

SIEMENS

SINUMERIK

SINUMERIK 808D, SINUMERIK 808D ADVANCED

Programming and Operating Manual (Turning)

User Manual

Legal information

Warning notice system

This manual contains notices you have to observe in order to ensure your personal safety, as well as to prevent damage to property. The notices referring to your personal safety are highlighted in the manual by a safety alert symbol, notices referring only to property damage have no safety alert symbol. These notices shown below are graded according to the degree of danger.

 DANGER
indicates that death or severe personal injury will result if proper precautions are not taken.
 WARNING
indicates that death or severe personal injury may result if proper precautions are not taken.
 CAUTION
indicates that minor personal injury can result if proper precautions are not taken.
NOTICE
indicates that property damage can result if proper precautions are not taken.

If more than one degree of danger is present, the warning notice representing the highest degree of danger will be used. A notice warning of injury to persons with a safety alert symbol may also include a warning relating to property damage.

Qualified Personnel

The product/system described in this documentation may be operated only by **personnel qualified** for the specific task in accordance with the relevant documentation, in particular its warning notices and safety instructions. Qualified personnel are those who, based on their training and experience, are capable of identifying risks and avoiding potential hazards when working with these products/systems.

Proper use of Siemens products

Note the following:

 WARNING
Siemens products may only be used for the applications described in the catalog and in the relevant technical documentation. If products and components from other manufacturers are used, these must be recommended or approved by Siemens. Proper transport, storage, installation, assembly, commissioning, operation and maintenance are required to ensure that the products operate safely and without any problems. The permissible ambient conditions must be complied with. The information in the relevant documentation must be observed.

Preface

Applicable products

This manual is valid for the following control systems:

Control system	Software version
SINUMERIK 808D ADVANCED T (Turning)	V4.8.1: PPU16x.3 with spindle/feed servo system
SINUMERIK 808D (Turning)	V4.8.1: PPU141.3 with feed servo system

Documentation components and target audience

End-user documentation	Target audience
Programming and Operating Manual (Turning)	Programmers and operators of turning machines
Programming and Operating Manual (Milling)	Programmers and operators of milling machines
Programming and Operating Manual (ISO Turning/Milling)	Programmers and operators of turning/milling machines
Programming and Operating Manual (Manual Machine Plus (MM+), Turning)	Programmers and operators of turning machines
Diagnostics Manual	Mechanical and electrical designers, commissioning engineers, machine operators, and service and maintenance personnel
Manufacturer/service documentation	Target audience
Commissioning Manual	Installation personnel, commissioning engineers, and service and maintenance personnel
Function Manual	Mechanical and electrical designers, technical professionals
Parameter Manual	Mechanical and electrical designers, technical professionals
Service Manual	Mechanical and electrical designers, technical professionals, commissioning engineers, and service and maintenance personnel
Readme file	
Third-party software - Licensing terms and copyright information	

My Documentation Manager (MDM)

Under the following link you will find information to individually compile your documentation based on the Siemens content:

www.siemens.com/mdm

Standard scope

This manual only describes the functionality of the standard version. Extensions or changes made by the machine tool manufacturer are documented by the machine tool manufacturer.

Technical support

Country	Hotline ¹⁾	Further service contact information:
Germany	+49 911 895 7222	- Worldwide Web site: https://support.industry.siemens.com/cs/ww/en/ - Chinese Web site: http://www.siemens.com.cn/808D
China	+86 400 810 4288	

¹⁾ You can find more hotline information at the worldwide Web site given above.

EC Declaration of Conformity

The EC Declaration of Conformity for the EMC Directive can be found on the Internet at <http://www.siemens.com/automation/service&support>.

Here, enter the number "67385845" as the search term or contact your local Siemens office.

Table of contents

	Preface.....	2
1	Fundamental safety instructions	7
1.1	General safety instructions	7
1.2	Warranty and liability for application examples.....	7
1.3	Industrial security	8
2	Introduction.....	9
2.1	Panel Processing Units (PPUs).....	9
2.1.1	PPU versions.....	9
2.1.2	PPU control elements.....	9
2.1.3	Screen layout	11
2.2	Machine Control Panels (MCPs)	12
2.2.1	MCP versions	12
2.2.2	MCP control elements	12
2.3	Coordinate systems.....	15
2.4	Protection levels	16
3	Common notices for machining operations	19
4	Switching on and referencing	19
5	Setting the user interface language	20
6	Setting up tools.....	20
6.1	Creating a tool	20
6.2	Creating/changing a cutting edge	22
6.3	Activating the tool and the spindle.....	23
6.4	Assigning the handwheel.....	24
6.4.1	Assigning the handwheel through the MCP.....	24
6.4.2	Assigning the handwheel through the PPU	25
6.5	Measuring the tool manually.....	26
6.5.1	Measuring the tool manually (without the Y axis)	26
6.5.2	Measuring the tool manually (with the Y axis)	28
6.6	Verifying the tool offset result	31
7	Creating part programs	32
7.1	Creating a part program	32
7.2	Editing the part program	34
7.2.1	Using a standard program structure	34
7.2.2	Editing a part program	34
7.2.3	Understanding frequently used programming instructions.....	40
7.2.3.1	Dimension units in inch or mm (G70/G71).....	40
7.2.3.2	Definition of work offset (G54 to G59, G500, G90/G91)	40
7.2.3.3	Rapid traverse (G00).....	41
7.2.3.4	Tool and traverse (T, D, F, G94/G95, S, M3/M4, G01).....	41
7.2.3.5	Tool radius compensation (G40, G41/G42).....	42
7.2.3.6	Turning circles and arcs (G02/G03)	42
7.2.3.7	Fixed point approach (G74/G75)	43
7.2.3.8	Spindle control.....	44
7.2.3.9	Setting dwell time in the program (G04)	44
8	Program simulation.....	45
8.1	Simulation prior to machining of the workpiece	45

8.2	Simultaneous recording prior to machining of the workpiece	47
9	Machining the workpiece	48
9.1	Executing a part program	48
9.2	Executing specified blocks	49
9.3	Correcting a part program	50
9.4	Simultaneous recording during machining of the workpiece	50
9.5	Entering the tool wear offsets	51
10	Other common operations	54
10.1	Program control functions	54
10.1.1	Program test	54
10.1.2	Dry run feedrate	54
10.1.3	Conditional stop	55
10.1.4	Skip block	55
10.1.5	Single block mode	55
10.1.6	Rapid traverse override	55
10.1.7	Auxiliary function lock	55
10.2	Setting up the workpiece	56
10.2.1	Measuring the workpiece	56
10.2.2	Entering/modifying workpiece offsets	57
10.3	Additional method for measuring the tool	58
10.4	Entering/modifying the setting data	59
10.5	Setting R parameters	63
10.6	Setting user data	63
10.7	Setting the date and time	64
10.8	Configuring the operating area after startup	65
10.9	Executing/transferring a part program from external	65
10.9.1	Executing/transferring through the USB interface	65
10.9.1.1	Executing from external (through USB interface)	65
10.9.1.2	Transferring from external (through USB interface)	66
10.9.2	Executing/transferring through the Ethernet connection	67
10.9.2.1	Configuring the network drive	67
10.9.2.2	Executing from external (through Ethernet connection)	69
10.9.2.3	Transferring from external (through Ethernet connection)	70
10.9.3	Configuring the firewall	71
10.10	Extending/deactivating the CNC lock function	72
10.11	Data backup	74
10.11.1	Internal data backup	74
10.11.2	External data backup	75
10.11.2.1	External data backup in a data archive	75
10.11.2.2	External data backup of separate files	76
10.12	Pocket calculator	77
11	Programming principles	79
11.1	Fundamentals of programming	79
11.1.1	Program names	79
11.1.2	Program structure	79
11.2	Programming instructions	80
11.2.1	Programming dimensions	80
11.2.2	Absolute/incremental dimensioning: G90, G91, AC, IC	80
11.2.3	Dimensions in metric units and inches: G71, G70, G710, G700	81
11.2.4	Radius/diameter dimensions: DIAMOF, DIAMON, DIAM90	82
11.2.5	Programmable work offset: TRANS, ATRANS	83
11.2.6	Programmable scale factor: SCALE, ASCALE	86
11.2.7	Workpiece coordinate system- settable work offset: G54 to G59, G500, G53, G153	87

11.3	Linear interpolation.....	88
11.3.1	Linear interpolation with rapid traverse: G0.....	88
11.3.2	Feedrate F.....	89
11.3.3	Linear interpolation with feedrate: G1.....	89
11.4	Circular interpolation.....	90
11.4.1	Circular interpolation: G2, G3.....	90
11.4.2	Circular interpolation via intermediate point: CIP.....	93
11.4.3	Circle with tangential transition: CT.....	94
11.5	Thread cutting.....	94
11.5.1	Thread cutting with constant lead: G33.....	94
11.5.2	Programmable run-in and run-out path for G33: DITS, DITE.....	96
11.5.3	Thread cutting with variable lead: G34, G35.....	97
11.5.4	Thread interpolation: G331, G332.....	98
11.6	Fixed point approach.....	99
11.6.1	Fixed point approach: G75.....	99
11.6.2	Reference point approach: G74.....	100
11.7	Acceleration control and exact stop/continuous path.....	100
11.7.1	Exact stop/continuous-path control mode: G9, G60, G64.....	100
11.7.2	Acceleration pattern: BRISK, SOFT.....	102
11.7.3	Dwell Time: G4.....	103
11.8	The third axis.....	103
11.9	Spindle movements.....	104
11.9.1	Spindle speed S, directions of rotation.....	104
11.9.2	Spindle positioning.....	105
11.9.2.1	Spindle positioning (SPOS, SPOSA, M19, M70, WAITS).....	105
11.9.2.2	Spindle positioning (SPOS, SPOSA, M19, M70, WAITS): Further information.....	109
11.9.3	Gear stages.....	110
11.10	Special turning functions.....	110
11.10.1	Constant cutting rate: G96, G97.....	110
11.10.2	Rounding, chamfer.....	112
11.10.3	Contour definition programming.....	114
11.11	Tool and tool offset.....	116
11.11.1	General information (turning).....	116
11.11.2	Tool T (turning).....	116
11.11.3	Tool offset number D (turning).....	117
11.11.4	Selecting the tool radius compensation: G41, G42.....	120
11.11.5	Corner behavior: G450, G451.....	122
11.11.6	Tool radius compensation OFF: G40.....	122
11.11.7	Special cases of the tool radius compensation.....	123
11.11.8	Example of tool radius compensation (turning).....	124
11.11.9	Special handling of tool compensation (turning).....	124
11.12	Miscellaneous function M.....	125
11.13	H function.....	126
11.14	Arithmetic parameters, LUD and PLC variables.....	126
11.14.1	Arithmetic parameter R.....	126
11.14.2	Local User Data (LUD).....	128
11.14.3	Reading and writing PLC variables.....	129
11.15	Program jumps.....	129
11.15.1	Unconditional program jumps.....	129
11.15.2	Conditional program jumps.....	130
11.15.3	Program example for jumps.....	131
11.15.4	Jump destination for program jumps.....	132
11.16	Subroutine technique.....	133
11.16.1	General information.....	133
11.16.2	Calling machining cycles (turning).....	134
11.16.3	Executing internal and external subroutines (CALL, EXTCALL).....	135
11.17	Timers and workpiece counters.....	136
11.17.1	Runtime timer.....	136
11.17.2	Workpiece counter.....	138

11.18	Kinematic transformation	140
11.18.1	Face milling on turned parts (TRANSMIT)	140
11.18.2	Cylinder surface transformation (TRACYL)	142
12	Cycles	149
12.1	Overview of cycles	149
12.2	Programming cycles.....	149
12.3	Graphical cycle programming in the program editor.....	150
12.4	Drilling cycles	151
12.4.1	General information.....	151
12.4.2	Requirements.....	152
12.4.3	Drilling, centering - CYCLE81	154
12.4.4	Drilling, counterboring - CYCLE82	156
12.4.5	Deep-hole drilling - CYCLE83	158
12.4.6	Rigid tapping - CYCLE84	162
12.4.7	Tapping with compensating chuck - CYCLE840	166
12.4.8	Reaming1 - CYCLE85.....	170
12.4.9	Boring - CYCLE86.....	172
12.5	Turning cycles.....	174
12.5.1	Requirements.....	174
12.5.2	Cutoff - CYCLE92	176
12.5.3	Groove - CYCLE93	178
12.5.4	Undercut (forms E and F to DIN) - CYCLE94	184
12.5.5	Cutting with relief cut - CYCLE95.....	188
12.5.6	Thread undercut - CYCLE96.....	200
12.5.7	Thread chaining - CYCLE98	204
12.5.8	Thread cutting - CYCLE99	209
12.6	Error messages and error handling.....	216
12.6.1	General Information	216
12.6.2	Error handling in the cycles.....	216
12.6.3	Overview of cycle alarms	216
12.6.4	Messages in the cycles	216
13	Machining typical turning parts.....	217
13.1	Operations before programming	217
13.2	Programming for machining a spherical surface (Example 1).....	217
13.3	Programming for machining contours (Example 2)	222
13.4	Programming for roughing, grooving and threading (Example 3).....	228
13.5	Programming for roughing and finishing (Example 4)	236
13.6	Program simulation and execution	241
A	Appendix	242
A.1	Operating area overview	242
A.1.1	Machining operating area.....	242
A.1.2	Program editing operating area.....	242
A.1.3	Offset operating area	243
A.1.4	Program management operating area	244
A.1.5	System data operating area	247
A.1.6	Alarm operating area.....	248
A.2	Operating mode overview	249
A.2.1	"AUTO" mode.....	249
A.2.2	"JOG" mode	251
A.2.2.1	Running the spindle manually	252
A.2.2.2	Executing M functions	253
A.2.2.3	Setting the relative coordinate system (REL)	254
A.2.2.4	Setting the JOG data.....	254
A.2.3	"MDA" mode.....	255
A.3	Activating the contour handwheel via the NC program	256
A.4	The help system.....	258

A.5	Operation wizard	260
A.6	Editing Chinese characters.....	261
A.7	Calculating contour elements	262
A.8	Free contour programming	266
A.8.1	Defining a start point.....	268
A.8.2	Programming contour element	269
A.8.3	Parameters for contour elements	272
A.8.4	Undercuts for turning technology.....	275
A.8.5	Specifying contour elements in polar coordinates	275
A.8.6	Cycle support	278
A.8.7	Programming example for turning application	278
A.9	Word structure and address	282
A.10	Character set.....	283
A.11	Block format	284
A.12	List of instructions.....	285

1 Fundamental safety instructions

1.1 General safety instructions

 WARNING
Danger to life if the safety instructions and residual risks are not observed If the safety instructions and residual risks in the associated hardware documentation are not observed, accidents involving severe injuries or death can occur. • Observe the safety instructions given in the hardware documentation. • Consider the residual risks for the risk evaluation.

 WARNING
Malfunctions of the machine as a result of incorrect or changed parameter settings As a result of incorrect or changed parameterization, machines can malfunction, which in turn can lead to injuries or death. • Protect the parameterization (parameter assignments) against unauthorized access. • Handle possible malfunctions by taking suitable measures, e.g. emergency stop or emergency off.

1.2 Warranty and liability for application examples

The application examples are not binding and do not claim to be complete regarding configuration, equipment or any eventuality which may arise. The application examples do not represent specific customer solutions, but are only intended to provide support for typical tasks. You are responsible for the proper operation of the described products. These application examples do not relieve you of your responsibility for safe handling when using, installing, operating and maintaining the equipment.

1.3 Industrial security

Note

Industrial security

Siemens provides products and solutions with industrial security functions that support the secure operation of plants, systems, machines and networks.

In order to protect plants, systems, machines and networks against cyber threats, it is necessary to implement – and continuously maintain – a holistic, state-of-the-art industrial security concept. Siemens products and solutions only represent one component of such a concept.

The customer is responsible for preventing unauthorized access to its plants, systems, machines and networks. Systems, machines and components should only be connected to the enterprise network or the internet if and to the extent necessary and with appropriate security measures (e.g. use of firewalls and network segmentation) in place.

Additionally, Siemens' guidance on appropriate security measures should be taken into account. For more information about industrial security, please visit:

Industrial security (<http://www.siemens.com/industrialsecurity>).

Siemens' products and solutions undergo continuous development to make them more secure. Siemens strongly recommends to apply product updates as soon as available and to always use the latest product versions. Use of product versions that are no longer supported, and failure to apply latest updates may increase customer's exposure to cyber threats.

To stay informed about product updates, subscribe to the Siemens Industrial Security RSS Feed at:

Industrial security (<http://www.siemens.com/industrialsecurity>).

WARNING

Unsafe operating states resulting from software manipulation

Software manipulations (e.g. viruses, trojans, malware or worms) can cause unsafe operating states in your system that may lead to death, serious injury, and property damage.

- Keep the software up to date.
- Incorporate the automation and drive components into a holistic, state-of-the-art industrial security concept for the installation or machine.
- Make sure that you include all installed products into the holistic industrial security concept.
- Protect files stored on exchangeable storage media from malicious software by with suitable protection measures, e.g. virus scanners.

2 Introduction

Note

Chapter "Machining typical turning parts (Page 217)" later in this manual provides you with typical machining examples including workpiece drawings, typical programs, and step-by-step operating instructions. It helps you to quickly understand CNC programming and operation through practical examples.

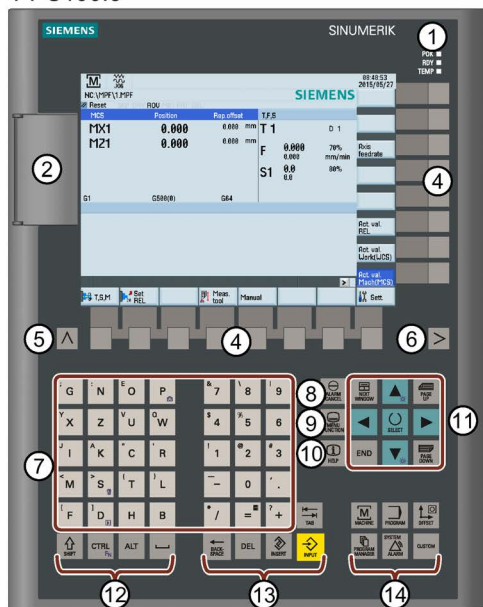
2.1 Panel Processing Units (PPUs)

2.1.1 PPU versions

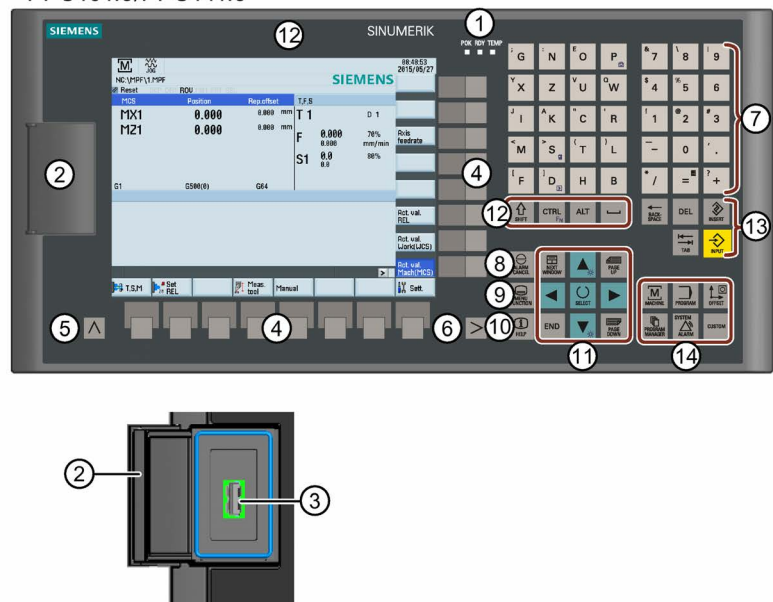
PPU version	Technology variant	Operator panel variant	Applicable control system
PPU161.3	Turning variant	Horizontal panel, with English keys	SINUMERIK 808D ADVANCED T (Turning)
		Horizontal panel, with Chinese keys	
	Milling variant	Horizontal panel, with English keys	SINUMERIK 808D ADVANCED M (Milling)
		Horizontal panel, with Chinese keys	
PPU160.3	Turning variant	Vertical panel, with English keys	SINUMERIK 808D ADVANCED T (Turning)
		Vertical panel, with Chinese keys	
	Milling variant	Vertical panel, with English keys	SINUMERIK 808D ADVANCED M (Milling)
		Vertical panel, with Chinese keys	
PPU141.3	Turning variant	Horizontal panel, with English keys	SINUMERIK 808D (Turning)
		Horizontal panel, with Chinese keys	
	Milling variant	Horizontal panel, with English keys	SINUMERIK 808D (Milling)
		Horizontal panel, with Chinese keys	

2.1.2 PPU control elements

PPU160.3



PPU161.3/PPU141.3



- | | |
|---|---|
| <p>① Status indicators</p> <p>② Protective cover for USB interface</p> <p>③ USB interface</p> <p>④ Horizontal and vertical keys
Call the corresponding vertical/horizontal softkeys on the screen</p> <p>⑤ Return key
Returns to the next higher-level menu</p> <p>⑥ Menu extension key
Calls the extended menu items</p> <p>⑦ Alphabetic and numeric keys</p> | <p>⑧ Alarm cancellation key
Cancels alarms and messages that are marked with the symbol on this key</p> <p>⑨ Wizard key
Guides you through performing basic commissioning and operation tasks</p> <p>⑩ Help key
Calls help information</p> <p>⑪ Cursor keys</p> <p>⑫ Modifier keys for use in key combinations</p> <p>⑬ Edit control keys</p> <p>⑭ Operating area keys</p> |
|---|---|

Additional information about keys

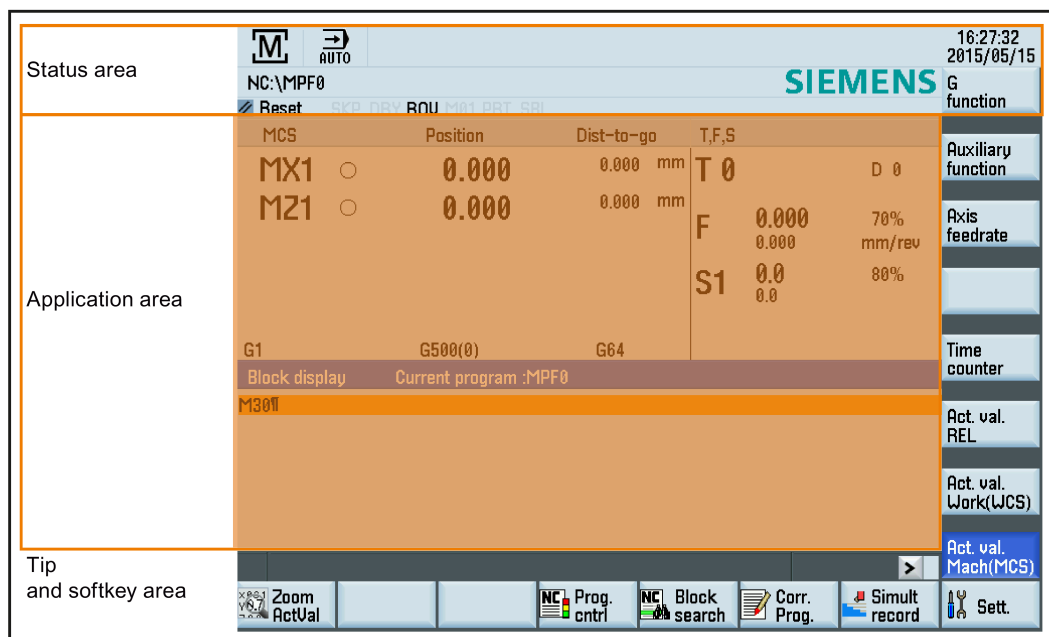
	<CTRL + P>: Creates a screenshot of the actual user interface
	<CTRL + S>: Saves startup archives and action logs to a USB stick
	<CTRL + D>: Displays pre-defined slides on the screen <ALT + D>: Saves action logs to a USB stick
	Calls the calculator function when you position the cursor on the desired input field
	<CTRL + cursor key>: Changes the screen backlight brightness by an increment of 10% (brightness range: 10% to 100%) Note that after you release the key combination, the HMI saves your brightness setting and the brightness bar disappears after a few seconds.
	Enters the upper character shown on the alphabetic/numeric keys concerned
	Opens the system data operating area
	In the program editor, pressing this key moves the cursor to the beginning of the selected program block.
	• Toggles between entries in the input field • Enters the "Set-up menu" dialog box at NC startup
	Enables user-defined extension applications, for example, generation of user dialog boxes with the EasyXLanguage function. For more information about this function, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Function Manual.

Status indicators



Indicator	Status and meaning
"POK"	Solid green: The power supply for the control system is switched on.
"RDY"	- Solid green: The control system is ready and the PLC is in run mode. - Solid red: The control system is in stop mode. - Solid orange: The PLC is in stop mode. - Flashing orange: The PLC is in power-up mode.
"TEMP"	- Off: The temperature of the control system is within the specified scope. - Solid orange: The temperature of the control system is beyond the scope.

2.1.3 Screen layout



Alarms and messages in the status area

Alarm and message area	
400006 ↓ 02 Loss of remanent PLC data ---	Displays active alarms with alarm text The alarm number is displayed in white lettering on a red background. The associated alarm text is shown in red lettering. An arrow indicates that several alarms are active. The number to the right of the arrow indicates the total number of active alarms. When more than one alarm is active, the display scrolls through the alarms in sequence. An acknowledgement symbol indicates the alarm cancel criterion.
READY TO START	Displays messages from NC programs Messages from NC programs do not have numbers and appear in green lettering.

2.2 Machine Control Panels (MCPs)

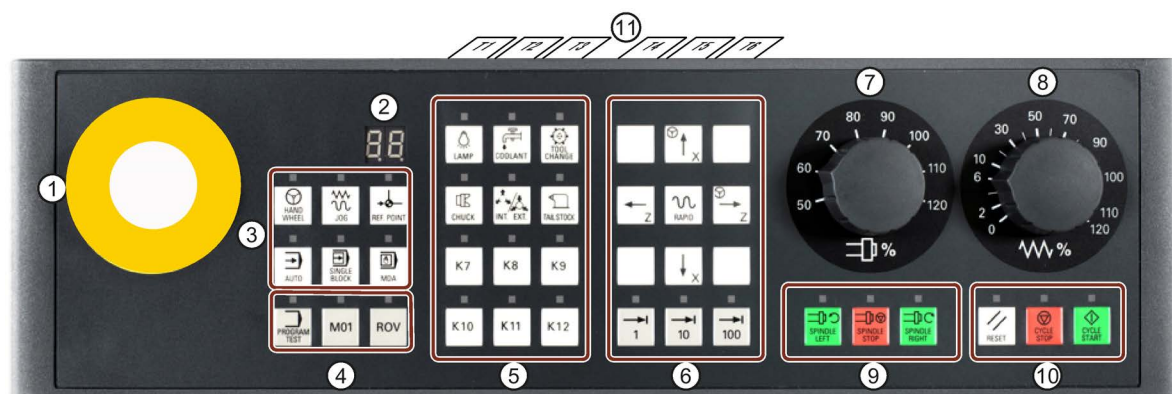
2.2.1 MCP versions

MCP version	Applicable control system
- Horizontal MCP, with English keys and override switches - Horizontal MCP, with Chinese keys and override switches - Vertical MCP, with English keys and a reserved slot for the handwheel - Vertical MCP, with Chinese keys and a reserved slot for the handwheel - Vertical MCP, with English keys and an override switch for the spindle - Vertical MCP, with Chinese keys and an override switch for the spindle	All MCP variants are applicable to any of the following control systems: - SINUMERIK 808D ADVANCED T (Turning) - SINUMERIK 808D ADVANCED M (Milling) - SINUMERIK 808D (Turning) - SINUMERIK 808D (Milling)
Horizontal MCP, with Chinese keys and a reserved slot for the handwheel	This MCP variant is applicable to the following control systems: - SINUMERIK 808D ADVANCED T (Turning) - SINUMERIK 808D (Turning)

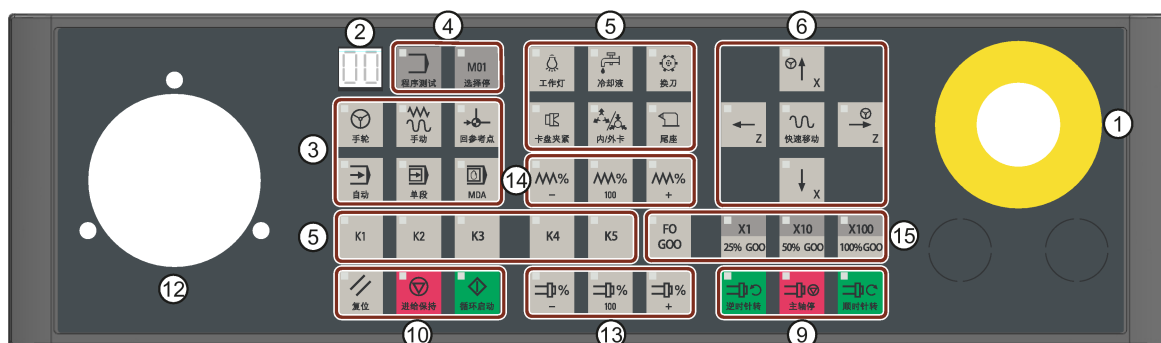
2.2.2 MCP control elements

Horizontal MCP

- Variant with override switches

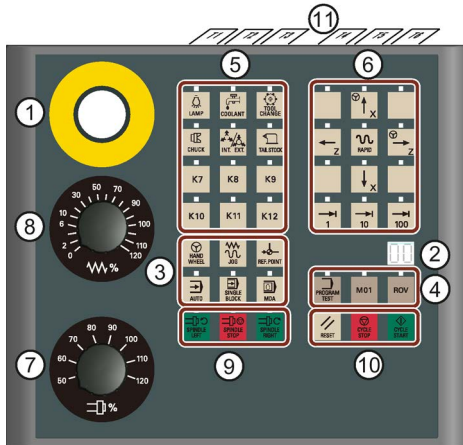


- Variant with a reserved slot for the handwheel

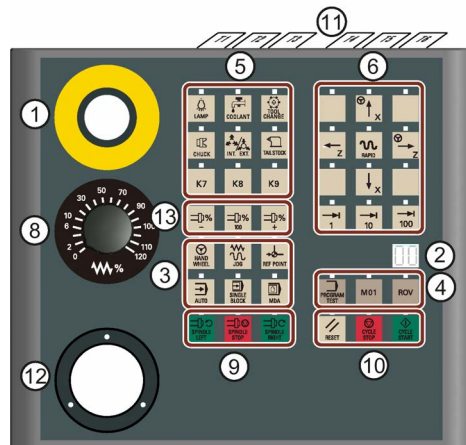


Vertical MCP

- Variant with an override switch for the spindle



- Variant with a reserved slot for the handwheel



- | | |
|---|--|
| ① Reserved hole for emergency stop button | ⑨ Spindle control keys |
| ② Tool number display
Displays the number of the currently active tool | ⑩ Keys for program start, stop, and reset |
| ③ Operating mode selection area | ⑪ Pre-defined labeling strips for keys of the MCP |
| ④ Program control keys | ⑫ Reserved slot for the handwheel |
| ⑤ User-defined keys | ⑬ Spindle override control keys |
| ⑥ Axis traversing keys | ⑭ Feedrate override control keys |
| ⑦ Spindle override switch | ⑮ Keys for increment override/rapid traverse override* |
| ⑧ Feedrate override switch | |

* The increment override can be activated in "JOG" mode, "Handwheel" mode, or "AUTO"/"MDA" mode with the contour handwheel activated. The rapid traverse override is effective only in "AUTO"/"MDA" mode with no contour handwheel activated. The "F0 G00" key can activate a rapid traverse override of 1% by default, which can be changed in the general machine data 12050[1], for example, 12050[1] = 0.15 as 15%.

Functions of user-defined keys

	Switches on/off the machine tool working lamp (always active independent of the machine operating mode) Indicator on: The working lamp on the machine tool is switched on. Indicator off: The working lamp on the machine tool is switched off.
	Switches on/off of the coolant supply (always active independent of the machine operating mode) Indicator on: The coolant supply is switched on. Indicator off: The coolant supply is switched off.
	Enables/disables the sequential tool changes (active only when the machine is in "JOG" mode) Indicator on: The machine starts sequential tool changes. Indicator off: The machine stops sequential tool changes.
	Controls the clamping/unclamping of the workpiece by the chuck (active only when the machine is in "JOG" mode and the spindle has stopped running) Indicator on: The chuck has clamped the workpiece. Indicator off: The chuck has unclamped the workpiece.

	Controls the internal/external clamping of the workpiece by the chuck (active only when the spindle has stopped running) Indicator on: The chuck has clamped the workpiece externally. Indicator off: The chuck has clamped the workpiece internally.
	Enables/disables the tailstock to hold the workpiece tightly (active only when the machine is in "JOG" mode) Indicator on: The tailstock moves towards the workpiece until it holds the workpiece tightly. Indicator off: The tailstock retracts from the workpiece to its original position.
K1 to K5, K7 to K12	Reserved keys. You can define the functions of these keys in PLC Programming Tool. For more information, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Commissioning Manual.

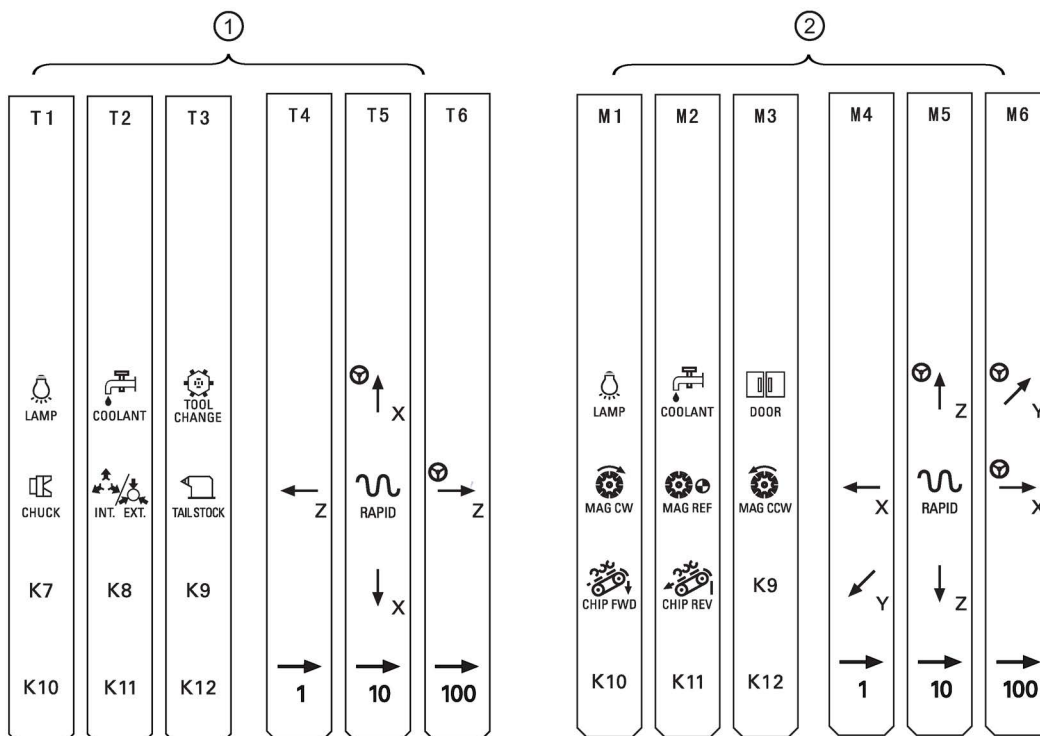
Pre-defined labeling strips for keys of the MCP

The MCP package includes two sets (six pieces each) of pre-defined labeling strips. One set (① as shown below) is for the turning variant of the control system and pre-inserted on the back of the MCP. The other set (② as shown below) is for the milling variant of the control system.

Note

For the horizontal MCP variant with a reserved slot for the handwheel, no pre-defined labeling strip is provided.

If your control system is a milling variant of the control system, replace the pre-inserted labeling strips with the milling-specific ones.



If the pre-defined labeling strips cannot meet your needs, proceed as follows to make customized labeling strips by using the A4-sized blank plastic sheet in the MCP package:

- Open the "Symbols for MCP customized key_808D" file in one of the following folders on the Toolbox DVD for the control system on your computer:
 - \examples\SINUMERIK_808D\MCP (for PPU141.3)
 - \examples\SINUMERIK_808D_ADVANCED\MCP (for PPU16x.3)
- Copy the desired symbols of keys from the file and then paste them to the desired locations for key labels in the "Strip_template_808D" file.

3. Print the final labeling strip template file on the blank plastic sheet.
4. Replace the pre-inserted labeling strips from the MCP with the customized ones.

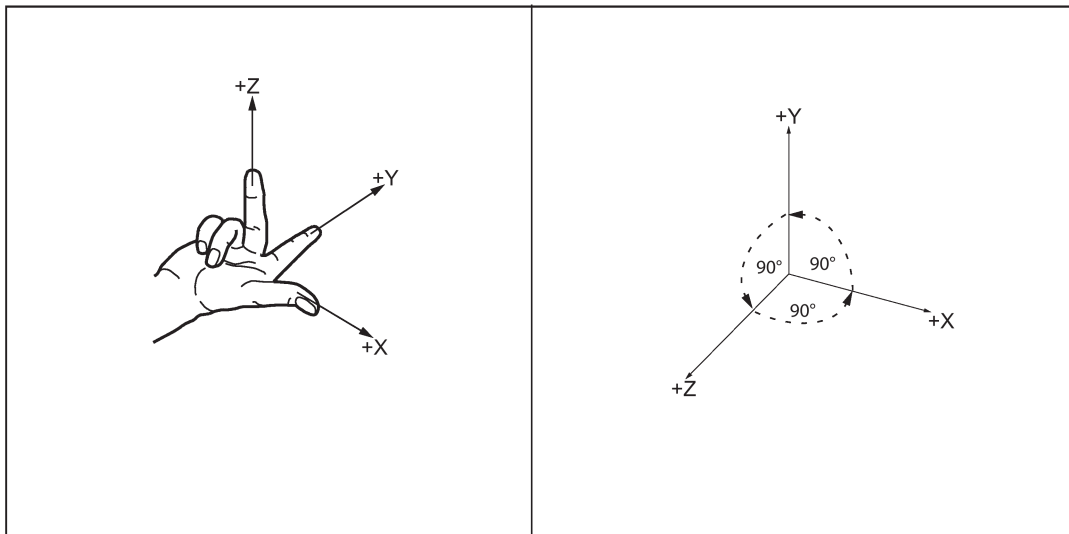
Note

The MCP information described in this manual is valid for the SINUMERIK 808D MCP.

2.3 Coordinate systems

As a rule, a coordinate system is formed from three mutually perpendicular coordinate axes. The positive directions of the coordinate axes are defined using the Cartesian coordinate system. The coordinate system is related to the workpiece and programming takes place independently of whether the tool or the workpiece is being traversed. When programming, it is always assumed that the tool traverses relative to the coordinate system of the workpiece, which is intended to be stationary.

The figure below illustrates how to determine the axis directions.

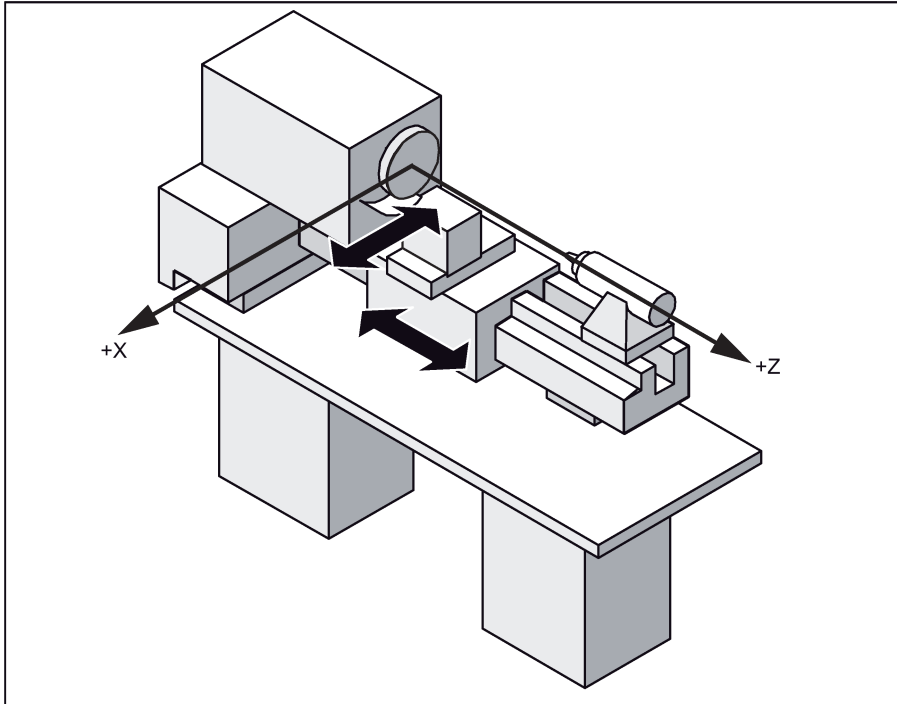


Machine coordinate system (MCS)

The orientation of the coordinate system relative to the machine depends on the machine type. It can be rotated in different positions.

The directions of the axes follow the Cartesian coordinate system. Seen from the front of the machine, the middle finger of the right hand points in the opposite direction to the infeed of the spindle.

The figure below shows an example of the machine coordinate system of a flat-body turning machine with a front tool holder.



The origin of this coordinate system is the machine zero, defined by the machine manufacturer.

Workpiece coordinate system (WCS)

WCS refers to the coordinate system which is used to describe the geometry of a workpiece in the workpiece program.

The origin of the coordinate system: the X axis is set at the rotating center of the workpiece while the Z axis is set at the surface of the workpiece.

Relative coordinate system (REL)

In addition to the machine and workpiece coordinate systems, the control system provides a relative coordinate system. This coordinate system can clear the current coordinate values at any position and has no influence on workpiece machining or the position of WCS. All axis movements are displayed relative to this position.

2.4 Protection levels

Overview

The control system provides a concept of protection levels for enabling data areas. Different protection levels control different access rights.

The control system delivered by Siemens is set by default to the lowest protection level 7 (without password). If the password is no longer known, the control system must be reinitialized with the default machine/drive data. All passwords are then reset to default passwords for this software release.

Note

Before you boot the control system with default machine/drive data, make sure that you have backed up your machine/drive data; otherwise, all data are lost after rebooting with default machine/drive data.

Protection level	Locked by	Area
0	Siemens password	Siemens, reserved
1	Manufacturer password	Machine manufacturers
2	Reserved	-

Protection level	Locked by	Area
3-6	End user password (Default password: "CUSTOMER")	End users
7	No password	End users

Protection level 1

Protection level 1 requires a manufacturer password. With this password entry, you can perform the following operations:

- Entering or changing part of the machine data and drive data
- Conducting NC and drive commissioning

Protection level 3-6

Protection level 3-6 requires an end user password. With this password entry, you can perform the following operations:

- Entering or changing part of the machine data
- Editing programs
- Setting offset values
- Measuring tools

Protection level 7

If you have not set the password or protection level interface signal, the system sets the protection level to 7 automatically. With this password entry, you can perform the following operations:

- Editing programs
- Setting offset values
- Measuring tools

You can also set protection level 7 through the PLC user interfaces. For more information, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Commissioning Manual.

In the menus listed below the input and modification of data depends on the set protection level:

- Tool offsets
- Work offsets
- Setting data
- Program creation/program correction

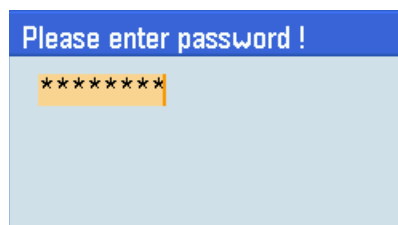
The number of machine data and drive data which can be read or modified depends on the protection level. You can set the protection level for these function areas with the display machine data (**USER_CLASS...**).

Setting the password



Set
password

1. Select the system data operating area.
2. Open the following password setting window and then enter the desired password (default end-user password: CUSTOMER):



Accept

3. Confirm the input.

Changing/deleting the password

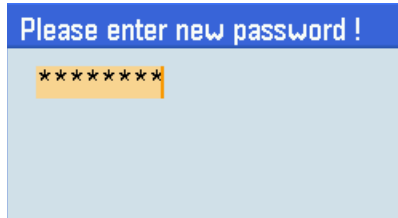
Note

To avoid unauthorized access to the controller, you must change the Siemens default passwords to your own ones.



Change
password

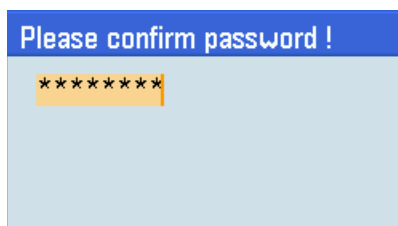
1. Select the system data operating area.
2. If you desire to change the existing password, press this softkey to open the following window and enter the new password:



If you desire to delete the existing password, proceed directly to Step 6.



3. Confirm the new password.
4. Enter the new password again in the following window:



5. Press this softkey to confirm the new password.

Delete
password

6. If you desire to delete the existing password, directly press this softkey.

3 Common notices for machining operations

To ensure the safety and correctness of machining, you must observe the following notices during machining operations:

- Reference point approach is required only for machine tools equipped with incremental encoders.
- Before running the spindle manually, make sure you have activated the tool.
- Before executing M functions, make sure all the axes are in safe positions.
- During handwheel assignment, make sure there is no obstacle when moving the tool to avoid tool collision.
- To guarantee the safety and correctness of machining, you must verify the tool setup result.
- You must distinguish the direction of tool wear compensation clearly before entering the tool wear offsets.
- Before machining the workpiece on the machine, make sure the following preconditions are met:
 - The PRT mode and the dry run feedrate are deactivated.
 - The feedrate override is 0%.
 - The safety door is closed.
- After changing important data, it is recommended to carry out an internal data backup immediately.

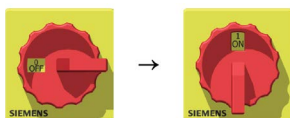
4 Switching on and referencing

Note

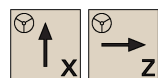
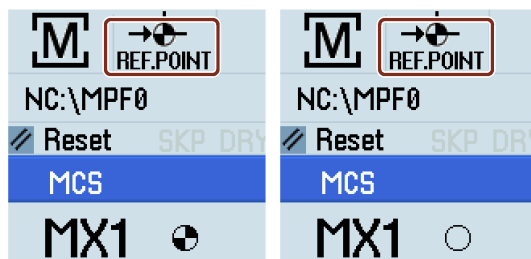
Reference point approach is required only for machine tools equipped with incremental encoders.

Operating sequence

1. Turn on the power supply for the machine tool.



2. Release all emergency stop buttons on the machine. By default, the control system is in reference point approach (REF.POINT) mode after startup.
 - If a machine axis is equipped with an absolute encoder, the axis is referenced automatically after the control system starts and therefore reference point approach is unnecessary. The circle symbol (lower-left figure) next to the axis identifier indicates that the axis is already referenced.
 - If a machine axis is equipped with an incremental encoder, the axis is not referenced after the control system starts. The circle symbol (lower-right figure) next to the axis identifier indicates that the axis has not been referenced. Proceed to Step 3.



3. Use the corresponding axis traversing keys to traverse the axes until the  symbol appears next to the axis identifier.

5 Setting the user interface language

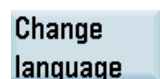
Note

The default user interface language is dependent on the PPU type. For a PPU with Chinese keys, the default language is Chinese after power-on; otherwise it's English. If you desire to change it, proceed through the following operating sequence; otherwise, skip this chapter and proceed to the next chapter for setting up tools.

Operating sequence



1. Select the system data operating area.



2. Open the user interface language selection window.



3. Use the cursor keys to select the desired language.



4. Press this softkey to confirm your selection and the system is automatically restarted to activate the selected language.

6 Setting up tools

6.1 Creating a tool

Note

The control system supports a maximum of 64 tools and 128 cutting edges. You can create a maximum of nine cutting edges for each tool. Before executing a part program, you must first create the desired tool in the tool list and proceed with the tool setting operations.

Operating sequence



1. Select the offset operating area.



2. Open the tool list window.



3. Open the lower-level menu for tool type selection.



4. Select a tool type (for example, a turning tool) with the corresponding softkey.

5. Enter the tool number and then select the corresponding tool edge position code in the following window according to the actual tool tip direction, the actual positive directions of the machine axes X and Z, as well as the description about tool edge positions given in the illustration under the screenshot below.

New turning tool

Tool No.:

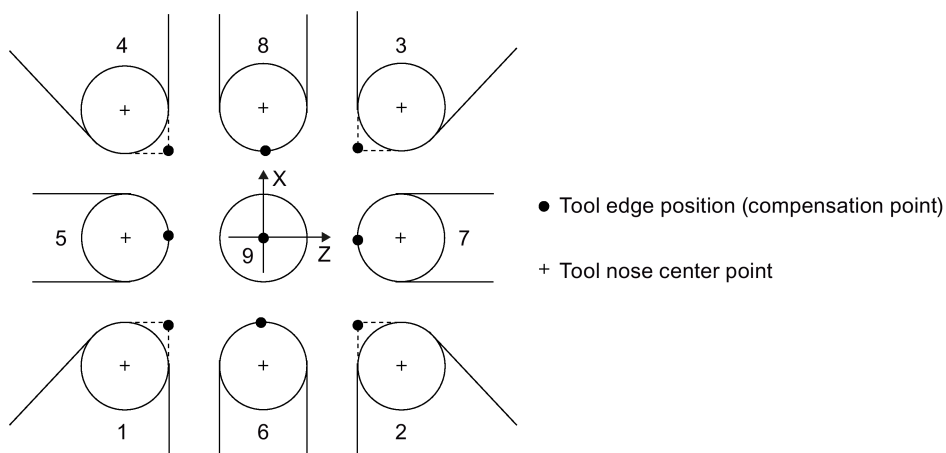
Edge position:

Identifier

Turning tool

Note:

The tool tip direction is the result of the correct tool settings as well as the loading orientation of the tool. The proper selection of tool edge positions has a direct influence on the correct tool nose radius compensation.





6. Press this softkey to confirm your settings. The window below shows the information of the new tool created. Set the desired tool data in this window.

Tool list									
REL				WCS				MCS	
X			0.000	X			0.000	MX1	0.000
Z			0.000	Z			0.000	MZ1	0.000
SP			0.000	SP			0.000	MSP1	0.000
① ②				③				④	⑤
Type	T	D	H	LengthX	LengthZ	Radius	Cle.ang.	Tip wid.	↗
	1	1	0	0.000	0.000	0.000	0.0	0.000	3

- Column ① indicates the tool offset number in Siemens mode.
- Column ② indicates the tool offset number in ISO mode.
- Column ③ (clearance angle) is visible only when you set bit 18 of MD19730[0] to 1 (MD19730[0] = 40000H). Note that MD19730 is visible only with the manufacturer password. Currently the clearance angle function can work only when the optional Manual Machine Plus function is licensed and activated. For more information about the clearance angle, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Function Manual.
- Column ④ is settable only for grooving tools.
- Column ⑤ is the input field for tool edge position codes.



7. Press this key to confirm your settings.

6.2 Creating/changing a cutting edge

Note

Before creating or changing cutting edges, you must first create tools (Page 20).

Operating sequence



1. Select the offset operating area.



2. Open the tool list window.



3. Select the tool to which you desire to add a cutting edge.



4. Open the menu items for cutting edge settings.



5. Press this softkey to add a new cutting edge for the selected tool. The control system automatically adds the new cutting edge to the tool list. Now you can enter different lengths and radii for each cutting edge.

Tool list									
REL				WCS				MCS	
X			0.000	X			0.000	MX1	0.000
Z			0.000	Z			0.000	MZ1	0.000
SP			0.000	SP			0.000	MSP1	0.000
Type	T	D	H	Length X	Length Z	Radius	Tip wid.	↗	
	1	1	0	0.000	0.000	0.000	0.000	3	
		2	0	0.000	0.000	0.000	0.000	3	

Reset
edge

Delete
edge

6. You can also press the corresponding softkey to reset or delete a cutting edge.

6.3 Activating the tool and the spindle

Note

A tool in the tool list is not active until you activate it in "MDA" mode or the "T, S, M" window.

Operating sequence



1. Select the machining operating area.



2. Switch to "JOG" mode.



3. Open the "T, S, M" window.

4. Enter the desired tool number (for example, 1) in the "T, S, M" window.

T,S,M	
Tool change	1 D1 Please m
Spindle speed	rpm
Spindle direction	U
Activate W0	U
Other M function	
Press 'CYCLE START' to activate above functio	
T,S,M	Set REL 28
	Meas. tool



5. Press this key to move the cursor to the input field for the spindle speed, and enter the desired speed, for example, 1000.



6. Press this key again to move the cursor to the input field for the spindle direction.



7. Use this key to select the spindle direction, for example, "M3".



8. Press this key on the MCP to activate the tool and the spindle.



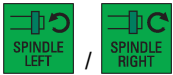
9. Press this key on the MCP to stop spindle rotation.



10. Press this softkey to return to the main screen of the machining operating area.

Note

You can alternatively activate the spindle using the following keys on the MCP:



For more information, see Section "Running the spindle manually (Page 252)".

6.4 Assigning the handwheel

If the machine manufacturer has assigned the handwheel for the control system, you can skip this section and directly measure the tool (Page 58) after activating the tool and the spindle (Page 23).

This control system provides the following two methods for assigning the handwheel:

- Assigning the handwheel through the MCP (Page 24)
- Assigning the handwheel through the PPU (Page 25)

Note

Make sure there is no obstacle when moving the tool to avoid tool collision.

6.4.1 Assigning the handwheel through the MCP

Precondition

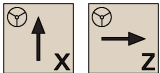
MD14512[16].7 = 0 (factory default)

Operating sequence

1. Select the machining operating area.



2. Press this key on the MCP to control the axis movement with external handwheels.



3. Press the desired axis traversing keys to assign the handwheel.



4. Press the increment keys on the MCP to select the required override increment.
 - 1: The override increment is 0.0005 mm (X)/0.001 mm (Z).
 - 10: The override increment is 0.005 mm (X)/0.010 mm (Z).
 - 100: The override increment is 0.050 mm (X)/0.100 mm (Z).

Note:

When the system is switched to REL or WCS, the override increment of the X axis equals that of the Z axis.

5. Rotate the handwheel to make the selected axis approach the workpiece.
6. Cut the surface of the workpiece to verify the handwheel assignment.
7. Close the window for handwheel assignment.



