Sinumerik 808D Programming and Operating Procedures for Turning
Version 2013-01
Basic knowledge of programming for turning is required, before operating of a machine!
<table>
<thead>
<tr>
<th>Notes</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Absolute value and incremental value</td>
<td>34</td>
</tr>
<tr>
<td>Editing part program</td>
<td>33</td>
</tr>
<tr>
<td>Executing function M</td>
<td>21</td>
</tr>
<tr>
<td>Calculator</td>
<td>89</td>
</tr>
<tr>
<td>Time change</td>
<td>79</td>
</tr>
<tr>
<td>Contour editor</td>
<td>41</td>
</tr>
<tr>
<td>Creating and measuring tools</td>
<td>13</td>
</tr>
<tr>
<td>Creating zero offsets</td>
<td>26</td>
</tr>
<tr>
<td>Cycles</td>
<td>41</td>
</tr>
<tr>
<td>Dry run</td>
<td>64</td>
</tr>
<tr>
<td>Jogging spindle</td>
<td>21</td>
</tr>
<tr>
<td>Help</td>
<td>77</td>
</tr>
<tr>
<td>List of programming functions</td>
<td>113</td>
</tr>
<tr>
<td>Tool wear</td>
<td>69</td>
</tr>
<tr>
<td>Manual start spindle</td>
<td>25</td>
</tr>
<tr>
<td>Manual tool change</td>
<td>16</td>
</tr>
<tr>
<td>MDA</td>
<td>83</td>
</tr>
<tr>
<td>Moving axis with handwheel</td>
<td>17</td>
</tr>
<tr>
<td>Part programming</td>
<td>31</td>
</tr>
<tr>
<td>Protection levels</td>
<td>7</td>
</tr>
<tr>
<td>Program execution</td>
<td>63</td>
</tr>
<tr>
<td>Breakpoint search</td>
<td>71</td>
</tr>
<tr>
<td>Reference point</td>
<td>10</td>
</tr>
<tr>
<td>RS232c and USB</td>
<td>75</td>
</tr>
<tr>
<td>Saving data</td>
<td>79</td>
</tr>
<tr>
<td>Simulation</td>
<td>59</td>
</tr>
<tr>
<td>Subprograms</td>
<td>84</td>
</tr>
<tr>
<td>Sample programs</td>
<td>93</td>
</tr>
<tr>
<td>Timers/counters</td>
<td>67</td>
</tr>
<tr>
<td>ISO mode</td>
<td>101</td>
</tr>
</tbody>
</table>
Unit Description

This unit describes the 808D PPU and MCP functionality, the coordinate system of a turning machine and how to enter passwords to access the system.

Unit Content

PPU Function of keyboard

MCP mode Changing

MCP Moving axis

MCP OEM keys

User interface

Machine coordinate system

Passwords

PPU

The 808D panel processing unit (PPU) is used to input data to the CNC and to navigate to operating areas of the system.

MCP mode Changing

Menu navigation

Operating area navigation

MCP mode Changing

Mode Navigation

The 808D machine control panel (MCP) is used to select the machine operating mode: JOG - MDA - AUTO
The 808D machine control panel (MCP) is used to control manual operation of the axis. The machine can be moved with the appropriate keys.

The 808D machine control panel (MCP) is used to control OEM machine functions. The machine functions can be activated with the appropriate keys.

808D (PPU) has eight vertical softkeys (abbr. SKs) on the right of the screen. These SKs can be activated with the corresponding button (located on the right).

808D (PPU) has eight horizontal SKs on the bottom of the screen. These SKs can be activated with the corresponding button (located below).
The Sinumerik 808D uses a coordinate system which is derived from the DIN 66217 standard. The system is an international standard and ensures compatibility between machines and coordinate programming. The primary function of the coordinate system is to ensure that the tool length and tool radius are calculated correctly in the respective axis.

**The machine zero point** (M) is determined by the machine manufacturer and cannot be changed.

**The workpiece zero point** (W) is the origin of the workpiece coordinate system.

**The reference point** (R) is used for synchronizing the measuring system.

**The tool holder reference point** (F) is used to determine the tool offset.

Passwords at the control are used to set the user’s right to access the system. Tasks such as “Basic Operating”, “Advanced Operating” and commissioning functions all depend on the passwords.

- **No password** Machine operator
- **Customer’s password** Advanced operator
- **Manufacturer’s password** OEM engineer

### Changing password

#### Step 1

1. The service mode is opened with the appropriate key combination.
2. In the service mode, the password can be activated and deactivated.

#### Step 2

1. Enter customer password
2. Change customer password
3. Delete customer password

**Usually the machine operator does not need to change the password.**
This unit describes how to switch the machine on and reference it.

Please note the explicit switching on rules as specified by the machine manufacturer.

**Step 1**

Turn on the main switch of the machine. The main switch is usually at the rear of the machine.

**Step 2**

Make sure you perform the following operation!

Release all the EMERGENCY STOP buttons on the machine!
After power on, the machine must first be referenced!

Step 1

After power on, the machine will be in the reference point approach mode (default).

If the axis is not referenced, the non-referenced symbol (circle) is displayed between the axis identifier and the value.

Step 2

The axes are referenced with the corresponding axis traversing keys.

The traversing direction keys are specified by the machine manufacturer.

Step 3

After completing the referencing procedure for all axes, the referenced symbol is displayed next to the axis identifier.

After returning to JOG mode, use the axis traversing keys to move the machine manually.

Now the machine can be operated in JOG mode.

During normal operation (JOG), the reference symbol is not shown on the screen.

End
Unit Description

This unit describes how to create and set up tools.

Unit Content

Create tool

- Tool edge position code
- Create tool edge
- Load tool into active position
- JOG spindle
- Measure tool
- Start spindle
- Execute M function
- Test tool offset results

Create tool

- Press "Offset" on the PPU.

Warning: A tool must have been created and measured before executing the program.

Step 1
Please make sure the system is in JOG mode.

Press "Offset" on the PPU.

Press the "Tool list" SK on the PPU.
Step 2

The range of tool numbers which can be created by this system is 1 ~32000. The machine can be loaded with a maximum of 64 tools / 128 tool edges.

Press the “New tool” SK on the PPU.

Select the type of required tool.

Enter “1” at “Tool No.”

Enter “3” at “Edge position”.

The correct “Edge position” selection directly determines the correct tool compensation which will be described in the next unit.

Press the “OK” SK on the PPU

Enter the “Radius” or “Tip width” as required.

Press the “Input” button on the PPU

Principle of correct tool edge position code selection: Select the corresponding tool edge position code according to actual tool point direction!

Observe the relationship between the tool point direction and the positive direction of the X axis and the Z axis.

Find the corresponding position relationship in the figure below and enter the number in “Edge position”; the red coordinate in the purple circle is the selected position code.

Note: Not every tool has eight position codes. All the options are shown above.

As to “turning tool” and “grooving tool”, 808D provide 4 edges (#1~4) which are shown on the left figure.

As to “drilling tool” and “tapping tool”, 808D provide only 1 edge (#7) which is also shown on the left figure.

Note that the tool tip direction here is the direction after the correct tool offset, not only the direction in tool loading. And the correctness of tool edge position code directly affects the correctness of the tool tip radius compensation!

The tool edge position code can also be changed in the position showed in the figure.
SEQUENCE

**Create tool edge**

A tool must have been created and selected before creating a tool edge.

**Step 1**

- Use "D" code to represent the tool edge. The system activates the No.1 tool edge as default at the beginning.
- Press the "Offset" key on the PPU.
- Press the "Edges" SK on the PPU.
- Use direction keys to select the tool which needs to add a tool edge.
- Use direction keys to select the tool which needs to add a tool edge.
- Press the "New edge" SK on the PPU.
- Press the "Tool list" SK on the PPU.

Example

Common tool edge position code choices are as follows:

![Tool edge position codes](image)

![Tool edge position codes](image)

**Tool Setup**

Press the "Tool list" SK on the PPU.

Press the "New edge" SK on the PPU.
### Tool Setup

**SEQUENCE**

**Step 2**
A new tool edge can be added in this way and different lengths and radii can be entered as required.

The red circle shows the actual active tool and tool edge, the purple circle shows how many tool edges have been created and the related data for each tool edge.

A maximum of nine tool edges can be created for each tool! Different tool lengths and radii can be saved in different tool edges as required. Please select the right tool edge for machining according to requirement!

![Tool Setup Diagram]

- Load tool into active position
- A tool must have been created in the system before it can be loaded into the active position.

1. Press the “Machine” key on the PPU.
2. Press the “T.S.M” SK on the PPU.
3. Enter tool number “1” in “T”.
4. Press “CYCLE START” on the MCP.
5. Load tool into active position.
6. A tool must have been created in the system before it can be loaded into the active position.
7. Press the “JOG” key on the MCP.
8. Step 2
   - The red circle shows the actual active tool and tool edge, the purple circle shows how many tool edges have been created and the related data for each tool edge.
   - A new tool edge can be added in this way and different lengths and radii can be entered as required.
   - Please select the right tool edge for machining according to requirement!

---

Programming and Operating — Turning

Page 16
Move machine with handwheel

Make sure there is no obstruction when moving the tool to avoid a crash.

A handwheel can control the axis motion instead of the “JOG” button.

Press the “Machine” key on the PPU.

Press the “Handwheel” key on the MCP.

Select the axis you want to move with the appropriate keys on the MCP

Select the required override increment according to the buttons on the right (this selection fits all axes)

The override increment is “0.001 mm”

The override increment is “0.010 mm”

The override increment is “0.100 mm”

The selected axis can now be moved with the handwheel.

Press “JOG” on the MCP to end the “Handwheel” function.

Notes: if set MD14512[16]=80, the system will deactivate the function of MCP for selecting the axis of handwheel, the user will have to activate “Handwheel” function with PPU softkey

Select the required axis on the right of the PPU; the selected axis is shown with a ✓

Under “WCS” or “MCS”, a handwheel will be shown beside the axis symbols, representing that the axis can be moved using handwheel.
Tool Setup

SEQUENCE

Start spindle

A tool must have been loaded and rotated to the position.

Start the spindle before adjusting tools as follows:

1. Press the “Machine” key on the PPU.
2. Press the “JOG” key on the MCP.
3. Press the “T.S.M” SK on the PPU.
4. Enter “500” at “Spindle speed”.
5. Select “M3” using the “Select” key on the PPU.
6. Press the “CYCLE START” key on the MCP.

Press the “Back” SK on the PPU.

Press “Reset” on the MCP to stop the spindle rotation.

Press the “Machine” key on the PPU.

Press the “JOG” key on the MCP.

Press the “Meas. tool” SK on the PPU.

Press the “Measure X” SK on the PPU.

Measure tool

A tool must have been created and loaded before it can be measured!

Step 1 Measure length: X

1. Press the “Machine” key on the PPU.
2. Press the “JOG” key on the MCP.
3. Press the “Meas. tool” SK on the PPU.
4. Press the “Measure X” SK on the PPU.
Use the traversing keys on the MCP to move the axis to the adjusted position.

**Note:** “X=0” or “Z=0” in the workpiece coordinate system is shown as “X0” / “Z0” in the following text.

Use the “Handwheel” key on the MCP and select a suitable feedrate override to move the tool to X0.

Move directly to zero point.

Enter 50 in “ø” (this is the diameter of the workpiece)

Press the “Set length X” SK on the PPU.
SEQUENCE

Step 2  Set length:  \( Z \)

Press the “Set length Z” SK on the PPU.

Use the traversing keys on the MCP to move the axis to the adjusted position.

Enter “0” in “Z0”
(this is the distance between the tool point and the zero point)

Move directly to zero point

Press the “Set length Z” SK on the PPU.

Press the “Back” SK on the PPU.

Use the “Handwheel” key on the MCP and select a suitable feedrate override to move the tool to \( Z0 \).
A tool must have been loaded and rotated to the position!

Press the “Machine” key on the PPU.

Press the “JOG” key on the MCP.

Press the spindle direction key on the MCP to start/stop the spindle.

Press “Spindle left” on the MCP to start the spindle in the counter-clockwise direction.

Press “Spindle stop” on the MCP to stop the spindle.

Press “Spindle right” on the MCP to start the spindle in the clockwise direction.

Execute M function

Please make sure all the machine axes are in safe positions before executing the M function!

Press the “Machine” key on the PPU.

Press the “JOG” key on the MCP.

Press the spindle direction key on the MCP to start/stop the spindle.

Press “Spindle left” on the MCP to start the spindle in the counter-clockwise direction.

Press “Spindle stop” on the MCP to stop the spindle.

Press “Spindle right” on the MCP to start the spindle in the clockwise direction.

Press the “Machine” key on the PPU.

Press the “T.S.M” SK on the PPU.

Use the direction key to move the highlighted cursor to “Other M function” and enter “8”. This will start the coolant.

Press the “Machine” key on the PPU.

