SIEMENS

SINUMERIK 802D

Turning
ISO Dialect T

Short Guide 09.2001 Edition

Valid for

Control Software Version
SINUMERIK 802D 1
SINUMERIK® Documentation

Printing history

Brief details of this edition and previous editions are listed below.

The status of each edition is shown by the code in the "Remarks" column.

**Status code in the "Remarks" column:**

A .... New documentation.
B .... Unrevised reprint with new order no.
C .... Revised edition with new status.

<table>
<thead>
<tr>
<th>Edition</th>
<th>Order No.</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>09.01</td>
<td>6FC5698-1AA60-0BP0</td>
<td>A</td>
</tr>
</tbody>
</table>

This manual is included in the documentation on CD-ROM (DOCONCD)

<table>
<thead>
<tr>
<th>Edition</th>
<th>Order No.</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>09.01</td>
<td>6FC5298-6CA00-0AG1</td>
<td>C</td>
</tr>
</tbody>
</table>

**Trademarks**

SIMATIC®, SIMATIC HMI®, SIMATIC NET®, SIROTEC®, SINUMERIK® and SIMODRIVE® are registered trademarks of the Siemens AG. Other product names used in this documentation might be trademarks which, if used by third parties, could infringe the rights of their owners.

Further information is available on the Internet under:
http://www.ad.siemens.de/sinumerik.

This publication was produced with Win Word V8.0 and Designer V7.0.

Other functions not described in this documentation might be executable in the control. This does not, however, represent an obligation to supply such functions with a new control or when servicing.

Subject to change without prior notice.

The reproduction, transmission or use of this document or its contents is not permitted without written authority. Offenders will be liable for damages. All rights, including rights created by patent grant or registration of a utility model or design, are reserved.

© Siemens AG, 2001. All rights reserved
Introduction

**How to use this document**
This document is a short guide describing all the important operating and programming steps.

For detailed descriptions of the operating and programming of SINUMERIK 802D, refer to:
- User Manual, Turning, Order No. 6FC5698-2AA00-0BP0
- User Manual, Milling, Order No. 6FC5698-2AA10-0BP0

**Method of description**
The method of description is as follows:

Operating
- Prerequisite
- Operating sequence

Programming
- Programming the function
- Meaning of the parameters
- Descriptive picture with an example of a workpiece
# Table of Contents

1. Setup 1-7
   - Activate ISO Dialect T, G291 ............................................1-8
   - Tool Offsets .................................................................1-9

2. Create/Edit Program 2-11
   - Create/Open Program .....................................................2-12
   - Insert/Edit Block ............................................................2-13
   - Copy/Insert/Delete Block ...............................................2-14
   - Block Search/Numbering ...............................................2-15
   - Start/Simulate Program .................................................2-16

3. Execute/Correct Program 3-17
   - Select/Trace Program .....................................................3-18
   - Correct Program .............................................................3-19
   - Block Search ..................................................................3-20

4. Program Positional Data 4-21
   - Absolute Dimension, Incremental Dimension ..................4-22

5. Program Axis Motions 5-25
   - Rapid Traverse, G0; Linear Interpolation, G1 .................5-26
   - Circular Interpolation, G2/G3 .........................................5-27
   - Thread Cutting, G32 .......................................................5-29
   - Contour Definitions: A, C, R ..........................................5-30

6. Tool Offsets 6-31
   - Call Tool ........................................................................6-32
   - Tool Nose Radius Offset, G41/G42 ..................................6-33

7. Program Preparatory Functions 7-35
   - Program Feed, G94 to G99 .............................................7-36
   - Program Spindle Motion ...................................................7-37
   - Subroutine Call, M98/M99 ..............................................7-38

8. Appendix 8-39
   - List of the M Commands ..................................................8-40
   - List of the G Functions ....................................................8-41
   - Cycle Alarms .................................................................8-43
   - Notes ............................................................................8-44

© Siemens AG, 2001. All rights reserved
SINUMERIK 802D Turning ISO Dialect T (ISD) - 09.01 Edition
1. Setup

Activate ISO Dialect T, G291 1-8

Tool Offsets 1-9
Activate ISO Dialect T, G291

<table>
<thead>
<tr>
<th>N10</th>
<th>G291</th>
</tr>
</thead>
<tbody>
<tr>
<td>G291</td>
<td>Activate ISO dialect T NC programming language</td>
</tr>
<tr>
<td>G290</td>
<td>Activate SIEMENS NC programming language</td>
</tr>
</tbody>
</table>

Machine OEM

Please observe the details supplied by the machine OEM before switching on the power and when switching from the Siemens programming language into the ISO dialect programming language.