8. Press this key to return to the main screen of the machining operating area.



6.4.2 Assigning the handwheel through the PPU

Precondition

Set MD14512[16].7 = 80 (default value: 0). With this setting, you can only assign the handwheel through the PPU.

Operating sequence (activating the handwheel control function on the PPU)

If this function is already active, skip the steps below and proceed directly to the operating sequence for assigning the handwheel through the PPU.

-   1. Select the system data operating area.
-  2. Open the machine data window.
-  3. Press this softkey to open the basic machine data list.
-  4. Press this softkey to open the search field and enter the machine data "14512".
-  5. Confirm your entry with this softkey. Then the system starts searching automatically.
-  6. Move the cursor to the input field for 14512[16].
-  7. Enter the value "80".
8. Confirm your entry.
-  9. Press this vertical softkey to activate the value change. Note that the control system restarts to accept the new value.

Operating sequence (assigning the handwheel through the PPU)

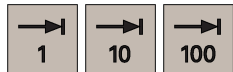
-  1. Select the machining operating area.
-  2. Press this key on the MCP.
-  3. Press this key to open the extended menu.
-  4. Open the handwheel assignment window.
5. Select the handwheel to be assigned with the cursor keys in the following window. You can assign a maximum of two handwheels.

Handwheel	
Number	Axis
1	X 
2	



6. Press the desired vertical softkey (<X> ... <MZ1>) or this key for the handwheel assignment. The handwheel symbol which appears to the left of the selected axis (for example, the X axis) indicates that you have assigned the handwheel to the selected axis.

MCS	Position
MX1	0.000



7. Press the increment keys on the MCP to select the required override increment.
 - 1: The override increment is 0.0005 mm (X)/0.001 mm (Z).
 - 10: The override increment is 0.005 mm (X)/0.010 mm (Z).
 - 100: The override increment is 0.050 mm (X)/0.100 mm (Z).

Note:

When the system is switched to REL or WCS, the override increment of the X axis equals that of the Z axis.

8. Rotate the handwheel to make the selected axis approach the workpiece.
9. Machine a surface to verify the handwheel assignment.
10. Press this softkey to close the window for handwheel assignment.



11. Press this key to return to the main screen of the machining operating area.

6.5 Measuring the tool manually

Precondition

You must first create a tool (Page 20) and activate it (Page 23) before measuring the tool. Make sure all the axes have been referenced (Page 19).

6.5.1 Measuring the tool manually (without the Y axis)

Note

This section takes the turning tool measurement in the offset operating area for example. For an alternative method of tool measurement, see Section "Additional method for measuring the tool (Page 58)".

If you have created other types of tools, proceed through the steps below to finish measuring all the tools.

Operating sequence

Measuring the X axis of the tool



1. Select the machining operating area.



2. Switch to handwheel control mode.



3. Select a suitable override feedrate, activate the spindle (Page 23) and then use the handwheel to move the tool to cut the external cylindrical surface of the workpiece for about 1 mm along the X axis and 15 mm along the Z axis. Then retract the tool along the Z axis.



4. Press this key to reset the system. Measure the diameter of the workpiece machined in the previous step with a calliper.



5. Select the offset operating area.

6. Move the cursor to the input field for Length X with the cursor keys.

Type	T	D	H	X	Z	Radius	Tip wid.	
	1	1	0	0.000	0.000	0.000	0.000	3

7. Press this softkey or the alphabetic key <X> to open the window for measuring the tool in the X direction and enter the workpiece diameter in the input field (for example, 50).

Meas. tool	T1	D1
X	Ø 50	

If you desire to use the relative coordinate, press this softkey.

Measure tool

Use rel. coordi.

OK

8. Press this softkey to confirm your input. The system calculates the offset and enters it in the geometry input field of the currently active tool automatically.

Tool list				Active T No.			
REL				T1 D1			
X		0.000	X	50.000	MX1	0.000	
Z		0.000	Z	0.000	MZ1	0.000	
SP		0.000	SP	0.000	MSP1	0.000	

Type	T	D	H	Length X	Length Z	Radius	Tip wid.	
	1	1	0	-50.000	0.000	0.000	0.000	3

Measuring the Z axis of the tool

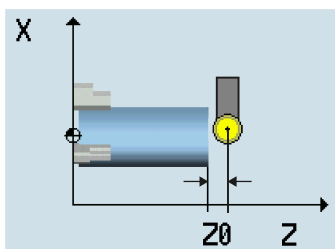
1. Move the cursor to the input field for Length Z with the cursor keys.

Type	T	D	H	X	Z	Radius	Tip wid.	
	1	1	0	-50.000	0.000	0.000	0.000	3

2. Press this softkey or the alphabetic key <Z> to open the window for measuring the tool in the Z direction, and then enter the distance between the tool tip and the workpiece edge in the input field, for example, "0". (This value is the thickness of a setting block if it is used.)

Meas. tool	T1	D1
Z	0	

Note that when measuring a large-radius tool, for example the button tool, you must take the tool radius into account when entering the offset value. See the following illustration for an example of measuring the Z axis of a button tool with edge position 8:



In the above example, you must enter the value of Z0 which is the distance between the button tool center and the workpiece edge in the offset input field.

Use rel.
coordi.

OK

If you desire to use the relative coordinate, press this softkey.

3. Press this softkey to confirm your input. The system calculates the offset and enters it in the geometry input field of the currently active tool automatically.

Tool list				Active T No.		T 1	D 1
REL				WCS		MCS	
X		0.000		X	50.000	MX1	0.000
Z		0.000		Z	0.000	MZ1	0.000
SP		0.000		SP	0.000	MSP1	0.000
Type	T	D	H	Length X	Length Z	Radius	Tip wid.
	1	1	0	-50.000	0.000	0.000	0.000

4. Repeat the above steps to finish measuring all the tools.

6.5.2 Measuring the tool manually (with the Y axis)

Note

If you have configured the Y axis and activated it, proceed through the steps below to measure axes X, Y and Z of the tool. For more information about how to configure the Y axis, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Commissioning Manual.

Operating sequence

Measuring the X axis of the tool



1. Select the machining operating area.



2. Switch to "MDA" mode and enter "SPOS=0" in the program editor window to fix the spindle.



3. Switch to handwheel control mode.



4. Select a suitable override feedrate.

5. Use the handwheel to move the tool to approach the workpiece and cut the surface of the workpiece for about 1 mm along the Y axis. Then retract the tool along the Y axis.

Note:

Make sure you do not move the tool along the X axis afterwards.



6. Switch to "MDA" mode and continue to enter "SPOS=180" in the program editor window.



7. Press this key to rotate the spindle 180 degrees.

8. Repeat Steps 3 to 5.

9. Measure the distance between the two cutting surfaces machined in the previous steps with a calliper.



10. Select the offset operating area.

Measure tool

11. Press the alphabetic key <X> or proceed through the following method to open the window for measuring the tool in the X direction:

Move the cursor to the input field for Length X with the cursor keys and then press this softkey.

Type	T	D	H	Length X	Length Y	Length Z	Radius	Tip wid.	↕
≡	1	1	0	0.000	0.000	0.000	5.000	0.000	7

12. Enter the distance measured in Step 9 in the input field, for example, 48.

Meas. tool	T1	D1
X	Ø 48	

If you desire to use the relative coordinate, press this softkey.

Use rel. coordi.



13. Press this softkey to confirm your input. The system calculates the offset and enters it in the geometry input field of the currently active tool automatically.

Tool list				Active T No.				T 1 D 1	
REL				WCS				MCS	
X		0.000		X	58.000			MX1	0.000
Y		0.000		Y	0.000			MY1	0.000
Z		0.000		Z	0.000			MZ1	0.000
SP1		0.000		SP1	0.000			MSP1	0.000

Type	T	D	H	Length X	Length Y	Length Z	Radius	Tip wid.	↕
≡	1	1	0	-29.000	0.000	0.000	5.000	0.000	7

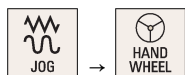
Measuring the Y axis of the tool



1. Select the machining operating area.



2. Switch to "MDA" mode and enter "SPOS=0" in the program editor window to fix the spindle.



3. Switch to handwheel control mode.



4. Select a suitable override feedrate.

5. Use the handwheel to move the tool to approach the workpiece and cut the surface of the workpiece for about 1 mm along the X axis. Then retract the tool along the X axis.

Note:

Make sure you do not move the tool along the Y axis afterwards.

6. Switch to "MDA" mode and continue to enter "SPOS=180" in the program editor window.



7. Press this key to rotate the spindle 180 degrees.



8. Repeat Steps 3 to 5.
9. Measure the distance between the two cutting surfaces machined in the previous steps with a calliper.
10. Select the offset operating area.



Measure tool

11. Press the alphabetic key <Y> or proceed through the following method to open the window for measuring the tool in the Y direction:

Move the cursor to the input field for Length Y with the cursor keys and then press this softkey.

Tool list				Active T No. T 1 D 1			
REL				WCS			
X		0.000		X	58.000		
Y		0.000		Y	0.000		
Z		0.000		Z	0.000		
SP1		0.000		SP1	0.000		
				MCS			
				MX1	0.000		
				MY1	0.000		
				MZ1	0.000		
				MSP1	0.000		
Type	T	D	H	Length X	Length Y	Length Z	Radius
	1	1	0	-29.000	0.000	0.000	5.000
							0.000
							7

12. Enter the distance measured in Step 9 in the input field, for example, "48".

Meas. tool				T1	D1
Y	Ø	48			

If you desire to use the relative coordinate, press this softkey.

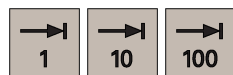
Use rel. coordi.



13. Press this softkey to confirm your input. The system calculates the offset and enters it in the geometry input field of the currently active tool automatically.

Tool list				Active T No. T 1 D 1			
REL				WCS			
X		0.000		X	58.000		
Y		0.000		Y	34.000		
Z		0.000		Z	0.000		
SP1		0.000		SP1	0.000		
				MCS			
				MX1	0.000		
				MY1	0.000		
				MZ1	0.000		
				MSP1	0.000		
Type	T	D	H	Length X	Length Y	Length Z	Radius
	1	1	0	-29.000	-29.000	0.000	5.000
							0.000
							7

Measuring the Z axis of the tool



1. Select the machining operating area.
2. Select a suitable override feedrate and use the handwheel to move the tool to scratch the required workpiece edge (or the edge of the setting block, if it is used) along the Z axis.
3. Select the offset operating area.
4. Press the alphabetic key <Z> or proceed through the following method to open the window for measuring the tool in the Z direction:

**Measure
tool**

Move the cursor to the input field for Length Z with the cursor keys and then press this softkey.

Tool list				Active T No. T 1 D 1					
REL		WCS		MCS					
X	0.000	X	58.000	MX1	0.000				
Y	0.000	Y	34.000	MY1	0.000				
Z	0.000	Z	0.000	MZ1	0.000				
SP1	0.000	SP1	0.000	MSP1	0.000				
Type	T	D	H	Length X	Length Y	Length Z	Radius	Tip wid.	↕
	1	1	0	-29.000	-29.000	0.000	5.000	0.000	7

- Enter the distance between the tool tip and the workpiece edge in the input field, for example, "0". (This value is the thickness of a setting block if it is used.)

Meas. tool	T1	D1
Z	0	

**Use rel.
coordi.**

If you desire to use the relative coordinate, press this softkey.

OK

- Press this softkey to confirm your input. The system calculates the offset and enters it in the geometry input field of the currently active tool automatically.

Tool list				Active T No. T 1 D 1					
REL		WCS		MCS					
X	0.000	X	58.000	MX1	0.000				
Y	0.000	Y	34.000	MY1	0.000				
Z	0.000	Z	0.000	MZ1	0.000				
SP1	0.000	SP1	0.000	MSP1	0.000				
Type	T	D	H	Length X	Length Y	Length Z	Radius	Tip wid.	↕
	1	1	0	-29.000	-29.000	0.000	5.000	0.000	7

- Repeat the above steps to finish measuring all the tools.

6.6 Verifying the tool offset result

To guarantee the safety and correctness of machining, you must verify the tool setup result.

Note

Before verifying the tool setup result, you must have finished all tool setup work as mentioned before.

Operating sequence



- Select the machining operating area.



- Switch to "MDA" mode.

**Delete
file**

- Press this softkey on the PPU.

- Enter the following test program (or you can create your own program):
T1 D1
G00 X0 Z5



Alternatively, you can press this softkey to load an existing part program from a system directory.



5. Press this key to activate the "ROV" function (indicator on: this function is active).

Note:

After the "ROV" function is active, the feedrate override switch can also control the speed of G00.



6. Rotate the feedrate override switch to 0%.

7. Press this key on the MCP.

Increase the feedrate override gradually to avoid accidents caused by fast axis movement. Observe whether the axis moves to the specified position.

For more information on the further softkey functions in "MDA" mode, see Section ""MDA" mode (Page 255)".

7 Creating part programs

7.1 Creating a part program

The control system can store a maximum of 300 part programs which include those created by the control system for certain functions such as MM+, TSM, and so on.

WARNING

Danger to life and/or damage to machine due to insecure part programs

Running an insecure part program on your machine may cause unexpected attacks to the machine, which in turn can lead to death, personal injuries, and/or machine damage.

- Make sure you use part programs that come from trusted sources.

Methods for creating part programs

You can create a part program with one of the following methods:

- Creating on a computer and transferring it to the PPU via USB interface (Page 66)
- Creating on a computer and transferring it to the PPU via Ethernet interface (Page 70)
- Creating directly on the PPU (see below for details)

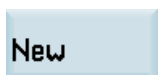
Creating a part program on the PPU



1. Select the program management operating area.



2. Press this softkey to enter the system directory for storing part programs.



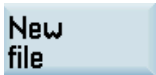
3. To directly create a new program file, press this softkey and go to Step 4.

To create a new program directory first, press this softkey and proceed as follows before you go to Step 4:

A. Press this softkey to open the window for creating a new directory.



New directory	
Type:	Directory DIR
Name:	



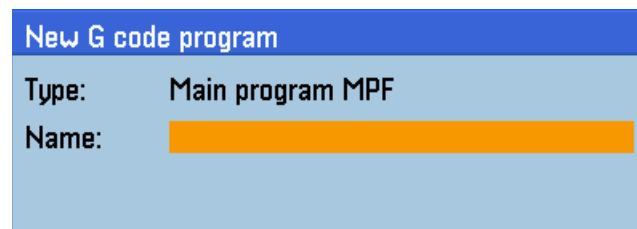
B. Enter a desired name for the new directory. Note that some special characters (see table at the end of this section) are invalid for the directory name.

C. Press this softkey to confirm your entry.

D. Select the new directory with the cursor keys.

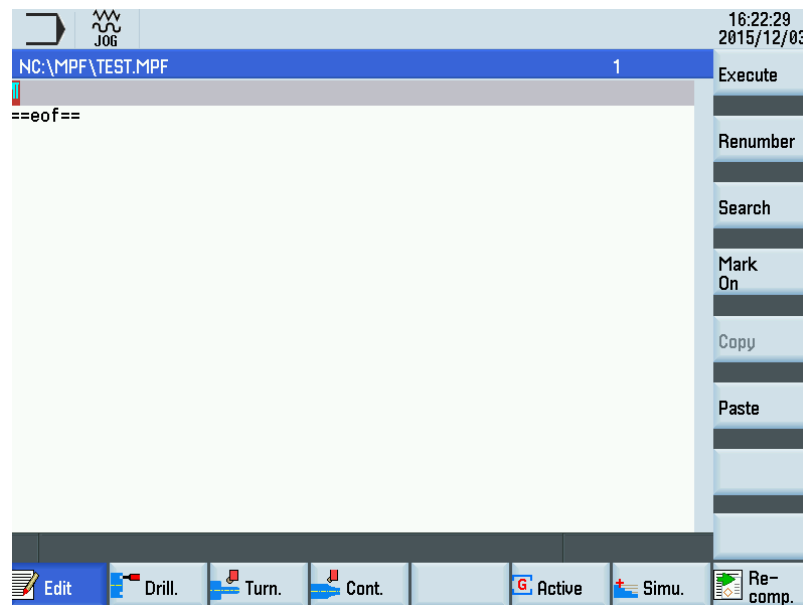
E. Press this key on the PPU to open the directory.

4. Press this softkey to open the window for creating a new program.



5. Enter the name of the new program. You can enter the file name extension ".MPF" (main program) or ".SPF" (subprogram) to define the program type. The control system identifies a program as a main program if you do not enter any file name extension. The character length of a program name is limited to 24 English characters or 12 Chinese characters. Note that some special characters (see table at the end of this section) are invalid for the program name.

6. Press this softkey to confirm your entry. The part program editor window opens automatically. Now you can edit the program text in the window. The control system saves your editing automatically.



Special characters invalid for program or directory names

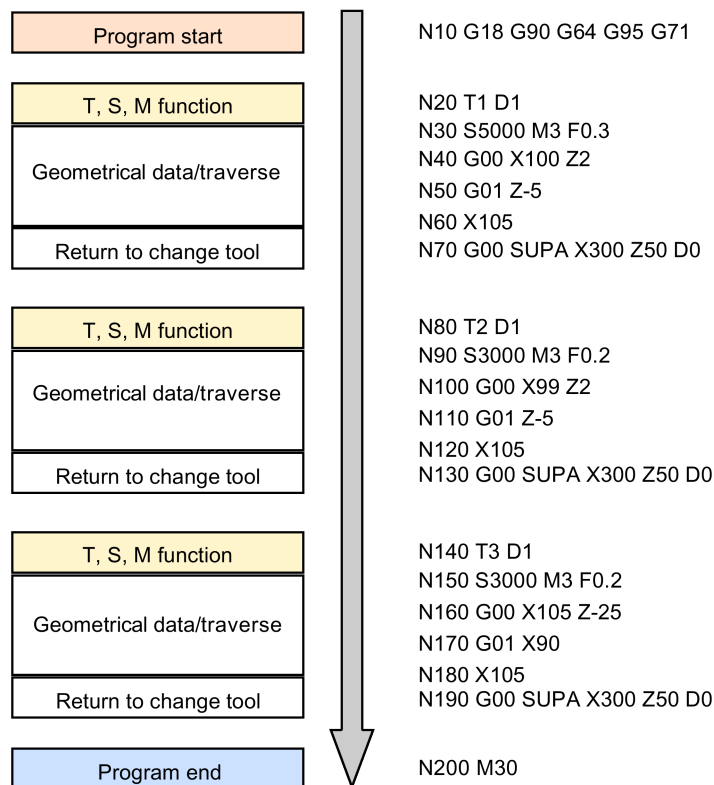
The control system does not support the use of the following special characters in program or directory names:

/	<	>	?	:	#	(	)	[	]	\$	!	-
+	^	\	*		;	&	%	@	=	~	`	Space

7.2 Editing the part program

7.2.1 Using a standard program structure

Using a standard program structure provides an easy way of part programming and a clear view of the machining sequences. Siemens recommends that you use the following program structure:



7.2.2 Editing a part program

Methods for editing part programs

You can edit a part program with one of the following methods:

- Editing on a computer and transferring it to the PPU via USB interface (Page 66)
- Editing on a computer and transferring it to the PPU via Ethernet interface (Page 70)
- Editing directly on the PPU (see below for details)

Editing an internal part program on the PPU

You can edit an internal part program only when it is not being executed. Note that any modification to the part program in the program editor window is stored immediately.

Note:

Steps 1 to 4: Search for a program file

Steps 5 to 9: Edit the selected program in the open program editor window

1. Select the program management operating area.



2. Press this softkey to enter the system directory for storing part programs.





3. Select the desired program file/directory in one of the following methods:

- Navigate to the program/directory with the cursor keys
- Open the search dialog box and enter the desired search term.

Note:

If you search for a program, the file name extension must be entered in the first input field of the dialog box below.

Search

Name:

Contained text:

☐ Include subordinate folders

☐ Case-sensitive

Search in:

/_N_MPF_DIR/

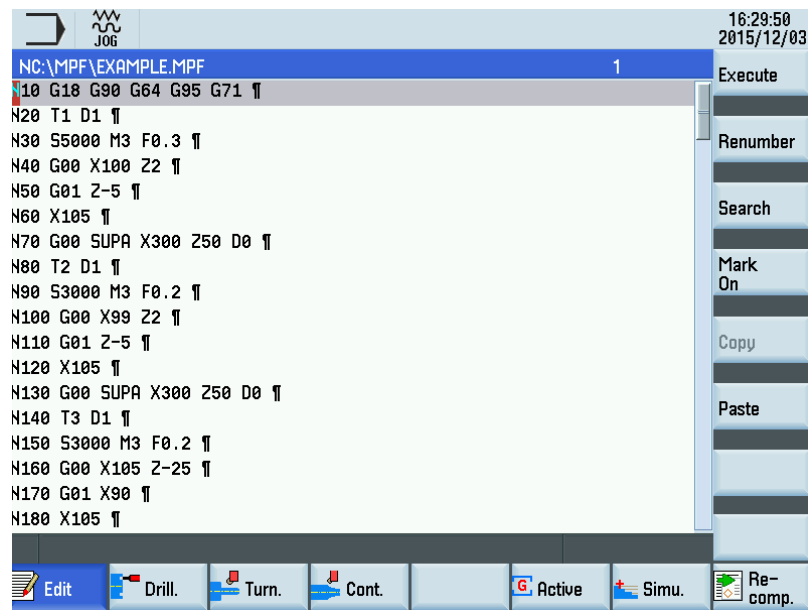
- On the PPU, press the alphabetic or numeric key that contains the first character of the desired program/directory name. The control system automatically highlights the first program/directory whose name starts with that character. If necessary, press the key continuously until you find the desired program/directory.



4. Press this key to open either the selected program in the program editor or the selected directory. In the latter case, perform Step 3 and then Step 4 until the selected program is opened in the program editor.
5. Edit the program text in the program editor window using the following keys on the PPU:



6. When necessary, select the following vertical softkeys to complete more program editing operations.



- **Renumbering program blocks**

Renumber

With this softkey, you can modify the block numbering (Nxx) of a program opened in the program editor window. After you press this softkey, the block number is inserted at the beginning of the program block in ascending order and is increased by an increment of 10 (for example, N10, N20, N30).

- **Searching in programs**

Search

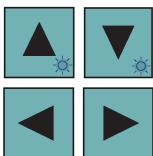
Pressing this softkey opens the search dialog box. You can use the search function to quickly arrive at points where you would like to make changes, for example, in very large programs. You can search with specified text or line number by selecting the corresponding softkey.



- **Copying/deleting/pasting program blocks**

Mark On

A. Press this softkey in the open program editor window to insert a marker.



B. Select the desired program blocks with the cursor.

Copy

C. Press this softkey to copy the selection to the buffer memory.

- OR -

DEL

Press this key to delete the selected program blocks and to copy them into the buffer memory.

Paste

D. Place the cursor on the desired insertion point in the program and press this softkey. The content of the buffer memory is pasted.



7. If you want to program cycles, press the corresponding softkey to open the desired cycle programming window. For more information, see Section "Cycles (Page 149)".



8. If you want to program contours, press this softkey to open the contour programming window. For more information, see Section "Free contour programming (Page 266)".



9. Press this key to return to the program management operating area after finishing editing.

Note:

When you are in the main screen of the program management operating area, you can select to make the following operations between programs:

- Searching for programs
- Copying/cutting/pasting programs
- Deleting/restoring programs
- Renaming programs

For more information, see Section "Program management operating area (Page 244)".

Editing an external part program on the PPU

You can edit an external part program directly on the PPU. Note that the program can be edited only when it is not being executed.

Note:

Steps 1 to 3: Search for a program file

Steps 4 to 7: Edit the selected program in the open program editor window



1. Select the program management operating area.



2. Press the desired softkey to enter the target program directory.



3. Follow the above steps 3 and 4 for editing an internal program to search for and open the desired program file.

4. Edit the program text in the program editor window using the following keys on the PPU:



5. When necessary, select the following vertical softkeys to complete more program editing operations.

- **Renumbering program blocks**

With this softkey, you can modify the block numbering (Nxx) of a program opened in the program editor window. After you press this softkey, the block number is inserted at the beginning of the program block in ascending order and is increased by an increment of 10 (for example, N10, N20, N30).

- **Searching and replacing in programs**

Pressing this softkey opens the search dialog box. You can use the search function to quickly arrive at points where you would like to make changes, for example, in very large programs.

You can search with specified text or line number by selecting the corresponding softkey as follows:



If more than one search result is found, you can also use the following softkey to check the remaining results one by one:



You can search and replace with specified text by using the following softkey:



If more than one search result is found, you can also use the following softkey to replace the remaining results one by one:



- **Copying/deleting/pasting program blocks**

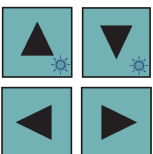
A. Press this softkey in the open program editor window to insert a marker.

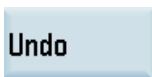
B. Select the desired program blocks with the cursor.

Renumber

Search

Mark
On





C. Press this softkey to copy the selection to the buffer memory.

- OR -

Press this key/softkey to delete the selected program blocks and to copy them into the buffer memory.

D. Place the cursor on the desired insertion point in the program and press this softkey. The content of the buffer memory is pasted.

- **Cutting a row of program blocks**

With this softkey, you can delete the entire program row where the cursor is placed and copy into the buffer memory. Then you can move the cursor to the desired position and press the following softkey to paste the cut program row at the current cursor position:



- **Undoing/redoing operations**

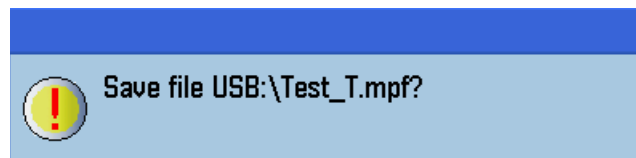
Pressing this softkey allows you to revoke your last operation.

Pressing this softkey allows you to restore your last operation that is revoked.

6. Save the program file after finishing editing through the following softkey operations:

 → 

If you switch to any other operating area after finishing editing without saving, a message appears to remind you to save the file, for example:



In this case, you may press either of the following softkeys as desired:

Pressing this softkey aborts the editing results and system switches to the desired operating area.

Pressing this softkey saves the editing results before the system switches to the desired operating area.



7. Press this key to return to the program management operating area.

Note:

When you are in the main screen of the program management operating area, you can select to make the following operations between programs:

- Searching for programs
- Copying/cutting/pasting programs
- Deleting programs
- Renaming programs

For more information, see Section "Program management operating area (Page 244)".

7.2.3 Understanding frequently used programming instructions

7.2.3.1 Dimension units in inch or mm (G70/G71)

Description	Programming example
G71: With G71 at the program start, both the geometrical data and the feedrates are evaluated as metric units.	N10 G18 G90 G54 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X100 Z1 N50 G01 X-0.5 N60 Z2 N70 G00 X200 Z50
G70: With G70 at the program start, the geometrical data is evaluated as inches, but the feedrates are not affected and remain as metric units.	N10 G18 G90 G54 G70 N20 T1 D1 N30 S5000 M3 G95 F0.2 N40 G00 X10 Z0.2 N50 G01 X-0.2 N60 Z0.2 N70 G00 X10 Z10

For more information, see Section "Dimensions in metric units and inches: G71, G70, G710, G700 (Page 81)".

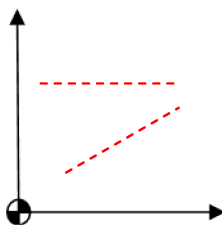
7.2.3.2 Definition of work offset (G54 to G59, G500, G90/G91)

Description	Illustration	Programming example
G500: All absolute path data corresponds to the current position. The position values are written in the G500 (basic) zero offset.		N10 G18 G90 G500 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X50 Z5 N50 G01 Z-5 N60 Z5 N70 G00 Z50 X100
G54 to G59: With G500 = 0, the offset for the workpiece can be stored in the workpiece offsets G54 to G59.		N10 G18 G90 G54 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X0 Z5 N50 G01 Z-5 N60 Z5 N70 G00 Z50 X100
G500 + G54: With G500 ≠ 0 activated, the value in G500 is added to the value in G54.		N10 G18 G90 G500 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 G54 X20 Z5 N50 G01 Z-5 N60 Z5 N70 G00 G53 Z50 X100

Description	Illustration	Programming example
G90: With G90 (absolute positioning) at the program start, the geometrical data refers to the zero of the coordinate system currently active in the program, usually with G54, G500, or G500 + G54.	-	N10 G18 G90 G54 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X100 Z5 N50 G01 Z-20 N60 Z5 N70 G00 Z500 X100
G91: With G91 (incremental positioning), you can add numerical value of path information (the incremental positioning with the current axis position as the start point) in the program. Subsequently, switch to absolute positioning with G90.	-	N10 G18 G90 G54 G70 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X3.93 Z0.196 N50 G01 G91 Z-0.787 N60 Z0.196 N70 G00 G90 Z19.68 X10

For more information, see Sections "Workpiece coordinate system- settable work offset: G54 to G59, G500, G53, G153 (Page 87)" and "Absolute/incremental dimensioning: G90, G91, AC, IC (Page 80)".

7.2.3.3 Rapid traverse (G00)

Description	Illustration	Programming example
G00: When G00 is active in the program, the axis will traverse at the maximum axis speed in a straight line.	 Straight line in any direction	N10 G18 G90 G54 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X50 Z5 N50 G01 Z-20 N60 Z5 N70 G00 Z500 X200

For more information, see Section "Linear interpolation with rapid traverse: G0 (Page 88)".

7.2.3.4 Tool and traverse (T, D, F, G94/G95, S, M3/M4, G01)

Description	Programming example
T, D: A new tool can be selected with the "T" command, and the "D" command is used to activate the tool length offset. G94/G95, F: The feedrate is defined with "F". G94 F defines feedrate in terms of time (mm/min) and G95 F defines feedrate in terms of spindle revolutions (mm/rev). S, M3/M4: The spindle speed is defined with "S". The spindle direction is defined with M3 (clockwise) and M4 (counter-clockwise). G01: When G01 is active in the program, the axis traverses at the programmed feedrate (as defined by G94 F or G95 F) in a straight line.	N10 G18 G90 G54 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X50 Z5 N50 G01 Z-5 N60 Z5 N70 G00 Z500 Z200

For more information, see sections as follows:

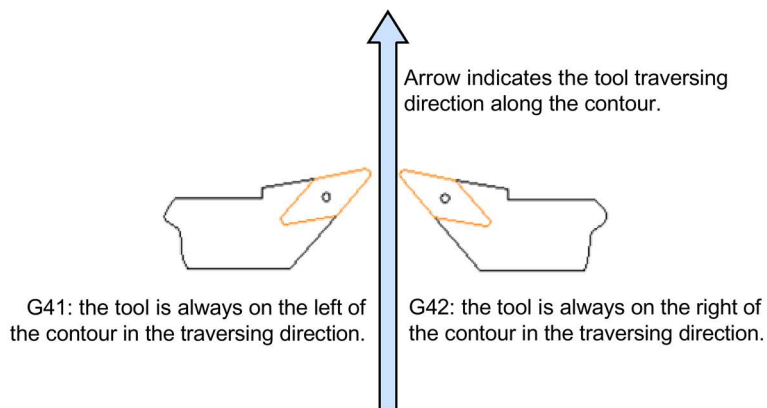
- Tool T (turning) (Page 116)
- Tool offset number D (turning) (Page 117)
- Feedrate F (Page 89)

- Spindle speed S, directions of rotation (Page 104)
- Linear interpolation with feedrate: G1 (Page 89)

7.2.3.5 Tool radius compensation (G40, G41/G42)

The tool radius compensation can be activated (G41/G42) or deactivated (G40) in contour programming.

- G41: tool radius compensation to the left of the contour
- G42: tool radius compensation to the right of the contour
- G40: tool radius compensation off



For more information, see Sections "Tool radius compensation OFF: G40 (Page 122)" and "Selecting the tool radius compensation: G41, G42 (Page 120)".

7.2.3.6 Turning circles and arcs (G02/G03)

The following gives an example of machining arc with specified program code:

```
N10 G18 G90 G500 G71
N20 T1 D1
N30 S5000 M3 G95 F0.3
N40 G00 X0 Z2
N50 G01 Z0
N60 G42 X50
N70 G00 X300 Z500
N80 G03 X75 Z-35 I-12 K-35
N90 G01 Z-130
N100 G40 X120 Z-140
```

* Note that N80 block as above can also be written as "N80 G03 X75 Z-35 CR=37".

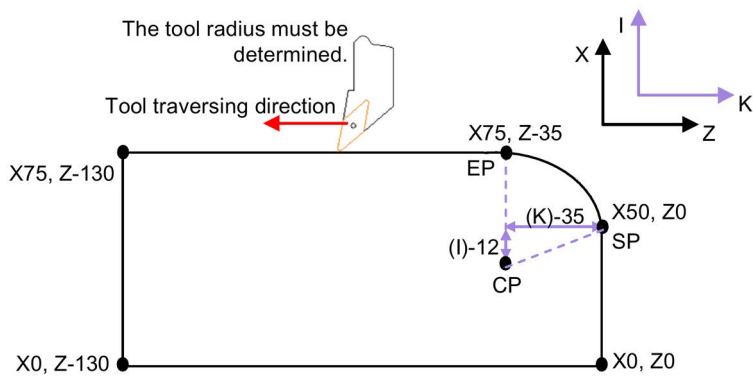
When turning circles and arcs, you must define the circle center point and the distance between the start point, end point, and the center point in the relative coordinate system.

When using the XZ coordinate system, the interpolation parameters I and K are available.

There are two common ways of defining circles and arcs:

- G02/G03 X... Z... I... K...
- G02/G03 X... Z... CR=...

Use positive value in CR with arcs $\leq 180^\circ$, and negative value with arcs $> 180^\circ$.



- SP Start point of circle
 CP Center point of circle
 EP End point of circle
 I Incremental distance from SP to CP in X axis
 K Incremental distance from SP to CP in Z axis
 G2 Traversing direction of the circle (clockwise)
 G3 Traversing direction of the circle (counter-clockwise)

For more information, see Section "Circular interpolation: G2, G3 (Page 90)".

7.2.3.7 Fixed point approach (G74/G75)

Description	Programming example
G74: By using G74, reference point can be approached automatically.	N10 G18 G90 G500 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X50 Z5 N50 G01 Z-5 N60 Z5 N70 G74 X=0 ;reference point
G75: By using G75, a fixed point on the machine defined by the manufacturer can be approached automatically.	N10 G18 G90 G500 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X50 Z5 N50 G01 Z-5 N60 Z5 N70 G74 Z=0 ;reference point N80 G75 X=0 ;fixed point

For more information, see Section "Fixed point approach (Page 99)".

7.2.3.8 Spindle control

Description	Programming example
M3: Spindle accelerates to the programmed speed in clockwise direction.	N10 G18 G90 G500 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3
M4: Spindle accelerates to the programmed speed in counter-clockwise direction.	N40 G00 X50 Z5 N50 G01 Z-5 N60 M5
M5: Spindle decelerates to stop.	N70 Z5 M4 N80 M5
M19: Sets the spindle to a specific angular position.	N90 M19 N100 G00 X200 Z50

For more information, see Section "Spindle movements (Page 104)".

7.2.3.9 Setting dwell time in the program (G04)

Description	Programming example
G04: G04 can be used to pause the tool movements during machining process. G04 F5: program dwells for 5 seconds This makes the workpiece surface much more smoother.	N10 G18 G90 G500 G71 N20 T1 D1 N30 S5000 M3 G95 F0.3 N40 G00 X50 Z5 N50 G01 Z-5 N60 G04 F5 N70 Z5 M4 N80 M5 N90 M19 N100 G00 X200 Z50

For more information, see Section "Dwell Time: G4 (Page 103)".

8 Program simulation

Simulation before machining

Before automatic machining, you need to perform the simulation to check whether the tool moves in the right way. During simulation, the current program is calculated in its entirety and the result displayed in graphic form. The result of programming is verified without traversing the machine axes. Incorrectly programmed machining steps are detected at an early stage and incorrect machining on the workpiece prevented.

You can choose one of the following methods for simulation:

- Simulation prior to machining of the workpiece (Page 45)
- Simultaneous recording prior to machining of the workpiece (Page 47)

Simulation during machining

If the view of the work space is blocked by coolant, for example, while the workpiece is being machined, you can track the program execution on the screen. For more information, see Section "Simultaneous recording during machining of the workpiece (Page 50)".

8.1 Simulation prior to machining of the workpiece

Before machining the workpiece on the machine, you have the option of performing a quick run-through in order to graphically display how the program is executed. This provides a simple way of checking the result of the programming.

Operating sequence

Note that the following Steps 1 to 3 describe how to open a desired program file on the PPU. If you are already in the program editor window, you can go to Step 4 directly.



1. Select the program management operating area.



2. Enter the system program directory and position the cursor on the program to be simulated.



3. Press this key and the program is opened in the program editor window.



4. Switch to "AUTO" mode.



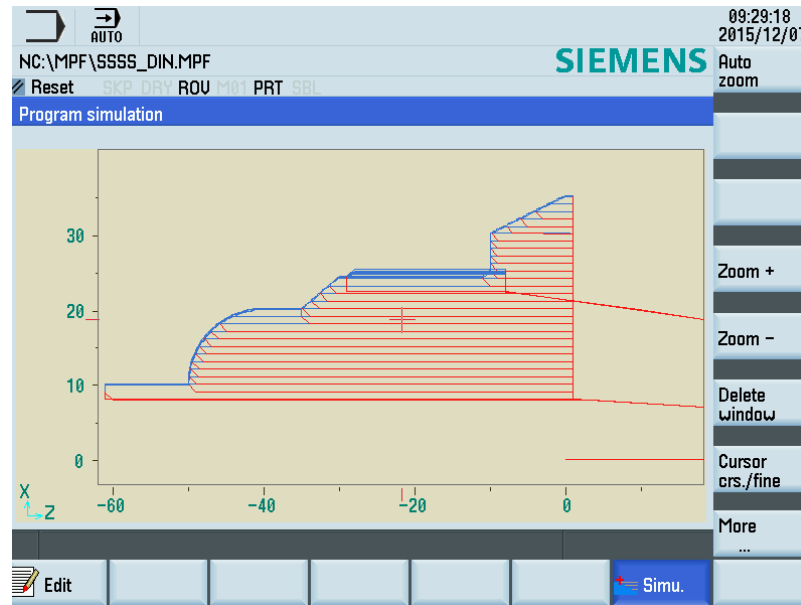
5. Press this softkey to open the program simulation window, and the program control mode PRT is automatically activated.



6. Press this key to start the simulation for the selected program.

The program execution is displayed graphically on the screen. The machine axes do not move.

7. You can also select the vertical softkeys as required in the simulation window as follows:



Auto
zoom

Scales the entire simulation track automatically to adapt to the size of the window

Zoom +

Zooms in the screen from the cursor position

Zoom -

Zooms out the screen from the cursor position

Delete
window

Deletes all the simulation tracks recorded up till now

Cursor
crs./fine

Makes the cursor move in large or small steps

Material
removal

Enables the material removal simulation of a defined blank

Show
blocks

Selects whether to show the blocks or not



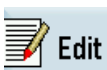
8. Press this key on the MCP if you wish to stop the simulation.
- OR -



Press this key on the MCP to cancel the simulation.



9. Press this key on the MCP to restart or continue the simulation.



10. After you finish the simulation, you can press this softkey to return to the program editor window.

8.2 Simultaneous recording prior to machining of the workpiece

Before machining the workpiece on the machine, you can graphically display the execution of the program on the screen to monitor the result of the programming.

Operating sequence



1. Select the program management operating area.



2. Enter the target program directory and position the cursor on the desired program.
3. Press this softkey and the system automatically changes to "AUTO" mode in the machining operating area.



4. Press this softkey to open the lower-level menu for program control.



5. Press this softkey to activate the PRT mode, and the program is executed without axis movement.



You can additionally press this softkey to replace the programmed feedrate with a dry run feedrate. For more information about setting the dry run feedrate, see Section "Dry run feedrate (Page 54)".



6. Press this softkey to open the simultaneous recording window.



7. Press this key to start the recording. The program execution is displayed graphically on the screen. The machine axes do not move.



8. Press this key on the MCP if you wish to stop the recording.
- OR -



Press this key on the MCP to cancel the recording.



9. Press this key on the MCP to restart or continue the recording.

Note

You must deactivate the PRT mode and the dry run feedrate activated in Steps 4 and 5 above before machining the workpiece on the machine; otherwise the machining cannot be performed due to the standstill of the axes and the spindle.

For more information about the softkeys in the simultaneous recording window, see Section "Simulation prior to machining of the workpiece (Page 45)".

9 Machining the workpiece

9.1 Executing a part program

Before starting a program, make sure that both the control system and the machine are set up, and the part program is verified with simulation and test. Observe the relevant safety notes of the machine manufacturer.



1. Select the program management operating area.



2. Select the desired program directory.
3. Select the program that you desire to execute.



4. Press this softkey. For some directories, press the following softkey instead:

Ext. execution

The system automatically changes to "AUTO" mode in the machining operating area after you press the softkey.



5. If desired, you can use this softkey to specify how you want the program to be executed (for more information, see Section "Program control functions (Page 54)").



6. Make sure the feedrate override is 0%. Press this key on the MCP to close the safety door. If this function is not available, close the door on the machine manually.



7. Press this key on the MCP to start the machining of the program.



8. Turn the feedrate override switch slowly to the desired value.
Pressing this key stops the execution of a part program. The program currently running is aborted. On the next program start, the machining starts from the beginning.



Pressing this key suspends the execution of a part program. The axes stop running while the spindle continues running. Press the following key again, and the program continues to run.



Note

To ensure the shortest machining time, do not perform the HMI operations during program running, for example, creating a new program or editing an external program on the PPU.

9.2 Executing specified blocks

If you would only like to perform a certain section of a program on the machine, then you do not need to start the program from the beginning. You can start the program from a specified program block in the following cases:

- After you stop or interrupt the program execution
- When you need to specify a target position, e.g. during remachining

Operating sequence

Proceed as follows to start machining from the last interruption point in the program:



1. Select the machining operating area.



2. Switch to "AUTO" mode.



3. Press this softkey to open the block search window.

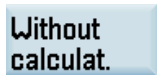


4. Press this softkey to load the interruption point, and the cursor moves to the beginning of the target block which is interrupted last time.



5. Press one of the following softkeys to set the condition for the block search:

- The program will continue from the line before the target block. The same calculations of the basic conditions (e.g. tool and cutting edge numbers, M functions, feedrate and spindle speed) in the previous blocks are carried out as during normal program operation, but the axes do not move.
- The program will continue from the target block containing the interruption point. The same calculations of the basic conditions in the previous blocks are carried out as during normal program operation, but the axes do not move.
- Block search without calculation of the basic conditions. All settings required for execution have to be programmed from the target block (e.g. feedrate, spindle speed, etc.).



For the block search to contour/end point, make sure you set MD11450 bit 2 to 1 to suppress the output of the auxiliary functions. Note that MD11450 is visible only with the manufacturer password.

6. Make sure the feedrate override is 0%.

7. Press this key on the MCP, and then an alarm 010208 appears for your confirmation whether to continue.



010208 Continue program with NC start 16:07

8. Press this key again to execute the program.



9. Turn the feedrate override switch on the MCP slowly to the desired value.

NOTICE

Damage to workpiece or machine tools

For the machining in ISO mode, you must program the desired feedrate from the target block after a block search without calculation. Otherwise, the axis will traverse at the maximum speed with G0 after you execute the program, which may cause damage to the workpiece or the machine tools.

9.3 Correcting a part program

As soon as a syntax error in the part program is detected by the control system, program execution is interrupted and the syntax error is displayed in the alarm line. In this case, you can correct the error directly in the machining operating area with the program correction function. When making program changes, the control system must be brought into the reset state.

Note

The program correction function is not available for program execution from external.

Operating sequence

Prerequisite: A part program is loaded in "AUTO" mode in the machining operating area.



1. Press this key on the MCP to place the program in the reset mode.



2. Press this softkey to activate the block editing window.



3. Make the necessary corrections. Any changes to the program will be stored immediately.
4. After you finish the program correction, press this softkey to switch back to the block display window.



5. Press this key on the MCP to restart the machining of the program.

9.4 Simultaneous recording during machining of the workpiece

You can follow machining of the workpiece on the screen while the program is being executed on the machine.

Operating sequence



1. Select the program management operating area.



2. Enter the target program directory and position the cursor on the desired program.



3. Press this softkey and the system automatically changes to "AUTO" mode in the machining operating area.



4. Press this softkey to open the simultaneous recording window.



5. Press this key to start the recording. The machining of the workpiece is started and graphically displayed on the screen.

9.5 Entering the tool wear offsets

Note

You must distinguish the direction of tool wear compensation clearly.

Two methods are available for entering the tool wear offsets: absolute input and incremental input (default: absolute input).

Operating sequence for absolute input



1. Select the offset operating area.



2. Open the tool wear window.



3. Use the cursor keys to select the required tools and their edges.

4. Enter the desired tool length wear parameter of axes X and Z as well as the tool radius wear parameter (range of value in mm: -9.999 to 9.999).

Positive value: the tool compensates in a direction of moving away from the workpiece.

Negative value: the tool compensates in a direction of approaching the workpiece.

Note:

If there is already a non-zero offset value in the input field (for example, the existing tool wear offset value of the X axis is "0.1") and you desire to increase the offset value, then you can either directly enter the sum of the existing offset value and the increment, or call the pocket calculator to work out the tool wear offset (proceed through Steps 6 to 9).

5. Press this key or move the cursor to activate the compensation.



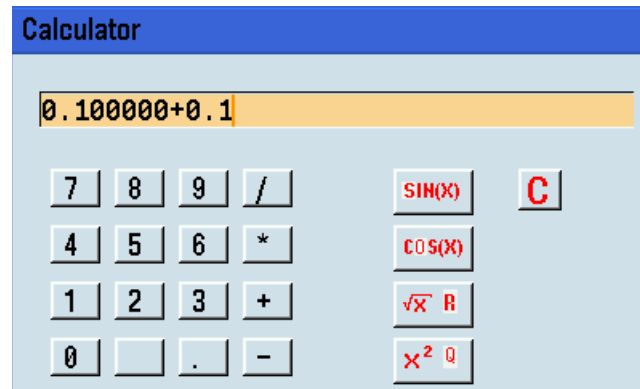
Tool wear < ABS input >					
Type	T	D	Length X	Wear Length Z	Radius
	1	1	0.200	0.100	0.400
		2	0.000	0.000	0.000

6. If you desire to use the pocket calculator, position the cursor on the desired input field and press this key. Then the pocket calculator is opened:

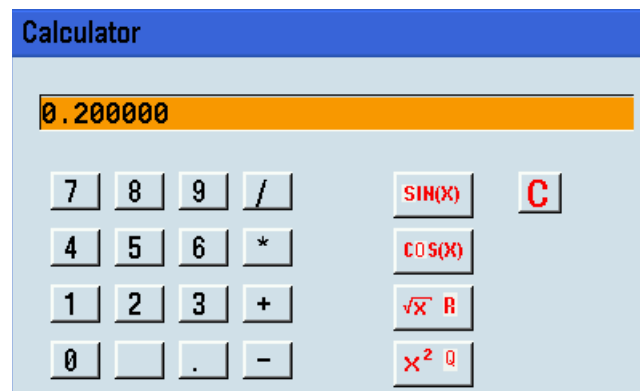


Calculator									
0.100000									
7	8	9	/	SIN(X)	C				
4	5	6	*	COS(X)					
1	2	3	+	√X R					
0		.	-	X ² Q					

7. Enter the desired arithmetic statement in the input line of the pocket calculator.



8. Press this key and the calculation result is displayed in the pocket calculator:



9. Press this softkey to enter the result in the input field at the current cursor position and close the pocket calculator automatically.



Tool wear < ABS input >					
Type	T	D	Length X	Wear Length Z	Radius
	1	1	0.200	0.100	0.400
		2	0.000	0.000	0.000

Note:

For more information about how to use the pocket calculator, see Section "Pocket calculator (Page 77)".

Operating sequence for incremental input



1. Select the offset operating area.



2. Open the tool wear window.



3. Position the cursor bar on the input field to be modified.

Tool wear < ABS input >						
Type	T	D	Wear			
			Length X	Length Z	Radius	
	1	1	0.100	0.100	0.400	
		2	0.000	0.000	0.000	

INC
input

4. Switch to incremental input.

5. Enter the desired increment value, for example, 0.1.

- Positive value: the tool compensates in a direction of moving away from the workpiece.
- Negative value: the tool compensates in a direction of approaching the workpiece.

Note:

Alternatively, you can call the pocket calculator to work out the increment value (see Steps 7 to 10).

6. Press this key or move the cursor to activate the compensation. Then the value displayed in the input field becomes the sum of the original value and the increment value.



Tool wear < * INC input >						
Type	T	D	Wear			
			Length X	Length Z	Radius	
	1	1	0.200	0.100	0.400	
		2	0.000	0.000	0.000	

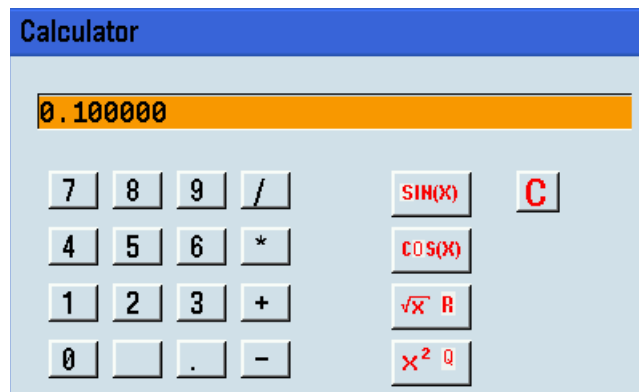
7. If you desire to use the pocket calculator, press this key.



Calculator									
7	8	9	/	SIN(X)	C				
4	5	6	*	COS(X)					
1	2	3	+	√X R					
0		.	-	X ² Q					

8. Enter the desired arithmetic statement in the input line of the pocket calculator.
9. Press this key and the calculation result (for example, 0.1) is displayed in the pocket calculator:





10. Press this softkey to add the calculation result to the value in the input field at the current cursor position and close the pocket calculator automatically.

Tool wear < * INC input >					
Type	T	D	Length X	Wear Length Z	Radius
	1	1	0.200	0.100	0.400
		2	0.000	0.000	0.000

Note:

For more information about how to use the pocket calculator, see Section "Pocket calculator (Page 77)".

10 Other common operations

10.1 Program control functions

You can perform further program control operations through the following operation:



10.1.1 Program test



With this softkey activated, the part program is executed with no axis or spindle movement. In this way, you can check the programmed axis positions and auxiliary function outputs of a part program. This softkey functions the same as the following key on the MCP:



10.1.2 Dry run feedrate



With this softkey activated, all traversing motions are performed with the dry run feedrate defined in SD42100 DRY_RUN_FEED. You can set the dry run feedrate in the JOG data setting window through the following operation:



10.1.3 Conditional stop

**Condit.
stop**

With this softkey activated, the processing of the program stops at the end of every block containing M01. In this way, you can check the result already obtained during the machining of a workpiece. This softkey functions the same as the following key on the MCP:



Note that in order to continue executing the program, press the following key again:



10.1.4 Skip block

Skip

It is possible to skip program blocks, which are not to be executed every time the program runs. With this softkey activated, the program blocks identified with a slash in front of the block number (e.g. "/N100") are skipped during machining. Several consecutive blocks can be skipped.

10.1.5 Single block mode

**SBL
fine**

With this softkey activated, the system can interrupt the machining of the workpiece after each program block. In this way, you can control the machining result block-by-block and check individual machining steps. This softkey functions the same as the following key on the MCP:



The program is advanced to the next block via the following key:



However, for the thread blocks (G33), a stop is performed at the end of the current thread block only with dry run feedrate activated.

10.1.6 Rapid traverse override

**ROV
active**

With this softkey activated, the traversing speed of the axes in the rapid traverse mode (G00) can be controlled via the feedrate override switch on the MCP.

This softkey functions the same as the following key on the MCP:



It is recommended that you keep this function activated for easy feedrate control during machining.

10.1.7 Auxiliary function lock

**Auxiliary
func. OFF**

With this softkey activated, the part program is executed with the spindle disabled and all auxiliary functions (see below) suppressed.

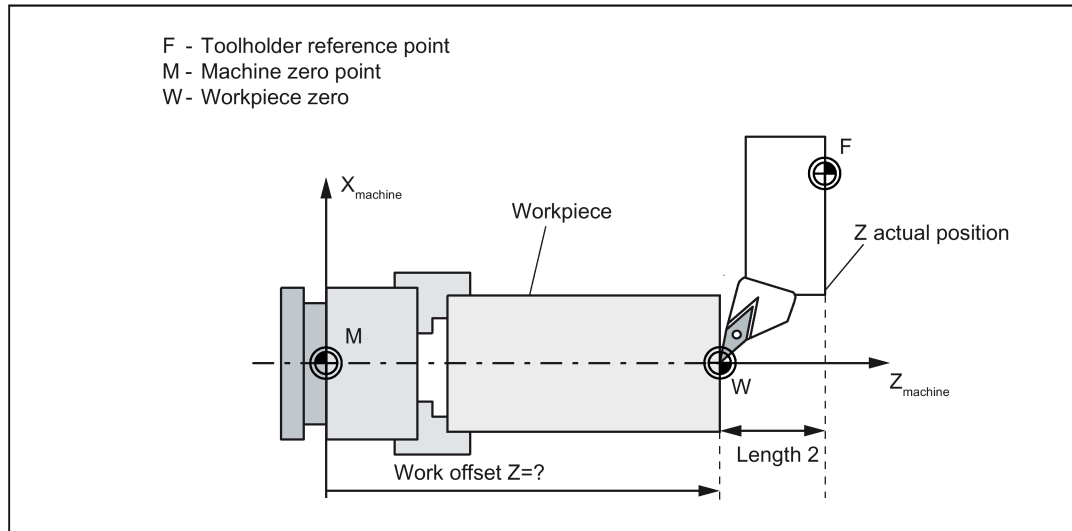
Auxiliary function	Address
Tool selection	T
Tool offset	D, DL
Feedrate	F
Spindle speed	S
M functions	M
H functions	H

10.2 Setting up the workpiece

10.2.1 Measuring the workpiece

Overview

Select the relevant offset panel (for example, G54) and the axis you want to determine for the offset. See the following example for how to determine the work offset in the Z axis.



Before measuring, you can start the spindle by following the steps in Section "Activating the tool and the spindle (Page 23)".

Operating sequence



1. Select the offset operating area.



2. Switch to "JOG" mode.



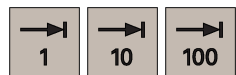
3. Open the list of work offsets.