Press the “T.S.M” SK on the PPU.

Use the direction key to move the highlighted cursor to “Other M function” and enter “8”. This will start the coolant.

You can see that the coolant function key on the MCP is active.

Press the “Reset” key on the MCP to stop the coolant function.

Press the “Back” SK on the PPU.
The tool setup and workpiece setup must have been performed correctly so that it can be tested as follows!

In order to ensure the machine safety and correctness, the results of the tool offset should be tested appropriately.

Press the “Machine” key on the PPU.

Press the “MDA” key on the MCP.

Press the “Delete file” SK on the PPU.

Enter the test program recommended on the right (can also be customized).

Press the “ROV” key to ensure the “ROV” function is active (lit up).

Note: The ROV function activates the feedrate override switch under the G00 function.

Press “CYCLE START” on the MCP.

Increase the feedrate override gradually to avoid accidents caused by an axis moving too fast and observe whether the axis moves to the set position.

Make sure the feedrate override on the MCP is at 0%!
Unit Description
This unit describes how to set the workpiece offset and test the tool results.

Unit Content
- Manual start spindle
- Create workpiece offset
- Test tool offset results
- End

SEQUENCE

A tool must be loaded and rotated to the position.

Before measuring, the spindle can be started as follows:

1. Press the “Machine” key on the PPU.
2. Press the “JOG” key on the MCP.
3. Press the “T.S.M” SK on the PPU.
4. Enter “500” in “Spindle speed” on the PPU.
5. Select “M3” as the “Spindle direction” using the “Select” key on the PPU.
6. Press “CYCLE START” on the MCP.

Press “CYCLE START” on the MCP.
Workpiece Setup

SEQUENCE

Press the “Reset” key on the MCP to stop the spindle rotation.

Press the “Back” SK on the PPU.

Create workpiece offset

A tool must have been created and measured before it can be used to set the workpiece offset.

Make sure the active tool is the measured tool!

Press the “Machine” key on the PPU.

Press the “JOG” key on the MCP.

Press the “Offset” key on the PPU.

Press the “Work offset” SK on the PPU.

Press the “Meas.work.” SK on the PPU.
**Workpiece Setup**

**SEQUENCE**

**Step 2**

Using a tool that has a measured “Tool length”, move the tool to a known position on the workpiece. Using either JOG or Handwheel, scratch an edge and then calculate the zero point of the workpiece. The process of setting the zero point (“Z0”) is described below.

Press the SK on the PPU to select the required setting axis.

Press the axis traverse keys to move the tool to the required setting position in the Z axis.

Press the “Handwheel” key on the MCP to move the tool to the Z0 position on the workpiece.

Enter tool number “1” in “T”.

Set “Save in” as “G54” (or other offset).

Set “Distance” as “0”.

Press the “Set work offset” SK on the PPU.

Repeat the operations to set the “X” zero point.

Press the “Back” SK on the PPU after measuring.
The tool setup and workpiece setup must have been performed correctly so that it can be tested as follows!

In order to ensure the machine safety and correctness, the results of the tool offset should be tested appropriately.

Press the “MDA” key on the MCP.

Press the “Delete file” SK on the PPU.

Enter the test program recommended on the right.

Press the “ROV” key to ensure the “ROV” function is active (the function is activated when the light on the key is on).

Note: The ROV function activates the feedrate override switch under the G00 function.

Make sure the feedrate override on the MCP is at 0%!

Press “CYCLE START” on the MCP.

Increase the feedrate override gradually to avoid accidents caused by an axis moving too fast and observe whether the axis moves to the set position.
Create Part Program Part 1

Content

Unit Description
This unit describes how to create and edit a part program, and get to know the most important CNC commands required to produce a work-piece.

Unit Content

Program structure
Create program
Edit program
Imperial and Metric system
Definition of target position
Rapid motion
Tools and motion
Behavior at corners
Turning circles and arcs
Moving to a fixed position
Controlling the spindle
Setting a delay in the program
End

BASIC THEORY

A standard program structure is not needed but is recommended in order to provide clarity for the machine operator. Siemens recommends the following structure:

<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>N5</td>
<td>G17</td>
</tr>
<tr>
<td>N10</td>
<td>G90</td>
</tr>
<tr>
<td>N15</td>
<td>G54</td>
</tr>
<tr>
<td>N20</td>
<td>G71</td>
</tr>
<tr>
<td>N25</td>
<td>S5000 M3 G95 F0.3</td>
</tr>
<tr>
<td>N30</td>
<td>G00 X100 Z2</td>
</tr>
<tr>
<td>N35</td>
<td>G01 Z-5</td>
</tr>
<tr>
<td>N40</td>
<td>T2 D1</td>
</tr>
<tr>
<td>N45</td>
<td>S3000 M3 G95 F0.2</td>
</tr>
<tr>
<td>N50</td>
<td>G00 X99 Z2</td>
</tr>
<tr>
<td>N55</td>
<td>G01 Z-5</td>
</tr>
<tr>
<td>N60</td>
<td>X105</td>
</tr>
<tr>
<td>N65</td>
<td>G00 SUPA X300 Z50 D0</td>
</tr>
<tr>
<td>N70</td>
<td>T3 D1</td>
</tr>
<tr>
<td>N75</td>
<td>S3000 M3 G95 F0.2</td>
</tr>
<tr>
<td>N80</td>
<td>G00 X105 Z-25</td>
</tr>
<tr>
<td>N85</td>
<td>G01 X90</td>
</tr>
<tr>
<td>N90</td>
<td>X105</td>
</tr>
<tr>
<td>N95</td>
<td>G00 SUPA X300 Z50 D0</td>
</tr>
<tr>
<td>M30</td>
<td></td>
</tr>
</tbody>
</table>

Setting a delay in the program
End/stop position
Create Part Program

The following sequence should be followed to create a part program:

Step 1
Programs can be created with the “program manager”. You can select the “program manager” using the key located on the PPU.

Step 2
Select NC as the storage location for the program. Programs can only be created in the NC.

Step 3
Create a new program with the “New” SK on the right of the PPU.

Step 4
You can choose “New” or “New directory”.
Choose “New” to create a program.
Choose “New directory” to create a file.

Step 5
Now the program is open and can be edited.

After editing the system will save it automatically.
The program shown in the editor can be created and edited with the correct keys.

**G71**
With G71 at the header, the geometry data will be in the metric unit system, feedrate in the default metric system.

```
N5 G17 G90 G54 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X10 Z1
N25 G01 X-0.5 Z2
N30 Z2
N35 G00 X200 Z50
```

**G70**
With G70 at the header, the geometry data will be in the imperial (inches) unit system, the feedrate in the default metric system.

```
N5 G17 G90 G54 G70
N10 T1 D1
N15 S5000 M3 G95 F0.2
N20 G00 X10 Z0.2
N25 G01 X-0.2 Z0.2
N30 Z0.2
N35 G00 X10 Z10
```
G500
All absolute path data will be relative to this position. The position is written in the G500 (basic) zero offset.

Or

G54 G55 G56 G57 G58 G59
With G500 = 0, the offset for the workpiece can be stored in the G54 workpiece offset.

Or

G500 + G54
With G500 <= 0 and is activated, the value in G500 will be added to the value in G54.

G90
Absolute positioning; with G90 at the beginning of the program, the geometry data which follows will be interpreted relative to the active zero point in the program, usually with G54 or G500 or G500 + G54.

N5 G17 G90 G500 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 Z5
N35 G00 Z50 X100

G91
Relative positioning; with G91 you can add an incremental value (G91 defined data is the relative positioning using the present position as the start point). Finally you should change the program to absolute positioning with G90.

N5 G17 G90 G54 G70
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X3.93 Z0.196
N25 G01 G91 Z-0.787
N30 Z0.196
N35 G00 Z9.68 X10
Rapid motion

G00
When G00 is active in the program, the axis will traverse at the maximum axis speed in a straight line.

Tools and motion

T1 D1
With the "T" command the new tool can be selected, the "D" command is used to activate the tool length offset. M06 is also available for machines with automatic tool changer.

Feedrate

- The feedrate is defined in the program with "F". Two types of feedrate are available:
  1. Feed per minute → G94
  2. Feed per revolution of the spindle → G95

Spindle speed

- The spindle speed is defined with "S" S5000

Feed type

- "M3" and "M4" define the spindle direction with M3 and M4, clockwise/counter-clockwise respectively.

G01
When G01 is active in the program, the axis will traverse at the programmed feedrate in a straight line, according to the feedrate type defined by G94 or G95.
Activation/deactivation of the tool radius compensation when working on the part contour.

G41 / G42 and G40

With G41/G42, the radius compensation of the tool will be done in the direction of travel.

G41: Compensation to left.

G42: Compensation to right.

G40: Compensation of the radius can be deactivated.

Arrow indicates the direction of tool motion along the contour.
The circle radius shown in the example on the right can be produced with the specified part program code. When milling circles and arcs, you must define the circle center point and the distance between the start point / end point and the center point on the relative coordinate. When working in the XZ coordinate system, the interpolation parameters I and K are available.

Two common types of defining circles and arcs:
①: G02/G03 X_Z_I_K_
②: G02/G03 X_Z_CR=_

Arcs \( \leq 180^\circ \), CR is a positive number
Arcs \( > 180^\circ \), CR is a negative number

For the example:

N5 G17 G90 G500 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X0 Z2
N25 G01 Z0
N30 G42 X50
N45 G03 X75 Z-35 I-12 K-35
N50 G01 Z-130
N60 G40 X120 Z-140
N35 G00 X300 Z500

Note:
N45 can also be written as follows
N45 G03 X75 Z-35 CR=37

SP = start point of circle
CP = center point of circle
EP = end point of circle

G2 = define circle direction in traversing direction = G2 clockwise
G3 = define circle direction in traversing direction = G3 counter-clockwise
Moving to a fixed position

Using the code **G74**, the machine can move to the reference point automatically.

```
N5 G17 G90 G500 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 Z5
N35 G74 X=0 ; reference point
```

Using the code **G75**, the machine can move to the fixed position defined by machine supplier automatically.

```
N5 G17 G90 G500 G71
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 Z5
N35 G74 Z=0 ; reference point
N40 G75 X=0 ; fixed point
```

Controlling the spindle

The following functions can be used to influence the operation of the spindle:

- **M3** accelerate to programmed speed clockwise.
- **M4** accelerate to programmed speed counter-clockwise.
- **M5** spindle decelerate to stop.
- **M19** orient the spindle to a specific angular position.

```
N10 T1 D1
N15 S5000 M3 G95 F0.3
N20 G00 X50 Z5
N25 G01 Z-5
N30 M5
N35 Z5 M4
N40 M5
N45 M19
N50 G00 X200 Z50
```

Setting a delay in the program

**G04** can be used to pause the tools’ movements during operation.

**G04 F5**: Program dwells for 5 s

This makes the surface of the workpiece much smoother.
Unit Description

This unit describes how to create and edit a part program, and get to know the most important CNC commands required to produce a workpiece.

Part 2

Unit Content

Contour turning cycle

Drilling holes

Tapping

Grooving

Thread cutting

Cutting off

Drilling center holes

End

BASIC THEORY

Contour turning cycle

Step 1

The easiest way to perform roughing/finishing along the contour is to use the “contour turning” cycle function. By selecting the “Turn.” SK, you can enter the cycle and set parameters.

The “Contour turning” SK can be found on the right vertical menu.

The related parameters can be set on the screen.
By selecting the “Attach contour” SK, the contour milling data can be inserted in the main program file behind the M30 command. You can edit and change it when selected. The sequence is as follows:

- Make sure the cursor is in the editing position (shown in the figure on the left).
- Press “Cont.” on the PPU to open the contour data setting window.
- Press “Attach contour” on the PPU to create contour information at the program end. The cursor moves to the contour editing position.
- Open the cycle data setting window and enter the name of the contour subprogram.
- Press “Attach contour” on the PPU to create contour information at the program end. The cursor moves to the contour editing position.
- Use the items on the PPU to select the direction and the contour shape and enter the corresponding coordinate parameters.

After opening the contour data setting window, please make the following settings:

- Enter appropriate start point coordinates based on the machining figure and select the appropriate approach.
- Press the “Accept element” SK on the PPU.

By selecting the “Attach contour” SK, the contour milling data can be inserted in the main program file behind the M30 command. You can edit and change it when selected. The sequence is as follows:

- Open the cycle data setting window and enter the name of the contour subprogram.
- Enter appropriate start point coordinates based on the machining figure and select the appropriate approach.
- Press the “Accept element” SK on the PPU.
- Use the items on the PPU to select the direction and the contour shape and enter the corresponding coordinate parameters.
The selected direction is shown at the top left side of PPU.

The meaning of the highlighted positions is shown at the bottom of the PPU screen.

Press the “Accept element” SK on the PPU.

Select different items to set the contour until you have finished editing the whole shape of the contour.