- The active tool,
- the tool offsets, and
- zero offsets

are retained when the ISO dialect programming language is active.

ISO dialect T

The "ISO Dialect T" NC programming language is a second programming language with a modified G Code command set.

Note

Only the program commands for ISO dialect T, Version A are described in this description. Any differences to ISO dialect T, Version B or C are indicated.
Tool Offsets

Select
OFFSET PARAM

Select OFFSET PARAM operating area.

Tool list

Select "Tool List" menu.

Functions

Delete tool
Delete tool offsets.

Search
Search for tool.

New tool
Create new tool.
Enter the new values.

Setting possibilities for the cutting edge position
## 2. Create/Edit Program

<table>
<thead>
<tr>
<th>Task</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create/Open Program</td>
<td>2-12</td>
</tr>
<tr>
<td>Insert/Edit Block</td>
<td>2-13</td>
</tr>
<tr>
<td>Copy/Insert/Delete Block</td>
<td>2-14</td>
</tr>
<tr>
<td>Block Search/Numbering</td>
<td>2-15</td>
</tr>
<tr>
<td>Start/Simulate Program</td>
<td>2-16</td>
</tr>
</tbody>
</table>
Create/Open Program

Create new program:
Select PROGRAM MANAGER operating area.
Select program directory.
Enter program name and confirm with OK.

Note:
The "SPF" file extension must be written explicitly for subroutines (e.g. TEST.SPF).

Open an existing program:
Select PROGRAM MANAGER operating area.
Select program directory.
Use the cursor to select the program in the program directory and open.

Note:
If the program is already open in the editor, it can be selected directly using the PROGRAM operating area key.
Insert/Edit Block

Insert new block

Prerequisite:
Existing program is open.

Use the cursor to select the line to be inserted.

Press the Input key.

Enter block.

Edit block

Prerequisite:
Existing program is open.

Select the block with the cursor and change it.

Note
If the program is already open in the editor, it can be selected directly using the PROGRAM operating area key.
Copy/Insert/Delete Block

Copy/insert

Prerequisite:
Existing program is open.

Use the cursor to select the required block or the position where the marking is to start.

Mark block

Enable marking mode (re-activation resets marking mode).

Use the cursor to select the end point of the marking.

Copy block

Copy the marked text into the clipboard.

Place the cursor at the required insertion point.

Insert block

Insert copied selection.

Note
Blocks can also be copied and inserted between different programs.

Delete

Prerequisite:
Existing program is open.

Use the cursor to select the required block or the position where the marking is to start.

Mark block

Enable marking mode.

Use the cursor to select the end point of the marking.

Delete block

Delete marked text.
Block Search/Numbering

Block search
Prerequisite: 
Existing program is open.

Enter search text.

You can choose between text 
or line number ("N..." must be 
entered for block number in the 
Text Search menu).

Start search.

Note
At the start of the search for text, it is possible to choose 
between
• Search from the cursor position, or
• Search from the block start.

Block numbering
Prerequisite: 
Program is open.

The block numbers of the 
complete program are 
renumbered in increments of 10.
Start/Simulate Program

Start program

Prerequisite:
Automatic mode is selected.
Existing program is open.

Select program to be executed.
NC start is used to start the program.

Simulate program
Select Simulation and start with NC-Start

Call submenu to show:
Show the complete workpiece (submenu of “Show…”).

Enlarge the size of the display.

Reduce the size of the display.

Select the start screen of the simulation.
Automatic scaling of the drawn tool path.

Change cursor increment.
Delete simulation display.