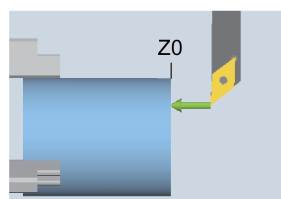
4. Open the window for measuring the work offsets. Note that this vertical softkey is active only in "JOG" mode.



5. Switch to handwheel mode.



6. Select a suitable override feedrate, and then use the handwheel to move the tool to scratch the required workpiece edge or the edge of the setting block (for example, Z0), if a setting block is used.





7. Select the offset plane to save in (for example, G54).



8. Enter the distance (for example, "0").
Press this key or move the cursor to confirm your input.

Workpiece measurement, edge		Distance to work zero			
	Save in	T 1	↔ 3	D 1	
	Basic offset	G54			
	Length Z		0.000	mm	
	Distance		0.000	mm	
	Offset	Z0	0.000	mm	

Set work offset

9. Press this vertical softkey. The work offset of the Z axis is calculated automatically and displayed in the offset field.

10.2.2 Entering/modifying workpiece offsets

Operating sequence

In case of any problems found when testing the tool offset result, you can proceed through the following steps to make tiny adjustment of values:



1. Select the offset operating area.



2. Open the list of work offsets. The list contains the values of the basic offset of the programmed work offset and the active scaling factors, the mirror status display and the total of all active work offsets.



3. Use the cursor keys to position the cursor bar in the input fields to be modified and enter the values.

	X	mm	Z	mm	SP	°
G500	0.000000		0.000000		0.000000	
G54	0.000000		0.000000		0.000000	
G55	0.000000		0.000000		0.000000	
G56	0.000000		0.000000		0.000000	
G57	0.000000		0.000000		0.000000	
G58	0.000000		0.000000		0.000000	
G59	0.000000		0.000000		0.000000	
Program	0.000000		0.000000		0.000000	
Scale	1.000000		1.000000		1.000000	
Mirror	0		0		0	
Total	0.000000		0.000000		0.000000	



4. Confirm your entries. The changes to the work offsets are activated immediately.

10.3 Additional method for measuring the tool

Note

- You must first create a tool (Page 20) and activate it (Page 23) before measuring the tool.
- This section takes the turning tool measurement for example. If you have created other types of tools, proceed through the following steps to finish the measurement of all the tools. This will make the tool change process easier.

Operating sequence

Measuring the X axis of the tool



1. Select the machining operating area.



2. Switch to "JOG" mode.



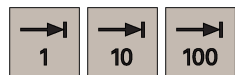
3. Open the manual tool measurement window.



4. Press this vertical softkey to measure the tool in the X direction.



5. Switch to handwheel control mode.



6. Select a suitable override feedrate, and use the handwheel to move the tool to cut the surface of the workpiece for about 1 mm along the X axis and 15 mm along the Z axis. Then retract the tool along the Z axis.

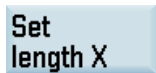


7. Press this key to reset the system. Measure the diameter of the workpiece machined in the previous step with a calliper.



8. Enter the workpiece diameter in the "Ø" field (for example, 50).

Tool measure manual		Workpiece diameter	
X		T 1	D 1
		Ø 50 mm	
Z	Length X(L)	0.000 mm	



9. Press this softkey. The system calculates the offset and enters it in the geometry input field of the currently active tool automatically.



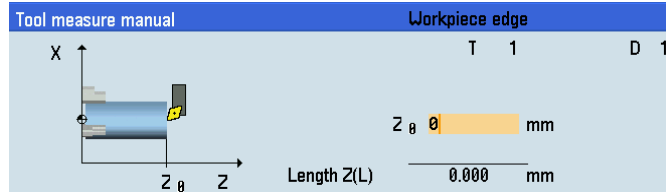
10. Press this softkey and you can see that the compensation data values have been automatically added to the tool data.

Tool list				Active T No. T 1 D 1			
REL		WCS		MCS			
X	0.000	X	50.000	MX1	0.000		
Z	0.000	Z	0.000	MZ1	0.000		
SP	0.000	SP	0.000	MSP1	0.000		
Type	T	D	H	Length X	Length Z	Radius	Tip wid.
	1	1	0	-50.000	0.000	0.000	0.000

Measuring the Z axis of the tool



1. Open the manual tool measurement window.
2. Press this vertical softkey to measure the tool in the Z direction.
3. Enter the distance between the tool tip and the workpiece edge in the "Z0" field, for example, "0". (This value is the thickness of a setting block if it is used.)



4. Press this softkey and then the system calculates and enters the offset in the geometry input field of the currently active tool automatically.
5. Press this softkey and you can see that the compensation data values have been automatically added to the tool data.

Tool list				Active T No. T 1 D 1			
REL				WCS			
X		0.000		X	50.000	MX1	0.000
Z		0.000		Z	0.000	MZ1	0.000
SP		0.000		SP	0.000	MSP1	0.000
Type	T	D	H	Length X	Length Z	Radius	Tip wid.
	1	1	0	-50.000	0.000	0.000	0.000

6. Repeat the above steps to finish measuring all the tools.

10.4 Entering/modifying the setting data

Entering/modifying the setting data

Operating sequence



1. Select the offset operating area.
2. Open the setting data window.
3. Open the window for setting the general data.
4. Position the cursor bar in the input fields to be modified and enter the values (see table below for the parameter descriptions).
5. Use this key or move the cursor to confirm your entries.

Parameters in the setting data window

Setting data			
Spindle data			
Minimum:	①	0.000	rpm
Maximum:	②	1000.000	rpm
Start angle			
Start angle for thread:	③	0.000	°

- ①/② A limitation of the spindle speed in the maximum (G26)/minimum (G25) fields can only be performed within the limit values defined in the machine data.
- ③ For thread cutting, a start position for the spindle is displayed as the start angle. A multiple thread can be cut by changing the angle when the thread cutting operation is repeated.

Setting the time counter

Operating sequence



1. Select the offset operating area.



2. Open the setting data window.



3. Open the time counter window.



4. Position the cursor bar in the input fields to be modified and enter the values (see table below for the parameter descriptions).



5. Use this key or move the cursor to confirm your entries.

Parameters in the window for timers and workpiece counters

Times / Counter		
Parts in total	①	0
Parts required	②	0
Part count	③	0
Run time	④	0000 H 00 M 00 S
Cycle time	⑤	0000 H 00 M 00 S
Cutting time	⑥	0000 H 00 M 00 S
Setup time	⑦	0002 H 44 M
Power-on time	⑧	0001 H 48 M

- ① The total number of workpieces produced (total actual)
- ② The number of workpieces required (workpiece setpoint)
- ③ The number of all workpieces produced since the starting time
- ④ The total run time of NC programs in "AUTO" mode and the run times of all programs between NC start and end of program/RESET. The timer is set to zero after each power up of the control system.
- ⑤ The run time of the selected NC program in seconds
The default value is 0 each time a new NC program starts up. MD27860 can be set to ensure that this value will be deleted even if there is a jump to the beginning of the program with GOTOs or in the event of ASUPs (used for tool change in "JOG" and "MM+" modes) and PROG_EVENTS starting.
- ⑥ Processing time in seconds
- ⑦ The time since the last system power-up with default values ("cold restart") in minutes
- ⑧ The time since the last normal system power-up ("warm restart") in minutes

Note

The timer is automatically reset to zero in case of a power-up of the control system with default values.

Setting the working area limitation

Operating sequence



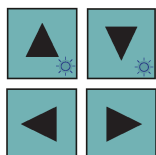
1. Select the offset operating area.



2. Open the setting data window.



3. Open the working area limitation window.



4. Position the cursor bar in the input fields to be modified and enter the required values.



5. Position the cursor bar in the fields for activating the entered values.
6. Press this key to activate or deactivate the entered minimum/maximum values.

Parameters in the window for working area limitation

Working area limitation					
Axis	Minimum	Active	Maximum	Active	Unit
① MX1	② -100000000.000	③ ☐ U	④ 100000000.000	⑤ ☐ U	⑥ mm
MZ1	-100000000.000	☐ U	100000000.000	☐ U	mm
MSP1	-100000000.000	☐ U	100000000.000	☐ U	°

- ① Displays the axes that exist in the machine coordinate system (MCS), workpiece coordinate system (WCS), or relative coordinate system (REL).
- ②/④ The minimum/maximum traversing distance specified for the axes.
- ③/⑤ Activates or deactivates the entered minimum/maximum values.
- ⑥ Unit for the traversing distance.

Modifying miscellaneous setting data

Operating sequence



1. Select the system data operating area.



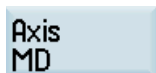
2. Open the window for machine data.



3. Open the window for the expert list.



4. Select a group of setting data you desire to modify.



5. Use these softkeys to search for your desired setting data with the data number/name.





Alternatively, you can position the cursor bar in the input fields to be modified and enter the values.

You can use the following softkeys to switch to the desired axis when modifying the axis-specific setting data.



6. Use this key or move the cursor to confirm your entries.

10.5 Setting R parameters

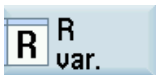
Functionality

The "R variables" start screen lists the R parameters that exist within the control system. You can set or query these global parameters in any program as required.

Operating sequence



1. Select the offset operating area.



2. Open the list of R parameters.



3. Press the cursor keys to navigate in the list, and enter the values in the input fields to be modified.



Note:

You can search for your desired R variable with this softkey. By default, the function searches the R number.



You can press this softkey to activate the option of searching by R name. Define the R name as desired, if necessary.



4. Press this key or move the cursor to confirm your entries.

For more information about R parameters, see Section "Arithmetic parameter R (Page 126)".

10.6 Setting user data

Functionality

The "User data" start screen lists the user data that exist within the control system. You can set or query these global parameters in any program as required.

Operating sequence



1. Select the offset operating area.



2. Open the list of user data.



3. Use the cursor keys to navigate in the list, and enter the values in the input fields to be modified.



Note:

You can search for your desired user data with this softkey.



You can press this softkey to continue searching your desired user data.

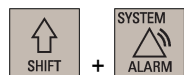


4. Use this key or move the cursor to confirm your entries.

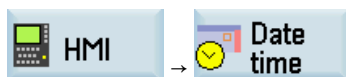
10.7 Setting the date and time

By default, the system date and time remain the factory settings. You can proceed through the following operating sequence to change the date and time as required.

Operating sequence



1. Select the system data operating area.
2. Open the date and time setting window through the following softkey operations:



3. Enter the date and time in the specified format.

Current	2015/05/19	11:02:16
Format	YYYY/MM/DD	HH:MM:SS
New	2015 / 05 / 19	11 : 05 : 35



4. Press this softkey to confirm your settings.

Note

The settings of date and time are effective for 30 days if the control system is powered off for a long time at 25 °C surrounding air temperature.

10.8 Configuring the operating area after startup

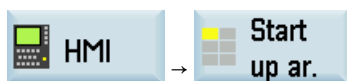
The machining operating area is displayed by default after the startup of the control system. Alternatively, you can select another operating area which you desire to enter after the system starts up.

Operating sequence

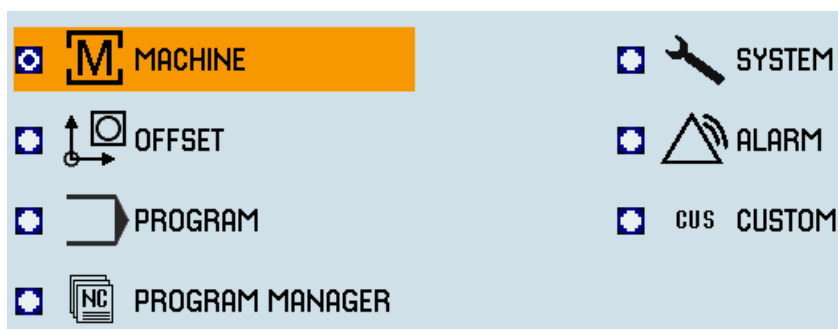


1. Select the system data operating area.

2. Open the window for setting the startup operating area through the following softkey operations:



3. Use the cursor keys to select the desired operating area in the following window:



Change
start ar.

4. Press this softkey to confirm your setting, and the configured operating system will display after the control system restarts.

10.9 Executing/transferring a part program from external

Note

You're recommended to use the Notepad editor with ANSI character encoding to create/edit part program files on the computer. Program files in other encoding formats (for example, Unicode) may cause unexpected errors after being imported into the control system.

10.9.1 Executing/transferring through the USB interface

The USB interface on the front of the PPU can be used to connect to a USB device, for example:

- An external USB memory stick, to transfer data between the USB stick and the control system
- An external USB keyboard which functions as an external NC keyboard

10.9.1.1 Executing from external (through USB interface)

Prerequisite: A USB memory stick (which includes the part program to be executed) is inserted in the front USB interface of the PPU.

Proceed as follows to execute a part program from external through the USB interface:



1. Select the program management operating area.



2. Press this softkey to enter the USB directory, and select the program file you desire to execute.

Ext. execution

3. To directly execute the program, press this softkey. The system automatically switches to "AUTO" mode in the machining operating area. The program is transferred to the buffer memory on the control system and then displayed in the following window:

Block display Current program :EXAMPLE.MPF

N5 G17 G90 G54 G71 ¶
N10 T1 D1 ¶
N15 S5000 M3 G95 F0.3 ¶
N20 G00 X100 Z2 ¶
N25 G01 Z-5 ¶
N30 X105 ¶
N35 G00 SUPA X300 Z50 D0 ¶



You can also press this key on the PPU to view the program blocks first before executing. Then press the following softkey to execute:

Execute



4. If desired, you can use this softkey to specify how you want the program to be executed. For more information of the program control, refer to Section "Program control functions (Page 54)".
5. Press this key to execute the program. The program is reloaded continuously.



Either at the end of the program or after pressing this key, the program is automatically removed from the control system.

Note

The USB directory is automatically identified if a USB stick is inserted.

Do not remove the USB stick during the external program execution through the USB interface.

10.9.1.2 Transferring from external (through USB interface)

Prerequisite: A USB memory stick (which includes the part program to be transferred) is inserted in the front USB interface of the PPU.

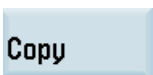
Proceed as follows to transfer a part program from external through the USB interface:



1. Select the program management operating area.



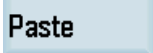
2. Press this softkey to enter the USB directory.



3. Select the program file you desire to transfer.
4. Press this softkey to copy the file to the buffer memory on the control system.



5. Enter the program directory.



6. Press this softkey to paste the copied file into the program directory.

10.9.2 Executing/transferring through the Ethernet connection

10.9.2.1 Configuring the network drive

A connected network drive allows you to access a shared directory on your computer from the control system. The network drive functions based on the Ethernet connection between the control system and a computer. The following Ethernet connections are possible:

- Direct connection: connecting the control system directly to a computer
- Network connection: integrating the control system into an existing Ethernet network

Establishing a direct connection

Proceed as follows to establish a direct connection:

1. Connect the control system with the computer using an Ethernet cable.
 2. Select the system data operating area.
-  + 
3. Press the extension key.
- 
4. Enter the main screen of the service control options through the following softkey operations:
-  → 
5. Press this softkey to set up a direct connection between the control system and the computer.
- 

The following dialog box pops up on the screen:

Link set up	
IP address:	169.254.11.22
Subnet mask:	255.255.0.0

Establishing a network connection

Proceed as follows to establish a network connection:

1. Connect the control system with the local network using an Ethernet cable.
 2. Select the system data operating area on the PPU.
-  + 
3. Press the extension key.
- 
4. Enter the main screen of the service control options through the following softkey operations:
-  → 
5. Press this softkey to enter the window for the network configuration.
- 

Direct
connect.

Note: make sure this vertical softkey is not selected.

6. Configure the network as required in the following window:

Network configuration	
Local data	
Protocol:	TCP / IP
DHCP:	Yes 
Cmpt. name:	NONAME_NCU
IP address:	176 16 202 200
Subnet mask:	255 255 255 0
Gateway:	

You can configure the DHCP with this key.

Note: if you select "No" for DHCP, you must enter the IP address (which must belong to the same network as that of your computer) and subnet mask manually.

7. Press this softkey to save the configuration. If you select "Yes" for DHCP, you also need to restart the control system to activate the network configuration.



Save

Creating and connecting a network drive

Proceed as follows to create and connect a network drive:

1. Share a directory on your local disk on your computer.
2. Select the program management operating area.
3. Press this softkey to go to the network drive directory.
4. Press this softkey to go to the window for configuring the network drives.
5. Press this key to select a drive identifier: N1, N2, or N3.



Netwo.
drive

Configure
net.drive





6. Move the cursor to the following input fields:

User:		①
Windows login		
Password:		②
Windows password (Case-sensitive. Toggle with ALT+L)		
Path:		③
Example: //Server/Share name		

①: Enter the user name of your Windows account

②: Enter the login password (case sensitive) of your Windows account

③: Enter the IP address of the server and the share name of the shared directory on your computer. Example: //140.231.196.90/808D



7. Press this softkey to confirm and the configured network drive appears on the screen as follows. The drive icon is yellow if the network drive is connected successfully; otherwise, the icon is gray.

 N1://140.231.196.90/808D

You can delete a selected network drive using this softkey.



Note

After you properly configure all the settings for the direct connection between the control system and the network drive, if the network drive connection is still invalid, contact your Windows system administrator for possible problems with your operating system configuration.

10.9.2.2 Executing from external (through Ethernet connection)

Prerequisites:

- An Ethernet connection has been established between the control system and the computer.
- A network drive (which includes the part program to be executed) has been created and connected.

Proceed as follows to execute a part program from external through the Ethernet connection:



1. Select the program management operating area.



2. Press this softkey to view the network drive(s) created.



3. Select the desired network drive (which includes the part program to be executed) and press this key to open it.
4. Select the program file you desire to execute.

Execute

- Press this softkey and the system automatically switches to "AUTO" mode in the machining operating area. The program is transferred to the buffer memory on the control system and then displayed in the following window:

```
Block display      Current program :EXAMPLE.MPF
N5 G17 G90 G54 G71 ¶
N10 T1 D1 ¶
N15 S5000 M3 G95 F0.3 ¶
N20 G00 X100 Z2 ¶
N25 G01 Z-5 ¶
N30 X105 ¶
N35 G00 SUPA X300 Z50 D0 ¶
```

NC Prog. ctrl

- If desired, you can use this softkey to specify how you want the program to be executed. For more information of the program control, refer to Section "Program control functions (Page 54)" .



- Press this key to execute the program. The program is reloaded continuously.



Either at the end of the program or after pressing this key, the program is automatically removed from the control system.

10.9.2.3 Transferring from external (through Ethernet connection)

Prerequisites:

- An Ethernet connection has been established between the control system and the computer.
- A network drive (which includes the part program to be transferred) has been created and connected.

Proceed as follows to transfer a part program from external through the Ethernet connection:



- Select the program management operating area.



- Press this softkey to view the network drive(s) created.



- Select the desired network drive (which includes the part program to be transferred) and press this key to open it.

Copy

- Select the program file you desire to transfer.
- Press this softkey to copy the file to the buffer memory on the control system.

NC NC

- Enter the program directory.

Paste

- Press this softkey to paste the copied file into the program directory.

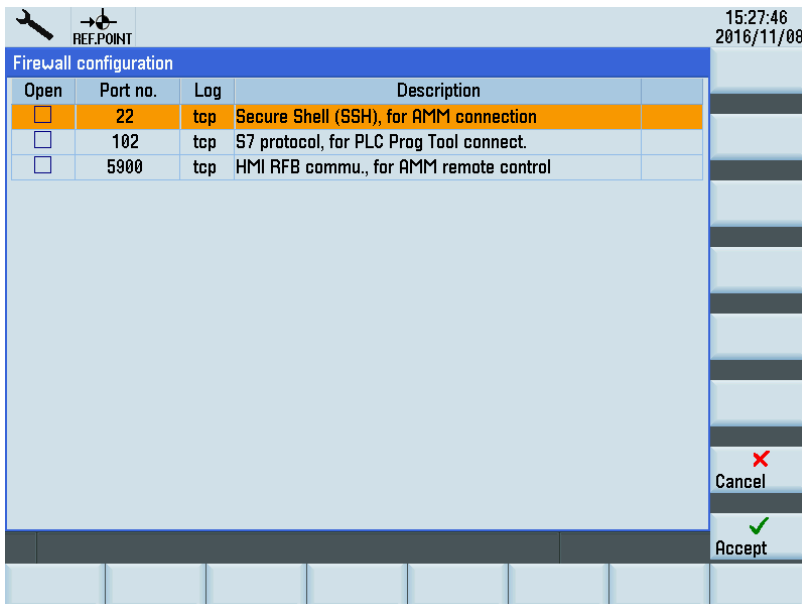
10.9.3 Configuring the firewall

Configuring the firewall

Secure access and communication is achieved through the security function of the integrated firewall. You can open the window for firewall configuration through the following operations:



Configurable ports are listed in the following window:



The ports are disabled by default and can be enabled when necessary. To change the port status, select the relevant port using the cursor keys and press either of the following keys to enable or disable the port:



WARNING

Death, severe personal injury and/or property damage due to improper firewall configuration

Improper firewall configuration may cause network security risks, for example, data leakage, virus invasion, and hacker attack. This may lead to incorrect parameterization or machine malfunction, which in turn can result in death, severe injuries and/or property damage.

- Do not use the control system inside a network infrastructure without an additional security product.
- You must make sure you disable all ports not needed in the firewall configuration.

Note

After you disable a communication port, the existing connection established earlier via this port will not be disabled until you manually disconnect it or the control system restarts/powers off. Therefore, it is recommended that you disconnect the established connections or restart the control system after disabling the ports.

10.10 Extending/deactivating the CNC lock function

To extend/deactivate the CNC lock function, you must import the corresponding activation/deactivation file (.clc format) provided by the machine manufacturer into the control system. The file can be imported either directly via an Ethernet connection (network drive) or alternatively via a storage medium (for example, USB memory stick). The control system must be in the reset state for the import.

Importing the activation/deactivation file

Proceed through the following steps to import the file:

- To import the file via USB, store the file in a USB memory stick and insert the USB memory stick into the USB interface at the front of the PPU.
 - To import the file via Ethernet, store the file in a shared folder (network drive) on your computer and connect the network drive via Ethernet connection (Page 67).

- Select the system data operating area.

- Press this key to view the extended softkeys.

- Open the license key dialog box through the following softkey operations:



- Press this softkey to open the file opening dialog box.

- Select the target directory (USB or network drive) and press this key to open it.

- Locate the desired activation/deactivation file and press this softkey to import.

After the file is imported successfully, you can check the CNC lock status on the HMI screen, for example:

- CNC lock function extended (with a new lock date):

License key

Enter the licence key to activate the option!
The option is activated after Restart!

CF card serial number: SPG2012060501317
Order No. of the NCU module: 6FC5812-2GY46-0YA0

CNC lock is activated:2017/05/01

- License key**

Enter the licence key to activate the option!
The option is activated after Restart!

CF card serial number: SPG2012060501317
Order No. of the NCU module: 6FC5812-2GY46-0YA0

CNC lock is activated.No lock data.

If an error occurs when importing the activation file, an error-specific alarm will be issued. The state of the CNC lock function remains unchanged.

We recommend that you create a complete commissioning archive over all control system components after deactivating the CNC lock function. If necessary, this commissioning archive can be used to recommission the control system without re-deactivating the CNC lock function.

If a date earlier than the actual date is set for activated CNC lock function, alarm 8065 is issued after NC restart and then NC start is disabled. In this case, you must correct the date and perform an NC restart again to clear the alarm.

If during the correcting, a future date is set inadvertently, alarm 8066 is issued. Provided no NC restart has been performed, the date can still be corrected. After NC restart, a date set in the future is considered as being an actual date and can no longer be reset.

Shorter service life

After NC restart, a future date set earlier than the lock date reduces the service life until the lock date. If a date equal to or later than the lock date is set, alarm 8064 is issued and the NC start disabled.

Make sure you set the date correctly prior to NC restart.

10.11 Data backup

10.11.1 Internal data backup

You can save the NC and PLC data of the volatile memory to the permanent memory of the control system.

Note

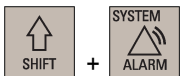
After changing important data, it is recommended to carry out an internal data backup immediately.

Backing up data internally

Prerequisite:

- A valid system password has been set on the control system.
- There is no program currently being executed.

Proceed through the following steps to save data:



1. Select the system data operating area.



2. Press this softkey to open the window for data saving.



3. Press this softkey to start saving. Do not carry out any operator actions while the data back-up is running.

Loading internally backed up data

There are two methods to load the saved data.

Method 1:

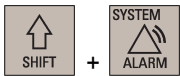


1. Press this key while the control system is booting.



2. Use the cursor keys to select "Reload saved user data" in the setup menu.
3. Press this key to confirm.

Method 2:



1. Select the system data operating area.



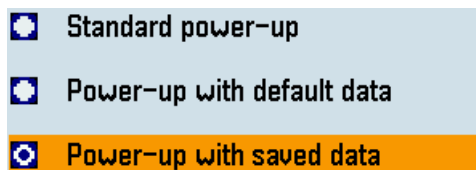
2. Press this softkey.



3. Enter the NC startup screen.



4. Use the cursor keys to select the third startup mode as follows:





5. Press this softkey to confirm. The control system restarts with the saved data.

Note

The following message is displayed on the screen after the control system starts up successfully with the saved data:



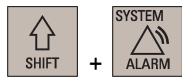
You must enter the password again after you have powered up the control system with the saved data.

10.11.2 External data backup

10.11.2.1 External data backup in a data archive

You can perform a complete data backup of the control system by creating a startup archive.

Backing up data externally in a data archive



1. Select the system data operating area.



2. Press this softkey to open the window for creating or restoring a startup archive.

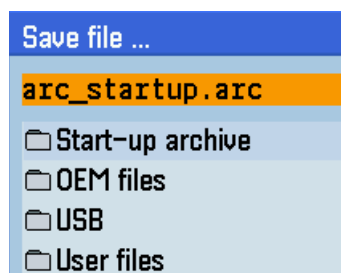
There are three options for creating a data archive. Note that options ① and ③ are visible only with the manufacturer password.



- ① Creates a data archive for series machine commissioning
- ② Creates a data archive for a complete system backup
- ③ Backs up the complete system data on the system CompactFlash Card (CF card)



3. Select option ② and press this softkey to confirm, and the dialog box for saving the archive file opens.



The name of the data archive is "arc_startup.arc" by default. You can also use your favorite name. The file name extension ".arc" must be entered.

4. Select your desired directory and press this key to open it.



Note that you can press this key on the PPU to toggle between the file name input field and the directory selection area.



5. Press this softkey to confirm and the archive information dialog box opens.



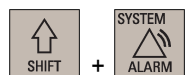
6. Specify the properties of the archive and press this softkey. The control system starts creation of the startup archive.

Note

Do not remove the USB stick in the process of data backup if you choose USB as the target directory.

Pressing <CTRL + S> when you are in any operating area creates a startup archive on the connected USB stick. In addition, it automatically saves the action log to the USB stick.

Restoring a startup archive



1. Select the system data operating area.



2. Press this softkey to open the startup archive window.



3. Select the following option and press this softkey to restore the startup archive:

Restore start-up archive



4. Select the backup path to locate the archive file and press this softkey to confirm.



5. Press this softkey to confirm the archive information.



6. Press this softkey to continue and start restoring the startup archive. The control system restarts to complete restoring the archive.

10.11.2.2 External data backup of separate files

In the program management operating area, program files or directories can be copied into another directory or onto a different drive by means of copying and pasting operations.

Operating sequence



1. Select the program management operating area.



2. Enter the program directory.



3. Select the program file or directory to be backed up.
4. Press this softkey to copy the file to the clipboard.



5. Select a desired directory or drive as the data target.
Backs up the files in the folder for storing the user cycles on the control system. This folder is visible with the manufacturer password.



Backs up the files onto an USB stick.



Backs up the files onto a computer. This requires a connected network drive on the control system. For more information, see section "Configuring the network drive (Page 67)".



Backs up the files in the folder for storing the manufacturer files on the control system. This folder is visible with the manufacturer password.



Backs up the files in the folder for storing end user files on the control system.



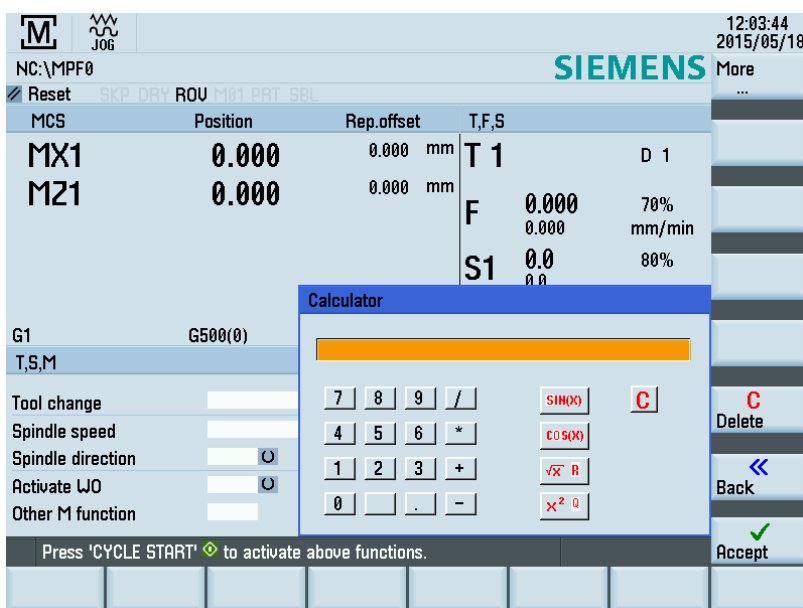
6. Press this softkey to paste the copied data into the current directory.

For more information about data backup, refer to the SINUMERIK 808D/SINUMERIK 808D ADVANCED Diagnostics Manual.

10.12 Pocket calculator



You can activate the calculator function by pressing this key on the PPU when you position the cursor on the desired input field to calculate the cycle relevant parameters, contour parameters, tool parameters, workpiece offsets, R parameters, etc.



For calculating, the four basic arithmetic operations are available, as well as the functions "sine", "cosine", "squaring" and "square root". A bracket function is provided to calculate nested terms. The bracket depth is unlimited.

If the input field is already occupied by a value, the function will accept this value into the input line of the pocket calculator.



Pressing this softkey empties the input line of the calculator.



After inputting a desired arithmetic statement in the input line of the calculator, pressing this key starts the calculation. The result is displayed in the pocket calculator.



Selecting this softkey enters the result in the input field at the current cursor position and closes the pocket calculator automatically.



Pressing this softkey aborts the calculation result (if any) and exits the pocket calculator.

Characters that may be entered

+, -, *, /	Basic arithmetic operations
S	Sine function The X value (in degrees) in front of the input cursor is replaced by the sin(X) value.
O	Cosine function The X value (in degrees) in front of the input cursor is replaced by the cos(X) value.
Q	Square function The X value in front of the input cursor is replaced by the X^2 value.
R	Square root function The X value in front of the input cursor is replaced by the $\sqrt{X}$ value.
()	Bracket function (X+Y)*Z

Calculation examples

Task	Input -> Result
100 + (67*3)	100+67*3 -> 301
sin(45_)	45 S -> 0.707107
cos(45_)	45 O -> 0.707107
4 ²	4 Q -> 16
$\sqrt{4}$	4 R -> 2
(34+3*2)*10	(34+3*2)*10 -> 400

To calculate auxiliary points on a contour, the pocket calculator offers the following functions:

- Calculating the tangential transition between a circle sector and a straight line
- Moving a point in the plane
- Converting polar coordinates to Cartesian coordinates
- Adding the second end point of a straight line/straight line contour section given from an angular relation

11 Programming principles

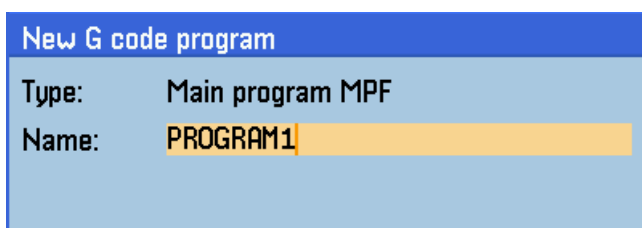
11.1 Fundamentals of programming

11.1.1 Program names

Each program must have a program name. The program name must follow the conventions below:

- Use a maximum of 24 letters or 12 Chinese characters for a program name (the character length of the file extension excluded)
- Separate the file extension only with a decimal point
- Enter the file extension ".SPF" if the current default program type is MPF (main program) and you desire to create a subprogram
- Enter the file extension ".MPF" if the current default program type is SPF (subprogram) and you desire to create a main program
- Do not enter the file extension if you desire to take the current default program type
- Avoid using special characters for program names.

Example



New G code program

Type: Main program MPF

Name: PROGRAM1

11.1.2 Program structure

Structure and content

The NC program consists of a sequence of **blocks** (see the table below). Each block represents a machining step. Instructions are written in the blocks in the form of **words**. The last block in the execution sequence contains a special word for the end of the program, for example, **M2**.

The following table shows you an example of the NC program structure.

Block	Word	Word	Word	...	; Comment
Block	N10	G0	X20	...	; First block
Block	N20	G2	Z37	...	; Second block
Block	N30	G91	...	...	; ...
Block	N40	...	...	...	
Block	N50	M2			; End of program

11.2 Programming instructions

11.2.1 Programming dimensions

In this section you will find descriptions of the commands, with which you can directly program dimensions taken from a drawing. This has the advantage that no extensive calculations have to be made for NC programming.

Note

The commands described in this section stand in most cases at the start of an NC program. The way, in which these functions are combined, is not intended to be a patent remedy. For example, the choice of working plane may be made at another point in the NC program. The real purpose of this and the following sections is to illustrate the conventional structure of an NC program.

Overview of typical dimensions

The basis of most NC programs is a drawing with concrete dimensions.

When implementing in a NC program, it is helpful to take over exactly the dimensions of a workpiece drawing into the machining program. These can be:

- Absolute dimension, G90 modally effective applies for all axes in the block, up to revocation by G91 in a following block.
- Absolute dimension, X=AC(value) only this value applies only for the stated axis and is not influenced by G90/G91. This is possible for all axes and also for SPOS, SPOSA spindle positionings, and interpolation parameters I, J, K.
- Absolute dimension, X=DC(value) directly approaching the position by the shortest route, only this value applies only for the stated rotary axis and is not influenced by G90/G91. This is also possible for SPOS, SPOSA spindle positionings.
- Absolute dimension, X=ACP(value) approaching the position in positive direction, only this value is set only for the rotary axis, the range of which is set to 0... < 360 degrees in the machine data.
- Absolute dimension, X=ACN(value) approaching the position in negative direction, only this value is set only for the rotary axis, the range of which is set to 0... < 360 degrees in the machine data.
- Incremental dimension, G91 modally effective applies for all axes in the block, until it is revoked by G90 in a following block.
- Incremental dimension, X=IC(value) only this value applies exclusively for the stated axis and is not influenced by G90/G91. This is possible for all axes and also for SPOS, SPOSA spindle positionings, and interpolation parameters I, J, K.
- Inch dimension, G70 applies for all linear axes in the block, until revoked by G71 in a following block.
- Metric dimension, G71 applies for all linear axes in the block, until revoked by G70 in a following block.
- Inch dimension as G70, however, G700 applies also for feedrate and length-related setting data.
- Metric dimension as G71, however, G710 applies also for feedrate and length-related setting data.
- Diameter programming, DIAMON on
- Diameter programming, DIAMOF off

Diameter programming, DIAM90 for traversing blocks with G90. Radius programming for traversing blocks with G91.

11.2.2 Absolute/incremental dimensioning: G90, G91, AC, IC

Functionality

With the instructions G90/G91, the written positional data X, Z, ... are evaluated as a coordinate point (G90) or as an axis position to traverse to (G91). G90/G91 applies for all axes.

Irrespective of G90/G91, certain positional data can be specified for certain blocks in absolute/incremental dimensions using AC/IC.

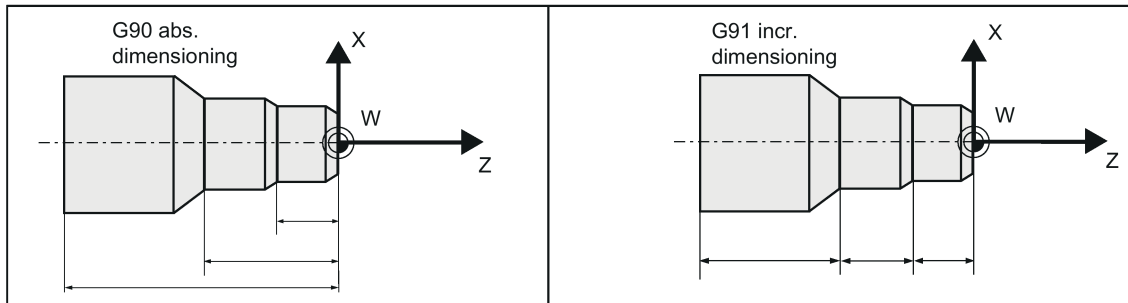
These instructions do **not determine the path** by which the end points are reached; this is provided by a G group (G0, G1, G2 and G3...). For more information, see Sections "Linear interpolation (Page 88)" and "Circular interpolation (Page 90)".

Programming

G90 ; Absolute dimension data
G91 ; Incremental dimension data

Z=AC(...) ; Absolute dimensioning for a certain axis (here: Z axis), non-modal
Z=IC(...) ; Incremental dimensioning for a certain axis (here: Z axis), non-modal

See the following different dimensioning types in the drawing:



Absolute dimensioning G90

With absolute dimensioning, the dimensioning data refers to the **zero of the coordinate system currently active** (workpiece or current workpiece coordinate system or machine coordinate system). This is dependent on which offsets are currently active: programmable, settable, or no offsets.

Upon program start, G90 is active for **all axes** and remains active until it is deselected in a subsequent block by G91 (incremental dimensioning data) (modally active).

Incremental dimensioning G91

With incremental dimensioning, the numerical value of the path information corresponds to the **axis path to be traversed**. The leading sign indicates the **traversing direction**.

G91 applies to all axes and can be deselected in a subsequent block by G90 (absolute dimensioning).

Specification with =AC(...), =IC(...)

After the end point coordinate, write an equality sign. The value must be put in round brackets.

Absolute dimensioning is also possible for center points using =AC(...). Otherwise, the reference point for the circle center is the circle starting point.

Programming example

```
N10 G90 X20 Z90 ; Absolute dimensions
N20 X75 Z=IC(-32) ; X-dimensions remain absolute, incremental Z dimension
N180 G91 X40 Z2 ; Switch-over to incremental dimensioning
N190 X-12 Z=AC(17) ; X-remains incremental dimensioning, Z-absolute
```

11.2.3 Dimensions in metric units and inches: G71, G70, G710, G700

Functionality

If workpiece dimensions that deviate from the base system settings of the control system are present (inch or mm), the dimensions can be entered directly in the program. The required conversion into the base system is performed by the control system.

Programming

G70 ; Inch dimensions
G71 ; Metric dimensions
G700 ; Inch dimensions, also for feedrate F
G710 ; Metric dimensions, also for feedrate F

Programming example

```
N10 G70 X10 Z30 ; Inch dimensions
N20 X40 Z50 ;G70 continues to act
N80 G71 X19 Z17.3 ; metric dimensioning from this point on
```

Information

Depending on the **default setting** you have chosen, the control system interprets all geometric values as either metric **or** inch dimensions. Tool offsets and settable work offsets including their displays are also to be understood as geometrical values; this also applies to the feed F in mm/min or inch/min.

The default setting can be set in machine data.

All examples provided in this manual assume the **metric default setting**.

G70 or G71 evaluates all geometrical data that directly refer to the **workpiece**, either as inches or metric units, for example:

- Positional data X, Z, ... for G0,G1,G2,G3,G33, CIP, CT
- Interpolation parameters I, K (also thread lead)
- Circle radius CR
- **Programmable** work offset (TRANS, ATRANS)

All remaining geometric parameters that are not direct workpiece parameters, such as feedrates, tool offsets, and **settable** work offsets, are not affected by **G70/G71**.

G700/G710, however, also affects the feedrate F (inch/min, inch/rev. or mm/min, mm/rev.).

11.2.4 Radius/diameter dimensions: DIAMOF, DIAMON, DIAM90

Functionality

For machining parts, the positional data for the **X axis** (transverse axis) is programmed as diameter dimensioning. When necessary, it is possible to switch to radius dimensioning in the program.

DIAMOF or DIAMON assesses the end point specification for the X axis as radius or diameter dimensioning. The actual value appears in the display accordingly for the workpiece coordinate system.

For DIAM90, irrespective of the traversing method (G90/G91), the actual value of the transverse axis is always displayed as a diameter. This also applies to reading of actual values in the workpiece coordinate system with MEAS, MEAW, \$P_EP[x] and \$AA_IW[x].

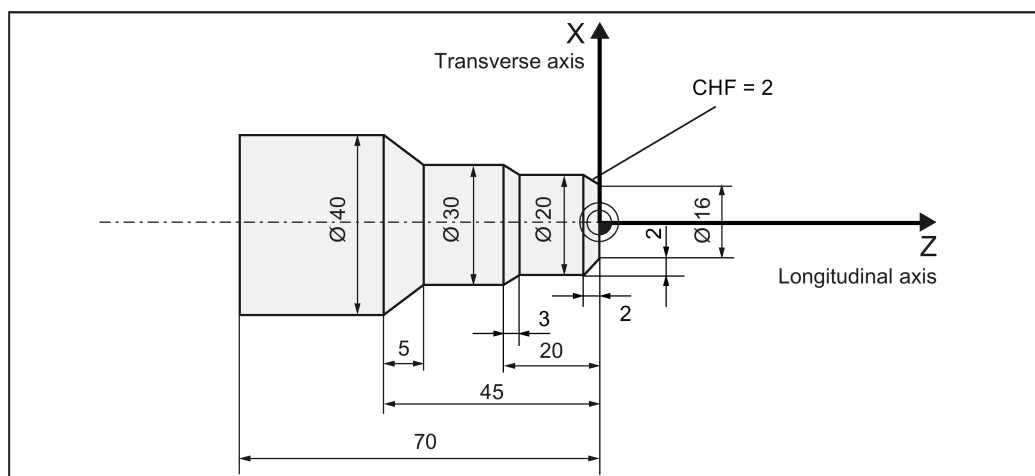
Programming

DIAMOF ; Radius dimensioning

DIAMON ; Diameter dimensioning

DIAM90 ; diameter dimensioning for G90, radius dimensioning for G91

See the following diameter and radius dimensioning for the traverse axis:



Programming examples

Diameter dimensions (DIMON)

```
DIAMON                      ; Diameter dimensions active
G1 X0 Z0 F0.2               ; Traverse to diameter position X0 and Z0
X20 CHF=2
Z-17
X30 Z-20
Z-40
X40 Z-45
Z-70
M30                          ; End of program
```

Radius dimensions (DIMOF)

```
DIMOF                       ; Diameter dimensions inactive
G1 X0 Z0 F0.2               ; Traverse to diameter position X0 and Z0
X10 CHF=2
Z-17
X15 Z-20
Z-40
X20 Z-45
Z-70
M30                          ; End of program
```

Note

A programmable offset with TRANS X... or ATRANS X... is always evaluated as radius dimensioning. Description of this function: see the next section.

11.2.5 Programmable work offset: TRANS, ATRANS

Functionality

The programmable work offset can be used:

- for recurring shapes/arrangements in various positions on the workpiece
- when selecting a new reference point for the dimensioning
- as a stock allowance when roughing

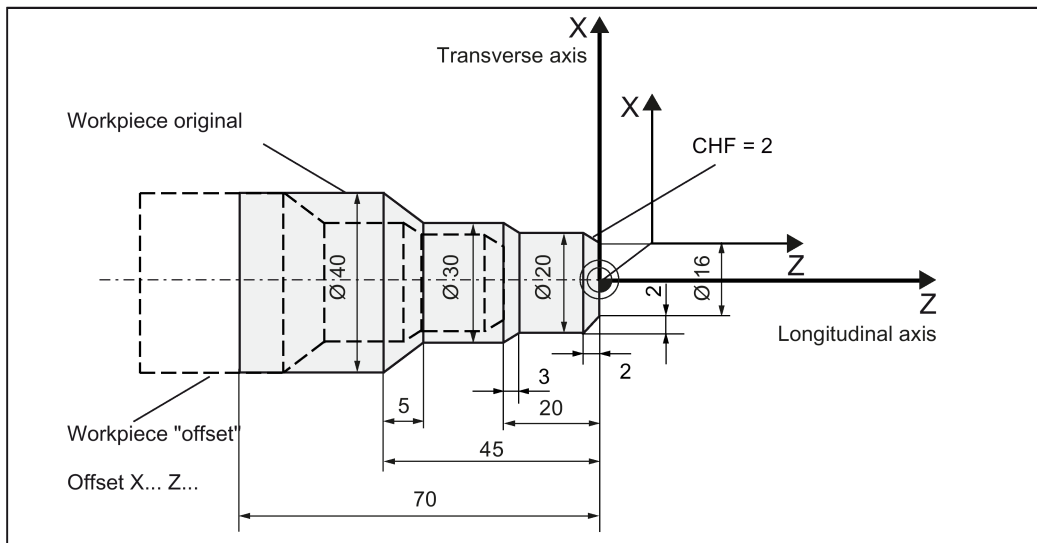
This results in the **current workpiece coordinate system**. The rewritten dimensions use this as a reference.

The offset is possible in all axes.

Note

In the X axis, the workpiece zero should be in the turning center due to the functions of diameter programming (DIAMON) and constant cutting speed (G96). For this reason, use no offset or only a small offset (e.g. as allowance) in the X axis.

See the following effect of the programmable offset:



Programming

TRANS Z... ; programmable offset, deletes old instructions for offsetting, rotation, scaling factor, mirroring
 ATRANS Z... ; programmable offset, additive to existing instructions
 TRANS ; without values: clears old instructions for offset, rotation, scaling factor, mirroring

The instructions that contain TRANS or ATRANS each require a separate block.

Programming example 1

```
DIAMON ; Diameter dimensions active
TRANS Z-20 ; Programmable offset, 5 mm in the Z axis
G1 X0 Z0 F0.2 ; Traverse to diameter position X0 and Z0
X16
X20 Z-3
Z-17
X30 Z-20
Z-40
X40 Z-45
Z-70
M30 ; End of program
```

Subroutine call: see Section "Subroutine technique (Page 133)".

Programming example 2

```
G90 G18 G500
T3D1
M4S1500
G0X50 Z10
CYCLE95( "CON1:CON1_E", 0.50000, 0.20000, 0.20000, ,0.20000, 0.20000, 0.15000, 9, , ,2.00000)
```

```
M4S1200
G0X100Z-10
R0=46
LAB1:
```

```

TRANS X=R0 Z-25
AROT RPL=-10
R1=-45
R2=14
R3=34
LAB:
TRANS X=R0 Z-25
AROT RPL=10
R5=R2*COS(R1)
R6=R3*SIN(R1)
G1 Z=R5 X=R6
R1=R1-0.5
IF R1>=-151 GOTOB LAB
R0=R0-0.5
IF R0>=40 GOTOB LAB1
G0X80
Z50
AROT
TRANS

G500
T5D1
M4S1000
G1F0.1
CYCLE93( 58.00000, -36.00000, 22.00000, 0.90000, , , , , , , 0.10000, 0.10000, 0.50000, 0.10000, 5, 2.00000)
G0X80
Z50

T3D1
M4S1500
R0=29
BB:
TRANS Z-52 X=R0
DIAMOF
R4=720
LL:
R1=(3.14159*R4)/180
R2=SIN(R4)
G1 X=R2 Z=R1
R4=R4-0.5
IF R4>=0 GOTOB LL
DIAMON

```

```

R0=R0-0.5
IF R0>=27 GOTOB BB
G0X80
Z50
M30

,*****CONTOUR*****
CON1:
X42Z0
X54Z-13
X58
Z-60
X60
M02
CON1_E,;***** CONTOUR ENDS *****

```

11.2.6 Programmable scale factor: SCALE, ASCALE

Functionality

A scale factor can be programmed for all axes with SCALE, ASCALE. The path is enlarged or reduced by this factor in the axis specified.

The currently set coordinate system is used as the reference for the scale change.

Programming

SCALE X... Z... ; Programmable scaling factor, clears old instructions for offset, rotation, scaling factor, mirroring

ASCALE X... Z... ; Programmable scaling factor, additive to existing instructions

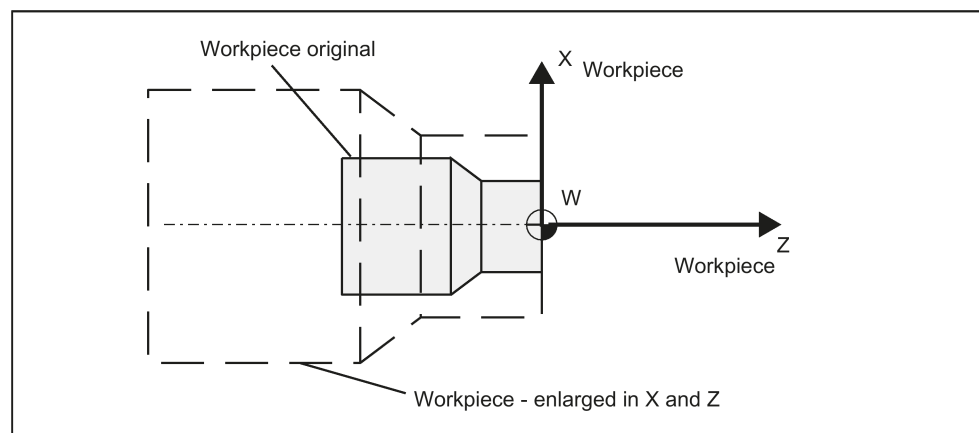
SCALE ; Without values: clears old instructions for offset, rotation, scaling factor, mirroring

The instructions that contain SCALE or ASCALE each require a separate block.

Notes

- For circles, the same factor should be used in both axes.
- If an ATRANS is programmed with SCALE/ASCALE active, these offset values are also scaled.

See the following example of a programmable scale factor:



Programming example

```
N10 L10 ; Programmed contour original
N20 SCALE X2 Z2 ; contour in X and Z enlarged 2 times
N30 L10
N40 ATRANS X2.5 Z1.8
N50 L10
N60 M30
```

Subroutine call - see Section "Subroutine technique (Page 133)".

Information

In addition to the programmable offset and the scale factor, the following functions exist:

- Programmable rotation ROT, AROT and
- programmable mirroring, MIRROR, AMIRROR.

These functions are primarily used in milling.

Examples of rotation and mirroring: see Section "List of instructions (Page 285)".

11.2.7 Workpiece coordinate system- settable work offset: G54 to G59, G500, G53, G153

Functionality

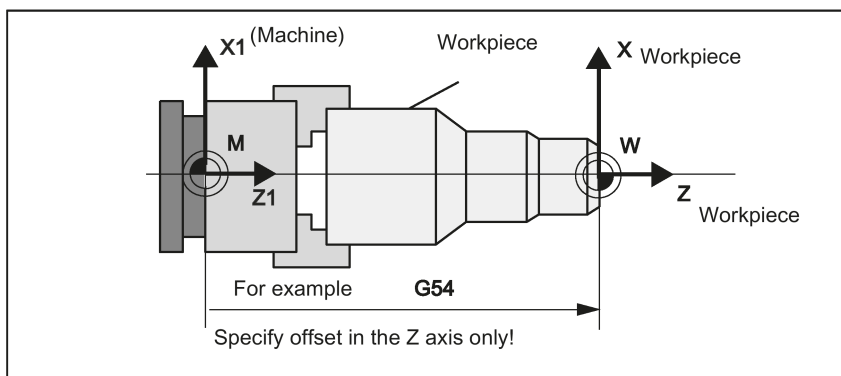
The adjustable work offset specifies the position of the workpiece zero on the machine (offset of the workpiece zero with respect to the machine zero point). This offset is determined upon clamping of the workpiece into the machine and must be entered in the corresponding data field by the operator. The value is activated by the program by selection from six possible groupings: G54 to G59.

Programming

G54 to G59 ; 1. to 6th settable work offset
G500 ; Settable work offset OFF - modal

G53 ; settable work offset OFF non-modal, also suppresses programmable offset
G153 ; As with G53; additionally suppresses base frame

See the following illustration for settable work offset:



Programming example

```
N10 G54 G0 X50 Z135
N20 X70 Z160
N30 T1 D1
N40 M3 S1000
N50 G0 X20 Z130
```

N60 G01 Z150 F0.12
N70 X50 F0.1
N80 G500 X100 Z170
N90 M30

11.3 Linear interpolation

11.3.1 Linear interpolation with rapid traverse: G0

Functionality

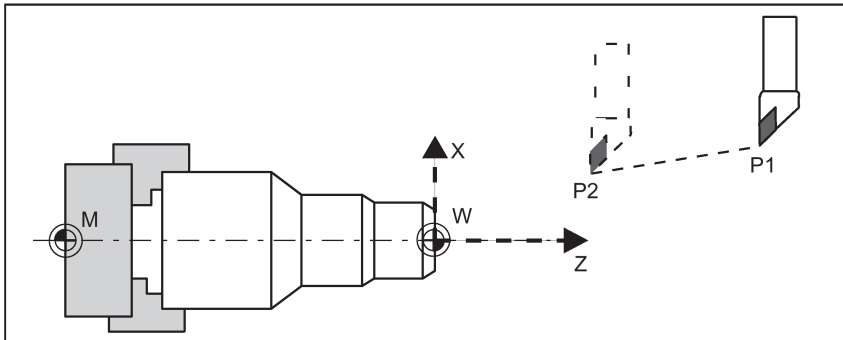
The rapid traverse movement G0 is used for fast positioning of the tool, however, **not for direct workpiece machining**. All axes can be traversed simultaneously - on a straight path.

For each axis, the maximum speed (rapid traverse) is defined in machine data. If only one axis traverses, it uses its rapid traverse. If two axes are traversed simultaneously, the path velocity (resulting velocity) is selected to achieve the **maximum possible path velocity** in consideration of both axes.

Any programmed feedrates (F word) are not relevant for G0.

G0 remains active until canceled by another instruction from this G group (G0, G1, G2, G3, ...).

See the following illustration for linear interpolation with rapid traverse from point P1 to P2:



Programming example

```
| N10 G0 X100 Z65
```

Note

Another option for linear programming is available with the angle specification ANG=. (For more information, see Section "Contour definition programming (Page 114)".)

Information

Another group of G functions exists for moving into the position (see Section "Exact stop/continuous-path control mode: G9, G60, G64 (Page 100)"). For G60 exact stop, a window with various precision values can be selected with another G group. For exact stop, an alternative instruction with non-modal effectiveness exists: G9. You should consider these options for adaptation to your positioning tasks.

11.3.2 Feedrate F

Functionality

The feed F is the **path velocity** and represents the value of the geometric sum of the velocity components of all axes involved. The axis velocities are determined from the share of the axis path in the overall path.