Press the “Accept element” SK on the PPU to enter the information in the main program (stored in the cursor position after the M30 command).

After completing the steps, the system will return to the Edit interface. Press “Technical interface” on the PPU to return to the interface for setting the cycle data.

After finishing the parameter settings of CYCLE95, press the "OK" SK on the PPU to insert the corresponding cycles in the main program.

With the “New” SK and “Contour milling”, the operation can be edited and saved in a subprogram.

The editing in the subprogram is the same as above.

After all the settings take effect, the selected cycle and set data will be transferred to corresponding part program automatically (for further information, see next page).

### Step 2  
**Radius and chamfers**

The radii and the chamfer can be produced using the contour editor, in conjunction with the roughing or finishing cycles.

Press the “Accept element” SK on the PPU to enter the information in the main program (stored in the cursor position after the M30 command).

RND = Radii

CHR = Chamfer

(specified side length of isosceles triangle with chamfer as base line)

CHF = Chamfer

(specified base line length of isosceles triangle with chamfer as base line)
**N170 CYCLE95("DEMO:DEMO_E", 2.5, 0.2, 0.1, 0.15, 0.35, 0.2, 0.15, 9, 0, 0)***

---CONTOUR---

**DEMO:**
- If_DigK contour definition begin - Don't change; "GP","RO","HD"
- G16 G90 DIAMON; "GP"
- G0 Z5 X16; "GP"
- G1 Z2 X20; "GP"
- Z-15; "GP"
- Z-16 RND=2.5; "GP"
- Z-20 RND=2.5; "GP"
- X25 CHR=1; "GP"
- Z-35; "GP"
- X40 CHR=1; "GP"
- Z-45; "GP"
- X50; "GP"
- CON,V64,2,0.0000,4.4,MST-1,2,AX:Z,AK,J; "GP","RO","HD"
- S,EX:0,EY:16,ASE:0; "GP","RO","HD"
- LA,EX:-20,"GP","RO","HD"
- LL,EX:-20,"GP","RO","HD"
- AB,IDX:8; "GP","RO","HD"
- LUEY:30; "GP","RO","HD"
- F,LFA1E:1; "GP","RO","HD"
- LL,DEX:-15,"GP","RO","HD"
- LUEY:40; "GP","RO","HD"
- F,LFA1E:1; "GP","RO","HD"
- LL,EX:-55,"GP","RO","HD"
- LUEY:50; "GP","RO","HD"
- Do horizontal complete machining externally

#End contour definition end - Don't change; "GP","RO","HD"

---CONTOUR ENDS---

**Create Part Program**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Meanings</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>NPP=DEMO:DEMO_E</td>
<td>Subprogram name: &quot;DEMO&quot; (&quot;DEMO_E&quot; is created automatically)</td>
<td>The first two positions of the name must be letters.</td>
</tr>
<tr>
<td>MID=2.5</td>
<td>Maximal feed depth 2.5 mm</td>
<td></td>
</tr>
<tr>
<td>FALZ=0.2</td>
<td>Finishing allowance at the vertical axis is 0.2 mm</td>
<td></td>
</tr>
<tr>
<td>FALX=0.1</td>
<td>Finishing allowance at the horizontal axis is 0.1 mm</td>
<td></td>
</tr>
<tr>
<td>FAL=0.15</td>
<td>Contour finishing allowance is 0.15 mm</td>
<td></td>
</tr>
<tr>
<td>FF1=0.35</td>
<td>Roughing feedrate is 0.35 mm/rev</td>
<td></td>
</tr>
<tr>
<td>FF2=0.2</td>
<td>Feedrate with back cut is 0.2 mm/rev</td>
<td></td>
</tr>
<tr>
<td>FF3=0.15</td>
<td>Finishing feedrate is 0.15 mm/rev</td>
<td></td>
</tr>
<tr>
<td>VARI=9</td>
<td>Do horizontal complete machining externally</td>
<td>For other parameters, please refer to the standard manual</td>
</tr>
</tbody>
</table>
;**************CONTOUR**************

DEMO:

;#7__DlgK contour definition begin - Don't change!
G18 G90 DIAMON;*GP*
G0 Z0 X16 ;*GP*
G1 Z-2 X20 ;*GP*
Z-15 ;*GP*
Z-16.493 X19.2 RND=2.5 ;*GP*
Z-20 RND=2.5 ;*GP*
X30 CHR=1 ;*GP*
Z-35 ;*GP*
X40 CHR=1 ;*GP*
Z-55 ;*GP*
X50 ;*GP*
X,CON,V64,2,0.0000,4,4,MST:1,2,AX:Z,X,K,I;*GP*;*RO*;*HD*
S,EX:0,EY:16,ASE:0;*GP*;*RO*;*HD*
LA,EX:-2,EY:20;*GP*;*RO*;*HD*
LL,EX:-20;*GP*;*RO*;*HD*
AB,IDX:8;*GP*;*RO*;*HD*
LU,EY:30;*GP*;*RO*;*HD*
F,LFASE:1;*GP*;*RO*;*HD*
LL,DEX:-15;*GP*;*RO*;*HD*
LU,EY:40;*GP*;*RO*;*HD*
F,LFASE:1;*GP*;*RO*;*HD*
LL,EX:-55;*GP*;*RO*;*HD*
LU,EY:50;*GP*;*RO*;*HD*

#End contour definition end - Don't change!;*GP*;*RO*;*HD*

DEMO_E;;*********** CONTOUR ENDS ***********
The easiest way to produce a groove is to use CY-CLE93.

The cycle can be found and parameterized with the “Turn.” SK.

Select “Groove” using the vertical SKs and parameterize the cycle according to requirement.

With the “OK” SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below. The machine will cut a groove at the position specified in the cycle.
## BASIC THEORY

### Parameters | Meanings | Remarks
---|---|---
SPD=30 | Starting coordinate at horizontal axis is 30 |  
SPL=-30.5 | Starting coordinate at vertical axis is -30.5 |  
WIDG=7 | Groove width is 7 mm |  
DIAG=5 | Groove depth is 5 mm |  
STA1=0 (range 0°~180°) | Angle between contour and vertical axis is 0° |  
ANK1=0 (range 0°~89.999°) | Angle between positive vertical axis and groove cliff near starting point is 0° |  
ANK2=0 (range 0°~89.999°) | Angle between positive vertical axis and groove incline away from starting point is 0° |  
RCO2=1 | Reverse angle length away from machining starting point is 1mm |  
RCI1=0 | Groove bottom with no reverse angle (near groove machining starting point) |  
RCI2=0 | Groove bottom with no reverse angle (away from groove machining starting point) |  
FAL1=0.2 | Finishing allowance at the bottom of groove is 0.2 mm |  
FAL2=0.1 | Finishing allowance at groove side is 0.1 mm |  
IDEP=2.5 | Feed depth is 2.5 mm |  
DTB=0.5 | Pause 0.5 s at the bottom of groove |  
VARI=11 | Use CHR to calculate the reverse angle | For other parameters please refer to the standard manual  

N230 CYCLE93( 30.00000, -30.50000, 7.00000, 5.00000, 0.00000, 0.00000, 0.00000, 0.00000, 1.00000, 1.00000, 0.00000, 0.20000, 0.10000, 2.50000, 0.50000, 11, )
The easiest way to cut a thread is to use CYCLE99.

The cycle can be found and parameterized with the “Turn” SK.

The relevant cycle can now be found using the vertical SKs on the right.

Select “Thread” and “Thread long.” using the vertical SKs and parameterize the cycle according to requirements.

With the “OK” SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below.

The machine will cut a thread at the position specified in the cycle.
**Parameters** | **Meanings** | **Remarks**
---|---|---
SPL=0 | Thread start point coordinate at vertical axis is 0 | 
FPL=-18 | Thread end point coordinate at vertical axis is -18 mm | 
DH1=20 | Thread diameter at start point is 20 mm | 
DH2=20 | Thread diameter at end point is 20 mm | 
APP=2 | Reverse distance is 2 mm | 
ROP=0 | End distance is 0 mm | 
TDEP=1 | Thread depth is 1 mm | 
FAL=0.01 | Finishing allowance is 0.01 mm | 
IANG=29 | Feed along the same face, feed angle is 29º | IANG<0: feed along two faces in turn | 
NSP=0 | In comparison with the starting point, the angle offset of the first thread cutting point is 0º | (range 0º~359.9999º) |
NRC=8 | Roughing cutting 8 times | 
NID=2 | Empty tool cutting steps 2 | 
PIT=2.5 | Thread distance is 2.5 mm | 
VARI=300103 | Machining externally, constant cross session | For other parameters, please refer to the standard manual |
NUMTH=1 | Thread number of multi-head thread is 1 | 
PITA=1 | Select data in the PIT and in mm | 
DMODE=0 | Thread types |
Drilling center holes

The easiest way to drill a center hole prior to drilling is to use either CYCLE81 or CYCLE82.

**CYCLE81**: Without dwell at current hole depth.

**CYCLE82**: With dwell at current hole depth.

The cycle can be found and parameterized with the “Drill.” SK.

The relevant cycle can now be found.

Select “Center drilling” using the vertical SKs, and then select “Center drilling” parameterize the cycle according to

With the “OK” SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below.

If there is no other operation, the machine will drill holes at the current position.

### BASIC THEORY

**N375 CYCLE82( 5.00000, 0.00000, 2.00000, -5.00000, 0.00000, 0.50000)**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Meanings</th>
</tr>
</thead>
<tbody>
<tr>
<td>RTP=5</td>
<td>Coordinate value of turning position is 5 (absolute)</td>
</tr>
<tr>
<td>RFP=0</td>
<td>Coordinate value of hole edge starting position under workpiece zero point surface is 0 (absolute)</td>
</tr>
<tr>
<td>SDID=2</td>
<td>Safety distance, feed path changes from quick feed to machine feed 2 mm away from RFP face</td>
</tr>
<tr>
<td>DP=-5</td>
<td>Coordinate position of final drilling depth is -5 mm (absolute)</td>
</tr>
<tr>
<td>DTB=0.5</td>
<td>Dwell of 0.5 sec at final drilling depth</td>
</tr>
</tbody>
</table>

**SIEMENS**
Create Part Program
Part 2

BASIC THEORY

Drilling holes

The easiest method to drill holes is with CYCLE81/82: Without/with dwell at current hole depth. CYCLE83: Each drilling operation needs a withdrawal distance during deep hole drilling. The cycle can be found and parameterized with the “Drill” SK.

The relevant cycle can now be found using the vertical SKs on the right. Select “Center drilling” using the vertical SKs, and then select “Center drilling” and parameterize the cycle according to requirements.

With the “OK” SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below. If there is no other operation, the machine will drill holes at the current position.

For RTP, RFP, SDIS, DP, DPR and DTB and related commands, see page 50.
With the “OK” SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below. If there is no other operation, the machine will drill holes at the current position.

The easiest way to tap a hole is to use CYCLE84: Solid tap holder and CYCLE840: With floating tap holder. The cycles can be found and parameterized using the “Drill” SK.

### Drill

The relevant cycle can now be found using the vertical SKs and then select “Rigid tapping.” and parameterize the cycle according to re-

<table>
<thead>
<tr>
<th>Cycle</th>
<th>Description</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>CYCLE84</td>
<td>Drilling centering</td>
<td>D0, D1, D2, D3, D4, D5, D6, D7, D8, D9</td>
</tr>
<tr>
<td>CYCLE840</td>
<td>Deep hole drilling</td>
<td>D0, D1, D2, D3, D4, D5, D6, D7, D8, D9</td>
</tr>
<tr>
<td>CYCLE850</td>
<td>Boring</td>
<td>D0, D1, D2, D3, D4, D5, D6, D7, D8, D9</td>
</tr>
<tr>
<td>CYCLE860</td>
<td>Tapping</td>
<td>D0, D1, D2, D3, D4, D5, D6, D7, D8, D9</td>
</tr>
<tr>
<td>CYCLE870</td>
<td>Thread</td>
<td>D0, D1, D2, D3, D4, D5, D6, D7, D8, D9</td>
</tr>
</tbody>
</table>

Select “Thread” using the vertical SKs and then select “Rigid tapping.” and parameterize the cycle according to re-
### Parameters, Meanings, Remarks

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Meaning</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>DTB=0.5</td>
<td>Pause 0.5 s during final tapping to thread depth (discontinuous cutting)</td>
<td></td>
</tr>
<tr>
<td>SDAC=3</td>
<td>Spindle state after cycle is M3</td>
<td>Enter values 4/5→M4/ M5</td>
</tr>
<tr>
<td>MPIT=12</td>
<td>Thread distance is same as values corresponding to the thread size M12</td>
<td>Negative value→rotate thread left</td>
</tr>
<tr>
<td>POSS=0</td>
<td>Spindle stops at 0º (unit: º)</td>
<td></td>
</tr>
<tr>
<td>SST=200</td>
<td>Tapping thread spindle speed is 200 r/min</td>
<td>Direction is opposite to SST SST1=0 →speed is same as SST</td>
</tr>
<tr>
<td>SST1=200</td>
<td>Retraction spindle speed is 200 r/min</td>
<td></td>
</tr>
<tr>
<td>AXN=3</td>
<td>AXN is tool axis, use Z axis under G17</td>
<td></td>
</tr>
<tr>
<td>VARI=0</td>
<td>Tapping is active</td>
<td></td>
</tr>
<tr>
<td>VRT=0</td>
<td>Retraction value during discontinuous cutting is 1 mm</td>
<td>VRT&gt;0→retraction value is fixed</td>
</tr>
</tbody>
</table>

Data in SST and SST1 control the spindle speed and the Z axis feed position synchronously. During execution of CYCLE 84 the switches of the feedrate override and the cycle stop (maintaining feed) switch are not active.