Return to edit modes.
3. Execute/Correct Program

Select/Trace Program 3-18
Correct Program 3-19
Block Search 3-20
## Select/Trace Program

<table>
<thead>
<tr>
<th>PROGRAM MANAGER</th>
<th>PROGRAMS</th>
<th>Execute</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Select PROGRAM MANAGER operating area. Select program directory.</td>
<td>Use the cursor to select the program in the program directory and select the program for execution.</td>
<td>Select &quot;Automatic&quot; mode.</td>
</tr>
<tr>
<td>Programs</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Note</td>
<td></td>
<td></td>
</tr>
<tr>
<td>At least the following conditions must be satisfied when the program is started:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>• No alarms pending.</td>
<td>• The feedrate enable is present.</td>
<td>• The spindle enable is present.</td>
</tr>
<tr>
<td>Trace machining on the screen</td>
<td>Possibly select the [M] POSITION operating area.</td>
<td>Start tracing.</td>
</tr>
<tr>
<td>Trace</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Note</td>
<td></td>
<td></td>
</tr>
<tr>
<td>As for the simulation, functions for various display settings are also available here (Zoom, To origin, ...).</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
## Correct Program

### NC stop

**Prerequisite:**
Program is being executed in Automatic.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Stop program" /></td>
<td>Stop program.</td>
</tr>
<tr>
<td><img src="image" alt="Program correction" /></td>
<td>Select Program correction.</td>
</tr>
<tr>
<td><img src="image" alt="Select block with the cursor and correct it" /></td>
<td>Select block with the cursor and correct it.</td>
</tr>
<tr>
<td><img src="image" alt="NC start is used to continue the program at the interrupt point" /></td>
<td>NC start is used to continue the program at the interrupt point.</td>
</tr>
</tbody>
</table>

### Notes
- After program interrupt (NC stop), the tool can be moved in manual operation (jog) away from the contour. The control stores the coordinates of the interrupt point.
- Corrections can only be made to those blocks that the control has not yet imported.

### NC reset

**Prerequisite:**
Program is being executed in Automatic.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Interrupt program" /></td>
<td>Interrupt program.</td>
</tr>
<tr>
<td><img src="image" alt="Program correction" /></td>
<td>Select Program correction.</td>
</tr>
<tr>
<td><img src="image" alt="Select block with the cursor and correct it" /></td>
<td>Select block with the cursor and correct it.</td>
</tr>
<tr>
<td><img src="image" alt="NC start is used to start the program at the beginning" /></td>
<td>NC start is used to start the program at the beginning</td>
</tr>
</tbody>
</table>

### Note
The control interrupts the execution should a system error occur in the parts program.
### Block Search

**Prerequisite:**
Program is selected in "Automatic" and is being executed.

- **Interrupt program.**
- **Select Block search.**
- **Possibly select the program level higher or lower.**
- **Select the block in the editor with the cursor or enter search text and start search.**
- **Enter changes.**
  - You have 2 possibilities for repositioning:
    - At the start of the contour
    - At the interrupt point.
  - Continue the program with NC start.

**Notice**
Tool changes are taken into consideration only when the tool is entered in the target block.
4. Program Positional Data

Absolute Dimension, Incremental Dimension 4-22
Absolute Dimension, Incremental Dimension

N 5 G0 X25 Z1
N10 G1 Z-7.5 F0.2
N20 G1 X40 Z-15
N30 G1 W-10
N40 G1 Z-35

The dimensioning is specified using the programming of the axis names:

\[ \text{X/Z} \quad \triangleq \quad \text{Absolute dimension} \]
\[ \text{U/W} \quad \triangleq \quad \text{Incremental dimension} \]

You can freely change between absolute and incremental dimension inputs from block to block.