The feedrate F is effective for the interpolation types G1, G2, G3, CIP, and CT and is retained until a new F word is written. For more information, see Sections "Linear interpolation with feedrate: G1 (Page 89)" and "Circular interpolation: G2, G3 (Page 90)".

Programming

F...

Remark: For **integer values**, the decimal point is not required, e.g.: F300

Unit of measure for F with G94, G95

The dimension unit for the F word is determined by G functions:

- G94 F as feedrate in **mm/min**
- G95 F as feedrate in **mm/rev.** (only relative to the spindle speed!)

Remark:

This unit of measure applies to metric dimensions. According to Section "Metric and inch dimensioning", settings with inch dimensioning are also possible.

Programming example

```
N10 G94 F310 ; Feedrate in mm/min
N20 G01 X60 Z60
N30 M5
N40 S200 M3 ; Spindle rotation
N50 G95 F0.8 ; Feedrate in mm/revolution
N60 G01 X100 Z100
N70 M30
```

Remark: Write a new F word if you change G94 - G95.

Information

The G group with G94, G95 also contains the functions G96, G97 for the constant cutting rate. These functions also influence the S word.

11.3.3 Linear interpolation with feedrate: G1

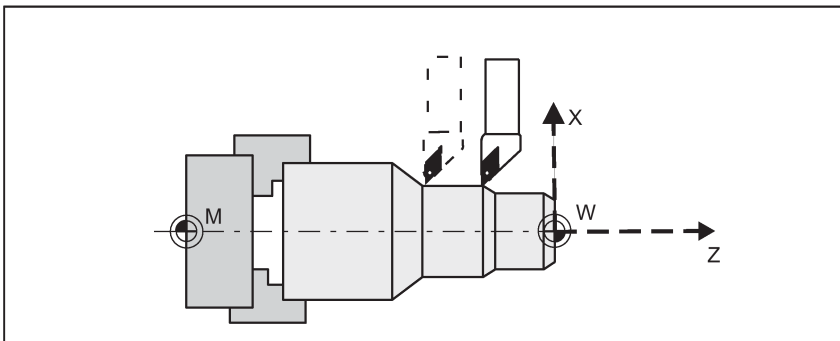
Functionality

The tool moves from the starting point to the end point along a straight path. For the **path velocity**, is determined by the programmed **F word**.

All the axes can be traversed simultaneously.

G1 remains active until canceled by another instruction from this G group (G0, G2, G3...).

See the following illustration for linear interpolation with G1:



Programming example

```

N05 G54 G0 G90 X40 Z200 S500 M3           ; The tool traverses in rapid traverse, spindle speed
                                           = 500 r.p.m., clockwise
N10 G1 Z120 F0.15                         ; Linear interpolation with feedrate 0.15
                                           mm/revolution
N15 X45 Z105
N20 Z80
N25 G0 X100                               ; Retraction in rapid traverse
N30 M2                                     ; End of program

```

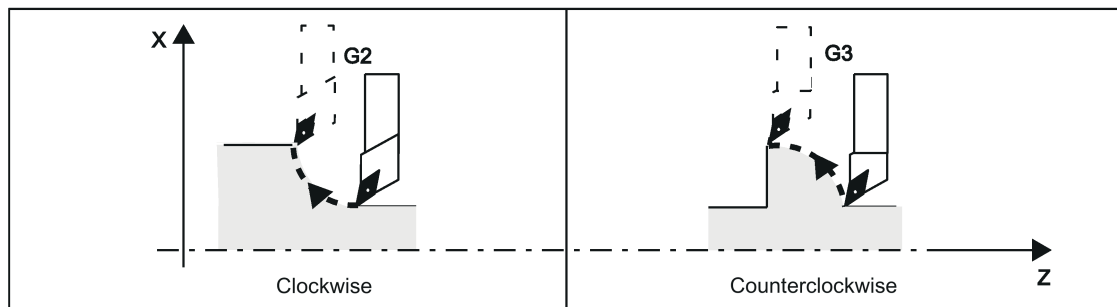
Note: Another option for linear programming is available with the angle specification ANG=.

11.4 Circular interpolation

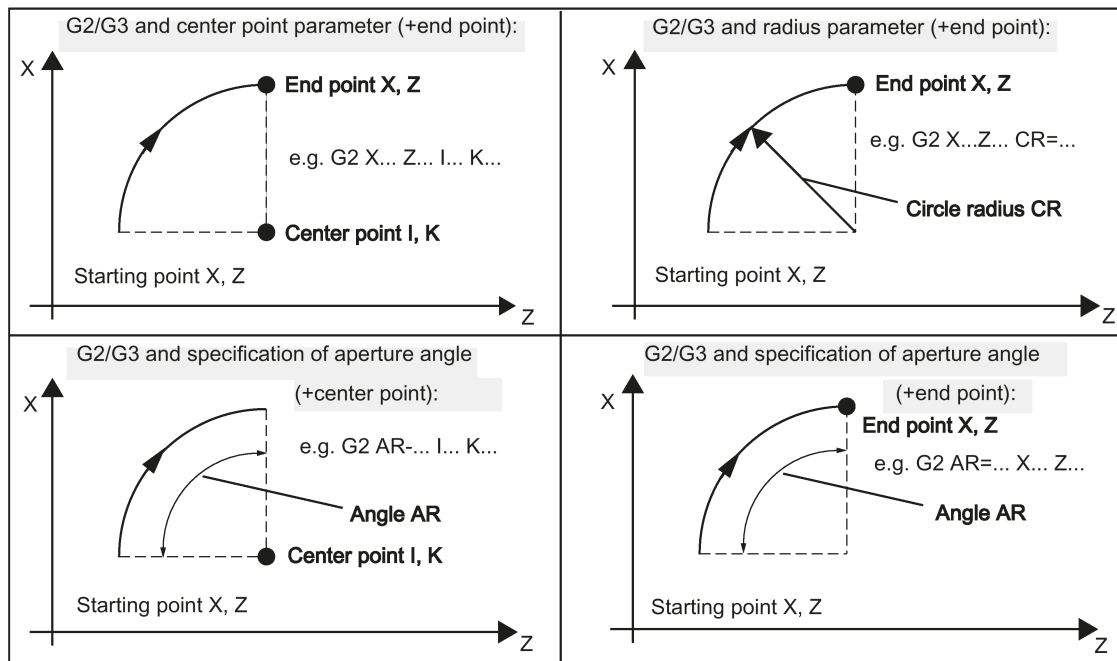
11.4.1 Circular interpolation: G2, G3

Functionality

The tool moves from the starting point to the end point along a circular path. The direction is determined by the G function:



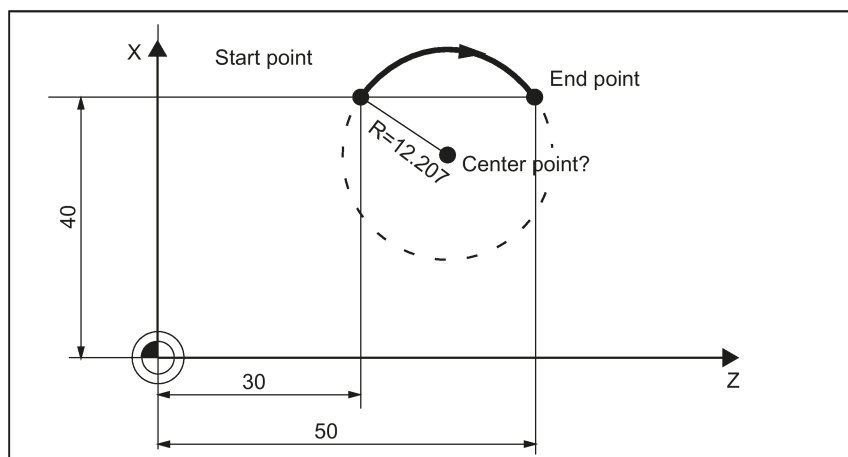
The description of the desired circle can be given in various ways:



G2/G3 remains active until canceled by another instruction from this G group (G0, G1...).

The **path velocity** is determined by the programmed **F word**.

Programming example: End point and radius specification

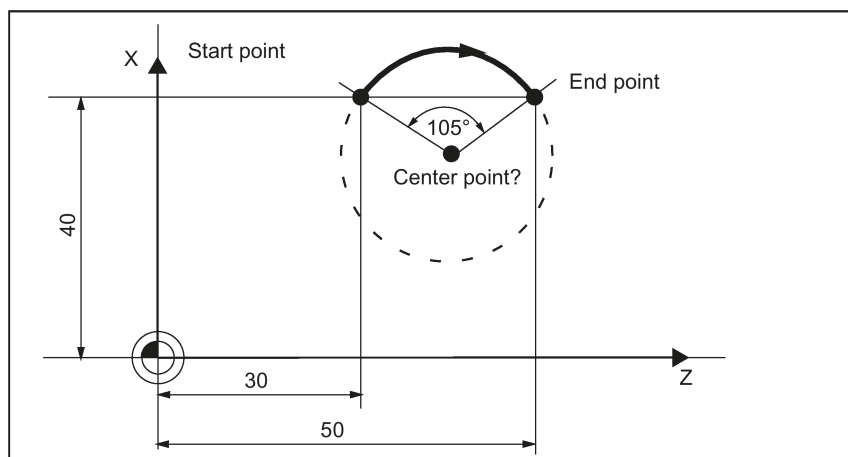


```
N5 G90 Z30 X40 ; Starting point circle for N10
N10 G2 Z50 X40 CR=12.207 ; End point and radius
```

Note

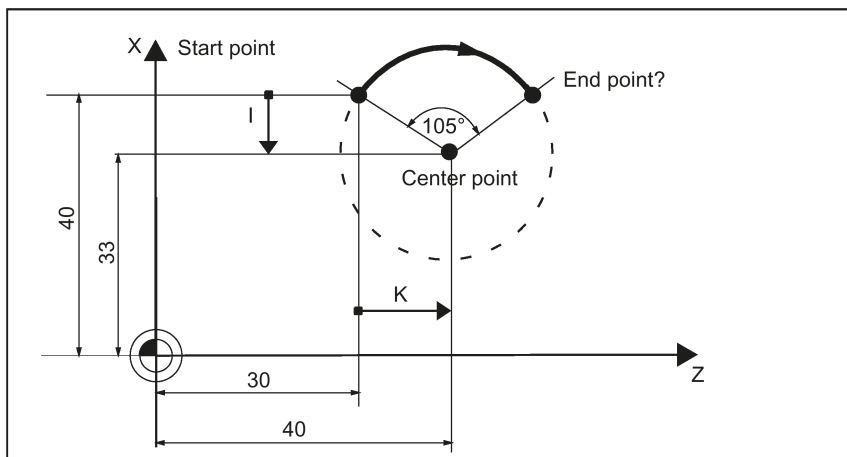
With a negative leading sign for the value with CR=-..., a circular segment larger than a semicircle is selected.

Programming example: Definition of end point and aperture angle



```
N5 G90 Z30 X40 ; Starting point circle for N10
N10 G2 Z50 X40 AR=105 ; Opening angle and end point
```

Programming example: Definition of center point and aperture angle



```
N5 G90 Z30 X40 ; Starting point circle for N10
N10 G2 K10 I=7 AR=105 ; Opening angle and center point
```

Note

Center point values refer to the circle starting point!

11.4.2 Circular interpolation via intermediate point: CIP

Functionality

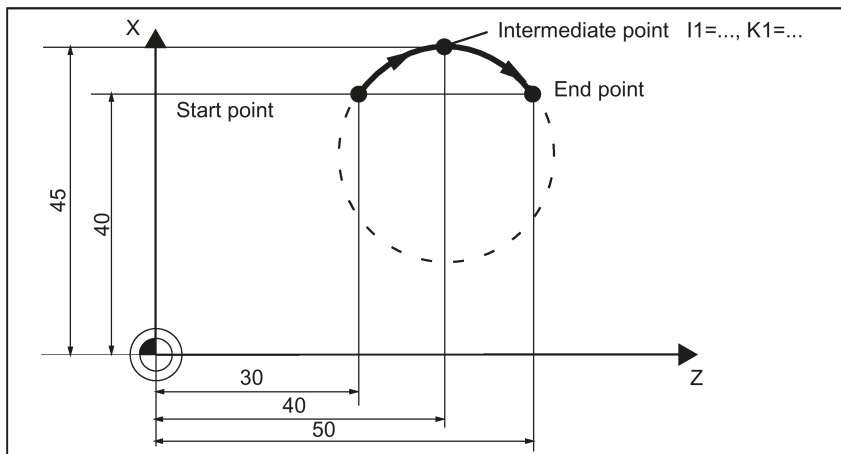
The direction of the circle results here from the position of the intermediate point (between starting and end points).

Specification of intermediate point: I1=... for the X axis, K1=... for the Z axis.

CIP remains active until canceled by another instruction from this G group (G0, G1...).

The configured dimensional data G90 or G91 applies to the end point **and** the intermediate point.

See the following illustration for circle with end point and intermediate point specification:



Programming example

```
N5 G90 Z30 X40 ; Starting point circle for N10
N10 CIP Z50 X40 K1=40 I1=45 ; End point and intermediate point
```

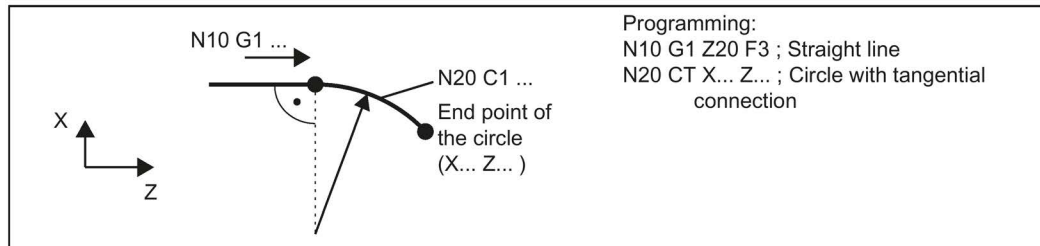
11.4.3 Circle with tangential transition: CT

Functionality

With CT and the programmed end point in the current plane (G18: Z/X plane), a circle is produced which tangentially connects to the previous path segment (circle or straight line).

This defines the radius and center point of the circle from the geometric relationships of the previous path section and the programmed circle end point.

See the following illustration for circle with tangential transition to the previous path section:



11.5 Thread cutting

11.5.1 Thread cutting with constant lead: G33

Functionality

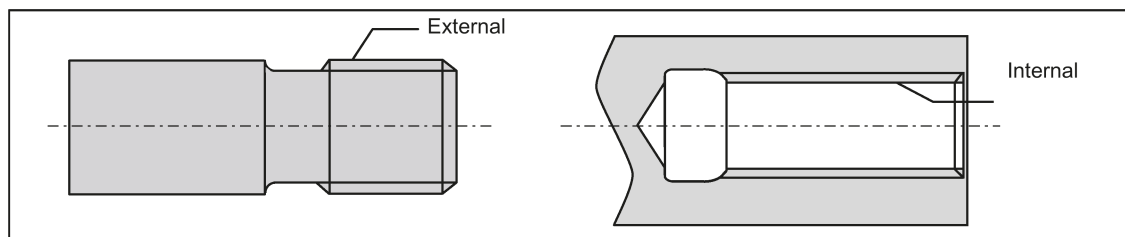
The function G33 can be used to machine threads with constant lead of the following type:

- Thread on cylindrical structures
- Thread on tapered structures
- External thread
- Single- and multiple-start thread
- Multi-block thread (series of threads)

This requires a spindle with position measuring system.

G33 remains active until canceled by another instruction from this G group (G0, G1, G2, G3...).

See the following illustration for external and internal thread with cylindrical thread:



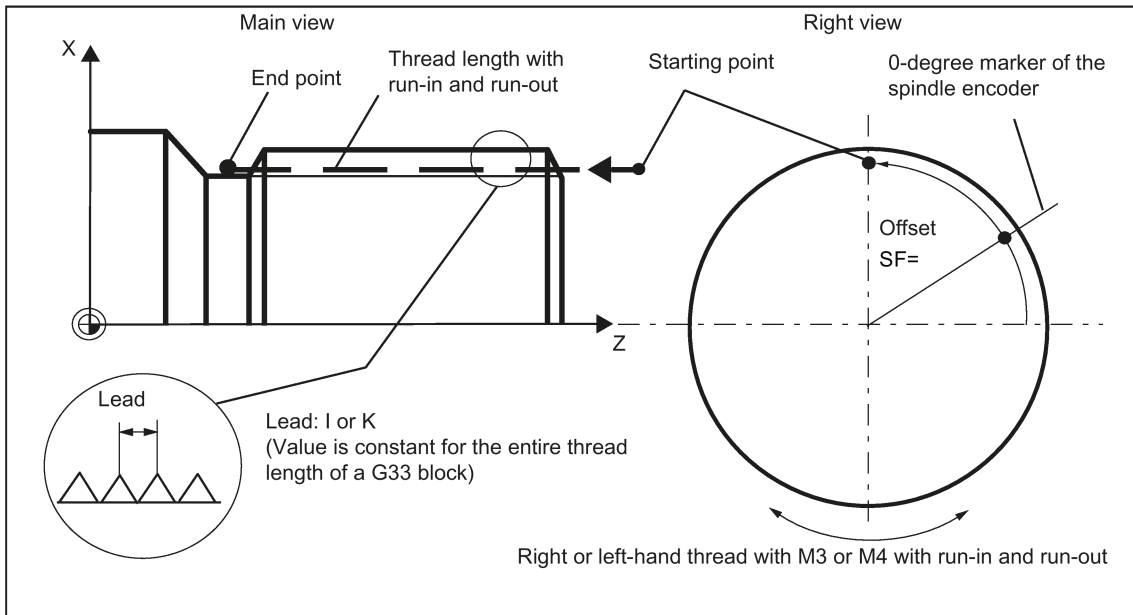
Right-hand or left-hand thread

Right-hand or left-hand thread is set with the rotation direction of the spindle (M3 right, M4 left). To do this, the rotation value must be programmed under address S or a rotation speed must be set.

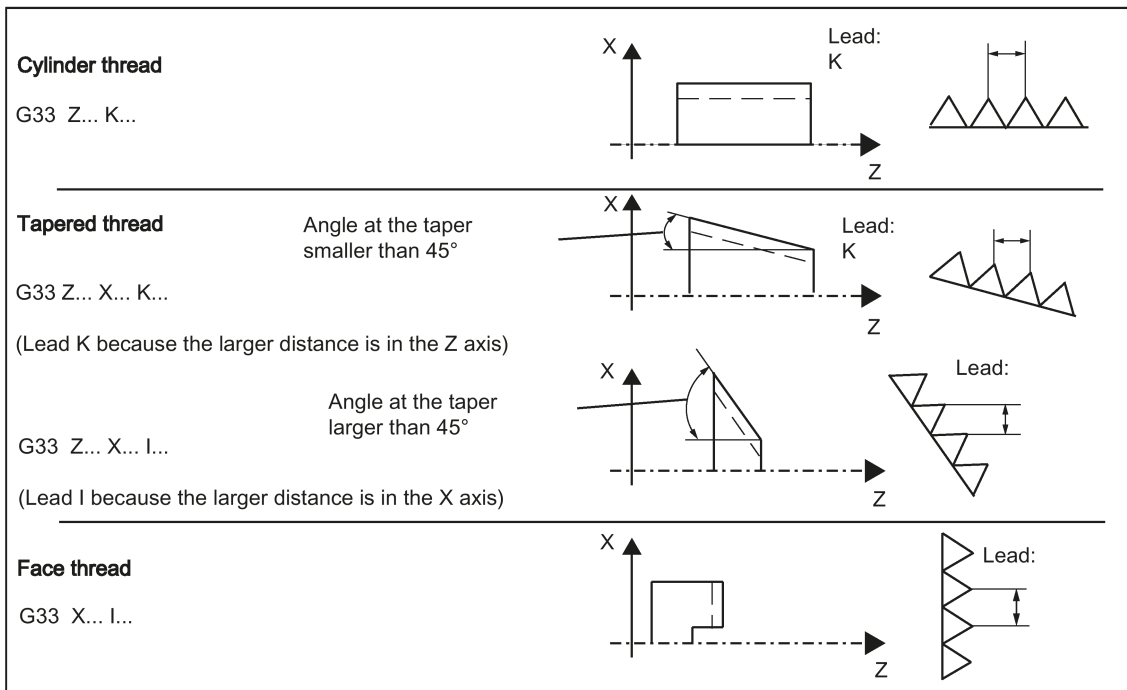
Programming

Remark: Run-in and run-out paths must be taken into account for the thread lengths.

See the following illustration for programmable values for the thread with G33:



See the following illustration for lead assignment for cylindrical, tapered and face thread:



Tapered thread

For tapered threads (2 axis values required), the corresponding lead address I or K of the axis with the **larger travel** (longer thread) must be used. A second lead is not defined.

Starting point offset SF=

A starting point offset is required for the spindle if a multiple-start thread or a thread in offset sections is to be machined. The starting point offset is programmed in the thread block with G33 under the address **SF** (absolute position). If no starting point offset SF is written, the value from the setting data "Starting angle of thread" (SD42000: THREAD_START_ANGLE) is active.

Note

A programmed value for SF must always be entered in the setting data.

Programming example

Cylindrical thread, double-thread, starting point offset 180 degrees, thread length (including run-in and run-out) 100 mm, thread lead 4 mm/rev.

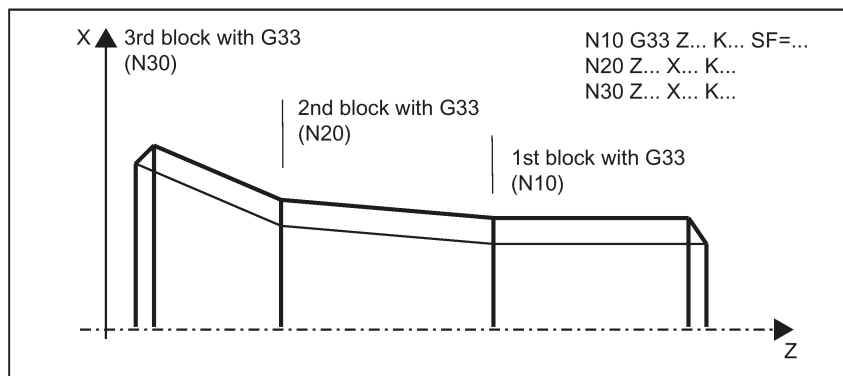
```
N10 G54 G0 G90 X50 Z0 S500 M3           ; Approach starting point, clockwise spindle rotation
N20 G33 Z-100 K4 SF=0                   ;Lead: 4 mm/rev
N30 G0 X54
N40 Z0
N50 X50
N60 G33 Z-100 K4 SF=180                 ; 2nd thread, offset by 180 degrees
N70 G0 X54
N80 Z0
N90 G0X50Z50
N100 M30
```

Multi-block thread

If multiple thread blocks are programmed consecutively (multi-block thread), it only makes sense to define a starting point offset in the first thread block.

Multi-block threads are connected automatically in G64 continuous path mode.

See the following example of multi-block thread chaining:



Axis velocity

When cutting threads with G33, you can determine the axis velocity based on the spindle speed and the thread lead. However, the maximum axis velocity (rapid traverse) defined in the machine data cannot be exceeded; otherwise, alarms will appear.

Information on the override switch

- The spindle speed override switch must remain unchanged for thread machining.
- The feedrate override switch has no meaning in this block.

11.5.2 Programmable run-in and run-out path for G33: DITS, DITE

Functionality

The run-in and run-out path must also be traversed to the required thread with thread G33. The starting and braking of the axis (both axes in case of a tapered thread) are performed in these areas. This path depends on the thread lead, spindle speed, and the axis dynamics (configuration).

If the available path for run-in or run-out is limited, it may be necessary to reduce the spindle speed so that this path is sufficient.

In this case, the run-in and run-out paths can be specified separately in the program to achieve favorable cutting values and

short machining time or to simplify the handling of this issue. If no values are specified, the values from the setting data apply. The specifications in the program are written into SD42010: THREAD_RAMP_DISP[0] ... [1].
 If this path is not sufficient for the traversing with the configured axis acceleration, the axis is overloaded in terms of acceleration. Alarm 22280 ("Programmed run-in path too short") is then issued for the thread run-in. The alarm is purely for information and has no effect on part program execution.
 The run-out path acts as a retracting distance at the end of the thread. This achieves a smooth change in the axis movement when retracting is in progress.

Programming

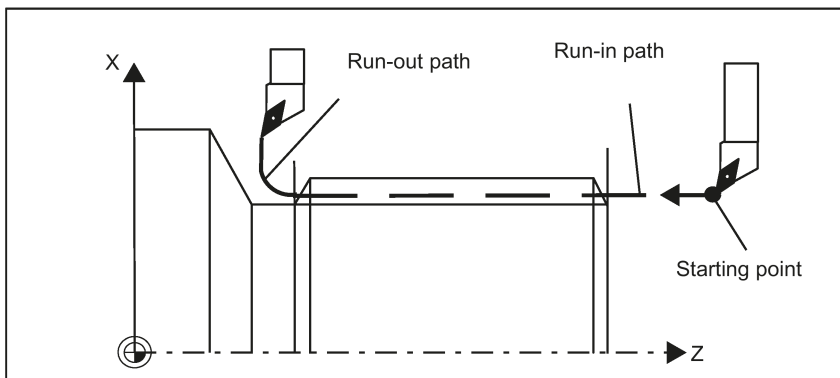
DITS=... ; Run-in path of the thread for G33
 DITE=... ; Run-out path of the thread for G33

Values for DITS and DITE or SD42010: THREAD_RAMP_DISP

- 1 ... < 0: Starting/braking of the feed axis is carried out with the configured acceleration.
Jerk according to current BRISK/SOFT programming.
- 0: Abrupt starting/braking of the feedrate axis on thread cutting.
- > 0: The run-in/run-out path of the thread is predefined for G33.
To avoid alarm 22280, the acceleration limits of the axis must be observed in case of very small run-in and run-out paths.

Note: The value of SD42010 after reset/program start is -1.

See the following illustration for run-in/run-out path for G33:



Programming example

```
N10 G54
N20 G90 G0 Z100 X10 M3 S500
N30 G33 Z50 K5 SF=180 DITS=4 DITE=2 ; run-in 4 mm, run-out 2 mm
N40 G0 X30
N50 G0 X100 Z100
N60 M5
N70 M30
```

11.5.3 Thread cutting with variable lead: G34, G35

Functionality

You can produce threads with variable lead in one block with G34 or G35:

- G34 ; Thread with (linearly) increasing lead
- G35 ; Thread with (linearly) decreasing lead

Both functions otherwise have the same functionality as G33 and have the same prerequisites.

G34 or G35 remain active until canceled by another instruction from this G group (G0, G1, G2, G3, G33...).

Thread lead:

- I or K; Starting thread lead in mm/rev., associated with X or Z axis

Lead change:

In the block with G34 or G35, the address F contains the meaning of the lead change:

The lead (mm per revolution) changes per revolution.

Programming: F ; lead change in mm/rev.².

Determining F

If you already know the starting and final lead of a thread, you can calculate the thread lead change F to be programmed according to the following equation:

$$F = \frac{|K_e^2 - K_a^2|}{2 \times L_G} \left[\text{mm} / U^2 \right]$$

Explanation:

K_e Thread lead of the axis end point coordinate [mm/rev]

K_a Initial thread lead (under I, K progr.) [mm/U]

L_G Thread length in [mm]

Programming

G34 Z... K... F... ;Cylindrical thread with increasing lead

G35 X... I... F... ;Face thread with decreasing lead

G35 Z... X... K... F... ;Taper thread with decreasing lead

Programming example

```
N10 M3 S40 ; Spindle CW
N20 G0 G54 G90 G64 Z10 X60 ; Approach starting point
N30 G33 Z-100 K5 SF=15 ; Thread, constant lead 5mm/rev
; Activation point at 15 degrees
N40 G35 Z-150 K5 F0.16 ; Starting pitch 5 mm/rev
; Pitch decreases 0.16 mm/rev
; Thread length 50 mm
; Desired lead at end of block 3 mm/rev
N50 G0 X80 ; Retraction in X
N60 Z120
N100 M2
```

11.5.4 Thread interpolation: G331, G332

Functionality

A position-controlled spindle with position measuring system is required.

With G331/G332, threads **without** compensating chuck can be drilled, if the dynamic response of the spindle and the axis allow this.

If a compensating chuck is used nevertheless, the position differences to be compensated for by the compensating chuck are reduced. This allows thread grinding at higher spindle speeds.

G331 applies for grinding, G332 for grinding in opposite direction.

The grinding depth is specified through the axis, e.g. Z; the thread lead through the corresponding interpolation parameter (here: K).

For G332, the same lead is programmed as for G331. The spindle direction of rotation is automatically reversed.

The spindle speed is programmed with S; without M3/M4.

Before thread grinding with G331/G332, the spindle must be brought into the closed-loop position controlled mode using SPOS=... .

Right-hand or left-hand thread

The **sign of the thread lead** determines the direction of spindle rotation:

positive: right-hand (as with M3)

Negative: Left-hand (as with M4)

Axis velocity

When cutting threads with G331/G332, you can determine the axis velocity based on the spindle speed and the thread lead. However, the maximum axis velocity (rapid traverse) defined in the machine data cannot be exceeded; otherwise, alarms will appear.

Programming example

Metric thread M5

Lead: 0.8 mm/rev.

Diameter of the bottom hole: 4.2 mm

```
N10 G54 G0 G90 X10 Z5          ; Approach starting point
N20 SPOS=0                     ; Spindle in position control
N30 G331 Z-25 K0.8 S600        ; Thread grinding, K positive = Clockwise rotation of spindle,
                                end point -25 mm
N40 G332 Z5 K0.8               ; Retraction
N50 G0 X10 Z5
N60 M30
```

11.6 Fixed point approach

11.6.1 Fixed point approach: G75

Functionality

By using G75, a fixed point on the machine, e.g. tool change point, can be approached. The position is stored permanently in the machine data for all axes. A maximum of four fixed points can be defined for each axis.

No offset is effective. The velocity of each axis is its rapid traverse.

G75 requires a separate block and acts non-modal. The machine axis identifier must be programmed!

In the part program block after G75, the previous G command of the "Interpolation type" group (G0, G1, G2, ...) is active again.

Programming

G75 FP=<n> X=0 Z=0

Note

FPn is referencing with axis machine data MD30600 \$MA_FIX_POINT_POS[n-1]. If no FP is programmed, then the first fixed point is selected.

Command	Significance
G75	Fixed point approach
FP=<n>	Fixed point that is to be approached. The fixed point number is specified: <n> Value range of <n>: 1, 2, 3, 4 MD30610\$NUM_FIX_POINT_POS should be set if fixed point number 3 or 4 is to be used. If no fixed point number is specified, fixed point 1 is approached automatically.
X=0 Z=0	Machine axes to be traversed to the fixed point. Specify the axes with value "0" with which the fixed point is to be approached simultaneously. Each axis is traversed with the maximum axial velocity.

Programming example

```
N05 G75 FP=1 X=0                ; Approach fixed point 1 in X
N10 G75 FP=2 Z=0                ; Approach fixed point 2 in Z, e.g. for tool
                                change
N30 M30                          ; End of program
```

Note

The programmed position values for X, Z (any value, here = 0) are ignored, but must still be written.

11.6.2 Reference point approach: G74

Functionality

The reference point can be approached in the NC program with G74. The direction and speed information of each axis is stored in the machine data.

G74 needs a separate block and is active based on the block mode. The machine axis identifier must be programmed! In the block after G74, the previous G command of the "Interpolation type" group (G0, G1, G2, ...) is active again.

Programming example

```
| N10 G74 X=0 Z=0
```

Note

The programmed position values for X, Z (here = 0) are ignored, but must still be written.

11.7 Acceleration control and exact stop/continuous path

11.7.1 Exact stop/continuous-path control mode: G9, G60, G64

Functionality

G functions are provided for optimum adaptation to different requirements to set the traversing behavior at the block borders and for block advancing. For example, you would like to quickly position with the axes or to machine contours over multiple blocks.

Programming

G60	; Exact stop, modal
G64	; Continuous-path mode
G9	; Exact stop, non-modal
G601	; Exact stop window fine
G602	; Exact stop window coarse

Exact stop G60, G9

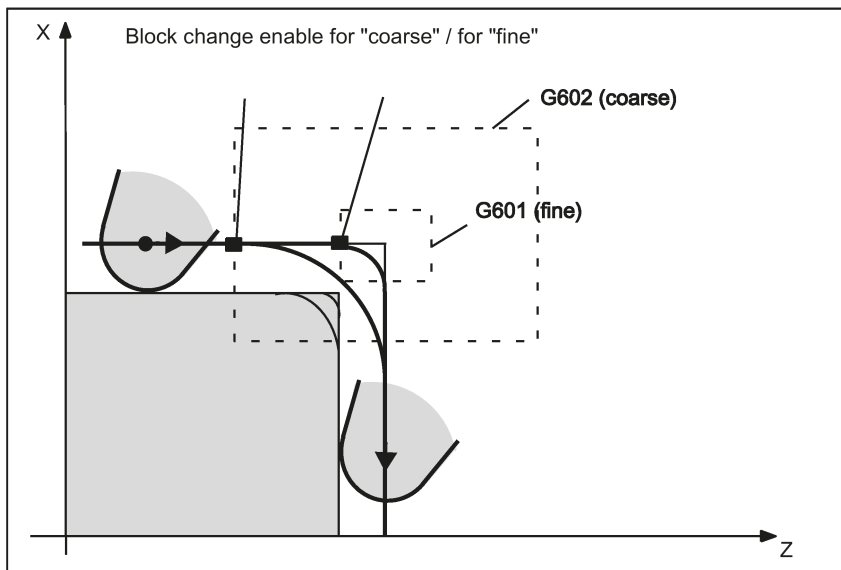
If the exact stop function (G60 or G9) is active, the velocity for reaching the exact end position at the end of a block is decelerated to zero.

Another modal G group can be used here to set when the traversing movement of this block is considered ended and the next block is started.

- G601 Exact stop window fine
Block advance takes place when all axes have reached the "Exact stop window fine" (value in the machine data).
- G602 Exact stop window coarse
Block advance takes place when all axes have reached the "Exact stop window coarse" (value in the machine data).

The selection of the exact stop window has a significant influence on the total time if many positioning operations are executed. Fine adjustments require more time.

See the following illustration for comparison of the G60 and G64 velocity behavior:



Programming example

```

N5 G602 ; Exact stop window coarse
N10 G0 G60 Z10 ; Exact stop modal
N20 X20 Z0 ;G60 continues to act
N30 X30 Z-40
N40 M3 S1000
N50 G1 G601 X35 Z-50 F0.12 ; Exact stop window fine
N60 G64 Z-65 ;Switching over to continuous-path mode
N70 X40 Z-70
N80 G0 G9 Z-80 ; Exact stop acts only in this block
N90 X45 Z-90 ;Again continuous-path mode
N100 M30

```

Remark: The G9 command only generates exact stop for the block in which it is programmed; G60, however, is effective until it is canceled by G64.

Continuous-path control mode G64

The objective of the continuous-path control mode is to avoid deceleration at the block boundaries and to switch **to the next block with a path velocity as constant as possible** (in the case of tangential transitions). The function works with **look-ahead velocity control** over several blocks.

For non-tangential transitions (corners), the velocity can be reduced rapidly enough so that the axes are subject to a relatively high velocity change over a short period of time. This may lead to a significant jerk (acceleration change). The size of the jerk can be limited by activating the SOFT function.

Programming example

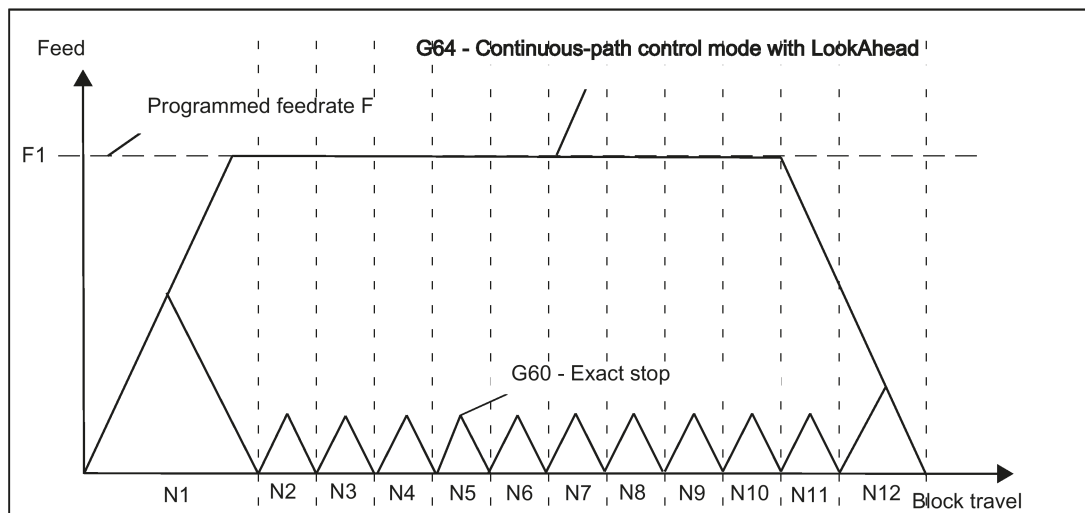
```

N10 G64 G1 Z5 F0.15 M3 S800 ; Continuous-path mode
N20 X20 Z0 ; Continuous-path control mode continues to be active
N30 Z-40
N40 G60 X30 Z-50 ; Switching over to exact stop
N50 X45 Z-70
N60 M30

```

Look-ahead velocity control

In the continuous-path control mode with G64, the control system determines the velocity control for several NC blocks in advance automatically. This enables acceleration and deceleration across multiple blocks with approximately tangential transitions. For paths that consist of short travels in the NC blocks, higher velocities can be achieved than without look ahead.



11.7.2 Acceleration pattern: BRISK, SOFT

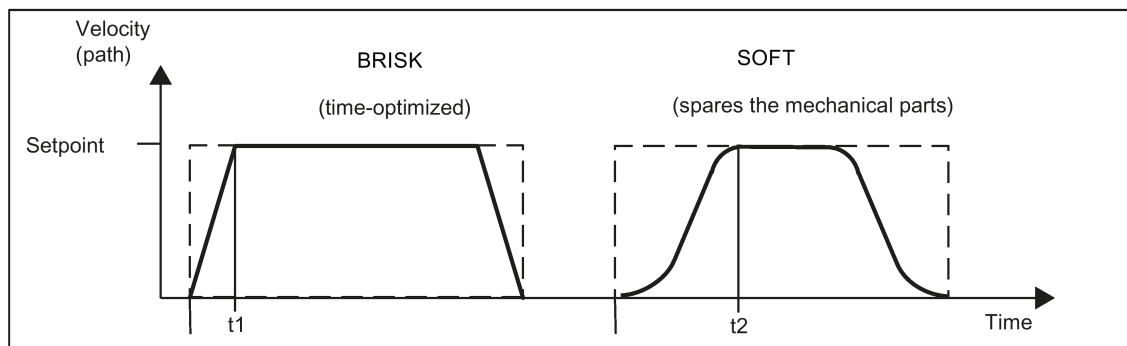
BRISK

The axes of the machine change their velocities using the maximum permissible acceleration value until reaching the final velocity. BRISK allows time-optimized working. The set velocity is reached in a short time. However, jumps are present in the acceleration pattern.

SOFT

The axes of the machine accelerate with nonlinear, constant curves until reaching the final velocity. With this jerk-free acceleration, SOFT allows for reduced machine load. The same behavior can also be applied to braking procedures.

See the following illustration for principle course of the path velocity when using BRISK or SOFT:



Programming

BRISK ; Jerking path acceleration
SOFT ; Jerk-limited path acceleration

Programming example

```
N10 M3 S200
N20 SOFT G1 X30 Z84 F6.5 ; Jerk-limited path acceleration
N30 X46 Z92
N40 BRISK X87 Z104 ; continuing with jerking path acceleration
N50 X95 Z110
N60 M30
```

11.7.3 Dwell Time: G4

Functionality

Between two NC blocks you can interrupt the machining process for a defined period by inserting your **own block** with G4; e.g. for relief cutting.

Words with F... or S... are only used for times in this block. Any previously programmed feedrate F or a spindle speed S remain valid.

Programming

G4 F... ; Dwell time in seconds
G4 S... ; Dwell time in spindle revolutions

Programming example

```
N5 G1 F3.8 Z-50 S300 M3 ;Feed F; spindle speed S
N10 G4 F2.5 ; Dwell time 2.5 seconds
N20 Z70
N30 G4 S30 ;dwelling 30 revolutions of the spindle, corresponds at
; S=300 rpm and 100 % speed override to: t=0.1 min
N40 X20 ;Feed and spindle speed remain effective
N50 M30
```

Remark

G4 S.. is only possible if a controlled spindle is available (if the speed specifications are also programmed via S...).

11.8 The third axis

Prerequisite

The control system itself must be designed for three axes.

Functionality

Depending on the machine design, the third axis can be required. These axes can be implemented as linear or rotary axes. The identifier for these axes is defined by the machine manufacturer (e.g. C).

For rotary axes, the traversing range can be configured between 0 ...<360 degrees (modulo behavior) or -360 degrees/+360 degrees if there is no modulo axis is present.

With an appropriate machine design, a 3rd axis can be traversed linear simultaneously with the remaining axes. If the axis is traversed in a block with G1 or G2/G3 with the remaining axes (X, Z), it does not receive a component of the feedrate F. Its speed conforms to the path time of axes X, Z. Its movement begins and ends with the remaining path axes. However, the speed cannot exceed the defined limit value.

If a block is programmed with this 3rd axis only, the axis will traverse using the active feedrate F when the G1 function is executed. If the axis is a rotary axis, the unit of measurement for F is degrees/min with G94 or degrees/rev. of the spindle with G95.

For these axes, offsets can be set (G54 ... G59) and programmed (TRANS, ATRANS).

Programming example

```
The 3rd axis is a rotary axis with the axis identifier C
N5 G94 ; feedrate F in mm/min or degrees/min
N10 G0 X10 Z30 C45 ; X-Z traverse path with rapid traverse, C at the same time
N20 G1 X12 Z33 C60 F400 ; X-Z traverse path at 400 mm/min, C at the same time
N30 G1 C90 F3000 ; Axis C traverses alone to position 90 degrees at a speed of 3000
degrees/min
```

Special instructions for rotary axes: DC, ACP, ACN

```
For example, for rotary axis A:
A=DC(...) ; Absolute dimensions, approach position directly (on the shortest
path)
A=ACP(...) ; Absolute dimensions, approach position in positive direction
A=ACN(...) ; Absolute dimensions, approach position in negative direction
```

Example:

```
N10 A=ACP(55.7) ; approach absolute position 55.7 degrees in positive direction
```

11.9 Spindle movements

11.9.1 Spindle speed S, directions of rotation

Functionality

The spindle speed is programmed under the address S in revolutions per minute, if the machine has a controlled spindle. The direction of rotation and the beginning or end of the movement are specified via M commands.

Programming

```
M3 ; Spindle clockwise
M4 ; Spindle counter-clockwise
M5 ; Spindle stop
```

Remark: For integer S values, the decimal point can be omitted, e.g. S270.

Information

If you write M3 or M4 in a **block with axis movements**, the M commands become active **before** the axis movements.

Standard setting: Axis movements will only start once the spindle has accelerated to speed (M3, M4). M5 is also issued before the axis movement. However, it does not wait for the spindle to stop. Axis motion already starts before the spindle comes to a standstill.

The spindle is stopped with the end of the program or with the following key:



At the beginning of the program, the spindle speed is zero (S0).

Note: Other settings can be configured via machine data.

Programming example

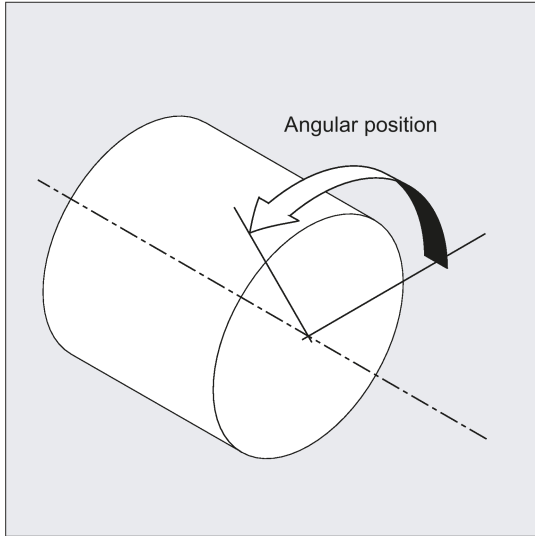
```
N10 G1 X70 Z20 F3 S270 M3 ; before the axis traversing X, Z the spindle accelerates to 270
                           r.p.m., clockwise
N20 X90 Z0
N30 Z-40
N40 M5
N50 M4 S290
N60 G1 X100 Z50
N70 S450 Z100 ; Speed change
N80 X150 Z150
N90 G0 Z180 M5 ; Z movement, spindle comes to a stop
N100 M30
```

11.9.2 Spindle positioning

11.9.2.1 Spindle positioning (SPOS, SPOSA, M19, M70, WAITS)

Functionality

SPOS, SPOSA or M19 can be used to set the spindle to specific angular positions, e.g. during tool change.



SPOS, SPOSA and M19 induce a temporary switchover to position-controlled mode until the next M3/M4/M5/M41 to M45.

Positioning in axis mode

The spindle can also be operated as a path axis, synchronized axis or positioning axis at the address defined in the machine data. When the axis identifier is specified, the spindle is in axis mode. M70 switches the spindle directly to axis mode.

End of positioning

The end-of-motion criterion when positioning the spindle can be programmed using FINEA, CORSEA, or IPOENDA.

The program advances to the next block if the end of motion criteria for the spindle or axes programmed in the current block plus the block change criterion for path interpolation are fulfilled.

Synchronization

In order to synchronize spindle movements, WAITS can be used to wait until the spindle position is reached.

Conditions

The spindle to be positioned must be capable of operation in position-controlled mode.

Programming

Position spindle:

SPOS=<value>

SPOSA=<value>

M19/M<n>=19

Switch spindle over to axis mode:

M70/M<n>=70

Define end-of-motion criterion:

FINEA/FINEA[S<n>]

COARSEA/COARSEA[S<n>]

IPOENDA/IPOENDA[S<n>]

IPOBRKA/IPOBRKA(<axis>[,<instant in time>]); Programming in a separate NC block.

Synchronize spindle movements:

WAITS/WAITS (<n>, <m>) ; Programming in a separate NC block.

Significance

SPOS/SPOSA:

Set spindle to specified angle

SPOS and SPOSA have the same functionality but differ in their block change behavior:

- SPOS delays the enabling of the NC block until the position has been reached.
- SPOSA enables the NC block even if the position has not been reached.

<value>: Angular position to which the spindle is to be set.

Unit: degrees

Type: REAL

The following options are available about programming the position approach mode:

=AC (<value>): Absolute dimensions

Range of values: 0 ... 359,9999

=IC (<value>): Incremental dimensions

Range of values: 0 ... ±99 999,999

=DC (<value>): Approach absolute value directly

=ACN (<value>): Absolute dimension, approach in negative direction

=ACP (<value>): Absolute dimension, approach in positive direction

=<value>: as DC (<value>)

M<n>=19:

Set the spindle (M19 or M0=19) or spindle number <n> (M<n>=19) to the angular position preset with SD43240 \$SA_M19_SPOS with the position approach mode preset in SD43250 \$SA_M19_SPOSMODE.

The NC block is not enabled until the position has been reached.

M<n>=70:

Switch the spindle (M70 or M0=70) or spindle number <n> (M<n>=70) over to axis mode.

No defined position is approached. The NC block is enabled after the switchover has been performed.

FINEA:

Motion end when "Exact stop fine" reached

COARSEA:

Motion end when "Exact stop coarse" reached

IPOENDA:

End of motion on reaching "interpolator stop"

IPOBRKA:

A block change is possible in the braking ramp.

<axis>: Channel axis identifier

<instant in time>: Instant in time of the block change with reference to the braking ramp

Unit: Percent

Range of values: 100 (application point of the braking ramp) to 0 (end of the braking ramp)

If a value is not assigned to the <instant in time> parameter, the current value of the setting data is applied:

SD43600 \$SA_IPOBRAKE_BLOCK_EXCHANGE

Note:

IPOBRKA with an instant in time of "0" is identical to IPOENDA.

WAITS:

Synchronization command for the specified spindle

The subsequent blocks are not processed until the specified spindle programmed in a previous NC block with SPOSA has reached its end position (with exact stop fine).

WAITS after M5: Wait for the spindle to come to a standstill.

WAITS after M3/M4: Wait for the spindle to reach its setpoint speed.

<n>, <m>: Number of the spindle to which the synchronization command is to be applied.

If the spindle number is not specified or if the spindle number is set to "0", WAITS will be applied to the spindle.

Note

Three spindle positions are possible for each NC block.

Note

With incremental dimensions `IC (<value>)`, spindle positioning can take place over several revolutions.

Note

If position control was activated with `SPCON` prior to `SPOS`, this remains active until `SPCOF` is issued.

Note

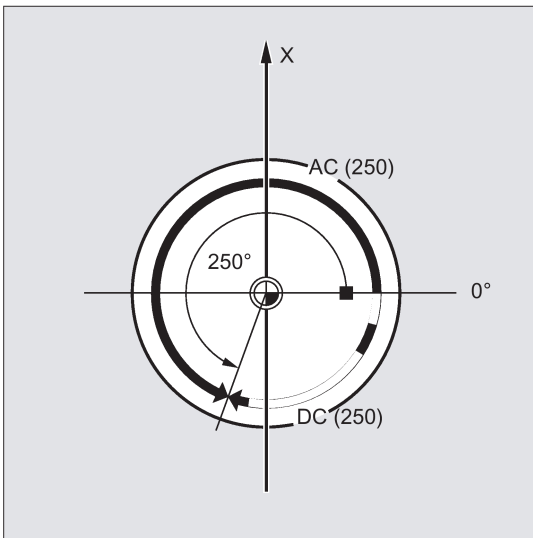
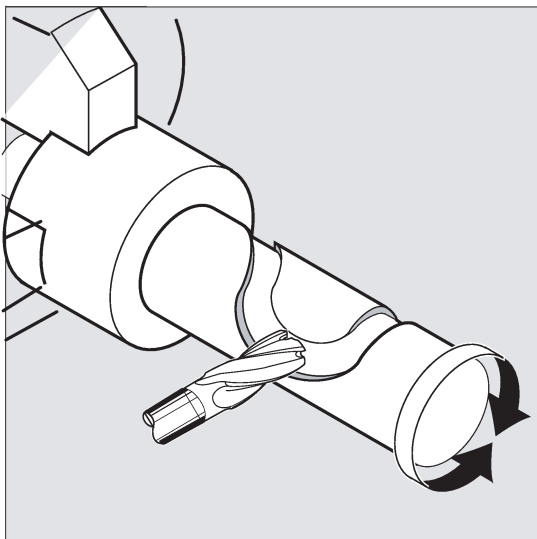
The control system detects the transition to axis mode automatically according to the program sequence. Explicit programming of M70 in the part program is, therefore, essentially no longer necessary. However, M70 can continue to be programmed, e.g. to increase the legibility of the part program.

Programming examples**Example 1: Position spindle with negative direction of rotation**

Spindle 1 is to be positioned at 250° with negative direction of rotation:

```
N10 SPOSA[1]=ACN(250)
```

The spindle is decelerated if necessary and accelerated in the opposite direction to that of the positioning movement.

**Example 2: Spindle positioning in axis mode**

Program variant 1:

```
N10 G0 X100 Z100
N20 M3 S500
N30 G0 X80 Z80
N40 G01 X60 Z60 F0.25
N50 SPOS=0
N60 X50 C180
N70 Z20 SPOS=90
N80 M30
```

Position control on, spindle 1 positioned to 0, axis mode can be used in the next block.

Spindle(C axis) is traversed with linear interpolation synchronous to X.

Spindle is positioned to 90 degrees.

Program variant 2:

```
N10 G0 X100 Z100
N20 M3 S500
N30 G0 X80 Z80
N40 G01 X60 Z60 F0.25
N50 M2=70
N60 X50 C180
N70 Z20 SPOS=90
N80 M30
```

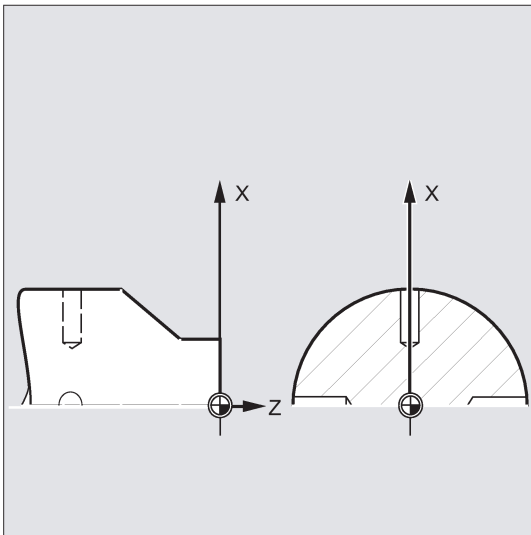
Spindle switches to axis mode.

Spindle(C axis) is traversed with linear interpolation synchronous to X.

Spindle is positioned to 90 degrees.

Example 3: Drill cross holes in turned part

Cross holes are to be drilled in this turned part. The running spindle is stopped at zero degrees and then successively turned through 90°, stopped and so on.



```
G0 X100 Z100
N110 S2=1000 M2=3
N120 SPOSA=DC(0)
N125 G0 X34 Z-35
N130 WAITS
N135 G1 G94 X10 F250
N140 G0 X34
N145 SPOS=IC(90)
N150 G1 X10
N155 G0 X34
N160 SPOS=AC(180)
N165 G1 X10
N170 G0 X34
```

Switch on cross drilling attachment.

Set main spindle to 0° immediately, the program will advance to the next block straight away.

Switch on the drill while the spindle is taking up position.

Wait for the main spindle to reach its position.

Feedrate in mm/min (G96 is suitable only for the multi-edge turning tool and synchronous spindle, but not for power tools on the cross slide.)

The spindle is positioned through 90° with read halt in a positive direction.

The spindle is positioned at 180° relative to the spindle zero point.

```

N175 SPOS=IC(90)
N180 G1 X10
N185 G0 X50
M30

```

The spindle turns in a positive direction through 90° from the absolute 180° position, ending up in the absolute 270° position.

11.9.2.2 Spindle positioning (SPOS, SPOSA, M19, M70, WAITS): Further information

Further information

Positioning with SPOSA

The block step enabling or program execution is not affected by SPOSA. The spindle positioning can be performed during execution of subsequent NC blocks. The program moves onto the next block if all the functions (except for spindle) programmed in the current block have reached their block end criterion. The spindle positioning operation may be programmed over several blocks (see WAITS).