For descriptions of RTP, RFP, SDIS, DP and DTB, please see page 50.
The easiest way to cut off a part is to use CYCLE92. The cycle can be found and parameterized using the “Turn.” SK.

The relevant cycle can now be found using the vertical SKs and parameterize the cycle according to requirements.

With the “OK” SK, the setting is activated and the selected cycle and data will be transferred to the part program automatically as shown below. The machine will cut off a part at the position specified in the cycle.
### Parameters | Meanings | Remarks
--- | --- | ---
DING1 | The speed is reduced at depth of 6 mm |
DING2 | When cutting off the final depth is -1 mm | Or can be set as the radii of reverse circle
RC | Width of reverse angle is 0.5 mm |
SV1 | Fixed cutting speed is 200 mm/min |
SV2 | Maximal spindle speed during fixed cutting is 2500 r/min | SDAC=4 → spindle rotation direction M4
SDAC=3 | Spindle rotation direction is M3 |
FF1=0.2 | Depth feedrate when reaching the reduced speed (DING1) |
FF2=0.08 | DING2 feedrate is 0.08 mm/min |
SS2=500 | Reduced spindle speed (until final depth) is 500 r/min |
VARI=0 | Retract to the position defined by SPD+SDIS | VARI=1 → no retraction
AMODE=11000 | Machining shape is reverse angle | AMODE=10000 → reverse circle

For descriptions of SDIS, see page 50
For descriptions of SPD and SPL, see page 47

N350 CYCLE92( 40.00000, -50.00000, 6.00000, -1.00000, 0.50000, 200.00000, 2500.00000, 3, 0.20000, 0.08000, 500.00000, 0, 0, 1, 0, 11000)
Unit Description

This unit describes how to simulate a part program before executing it in AUTO mode.

Unit Content

Simulate program (axis do not move)

A part program must have been created before it can be tested using “Simulation”.

Step 1

The part program must be opened using the “Program Manager”.

The part program must be opened using the “Program Manager”.

Simulate program (axis do not move)
Step 2
Press the “Simu.” SK on the PPU.

If the control is not in the correct mode, a message will be displayed at the bottom of the screen.

If this message is displayed at the bottom of the screen, press the “AUTO” mode key on the MCP.

Step 3
Press the “CYCLE START” key on the MCP.

Press the “Edit” SK on the PPU to return to the program.
**Unit Description**

This unit describes how to load the program in “AUTO” mode and test the part program at fixed speed.

**Unit Content**

- Program Execution
- Dry Run

---

**Program Execution**

Before the part program can be loaded and executed in AUTO mode, it must be tested using the simulation function in “Edit”.

Press the “Execute” SK on the PPU.

The control is now in AUTO mode with the current opened program storage path being displayed and the AUTO lamp on the MCP is on.

Now the program is ready to start and the actual operation will be described in the next section!
Step 1

The data in the “Dry run feedrate” must first be set and checked!

Press the “Offset” key on the PPU.

Press the “Sett. data” SK on the PPU.

Use the traversing key to move to the required position. The position is now highlighted.

Enter the required feedrate in mm/min, enter “2000” in the example.

Press the “Input” key of the PPU.

Note: The following operation is based on the finished “program execution”.

Step 2

Make sure the feedrate override on the MCP is 0%.

Press the “Machine” key on the PPU.

Press the “Prog. cont.” SK on the PPU.

Press the “Dry run feedrate” SK on the PPU.

Note: The “DRY” symbol is shown and the “Dry run feedrate” SK is highlighted in blue.

Press the “Back” SK on the PPU.

Press “Door” on the MCP to close the door of the machine. (If you don’t use this function, just close the door in the machine manually.)

Press “CYCLE START” on the MCP to execute the program.

Turn the feedrate override gradually to the required value.

After finishing the dry run, please turn the changed offset back to the original value in order to avoid affecting the actual machining!
Unit Description

This unit describes how to use the Time counter function and how to machine pieces and the compensation setting for the tool wear.

Unit Content

- Time Counter
- Machining Pieces
- Tool Wear
- End

SEQUENCE

Make sure the machine has been referenced before machining workpieces!

Step 1

Press the “Machine” key on the PPU.

Press the “Auto” key on the MCP.

Press the “Time counter” SK on the PPU.
“Cycle time” shows how long the program has been running.

“Remaining time” shows how much time remains before the program ends.

The “Remaining time” can only be counted after a successful cycle run of a part program!

Step 2

Select “Yes” or “No” to decide whether to activate the counter (press the “Select” key to activate the choice).

Enter the number of workpieces you require to be machined in “Required”.

“Actual” shows the number of workpieces that have been machined.

Make sure that only “AUTO” mode and “ROV” mode are active.

Note: M01 function → program will stop at the position where there is M01 code.

Make sure that the feedrate override on the MCP is 0%!

Press “Door” on the MCP to close the door of the machine. (If you don’t use this function, just close the door on the machine manually.)

Press “CYCLE START” on the MCP to execute the program.

Turn the feedrate override gradually to the required value.
The tool wear compensation must distinguish the direction of compensation clearly!

**Step 1**

Press the “Offset” key on the PPU.

Press the “Tool wear” SK on the PPU.

Use the direction keys to select the required tools and their edges.

**Step 2**

Set the tool length wear parameter of axis X in "Length X", the sign determines the direction of wear compensation.

Set the tool length wear parameter of axis Z in "Length Z", the sign determines the direction of wear compensation.

Positive value: The tool moves away from the workpiece

Negative value: The tool moves closer to the workpiece

Press “Input” on the PPU to activate the compensation

Set the tool radius wear parameter in “Radius”, the sign determines the direction of wear compensation.

Positive value: The tool moves away from the workpiece (set radius is larger than the actual radius)

Negative value: The tool moves closer to workpiece (set radius is smaller than the actual radius)

Press “Input” on the PPU to activate the compensation
<table>
<thead>
<tr>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>
## Unit Description

This unit describes how to restart the part program after a tool has been changed due to damage, or remachining has to be performed.

## Unit Content

**Block search**

Press the “Machine” key on the PPU.

Press the “Auto” key on the MCP.

Press the “Block search” SK on the PPU.

Press the “Inter. point” SK on the PPU and the cursor will move to the last interrupted program line.

**Note:** The cursor can be moved to the required program block with the traversing keys.

Press the “To end point” SK on the PPU (or select “To contour” as required).

**Note:** The “To contour” and “To end point” functions.

“To contour”: The program will continue from the line before the breakpoint.

“To end point”: The program will continue from the line with the breakpoint.
Press the “Back” SK on the PPU.

The feedrate override must always be set to 0%!
Make sure the correct tool is selected before continuing!

Press the “CYCLE START” key on the PPU to execute the program.

Alarm 010208 is shown at the top prompting to press the “CYCLE START” key to continue the program.

Press the “CYCLE START” key on the MCP to execute the program.

Turn the feedrate override on the MCP gradually to the required value.
Unit Description

This unit describes how to perform simple tasks on the machine and provides some additional information which may be required to operate the machine correctly.

Unit Content

RS232c + USB

Help

R parameters

Time change

Save data

Gear change

End

RS232c is used to transfer the programs to and from the NC.

Step 1

It is recommended to use the “Sinucom PCIN” communication SW provided by Siemens to transfer the standard program.

Adjust the parameter settings on the PPU to match the settings of the communication SW on PC.

Press “Program Manager” on the PPU.

Press the “RS232” SK on the PPU.

Press the “Settings” SK on the PPU.

Adjust the parameters in “Communication settings” to match the settings of communication SW on PC.

- Device
- Baud rate
- Stop bits
- Parity
- Data bits
- End of transmit
- Confirm overwrite

Press the “Save” SK on the PPU.

Press the “Back” SK on the PPU.
SEQUENCE

Step 2  Transfer a part program to a PC from the PPU.

Press the “NC” SK on the PPU.

Use “Cursor + Select” to select the required part program. The selected program will be highlighted.

Press the “Copy” SK on the PPU.

Press the “RS232” SK on the PPU.

Check the interface setting and start the communication software to receive the program from PC.

(Press “Receive Data” on SINUCOM PCIN to start the receive function.)

Press the “Send” SK on the PPU.

The PPU will display a window showing the progress of the transfer.

If there is a problem during transfer of the part program, a window will be displayed.

Step 3  Transfer a part program to the PPU from a PC.

Press the “RS232” SK on the PPU.

Press the “Receive” SK on the PPU.

The PPU will display a window showing the progress of the transfer.

Check the interface setting and start the communication software to send the program from PC.

(Press “Send Data” on SINUCOM PCIN to send data.)

The PPU will display a window showing the progress of the transfer.
“USB” is used to transfer the programs to and from the NC.

**Step 4**

Use the “Copy” “Paste” SKs to transfer the part program from NC to USB.

Connect a USB device with sufficient memory to the USB interface on the PPU.

Press the “NC” SK on the PPU.

Use “Cursor + Select” to select the required part program. The selected program will be highlighted.

Press the “Copy” SK on the PPU.

Press the “USB” SK on the PPU.

Press the “Paste” SK on the PPU.

**Step 5**

Use the “Copy” and “Paste” SKs to transfer the part program from USB to NC.

Connect the USB device with the stored target programs to the USB interface on the PPU.

Press the “USB” SK on the PPU.

Use “Cursor + Select” to select the required part program. The selected program will be highlighted.

Press the “Copy” SK on the PPU.

Press the “NC” SK on the PPU.

Press the “Paste” SK on the PPU.

---

The PPU has an online help which shows the contents of standard documents.

Press the “Help” key on the PPU.

Press the “Cur. Topic” SK on the PPU.

The help information related to the current topic will be shown on screen.

Press the “OEM Manual” SK on the PPU.

The online help manual of the OEM will be shown on the screen.

Press the “TOC” SK on the PPU.

The online help from the Siemens manual will be shown.
The arithmetic parameters are used in a part program for value assignment, and also for some necessary value calculations. The required values can be set or calculated by the control system during program execution. Some of the common arithmetic functions are shown below:

<table>
<thead>
<tr>
<th>Arithmetic parameters</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>+</td>
<td>Addition</td>
</tr>
<tr>
<td>-</td>
<td>Subtraction</td>
</tr>
<tr>
<td>*</td>
<td>Multiplication</td>
</tr>
<tr>
<td>/</td>
<td>Division</td>
</tr>
<tr>
<td>=</td>
<td>Equals</td>
</tr>
<tr>
<td>Sin()</td>
<td>Sine</td>
</tr>
<tr>
<td>COS()</td>
<td>Cosine</td>
</tr>
<tr>
<td>TAN()</td>
<td>Tangent</td>
</tr>
<tr>
<td>ASIN()</td>
<td>Arcsine</td>
</tr>
<tr>
<td>ACOS()</td>
<td>Arccosine</td>
</tr>
<tr>
<td>ATAN2( , )</td>
<td>Arctangent²</td>
</tr>
<tr>
<td>SQRT()</td>
<td>Square root</td>
</tr>
<tr>
<td>ABS()</td>
<td>Absolute value</td>
</tr>
</tbody>
</table>

Note:
Preprocessing stop
Programming the STOPRE command in a block will stop block preprocessing and buffering. The following block is not executed until all preprocessed and saved blocks have been executed in full. The preceding block is stopped in exact stop (as with G9).
You can change the time on the control if required when the clocks change from summer time to winter time.

Press “Shift” and “Alarm” on the PPU simultaneously.

Make sure the password is set to the “CUSTOMER” access level.

Press the “Date time” SK on the PPU.

Press the “Cancel” SK on the PPU to abort the operation.

“Save data” enables the complete system to be backed up on the system CF card so that there is a system backup available to the operator.

Press “Shift” and “Alarm” on the PPU simultaneously.

Make sure the password is set to the “CUSTOMER” access level.

Press the “Save data” SK on the PPU.
Gear stages M40, M41, M42, M43, M44 and M45 are available.