Absolute and incremental dimensioning
**Absolute Dimension, Incremental Dimension**

**ISO systems B and C: G90, G91**

<table>
<thead>
<tr>
<th>Block</th>
<th>Command</th>
<th>Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>N 5</td>
<td>G0</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G90</td>
<td>X25 Z1</td>
</tr>
<tr>
<td>N 10</td>
<td>G1</td>
<td>Z 7.5 F0.2</td>
</tr>
<tr>
<td>N 20</td>
<td>G1</td>
<td>X 40 Z -15</td>
</tr>
<tr>
<td>N 30</td>
<td>G1 G91</td>
<td>Z -10</td>
</tr>
<tr>
<td>N 40</td>
<td>G1 G90</td>
<td>Z -35</td>
</tr>
</tbody>
</table>

- **G90**: Absolute dimension input; all values refer to the current workpiece zero offset.
- **G91**: Incremental dimension input; each dimension refers to the most recently entered contour point.

**Note**: G90, G91 apply in the block starting at the programmed location and not in the complete block.

Incremental dimension or absolute dimension in ISO dialect B or C
5. Program Axis Motions

Rapid Traverse, G0; Linear Interpolation, G1 5-26
Circular Interpolation, G2/G3 5-27
Thread Cutting, G32 5-29
Contour Definitions: A, C, R 5-30
Rapid Traverse, G0; Linear Interpolation, G1

N20 G0 X25 Z1
N30 G1 Z-7,5 F200
N... ... ...
N80 G0 X70 Z15

X, Z Coordinates of the target point
F Feedrate value

Fast positioning of the tool in rapid traverse during turning
Circular Interpolation, G2/G3

Programming the center point

N10  G0 X12 Z0
N20  G1 X40 Z-25 F0.2
N30  G3 X70 Z-75 I-3.335 K-29.25

X, Z  Coordinates of the circle end point
I, K  Interpolation parameters (directions: I in X, K in Z) to determine the circle center point

The tool travels in clockwise or counterclockwise direction for G2 and G3, respectively, viewed in the direction of the third coordinate axis.

Manufacturing a spherical bolt
Circular Interpolation, G2/G3

Program radius

N20 M3 S1000 G0 X68 Z102
N30 M3 S1000 G3 X20 Z150 R48 F5

X, Z, End point value
R Circle radius
F Feedrate value

Notice
Radius programming is not permitted for a traversal angle of 360°.

Radius programming in accordance with the drawing
Thread Cutting, G32

N20  G32 Z22 K2
Z, X  Thread end point
K  Pitch for cylinder thread
I  Pitch for face thread
I  Pitch for taper thread (taper angle > 45°)
K  Pitch for taper thread
   (taper angle < 45°)
SF  Start point offset in degrees

Right-hand or left-hand threads are programmed by specifying the direction of spindle rotation M3/M4. The direction of spindle rotation and speed must be programmed in the block prior to G32.

To program taper threads, enter the X and Z coordinates for G32. Multiple-start threads can be programmed with offset start points (SF=...).

Note
The G command is G33 in the ISO dialect, version B/C.
Contour Definitions: A, C, R

N20 A140 C7.5
N30 X80 Z70 A95 R10
N40 X70 Z50

A Angle of the first or second straight line relative to the 1st axis (Z)
C Chamfer
R Rounded corner
X1, Z1 Initial coordinates of the first straight line
X2, Z2 End point coordinates of the first straight line or start point of the second straight line
X3, Z3 End point coordinates of the second straight line or start point of the third straight line
X4, Z4 End point coordinates of the third straight line

The intersection point of the straight lines can be made as a corner, rounded corner or chamfer. The end point of the third straight line must always be programmed using Cartesian coordinates.
6. Tool Offsets

Call Tool 6-32

Tool Nose Radius Offset, G41/G42 6-33
N10 Txx01

Call tool with the number xx and the offset number 01.

Offset value for tool nose for the plunge-cutter
### Tool Nose Radius Offset, G41/G42

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>G41</td>
<td>Call the radius offset; tool in travel direction at the left-hand side of the contour</td>
</tr>
<tr>
<td>G42</td>
<td>Call the radius offset; tool in travel direction at the right-hand side of the contour</td>
</tr>
<tr>
<td>G40</td>
<td>Deselect the radius offset</td>
</tr>
</tbody>
</table>

At least one axis of the selected working plane must be programmed in the NC block with G40/G41/G42.