Note

If a command, which implicitly causes a preprocessing stop, is read in a following block, execution of this block is delayed until the positioning spindle is stationary.

Positioning with SPOS/M19

The block step enabling condition is met when all functions programmed in the block reach their end-of-block criterion (e.g. all auxiliary functions acknowledged by the PLC, all axes at their end point) and the spindle reaches the programmed position.

Velocity of the movements:

The velocity and the delay response for positioning are stored in the machine data. The configured values can be modified by programming or by synchronized actions.

Specification of spindle positions:

As the G90/G91 commands are not effective here, the corresponding dimensions apply explicitly, e.g. AC, IC, DC, ACN, ACP. If no specifications are made, traversing automatically takes place as for DC.

Synchronize spindle movements with WAITS

WAITS can be used to identify a point at which the NC program waits until the spindle programmed with SPOSA in a previous NC block reaches its position.

Example:

```

N10 SPOSA[1]=180 SPOSA[1]=0
G01 X34
G00 X10
N40 WAITS(1)                                ;The block waits until the spindle 1 has reached the
                                           position specified in block N10.

```

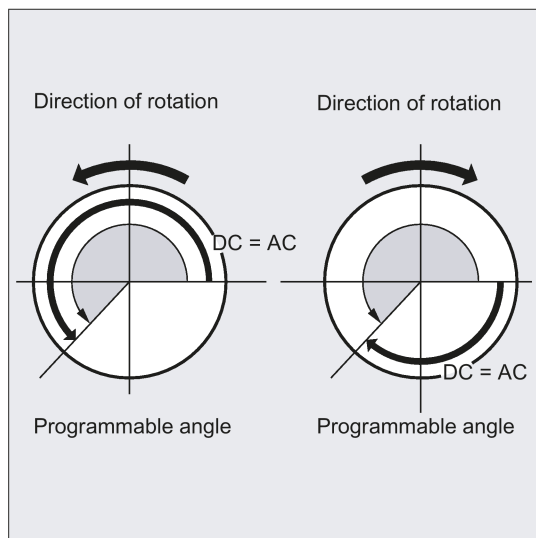
WAITS can be used after M5 to wait until the spindle(s) has (have) stopped. WAITS can be used after M3/M4 to wait until the spindle has reached the specified speed/direction of rotation.

Note

If the spindle has not yet been synchronized with synchronization marks, the positive direction of rotation is taken from the machine data (state on delivery).

Position spindle from rotation (M3/M4)

When M3 or M4 is active, the spindle comes to a standstill at the programmed value.



There is no difference between **DC** and **AC** dimensioning. In both cases, rotation continues in the direction selected by **M3/M4** until the absolute end position is reached. With **ACN** and **ACP**, deceleration takes place if necessary, and the appropriate approach direction is taken. With **IC**, the spindle rotates additionally to the specified value starting at the current spindle position.

Position a spindle from standstill (M5)

The exact programmed distance is traversed from standstill (**M5**).

11.9.3 Gear stages

Functionality

Up to 5 gear stages can be configured for a spindle for speed / torque adaptation.

Programming

The relevant gear stage is selected in the program via **M** commands:

M40 ; Automatic gear stage selection
M41 to M45 ; Gear stages 1 to 5

11.10 Special turning functions

11.10.1 Constant cutting rate: G96, G97

Functionality

Requirement: A controlled spindle must be present.

With activated **G96** function, the spindle speed is adapted to the currently machined workpiece diameter (transverse axis) such that a programmed cutting rate **S** remains constant on the tool edge:
 Spindle speed times diameter = constant.

The **S** word is evaluated as the cutting rate as of the block with **G96**. **G96** is modally effective until cancellation by another **G** function of the group (**G94**, **G95**, **G97**).

Programming

G96 S... **LIMS=...** **F...** ; Constant cutting speed ON
G97 ; Constant cutting speed OFF

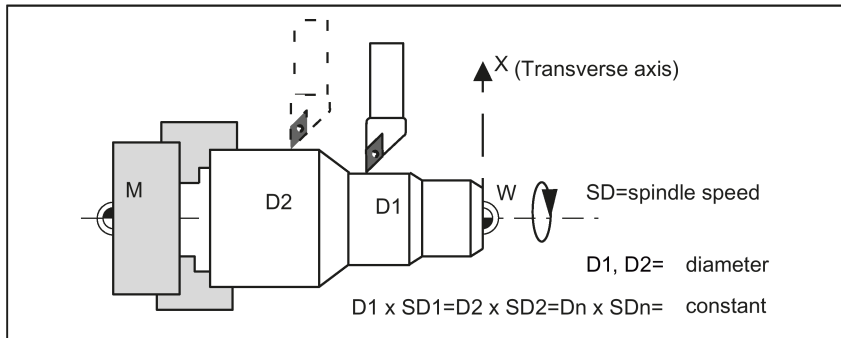
S ; Cutting rate, unit of measurement m/min.
LIMS= ; Upper limit speed of the spindle with **G96**, **G97** effective

F ; Feedrate in mm/revolution - as for G95

Remark:

If G94 instead of G95 was active before, a new appropriate F value must be written.

See the following illustration for constant cutting rate G96:



Rapid traverse

With rapid traverse G0, there is no change in speed.

Exception: If the contour is approached at rapid traverse and the next block contains an interpolation type G1 or G2, G3, CIP, CT (contour block), then the speed for the contour block is applied already in the approach block with G0.

Upper speed limit LIMS=

When machining from large to small diameters, the spindle speed can increase significantly. In this case, it is recommended to program the upper spindle speed limitation LIMS=... . LIMS is only effective with G96 and G97.

By programming LIMS=..., the value entered into the setting data (SD43230: SPIND_MAX_VELO_LIMS) is overwritten. This SD takes effect when LIMS is not written.

The upper limit speed programmed with G26 or defined via machine data cannot be overwritten with LIMS=.

Deactivate constant cutting rate: G97

The function "Constant cutting rate" is deactivated by G97. If G97 is active, a programmed **S word** is given in RPM as the **spindle speed** .

If no new S word is programmed, the spindle turns at the last defined speed with G96 function active.

Programming example

```
N10 M3 S1000 ; Spindle's direction of rotation
N20 G96 S120 LIMS=2500 ; Activate constant cutting speed, 120 m/min, speed limit 2,500
                        r.p.m.
N30 G0 X150 ; no change in speed, because block N31 with G0
N40 X50 Z20 ; no change in speed, because block N32 with G0
N50 X40 ; Approach on contour, new speed is automatically set as is required
        for the beginning of block N40
N60 G1 F0.2 X32 Z25 ; Feedrate 0.2 mm/revolution
N70 X50 Z50
N80 G97 X10 Z20 ; Deactivating constant cutting rate
N90 S600 ; new spindle speed, r.p.m.
N100 M30
```

Information

The G96 function can also be deactivated with G94 or G95 (same G group). In this case, the last **programmed** spindle speed S is active for the remaining machining sequence if no new S word is programmed.

The programmable offset TRANS or ATRANS (see Section "Programmable work offset: TRANS, ATRANS (Page 83)") should not be used on the transverse axis X or used only with low values. The workpiece zero point should be located at the turning center. Only then is the exact function of G96 guaranteed.

11.10.2 Rounding, chamfer

Functionality

You can insert the chamfer (CHF or CHR) or rounding (RND) elements into a contour corner. If you wish to round several contour corners sequentially by the same method, use "Modal rounding" (RNDM).

You can program the feedrate for the chamfer/rounding with FRC (non-modal) or FRCM (modal). If FRC/FRCM is not programmed, the normal feedrate F is applied.

Programming

CHF=... ; Insert chamfer, value: Length of chamfer
CHR=... ; Insert chamfer, value: Side length of the chamfer
RND=... ; Insert rounding, value: Radius of chamfer
RNDM=... ; Modal rounding:
Value >0: Radius of chamfer, modal rounding ON
This rounding is inserted in all contour corners.
Value = 0: Modal rounding OFF...
FRC=... ; Non-modal feedrate for chamfer/rounding
Value >0, feedrate in mm/min (G94) or mm/rev. (G95)
FRCM=... ; Modal feedrate for chamfer/rounding:
Value >0: Feedrate in mm/min (G94) or mm/rev. (G95),
Modal feedrate for chamfer/rounding ON
Value = 0: Modal feedrate for chamfer/rounding OFF
Feedrate F applies to the chamfer/rounding.

Information

The chamfer/rounding functions are executed in the current planes G18 to G19.

The appropriate instruction CHF= ... or CHR=... or RND=... or RNDM=... is written in the block with axis movements leading to the corner.

The programmed value for chamfer and rounding is automatically reduced if the contour length of an involved block is insufficient.

No chamfer/rounding is inserted, if

- more than three blocks in the connection are programmed that do not contain any information for traversing in the plane,
- or a plane change is carried out.

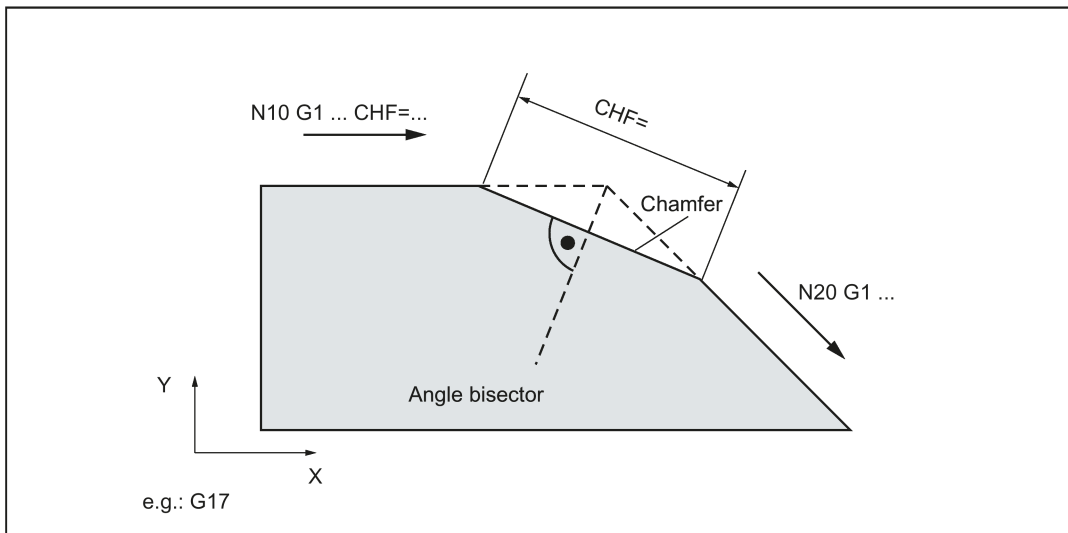
F, FRC, FRCM are not active when a chamfer is traversed with G0.

If the feedrate F is active for chamfer/rounding, it is by default the value from the block which leads away from the corner. Other settings can be configured via machine data.

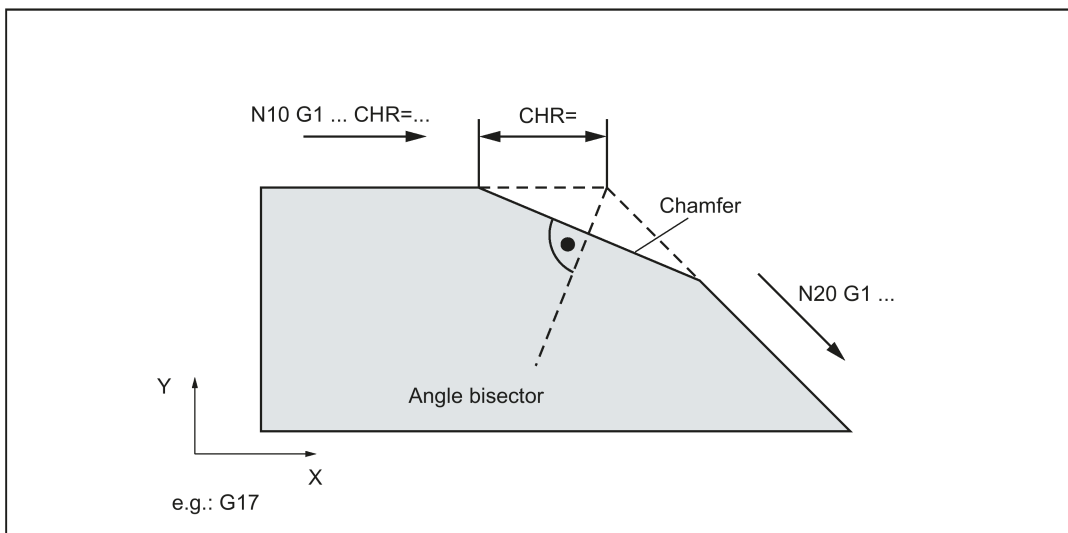
Chamfer CHF or CHR

A linear contour element is inserted between **linear and circle contours** in any combination. The edge is broken.

See the following illustration for inserting a chamfer with CHF using the example: between two straight lines.



See the following illustration for inserting a chamfer with CHR using the example: Between two straight lines.



Programming examples of chamfer

```

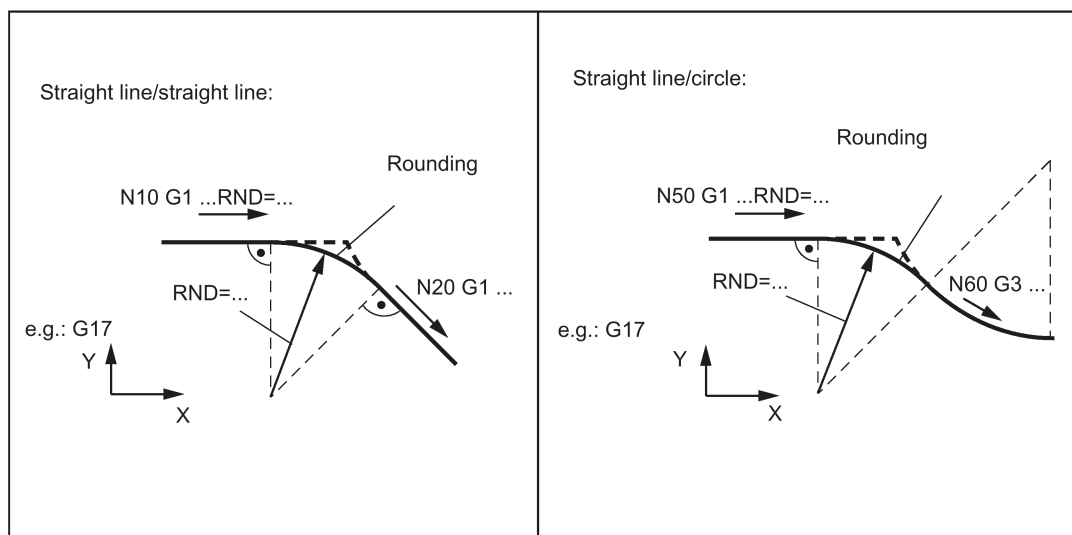
N10 G0 X100 Z100 G94 F100
N20 G1 X80 CHF=5                ; Insert chamfer with chamfer length of 5 mm
N30 X50 Z60
N40 X40 Z50
N50 G1 X30 CHR=7                ; Insert chamfer with leg length of 7 mm
N60 X10 Z20
N70 X0 Z0
N80 G1 FRC=200 X100 CHR=4       ; Insert chamfer with feedrate FRC
N90 X120 Z20
N100 M30

```

Rounding RND or RNDM

A circle contour element can be inserted with tangential connection between the **linear and circle contours** in any combination.

See the following illustration for examples for inserting roundings:



Programming examples for rounding

```

N10 G0 X100 Z100 G94 F100
N20 G1 X80 RND=8                ; Insert 1 rounding with radius 8 mm, feedrate F
N30 X60 Z70
N40 X50 Z50
N50 G1 X40 FRCM= 200 RNDM=7.3   ; Modal rounding, radius 7.3 mm with special feedrate FRCM
                                   (modal)
N60 G1 X20 Z10                  ; continue inserting this rounding - to N70
N70 G1 X0 Z-45 RNDM=0           ; Modal rounding OFF
N80 M30

```

11.10.3 Contour definition programming

Functionality

If direct end point values for the contour are not visible in a machining drawing, angle values can also be used for straight line determination. In a contour corner, you can insert the elements chamfer or rounding. The corresponding instruction **CHR= ...** or **RND=...** is written in the block that leads to the corner.

Contour definition programming can be used in blocks with **G0** or **G1**.

Theoretically, any number of straight-line blocks can be combined and a rounding or chamfer inserted in between. Every straight line must be clearly identified by point values and/or angle values.

Programming

ANG=... ; Specification of angle to define a straight line
RND=... ; Insert rounding, value: Radius of chamfer
CHR=... ; Insert chamfer, value: Side length of the chamfer

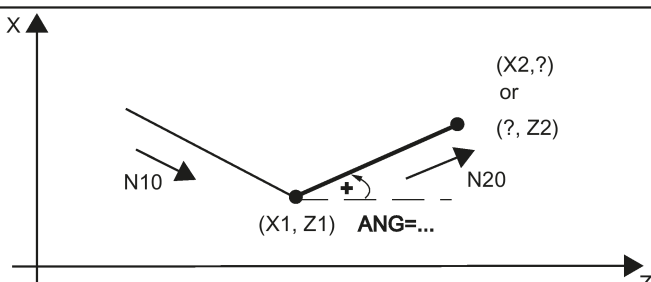
Information

If radius and chamfer are programmed in one block, only the radius is inserted regardless of the programming sequence.

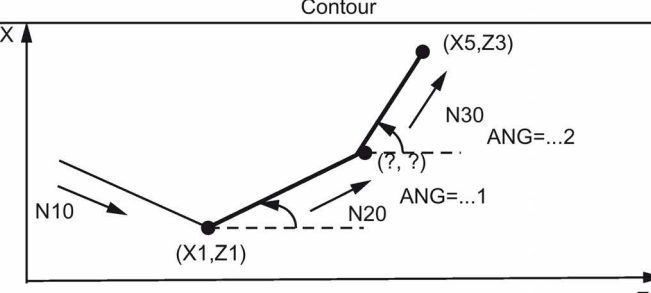
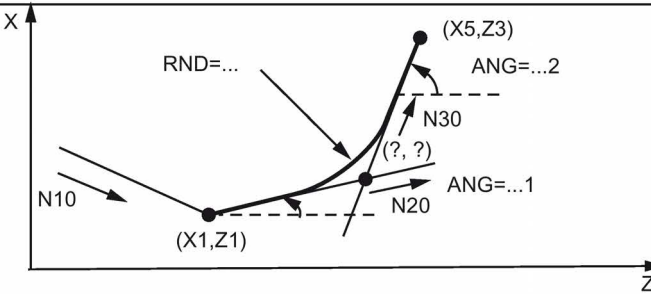
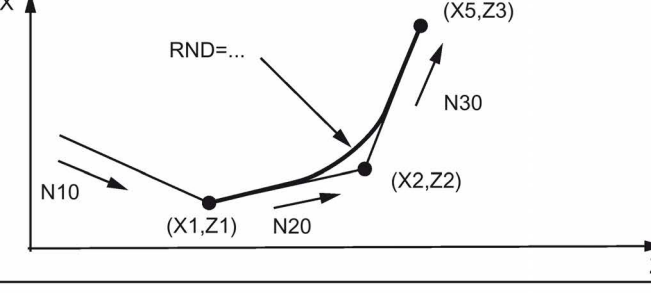
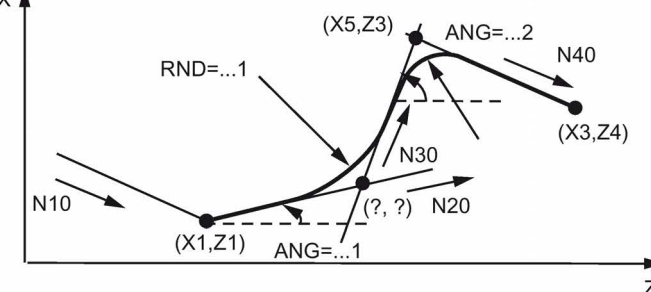
Angle ANG=

An angle can be entered to uniquely define the straight line path if only one end point coordinate of the plane is known for a straight line or for contours across multiple blocks the cumulative end point. The angle is always referred to the Z axis (normal case: G18 active). Positive angles are aligned counter-clockwise.

See the following illustration for angle value for determination of a straight line:

Contour	Programming
	End point in N20 not fully known N10 G1 X1 Z1 N20 X2 ANG=... or: N10 G1 X1 Z1 N20 Z2 ANG=... The values are only symbolic

See the following illustration for examples of multi-block contours:

Contour	Programming
	End point in N20 unknown N10 G1 X1 Z1 N20 ANG=30 N30 X5 Z3 ANG=60 N40 M30
	End point in N20 unknown, insert rounding: N10 G1 X1 Z1 N20 ANG=30 RND=0.1 N30 X5 Z3 ANG=60 analog Inserting a chamfer: N10 G1 X1 Z1 N20 ANG=30 CHR=0.1 N30 X5 Z3 ANG=60
	End point in N20 known Inserting a rounding: N10 G1 X1 Z1 N20 X2 Z2 RND=0.5 N30 X5 Z3 analog Inserting a chamfer: N10 G1 X1 Z1 N20 X2 Z2 CHR=0.2 N30 X5 Z3
	End point in N20 unknown Inserting a rounding: N10 G1 X1 Z1 N20 ANG=30 RND=0.3 N30 X5 Z3 ANG=60 RND=0.3 N40 X3 Z4 analog Inserting a chamfer: N10 G1 X1 Z1 N20 ANG=30 CHR=0.3 N30 X5 Z3 ANG=60 CHR=0.3 N40 X3 Z4 N50 M30

11.11 Tool and tool offset

11.11.1 General information (turning)

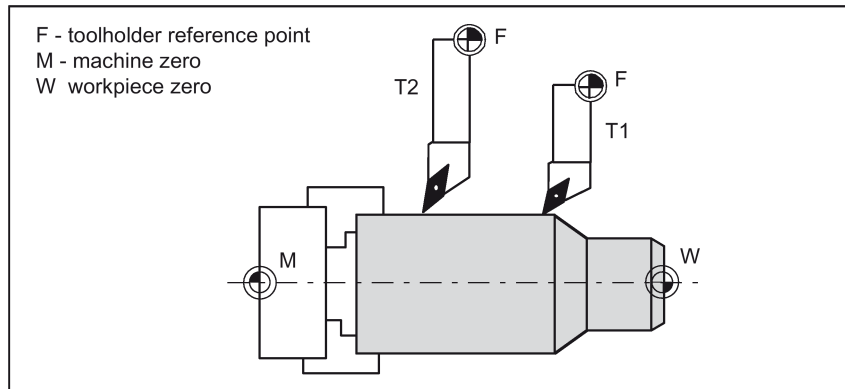
Functionality

During program creation for the workpiece machining, you do not have to take tool lengths or cutting radius into consideration. You program the workpiece dimensions directly, e.g. according to the drawing.

The tool data must be entered separately in a special data area.

In the program, you will merely call the required tool with its offset data. The control system performs the required path compensations based on the data to create the described workpiece.

See the following illustration for machining a workpiece with different tool dimensions:



11.11.2 Tool T (turning)

Functionality

The tool selection takes place when the T word is programmed. Whether this is a **tool change** or only a **preselection** is defined in the machine data:

- A tool change (tool call) takes place directly with the T word (e.g. typical for tool turrets on turning machines)

or

- The change takes place after the preselection with the T word by an additional instruction **M6**.

Note:

If a certain tool was activated, it remains stored as an active tool even beyond the end of the program and after switching off/switching on the control system.

If you change a tool manually, input the change also in the control system so that the control system 'knows' the correct tool. For example, you can start a block with the new T word in MDA mode.

Programming example

Tool change without M6

```
N10 T1
N20 T3
N30 T2
N40 T6
N50 T7
N60 T5
N70 T588
N80 M30
```

The control system can store a maximum of 64 tools.

11.11.3 Tool offset number D (turning)

Functionality

It is possible to assign 1 to 9 data fields with different tool offset blocks (for multiple cutting edges) to a specific tool. If a special cutting edge is required, then it can be programmed using D and the appropriate number.

If a D word is not written, **D1 is automatically effective**.

If **D0** is programmed, the offsets for the tool are **ineffective**.

Programming

D... ; Tool offset number: 1 ... 9, D0: No offsets active!

A maximum of 64 data fields (D numbers) for tool offset blocks can be stored in the control system simultaneously:

T1	D1	D2	D3		D9
T2	D1				
T3	D1				
T6	D1	D2	D3		
T8	D1	D2			

Each tool has its own compensation block - a max. of nine

Information

Tool length compensations become effective **immediately** when the tool is active; when no D number was programmed with the values of D1.

The compensation is retracted with the first programmed traversing of the associated length compensation axis.

A **tool radius compensation** must also be activated by G41/G42.

Programming example

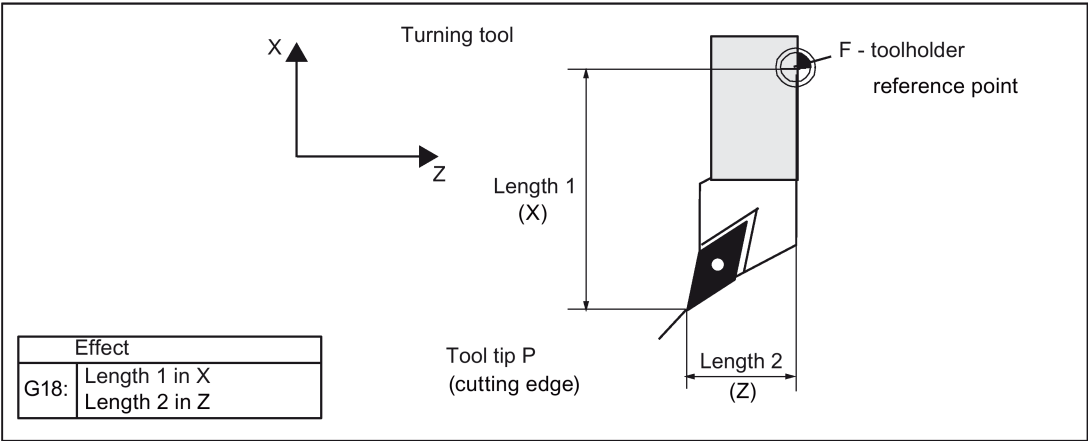
Tool change:

```
N10 T1          ; Tool 1 is activated with the associated D1
N20 G0 X100     ; The length offset compensation is overlaid here
N30 Z100
N40 T4 D2       ; Load tool 4, D2 from T4 is active
N50 X50 Z50
N60 G0 Z62
N70 D1          ; D1 for tool 4 active, only cutting edge changed
N80 M30
```

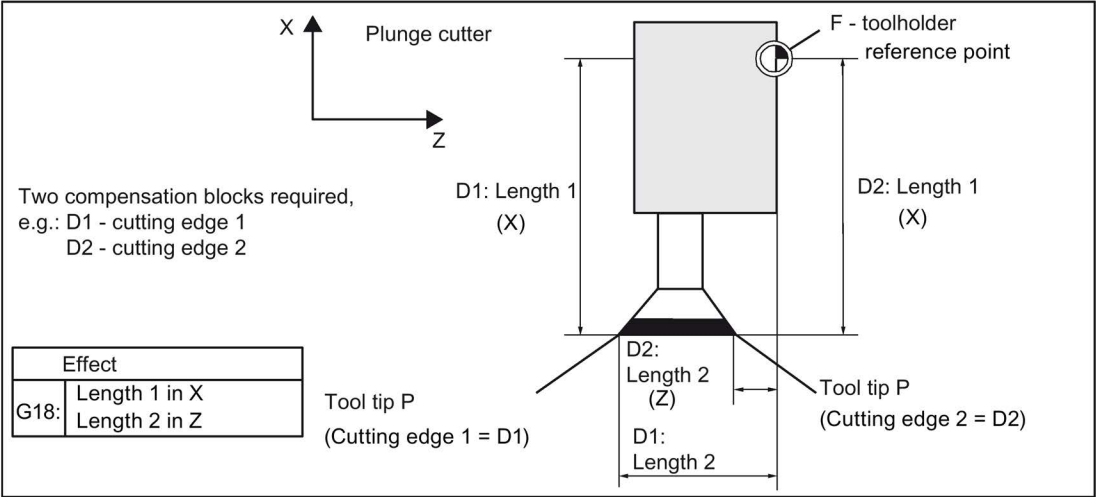
Contents of a compensation memory

- Geometrical dimensions: Length, radius.
They consist of several components (geometry, wear). The control system takes into account the components to obtain a resulting dimension (e.g. overall length 1, total radius). The respective overall dimension becomes active when the offset memory is activated.
The way in which these values are computed in the axes is determined by the tool type and the current plane G17, G18, G19.
- Tool type
The tool type (drill or turning tool) determines which geometry data are required and how they will be calculated.
- Cutting edge position
For the "turning tool" tool type, you must also enter the cutting edge position.

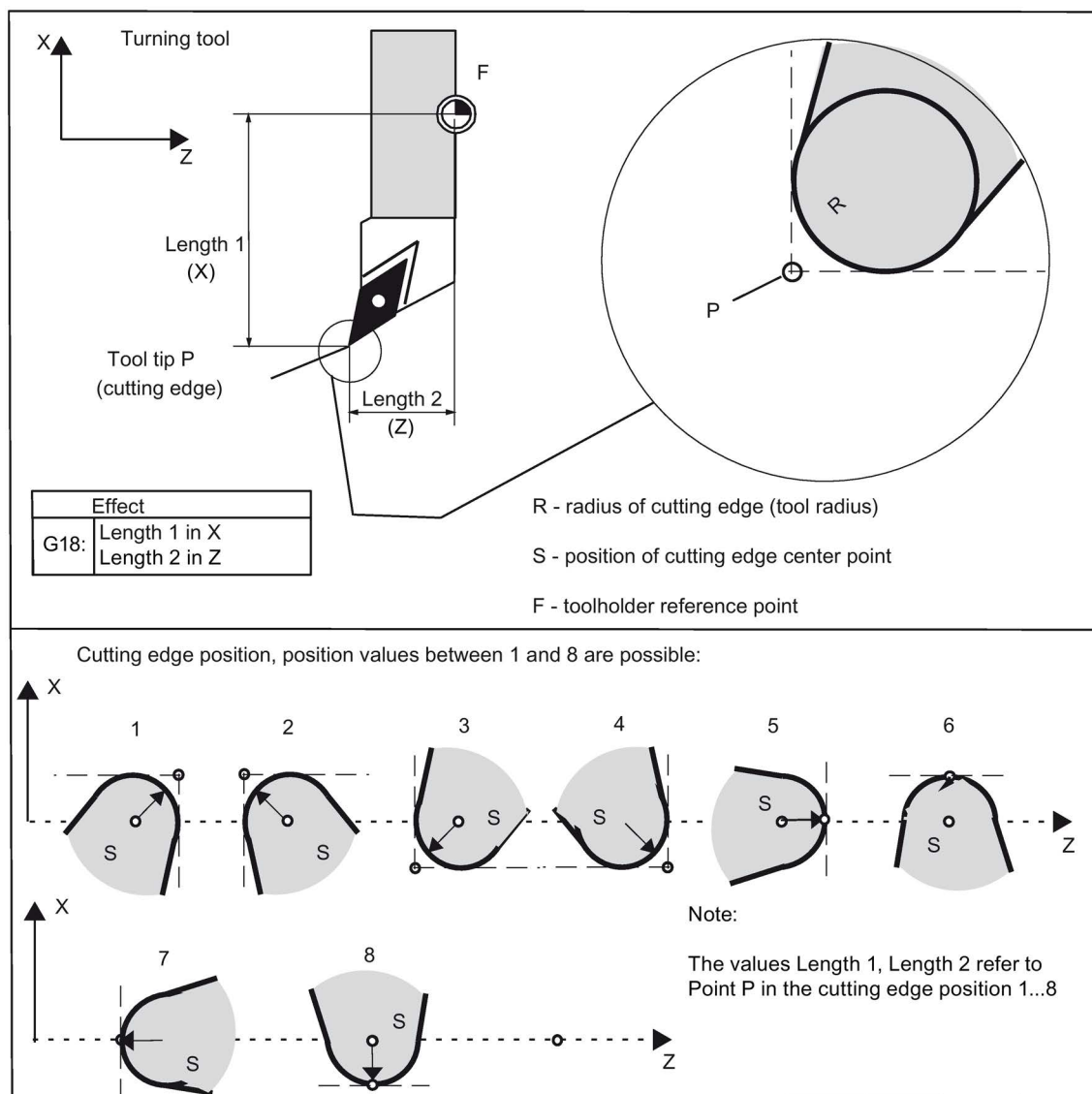
The following figures provide information on the required tool parameters for the respective tool type.



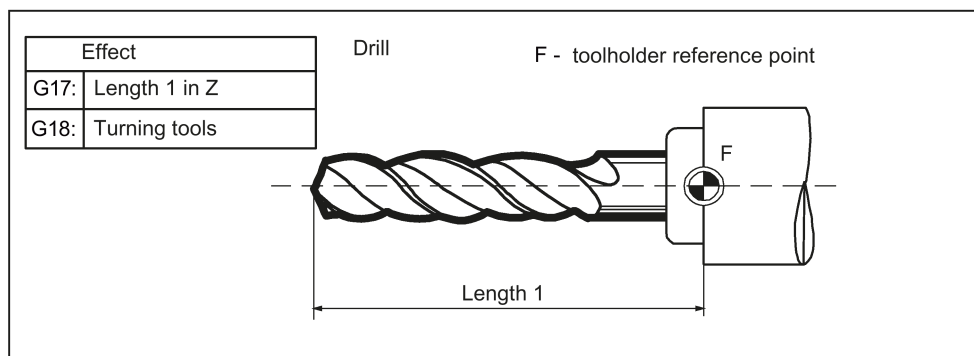
See the following illustration for turning tool with two cutting edges D1 and D2_length compensation:



See the following illustration for compensations for turning tool with tool radius compensation:



See the following illustration for effect of the compensation for the drill:

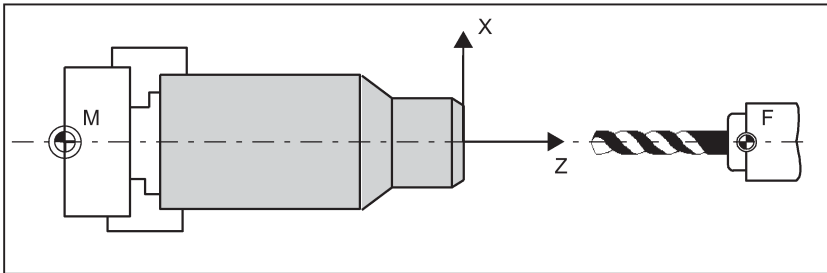


Center hole

Switch to G17 for application of a center hole. This makes the length compensation take effect for the drill in the Z axis. After drilling, the normal compensation for turning tools takes effect again with G18.

Programming example

```
N10 T3 D1 ;Drill
N20 G17 G1 F1 Z0 M3 S100 ; Tool length offset effective in Z axis
N30 Z-15
N40 G18 M30 ; Drilling terminated
```



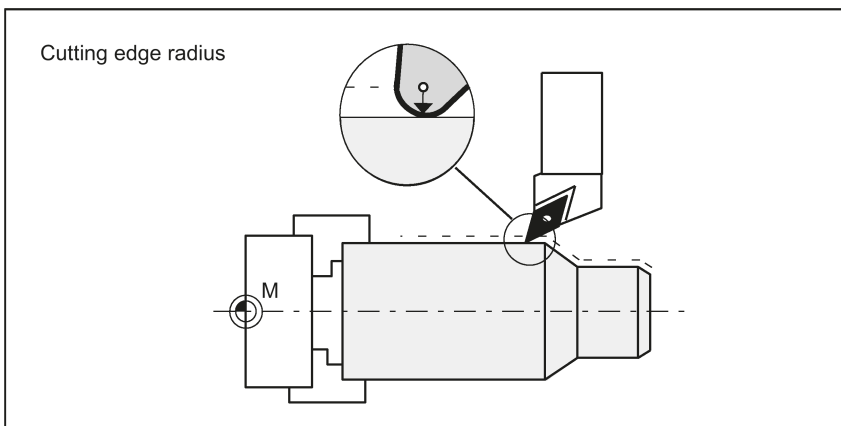
11.11.4 Selecting the tool radius compensation: G41, G42

Functionality

A tool with a corresponding D number must be active. The tool radius offset (cutting edge radius offset) is activated by G41/G42. The controller automatically calculates the required equidistant tool paths for the programmed contour for the respective current tool radius.

G18 must be active.

See the following illustration for tool radius compensation (cutter radius compensation):



Programming

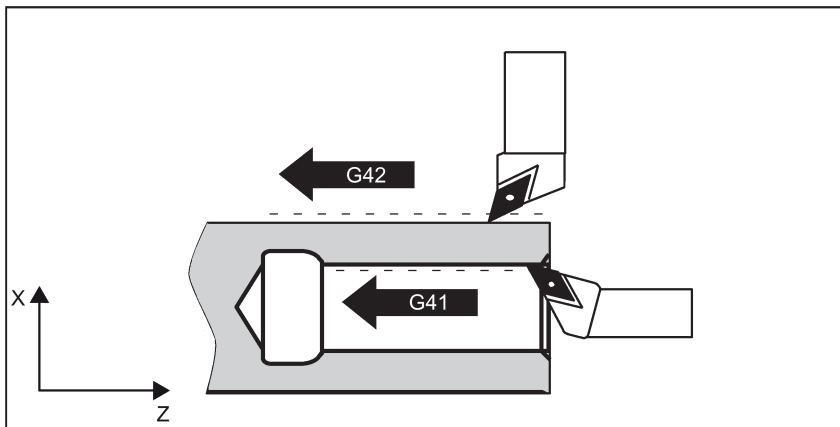
G41 X... Z... ; Tool radius compensation left of contour

G42 X... Z... ; Tool radius compensation right of contour

Remark: The selection can only be made for linear interpolation (G0, G1).

Program both axes. If you only specify one axis, the second axis is automatically completed with the last programmed value.

See the following illustration for compensation to the right/left of the contour:

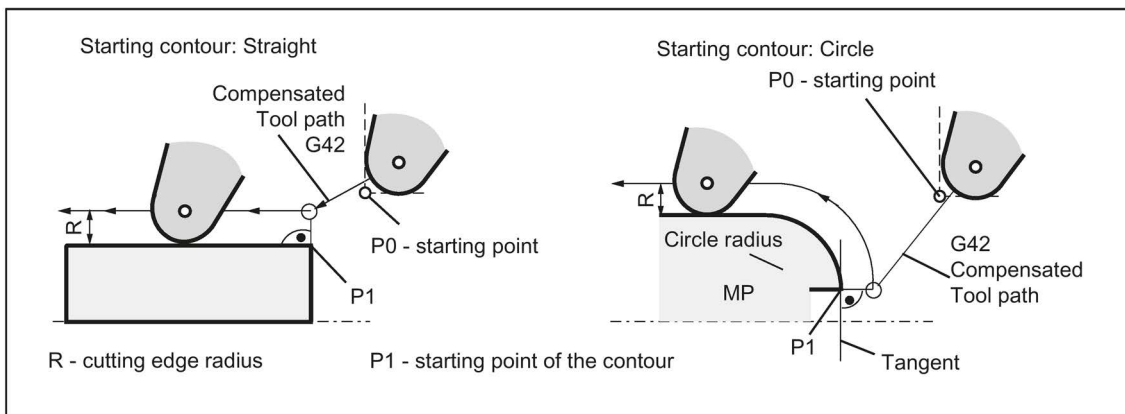


Starting the compensation

The tool approaches the contour on a straight line and positions itself vertically to the path tangent in the starting point of the contour.

Select the start point so as to ensure collision-free traversing.

See the following illustration for start of the tool radius compensation with the example G42:



The tool tip goes around the left of the workpiece when the tool runs clockwise using G41; The tool tip goes around the right of the workpiece when the tool runs counter-clockwise using G42.

Information

As a rule, the block with G41/G42 is followed by a block with workpiece contour description. If, however, the block with G41/G42 is followed by blocks without contour description, a maximum of five such blocks (for example, M commands) are allowed; otherwise, the compensation will be interrupted.

Programming example

```

N10 T4 D1
N20 G18           ; Working plan
N30 G0 X60 Z50    ; P0 - starting point
N40 G0 G42 X50 Z0  ; Selecting right of contour, P1
N50 S1000 M3       ; M command
N60 M08           ; M command
N70 G1 Z-30 F0.15  ; Starting contour
N80 X60 Z10
N90 M30

```

11.11.5 Corner behavior: G450, G451

Functionality

Using the functions G450 and G451, you can set the behavior for non-continuous transition from one contour element to another contour element (corner behavior) when G41/G42 is active.

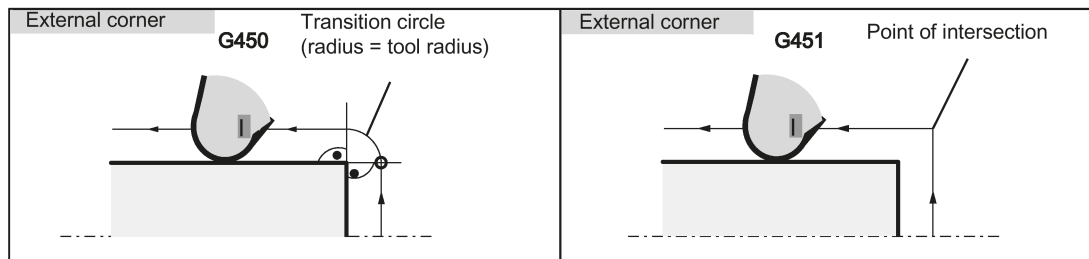
Internal and external corners are detected automatically by the control system. For internal corners, the intersection of the equidistant paths is always approached.

Programming

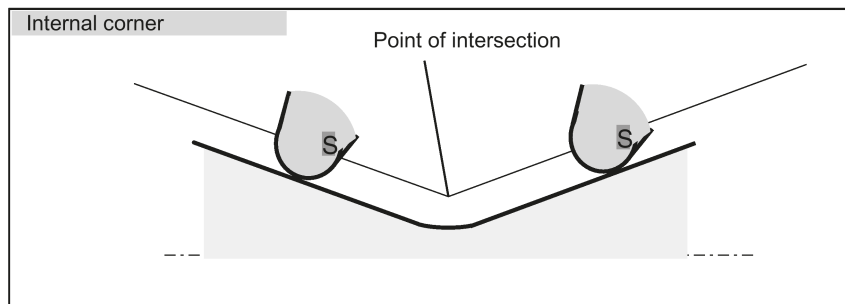
G450 ;Transition circle

G451 ;Point of intersection

See the following illustration for corner behavior at an external corner:



See the following illustration for corner behavior at an internal corner:



Transition circle G450

The tool center point travels around the workpiece external corner in an arc with the tool radius. In view of the data, for example, as far as the feedrate value is concerned, the transition circle belongs to the next block containing traversing movements.

Point of intersection G451

For a G451 intersection of the equidistant paths, the point (intersection) that results from the center point paths of the tool (circle or straight line) is approached.

11.11.6 Tool radius compensation OFF: G40

Functionality

The compensation mode (G41/G42) is deselected with G40. G40 is also the switch-on position at the beginning of the program.

The tool ends the **block before G40** in the normal end position (compensation vector vertical to the tangent in the end point); independently of the start angle.

If G40 is active, the reference point is the tool tip. The tool tip then travels to the programmed point upon deselection.

Always select the end point of the G40 block such that collision-free traversing is guaranteed!

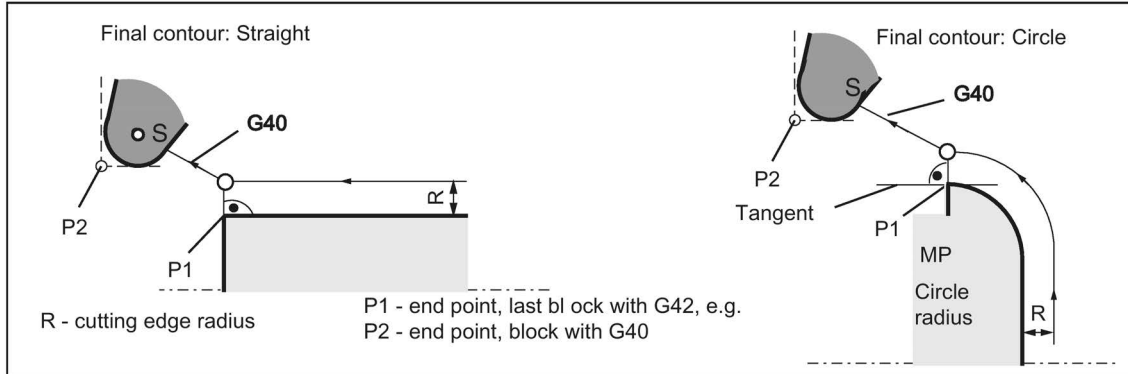
Programming

G40 X... Z... ; Tool radius compensation OFF

Remark: The compensation mode can only be deselected with linear interpolation (G0, G1).

Program both axes. If you only specify one axis, the second axis is automatically completed with the last programmed value.

See the following illustration for ending the tool radius compensation with G40:



Programming example

```
N10 T4 D1 M3 S1000 F0.1
N20 G0 X50 Z50
N30 G1 G42 X30 Z40
N40 G2 X20 Z20 R15
N50 G1 X10 Z10
N60 G40 G1 X0 Z0 ;Last block on the contour, circle or straight line, P1
N70 M30 ;Switch off tool radius compensation,P2
```

11.11.7 Special cases of the tool radius compensation

Change of the compensation direction

The G41 ↔ G42 compensation direction can be changed without writing G40 in between.

The last block that uses the old compensation direction will end at the normal end position of the compensation vector in the end point. The new compensation direction is executed as a compensation start (default setting at starting point).

Repetition of G41, G41 or G42, G42

The same compensation can again be programmed without writing G40 in between.

The last block before the new compensation call will end at the normal position of the compensation vector in the end point. The new compensation is carried out as a compensation start (behavior as described for change in compensation direction).

Changing the offset number D

The offset number D can be changed in the compensation mode. A modified tool radius is active with effect from the block in which the new D number is programmed. Its complete modification is only achieved at the end of the block. In other words: The modification is traversed continuously over the entire block, also for circular interpolation.

Cancellation of compensation by M2

If the offset mode is canceled with M2 (program end) without writing the command G40, the last block with coordinates ends in the normal offset vector setting. **No** compensating movement is executed. The program ends with this tool position.

Critical machining cases

When programming, pay special attention to cases where the contour path for inner corners is smaller than the tool radius; and smaller than the diameter for two successive inner corners.

Such cases should be avoided.

Also check over multiple blocks that the contour contains no "bottlenecks".

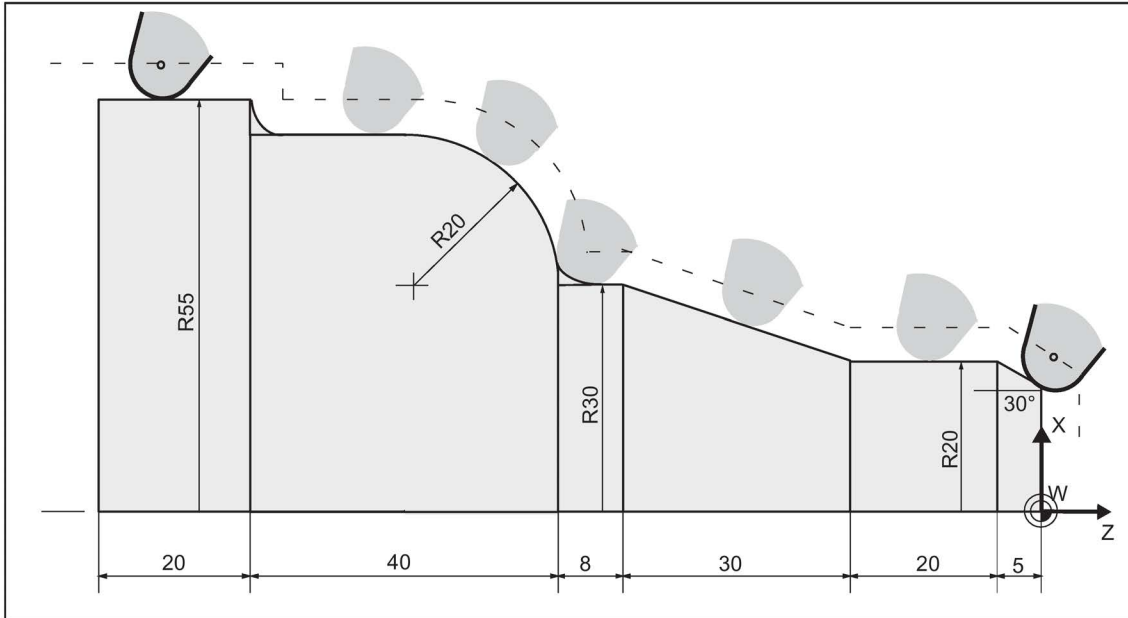
When carrying out a test/dry run, use the largest tool radius you are offered.

Acute contour angles

If very sharp outside corners occur in the contour with active G451 intersection, the control system automatically switches to transition circle. This avoids long idle motions.

11.11.8 Example of tool radius compensation (turning)

See the following example of tool radius compensation, cutting edge radius shown magnified:



Programming example

```

N1                                     ; Contour cut
N2 T1                                 ; Tool 1 with offset D1
N10 DIAMOF F0.15 S1000 M3             ; Radius dimension, technological values
N15 G54 G0 G90 X100 Z15
N20 X0 Z6
N30 G1 G42 G451 X0 Z0                 ; Start compensation mode
N40 G91 X20 CHF=(5* 1.1223 )          ; Insert chamfer, 30 degrees
N50 Z-25
N60 X10 Z-30
N70 Z-8
N80 G3 X20 Z-20 CR=20
N90 G1 Z-20
N95 X5
N100 Z-25
N110 G40 G0 G90 X100                 ; Terminate compensation mode
N120 M2

```

11.11.9 Special handling of tool compensation (turning)

Influence of setting data

With the use of the following setting data, the operator/programmer can influence the calculation of the **length compensation** of the tool used:

- SD 42940: TOOL_LENGTH_CONST
(Assignment of tool-length components to geometry axes)
- SD 42950: TOOL_LENGTH_TYPE
(Assignment of the tool-length components independent of tool type)

Note

The modified setting data will become effective with the next cutting edge selection.

Examples

With SD 42950: TOOL_LENGTH_TYPE =2

a milling tool used is taken into account in length compensation as a turning tool:

- G17: Length 1 in Y axis, length 2 in X axis
- G18: Length 1 in X axis, length 2 in Z axis
- G19: Length 1 in Z axis, length 2 in Y axis

With SD 42940: TOOL_LENGTH_CONST =18

the length assignment is performed in all planes G17 to G19 as for G18:

- Length 1 in X axis, length 2 in Z axis

Setting data in the program

In addition to setting of setting data via operator input, these can also be written in the program.

Programming example

```
N10 $MC_TOOL_LENGTH_TYPE=2  
N20 $MC_TOOL_LENGTH_CONST=18
```

11.12 Miscellaneous function M

Functionality

The miscellaneous function M initiates switching operations, such as "Coolant ON/OFF" and other functions.

A small part of M functions have already been assigned a fixed functionality by the manufacturer of the control system. The functions not yet assigned fixed functions are reserved for free use of the machine manufacturer.

Programming

M... ;Max. 5 M functions per block

Effect**Activation in blocks with axis movements:**

If the functions **M0**, **M1**, **M2** are contained in a block with traversing movements of the axes, these M functions become effective **after the traversing movements**.

The functions M3, M4, M5 are output to the internal interface (PLC) before the traversing movements. The axis movements only begin once the controlled spindle has ramped up for M3, M4. For M5, however, the spindle standstill is not waited for. The axis movements already begin before the spindle stops (default setting).

The remaining M functions are output to the PLC with the traversing movements.

If you would like to program an M function directly before or after an axis movement, insert a separate block with this M function.

Note

The M function interrupts the G64 continuous path mode and generates exact stop.

Programming example

```
N10 S1000  
N20 G1 X50 F0.1 M3 ;M function in the block with axis movement, spindle acceler-  
ates before the X axis movement  
N180 M78 M67 M10 M12 M37 ;Max. 5 M functions in the block  
M30
```

Note

In addition to the M and H functions, T, D, and S functions can also be transferred to the PLC (Programmable Logic Controller). In all, a maximum of 10 such function outputs are possible in a block.

11.13 H function

Functionality

With H functions, floating point data (REAL data type - as with arithmetic parameters, see Section "Arithmetic parameter R (Page 126)") can be transferred from the program to the PLC.

The meaning of the values for a given H function is defined by the machine manufacturer.

Programming

H0=... to H9999=... ;Max. 3 H functions per block

Programming example

```
N10 H1=1.987 H2=978.123 H3=4 ;3 H functions in block
N20 G0 X71.3 H99=-8978.234 ;With axis movements in block
N30 H5 ;Corresponds to H0=5.0
```

Note

In addition to the M and H functions, T, D and S functions can also be transferred to PLC (Programmable Logic Controller). In all, a maximum of 10 function outputs of this type are possible in a part program block.

11.14 Arithmetic parameters, LUD and PLC variables

11.14.1 Arithmetic parameter R

Functionality

The arithmetic parameters are used if an NC program is not only to be valid for values assigned once, or if you must calculate values. The required values can be calculated or set by the control system during program execution.

Another possibility consists of setting the arithmetic parameter values by operator inputs. If values have been assigned to the arithmetic parameters, they can be assigned to other variable-setting NC addresses in the program.

Programming

R0=... to R299=... ;Assign values to the arithmetic parameters
R[R0]=... ;Indirect programming: Assign a value to the arithmetic parameter R, whose number can be found, e.g. in R0
X=R0 ;Assign arithmetic parameters to the NC addresses, e.g. for the X axis

Value assignments

You can assign values in the following range to the R parameters:

±(0.000 0001 ... 9999 9999)
(8 decimal places, arithmetic sign, and decimal point)

The decimal point can be omitted for integer values. A plus sign can always be omitted.

Example:

R0=3.5678 R1=-37.3 R2=2 R3=-7 R4=-45678.123

Use the **exponential notation** to assign an extended range of numbers:

± (10⁻³⁰⁰ ... 10⁺³⁰⁰)

The value of the exponent is written after the **EX** characters; maximum total number of characters: 10 (including leading signs and decimal point)

Range of values for EX: -300 to +300

Example:

R0=-0.1EX-5	;Meaning: R0 = -0.000 001
R1=1.874EX8	;Meaning: R1 = 187 400 000

Note

There can be several assignments in one block incl. assignments of arithmetic expressions.