- **M40** Automatic gear selection
- **M41** Gear stage 1
- **M42** Gear stage 2
- **M43** Gear stage 3
- **M44** Gear stage 4
- **M45** Gear stage 5

**Example:**

The machine tool manufacturer specifies a speed range for each gear stage:

- **S0...500** Gear stage 1 $\rightarrow$ M41
- **S400..1200** Gear stage 2 $\rightarrow$ M42
- **S1000..2000** Gear stage 3 $\rightarrow$ M43

If the operator is manually selecting the gear stage in the part program, it is the operator’s responsibility to select the correct gear stage according to the required speed.

When a machine has a manual gearbox on the spindle, it is the responsibility of the operator to change gear at the correct place in the part program.

If the machine tool manufacturer has fitted an automatic gearbox, the following M-codes can be used to change gear in the part program.

Press the “OK” SK on the PPU.

*WARNING: While the control is saving data to the system, do not operate or switch off the control!*
Unit Description
This unit describes how to perform simple tasks on the machine and provides some additional information which may be required to operate the machine correctly.

Part 2

Unit Content

- MDA
- M/H function
- Sub program
- Polar coordinates
- Additive workpiece offsets
- Scaling
- Program jump
- Program skip
- Calculator
- End

SEQUENCE

In MDA mode, you can enter and execute single and multiple lines of NC codes.

Use MDA to move the axis to a fixed position.

Press the “Machine” key on the PPU.

Press the “MDA” key on the PPU.

Press the “Delete file” SK on the PPU.

Enter correct NC code to move the axis to the required position...

Make sure the feedrate override on the MCP is at 0%!

Press “CYCLE START” on the MCP to execute the MDA program.

Turn the feedrate override on the MCP gradually to the required value.
Frequently used machining sequences, e.g. certain contour shapes, are stored in subprograms. These subprograms are called at the appropriate locations in the main program and then executed.

The structure of a subprogram is identical to that of the main program, but a subprogram contains M17 - end of program in the last block of the program sequence. This means a return to the program level where the subprogram was called.

The subprogram should be given a unique name enabling it to be selected from several subprograms. When you create the program, the program name may be freely selected. However, the following rule should be observed:

The name can contain letters, numbers and underscores and should be between 2 and 8 characters long.

Example: LRAHMEN7
In addition to the common specification in Cartesian coordinates (X, Z), the points of a workpiece can also be specified using polar coordinates. Polar coordinates are also helpful if a workpiece or a part of it is dimensioned from a central point (pole) with specification of the radius and the angle.

The polar coordinates refer to the plane activated with G17 to G19. In addition, the third axis perpendicular to this plane can be specified. When doing so, spatial specifications can be programmed as cylindrical coordinates.

The polar radius \( R_P = \) specifies the distance of the point to the pole. It is saved and must only be written in blocks in which it changes, after the pole or the plane has been changed.

The polar angle \( \theta_P = \) is always referred to the horizontal axis (abscissa) of the plane (for example, with G18: X axis). Positive or negative angle specifications are possible. The positive angle is defined as follows: Starting from the plus direction of X axis and rotates CCW.

It is saved and must only be written in blocks in which it changes, after the pole or the plane has been changed.
Additional Information
Part 2

SEQUENCE

G110 Pole specification relative to the setpoint position last programmed
   (in the plane, e.g. with G18: Z/X)
   (when using G110, please always take the current position of the
    tool as the reference point to specify the new pole)
G111 Pole specification relative to the origin of the current workpiece
   coordinate system (in the plane, e.g. with G18: Z/X)
G112 Pole specification, relative to the last valid pole; retain plane

Programming example

N10 G18 ; Z/X plane
N20 G111 X17 Z36 ; pole coordinates in the current workpiece
   coordinate system...
N80 G112 AP=45 RP=27.8 ; new pole, relative to the last pole as a
   polar coordinate
N90 ... AP=12.5 RP=47.679 ; polar coordinate
N100 ... AP=26.3 RP=7.344 Z4 ; polar coordinate and Z axis (= cylinder
   coordinate)

Example: G18: Z/X plane

The programmable workpiece offsets TRANS and ATRANS can be used
in the following cases:

- For recurring shapes/arrangements in various positions on the work-
  piece
- When selecting a new reference point for dimensioning

This results in the current workpiece coordinate system.

TRANS X... Z... ; programmable offset (absolute)
ATRANS X... Z... ; programmable offset, additive to existing offset
   (incremental)
TRANS ; without values, clears old commands for offset

Programming example
N20 TRANS X20.0 Z15.0 programmable offset
L10 subprogram call
A scale factor can be programmed for all axes with SCALE, ASCALE. The path is enlarged or reduced by this factor in the specified axis. The currently set coordinate system is used as the reference for the scale change.

SCALE X... Z... ; programmable rotation offset (absolute)
ASCALE X... Z... ; programmable offset, additive to existing offset (incremental)

If a program contains SCALE or ASCALE, this must be programmed in a separate block.

Programming example
N10 G17
N20 SCALE X2.0 Z2.0 ; contour is enlarged two times in X and Z
L10 subprogram call

NC programs process their blocks in the sequence in which they were arranged when they were written. The processing sequence can be changed by introducing program jumps. The jump destination can be a block with a label or with a block number. This block must be located within the program. The unconditional jump command requires a separate block.

GOTO F+ label: Jump forward (in the direction of the end block of the program)
GOTO B+ label: Jump backward (in the direction of the start block of the program)
Label: Name of the selected string (standing for the required jump program block) or block number
Program execution

N10 G0 X...Z...
...
...
N40 GOTO LABEL0; jumps to label LABEL0
...
...
N70 LABEL0: R1=R2+R3
N80 GOTO LABEL1; jumps to label LABEL1
N90 LABEL2:
M30; program ends
N110 LABEL1:
...
N130 GOTO LABEL2; jumps to label LABEL2

Unconditional jump example

";" code
Using ";" code at the beginning of the block can skip this string. ";" can also be used to add remarks to the block.
See the figure on the right for an example of use.

Method 1

Using ";" code at the beginning of the program block N95, this string will be skipped without execution.

Using ";" code to add a remark to the N85 function, without any influence on the execution.

Program skip

N5 G90 G500 G71
N10 T1 D1 M6
N15 S3000 M3 G94 F300
N20 G00 X50 Z5
N25 G01 Z-20
N30 Z5
...
N85 T2 D1 M6
N90 S3000 M3 G94 F300
; N95 G00 X60 Z10
...
**SEQUENCE**

**Method 2**

Press the “Machine” key on the PPU.

Press the “Auto” key on the MCP.

Press the “Prog cont.” SK on the PPU.

Press the “Skip” SK on the PPU.

You can use the calculator to calculate contour elements, values in the program editor, tool offsets and workpiece offsets and enter the results on the screen.

Press the “=” on the PPU.

When “SKP” is displayed (red circle), the skip function has been activated. After activating “SKP”, using “/” at the beginning of the program string (shown in purple circle), the string will be skipped without influencing the execution.
SEQUENCE

- Press this SK to delete the contents in the calculator.
- Press this SK to exit the calculator screen.
- Use this SK to accept the input and write the values to the required position.
  - If the input field is already occupied by a value, the calculator will take this value into the input line.
  - Use the “Accept” SK to enter the result in the input field at the current cursor position of the part program editor. The calculator will then close automatically.
Unit Description

This unit shows three typical program examples of frequently used turning cycles and the corresponding machining diagrams with detailed explanations.

Unit Content

- Turning program 1
- Turning program 2
- Turning program 3
- End

Tool information:
- T1 Turning tool D0.8
- T2 Turning tool D0.8

Note: All the program examples in this book are only for reference. If you want to perform actual operations, please adjust the tool offset, coordinate moving range, workpiece plane settings, etc. according to the actual machine conditions!

Make sure all the preparations and safety measures have been performed before machining!
N10  G00 G90 G95 G40 G71
N20  LIMS=4500
N30  T1 D1
; ========Start face turning========
N40  G00 G90 G95 G40 G71
N50  G00 X60 Z0
N60  G01 X-2 F0.35
N70  G00 Z2
N80  G00 X60
; ========End face turning========

N90  CYCLE95( "CON1:CON1_E", 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 9, , )

N100 T2 D1
N110  G96 S250 M03 M08
; ==Start contour turning finishing with back cut=================
N120 CYCLE95( "CON2:CON2_E", 0.5, , 0.2, 0.4, 0.3, 0.2, 9, , )
N130 M30

N10  spindle feedrate in mm/r
N20  set spindle upper limit 4500 r/min
N30  ; ========Start face turning========
N40  constant cutting speed 250 m/min
N50  N60  feedrate is 0.35 mm/r
N70  N80  ; ========End face turning========

; ==Start contour turning roughing without back cut==
N90  CYCLE95( "CON1:CON1_E", 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 9, , )

N100 T2 D1
N110  G96 S250 M03 M08
; ==Start contour turning finishing with back cut=================
N120 CYCLE95( "CON2:CON2_E", 0.5, , 0.2, 0.4, 0.3, 0.2, 9, , )
N130 M30

CON1:
; #7_DlgK contour definition begin - Don't change!
; GP*: RO*: HD*
G18 G90 DIAMON;*GP*
G0 Z2 X16;*GP*
G1 Z-2 X20;*GP*
Z-20;*GP*
X35 RND=2;*GP*
X55 CHR=2;*GP*
Z-70;*GP*
:CON,V64,2,0.0000,6,6,1.2,AX:Z,X,K,I;*GP*;*RO*;*HD*
;S,EX:0,EY:16,ASE:0:*GP*;*RO*;*HD*
;LL,EX:-20:*GP*;*RO*;*HD*
;LU,EY:35:*GP*;*RO*;*HD*
;R,ROUND:2;*GP*;*RO*;*HD*
;LL,DEX:-30:*GP*;*RO*;*HD*
;LL,EX:70:*GP*;*RO*;*HD*
;#End contour definition end - Don't change!
CON1_E:;******* CONTOUR ENDS ************

CON2:
; #7_DlgK contour definition begin - Don't change!
; GP*: RO*: HD*
G18 G90 DIAMON;*GP*
G0 Z-22.5 X35;*GP*
G2 Z-49.5 K=AC(-35) I=AC(89.544);*GP*
G1 Z-49.5;*GP*
:CON,V64,2,0.0000,1,1,1.2,AX:Z,X,K,I;*GP*;*RO*;*HD*
;S,EX:-22.5,EY:35,ASE:0:*GP*;*RO*;*HD*
;ACW,DIA:0/235,DEX:-25,DEY:0,RAD:30:*GP*;*RO*;*HD*
;LL,DEX:-2;*GP*;*RO*;*HD*
;#End contour definition end - Don't change!
CON2_E:;******* CONTOUR ENDS ************

This program is additional description information created by the system automatically after finishing the programming of the rough cutting CYCLE95 and does not affect the system execution.
Make sure all the preparations and safety measures have been performed before machining!

<table>
<thead>
<tr>
<th>Line</th>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>N10</td>
<td>G54 G00 G90 G95 G40 G71</td>
<td>; ========Start face turning=======</td>
</tr>
<tr>
<td>N20</td>
<td>LIMS=4500</td>
<td>set spindle upper limit 4500 r/min</td>
</tr>
<tr>
<td>N30</td>
<td>T1 D1</td>
<td>constant cutting speed 250 m/min</td>
</tr>
<tr>
<td>N40</td>
<td>G96 S250 M03 M08</td>
<td>; ========Start face turning=======</td>
</tr>
<tr>
<td>N50</td>
<td>G00 X35 Z0</td>
<td>spindle feedrate in mm/r</td>
</tr>
<tr>
<td>N60</td>
<td>G01 X-2 F0.35</td>
<td>set spindle upper limit 4500 r/min</td>
</tr>
<tr>
<td>N70</td>
<td>G00 Z2</td>
<td>feedrate is 0.35 mm/r</td>
</tr>
<tr>
<td>N80</td>
<td>G00 X35</td>
<td>; ========End face turning========</td>
</tr>
<tr>
<td>N90</td>
<td>T13 D1</td>
<td>spindle speed 1000 r/min, X/Y plane</td>
</tr>
<tr>
<td>N100</td>
<td>CYCLE83 (10, 0, 2, -23, 0, -10, 5, 1, 0, 1, 5, 0, 0)</td>
<td>; =========Start drilling==========</td>
</tr>
<tr>
<td>N110</td>
<td>CYCLE83 (10, 0, 2, -23, 0, -10, 5, 1, 0, 1, 5, 0, 0)</td>
<td>spindle speed 1000 r/min, X/Y plane</td>
</tr>
<tr>
<td>N120</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>; ==Start contour turning roughing==</td>
</tr>
<tr>
<td>N130</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>maximal feed depth 1.5 mm,</td>
</tr>
<tr>
<td>N140</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>vertical axis finishing allowance 0.2 mm,</td>
</tr>
<tr>
<td>N150</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>horizontal axis finishing allowance 0.1</td>
</tr>
<tr>
<td>N160</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>mm, roughing feedrate 0.5 mm/r,</td>
</tr>
<tr>
<td>N170</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>feedrate 0.3 mm/r with back cut, finishing</td>
</tr>
<tr>
<td>N180</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>feedrate 0.2 mm/r, feed along positive</td>
</tr>
<tr>
<td>N190</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>direction of Z axis to do complete ma-</td>
</tr>
<tr>
<td>N200</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>chining</td>
</tr>
<tr>
<td>N210</td>
<td>CYCLE95(“CON1:CON1_E”, 1.5, 0.2, 0.1, 0.5, 0.3, 0.2, 11, , ,)</td>
<td>; ==End contour turning roughing===</td>
</tr>
</tbody>
</table>

