13.2

Millimeter (metric):
An example is a four flute end mill, intended to feed at .040mm per flute, cutting at 3000rpm.

\[
\text{MPM (MM per MIN)} = \text{RPM} \times \text{MMPR (MM per REV)}
\]

\[
\text{MPM} = 3000 \times (4 \times .040mm)
\]

\[
\text{MPM} = 3000 \times .16mm
\]

\[
\text{MPM} = 480
\]

F480.0

Feed rates that are too high generally cause the tool to wear prematurely.

Machining centers can also be programmed in the feed per revolution mode (G95).
T1  1/2 x 45DEG SPOT DRILL  150SFM  .005CL
T2  5/16 DRILL  80SFM  .003CL
T3  .201 DRILL FOR 1/4-20 TAP  100SFM  .002CL
T4  1/4-20 TAP  50SFM
T5  27/64 DRILL FOR 1/2-13 TAP  100SFM  .004CL
T6  1/2-13 TAP  25SFM

1/2-13 Thru

5/16 Thru on a 2.00 BC
4 Places as Shown

1/4-20 .500 deep Typ(4)

2.80 250

4.00

5.50

1.50

3.5000

1.00 1.00 1.00

7.00

2.00
1/2-13 Thru 6 places

R 0.500 typ(4)

.250 dia .750 deep on a 2.500 bolt circle 6 places equally spaced

Hartwig Inc.

TOLERANCES
X 4+/-.03
XX 4+/-.05
XXX 4+/-.075
XXXX 4+/-.100

ANODES 4+/-.05 DEE

DRAWN: 1-1-06
CHECKED: 1-1-06
REV: 1/0

JOB: 20
DRAW: RU
Use a 1/2 E.M.
Profile ID
BASIC OKUMA MACHINING CENTER PROGRAMMING CLASS

Hartwig, Inc. desires to provide the best possible training experience possible. To do that we need to be graded on what we provide. Please take a minute to rate us 1 to 5.

1 POOR  
2 BELOW AVERAGE  
3 AVERAGE  
4 VERY GOOD  
5 EXCELLENT

In your daily job responsibilities, please check all that apply:

___ Tooling Selection  
___ Process Editing  
___ Program Creation  
___ Production/Machine Operation  
___ Machine Setup  
___ Inspection

Other: (please describe) __________________________________________________________

1. What specific thing did you learn that will help you the most?

____________________________________________________________________________

2. Rate how the information was covered. (1-5) ____________________________

3. Were the instructors prepared? (1-5) ____________________________

4. Would you call these instructors in the future for advice? (1-5) _____________

5. Would you recommend this or a similar class? (1-5) _________________

6. Please rate your overall satisfaction with the class (1-5) _______________

7. What other classes would like Hartwig, Inc. to offer?

____________________________________________________________________________

8. Any other comments or suggestions do you have to improve our classes?

____________________________________________________________________________
9. Check any other items you would like information on.

- Customized training at your site
- Upcoming Classes
- Machine Preventative Maintenance Programs
- Turnkeys

Web site
- Contract Programming
- Upcoming Shows

10. Information or literature on:

- CNC Lathes
- CNC Mills

Grinders
- Coordinate Measuring Machines

11. Who else; that you know, would benefit from this class?

 Company ___________________ Name ___________________
 Company ___________________ Name ___________________

Your Name ________________________________
Your Email ________________________________
Company Name _______________________________

Thanks very much for attending this event.