G0 or G1 must be used to select and deselect the offset in a program block. The offset acts only in the programmed working plane.

a = without tool nose radius offset  
b = with tool nose radius offset

![Diagram](Image)

Tool nose radius offset for the machining of inclinations and circular arcs
## 7. Program Preparatory Functions

<table>
<thead>
<tr>
<th>Function</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Program Feed, G94 to G99</td>
<td>7-36</td>
</tr>
<tr>
<td>Program Spindle Motion</td>
<td>7-37</td>
</tr>
<tr>
<td>Subroutine Call, M98/M99</td>
<td>7-38</td>
</tr>
</tbody>
</table>
Program Feed, G94 to G99

N5  G90  G00 X... Y... Z...
N10  G98  F500  G01...M3

G98 F  Constant speed in rpm and feed in mm/min
       (ISO dialect, version B/C: G94)
G99 F  Constant speed in rpm and feed in
       mm/revolution (ISO dialect, version B/C: G95)
G96 S  Constant cutting speed in m/min, and
       F  Feedrate in mm/revolution
G97  Switch off G96, save the last speed setpoint of
       G96 as constant speed.

The machine OEM specifies the maximum values for feed and speed values.

Control of the speed for constant cutting speed
**Program Spindle Motion**

N05 ...
N10 G1 F300 X70 Y20 **S270 M3**

- **S**: Spindle speed in rpm
- **M3**: Clockwise direction of rotation
- **M4**: Counterclockwise direction of rotation
- **M5**: Spindle stop
- **M19**: Spindle positioning

If the M commands are programmed in a block with axis motion, the commands act prior to the axis motion.

![Programming the spindle direction of rotation](image-url)
Subroutine Call, M98/M99

N20  M98 Pxxxxyyyy
N40  M99 Pxxxx

M98 Pxxxxyyyy Subroutine call: A subroutine with the number yyy is repeated xxx times.

M99 Pxxxx Subroutine end: Return to the main program at block number N....

The subroutine call must be made in a dedicated NC block.
## 8. Appendix

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>List of the M Commands</td>
<td>8-40</td>
</tr>
<tr>
<td>List of the G Functions</td>
<td>8-41</td>
</tr>
<tr>
<td>Cycle Alarms</td>
<td>8-43</td>
</tr>
<tr>
<td>Notes</td>
<td>8-44</td>
</tr>
</tbody>
</table>
## List of the M Commands

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>M0</td>
<td>Programmed stop</td>
</tr>
<tr>
<td>M1</td>
<td>Optional stop</td>
</tr>
<tr>
<td>M2</td>
<td>Program end (main program)</td>
</tr>
<tr>
<td>M30</td>
<td>Program end as for M2</td>
</tr>
<tr>
<td>M17</td>
<td>Subroutine end</td>
</tr>
<tr>
<td>M98</td>
<td>Subroutine call</td>
</tr>
<tr>
<td>M99</td>
<td>Subroutine end</td>
</tr>
<tr>
<td>M3</td>
<td>Clockwise rotating spindle</td>
</tr>
<tr>
<td>M4</td>
<td>Counterclockwise rotating spindle</td>
</tr>
<tr>
<td>M5</td>
<td>Spindle stop</td>
</tr>
<tr>
<td>M6</td>
<td>Tool change</td>
</tr>
<tr>
<td>M19</td>
<td>Spindle positioning</td>
</tr>
<tr>
<td>M70</td>
<td>Reserved for Siemens</td>
</tr>
<tr>
<td>M40</td>
<td>Automatic gearbox switching</td>
</tr>
<tr>
<td>M41</td>
<td>Gear stage 1</td>
</tr>
<tr>
<td>M42</td>
<td>Gear stage 2</td>
</tr>
<tr>
<td>M43</td>
<td>Gear stage 3</td>
</tr>
<tr>
<td>M44</td>
<td>Gear stage 4</td>
</tr>
<tr>
<td>M45</td>
<td>Gear stage 5</td>
</tr>
</tbody>
</table>

### Machine OEM

The machine OEM assigns M commands, for example with switching functions to control clamping devices or to activate/deactivate additional machine functions, etc. Please observe the details supplied by the machine OEM.
### List of the G Functions