**Tool Information:**
- T1: Turning tool D0.8
- T10: Turning tool D0.8
- T110: Grooving tool D0.2
- Tool tip width 3
CON1:

;#7_DlgK contour definition begin -
Don't change!;"GP";"RO";"HD"
G18 G90 DIAMON;"GP"
G0 Z0 X27 ;"GP"
G1 Z-.89 X24.11 ;"GP"
Z-9 X16 ;"GP"
Z-21 ;"GP"
X10 ;"GP"
;CON,V64,2,0.0000,4,4,MST:1,2,AX:Z,X
,K,I,"GP";"RO";"HD"
;S,EX:0,EY:27,ASE:0;"GP";"RO";"HD"
;LA,EX:-.89,EY:24.11;"GP";"RO";"HD"
;LA,DEX:-8.11,EY:16;"GP";"RO";"HD"
;LL,EX:-21;"GP";"RO";"HD"
;LD,EY:10;"GP";"RO";"HD"
#End contour definition end - Don't change!;"GP";"RO";"HD"
CON1_E:,******CONTOUR ENDS *****

This program is additional description information created by the system automatically after finishing the programming of the rough cutting CYCLE95 and does not affect the system execution.
### Tool information:

<table>
<thead>
<tr>
<th>Tool</th>
<th>Diameter (mm)</th>
<th>Length (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>D0.8</td>
<td>20</td>
</tr>
<tr>
<td>T2</td>
<td>D0.8</td>
<td>20</td>
</tr>
<tr>
<td>T3</td>
<td>D0.2</td>
<td>20</td>
</tr>
<tr>
<td>T4</td>
<td>D0.8</td>
<td>20</td>
</tr>
<tr>
<td>T5</td>
<td>D0.2</td>
<td>20</td>
</tr>
<tr>
<td>T6</td>
<td>D10</td>
<td>20</td>
</tr>
<tr>
<td>T7</td>
<td>D10</td>
<td>20</td>
</tr>
<tr>
<td>T8</td>
<td>D12</td>
<td>20</td>
</tr>
</tbody>
</table>

### Machining Process

- **N10** G00 G90 G95 G40 G71
- **N20** LIMS=4500
- **N30** T1 D1 ;ROUGH TURN
- **N40** G96 S250 M03 M08
- **N50** G00 X52.0 Z0.1
- **N60** G01 X-2.0 F0.35
- **N70** G00 X52.0 Z2.0
- **N80** CYCLE95("DEMO:DEMO_E", 2.5, 0.2, 0.1, 0.15, 0.35, 0.2, 0.15, 9, , , )
- **N90** G00 G40 X500.0 Z500.0
- **N100** M01
- **N110** T2 D1 ;FINISH TURN
- **N120** G96 S350 M03 M08
- **N130** G00 X22.0 Z0.0
- **N140** G01 X-2.0 F0.15
- **N150** G00 Z2.0
- **N160** X52.0
- **N170** CYCLE95("DEMO:DEMO_E", , , , , 0.15, 5, , , )
- **N180** G00 G40 X500.0 Z500.0
- **N190** M01
- **N200** T3 D1 ;GROOVE
- **N210** G96 S200 M03 M08
- **N220** G00 X55.0 Z0.
- **N230** CYCLE93(30, -30.5, 7, 5, 0, 0, 0, 1, 1, , 0, 0.2, 0.1, 2.5, 0.5, 11, )
- **N240** G00 G40 X500.0 Z500.0
- **N250** M01

- **N10** spindle feedrate in mm/r
- **N20** set spindle upper limit 4500 r/min
- **N30** constant cutting speed 250 m/min
- **N40** feedrate is 0.35 mm/r
- **N50** maximal feed depth 2.5 mm,
- **N60** vertical axis finishing allowance 0.2 mm,
- **N70** horizontal axis finishing allowance 0.1 mm,
- **N80** contour finishing allowance 0.15 mm,
- **N90** roughing feedrate 0.35 mm/r,
- **N100** feedrate 0.2 mm/r with back cut,

---

Part of the cycles in the program are taken as examples in Section 5, “Create Part Program Part 2”!
Programming and Operating — Turning

Page 98

Machining Process

; =========START THREAD=========
N260 T4 D1 ; THREAD
N270 G95 S150 M03 M08
N280 G80 X500 Z10.0
N290 CYCLE99( 0, 20, -18, 20, 0, 1, 0.01, 29, 0, 8, 2, 2.5, 300103, 1, 0, 0, 0, 0, 0, 0, 1, 0, 0, 0, 0 )
N300 G00 G40 X500.0 Z500.0
N310 M01

; =========CENTER DRILL=========
N355 T6 D1 ; CENTER DRILL
N360 G95 S1000 M03 M08
N370 G17 G00 X0 Z5
N375 CYCLE82( 5, 0, 2, -5, 0, 0.5 )
N380 G00 G40 X500 Z500

; =========DRILL========
N390 T7 D1 ; DRILL
N400 G95 S1000 M03 M08
N410 G100 X0 Z5
N420 CYCLE82( 5, 0, 2, -20, 0, 0.5 )
N430 G00 G40 X500 Z500

; =========CUT OFF========
N320 T5 D1 ; CUT-OFF
N330 G18 G96 S200 M03 M08
N340 G00 X550.0 Z10.0
N350 CYCLE92( 40, -50, 6, -1, 0.5, 200, 2500, 3, 0.2, 0.08, 500, 0, 0, 1, 0, 11000 )
N351 G00 G40 X500 Z500
N360 G00 G40 X500.0 Z500.0
N370 M30

; =========THREAD=========
N380
g95 → spindle feedrate in mm/r
N390 size of thread 2.5 mm, on Z
N400 axis start point → end point: 0 → 20,
N410 diameter at start point / end point are
N420 both 20 mm, reverse distance 2 mm,
N430 ending distance 0 mm, thread depth 1
N440 mm, finishing allowance 0.01 mm, feed
N450 angle 29°, first thread start point offset
N460 0 mm, rough cutting 8 times, idle tool
N470 cutting 2 mm, thread machining path is
N480 thread string number 1
N490 (discontinuous drilling)

; =========START CENTER DRILLING=========
N500 G40 → cancel tool radius compensa-
N510 tion
N520 N310 delay changing tool

; =========START DRILLING=========
N530
drilling depth 5 mm, delay time
N540 at final drilling depth is 0.5 s
N550 (discontinuous drilling)
N560

; =========START TAPPING HOLE=========
N570
tapping depth 18 mm, deep
N580 drilling delays 0.5 s (discontinuous drill-
N590 ing), spindle rotating direction is M3
N600 when withdrawn, thread size is M12,
N610 spindle stop position is 0°, tapping speed
N620 and turning speed are both 200 mm/min,
N630 tool axis is Z axis, machining way is
N640 tapping, withdraw path is 1 mm
N650 (discontinuous drilling)
N660

; =========CUT OFF========
N320
tapping, withdraw path is
N330 tool axis is
N340 (discontinuous drilling)
N350 spindle stop position is 0
N360 when withdrawn, thread size is M12,
N370 spindle rotating direction is M3, feedrate depth is
N380 0.2 mm/min when rotational speed is
N390 reached, reduced feedrate (until the final depth) is 0.08 mm/min, reduced speed
N400 (until the final depth) is 500 r/min, machining path returns to basic plane, alter-
N410 native mode is reverse angle.

; =========CUT OFF========
N320
tapping, withdraw path is
N330 tool axis is
N340 (discontinuous drilling)
N350 spindle stop position is 0
N360 when withdrawn, thread size is M12,
N370 spindle rotating direction is M3, feedrate depth is
N380 0.2 mm/min when rotational speed is
N390 reached, reduced feedrate (until the final depth) is 0.08 mm/min, reduced speed
N400 (until the final depth) is 500 r/min, machining path returns to basic plane, alter-
N410 native mode is reverse angle.

; =========START DRILLING=========
N390
drilling depth 20 mm, delay time
N400 at final drilling depth is 0.5 s
N410 (discontinuous drilling)
N420

; =========START TAPPING HOLE=========
N440
tapping depth 18 mm, deep
N450 drilling delays 0.5 s (discontinuous drill-
N460 ing), spindle rotating direction is M3
N470 when withdrawn, thread size is M12,
N480 spindle stop position is 0°, tapping speed
N490 and turning speed are both 200 mm/min,
N500 tool axis is Z axis, machining way is
N510 tapping, withdraw path is 1 mm
N520 (discontinuous drilling)
N530

; =========CONTOUR========
N490 18 mm, deep drill-
N500 ing delays 0.5 s (discontinuous drill-
N510 ing), spindle rotating direction is M3
N520 when withdrawn, thread size is M12,
N530 spindle stop position is 0°, tapping speed
N540 and turning speed are both 200 mm/min,
N550 tool axis is Z axis, machining way is tapping, withdraw path is 1 mm (discontinuous drilling)

; =========START THREAD=========
N260 G95 → spindle feedrate in mm/r
N270 size of thread 2.5 mm, on Z
N280 axis start point → end point: 0 → 20,
N290 diameter at start point / end point are
N300 both 20 mm, reverse distance 2 mm,
N310 ending distance 0 mm, thread depth 1
N320 mm, finishing allowance 0.01 mm, feed
N330 angle 29°, first thread start point offset
N340 0 mm, rough cutting 8 times, idle tool
N350 cutting 2 mm, thread machining path is thread string number 1
N360 (discontinuous drilling)

; =========START CENTER DRILLING=========
N390
drilling depth 5 mm, delay time
N400 at final drilling depth is 0.5 s
N410 (discontinuous drilling)
N420

; =========START DRILLING=========
N430
drilling depth 20 mm, delay time
N440 at final drilling depth is 0.5 s
N450 (discontinuous drilling)
N460

; =========START TAPPING HOLE=========
N470
tapping depth 18 mm, deep
N480 drilling delays 0.5 s (discontinuous drill-
N490 ing), spindle rotating direction is M3
N500 when withdrawn, thread size is M12,
N510 spindle stop position is 0°, tapping speed
N520 and turning speed are both 200 mm/min,
N530 tool axis is Z axis, machining way is tapping, withdraw path is 1 mm
N540 (discontinuous drilling)
N550

; =========CONTOUR========
N490 18 mm, deep drill-
N500 ing delays 0.5 s (discontinuous drill-
N510 ing), spindle rotating direction is M3
N520 when withdrawn, thread size is M12,
N530 spindle stop position is 0°, tapping speed
N540 and turning speed are both 200 mm/min,
N550 tool axis is Z axis, machining way is tapping, withdraw path is 1 mm (discontinuous drilling)
N560

; =========CONTOUR========
N490 18 mm, deep drill-
N500 ing delays 0.5 s (discontinuous drill-
N510 ing), spindle rotating direction is M3
N520 when withdrawn, thread size is M12,
N530 spindle stop position is 0°, tapping speed
N540 and turning speed are both 200 mm/min,
N550 tool axis is Z axis, machining way is tapping, withdraw path is 1 mm (discontinuous drilling)
Module Description
This unit describes the ISO operating functions in 808D, compares the similarities and differences of the machining code in DIN mode and ISO mode and shows how to transfer and implement the ISO machining program. All the ISO codes described in this unit can be implemented in the ISO mode of the 808D system.

Module Contents
- ISO function switch
- ISO code explanation
- ISO program transfer and operation

BASIC THEORY
Siemens standard machining codes are implemented in DIN mode. The 808D also provides appropriate functions for implementing the ISO commands, but the ISO mode must be activated during operation.

ISO function switch

Method 1
Press the “Shift” + “System - Alarm” keys on the PPU. Input the manufacturer’s password (“SUNRISE”)

Press the “ISO mode” SK on the right.

A dialog box appears prompting whether to activate the new setting. Select the “OK” SK to activate it.
After pressing “OK”, the system restarts automatically. After restarting, press “Shift” + “System - Alarm” again and if the symbol in the red circle appears, ISO mode is already activated.

A red ISO appears at the top of the screen and the ISO mode button on the right is highlighted in blue.

Method 2

When using method 2 to activate the ISO mode, it will exit ISO mode and return to the default DIN mode via “Reset” button or after finishing the machining program.

Insert G291 in the first line of the ISO part program to be executed and insert G290 in front of M30.

If ISO is displayed at the top of the screen, it is activated.