Assignments to other addresses

The flexibility of an NC program lies in assigning these arithmetic parameters or expressions with arithmetic parameters to other NC addresses. Values, arithmetic expressions and arithmetic parameters can be assigned to all addresses; **Exception: addresses N, G, and L.**

When assigning, write the "=" sign after the address character. It is also possible to have an assignment with a minus sign.

A separate block is required for assignments to axis addresses (traversing instructions).

Example:

N10 G0 X=R2	;Assignment to X axis
-------------	-----------------------

Arithmetic operations/arithmetic functions

When operators/arithmetic functions are used, it is imperative to use conventional mathematical notation. Machining priorities are set with round brackets. Otherwise, multiplication and division take precedence over addition and subtraction.

Degrees are used for the trigonometric functions.

Programming example: Calculating with R parameters

N10 R1= R1+1	;The new R1 is calculated from the old R1 plus 1
N20 R1=R2+R3 R4=R5-R6 R7=R8*R9 R10=R11/R12	
N30 R13=SIN(25.3)	;R13 equals sine of 25.3 degrees
N40 R14=R1*R2+R3	;Multiplication and division take precedence over addition or subtraction R14=(R1*R2)+R3
N50 R14=R3+R2*R1	;Result, the same as block N40
N60 R15=SQRT(R1*R1+R2*R2)	;Meaning:
N70 R1= -R1	;The new R1 is the negative old R1

Programming example: Assign R parameters to the axes

N10 G1 G91 G94 X=R1 Z=R2 F300	;Separate blocks (traversing blocks)
N20 Z=R3	
N30 X=-R4	
N40 Z= SIN(25.3)-R5	;With arithmetic operations
M30	

Programming example: Indirect programming

N10 R1=5	;Assigning R1 directly value 5 (integer)
R2=6	
R1=R2-1	
N100 R[R1]=27.123	;Indirectly assign R5 the value 27.123
M30	

11.14.2 Local User Data (LUD)

Functionality

The operator/programmer (user) can define his/her own variable in the program from various data types (LUD = Local User Data). These variables are only available in the program in which they were defined. The definition takes place immediately at the start of the program and can also be associated with a value assignment at the same time. Otherwise the starting value is zero.

The name of a variable can be defined by the programmer. The naming is subject to the following rules:

- A maximum of 31 characters can be used.
- It is imperative to use letters for the first two characters; the remaining characters can be either letters, underscore or digits.
- Do not use a name already used in the control system (NC addresses, keywords, names of programs, subroutines, etc.).

Programming / data types

DEF BOOL varname1	;Boolean type, values: TRUE (=1), FALSE (=0)
DEF CHAR varname2	;Char type, 1 ASCII code character: "a", "b", ... ;Numerical code value: 0 ... 255
DEF INT varname3	;Integer type, integer values, 32 bit value range: ;-2 147 483 648 through +2 147 483 647 (decimal)
DEF REAL varname4	;Real type, natural number (like arithmetic parameter R), ;Value range: $\pm(0.000\ 0001 \dots 9999\ 9999)$;(8 decimal places, arithmetic sign and decimal point) or ;Exponential notation: $\pm$ (10 to power of -300 ... 10 to power of +300)
DEF STRING[string length] varname41	; STRING type, [string length]: Maximum number of characters

Each data type requires its own program line. However, several variables of the same type can be defined in one line.

Example:

```
| DEF INT PVAR1, PVAR2, PVAR3=12, PVAR4           ;4 type INT variables
```

Example for STRING type with assignment:

```
| DEF STRING[12] PVAR="Hello"                     ; Define variable PVAR with a maximum of 12 characters and assign string "Hello"
```

Fields

In addition to the individual variables, one or two-dimensional fields of variables of these data types can also be defined:

DEF INT PVAR5[n]	;One-dimensional field, type INT, n: integer
DEF INT PVAR6[n,m]	;Two-dimensional field, type INT, n, m: integer

Example:

```
| DEF INT PVAR7[3]                               ;Field with 3 elements of the type INT
```

Within the program, the individual field elements can be reached via the field index and can be treated like individual variables. The field index runs from 0 to a small number of the elements.

Example:

```
| N10 PVAR7[2]=24                                ;The third field element (with index 2) is assigned the value 24.
```

Value assignment for field with SET instruction:

```
| N20 PVAR5[2]=SET(1,2,3)                        ;After the 3rd field element, different values are assigned.
```

Value assignment for field with REP instruction:

```
| N20 PVAR7[4]=REP(2)                            ;After field element [4] - all are assigned the same value, here 2.
```

11.14.3 Reading and writing PLC variables

Functionality

To allow rapid data exchange between NC and PLC, a special data area exists in the PLC user interface with a length of 512 bytes. In this area, PLC data are compatible in data type and position offset. In the NC program, these compatible PLC variables can be read or written.

To this end, special system variables are provided:

\$A_DBB[n]	;Data byte (8-bit value)
\$A_DBW[n]	;Data word (16-bit value)
\$A_DBD[n]	;Data double-word (32-bit value)
\$A_DBR[n]	;REAL data (32-bit value)

"n" stands here for the position offset (start of data area to start of variable) in bytes

Programming example

```
| R1=$A_DBR[4] ;Reading a REAL value, offset 4 (starts at byte 4 of range)
```

Note

The reading of variables generates a preprocessing stop (internal STOPRE).

Note

Writing of PLC tags is generally limited to a maximum of three tags (elements).

Where PLC tags are to be written in rapid succession, one element will be required per write operation.

If more write operations are to be executed than there are elements available, then block transfer will be required (a preprocessing stop may need to be triggered).

Example:

```
$A_DBB[1]=1 $A_DBB[2]=2 $A_DBB[3]=3  
STOPRE  
$A_DBB[4]=4
```

11.15 Program jumps

11.15.1 Unconditional program jumps

Functionality

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.

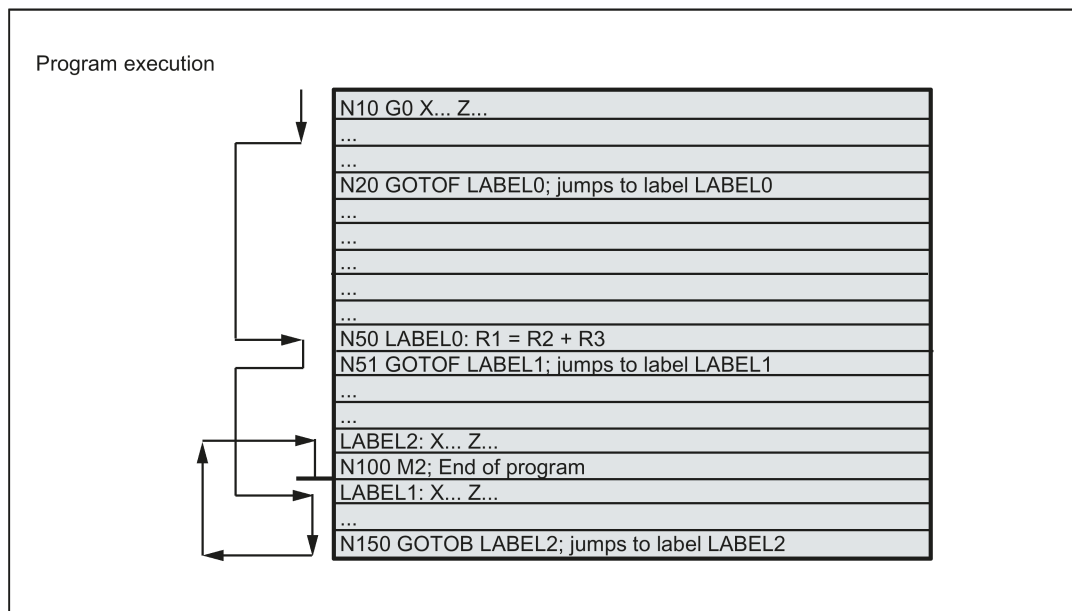
The jump destination can be a block with a **label** or with a **block number**. This block must be located within the program.

The unconditional jump instruction requires a separate block.

Programming

GOTOF label	;Jump forward (in the direction of the last block of the program)
GOTOB label	;Jump backwards (in the direction of the first block of the program)
Label	;Selected string for the label (jump label) or block number

See the following illustration for an example of unconditional jumps:



11.15.2 Conditional program jumps

Functionality

Jump conditions are formulated after the **IF instruction**. If the jump condition (**value not zero**) is satisfied, the jump takes place.

The jump destination can be a block with a **label** or with a **block number**. This block must be located within the program.

Conditional jump instructions require a separate block. Several conditional jump instructions can be located in the same block.

By using conditional program jumps, you can also considerably shorten the program, if necessary.

Programming

IF condition GOTOF label	;Jump forward
IF condition GOTOB label	;Jump backwards
GOTOF	;Jump direction forward (in the direction of the last block of the program)
GOTOB	;Jump direction backwards (in the direction of the first block of the program)
Label	;Selected string for the label (jump label) or block number
IF	;Introduction of the jump condition
Condition	;Arithmetic parameter, arithmetic expression for formulating the condition

Comparison operations

Operators	Meaning
= =	Equal to
< >	Not equal to
>	greater than
<	less than
> =	greater than or equal to
< =	less than or equal to

The comparison operations support formulating of a jump condition. Arithmetic expressions can also be compared.

The result of comparison operations is "satisfied" or "not satisfied." "Not satisfied" sets the value to zero.

Programming example for comparison operators

```
R1>1 ;R1 greater than 1
1 < R1 ;1 less than R1
R1<R2+R3 ;R1 less than R2 plus R3
R6>=SIN( R7*R7) ; R6 greater than or equal to SIN (R7) squared
```

Programming example

```
N10 IF R1 GOTOF LABEL1 ;If R1 is not null then go to the block having LABEL1
G0 X30 Z30
N90 LABEL1: G0 X50 Z50
N100 IF R1>1 GOTOF LABEL2 ;If R1 is greater than 1 then go to the block having LABEL2
G0 X40 Z40
N150 LABEL2: G0 X60 Z60
G0 X70 Z70
N800 LABEL3: G0 X80 Z80
G0 X100 Z100
N1000 IF R45==R7+1 GOTOB LABEL3 ;If R45 is equal to R7 plus 1 then go to the block having LABEL3
M30
Several conditional jumps in the block:
N10 MC1: G0 X20 Z20
N20 G0 X0 Z0
N30 IF R1==1 GOTOB MC1 IF R1==2 GOTOF MA2 ...
N40 G0 X10 Z10
N50 MA2: G0 X50 Z50
N60 M30
```

Note

The jump is executed for the first fulfilled condition.

11.15.3 Program example for jumps

Task

Approaching points on a circle segment:

Existing conditions:

Start angle: 30° in R1

Circle radius: 32 mm in R2

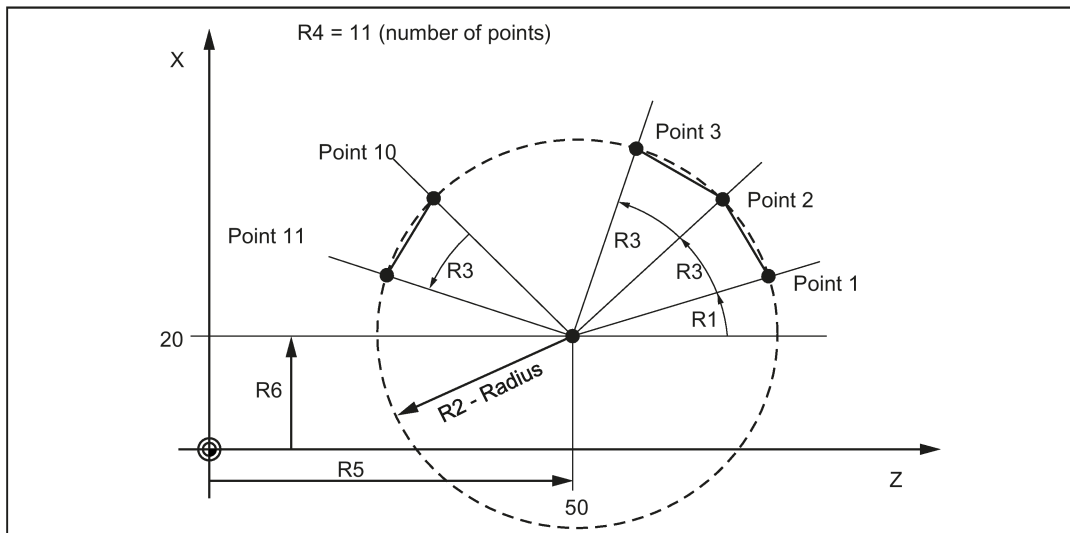
Position spacing: 10° in R3

Number of points: 11 in R4

Position of circle center in Z: 50 mm in R5

Position of circle center in X: 20 mm in R6

See the following illustration for linear approach of points on a circle segment:



Programming example

```

N10 R1=30 R2=32 R3=10 R4=11 R5=50 R6=20          ;Assignment of initial values
N20 MC1: G0 Z=R2*COS (R1)+R5 X=R2*SIN(R1)+R6      ;Calculation and assignment to axis addresses
N30 R1=R1+R3 R4= R4-1
N40 IF R4 > 0 GOTOB MC1
N50 M2

```

Explanation

In block N10, the starting conditions are assigned to the corresponding arithmetic parameters. The calculation of the coordinates in X and Z and the processing takes place in N20.

In block N30, R1 is incremented by the clearance angle R3, and R4 is decremented by 1.

If $R4 > 0$, N20 is executed again; otherwise, N50 with End of program.

11.15.4 Jump destination for program jumps

Functionality

A **label** or a **block number** serve to mark blocks as jump destinations for program jumps. Program jumps can be used to branch to the program sequence.

Labels can be freely selected, but must contain a minimum of 2 and a maximum of 8 letters or numbers of which the **first two characters must be letters** or underscore characters.

Labels that are in the block that serves as the jump destination are **ended by a colon**. They are always at the start of a block. If a block number is also present, the label is located **after the block number**.

Labels must be unique within a program.

Programming example

```

N10 LABEL1: G1 X20          ;LABEL1 is the label, jump destination
N20 G0 X10 Z10
TR789: G0 X10 Z20          ;TR789 is the label, jump destination
G0 X30 Z30                 - No block number existing
N100 G0 X40 Z40            ;Block number can be jump target
M30

```

11.16 Subroutine technique

11.16.1 General information

Usage

Basically, there is no difference between a main program and a subroutine.

Frequently recurring machining sequences are stored in subroutines, e.g. certain contour shapes. These subroutines are called at the appropriate locations in the main program and then executed.

One form of a subroutine is the **machining cycle**. Machining cycles contain universally valid machining scenarios. By assigning values via included transfer parameters, you can adapt the subroutine to your specific application.

Layout

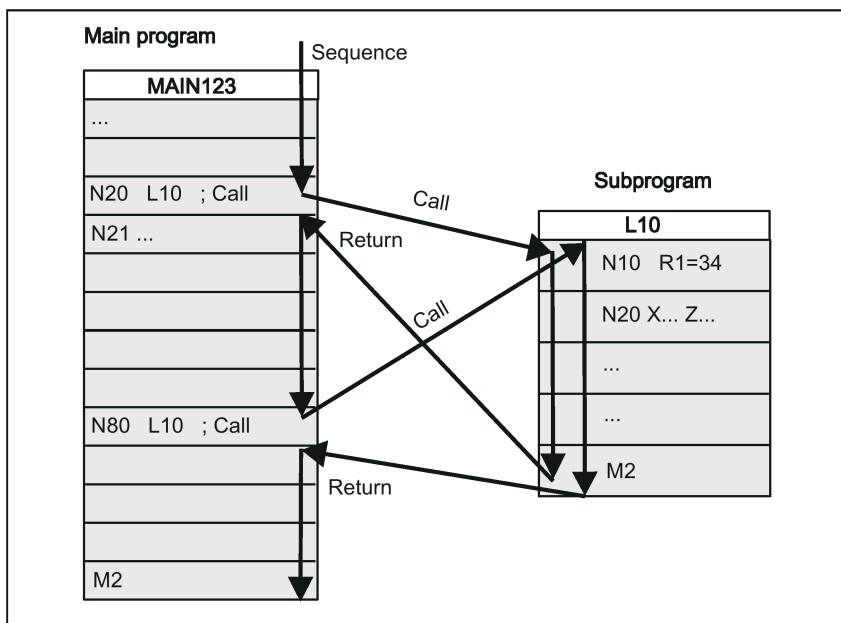
The structure of a subroutine is identical to that of a main program (see Section "Program structure (Page 79)"). Like main programs, subroutines contain **M2 - end of program** in the last block of the program sequence. This means a return to the program level where the subroutine was called from.

End of program

The end instruction **RET** can also be used instead of the M2 program end in the subroutine.

The RET instruction is used when G64 continuous-path mode is not to be interrupted by a return. With M2, G64 is interrupted and exact stop is initiated.

See the following example of a sequence when a subroutine is called in a two-channel manner:



Subroutine name

The subprogram is given a unique name allowing it to be selected from several subroutines. When you create the program, the program name may be freely selected provided the following conventions are observed:

The same rules apply as for the names of main programs.

Example: **BUCHSE7**

It is also possible to use the address word **L...** in subroutines. The value can have 7 decimal places (integers only).

Please observe: With address L, leading zeros are meaningful for differentiation.

Example: **L128** is not **L0128** or **L00128**.

These are 3 different subroutines.

Note: The subroutine name **LL6** is reserved for tool change.

Subroutine call

Subroutines are called in a program (main or subprogram) with their names. To do this, a separate block is required.

Example:

```
N10 L785 ; Subprogram call L785
N20 SHAFT7 ; Subprogram call SHAFT7
```

Program repetition P...

If a subroutine is to be executed several times in succession, write the number of times it is to be executed in the block of the call after the subroutine name under the **address P**. A maximum of **9,999 cycles** are possible (P1 ... P9999).

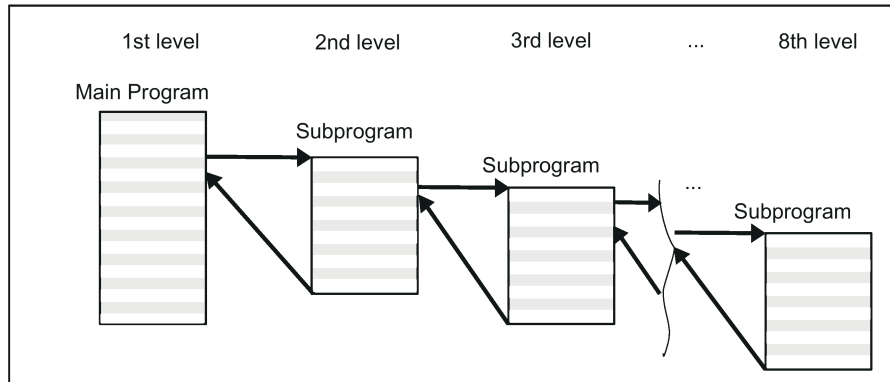
Example:

```
N10 L785 P3 ; Subprogram call L785, 3 cycles
```

Nesting depth

Subroutines can also be called from a subroutine, not only from a main program. In total, up to **8 program levels** are available for this type of nested call, including the main program level.

See the following illustration for execution with 8 program levels:



Information

Modal G functions can be changed in the subroutine, e.g. G90 -> G91. When returning to the calling program, ensure that all modal functions are set the way you need them to be.

Please make sure that the values of your arithmetic parameters used in upper program levels are not inadvertently changed in lower program levels.

When working with SIEMENS cycles, up to 7 program levels are needed.

11.16.2 Calling machining cycles (turning)

Functionality

Cycles are technology subroutines that realize a certain machining process. Adaptation to the particular problem is performed directly via supplying parameters/values when calling the respective cycle.

Programming example

```
N10 DEF REAL RTP, RFP, SDIS, DP, DTB
N20 G18 X100 Z100
N30 M3 S100 F0.1
N40 G17 X0
N50 CYCLE83(110, 90, 0, -80, 0, -10, 0, ;Call of cycle 83, transfer values directly,
0, 0, 0, 1, 0) ;separate block
N60 G0 X100 Z100
N70 RTP=100 RFP= 95.5 SDIS=2.4, DP=-20, ;Set transfer parameters for cycle 82
DTB=3
N80 CYCLE82(RTP, RFP,SDIS, DP, , DTB) ;Call of cycle 82, separate block
N90 M30
```

11.16.3 Executing internal and external subroutines (CALL, EXTCALL)

Function

- With the `CALL` command, you can reload and execute programs stored in the NC directory.
- With the `EXTCALL` command, you can reload and execute programs stored on an external USB memory stick.

Machine data and setting data

The following machine data and setting data is used for the `EXTCALL` command:

- MD18362 \$MN_MM_EXT_PROG_NUM
Number of program levels that can be processed simultaneously from external
- SD42700 \$SC_EXT_PROG_PATH
Program path for external subroutine call EXTCALL

Note

When using SD42700 \$SC_EXT_PROGRAM_PATH, all subprograms called with EXTCALL are searched under this path.

Programming

- `CALL "<Path/MPF.DIR/program name>"`
- `EXTCALL`
 - Programming with path specification in SD42700 EXT_PROGRAM_PATH
("`<program name>`")
 - Programming without path specification in SD42700 EXT_PROGRAM_PATH
("`<path\program name>`")

Parameter

- **For internal subroutines**

CALL	; Keyword for subroutine call
<Path/program name>	; Constant/variable of STRING type
Example: CALL "//NC:/MPF.DIR/Program1.MPF"	

- **For external subroutines**

- Programming with path specification in SD42700 EXT_PROGRAM_PATH

EXTCALL	; Keyword for subroutine call
<program name>	; Constant/variable of STRING type
Example: EXTCALL ("Program2.MPF")	

- Programming without path specification in SD42700 EXT_PROGRAM_PATH

EXTCALL	; Keyword for subroutine call
<Path\program name>	; Constant/variable of STRING type
Example: EXTCALL ("USB:\EXTERNE_UP\Program2.MPF")	

Note

Internal and external subroutines must not contain jump statements such as GOTO, GOTOB, CASE, FOR, LOOP, WHILE, or REPEAT.

IF-ELSE-ENDIF constructions are possible.

Subroutine calls and nested CALL and EXTCALL calls may be used.

RESET, POWER ON

RESET and POWER ON can cause the interruption of internal and external subroutine calls.

Examples

- **Calling an internal subroutine**

The "Main.mpf" main program stored on the USB memory stick is selected for execution:

N10 G4 F0.5	
N20 MSG("CALL Turning")	
N30 CALL "//NC:/MPF.DIR/BOHRUNG.SPF"	
N40 M30	

The "BOHRUNG.SPF" subprogram stored in the NC directory is to be reloaded:

N10 G1 F1000 G94	
N20 X=10 Z=10	
N30 G0 X50 Z50	
...	
...	
N999999 M17	

- **Calling an external subroutine**

The "Main.mpf" main program stored in the NC directory is selected for execution:

N10 G0 X0 Z0	
N20 EXTCALL ("USB:\BOHRUNG.SPF") *	
N30 G0 X100 Z100	
N40 M30	

* If you have set SD 42700 as "USB:\", you only need to enter "EXTCALL ("BOHRUNG.SPF")" alternatively.

The "BOHRUNG.SPF" subprogram stored on the USB memory stick is to be reloaded:

N10 G1 F1000 G94	
N20 X=10 Z=10	
N30 G0 X50 Z50	
...	
...	
N999999 M17	

11.17 Timers and workpiece counters

11.17.1 Runtime timer

Functionality

The timers are prepared as system variables (\$A...) that can be used for monitoring the technological processes in the program or only in the display.

These timers are read-only. There are timers that are always active. Others can be deactivated via machine data.

Timers that are always active

- **\$AN_SETUP_TIME**

Time since the last system power-up with default values (in minutes)

It is automatically reset in the case of a system power-up with default values.

- **\$AN_POWERON_TIME**

Time since the last system power-up (in minutes)

It is reset to zero automatically after each power-up of the control system.

Timers that can be activated machine data

The following timers are activated via machine data (default setting).

Each active run-time measurement is automatically interrupted in the stopped program state or for feedrate-override-zero.

The behavior of the activated timers for active dry run feedrate and program testing can be specified using machine data.

- **\$AC_OPERATING_TIME**

Total execution time in seconds of NC programs in "AUTO" mode

In "AUTO" mode, the runtimes of all programs between program start and end are summed up. The timer is zeroed with each power-up of the control system.

- **\$AC_CYCLE_TIME**

Runtime of the selected NC program (in seconds)

The runtime between program start and end is measured in the selected NC program. The timer is reset with the start of a new NC program.

- **\$AC_CUTTING_TIME**

Tool action time (in seconds)

The runtime of the path axes is measured in all NC programs between program start and end without rapid traverse active and with the tool active (default setting).

The measurement is interrupted when a dwell time is active.

The timer is automatically set to zero after each power-up of the control system.

Programming example

```
N10 IF $AC_CUTTING_TIME>=R10 GOTOF WZZEIT          ; Tool operation time limit value?
G0 X50 Z50
N80 WZZEIT:G0 X60 Z60
N90 MSG("Tool action time: Limit value reached")
N100 M0
M30
```

Display

The content of the active system variables is visible in the window opened through the following key operations:



Window display:

Times / Counter		
Parts in total	①	0
Parts required	②	0
Part count	③	0
Run time	④	0000 H 00 M 00 S
Cycle time	⑤	0000 H 00 M 00 S
Cutting time	⑥	0000 H 00 M 00 S
Setup time	⑦	0002 H 44 M
Power-on time	⑧	0001 H 48 M

① = \$AC_TOTAL_PARTS

② = \$AC_REQUIRED_PARTS

③ = \$AC_ACTUAL_PARTS

⑤ = \$AC_CYCLE_TIME

⑥ = \$AC_CUTTING_TIME

⑦ = \$AN_SETUP_TIME

\$AC_SPECIAL_PARTS is not available for display.

④ = \$AC_OPERATING_TIME

⑧ = \$AN_POWERON_TIME

You can also view the time counter information through the following key operations:



11.17.2 Workpiece counter

Functionality

The "Workpiece counter" function provides counters for counting workpieces.

These counters exist as system variables with write and read access from the program or via operator input (observe the protection level for writing).

You can use the machine data to control counter activation, counter reset timing and the counting algorithm.

Counters

- **\$AC_REQUIRED_PARTS**

Number of workpieces required (workpiece setpoint)

In this counter you can define the number of workpieces at which the actual workpiece counter \$AC_ACTUAL_PARTS is reset to zero.

The generation of the display alarm 21800 "Workpiece setpoint reached" can be activated via machine data.

- **\$AC_TOTAL_PARTS**

Total number of workpieces produced (total actual)

The counter specifies the total number of all workpieces produced since the start time.

The counter is set to zero upon every booting of the control system automatically.

- **\$AC_ACTUAL_PARTS**

Number of actual workpieces (actual)

This counter registers the number of all workpieces produced since the starting time. When the workpiece setpoint is reached (\$AC_REQUIRED_PARTS, value greater than zero), the counter is automatically zeroed.

- **\$AC_SPECIAL_PARTS**

Number of workpieces specified by the user

This counter allows users to make a workpiece counting in accordance with their own definition. Alarm output can be defined for the case of identity with \$AC_REQUIRED_PARTS (workpiece target). Users must reset the counter themselves.

Programming example

```
N10 IF $AC_TOTAL_PARTS==R15 GOTOF SIST ; Count reached?
G0 X50 Z50
N80 SIST:G0 X60 Z60
N90 MSG("Workpiece setpoint reached")
N100 M0
M30
```

Display

The content of the active system variables is visible in the window opened through the following key operations:



Window display:

Times / Counter		
Parts in total	①	0
Parts required	②	0
Part count	③	0
Run time	④	0000 H 00 M 00 s
Cycle time	⑤	0000 H 00 M 00 s
Cutting time	⑥	0000 H 00 M 00 s
Setup time	⑦	0002 H 44 M
Power-on time	⑧	0001 H 48 M

① = \$AC_TOTAL_PARTS

② = \$AC_REQUIRED_PARTS

③ = \$AC_ACTUAL_PARTS

⑤ = \$AC_CYCLE_TIME

⑥ = \$AC_CUTTING_TIME

⑦ = \$AN_SETUP_TIME

\$AC_SPECIAL_PARTS is not available for display.

④ = \$AC_OPERATING_TIME

⑧ = \$AN_POWERON_TIME

You can also select whether to activate the workpiece counter function through the following key operations:



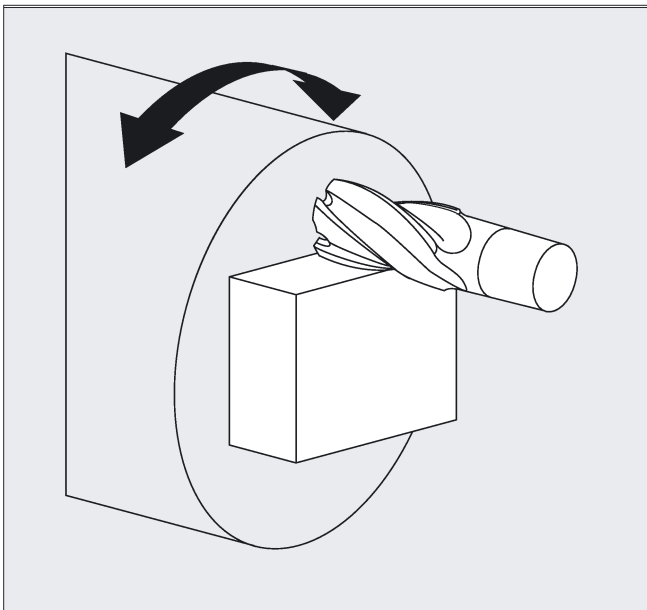
11.18 Kinematic transformation

11.18.1 Face milling on turned parts (TRANSMIT)

Function

The TRANSMIT function enables the following:

- Face machining on turned parts in the turning clamp (drill-holes, contours).
- A cartesian coordinate system can be used to program these machining operations.
- The controller maps the programmed traversing movements of the Cartesian coordinate system onto the traversing movements of real machine axes (standard situation):
 - Rotary axis:
The main spindle functions here as the machine rotary axis.
 - Infeed axis perpendicular to rotary axis
 - Longitudinal axis parallel to rotary axis
 - The linear axes are positioned perpendicular to one another.
- A tool center offset relative to the turning center is permitted.
- The velocity control makes allowance for the limits defined for the rotations.
- In addition to the tool length compensation, it is also possible to work with the tool radius compensation (G41, G42).



TRANSMIT transformation type

TRANSMIT in the standard case with (TRAFO_TYPE_n = 256)

Syntax

TRANSMIT

TRAFOOF

Rotary axis

The rotary axis cannot be programmed as it is occupied by a geometry axis and thus cannot be programmed directly as channel axis.

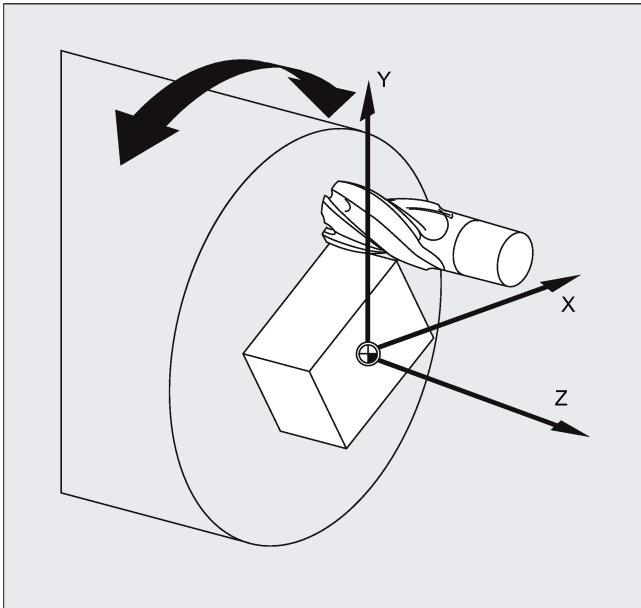
Meaning

TRANSMIT:	Activates the first declared TRANSMIT function. This function is also designated as polar transformation.
TRAFOOF:	Deactivates an active transformation.
OFFN:	Offset contour normal: Distance of the face machining from the programmed reference contour.

Note

An active TRANSMIT transformation is likewise deactivated if one of the other transformations is activated in the relevant channel (e.g. TRACYL).

Example



Program code	Comment
N10 T1 D1 G54 G17 G90 F1000 G94	; Tool selection
N20 G0 X20 Z10 SPOS=45	; Approach the starting position
N30 SETMS(2)	; Set the second spindle as the main spindle
N40 M3 S2000	; Run the spindle
N50 TRANSMIT	; Activate TRANSMIT function
N60 ROT RPL=-45	; Set frame
N70 DIAMOF	
N80 G1 X10 Y-10 G41 OFFN=1OFFN	; Square roughing; 1 mm tolerance
N90 X-10	
N100 Y10	
N110 X10	
N120 Y-10	
N130 G1 Z20 G40 OFFN=0	; Tool change
N140 T2 D1 X15 Y-15	
N150 Z10 G41	
N160 G1 X10 Y-10	; Square finishing
N170 X-10	
N180 Y10	
N190 X10	
N200 Y-10	
N210 Z20 G40	; Deselect frame
N220 TRAFOOF	
N230 SETMS(1)	; Set the first spindle back
N240 G0 X20 Z10 SPOS=45	; Approach the starting position
N250 M30	

Description

Pole

There are two ways of passing through the pole:

- Traverse the linear axis alone
- Traverse to the pole, rotate the rotary axis at the pole and traveling away from the pole

Make the selection using MD 24911 and 24951.

Information

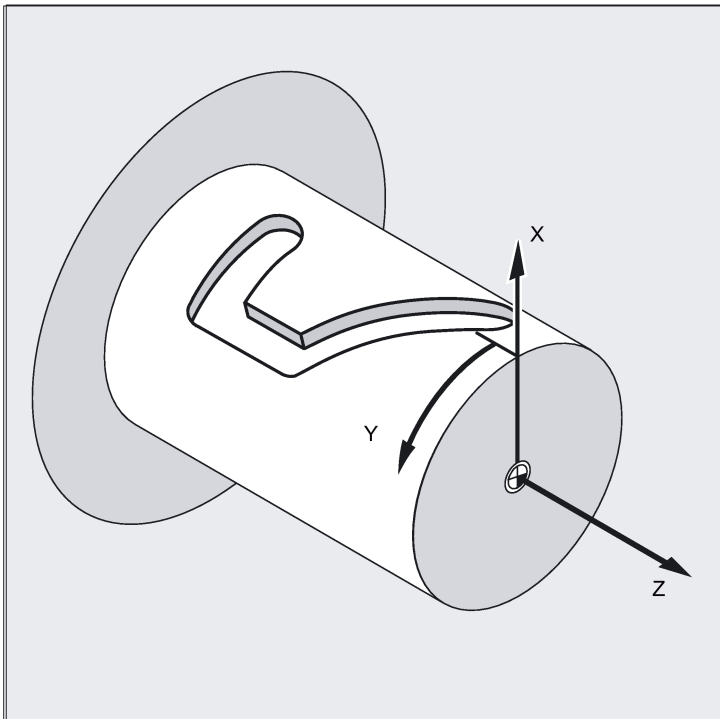
The turning center with X0/Y0 is designated as the pole. Workpiece machining operations close to the pole are not recommended since these may require sharp feedrate reductions to prevent overloading of the rotary axis. Avoid selecting TRANSMIT when the tool is positioned exactly on the pole. Ensure that the path of the tool center point does not travel through the X0/Y0 pole.

11.18.2 Cylinder surface transformation (TRACYL)

Functionality

- The TRACYL cylinder surface transformation function can be used to machine:
 - Longitudinal grooves on cylindrical bodies
 - Transverse grooves on cylindrical objects
 - Grooves with any path on cylindrical bodies

The path of the grooves is programmed with reference to the unwrapped, level surface of the cylinder.



- The control system transforms the programmed traversing movements in the Cartesian coordinate X, Y, Z system into traversing movements of the real machine axes. The main spindle functions here as the machine rotary axis.
- TRACYL must be configured using special machine data. The rotary axis position at which the value Y=0 is also defined here.
- If the machine has a real machine Y-axis(YM), an expanded TRACYL variant can also be configured. This allows grooves with groove wall offset to be produced: the groove wall and base are perpendicular to each other - even if the milling tool's diameter is smaller than the groove width. This is otherwise only possible with exact fitting milling cutters.

TRACYL transformation types

There are three forms of cylinder surface coordinate transformation:

- TRACYL without groove wall offset (TRAFO_TYPE_n=512)
- TRACYL with groove wall offset: (TRAFO_TYPE_n=513)
- TRACYL with additional linear axis and groove wall offset: (TRAFO_TYPE_n=514)
The groove wall offset is parameterized with TRACYL using the third parameter.

For cylinder peripheral curve transformation with groove side compensation, the axis used for compensation should be positioned at zero (y=0), so that the groove centric to the programmed groove center line is finished.

Axis utilization

The following axes cannot be used as a positioning axis or a reciprocating axis:

- The geometry axis in the peripheral direction of the cylinder peripheral surface (Y axis)
- The additional linear axis for groove side compensation (Z axis).

Programming

TRACYL(d) or TRACYL(d, n) or for transformation type 514

TRACYL(d, n, groove side offset)

TRAFOOF

Rotary axis

The rotary axis cannot be programmed as it is occupied by a geometry axis and thus cannot be programmed directly as channel axis.

Meaning

TRACYL(d)	Activates the first TRACYL function specified in the channel machine data. d is the parameter for the working diameter.
TRACYL (d, n)	Activates the n-th TRACYL function specified in the channel machine data. The maximum for n is 2, TRACYL(d,1) corresponds to TRACYL(d).
D	Value for the working diameter. The working diameter is double the distance between the tool tip and the turning center. This diameter must always be specified and be larger than 1.
n	Optional 2nd parameter for the TRACYL data block 1 (preselected) or 2.
Slot side compensation	Optional 3rd parameter whose value for TRACYL is preselected using the mode for machine data. Value range: 0: Transformation type 514 without groove wall offset as previous 1: Transformation type 514 with groove wall offset
TRAFOOF	Transformation OFF (BCS and MCS are once again identical).
OFFN	Offset contour normal: Distance of the groove side from the programmed reference contour.

Note

An active TRACYL transformation is likewise deactivated if one of the other transformations is activated in the relevant channel (e.g. TRANSMIT).

OFFN address

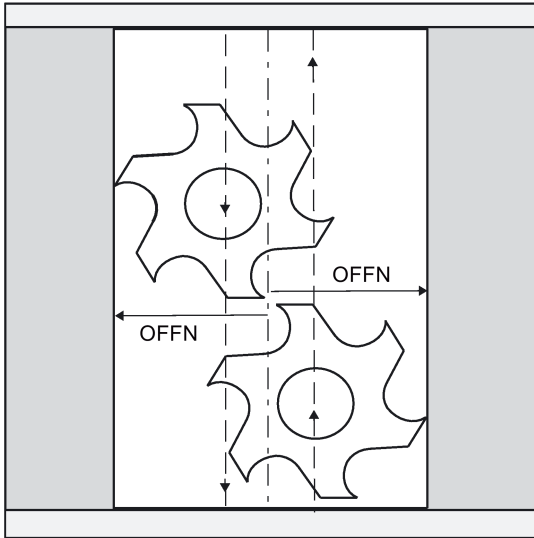
Distance from the groove side wall to the programmed path.

The groove center line is generally programmed. OFFN defines the (half) groove width for activated milling cutter radius compensation (G41, G42).

Programming: OFFN=...; distance in mm

Note

Set OFFN=0 once the groove has been completed. OFFN is also used outside of TRACYL - for offset programming in combination with G41, G42.

**Example: Tool definition**

The following example is suitable for testing the parameterization of the TRACYL cylinder transformation:

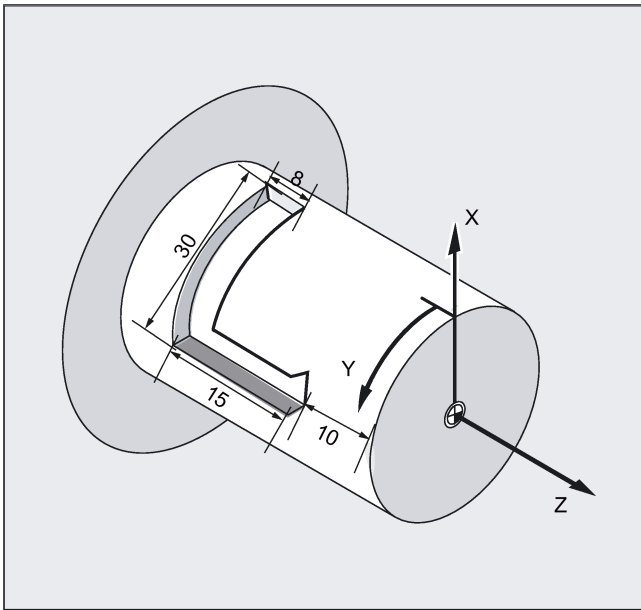
Program code	Comment
Tool parameters	Meaning
Number (DP)	
\$TC_DP1[1,1]=120	Tool type (Milling tool)
\$TC_DP2[1,1]=0	Cutting edge position (Only for turning tools)

Program code	Comment
Geometry	Length compensation
\$TC_DP3[1,1]=8	Length offset vector (Calculation acc. to type and plane)
\$TC_DP4[1,1]=9	
\$TC_DP5[1,1]=7	

Program code	Comment
Geometry	Radius
\$TC_DP6[1,1]=6	Tool radius
\$TC_DP7[1,1]=0	Slot width b for slotting saw, rounding radius for milling tools
\$TC_DP8[1,1]=0	Projection k (For slotting saw only)
\$TC_DP9[1,1]=0	
\$TC_DP10[1,1]=0	
\$TC_DP11[1,1]=0	Angle for taper milling tools

Program code	Comment
Wear	Length and radius compensation
\$TC_DP12[1,1]=0	Remaining parameters to \$TC_DP24=0 (Tool base dimension/adapter)

Example: Making a hook-shaped groove



Activate cylinder surface transformation:

Required tool: T1 milling tool, radius=3 mm, edge position=8.

Program code	Comment
N10 T1 D1 G54 G90 G94	; Tool selection, clamping compensation
F1000	
N20 SPOS=0	; Approach the starting position
N30 SETMS(2)	; Set the second spindle as the main spindle
N40 M3 S2000	; Run the spindle
N50 DIAMOF	; Change the diameter dimensioning to radius dimensioning
N60 G0 X23 Z105	
N70 TRACYL (20)	; Activate cylinder surface ;transformation
N80 G19	; Plane selection

Machining a hook-shaped groove:

Program code	Comment
N90 G1 Y0 Z-10	; Approach starting position
N100 G42 OFFN=-4.5	; Tool radius compensation right of contour on
N110 X19 F500	
N120 Z-25	
N130 Y30	
N140 OFFN=-3.5	
N150 Y0	
N160 Z-10	
N170 X25	
N180 TRAFOF	
N190 DIAMON	; Diameter dimensioning
N200 G40	; Tool radius compensation off
N210 G0 X80 Z100	; Retraction in rapid traverse
N220 M30	; End of program

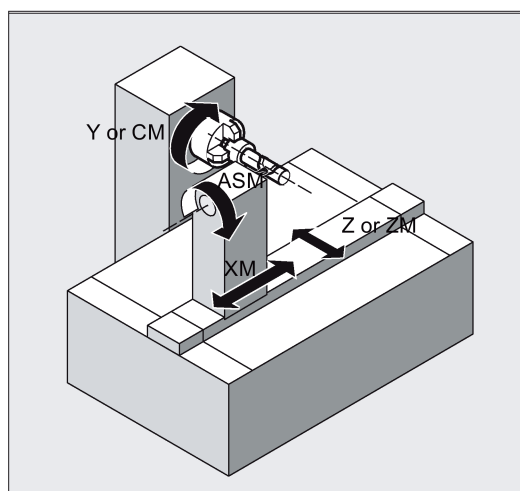
Description

Without groove wall offset (transformation type 512)

The controller transforms the programmed traversing movements of the cylinder coordinate system to the traversing movements of the real machine axes:

- Rotary axis
- Infeed axis perpendicular to rotary axis
- Longitudinal axis parallel to rotary axis

The linear axes are positioned perpendicular to one another. The infeed axis cuts the rotary axis.

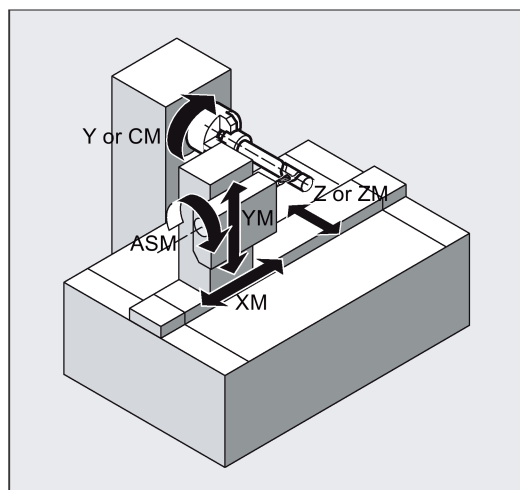


With groove wall offset (transformation type 513)

Kinematics as above, but an additional longitudinal axis parallels to the peripheral direction.

The linear axes are positioned perpendicular to one another.

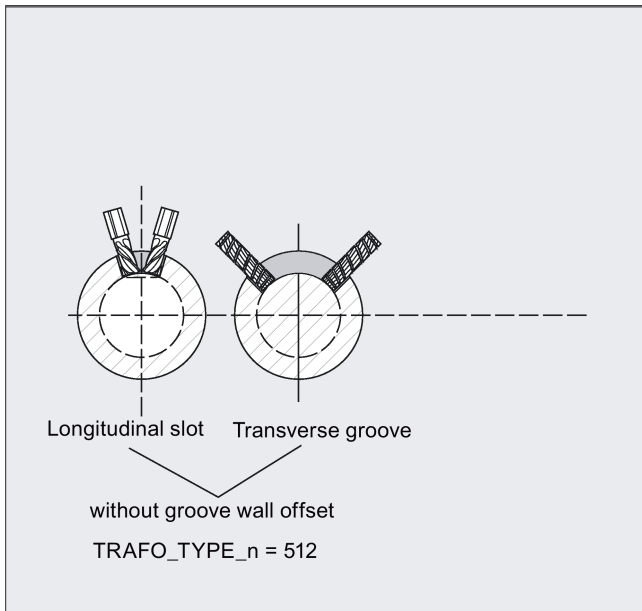
The velocity control makes allowance for the limits defined for the rotations.



Groove traversing-section

In the case of axis configuration 1, longitudinal grooves along the rotary axis are subject to parallel limits only if the groove width corresponds exactly to the tool radius.

Grooves in parallel to the periphery (transverse grooves) are not parallel at the beginning and end.



With additional linear axis and groove wall offset (transformation type 514)

On a machine with a second linear axis, this transformation variant makes use of redundancy in order to perform improved tool compensation. The following conditions then apply to the second linear axis:

- A smaller working area
- The second linear axis should not be used for traversing the part program.

Certain machine data settings are assumed for the part program and the assignment of the corresponding axes in the BCS or MCS.

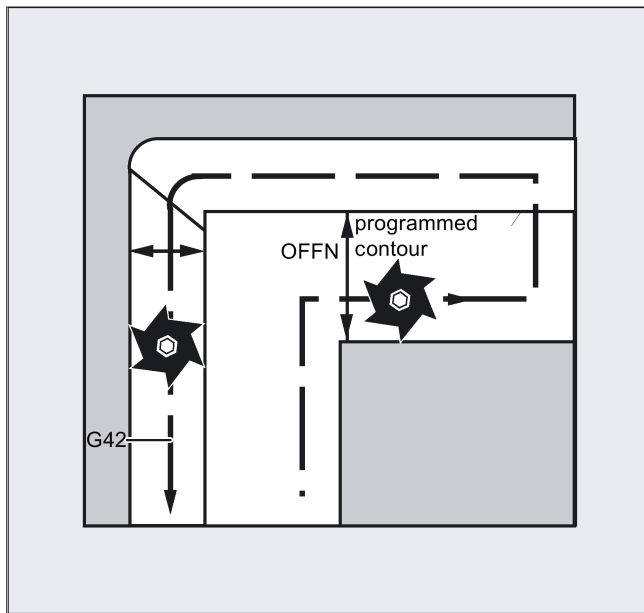
For more information, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Function Manual.

Offset contour normal OFFN (transformation type 513)

To mill grooves with TRACYL, the following is programmed:

- Groove center line in the part program
- Half the groove width programmed using OFFN.

To avoid damage to the groove side OFFN acts only when the tool radius compensation is active. Furthermore, OFFN should also be $\geq$ the tool radius to avoid damage occurring to the opposite side of the groove.



A part program for milling a groove generally comprises the following steps:

1. Selecting a tool
2. Select TRACYL
3. Select suitable coordinate offset (frame)
4. Positioning
5. Program OFFN
6. Select TRC
7. Approach block (position TRC and approach groove side)
8. Groove center line contour
9. Deselect TRC
10. Retraction block (retract TRC and move away from groove side)
11. Positioning
12. Deselect OFFN
13. TRAFOOF
14. Re-select original coordinate shift (frame)

Special features

- **TRC selection:**
TRC is not programmed in relation to the groove side, but relative to the programmed groove center line. To prevent the tool traveling to the left of the groove side, G42 is entered (instead of G41). You avoid this if in OFFN, the groove width is entered with a negative sign.
- **OFFN acts differently with TRACYL than it does without TRACYL.** As, even without TRACYL, OFFN is included when TRC is active, OFFN should be reset to zero after TRAFOOF.
- **It is possible to change OFFN within a part program.** This could be used to shift the groove center line from the center (see diagram).
- **Guiding grooves:**
TRACYL does not create the same groove for guiding grooves as it would be with a tool with the diameter producing the width of the groove. It is basically not possible to create the same groove side geometry with a smaller cylindrical tool as it is with a larger one. TRACYL minimizes the error. To avoid problems of accuracy, the tool radius should only be slightly smaller than half the groove width.

Note**OFFN and TRC**

With TRAFO_TYPE_n = 512, the value is effective under OFFN as an allowance for TRC. With TRAFO_TYPE_n = 513, half the groove width is programmed in OFFN. The contour is retracted with OFFN-TRC.

12 Cycles

12.1 Overview of cycles

Cycles are generally applicable technology subroutines that you can use to carry out a specific machining process, such as tapping. These cycles are adapted to individual tasks by parameter assignment.

Drilling cycles and turning cycles

You can carry out the following standard cycles by using the control system:

- **Drilling cycles**
 - CYCLE81: Drilling, centering
 - CYCLE82: Drilling, counterboring
 - CYCLE83: Deep-hole drilling
 - CYCLE84: Rigid tapping
 - CYCLE840: Tapping with compensating chuck
 - CYCLE85: Reaming 1
 - CYCLE86: Boring
- **Turning cycles**
 - CYCLE92: Cut-off
 - CYCLE93: Recess
 - CYCLE94: Undercut (DIN form E and F)
 - CYCLE95: Stock removal with relief cutting
 - CYCLE96: Thread undercut
 - CYCLE98: Thread chain
 - CYCLE99: Thread cutting

12.2 Programming cycles

A standard cycle is defined as a subroutine with name and parameter list.

Call and return conditions

The G functions effective prior to the cycle call and the programmable offsets remain active beyond the cycle.

The machining plane G17 for drilling cycles or G18 for turning cycles is defined before calling the cycle.

With drilling cycles, the drilling operation is carried out in the axis standing vertically to the current plane.

Messages output during execution of a cycle

Messages about the state of machining are displayed on the screen of the control system during program execution.

These messages do not interrupt the program execution and continue to be displayed on the screen until the next message appears.

The message texts and their meaning are listed together with the cycle to which they refer.

Block display during execution of a cycle

During the cycle execution, the cursor is always located on the program blocks of the cycle.

Cycle call and parameter list

The defining parameters for the cycles can be transferred via the parameter list when the cycle is called.

Note

Cycle calls must always be programmed in a separate block.

Basic instructions with regard to the assignment of standard cycle parameters

Each defined parameter of a cycle has a certain data type. The parameter being used must be specified when the cycle is called. The following can be transferred in the parameter list:

- R parameters (only numerical values)
- Constants

If R parameters are used in the parameter list, values must first be assigned to them in the program to be called. Proceed through either of the following methods to call the cycles:

- using an incomplete parameter list
- omitting parameters

If transfer parameters are omitted at the end of the parameter list, the parameter list must be prematurely ended with ")". If any parameters are to be omitted within the list, a comma "... , ..." must be written as a placeholder.

No plausibility checks are made for parameter values with a limited range of values unless an error response has been specifically described for a cycle.

When calling the cycle, if the parameter list contains more entries than parameters are defined in the cycle, the general NC alarm 12340 "Too many parameters" is displayed and the cycle is not executed.

Note

Axis-specific and channel-specific machine data of the spindle must be configured.

Cycle call

The individual methods for writing a cycle are shown in the programming examples provided for the individual cycles.

Simulation of cycles

Programs with cycle calls can be tested first in simulation.

During simulation, the traversing movements of the cycle are visualized on the screen.

12.3 Graphical cycle programming in the program editor

The program editor in the control system provides cycle programming. You can enter parameters in the cycle and call the cycle in the program.

Function

The cycle support consists of three components:

- Cycle selection
- Input screens for parameter assignment
- Help display per cycle

Operating the cycle support

To add a cycle call to the program, proceed through the following steps:



1. Select the program editing operating area.



2. Select a cycle type with the corresponding horizontal softkey to open the lower-level vertical softkey bar until the desired input screen form with the help display appears on the screen.
3. Enter the values directly (numerical values) or indirectly (R parameters, for example, R27, or expressions consisting of R parameters, for example, R27 + 10).
If numerical values are entered, the control system will automatically perform a check to see whether the value lies within the permitted range.
4. Use this key to select values for some parameters that may have only a few values for selection.
5. For drilling cycles, it is also possible to call a cycle modally with this key. To deselect the modal call, move the cursor to the next blank line of the program and press the softkey below:
6. Press this softkey to confirm your input. To cancel the input, press the softkey below:

Recompiling

Recompiling of program codes serves to make modifications to an existing program using the cycle support.



Position the cursor on the program line for the cycle to be modified and press this softkey. This reopens the input screen of parameter assignment for the cycle, and you can modify and accept the parameter values.

12.4 Drilling cycles

12.4.1 General information

Drilling cycles are motional sequences specified according to DIN 66025 for drilling, boring, tapping, etc.

They are called in the form of a subroutine with a defined name and a parameter list.

They all follow a different technological procedure and are therefore parameterized differently.

The drilling cycles can be modally effective, i.e. they are executed at the end of each block which contains motion commands.