<table>
<thead>
<tr>
<th>Version A/B/C</th>
<th>Function</th>
<th>M/S</th>
<th>Initial setting</th>
<th>Group</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>G0</td>
<td>Rapid traverse</td>
<td>M</td>
<td>X</td>
</tr>
<tr>
<td></td>
<td>G1</td>
<td>Linear interpolation</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G2</td>
<td>Circular interpolation in clockwise direction</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G3</td>
<td>Circular interpolation in counterclockwise direction</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G4 *)</td>
<td>Dwell time</td>
<td>S</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G10 *)</td>
<td>Load zero offset/tool offset</td>
<td>S</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G11 *)</td>
<td>End loading of zero offset/tool offset</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G15 *)</td>
<td>Select machining plane Z-X</td>
<td>M</td>
<td>X</td>
</tr>
<tr>
<td></td>
<td>G20/70</td>
<td>Input system in inches</td>
<td>M</td>
<td>X</td>
</tr>
<tr>
<td></td>
<td>G21/71</td>
<td>Metric input system</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G28 *)</td>
<td>Reference point approach</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G30 *)</td>
<td>Reference point 2nd, 3rd, 4th ref. point approach</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G31 *)</td>
<td>Measure using switching pushbutton</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G32/33</td>
<td>Thread cutting with constant pitch</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G34 *)</td>
<td>Tool radius offset OFF</td>
<td>M</td>
<td>X</td>
</tr>
<tr>
<td></td>
<td>G41</td>
<td>Tool radius offset to the left of the contour ON</td>
<td>S</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G42</td>
<td>Tool radius offset to the right of the contour ON</td>
<td>S</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G50/92</td>
<td>Set actual value memory</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G70/72</td>
<td>Finishing</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G71/73</td>
<td>Cutting longitudinal axis</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G72/74</td>
<td>Cutting traverse axis</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G73/75</td>
<td>Contour repetition</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G74/76</td>
<td>Drill deep-hole and plunge cutting in longitudinal axis</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G75/77</td>
<td>Drill deep-hole and plunge cutting in traverse axis</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G76 *)</td>
<td>Multiple thread cutting</td>
<td>M</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G80 *)</td>
<td>Cycle OFF</td>
<td>S</td>
<td></td>
</tr>
</tbody>
</table>
### List of the G Functions

<table>
<thead>
<tr>
<th>Version (A/B/C)</th>
<th>Function</th>
<th>M/S</th>
<th>Initial setting</th>
<th>Group</th>
</tr>
</thead>
<tbody>
<tr>
<td>G83 *)</td>
<td>Front face deep-hole drilling</td>
<td>M</td>
<td>9</td>
<td></td>
</tr>
<tr>
<td>G84 *)</td>
<td>Front face tapping</td>
<td>M</td>
<td>9</td>
<td></td>
</tr>
<tr>
<td>G85 *)</td>
<td>Front face drilling</td>
<td>S</td>
<td>9</td>
<td></td>
</tr>
<tr>
<td>G87 *)</td>
<td>Side deep-hole drilling</td>
<td>M</td>
<td>9</td>
<td></td>
</tr>
<tr>
<td>G88 *)</td>
<td>Side tapping</td>
<td>M</td>
<td>9</td>
<td></td>
</tr>
<tr>
<td>G89 *)</td>
<td>Side drilling</td>
<td>M</td>
<td>9</td>
<td></td>
</tr>
<tr>
<td>G90/77/20 *)</td>
<td>Outside-inside diameter simple – longitudinal turning cycle</td>
<td>M</td>
<td>18</td>
<td></td>
</tr>
<tr>
<td>G92/78/21 *)</td>
<td>Simple – Thread cutting</td>
<td>S</td>
<td>18</td>
<td></td>
</tr>
<tr>
<td>G94/79/24 *)</td>
<td>Simple – End face turning</td>
<td>M</td>
<td>18</td>
<td></td>
</tr>
<tr>
<td>G96</td>
<td>Constant cutting speed ON</td>
<td>M</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>G97</td>
<td>Constant cutting speed OFF</td>
<td>M</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>G98/94/94</td>
<td>Feedrate in mm/min, inch/min</td>
<td>M</td>
<td>5</td>
<td></td>
</tr>
<tr>
<td>G99/95/95</td>
<td>Feedrate in mm/rev, inch/rev</td>
<td>M</td>
<td>5</td>
<td></td>
</tr>
<tr>
<td>G–/98/98 *)</td>
<td>Return to starting point for fixed cycles</td>
<td>M</td>
<td>10</td>
<td></td>
</tr>
<tr>
<td>G–/99/99 *)</td>
<td>Return to point R for fixed cycles</td>
<td>M</td>
<td>10</td>
<td></td>
</tr>
<tr>
<td>G290</td>
<td>Deselect ISO Dialect programming</td>
<td>M</td>
<td>31</td>
<td></td>
</tr>
<tr>
<td>G291</td>
<td>Select ISO Dialect programming</td>
<td>M</td>
<td>31</td>
<td></td>
</tr>
<tr>
<td>G–/90/90</td>
<td>Absolute programming</td>
<td>M</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>G–/91/91</td>
<td>Incremental programming</td>
<td>M</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Subroutine call: Refer to M98</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Subroutine end: Refer to M99</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*) These commands are not described in this document
## Cycle Alarms