ISO code explanation

All the ISO codes described in this unit can be implemented in the ISO mode of the 808D system!

<table>
<thead>
<tr>
<th>ISO code</th>
<th>Description</th>
<th>Compare with DIN</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>Orientation (rapid traverse)</td>
<td>As DIN</td>
</tr>
<tr>
<td>G1</td>
<td>Linear difference</td>
<td>As DIN</td>
</tr>
<tr>
<td>G17/G18/G19</td>
<td>XY plane / ZX plane / YZ plane</td>
<td>As DIN</td>
</tr>
<tr>
<td>G20/G21</td>
<td>Input in inch/mm</td>
<td>G70/G71</td>
</tr>
<tr>
<td>G32</td>
<td>Equal lead thread cutting</td>
<td>G33</td>
</tr>
<tr>
<td>G41/G42/G40</td>
<td>Left tool tip radius compensation / right tool tip radius compensation / cancel tool radius compensation</td>
<td>As DIN</td>
</tr>
<tr>
<td>G54 ~ G59</td>
<td>Select workpiece coordinate system</td>
<td>As DIN</td>
</tr>
<tr>
<td>G80</td>
<td>Cancel fixed cycle</td>
<td></td>
</tr>
<tr>
<td>G98/G99</td>
<td>Feedrate F in mm/min / mm/r</td>
<td>G94/G95</td>
</tr>
<tr>
<td>S</td>
<td>Spindle speed</td>
<td>As DIN</td>
</tr>
<tr>
<td>R</td>
<td>Reverse circle</td>
<td>RND</td>
</tr>
<tr>
<td>, C</td>
<td>Reverse bevel angle (note the form there must be “,”, “ before C parameter)</td>
<td>CHF/CHR</td>
</tr>
<tr>
<td>M3/M4/M5</td>
<td>Spindle right / spindle left / spindle stop</td>
<td>As DIN</td>
</tr>
<tr>
<td>M98 P <em>L</em></td>
<td>Subprogram call (P+ subprogram name/ L+ times)</td>
<td>Program name +L_</td>
</tr>
<tr>
<td>M99</td>
<td>Subprogram end</td>
<td>M17</td>
</tr>
</tbody>
</table>
BASIC THEORY

Tool function T code

T code has two functions:
①→change automatically
②→execute tool offset

Code form T ΔΔ OO
ΔΔ: Enter target tool number
OO: Input tool offset number

Note: When using G291 to activate ISO mode, you must set machine data MD10890=0, or the tool path can not be implemented.

Table: Tool offset number

<table>
<thead>
<tr>
<th>Tool offset number</th>
<th>X</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>00</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>01</td>
<td>0.000</td>
<td>0.000</td>
</tr>
<tr>
<td>02</td>
<td>12.000</td>
<td>-23.000</td>
</tr>
<tr>
<td>03</td>
<td>24.560</td>
<td>13.542</td>
</tr>
</tbody>
</table>

Note: When using G291 to activate ISO mode, you must set machine data MD10890=0, or the tool path can not be implemented.

M3 S2000;  spindle rotation
G98 F500 G01 X100;  feedrate is 500 mm/min
G92 X50 W-20 F2 ;F is the thread lead
G04 X2.0 ;delay 2 s
G99 G01 U10 F0.01 ;feedrate is 0.01 mm/r
G00 G80 Z50 M30 ;cancel this cycle
M5 ;spindle rotation stop
M30

When specifying circle radii with parameter R
Circles less than 180° are assigned positive values
G02 X6.0 Y2.0 R50.0

Circles greater than 180° are assigned negative values
G02 X6.0 Y2.0 R-50.0

Code G02 and G03

G02 circular interpolation CW
Motion path: Start point→end point
CW (rear tool coordinate system) / CCW (front tool coordinate system)

G03 circular interpolation CCW
Motion path: Start point→end point
CCW (rear tool coordinate system) / CW (front tool coordinate system)

You can specify the circle end point in the following X/Z address for both. You can also describe circle radii with I, K incremental or use parameter R to specify radii directly.

When specifying circle radii with parameter R

Circles less than 180° are assigned positive values
G02 X6.0 Y2.0 R50.0

Circles greater than 180° are assigned negative values
G02 X6.0 Y2.0 R-50.0
Frequently used letter meanings of typical fixed cycle codes in ISO mode

<table>
<thead>
<tr>
<th>P.</th>
<th>Descriptions</th>
<th>Unit</th>
<th>Applied range and note</th>
</tr>
</thead>
<tbody>
<tr>
<td>X/Z</td>
<td>Cutting end point X/Z absolute coordinate values</td>
<td>mm</td>
<td>G90 / G94 / G74</td>
</tr>
<tr>
<td></td>
<td>Absolute coordinate difference between start point and end point at X/Z</td>
<td>mm</td>
<td>G90 / G94 / G74</td>
</tr>
<tr>
<td></td>
<td>X/Z tool retraction / finishing allowance</td>
<td>mm</td>
<td>G73</td>
</tr>
<tr>
<td>U/W</td>
<td>Radii—difference between start point and end point</td>
<td>G90 / G94 / G92</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Each radial/shaft (X/Z axis) tool retraction e</td>
<td>mm</td>
<td>G71 / G72 / G74 / G75</td>
</tr>
<tr>
<td></td>
<td>Cutting times d</td>
<td>G73</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Thread finishing d / thread cone i</td>
<td>mm</td>
<td>G76</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>P.</th>
<th>Descriptions</th>
<th>Unit</th>
<th>Applied range and note</th>
</tr>
</thead>
<tbody>
<tr>
<td>P</td>
<td>Single radial cutting cycle at X axis Δi</td>
<td>0.001mm</td>
<td>G74</td>
</tr>
<tr>
<td></td>
<td>Feed at X axis Δi</td>
<td>0.001mm</td>
<td>G75</td>
</tr>
<tr>
<td>Q</td>
<td>Thread finishing turning time m / thread retraction length r</td>
<td>time / 0.1 times thread lead / 0.001mm</td>
<td>G76</td>
</tr>
<tr>
<td></td>
<td>Angle between two nearby thread teeth a / thread tooth height k</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Feed at Z axis Δk</td>
<td>0.001 mm</td>
<td>G74</td>
</tr>
<tr>
<td>Q</td>
<td>Single radial cutting cycle at Z axis Δk</td>
<td>0.001 mm</td>
<td>G75</td>
</tr>
<tr>
<td></td>
<td>Minimum roughing thread Δdmin</td>
<td>0.001 mm</td>
<td>G76</td>
</tr>
<tr>
<td>F</td>
<td>Cutting feed speed</td>
<td>mm</td>
<td>G90 / G71 / G72</td>
</tr>
<tr>
<td></td>
<td>Thread lead in metric system F(I)</td>
<td>mm</td>
<td>G92 / G76</td>
</tr>
<tr>
<td>I</td>
<td>Thread teeth/inch in inch system</td>
<td></td>
<td>G92 / G76</td>
</tr>
</tbody>
</table>

Brief introduction of typical fixed cycle codes in ISO mode

For the meaning of letters when programming typical fixed cycles, please refer the figure on the left!

G90 shaft cutting cycle

Programming structure:
G90  X / U—Z / W—F;

Cone cutting
G90  X / U—Z / W—R—F;

Note: Please follow the specified structures when programming!
BASIC THEORY

G71 shaft roughing cycle

Programming structures:
G71  U(Δd) — R(e);
G71  P(ns) — Q(nf) — U(Δu) — W(Δw) — F— S— T
N(ns)…
... 
N(nf);

P(ns) / Q(nf): Indicating start/end point of
finishing program block path

Note: Please follow the specified structures when programming!

G71 sample program:

O0004:
G00 X200 Z10 M3 S800
G71 U2 R1
; each feed in 4 mm, retraction 2 mm
G71 P80 Q120 U0.5 W0.2 F200
; for a ~ e roughing, X axis allowance 1 mm
Z axis allowance 2 mm
N80 G00 X40 S1200
G01 Z-30 F100
; machine a→b
X60 W-30
; machine b→c
W-20
; machine c→d
N120 X100 W-10
; machine d→e
G70 P80 Q120
; finishing a→e
M30

G72 radical roughing cycle

Programming structures:
G72  W(Δd) — R(e);
G72  P(ns) — Q(nf) — U(Δu) — W(Δw) — F— S— T;
N(ns)…
... 
N(nf);

P(ns) / Q(nf): Indicating start/end point of
finishing program block path

Note: Please follow the specified structures when programming!

G72 sample program:

O0005:
G00 X176 Z10 M3 S500
G72 W2.0 R0.5
G72 P10 Q20 U0.2 W0.1 F300
N10 G00 X-55 S800
G01 X160 F120
X80 W20
; machine a→b
W15
; machine b→c
N20 X40 W20
; machine c→a
G70 P10 Q20
; finishing a→d
M30
G73 closed cutting cycle

Programming structures:
G73 U(Δi) — W(Δk) — R(d);
G73 P(ns) — Q(nf) — U(Δu) — W(Δw) — F — S — T;
N(ns)...
...
N(nf);

P(ns) / Q(nf): Indicating start/end point of finishing program block path

Note: Please follow the specified structures when programming!

G70 finishing cycle

Programming structures:
G70 P(ns) — Q(nf);
P(ns) / Q(nf): Indicating start/end point of finishing program block path

Note: T / S / F used in G70 must be specified in G71/G72/G73 fixed cycles before G70.

G94 radical cutting cycle

Programming structures:
G94 X / U — Z / W — F;
Cone face cutting
G94 X / U — Z / W — R — F;

G94 sample program:
O0003;
G00 X130 Z5 M3 S500
G73 P14 Q19 U0.5 W0.3 F0.3;
G01 W-20 F0.15 S600
X120 W-10
W-20
G02 X160 W-20 R20
N14 G00 X80 W-40
G01 W-20 F0.15 S600
X120 W-10
W-20
G02 X160 W-20 R20
N19 G01 X180 W-10
G70 P14 Q19 ; finishing
M30

Note: Please follow the specified structures when programming!

G73 sample program:
O0006;
G99 G00 X200 Z10 M3 S500
G73 U1.0 W1.0 R3;
; tool retraction at X axis 0.2 mm, at Z axis 1 mm
G73 P14 Q19 U0.5 W0.3 F0.3;
; roughing, keep 0.5 mm finishing allowance at X axis and 0.3 mm at Z axis
N14 G00 X80 W-40
G01 W-20 F0.15 S600
X120 W-10
W-20
G02 X160 W-20 R20
N19 G01 X180 W-10
G70 P14 Q19 ; finishing
M30

Note: Please follow the specified structures when programming!
G74 shaft grooving multi-cycles

Programming structures:
G74 R(e);
G74 X / U—Z / W—P(Δi)—Q(Δk)—R(Δd)—F;

Note: Please follow the specified structures when programming!

G74 sample program:
O007;
M3 S1500
G0 X40 Z5
G74 R0.5
; set each radical tool retraction 0.5 mm
G74 X20 Z60 P3000 Q5000 F50
; Z axis feed in 5 mm each time, tool retraction 0.5 mm, back to start point (Z5) after feeding to end point (Z60), then X axis feed in 3 mm, repeat the process till the end
M30

G75 radical grooving multi-cycles

Programming structures:
G75 R(e);
G75 X / U—Z / W—P(Δi)—Q(Δk)—R(Δd)—F;

Note: Please follow the specified structures when programming!

G75 sample program:
O008;
M3 S500
G0 X125 Z-20
G75 R0.5
; set each radical tool retraction 0.5 mm
G74 X40 Z-50 P6000 Q3000 F150
; X axis feed in 6 mm each time, tool retraction 0.5 mm, back to start point (X125) after feeding to end point (X40), then Z axis feed in 3 mm, repeat the process till the end
G0 X150 Z50
M30
**ISO Mode**

**BASIC THEORY**

**G92 thread cutting cycle**

Programming structures:

- Straight thread cutting cycle in mm
  
  G92 X / U—Z / W—F;

- Straight thread cutting cycle in inches
  
  G92 X / U—Z / W—I;

- Cone thread cutting cycle in mm
  
  G92 X / U—Z / W—R—F;

- Cone thread cutting cycle in inches
  
  G92 X /

**Note:** Please follow the specified structures when programming!

**G92 sample program:**

O0012;
M3 S1500
G0 X150 Z50 T0101; thread tool
G0 X65 Z5
G92 X58.7 Z-28 F3
; machining thread, divided into 4 cutting times, 1st cut: 1.3 mm
X57.7; 2nd cut: 1 mm
X57; 3rd cut: 0.7 mm
X56.9; 4th cut: 0.1 mm
M30

**G76 thread cutting multi-cycles**

Programming structures:

- G76 P(m)(r)(a)—Q(Δd_min)—R(d);
- G76 X / U—Z / W—R(i)—P(k)—Q(Δd)—F(I);

**Note:** Please follow the specified structures when programming!

**G76 sample program:**

O0013;
M3 S3000
G0 X150 Z50 T0101
G76 P020560 Q150 R0.1
; finishing repeat times 2, reverse width 0.5 mm, tool angle 60°,
minimum cutting depth 0.15 mm,
finishing allowance 0.1 mm
G76 X60.64 Z-62 P3680 Q1800 F6
; thread teeth height 3.68 mm, first thread cutting depth 1.8 mm,
thread lead 6 mm
G00 X100 Z50
M30

**G92 Example**

[Diagram of thread cutting cycle with dimensions and annotations]
Programming and Operating — Turning

**ISO Mode**

---

**BASIC THEORY**

**ISO program transfer and operation**

The ISO mode function provided by the 808D can easily operate the existing ISO program!