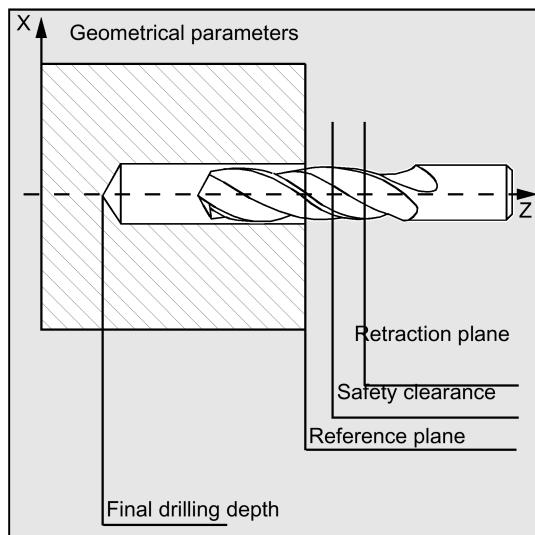
There are two types of parameters:

- Geometrical parameters
- Machining parameters

The geometrical parameters are identical with all drilling cycles. They define the reference and retraction planes, the safety clearance and the absolute or relative final drilling depth. Geometrical parameters are assigned once during the first drilling cycle CYCLE82.

The machining parameters have a different meaning and effect in the individual cycles. They are therefore programmed in each cycle separately.

See the following illustration for geometrical parameters:



12.4.2 Requirements

Call and return conditions

Drilling cycles are programmed independently of the actual axis names. The drilling position must be approached in the higher-level program before the cycle is called.

The required values for feedrate, spindle speed and direction of spindle rotation must be programmed in the part program if there are no defining parameters in the drilling cycle.

The G functions and the current data record active before the cycle was called remain active beyond the cycle.

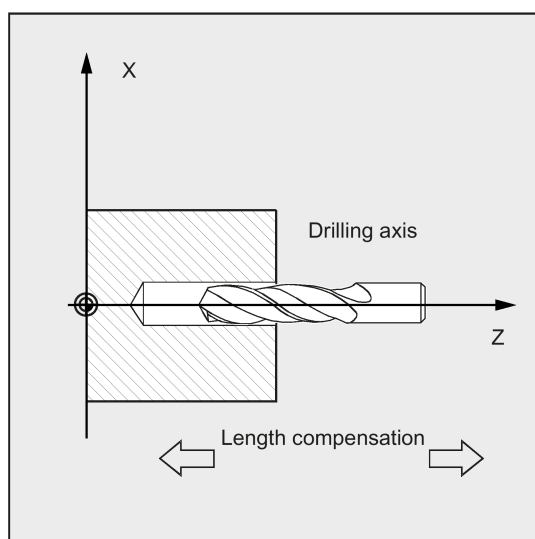
Plane definition

In the case of drilling cycles, it is generally assumed that the current workpiece coordinate system in which the machining operation is to be performed is to be defined by selecting plane G17 and activating a programmable offset. The drilling axis is always the axis of this coordinate system which stands vertically to the current plane.

A tool length compensation must be selected before the cycle is called. Its effect is always perpendicular to the selected plane and remains active even after the end of the cycle.

In turning, the drilling axis is thus the Z axis. Drilling is performed to the end face of the workpiece.

See the following illustration for drilling axis when turning:



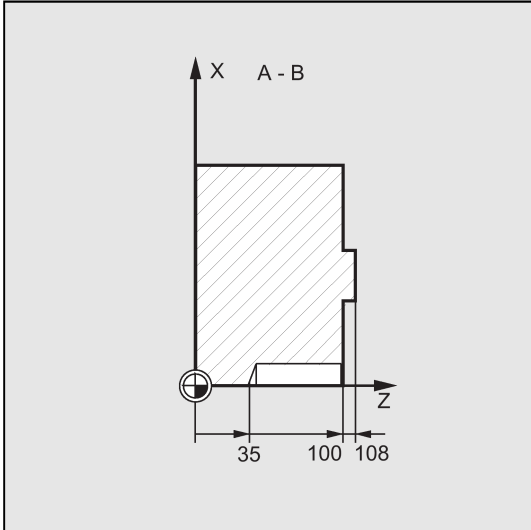
Dwell time programming

The parameters for dwell times in the drilling cycles are always assigned to the F word and must therefore be assigned with values in seconds. Any deviations from this procedure must be expressly stated.

Special features when using drilling cycles on a turning machine

Simple turning machines without driven tools can apply drilling cycles only for drilling on the end face (with Z-axis) in the turning center. These drilling cycles must always be called in the G17 plane.

See the following example for drilling centering _drilling on turning center without tool:

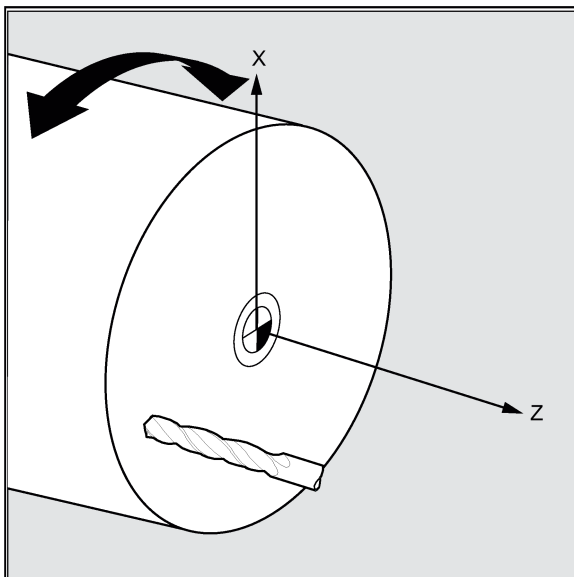


Turning machines with driven tools can also drill off-center on the end face or on the peripheral surface if the machine setup permits this.

The following must be observed when drilling off-center on the end face:

- Working plane is G17 - Z is the resulting tool axis.
- The spindle of the driven tool must be declared to the master spindle (SETMS command).
- The drilling position can be programmed either with X and the C-axis or, if TRANSMIT is active, with Y and Z.

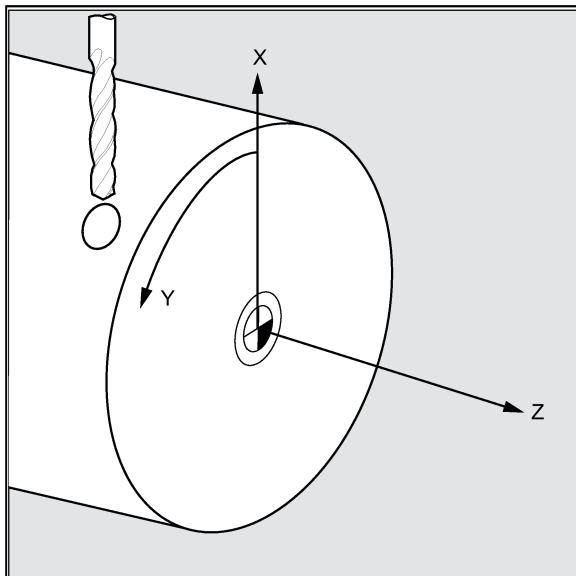
See the following illustration for drilling on end face with a driven tool:



The following must be observed when drilling on the peripheral surface:

- Working plane is G19 - X is the resulting tool axis.

- The spindle of the driven tool must be declared to the master spindle (SETMS command).
- The drilling position can be programmed either with Z and the C-axis or, if TRACYL is active, with X and Z.
See the following illustration for drilling on peripheral surface with a driven tool:



12.4.3 Drilling, centering - CYCLE81

Programming

CYCLE81 (RTP, RFP, SDIS, DP, DPR)

Parameters

Parameter	Data type	Description
RTP	REAL	Retraction plane (absolute)
RFP	REAL	Reference plane (absolute)
SDIS	REAL	Safety clearance (enter without sign)
DP	REAL	Final drilling depth (absolute)
DPR	REAL	Final drilling depth relative to the reference plane (enter without sign)

Function

The tool drills at the programmed spindle speed and feedrate to the entered final drilling depth.

Sequence

Position reached prior to cycle start:

The drilling position is the position in the two axes of the selected plane.

The cycle creates the following sequence of motions:

- Approaching the position of safety distance relative to the reference plane with G0
- Traversing to the final drilling depth with the feedrate (G1) programmed prior to the cycle call
- Retracting to the retraction plane with G0

Explanation of the parameters

RFP and RTP (reference plane and retraction plane)

Normally, reference plane (RFP) and return plane (RTP) have different values. The cycle assumes that the retraction plane precedes the reference plane. This means that the distance from the retraction plane to the final drilling depth is larger than the distance from the reference plane to the final drilling depth.

SDIS (safety clearance)

The safety clearance (SDIS) acts with reference to the reference plane. This is brought forward by the safety clearance.

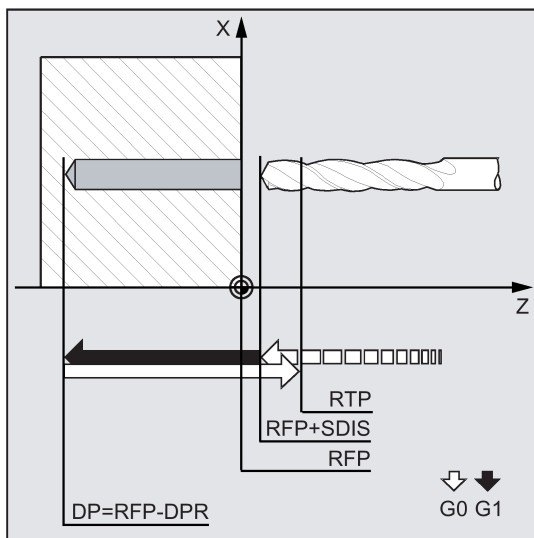
The direction in which the safety clearance is active is automatically determined by the cycle.

DP and DPR (final drilling depth)

The final drilling depth can be specified either absolute (DP) or relative (DPR) to the reference plane.

With relative specification, the cycle will calculate the resulting depth automatically using the positions of reference and retraction planes.

See the following illustration for final drilling depth:



Note

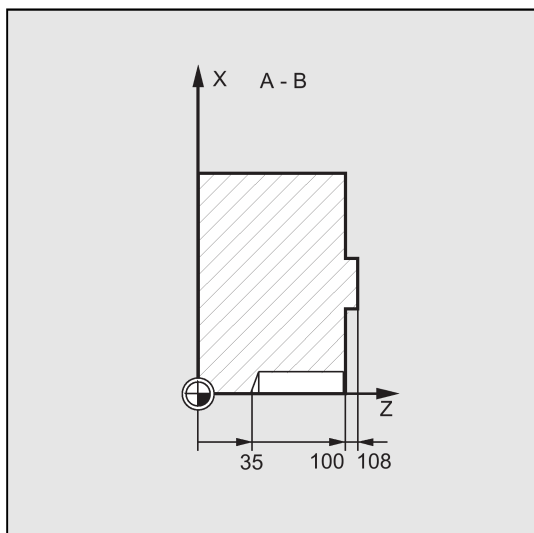
If a value is entered both for DP and for DPR, the final drilling depth is derived from DPR. If this differs from the absolute depth programmed via DP, the message "Depth: Corresponding to value for relative depth" is output in the message line.

If the values for reference and retraction planes are identical, a relative depth specification is not permitted. The error message 61101 "Reference plane defined incorrectly" is output and the cycle is not executed. This error message is also output if the retraction plane is located after the reference plane, i.e. its distance to the final drilling depth is smaller.

Programming example: Drilling_centering

This program produces three drill holes using the CYCLE81 drilling cycle. The drilling axis is always the Z axis.

See the following example for drilling_centering_drilling on turning_center without tool:



```

N10 G0 G17 G90 F200 S300 M3
N20 D1 T3 Z110
N30 X0
N40 CYCLE81(110, 100, 2, 35,)
M30

```

```

; Specification of technology values
; Approach retraction plane
; Approach drilling position
; Cycle call with absolute end drilling depth, safety
clearance and incomplete parameter list
; Program end

```

12.4.4 Drilling, counterboring - CYCLE82

Programming

CYCLE82 (RTP, RFP, SDIS, DP, DPR, DTB)

Parameters

Parameter	Data type	Description
RTP	REAL	Retraction plane (absolute)
RFP	REAL	Reference plane (absolute)
SDIS	REAL	Safety clearance (enter without sign)
DP	REAL	Final drilling depth (absolute)
DPR	REAL	Final drilling depth relative to the reference plane (enter without sign)
DTB	REAL	Dwell time at final drilling depth (chip breakage)

Function

The tool drills at the programmed spindle speed and feedrate to the entered final drilling depth. A dwell time can be allowed to elapse when the final drilling depth has been reached.

Sequence

Position reached prior to cycle start:

The drilling position is the position in the two axes of the selected plane.

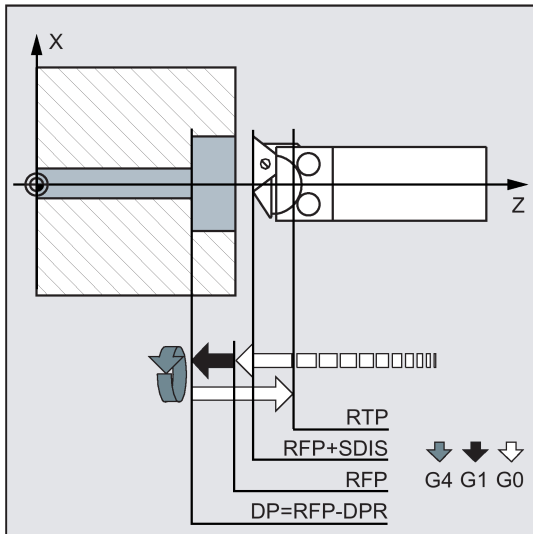
The cycle creates the following sequence of motions:

- Approaching the position of safety distance relative to the reference plane with G0
- Traversing to the final drilling depth with the feedrate (G1) programmed prior to the cycle call
- Dwell time at final drilling depth
- Retraction to the retraction plane with G0

Explanation of the parameters

For an explanation of the parameters RTP, RFP, SDIS, DP, and DPR, see Section "Drilling, centering - CYCLE81 (Page 154)".

See the following parameters for CYCLE82:



DTB (dwell time)

The dwell time to the final drilling depth (chip breakage) is programmed under DTB in seconds.

Programming example 1: Boring_counterboring

The program machines a single hole of a depth of 20 mm at position X0 with cycle CYCLE82.

The dwell time programmed is 3 s, the safety clearance in the drilling axis Z is 2.4 mm.

```
N10 G0 G90 G54 F2 S300 M3 ; Specification of technology values
N20 D1 T6 Z50 ; Approach retraction plane
N30 G17 X0 ; Approach drilling position
N40 CYCLE82 (3, 1.1, 2.4, -20, , 3) ; Cycle call with absolute final drilling depth and safety
clearance
N50 M2 ; End of program
```

Programming example 2

Proceed through the following steps:



1. Select the program editing operating area.



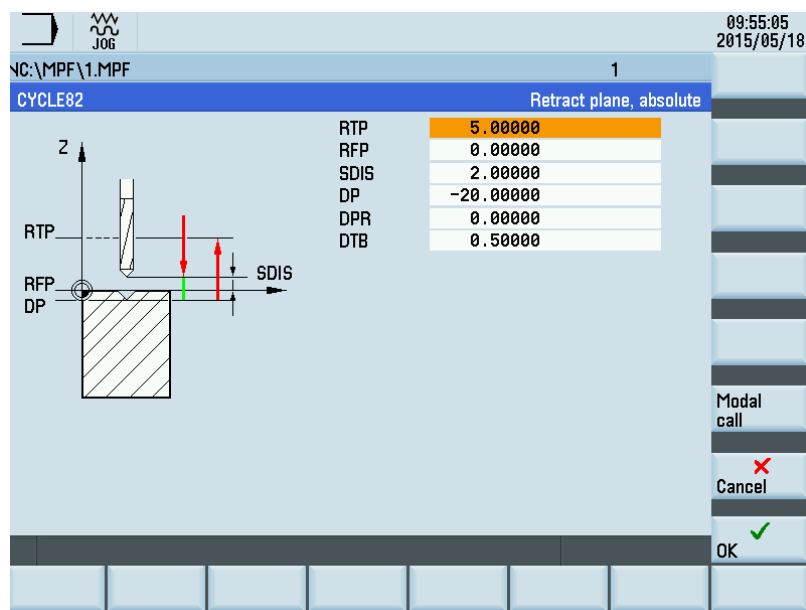
2. Open the vertical softkey bar for available drilling cycles.



3. Press this softkey from the vertical softkey bar.



4. Press this softkey to open the window for CYCLE82. Parameterize the cycle as desired.



5. Confirm your settings with this softkey. The cycle is then automatically transferred to the program editor as a separate block.

```
NC:\MPF\1.MPF
CYCLE82( 5, 0, 2, -20, 0, 0.5)
==eof==
```

12.4.5 Deep-hole drilling - CYCLE83

Programming

CYCLE83 (RTP, RFP, SDIS, DP, DPR, FDEP, FDPR, DAM, DTB, DTS, FRF, VARI, AXN, MDEP, VRT, DTD, DIS1)

Parameters

Parameter	Data type	Description
RTP	REAL	Retraction plane (absolute)
RFP	REAL	Reference plane (absolute)
SDIS	REAL	Safety clearance (enter without sign)
DP	REAL	Final drilling depth (absolute)
DPR	REAL	Final drilling depth relative to the reference plane (enter without sign)
FDEP	REAL	First drilling depth (absolute)
FDPR	REAL	First drilling depth relative to the reference plane (enter without sign)
DAM	REAL	Amount of degression (enter without sign) Values: >0: degression as value <0: degression factor =0: no degression
DTB	REAL	Dwell time at drilling depth (chip breakage) Values: >0: in seconds <0: in revolutions

Parameter	Data type	Description
DTS	REAL	Dwell time at starting point and for chip removal Values: >0: in seconds <0: in revolutions
FRF	REAL	Feedrate factor for the first drilling depth (enter without sign) Range of values: 0.001 ... 1
VARI	INT	Machining type: Chip breakage=0, Chip removal=1
AXN	INT	Tool axis (values: 1 = 1st geometrical axis; 2 = 2nd geometrical axis; 3 = 3rd geometrical axis)
MDEP	REAL	Minimum drilling depth (only in connection with degression factor)
VRT	REAL	Variable return path with chip breakage (VARI=0) Values: >0: if traction value =0: retraction value 1mm set
DTD	REAL	Dwell time at final drilling depth Values: >0: in seconds <0: in revolutions =0: value same as DTB
DIS1	REAL	Programmable limit distance for reinsertion in the drill hole (for chip removal VARI=1) Values: >0: programmable value applies =0: automatic calculation

Function

The tool drills at the programmed spindle speed and feedrate to the entered final drilling depth.

Deep hole drilling is performed with a depth infeed of a maximum definable depth executed several times, increasing gradually until the final drilling depth is reached.

The drill can either be retracted to the reference plane + safety clearance after every infeed depth for swarf removal or retracted by 1 mm each time.

Sequence

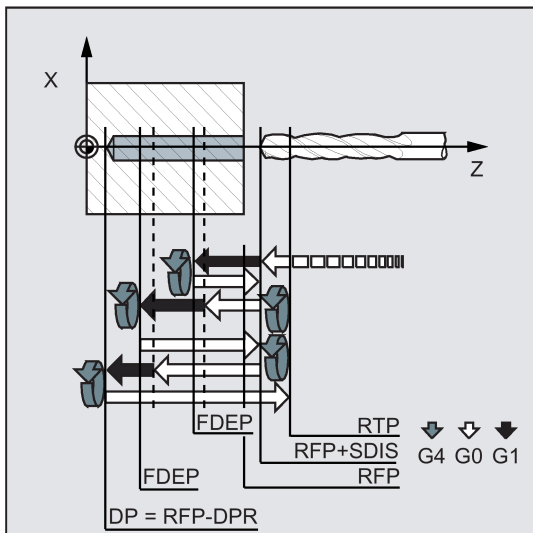
Position reached prior to cycle start:

The drilling position is the position in the two axes of the selected plane.

The cycle creates the sequence below:

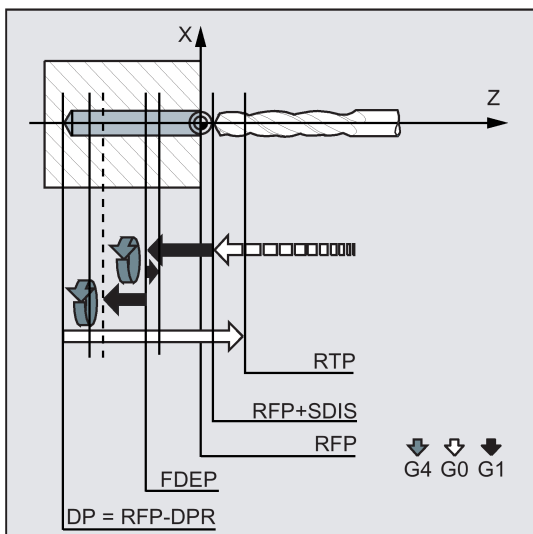
Deep-hole drilling with chip removal (VARI=1):

- Approaching the position of safety distance relative to the reference plane with G0
- Traversing to the first drilling depth with G1, the feedrate for which is derived from the feedrate defined with the program call which is subject to parameter FRF (feedrate factor)
- Dwell time at final drilling depth (parameter DTB)
- Retracting to a position of safety distance relative to the reference plane for swarf removal with G0
- Dwell time at the starting point (parameter DTS)
- Approach of the drilling depth last reached, reduced by anticipation distance with G0
- Traversing to the next drilling depth with G1 (sequential motions is continued until the final drilling depth is reached)
- Retracting to the retraction plane with G0



Deep-hole drilling with chip breakage (VARI=0):

- Approaching to a position of safety distance relative to the reference plane with G0
- Traversing to the first drilling depth with G1, the feedrate for which is derived from the feedrate defined with the program call which is subject to parameter FRF (feedrate factor)
- Dwell time at final drilling depth (parameter DTB)
- Retraction by 1 mm from the current drilling depth with G1 and the feedrate programmed in the calling program (for chip breaking)
- Traversing to the next drilling depth with G1 and the programmed feedrate (sequence of motions is continued until the final drilling depth is reached)
- Retracting to the retraction plane with G0



Explanation of the parameters

For an explanation of the parameters RTP, RFP, SDIS, DP, and DPR, see Section "Drilling, centering - CYCLE81 (Page 154)".

Interrelation of the parameters DP (or DPR), FDEP (or FDPR) and DAM

The intermediate drilling depth are calculated in the cycle on the basis of final drilling depth, first drilling depth and amount of degression as follows:

- In the first step, the depth parameterized with the first drilling depth is traversed as long as it does not exceed the total drilling depth.

- From the second drilling depth on, the drilling stroke is obtained by subtracting the amount of degression from the stroke of the last drilling depth, provided that the latter is greater than the programmed amount of degression.
- The next drilling strokes correspond to the amount of degression, as long as the remaining depth is greater than twice the amount of degression.
- The last two drilling strokes are divided and traversed equally and are therefore always greater than half of the amount of degression.
- If the value for the first drilling depth is incompatible with the total depth, the error message 61107 "First drilling depth defined incorrectly" is output and the cycle is not executed.

The FDPR parameter has the same effect in the cycle as the DPR parameter. If the values for the reference and retraction planes are identical, the first drilling depth can be defined as a relative value.

If the first drilling depth is programmed larger than the final drilling depth, the final drilling depth is never exceeded. The cycle will reduce the first drilling depth automatically as far as the final drilling depth is reached when drilling only once, and will therefore drill only once.

DTB (dwell time)

The dwell time to the final drilling depth (chip breakage) is programmed under DTB in seconds.

DTS (dwell time)

The dwell time at the starting point is only performed if VARI=1 (chip removal).

FRF (feedrate factor)

With this parameter, you can enter a reduction factor for the active feedrate which only applies to the approach to the first drilling depth in the cycle.

VARI (machining type)

If parameter VARI=0 is set, the drill retracts 1 mm after reaching each drilling depth for chip breaking. If VARI=1 (for chip removal), the drill traverses in each case to the reference plane shifted by the amount of the safety clearance.

Note

The anticipation distance is calculated internally in the cycle as follows:

- If the drilling depth is 30 mm, the value of the anticipation distance is always 0.6 mm.
 - For larger drilling depths, the formula drilling depth /50 is used (maximum value 7 mm).
-

AXN (tool axis)

By programming the drilling axis via AXN, it is possible to omit the switchover from plane G18 to G17 when the deep-hole drilling cycle is used on turning machines.

The identifiers have the following meanings:

AXN=1	First axis of the current plane
AXN=2	Second axis of the current plane
AXN=3	Third axis of the current plane

For example, to machine a center hole (in Z) in the G18 plane, you program:

G18

AXN=1

MDEP (minimum drilling depth)

You can define a minimum drilling depth for drill stroke calculations based on a degression factor. If the calculated drilling stroke becomes shorter than the minimum drilling depth, the remaining depth is machined in strokes equaling the length of the minimum drilling depth.

VRT (variable retraction value for chip breakage with VARI=0)

You can program the retraction path for chip breaking.

DTD (dwell time at final drilling depth)

The dwell time at final drilling depth can be entered in seconds or revolutions.

DIS1 (programmable limit distance for VARI=1)

The limit distance after re-insertion in the hole can be programmed.

The limit distance is calculated within the cycle as follows:

- Up to a drilling depth of 30 mm, the value is set to 0.6 mm.
- For larger drilling depths, the limit distance is the result of $(RFP + SDIS - \text{current depth}) / 50$. If this calculated value > 7 , a limit of 7 mm, maximum, is applied.

Programming example: Deep-hole drilling

This program executes the cycle CYCLE83 at the position X0. The first drill hole is drilled with a dwell time zero and machining type chip breaking. The final drilling depth and the first drilling depth are entered as absolute values. The drilling axis is the Z axis.

```

N10 G0 G54 G90 F5 S500 M4           ;Specification of technology values
N20 D1 T6 Z50                       ; Approach retraction plane
N30 G17 X0                           ; Approach drilling position
N40 CYCLE83(3.3, 0, 0, -80, 0, -10, 0, 0, 0, 1, 0) ;Call of cycle, depth parameters with absolute values
N50 M2                               ; End of program

```

12.4.6 Rigid tapping - CYCLE84

Programming

CYCLE84 (RTP, RFP, SDIS, DP, DPR, DTB, SDAC, MPIT, PIT, POSS, SST, SST1, AXN, 0, 0, VARI, DAM, VRT)

Parameters

Parameter	Data type	Description
RTP	REAL	Retraction plane (absolute)
RFP	REAL	Reference plane (absolute)
SDIS	REAL	Safety clearance (enter without sign)
DP	REAL	Final drilling depth (absolute)
DPR	REAL	Final drilling depth relative to the reference plane (enter without sign)
DTB	REAL	Dwell time at thread depth (chip breakage)
SDAC	INT	Direction of rotation after end of cycle Values: 3, 4 or 5 (for M3, M4 or M5)
MPIT	REAL	Thread lead as a thread size (signed): Range of values 3 (for M3) to 48 (for M48); the sign determines the direction of rotation in the thread
PIT	REAL	Thread lead as a value (signed) Range of values: 0.001 ... 2000.000 mm; the sign determines the direction of rotation in the thread
POSS	REAL	Spindle position for oriented spindle stop in the cycle (in degrees)
SST	REAL	Speed for tapping
SST1	REAL	Speed for retraction
AXN	INT	Tool axis (values ¹⁾ : 1 = 1st axis of the current plane; 2 = 2nd axis of the current plane; 3 = 3rd axis of the current plane)
PSYS	INT	Internal parameter; only the default value 0 is possible
PSYS	INT	Internal parameter; only the default value 0 is possible
VARI	INT	Machining type (values: 0 = Tapping in one pass; 1 = Deep-hole tapping with chip breakage; 2 = Deep-hole tapping with chip removal)
DAM	REAL	Incremental drilling depth value range: 0 ≤ Max. value
VRT	REAL	Variable retraction value for chip breakage value range: 0 ≤ Max. value

¹⁾ The definition of the 1st, 2nd, and 3rd axes depends upon the current plane selected.

Function

The tool drills at the programmed spindle speed and feedrate to the entered final thread depth.

CYCLE84 can be used to make tapped holes without compensating chuck. For tapping with compensating chuck, a separate cycle CYCLE840 is provided.

Note

CYCLE84 can be used if the spindle to be used for the boring operation is technically able to be operated in the position-controlled spindle operation.

Sequence

Position reached prior to cycle start:

The drilling position is the position in the two axes of the selected plane.

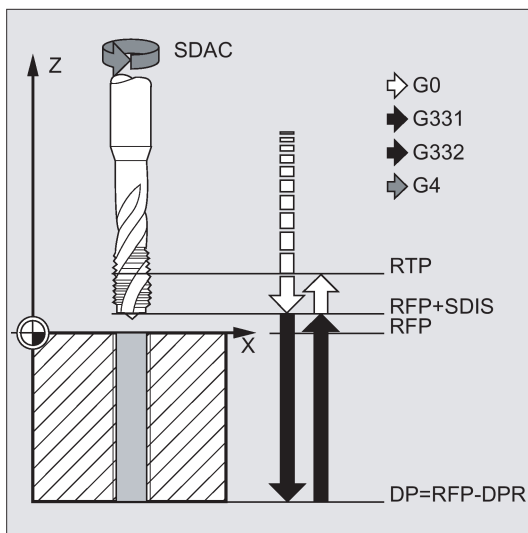
The cycle creates the following sequence of motions:

- Approaching the position of safety distance relative to the reference plane with G0
- Oriented spindle stop (value in the parameter POSS) and switching the spindle to axis mode
- Tapping to final drilling depth and speed SST
- Dwell time at thread depth (parameter DTB)
- Retraction to the reference plane brought forward by the safety clearance, speed SST1 and direction reversal
- Retraction to the retraction plane with G0; spindle mode is reinitiated by reprogramming the spindle speed active before the cycle was called and the direction of rotation programmed under SDAC

Explanation of the parameters

For more information about the parameters RTP, RFP, SDIS, DP, DPR, see Section "Drilling, centering - CYCLE81 (Page 154)".

See the following parameters for CYCLE84:



DTB (dwell time)

The dwell time must be programmed in seconds. When tapping blind holes, it is recommended that you omit the dwell time.

SDAC (direction of rotation after end of cycle)

Under SDAC, the direction of rotation after end of cycle is programmed.

For tapping, the direction is changed automatically by the cycle.

MPIT and PIT (thread lead as a thread size and as a value)

The value for the thread lead can be defined either as the thread size (for metric threads between M3 and M48 only) or as a value (distance from one thread turn to the next as a numerical value). Any parameters not required are omitted in the call or assigned the value zero.

RH or LH threads are defined by the sign of the lead parameters:

- Positive value → right (same as M3)
- Negative value → left (same as M4)

If the two lead parameters have conflicting values, alarm 61001 "Thread lead wrong" is generated by the cycle and cycle execution is aborted.

POSS (spindle position)

Before tapping, the spindle is stopped with orientation in the cycle by using the command SPOS and switched to position control.

The spindle position for this spindle stop is programmed under POSS.

SST (speed)

Parameter SST contains the spindle speed for the tapping block with G331.

SST1 (retraction speed)

The speed for retraction from the tapped hole is programmed under SST1.

If this parameter is assigned the value zero, retraction is carried out at the speed programmed under SST.

AXN (tool axis)

By programming the drilling axis via AXN, it is possible to omit the switchover from plane G18 to G17 when the deep-hole drilling cycle is used on turning machines.

The identifiers have the following meanings:

Plane	Designation	Infeed direction
X/Y	G17	AXN=1: 1st axis of the current plane is X AXN=2: 2nd axis of the current plane is Y ¹⁾ AXN=3: 3rd axis of the current plane is Z
Z/X	G18	AXN=1: 1st axis of the current plane is Z AXN=2: 2nd axis of the current plane is X AXN=3: 3rd axis of the current plane is Y ¹⁾
Y/Z	G19	AXN=1: 1st axis of the current plane is Y ¹⁾ AXN=2: 2nd axis of the current plane is Z AXN=3: 3rd axis of the current plane is X

¹⁾ If Y-axis is present.

For example, to machine a center hole (in Z) in the G17 plane, you can program:

```
G17
AXN=3
```

Deep-hole tapping: VARI, DAM, VRT

With the VARI parameter, it is possible to distinguish between simple tapping (VARI = 0) and deep-hole tapping (VARI ≠ 0).

In conjunction with deep-hole tapping, it is possible to choose between chip breaking (retraction by variable distance from current drilling depth, parameter VRT, VARI = 1) and chip removal (withdrawal from reference plane VARI = 2). These functions work analogously to the normal deep-hole drilling cycle CYCLE83.

The incremental drilling depth for one pass is specified via parameter DAM. The cycle internally calculates the intermediate depth as follows:

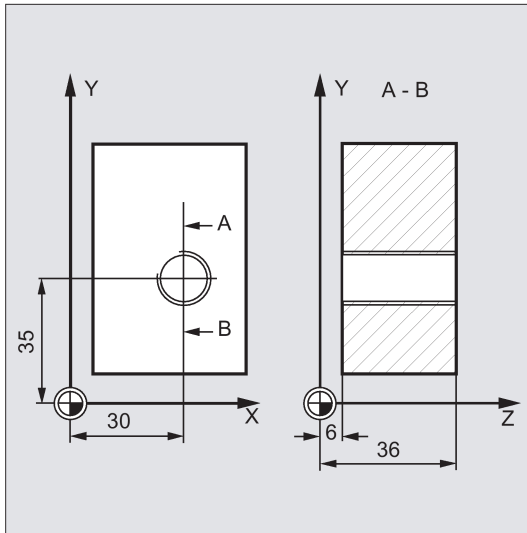
- The programmed incremental drilling depth is executed in each step until the rest up to the final drilling depth is less than (<) 2 x DAM
- The remaining drilling depth is halved and executed in two steps. Thus, the minimum drilling depth is not smaller than DAM / 2.

Note

The direction of rotation when tapping in the cycle is always reversed automatically.

Programming example 1: Rigid tapping

A thread is tapped without compensating chuck at position X30 Y35 in the XY plane; the tapping axis is the Z axis. No dwell time is programmed; the depth is programmed as a relative value. The parameters for the direction of rotation and for the lead must be assigned values. A metric thread M5 is tapped.



```
N10 G0 G90 T11 D1
N20 G17 X30 Y35 Z40
N30 CYCLE84(40, 36, 2, , 30, , 3, 5, , 90, 200, 500, 3, 0,
0,0, ,0.00000)
N40 M02
```

```
; Specification of technology values
; Approach drilling position
Cycle call; parameter PIT has been omitted; no value is entered for the absolute depth or the dwell time; spindle stop at 90 degrees; speed for tapping is 200, speed for retraction is 500
; End of program
```

Programming example 2

Proceed through the following steps:



1. Select the program editing operating area.



2. Open the vertical softkey bar for available drilling cycles.



3. Press this softkey to open the lower-level softkey bar.

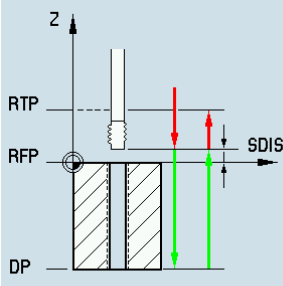


4. Press this softkey to open the window for CYCLE84. Parameterize the cycle as desired.

15:16:48
2016/01/22

NC:\MPF\1.MPF 1

CYCLE84 Retract plane, absolute



RTP	5.00000
RFP	
SDIS	2.00000
DP	-18.00000
DPR	0.00000
DTB	0.50000
SDAC	3
MPIT	12.00000
PIT	
POSS	0.00000
SST	200.00000
SST1	200.00000
AXN	3
PSYS	0
PSYS	0
UARI	0
DAM	
URT	1.00000

Modal call
Cancel
OK



5. Confirm your settings with this softkey. The cycle is then automatically transferred to the program editor as a separate block.

NC:\MPF\1.MPF 2

CYCLE84(5, ,2, -18, 0, 0.5, 3, 12, ,0, 200, 200, 3, 0, 0, 0, ,1)

==eof==

12.4.7 Tapping with compensating chuck - CYCLE840

Programming

CYCLE840 (RTP, RFP, SDIS, DP, DPR, DTB, SDR, SDAC, ENC, MPIT, PIT, AXN)

Parameters

Parameter	Data type	Description
RTP	REAL	Retraction plane (absolute)
RFP	REAL	Reference plane (absolute)
SDIS	REAL	Safety clearance (enter without sign)
DP	REAL	Final drilling depth (absolute)
DPR	REAL	Final drilling depth relative to the reference plane (enter without sign)
DTB	REAL	Dwell time at thread depth (chip breakage)
SDR	INT	Direction of rotation for retraction Values: 0 (automatic direction reversal), 3 or 4 (for M3 or M4)
SDAC	INT	Direction of rotation after end of cycle Values: 3, 4 or 5 (for M3, M4 or M5)
ENC	INT	Tapping with/without encoder Values: 0 = with encoder, 1 = without encoder
MPIT	REAL	Thread lead as a thread size (signed): Range of values: 0 (for M0) to 48 (for M48)
PST	REAL	Thread lead as a value (signed) Range of values: 0.001 ... 2000.000 mm

Parameter	Data type	Description
AXN	INT	Tool axis (values ¹⁾ : 1 = 1st axis of the current plane; 2 = 2nd axis of the current plane; 3 = 3rd axis of the current plane)

¹⁾ The definition of the 1st, 2nd, and 3rd axes depends upon the current plane selected.

Function

The tool drills at the programmed spindle speed and feedrate to the entered final thread depth.

You use this cycle to perform tapping with the compensating chuck:

- without encoder
- with encoder

Sequence

Tapping with compensating chuck without encoder

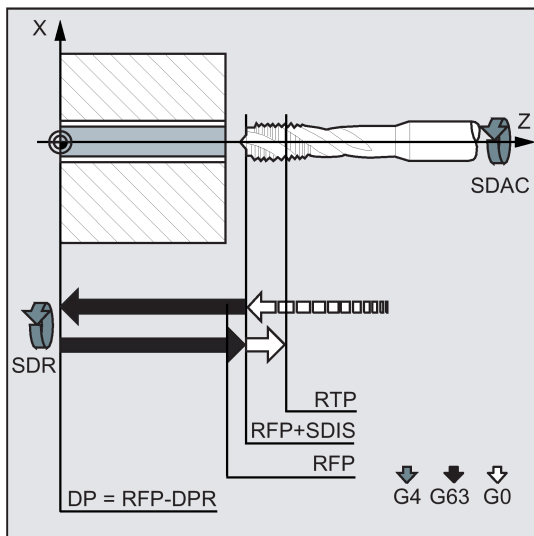
Position reached prior to cycle start:

The drilling position is the position in the two axes of the selected plane.

The cycle creates the sequence of motions below:

- Approaching the position of safety distance relative to the reference plane with G0
- Tapping to the final drilling depth
- Dwell time at tapping depth (parameter DTB)
- Retraction to the reference plane brought forward by the safety clearance
- Retraction to the retraction plane with G0

See the following parameters for CYCLE840 without encoder:



Sequence

Tapping with compensating chuck with encoder

Position reached prior to cycle start:

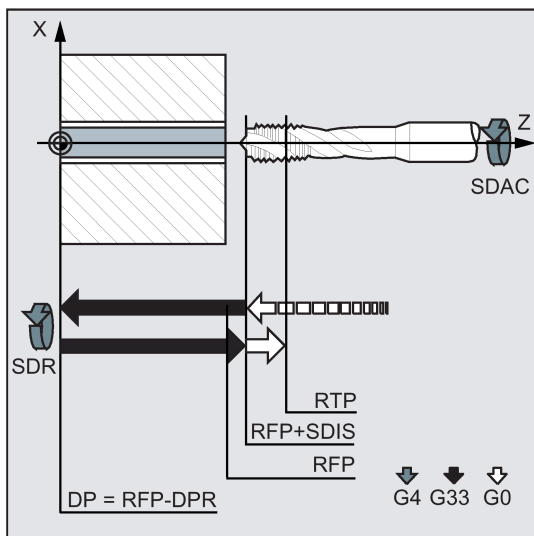
The drilling position is the position in the two axes of the selected plane.

The cycle creates the sequence of motions below:

- Approach of the reference plane brought forward by the safety clearance by using G0
- Tapping to the final drilling depth

- Dwell time at thread depth (parameter DTB)
- Retraction to the reference plane brought forward by the safety clearance
- Retraction to the retraction plane with G0

See the following parameters for CYCLE840 with encoder:



Explanation of the parameters

For an explanation of the parameters RTP, RFP, SDIS, DP, and DPR, see Section "Drilling, centering - CYCLE81 (Page 154)".

DTB (dwell time)

The dwell time must be programmed in seconds. It is only effective in tapping without encoder.

SDR (direction of rotation for retraction)

SDR=0 must be set if the spindle direction is to reverse automatically.

If the machine data is defined such that no encoder is set (in this case, machine data MD30200 \$MA_NUM_ENCS is 0), the parameter must be assigned the value 3 or 4 for the direction of rotation; otherwise, alarm 61202 "No spindle direction programmed" is output and the cycle is aborted.

SDAC (direction of rotation)

Because the cycle can also be called modally (see Section "Graphical cycle programming in the program editor (Page 150)"), it requires a direction of rotation for tapping further threaded holes. This is programmed in parameter SDAC and corresponds to the direction of rotation programmed before the first call in the higher-level program. If SDR=0, the value assigned to SDAC has no meaning in the cycle and can be omitted in the parameterization.

ENC (tapping)

If tapping is to be performed without encoder although an encoder exists, parameter ENC must be assigned value 1.

If, however, no encoder is installed and the parameter is assigned the value 0, it is ignored in the cycle.

MPIT and PIT (thread lead as a thread size and as a value)

The parameter for the lead is only relevant if tapping is performed with encoder. The cycle calculates the feedrate from the spindle speed and the lead.

The value for the thread lead can be defined either as the thread size (for metric threads between M3 and M48 only) or as a value (distance from one thread turn to the next as a numerical value). The parameter not required in each case is omitted in the call or is assigned the value zero.

If the two lead parameters have conflicting values, alarm 61001 "Thread lead wrong" is generated by the cycle and cycle execution is interrupted.

Note

Depending on the settings in machine data MD30200 \$MA_NUM_ENCS, the cycle selects whether tapping is to be performed with or without encoder.

The direction of rotation for the spindle must be programmed with M3 or M4.

In thread blocks with G63, the values of the feedrate override switch and spindle speed override switch are frozen to 100%.

A longer compensating chuck is usually required for tapping without encoder.

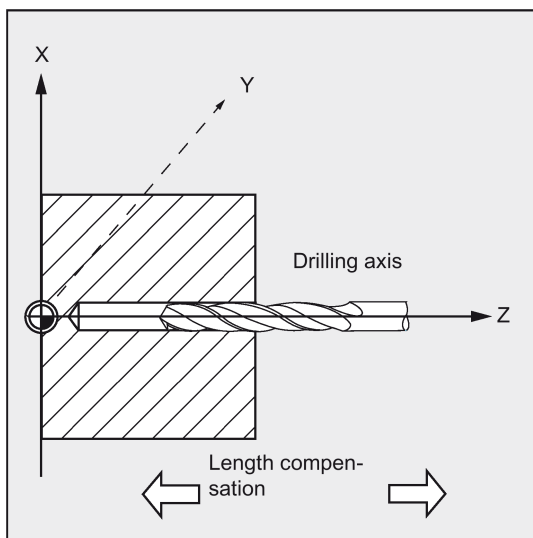
AXN (tool axis)

The following figure presents the options for the drilling axes to be selected.

With G18:

- AXN=1 ;Corresponds to Z
- AXN=2 ;Corresponds to X
- AXN=3 ;Corresponds to Y (if Y axis is present)

See the following illustration for drilling axis of G18:



Using AXN (number of the drilling axis) to program the drilling axis enables the drilling axis to be directly programmed.

Plane	Designation	Infeed direction
X/Y	G17	AXN=1: 1st axis of the current plane is X AXN=2: 2nd axis of the current plane is Y ¹⁾ AXN=3: 3rd axis of the current plane is Z
Z/X	G18	AXN=1: 1st axis of the current plane is Z AXN=2: 2nd axis of the current plane is X AXN=3: 3rd axis of the current plane is Y ¹⁾
Y/Z	G19	AXN=1: 1st axis of the current plane is Y ¹⁾ AXN=2: 2nd axis of the current plane is Z AXN=3: 3rd axis of the current plane is X

¹⁾ If Y-axis is present.

For example, to machine a center hole (in Z) in the G17 plane, you program:

G17

AXN=3

Programming example: Tapping without encoder

Tapping is carried out without encoder at position X0; the drilling axis is the Z axis. The parameters SDR and SDAC for the direction of rotation must be assigned; parameter ENC is assigned the value 1, the value for the depth is the absolute value. Lead parameter PIT can be omitted. A compensating chuck is used in machining.

```
N10 G90 G0 G54 D1 T6 S500 M3 ; Specification of technology values
N20 G17 X0 Z60 ; Approach drilling position
N30 G1 F200 ; Setting the path feedrate
N40 CYCLE840(3, 0, , -15, 0, 1, 4, 3, 1, , , 3) ; Cycle call; dwell time 1 s; direction
; of rotation for retraction M4; direction
; of rotation after cycle M3; no safety
; clearance
; The MPIT and PIT parameters have been
; omitted.
N50 M2 ; End of program
```

Programming example: Tapping with encoder

This program is used for tapping with encoder at position X0. The drilling axis is the Z axis. The lead parameter must be defined, automatic reversal of the direction of rotation is programmed. A compensating chuck is used in machining.

```
N10 G90 G0 G54 D1 T6 S500 M3 ; Specification of technology values
N20 G17 X0 Z60 ; Approach drilling position
N30 G1 F200 ; Setting the path feedrate
N40 CYCLE840(3, 0, , -15, 0, 0, , , 0, 3.5, , 3) ; Cycle call without safety clearance
N50 M2 ; End of program
```

12.4.8 Reaming1 - CYCLE85

Programming

CYCLE85 (RTP, RFP, SDIS, DP, DPR, DTB, FFR, RFF)

Parameters

Parameter	Data type	Description
RTP	REAL	Retraction plane (absolute)
RFP	REAL	Reference plane (absolute)
SDIS	REAL	Safety clearance (enter without sign)
DP	REAL	Final drilling depth (absolute)
DPR	REAL	Final drilling depth relative to the reference plane (enter without sign)
DTB	REAL	Dwell time at final drilling depth (chip breakage)
FFR	REAL	Feedrate
RFF	REAL	Retraction feedrate

Function

The tool drills at the programmed spindle speed and feedrate velocity to the entered final drilling depth.

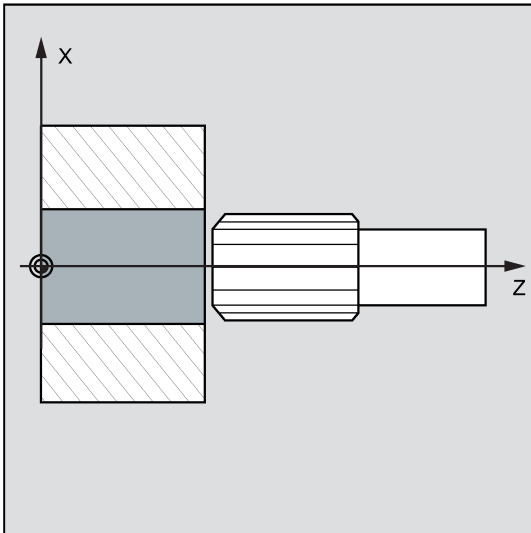
The inward and outward movement is performed at the feedrate assigned to FFR and RFF respectively.

Sequence

Position reached prior to cycle start:

The drilling position is the position in the two axes of the selected plane.

See the following illustration for sequence of operations:



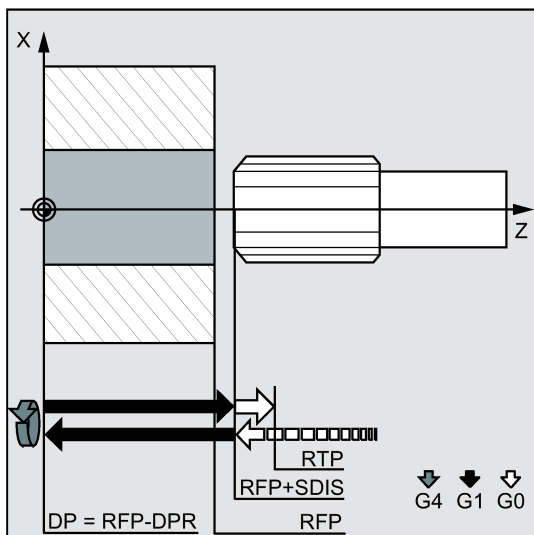
The cycle creates the following sequence of motions:

- Approaching the position of safety distance relative to the reference plane with G0
- Traversing to the final drilling depth with G1 and at the feedrate programmed under the parameter FFR
- Dwell time at final drilling depth
- Retracting to the position of safety distance relative to the reference plane with G1 and the retraction feedrate defined by the parameter RFF
- Retracting to the retraction plane with G0

Explanation of the parameters

For an explanation of the parameters RTP, RFP, SDIS, DP, and DPR, see Section "Drilling, centering - CYCLE81 (Page 154)".

See the following parameters for CYCLE85:



DTB (dwell time)

The dwell time to the final drilling depth is programmed under DTB in seconds.

FFR (feedrate)

The feedrate value programmed under FFR is active in drilling.

RFF (retraction feedrate)

The feedrate value programmed under RFF is active when retracting from the hole to the reference plane + safety clearance.

Programming example: First boring pass

CYCLE85 is called at Z70 X0. The drilling axis is the Z axis. The value for the final drilling depth in the cycle call is programmed as a relative value; no dwell time is programmed. Z0 is on the surface of the workpiece.

```
N10 G90 G0 S300 M3
N20 T3 G17 G54 Z70 X0 ; Approach drilling position
N30 CYCLE85(10, 2, 2, , 25, , 300, 450) ; Cycle call, no dwell time programmed
N40 M2 ; End of program
```

12.4.9 Boring - CYCLE86

Programming

CYCLE86 (RTP, RFP, SDIS, DP, DPR, DTB, SDIR, RPA, 0, RPAP, POSS)

Parameters

Parameter	Data type	Description
RTP	REAL	Retraction plane (absolute)
RFP	REAL	Reference plane (absolute)
SDIS	REAL	Safety clearance (enter without sign)
DP	REAL	Final drilling depth (absolute)
DPR	REAL	Final drilling depth relative to the reference plane (enter without sign)
DTB	REAL	Dwell time at final drilling depth (chip breakage)
SDIR	INT	Direction of rotation Values: 3 (for M3), 4 (for M4)
RPA	INT	Retraction path along the 1st axis of the plane (incremental, enter with sign)
PSYS	INT	Internal parameter; only the default value 0 is possible
RPAP	REAL	Retraction path in drilling axis (incremental, enter with sign)
POSS	REAL	Spindle position for oriented spindle stop in the cycle (in degrees)

Function

The cycle supports boring of holes with a boring bar.

The tool drills at the programmed spindle speed and feedrate velocity up to the entered drilling depth.

With boring 2, oriented spindle stop is activated once the drilling depth has been reached. Then, the programmed retraction positions are approached in rapid traverse, and from there the retraction plane is approached.

CYCLE86 can be applied on a turning machine only with TRANSMIT in the G17 plane and with a driven tool. For more information about TRANSMIT function, see Section "Face milling on turned parts (TRANSMIT) (Page 140)".

Sequence

Position reached prior to cycle start:

The drilling position is the position in the two axes of the selected plane.

The cycle creates the following sequence of motions:

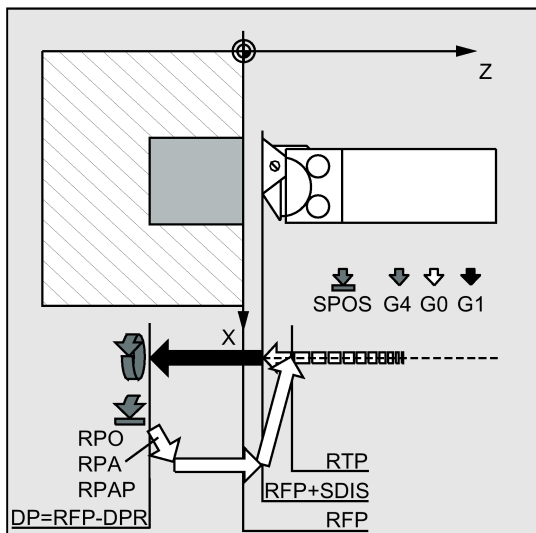
- Approaching the position of safety distance relative to the reference plane with G0
- Traversing to final drilling depth with G1 and the feedrate programmed prior to the cycle call
- Dwell time to final drilling depth
- Oriented spindle stop at the spindle position programmed under POSS
- Traverse retraction path in up to three axes with G0
- Retracting in the boring axis to the position of safety distance relative to the reference plane with G0

- Retracting to the retraction plane with G0 (initial drilling position in both axes of the plane)

Explanation of the parameters

For the explanation of the parameters RTP, RFP, SDIS, DP, and DPR, see Section "Drilling, centering - CYCLE81 (Page 154)".

See the following parameters for CYCLE86:



DTB (dwell time)

The dwell time to the final drilling depth (chip breakage) is programmed under DTB in seconds.

SDIR (direction of rotation)

With this parameter, you determine the direction of rotation with which boring is performed in the cycle. If values other than 3 or 4 (M3/M4) are generated, alarm 61102 "No spindle direction programmed" is generated and the cycle is not executed.

RPA (retraction path along the 1st axis)

Use this parameter to define a retraction motion along the 1st axis (abscissa), which is performed after reaching the final drilling depth and oriented spindle stop.

RPAP (retraction path along the boring axis)

You use this parameter to define a retraction movement along the drilling axis, which is executed after the final drilling axis has been reached and oriented spindle stop has been performed.

POSS (spindle position)

Use POSS to program the spindle position for the oriented spindle stop in degrees, which is performed after the final drilling depth has been reached.

Note

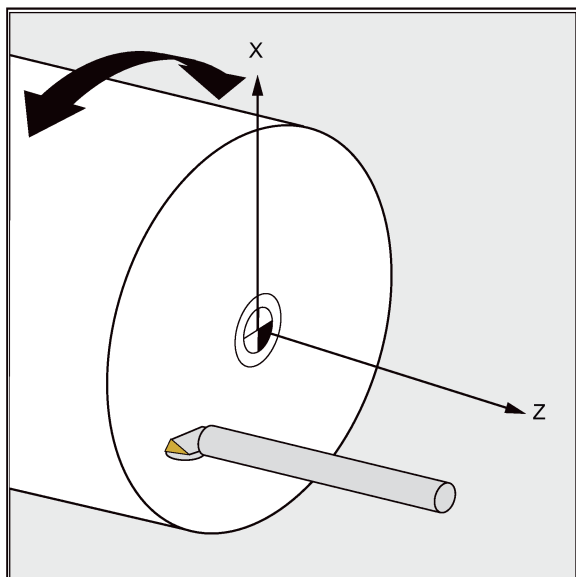
It is possible to stop the active spindle with orientation. The angular value is programmed using a transfer parameter.