<table>
<thead>
<tr>
<th>Alarm no.</th>
<th>Alarm text</th>
<th>Explanation/Remedy</th>
</tr>
</thead>
<tbody>
<tr>
<td>61003</td>
<td>No feed programmed in the cycle</td>
<td>Remedy: Program feed</td>
</tr>
<tr>
<td>61102</td>
<td>No spindle direction programmed</td>
<td>Remedy: Program spindle direction</td>
</tr>
</tbody>
</table>
| 61800     | ISO dialect NC programming language has not been activated.  
Turn has not been activated for G50/51 polygon turning (cycle 3512). | Remedy: Set MD 10880 MM_EXTERN_CNC_SYSTEM to 1.  
Remedy: Set MD 10880 MM_EXTERN_CNC_SYSTEM to 2. |
| 61801     | Incorrect or undefined G Code selected. | Remedy: Set correct G Code |
| 61802     | Programming error for G28: an axis programmed in the block is a spindle. | Remedy: Change program appropriately. |
| 61803     | Programming error for G28: programmed axis has not been defined in MD or does not exist.  
Note: Because a max. of 5 axes can be defined for SINUMERIK 802D, the cycle cannot find axes when more have been defined in the MDs. | Remedy: Change program or define axis in the MD |
| 61805     | Only for ISO dialect A: X and U, Z and W, Y and V or C and H have been programmed at the same time. | Remedy: Change program appropriately. |
| 61808     | Final drilling depth or single drilling depth not programmed | Remedy: Change program appropriately. |
| 61812     | Programming error for G50/51 polygon turning (cycle 3512):  
Value for P or Q has not been programmed or = 0 | Remedy: Change program appropriately. |
| 61814     | Programming error: calling the drilling cycles with polar coordinates (G15/G16) is not permitted. | Remedy: Change program appropriately. |
| 61816     | Programming error for G27: Reached position does not agree with the reference point. | Remedy: Deselect zero offsets, tool offsets and restart G27. |
Notes

You can enter your user-specific functions here.
To
SIEMENS AG
A&D MC BMS
P.O. Box 3180
D-91050 Erlangen
Germany
(Phone +49-180-5050-222 [Hotline]
Fax +49-9131-98-2176
E-mail: motioncontrol.docu@erlfsiemens.de)

<table>
<thead>
<tr>
<th>Suggestions</th>
<th>Corrections</th>
</tr>
</thead>
<tbody>
<tr>
<td>For Publication/Manual:</td>
<td>SINUMERIK 802D</td>
</tr>
<tr>
<td>Turning</td>
<td>ISO Dialect T</td>
</tr>
<tr>
<td>Short Guide</td>
<td>User Documentation</td>
</tr>
</tbody>
</table>

Suggestions and/or corrections