**Step 1** Transfer ISO files in USB device to 808D.

Connect the USB device with the stored target programs to the USB interface on the PPU.

Press the “USB” SK on the PPU.

Use the “Cursor + Select” keys to select the required program which is then highlighted.

Press the “Copy” SK on the PPU.

Press the “NC” SK on the PPU.

Press the “Paste” SK on the PPU.

A specified ISO program is then stored in the 808D system and can be edited and executed as described above.

**Step 2** Make the necessary changes to the ISO programs.

Programs in ISO mode in the 808D have their own rules. Suitable changes must be made at the appropriate positions so that you can run the ISO programs!

---

**Beginning of the program**

Common ISO programs:

- Beginning is “O”

ISO mode in 808D:

- Not compatible with programs beginning with “O”

**T code**

Common ISO programs:

- The default active tool offset number is same as the tool number

ISO mode in 808D:

- Tool active method T ΔΔ OO

No matter what the tool number is, the default active tool offset is 01

**Note:**

1. If you use the SKs on the PPU to activate ISO mode, you can use T0701 directly.
2. If you use G291 to activate ISO mode, you must set machine parameter MD10890=0 first and then you can use T0701.

No matter which way to activate ISO mode, the default active tool offset number is 01. If you want to use T0707 further, you must create tool edge number 7 in the 7th tool (each tool has a maximum of nine tool edges).
ISO Mode

BASIC THEORY

G90/G94 与 G71

ISO mode in 808D:
Must add relevant codes of G00/G01 between two cycles, or alarms will be displayed.

Common ISO programs:
Two cycles can be executed continuously.

Note: Alarm numbers 10255/15100/14082/10932 are available.

F / T / S in G71~G75

ISO mode in 808D:
F must be edited in the 2nd line.

Common ISO programs:
F position is optional.

Note: Alarm number 61812 is available.

F / T / S in G70

ISO mode in 808D:
Must be edited between G71 cycle blocks (N100~N200).

Common ISO programs:
① can be edited in G71 cycle blocks (N100 ~ N200)
② or can be edited in line G70.

Reversing angle and reversing circle

Common ISO programs:
ISO mode in 808D:

Linear reversing angle code: L

Circle reversing angle code: D

Note: If the L/D command is used in ISO mode in the 808D, the system will automatically skip the program line in which it lies without any operation.
Step 3  Program execution.

Make sure the current system is in ISO mode!
Make sure all preparations and safety measures have been performed!

Operate as described above.

Tool and workpiece setup → simulation → test → machining

Step 4  Transfer the ISO files in the 808D to the USB device.

Connect the USB device with sufficient memory to the USB interface on the PPU.

Press the “NC” SK on the PPU.

Use the “Cursor + Select” keys to select the required program which is then highlighted.

Press the “Copy” SK on the PPU.

Press the “USB” SK on the PPU.

Press the “Paste” SK on the

A specified ISO program is then stored in the USB and can be executed as required.

Step 5  Sample program (target workpiece is the same as in Section 5 “Create Part Program Part 2”).

ISO programs can be executed in the 808D as follows:

T3 M3 S500
G0 X22 Z4
G92 X20 Z-18 F2.5
X19
X18.5
X17
X16.8
X16.75
X16.75
G0 X50
Z50

T2 M3 S400 F0.2
G0 X32 Z-24
G75 R2
G75 X20 Z-31 P3000 Q3000
G0 X50
Z50
G0 X0 Z5

T5
M3 S500 F0.2
G74 R1
G74 X0 Z-21 P1000 Q5000 F0.2
G0 Z50
X50
G290
M30

Note: This program opens/exits ISO mode with the G291/G290 command. It is recommended to use the first method to open ISO mode — using the ISO mode active button on the PPU (described above).
### Group 1: Modally valid motion commands

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G00</td>
<td>Rapid traverse</td>
</tr>
<tr>
<td>G01</td>
<td>Linear interpolation</td>
</tr>
<tr>
<td>G02</td>
<td>Circular interpolation CW</td>
</tr>
<tr>
<td>G03</td>
<td>Circular interpolation CCW</td>
</tr>
<tr>
<td>CIP</td>
<td>Circular interpolation through intermediate point</td>
</tr>
<tr>
<td>CT</td>
<td>Circular interpolation: tangential transition</td>
</tr>
<tr>
<td>G33</td>
<td>Thread cutting with constant lead</td>
</tr>
<tr>
<td>G331</td>
<td>Thread interpolation</td>
</tr>
<tr>
<td>G332</td>
<td>Thread interpolation — Retraction</td>
</tr>
</tbody>
</table>

### Group 2: Non-modally valid motion. dwell

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G04</td>
<td>Dwell time preset</td>
</tr>
<tr>
<td>G63</td>
<td>Tapping without synchronization</td>
</tr>
<tr>
<td>G74</td>
<td>Reference point approach with synchronization</td>
</tr>
<tr>
<td>G75</td>
<td>Fixed point approach</td>
</tr>
<tr>
<td>G147</td>
<td>SAR - approach with a straight line</td>
</tr>
<tr>
<td>G148</td>
<td>SAR - retract with a straight line</td>
</tr>
<tr>
<td>G247</td>
<td>SAR - approach with a quadrant</td>
</tr>
<tr>
<td>G248</td>
<td>SAR - retract with a quadrant</td>
</tr>
<tr>
<td>G347</td>
<td>SAR - approach with a semicircle</td>
</tr>
<tr>
<td>G348</td>
<td>SAR - retract with a semicircle</td>
</tr>
</tbody>
</table>
### Group 3: Programmable frame

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>TRANS</td>
<td>Translation</td>
</tr>
<tr>
<td>ROT</td>
<td>Rotation</td>
</tr>
<tr>
<td>SCALE</td>
<td>Programmable scaling factor</td>
</tr>
<tr>
<td>MIRROR</td>
<td>Programmable mirroring</td>
</tr>
<tr>
<td>ATRANS</td>
<td>Additive translation</td>
</tr>
<tr>
<td>AROT</td>
<td>Additive programmable rotation</td>
</tr>
<tr>
<td>ASCALE</td>
<td>Additive programmable scaling factor</td>
</tr>
<tr>
<td>AMIRROR</td>
<td>Additive programmable mirroring</td>
</tr>
<tr>
<td>G110</td>
<td>Pole specification relative to the last programmed setpoint position</td>
</tr>
<tr>
<td>G111</td>
<td>Pole specification relative to origin of current workpiece coordinate system</td>
</tr>
<tr>
<td>G112</td>
<td>Pole specification relative to the last valid POLE</td>
</tr>
</tbody>
</table>

### Group 6: Plane selection

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G17</td>
<td>X/Y plane</td>
</tr>
<tr>
<td>G18 *</td>
<td>Z/X plane</td>
</tr>
<tr>
<td>G19</td>
<td>Y/Z plane</td>
</tr>
</tbody>
</table>

### Group 7: Tool radius compensation

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G40 *</td>
<td>Tool radius compensation OFF</td>
</tr>
<tr>
<td>G41</td>
<td>Tool radius compensation, left of contour</td>
</tr>
<tr>
<td>G42</td>
<td>Tool radius compensation, right of contour</td>
</tr>
</tbody>
</table>

### Group 8: Settable zero offset

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G500 *</td>
<td>Settable zero offset OFF</td>
</tr>
<tr>
<td>G54</td>
<td>1st settable zero offset</td>
</tr>
<tr>
<td>G55</td>
<td>2nd settable zero offset</td>
</tr>
<tr>
<td>G56</td>
<td>3rd settable zero offset</td>
</tr>
<tr>
<td>G57</td>
<td>4th settable zero offset</td>
</tr>
<tr>
<td>G58</td>
<td>5th settable zero offset</td>
</tr>
<tr>
<td>G59</td>
<td>6th settable zero offset</td>
</tr>
</tbody>
</table>

### Group 9: Frame suppression

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G53</td>
<td>Non-modal skipping of the settable zero offset</td>
</tr>
<tr>
<td>G153</td>
<td>Non-modal skipping of the settable zero offset including base frame suppression</td>
</tr>
</tbody>
</table>

### Group 10: Exact stop-continuous-path mode

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G60 *</td>
<td>Exact positioning</td>
</tr>
<tr>
<td>G64</td>
<td>Continuous-path mode</td>
</tr>
</tbody>
</table>

### Group 11: Exact stop, non-modal

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G09</td>
<td>Non-modal exact stop</td>
</tr>
</tbody>
</table>

### Group 12: Exact stop window modally effective

<table>
<thead>
<tr>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G601 *</td>
<td>Exact stop window</td>
</tr>
<tr>
<td>G602</td>
<td>G60, G9 course stop window</td>
</tr>
</tbody>
</table>
### Appendix

<table>
<thead>
<tr>
<th>Group 13: Workpiece measuring inch/metric</th>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G70</td>
<td>Inch dimension data input</td>
<td></td>
</tr>
<tr>
<td>G71 *</td>
<td>Metric dimension data input</td>
<td></td>
</tr>
<tr>
<td>G700</td>
<td>Inch dimension data input, also for feedrate F</td>
<td></td>
</tr>
<tr>
<td>G710</td>
<td>Metric dimension data input, also feedrate F</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Group 14: Absolute/incremental dimension modally effective</th>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G90 *</td>
<td>Absolute dimension data input</td>
<td></td>
</tr>
<tr>
<td>G91</td>
<td>Incremental dimension data input</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Group 15: Feedrate spindle feedrate modally effective</th>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G94</td>
<td>Feedrate F in mm/min</td>
<td></td>
</tr>
<tr>
<td>G95 *</td>
<td>Spindle feedrate in mm/r</td>
<td></td>
</tr>
<tr>
<td>G96</td>
<td>Constant cutting rate ON (in mm/r m/min)</td>
<td></td>
</tr>
<tr>
<td>G97</td>
<td>Constant cutting OFF</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Group 16: Feedrate override modally effective</th>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFC *</td>
<td>Feedrate override with circle ON</td>
<td></td>
</tr>
<tr>
<td>CFTCP</td>
<td>Feedrate override OFF</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Group 18: Behavior at corner when working with tool radius compensation</th>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G450 *</td>
<td>Transition circle</td>
<td></td>
</tr>
<tr>
<td>G451</td>
<td>Point intersection</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Group 44: Path segmentation with SAR modally effective</th>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G340 *</td>
<td>Approach and retraction in space (SAR)</td>
<td></td>
</tr>
<tr>
<td>G341</td>
<td>Approach and retraction in plane (SAR)</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Group 47: External NC languages modally effective</th>
<th>Name</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>G290 *</td>
<td>Siemens mode</td>
<td></td>
</tr>
<tr>
<td>G291</td>
<td>External mode</td>
<td></td>
</tr>
</tbody>
</table>
Technical Support

If you have any questions about this product or this manual, please contact the hotline:

<table>
<thead>
<tr>
<th>Phone</th>
<th>+86 1064 719990</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fax</td>
<td>+86 1064 719991</td>
</tr>
<tr>
<td>E-mail</td>
<td><a href="mailto:4008104288.cn@siemens.com">4008104288.cn@siemens.com</a></td>
</tr>
</tbody>
</table>

SINUMERIK internet address

Further product information can be found at the following website:

http://www.siemens.com/sinumerik
Everything ever wanted to know about SINUMERIK 808D:

Everything about shopfloor manufacturing:
www.siemens.com/cnc4you

Everything about the SINUMERIK Manufacturing Excellence portfolio of services:
www.siemens.com/sinumerik/manufacturing-excellence

Information about CNC training:
www.siemens.com/sinumerik/training

Siemens AG
Industry Sector
Motion Control Systems
P.O.Box 3180
91050 ERLANGEN
GERMANY

The information provided in this brochure contains merely general descriptions or characteristics of performance which in actual case of use do not always apply as described or which may change as a result of further development of the products. An obligation to provide the respective characteristics shall only exist if expressly agreed in the terms of contract. All product designations may be trademarks or product names of Siemens AG or supplier companies whose use by third parties for their own purposes could violate the rights of the owners.

Subject to change without prior notice
Order No.:
Dispostelle 06311
WÜ/35557 WERK.52.2.01 WS 11113.0
Printed in Germany
© Siemens AG 2012