CYCLE86 can be used only if the spindle to be used for the boring operation is technically able to go into position-controlled spindle operation.

Programming example: Second boring pass

CYCLE86 will be used to drill on the end face in the X-Y plane at position X20 Y20. The drilling axis is the Z axis. The final drilling depth is programmed as an absolute value; no safety clearance is specified. The dwell time at the final drilling depth is 2 s. The top edge of the workpiece is positioned at Z10. In the cycle, the spindle is to rotate with M3 and to stop at 45 degrees.

See the following example of second boring:



```

N10 G0 G90 X0 Z100 SPOS=0          ; Approach start position
N15 SETMS(2)                        ; Master spindle is now the milling spindle
N20 TRANSMIT                        ; Activate TRANSMIT function
N35 T10 D1                          ; Load tool
N50 G17 G0 G90 X20 Y20              ; Drilling position
N60 S800 M3 F500
N70 CYCLE86(112, 110, , 77, 0, 2, 3, -1, 0, 1, 45) ; Cycle call with absolute drilling depth
N80 G0 Z100
N90 TRAFOOF                          ; Switch off TRANSMIT
N95 SETMS                          ; Master spindle is now the main spindle again
N200 M2                             ; End of program

```

12.5 Turning cycles

12.5.1 Requirements

Call and return conditions

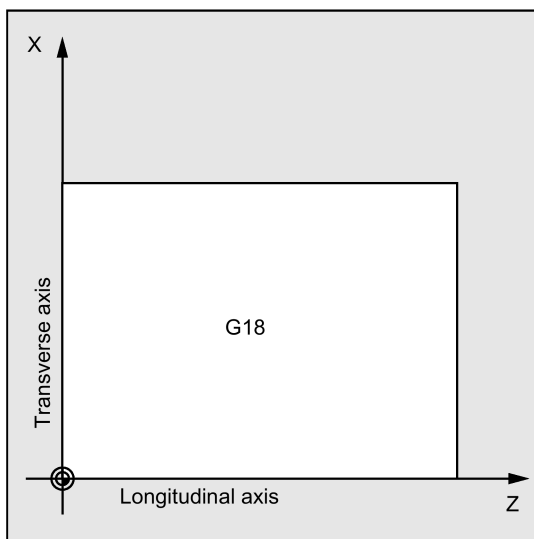
The G functions effective prior to the cycle call remain active beyond the cycle.

Plane definition

The machining plane must be defined prior to the cycle call. With turning, it is usually the G18 (ZX plane). The two axes of the current plane in turning are hereinafter referred to as the longitudinal axis (first axis of this plane) and transverse axis (second axis of this plane).

In the turning cycles, with diameter programming active, the second axis is taken into account as the transverse axis in all cases.

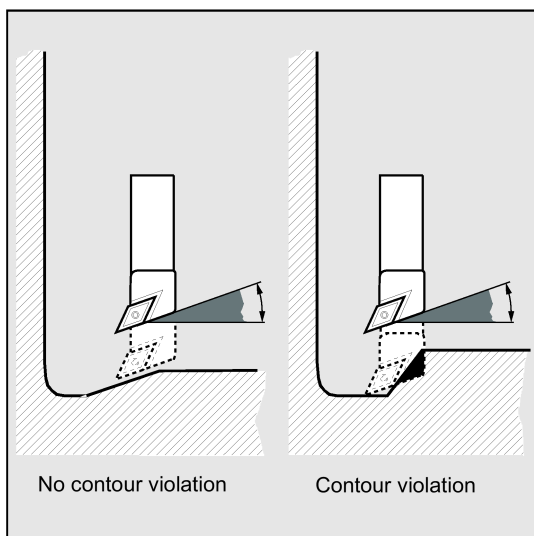
See the following illustration for G18:



Contour monitoring relative to the clearance angle of the tool

Certain turning cycles in which traversing motions with relief cutting are generated monitor the clearance angle of the active tool for a possible contour violation. This angle is entered in the tool compensation as a value (in the D offset under the parameter DP24). A value between 1 and 90 degrees (0=no monitoring) without sign must be specified for the angle.

Longitudinal contour monitoring:



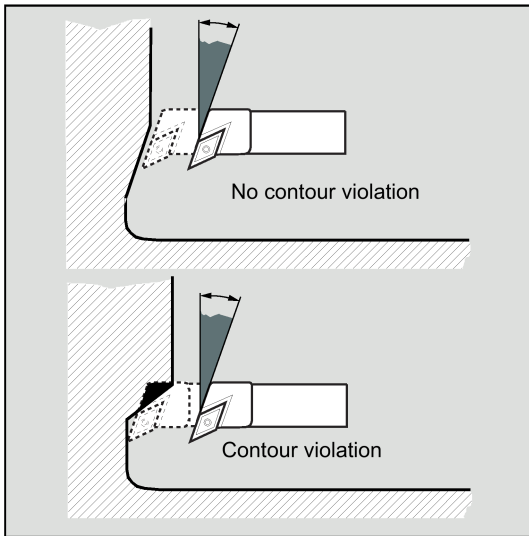
When entering the tool clearance angle, note that this depends on the machining type 'longitudinal' or 'face'. If you want to use one tool for longitudinal and face machining, two tool compensations must be used in the case of different tool clearance angles.

The cycle will check whether the programmed contour can be machined with the selected tool.

If the machining is not possible using this tool, the cycle will abort and an error message is output (in stock removal); or alternatively, the contour is continued to be machined and a message is output (with undercut cycles). In this case, the contour is determined by the cutting edge geometry.

If the tool clearance angle is specified with zero in the tool compensation, this monitoring will not be performed. For more information about the reactions, see the individual cycles.

Planar contour monitoring:



12.5.2 Cutoff - CYCLE92

Programming

CYCLE92 (SPD, SPL, DIAG1, DIAG2, RC, SDIS, SV1, SV2, SDAC, FF1, FF2, SS2, 0, VARI, 1, 0, AMODE)

Parameters

Parameter	Data type	Description
SPD	REAL	Starting point along facing axis (absolute, always diameter)
SPL	REAL	Starting point in the longitudinal axis (absolute)
DIAG1	REAL	Depth for speed reduction $\varnothing$ (absolute)
DIAG2	REAL	Final depth $\varnothing$ (absolute)
RC	REAL	Chamfer width or rounding radius
SDIS	REAL	Safety clearance (to be added to the reference point; enter without sign)
SV1	REAL	Constant cutting rate V
SV2	REAL	Maximum speed at constant cutting speed
SDAC	INT	Direction of spindle rotation (values: 3 = M3; 4 = M4)
FF1	REAL	Feed to depth for speed reduction
FF2	REAL	Reduced feed up to final depth, mm/rev
SS2	REAL	Reduced speed up to final depth
PSYS	INT	Internal parameter; only the default value 0 is possible
VARI	INT	Machining type (values: 0 = Retraction to SPD + SDIS; 1 = No retraction)
PSYS	INT	Internal parameter; only the default value 1 is possible
PSYS	INT	Internal parameter; only the default value 0 is possible
AMODE	INT	Alternate mode: radius or chamfer (values: 10000 = radius; 11000 = chamfer)

Function

CYCLE92 is used to cut off balanced parts (for example, screws, bolts, or pipes) dynamically.

You can program a chamfer or rounding on the edge of the machined part. You can machine at a constant cutting rate V or speed S up to a depth DIAG1, from which point the workpiece is machined at a constant speed. As of depth DIAG2, you can also program a reduced feedrate FF2 or a reduced speed SS2, in order to adapt the velocity to the smaller diameter.

Use parameter DIAG2 to enter the final depth that you wish to reach with the cut-off. With pipes, for example, you do not need to cut-off until you reach the center; cutting off slightly more than the wall thickness of the pipe is sufficient.

Sequence

1. The tool first moves to the starting point calculated internally in the cycle at rapid traverse.
2. The chamfer or radius is machined at the machining feedrate.
3. Cut-off down to depth DIAG1 is performed at the machining feedrate.
4. Cut-off is continued down to depth DIAG2 at reduced feedrate FF2 and reduced spindle speed SS2.
5. The tool moves back to the safety distance at rapid traverse.

Programming example 1

```

N10 G0 G90 Z30 X100 T5 D1 S1000 M3           ;Starting point before the beginning of
                                                the cycle
N20 G95 F0.2                                   ; Specification of technology values
N30 CYCLE92(60, -30, 40, -2, 2, 1, 800, 200,3,1,1,300, 0, ; Cycle call
0, 1, 0, 11000)
N40 G0 G90 X100 Z30                           ; Next position
N50 M02                                         ; End of program

```

Programming example 2

The cycle can be found and parameterized from the main screen of the turning cycles.



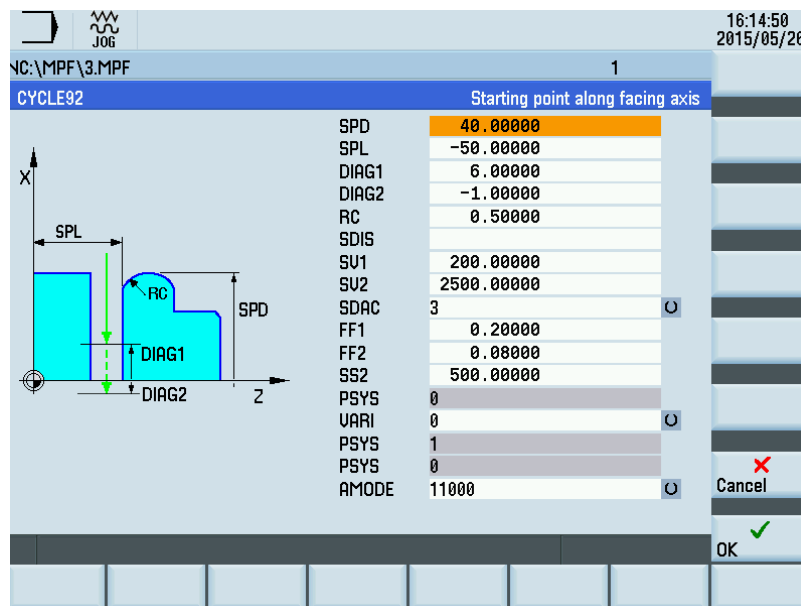
1. Select the program editing operating area.



2. Open the vertical softkey bar for available turning cycles.



3. Press this softkey to open the window for CYCLE92. Parameterize the cycle as desired.





4. Confirm your settings with this softkey. The cycle is then automatically transferred to the program editor as a separate block.

```
NC:\MPF\1.MPF 2
CYCLE92( 40, -50, 6, -1, 0.5, , 200, 2500, 3, 0.2, 0.08, 500, 0, 0,
1, 0, 11000)¶
==eof==
```

12.5.3 Groove - CYCLE93

Programming

CYCLE93 (SPD, SPL, WIDG, DIAG, STA1, ANG1, ANG2, RCO1, RCO2, RCI1, RCI2, FAL1, FAL2, IDEP, DTB, VARI, _VRT)

Parameters

Parameter	Data type	Description
SPD	REAL	Starting point along facing axis
SPL	REAL	Starting point along longitudinal axis
WIDG	REAL	Groove width (enter without sign)
DIAG	REAL	Groove depth (enter without sign)
STA1	REAL	Longitudinal: 0<=STA<=180, face: STA=90
ANG1	REAL	Flank angle 1: on the side of the groove determined by the starting point (enter without sign) Range of values: 0<=ANG1<89.999 degrees
ANG2	REAL	Flank angle 2: on the other side (enter without sign) Range of values: 0<=ANG2<89.999
RCO1	REAL	Radius/chamfer 1, externally: on the side determined by the starting point
RCO2	REAL	Radius/chamfer 2, externally
RCI1	REAL	Radius/chamfer 1, internally: on the starting point side
RCI2	REAL	Radius/chamfer 2, internally
FAL1	REAL	Finishing allowance at the recess base
FAL2	REAL	Finishing allowance at the flanks
IDEP	REAL	Infeed depth (enter without sign)
DTB	REAL	Dwell time at recess base
VARI	INT	Machining type Range of values: 1...8 and 11...18
_VRT	REAL	Variable retraction path for grooving, incremental (enter without sign)

Function

The grooving cycle can be used to carry out symmetrical and asymmetrical grooves for longitudinal and face machining at any straight contour elements. External and internal grooves can be produced.

Sequence

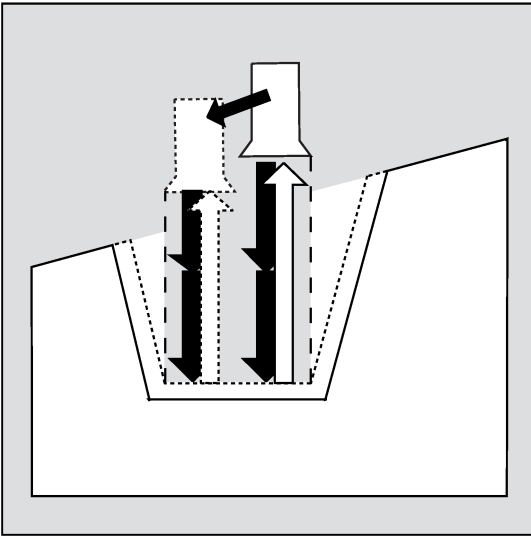
The infeed in the depth (towards the groove base) and in the width (from groove to groove) are calculated in the cycle internally and distributed equally with the maximum possible value.

When grooving at oblique faces, the tool will traverse from one groove to the next on the shortest path, that is, parallel to the cone at which the groove is machined. During this process, a safety clearance to the contour is calculated by the cycle internally.

Step 1

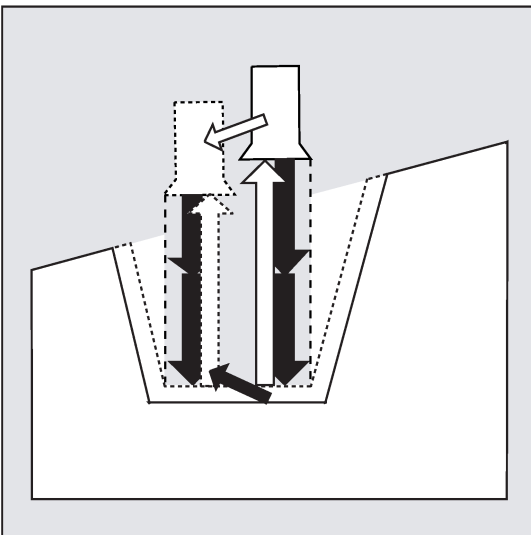
Paraxial roughing down to the base of the groove in single infeed steps.

After each infeed, the tool is retracted for chip breaking.



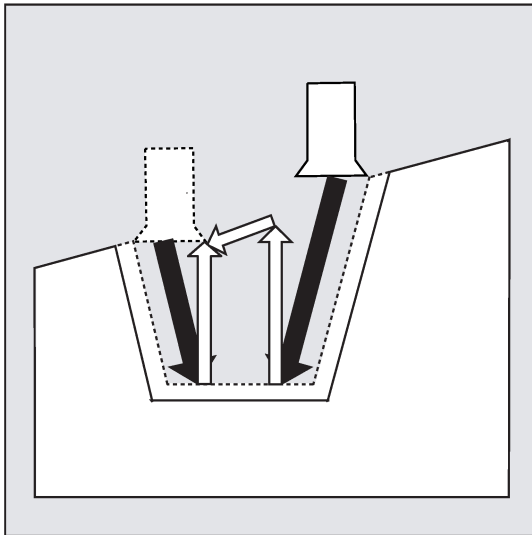
Step 2

The groove is machined vertically to the infeed direction in one or several steps whereby each step, in turn, is divided according to the infeed depth. From the second cut along the groove width onwards, the tool will retract by 1 mm before each retraction.



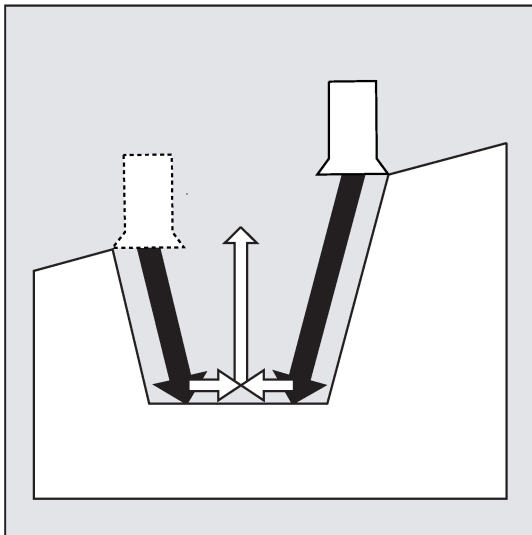
Step 3

Machining of the flanks in one step if angles are programmed under ANG1 or ANG2. Infeed along the groove width is carried out in several steps if the flank width is larger.



Step 4

Stock removal of the finishing allowance parallel to the contour from the edge to the groove center. During this operation, the tool radius compensation is selected and deselected by the cycle automatically.



Explanation of parameters

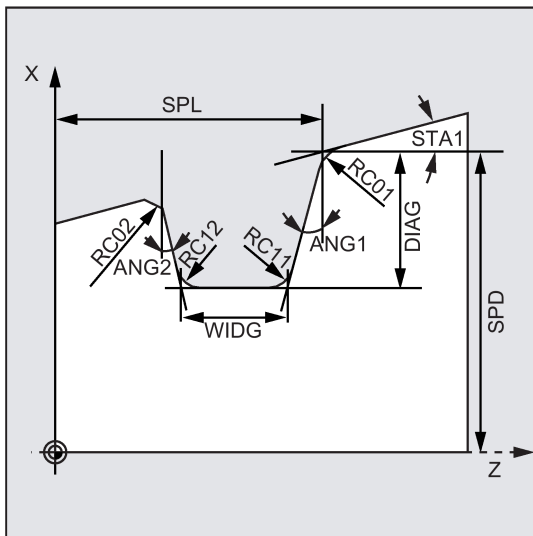
SPD and SPL (starting point)

These coordinates can be used to define the starting point of a groove starting from which the form is calculated in the cycle. The cycle determines its own starting point. For an external groove, movement begins in the direction of the longitudinal axis, for an internal groove in the direction of the facing axis.

Grooves at bent contour elements can be realized differently. Depending on the form and radius of the bend, either a paraxial straight line can be laid over the maximum of the bend or a tangential oblique line can be created in a point of the edge points of the groove.

Radii and chamfers at the groove edge make sense with bent contours only if the appropriate edge point is on the straight line specified for the cycle.

See the following illustration for the parameters for CYCLE93:

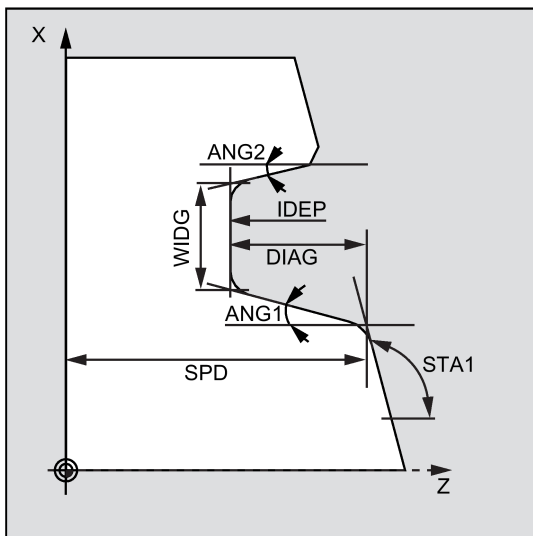


WIDG and DIAG (groove width and groove depth)

The parameters groove width (WIDG) and groove depth (DIAG) are used to define the form of the groove. In its calculation, the cycle always assumes the point programmed under SPD and SPL.

If the groove width is larger than that of the active tool, the width is removed in several steps. When doing so, the whole width is distributed by the cycle equally. The maximum infeed is 95% of the tool width after deduction of the cutting edge radii. This provides a cutting overlap.

If the programmed groove width is smaller than the real tool width, the error message 61602 "Tool width defined incorrectly" and machining is aborted. The alarm will also appear if a cutting edge width equal to zero is detected in the cycle.



STA1 (angle)

Use the STA1 parameter to program the angle of the oblique line at which the groove is to be machined. The angle can assume values between 0 and 180 degrees and always refers to the longitudinal axis.

Note

For transverse grooving, STA1 is usually 90 degrees.

ANG1 and ANG2 (flank angle)

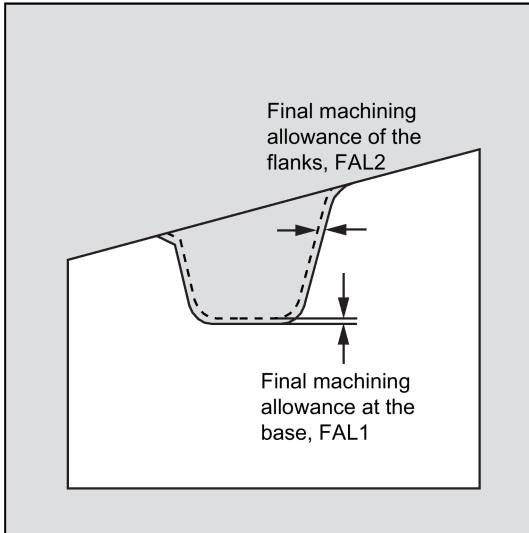
Asymmetric grooves can be described by flank angles specified separately. The angles can assume values between 0 and 89.999 degrees.

RCO1, RCO2 and RCI1, RCI2 (radius/chamfer)

The form of the groove can be modified by entering radii/chamfers at the margin or at the base. **It is imperative to enter the radii with positive sign, and the chamfers with negative sign.**

FAL1 and FAL2 (finishing allowance)

It is possible to program separate finishing allowances for groove base and flanks. During roughing, stock removal is carried out up to these finishing allowances. The same tool is then used to machine a contour-parallel cut along the final contour.



IDEF (infeed depth)

You can program an infeed depth to divide the paraxial grooving into several depth infeeds. After each infeed, the tool is retracted by 1 mm for chip breaking.

The IDEF parameter must be programmed in all cases.

DTB (dwell time)

The dwell time at the groove base should be selected such that at least one spindle revolution is carried out. It is programmed in seconds.

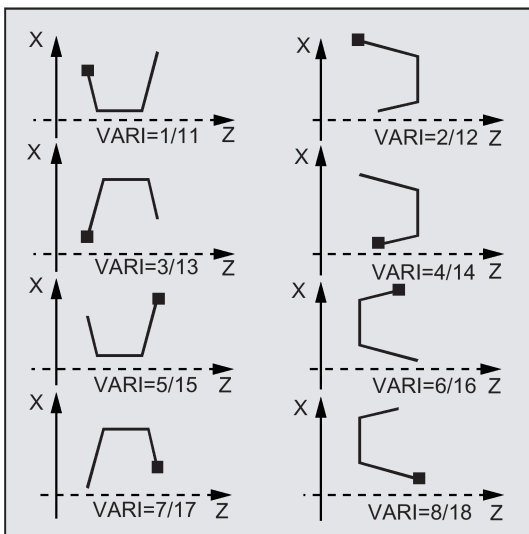
VARI (machining type)

The machining type of the groove is defined with the units digit of the VARI parameter. It can assume the values indicated in the illustration.

The tens digit of parameter VARI determines how the chamfers are taken into account.

VARI 1...8: Chamfers are calculated as CHF

VARI 11...18: Chamfers are calculated as CHR



If the parameter has a different value, the cycle will abort with alarm 61002 "Machining type defined incorrectly".

The cycle carries out a contour monitoring such that a reasonable groove contour results. This is not the case if the radii/chamfers come into contact or intersect at the groove base or if you try to carry out a face grooving operation at a contour segment located parallel to the longitudinal axis. In such cases, the cycle will abort with alarm 61603 "Groove form defined incorrectly".

_VRT (variable retraction path)

The retraction path can be programmed in the _VRT parameter on the basis of the outside or inside diameter of the groove.

For VRT=0 (parameter not programmed), the tool is retracted by 1 mm. The retraction path is always measured according to the programmed system of units, inch or metric.

Note

Before calling the grooving cycle, a double-edged tool must be enabled. The offset values for the two cutting edges must be stored in two successive D numbers of the tool whereby the first of which must be activated prior to the first cycle call. The cycle itself defines for which machining step it will use which of the two tool compensation values and will also enable them automatically. After completion of the cycle, the tool compensation number programmed prior to the cycle call is active again. If no D number is programmed for a tool compensation when the cycle is called, the execution of the cycle is aborted with the alarm 61000 "No tool compensation active".

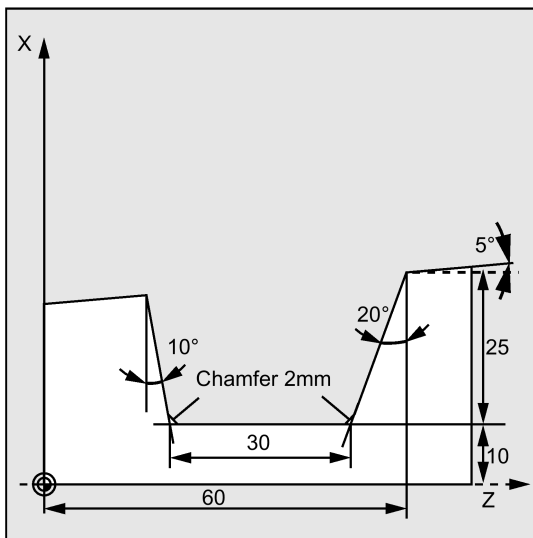
Programming example 1: Plunge-cutting

This program is used to produce a groove externally at an oblique line in the longitudinal direction.

The starting point is on the right-hand side at X70 Z60.

The cycle will use the grooving tool T5 (tip width: 4 mm). The cutting tool must be defined accordingly.

See the following illustration for the example for plunge-cutting:



Programming example 2



1. Select the program editing operating area.



2. Open the vertical softkey bar for available turning cycles.



3. Press this softkey to open the window for CYCLE93. Parameterize the cycle as desired.

16:34:54
2015/07/20

NC:\MPF\1.MPF 1

CYCLE93 Starting point along facing axis

SPD	70.00000
SPL	60.00000
WIDG	30.00000
DIAG	25.00000
STA1	5.00000
ANG1	20.00000
ANG2	10.00000
RCO1	0.00000
RCO2	0.00000
RCI1	-2.00000
RCI2	-2.00000
FAL1	1.00000
FAL2	1.00000
IDEP	5.00000
DTB	1.00000
VARI	5
_URT	0.20000

Cancel OK



4. Confirm your settings with this softkey. The cycle is then automatically transferred to the program editor as a separate block.

```
NC:\MPF\1.MPF 2
CYCLE93( 70, 60, 30, 25, 5, 20, 10, 0, 0, -2, -2, 1, 1, 5, 1, 5, 0.2
)
==eof==
```

```
N10 G0 G90 Z100 X100 T5 D1 S400 M3 ;Starting point before the beginning of the cycle
N20 G95 F0.2 ; Specification of technology values
N30 CYCLE93(70, 60, 30, 25, 5, 20, 10, 0, 0, -2, - ; Cycle call
2, 1, 1, 5, 1, 5,0.2) Retraction distance of 0.2 mm programmed
N40 G0 G90 X100 Z100 ; Next position
N50 M02 ; End of program
```

12.5.4 Undercut (forms E and F to DIN) - CYCLE94

Programming

CYCLE94 (SPD, SPL, FORM, VARI)

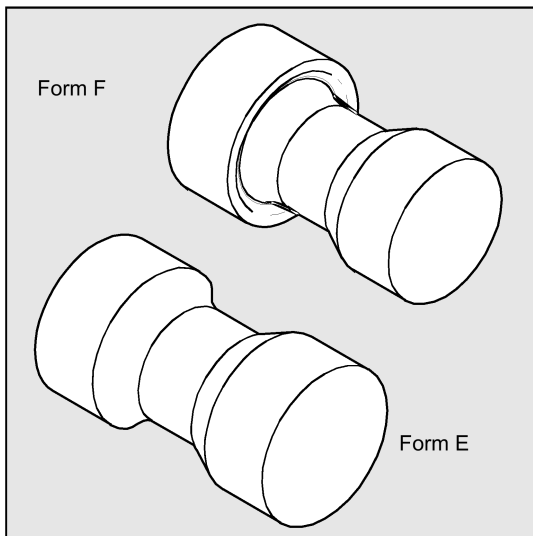
Parameters

Parameter	Data type	Description
SPD	REAL	Starting point along facing axis (enter without sign)
SPL	REAL	Starting point of the tool compensation in the longitudinal axis (enter without sign)
FORM	CHAR	Definition of the form Values: E (for form E), F (for form F)
VARI	INT	Specification of undercut position Values: 0: according to tool cutting edge position 1...4: define position

Function

This cycle can be used to perform undercuts to DIN509 of forms E and F with standard requirements at a finished diameter of >3 mm.

See the following illustration for undercut Form F and Form E:



Sequence

Position reached prior to cycle start:

The starting position can be any position from which the undercut can be approached without collision.

The cycle creates the following sequence of motions:

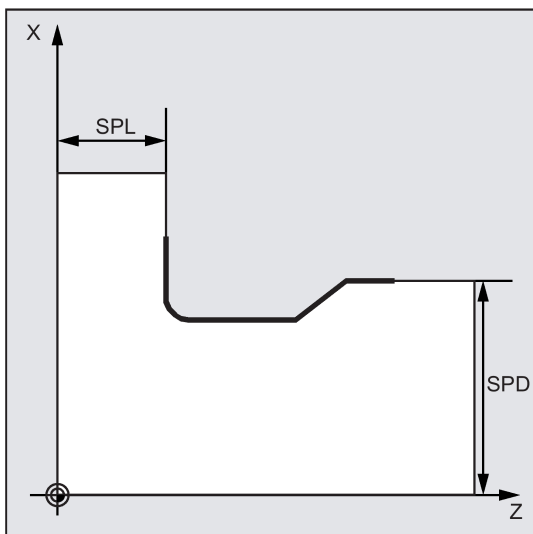
- Approach of the starting point determined in the cycle by using G0
- Selection of the cutter radius compensation according to the active tool point direction and traveling along the undercut contour at the feedrate programmed prior to the cycle call
- Retraction to the starting point with G0 and deselection of the cutter radius compensation with G40

Explanation of the parameters

SPD and SPL (starting point)

Use the parameter SPD to specify the finished part diameter for the undercut. The SPL parameter defines the finished dimension in the longitudinal axis.

If a final diameter of <3 mm results for the value programmed for SPD, the cycle is canceled, and alarm 61601 "Finished part diameter too small" is issued.

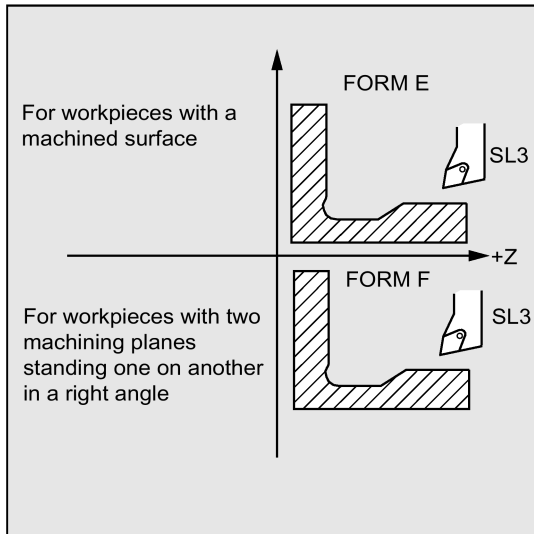


FORM (definition)

Form E and form F are fixed in DIN509 and must be defined using this parameter.

If the parameter has a value other than E or F, the cycle aborts and creates alarm 61609 "Form defined incorrectly".

See the following illustration for Form E and Form F:



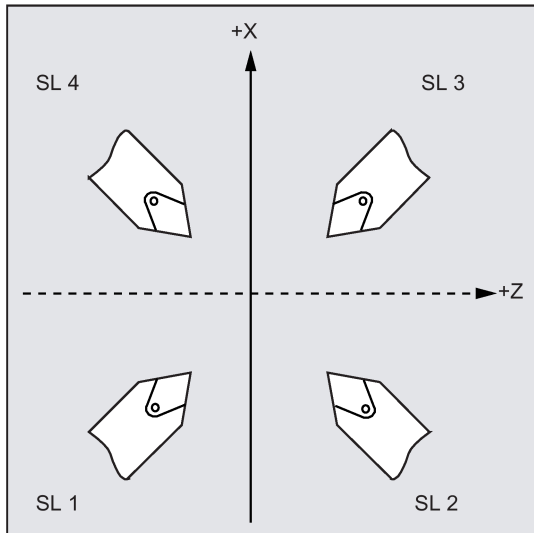
VARI (undercut position)

The position of the undercut can be either specified directly or derived from the tool point direction with the _VARI parameter.

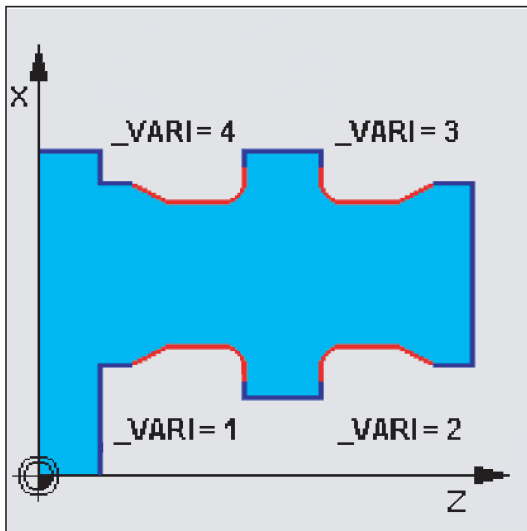
VARI=0: According to tool point direction

The tool point direction is determined by the cycle automatically from the active tool compensation. The cycle can operate with the tool point directions 1 ... 4.

If the cycle detects any of the tool point directions 5 ... 9, the alarm 61608 "Wrong tool point direction programmed" and the cycle is aborted.



VARI=1...4: Definition of undercut position



For $VARI \neq 0$, the following applies:

- The actual cutting edge position is not checked, i.e., all positions can be used if technologically suitable.

The clearance angle of the active tool is monitored in the cycle if an appropriate value is specified in the appropriate parameter of the tool compensation. If it turns out that the form of the undercut cannot be machined using the selected tool since its tool clearance angle is too small, then the message "Changed form of undercut" is displayed on the control system. The machining, however, is continued.

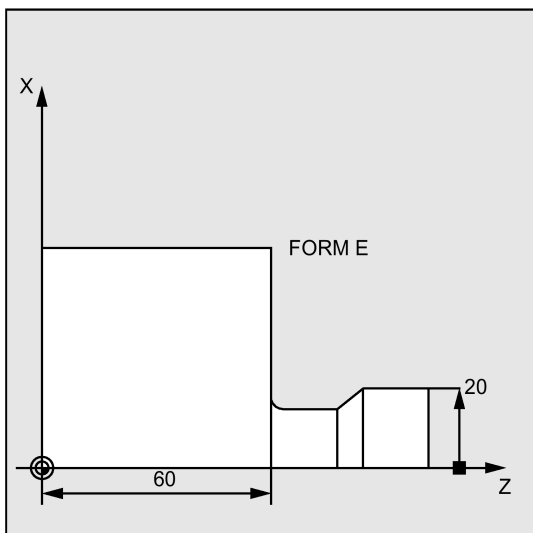
The cycle determines its starting point automatically. This is by 2 mm away from the end diameter and by 10 mm away from the finishing dimension in the longitudinal axis. The position of this starting point referred to the programmed coordinate values is determined by the tool point direction of the active tool.

Note

Before calling the cycle, a tool compensation must be activated; otherwise, the cycle is aborted after alarm 61000 "No tool compensation active" has been output.

Programming example: Undercut form E

This program can be used to program an undercut of form E.



```
N10 T1 D1 S300 M3 G95 F0.3
N20 G0 G90 Z100 X50
N30 CYCLE94(20, 60, "E",)
N40 G90 G0 Z100 X50
N50 M02
```

```
; Specification of technology values
; Selection of starting position
; Cycle call
; Approach next position
; End of program
```

12.5.5 Cutting with relief cut - CYCLE95

Programming

CYCLE95 (NPP, MID, FALZ, FALX, FAL, FF1, FF2, FF3, VARI, DT, DAM, _VRT)

Parameters

Parameter	Data type	Description
NPP	STRING	Name of contour subroutine
MID	REAL	Infeed depth (enter without sign)
FALZ	REAL	Finishing allowance in the longitudinal axis (enter without sign)
FALX	REAL	Finishing allowance in the transverse axis (enter without sign)
FAL	REAL	Finishing allowance suitable for contour (enter without sign)
FF1	REAL	Feedrate for roughing without relief cut
FF2	REAL	Feedrate for plunging into relief cut elements
FF3	REAL	Feedrate for finishing cut
VARI	REAL	Machining type Range of values: 1 ... 12
DT	REAL	Dwell time fore chip breaking when roughing
DAM	REAL	Path length after which each roughing step is interrupted for chip breaking
_VRT	REAL	Lift-off distance from contour when roughing, incremental (to be entered without sign)

Function

Using the rough turning cycle, you can produce a contour, which has been programmed in a subroutine, from a blank by paraxial stock removal. The contour may contain relief cut elements. With this cycle, It is possible to machine internal/external contours by using face/longitudinal machining. The technology can be freely selected (roughing, finishing, complete machining). When roughing the contour, paraxial cuts from the maximum programmed infeed depth are programmed and burrs are also removed parallel to the contour after an intersection point with the contour has been reached. Roughing is performed up to the final machining allowance programmed.

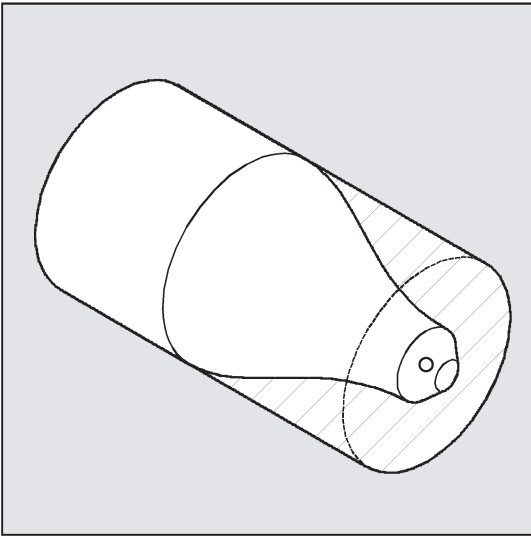
When the contour is machined through finishing and roughing, finishing is performed in the same direction as roughing, dependent on the setting of the VARI parameter; however, if the contour is machined through finishing only, the direction of machining is determined by the programmed direction, independent of the setting of the VARI parameter. The tool radius compensation can be selected (G41/G42) and deselected (G40) via the following USER DATA:

- _ZSD[13] = 0: roughing and finishing in G41/G42 mode
- _ZSD[13] = 1 (default): roughing in G40 mode and finishing in G41/G42 mode

For the machining with a large-radius tool, for example the button tool, you can set _ZSD[13] to 0 to activate the tool radius compensation in roughing by programming as follows in "MDA" mode:

```
MDI - Block
_ZSD[13]=0
==eof==
```

See the following illustration for CYCLE95:



Sequence

Position reached prior to cycle start:

The starting position is any position from which the starting point of the contour can be approached without collision.

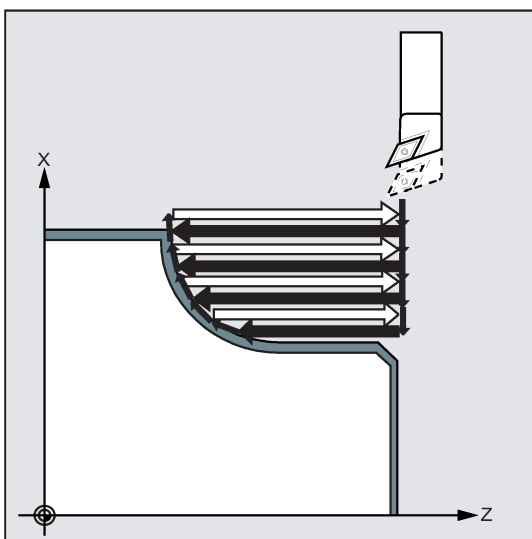
The cycle creates the following sequence of motions:

The cycle starting point is calculated internally and approached with G0 in both axes at the same time.

Roughing without relief cut elements:

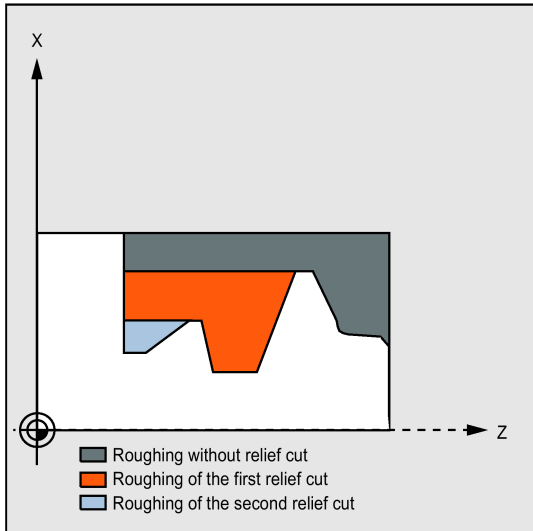
- The paraxial infeed to the current depth is calculated internally and approached with G0.
- Approach of paraxial roughing intersection point with G1 and at feedrate FF1.
- Rounding parallel to the contour along the contour + finishing allowance with G1/G2/G3 and FF1.
- Lift-off by the amount programmed under _VRT in each axis and retraction with G0.
- This sequence is repeated until the total depth of the machining step is reached.
- When roughing without relief cut elements, retraction to the cycle starting point is carried out axis by axis.

See the following illustration for the sequence of operation for CYCLE95:



Roughing the relief cut elements:

- Approach of the starting point for the next relief cut axis by axis with G0. When doing so, an additional cycle-internal safety clearance is observed.
- Infeed along the contour + finishing allowance with G1/G2/G3 and FF2.
- Approach of paraxial roughing intersection point with G1 and at feedrate FF1.
- Turning along the contour, carrying out retraction and return as with the first machining step.
- If there are further relief cut elements, this sequence is repeated for each relief cut.



Finishing:

- The cycle starting point is approached axis by axis with G0.
- The contour starting point is approached with G0 in both axes at the same time.
- Finishing along the contour with G1/G2/G3 and FF3
- Retraction to the starting point with both axes and G0

Explanation of the parameters

NPP (name)

This parameter is used to specify the contour name. You can define the contour as a subroutine or a section of the called program.

- Defining the contour as a subroutine
NPP = name of the subroutine
 - If the subroutine already exists, specify a name, and then continue.
 - If the subroutine does not yet exist, specify a name and then press the following softkey:

New
file

A program with the entered name is created and the program automatically jumps to the contour editor.

- Use the following softkey to confirm your input and return to the screen form for this cycle:

Tech
interface

- Defining the contour as a section of the called program
NPP = name of the starting label: name of the end label

Input:

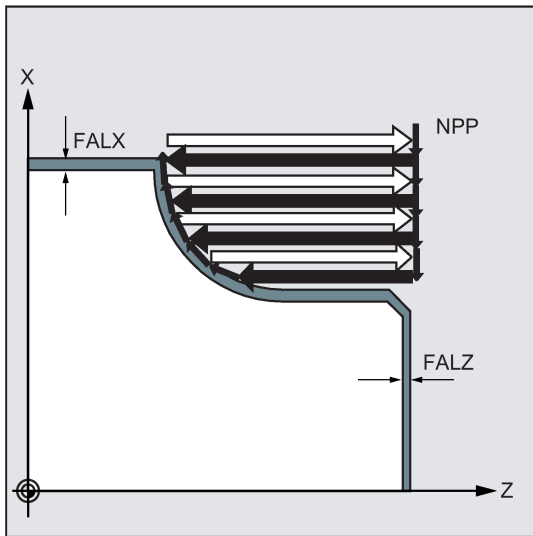
- If the contour is not yet described, specify the name of the starting label and press the following softkey. If the contour is already described (name of starting label: name of end label), directly press the following softkey:

**Attach
contour**

The control system automatically creates starting and end labels from the name entered and the program jumps to the contour editor.

- Use the following softkey to confirm your input and return to the screen form for this cycle:

**Tech
interface**



Examples:

```
NPP=CONTOUR_1      ;The rough turning contour is the complete program CONTOUR_1.
NPP=START:END      ;The rough turning contour is defined as a section in the calling program, which
                   ;starts from the block containing label START to the block containing label END.
```

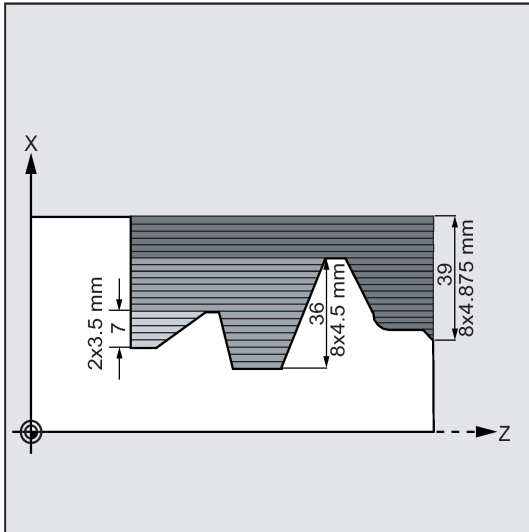
MID (infeed depth)

The MID parameter is used to define the maximum possible infeed depth for the roughing process.

The cycle automatically calculates the current infeed depth used for roughing.

With contours containing relief cut elements, the roughing process is divided by the cycle into individual roughing sections. The cycle calculates a new current infeed depth for each roughing section. This infeed depth is always between the programmed infeed depth and the half of its value. The number of required roughing steps is determined on the basis of the total depth of a roughing section and of the programmed maximum infeed depth to which the total depth to be machined is distributed equally. This provides optimum cutting conditions. For roughing this contour, the machining steps shown in the illustration result.

See the following illustration for the example of infeed depth:



Machining section 1 has a total depth of 39 mm. If the maximum infeed depth is 5 mm, eight roughing cuts are required. These are carried out with an infeed of 4.875 mm.

In machining step 2, 8 roughing steps, too, are carried out with an infeed of 4.5 mm each (total difference 36 mm).

In machining step 3, two roughing passes are carried out with a current infeed of 3.5 (total difference 7 mm).

FAL, FALZ and FALX (finishing allowance)

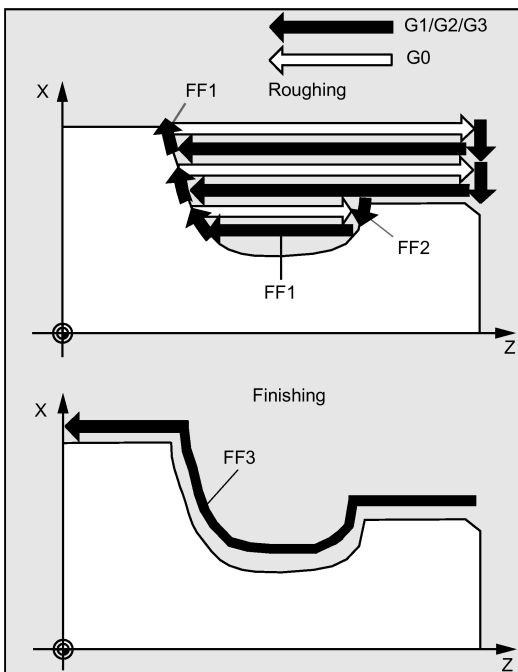
A finishing allowance for roughing can be specified either using the parameters FALZ and FALX if you want to specify different finishing allowances axis-specifically or via the parameter FAL for a finishing allowance that follows the contour. In this case, this value is taken into account in both axes as a finishing allowance.

No plausibility check is carried out for the programmed values. In other words: If all three parameters are assigned values, all these finishing allowances are taken into account by the cycle. It is, however, reasonable to decide either on the one or other form of definition of a finishing allowance.

Roughing is always carried out up to these finishing allowances. The resulting residual corner is also removed parallel to the contour after each paraxial roughing process immediately so that no additional residual corner cut is required after completion of roughing. If no finishing allowances are programmed, stock is removed when roughing up to the final contour.

FF1, FF2 and FF3 (feedrate)

It is possible to specify different feedrates for the individual machining steps, as shown in Figure NO TAG.



VARI (machining type)

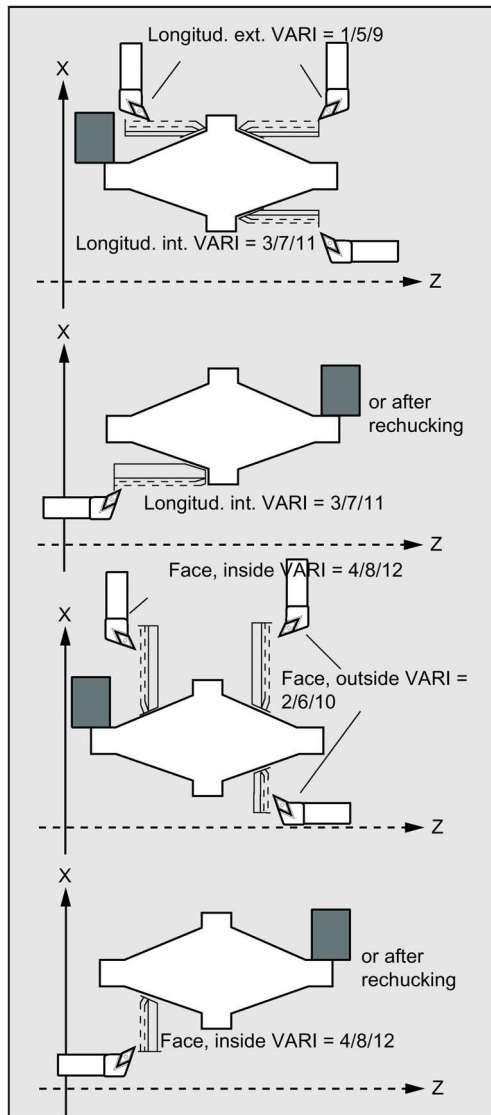
The machining type can be found in the table below.

Value	Longitudinal/face	Ext./int.	Roughing/finishing/complete
1	L	O	Roughing
2	P	O	Roughing
3	L	I	Roughing
4	P	I	Roughing
5	L	O	Finishing
6	P	O	Finishing
7	L	I	Finishing
8	P	I	Finishing
9	L	O	Complete machining
10	P	O	Complete machining
11	L	I	Complete machining
12	P	I	Complete machining

In longitudinal machining, the infeed is always carried out along the transversal axis, and in face machining - along the longitudinal axis.

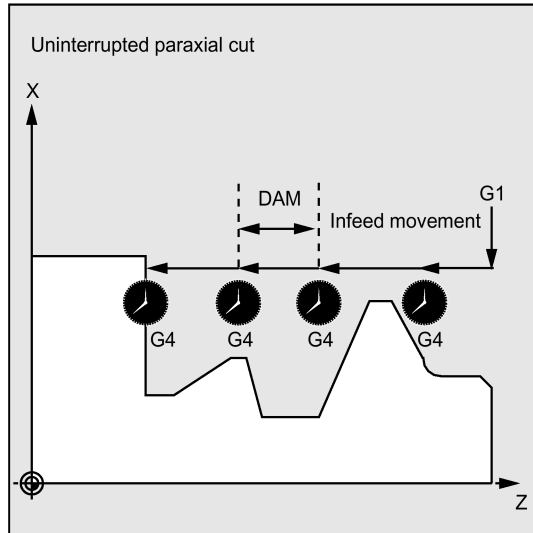
For external contour, carry out the infeed in the negative direction of the axis; for internal contour, carry out the infeed in the positive direction of the axis.

The VARI parameter is subjected to a plausibility check. If its value is not in the range 1 ... 12 when the cycle is called, the cycle is aborted with alarm 61002 "Machining type defined incorrectly".



DT and DAM (dwell time and path length)

These parameters can be used to achieve an interruption of the individual roughing steps after certain distances traversed in order to carry out chip breaking. These parameters are only relevant for roughing. The parameter DAM is used to define the maximum distance after which chip breaking is to be carried out. In DT, an appropriate dwell time (in seconds) can be programmed which is carried out at each of the cut interruption points. If no distance is specified for the cut interruption (DAM=0), uninterrupted roughing steps without dwell times are created.



_VRT (lift-off distance)

Parameter _VRT can be used to program the amount by which the tool is retracted in both axes when roughing.

For _VRT=0 (parameter not programmed), the tool will retract by 1 mm.

Contour definition

The contour must contain at least 3 blocks with motions in the two axes of the machining plane.

If the contour program is too short, the cycle is aborted after the alarms 10933 "Number of contour blocks contained in the contour program not sufficient" and 61606 "Error in contour preparation" have been output.

If a contour contains too many contour elements, the cycle is canceled, and alarm 10934 "Contour table overflow" is issued. In this case, the contour must be split over several contour sections, and the cycle for each section must be called separately.

If the maximum diameter is not at the programmed end or starting point of the contour, the cycle will automatically add an axis-parallel straight line to complete the contour maximum, and this part is removed as the undercut.

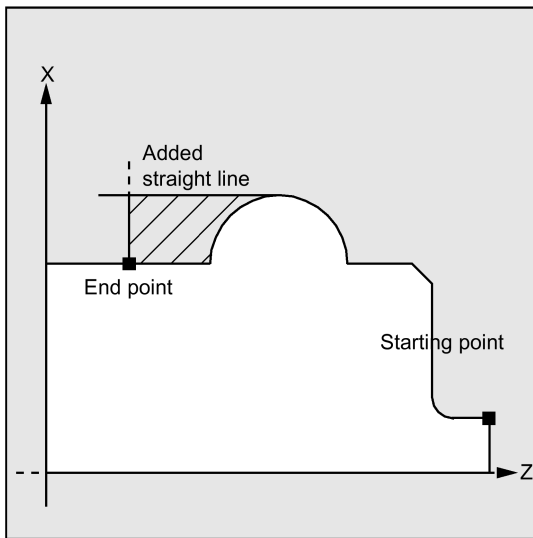
Relief cut elements can be connected directly one after the other. Blocks without motions in the plane can be written without restrictions.

In the cycle, all traversing blocks are prepared for the first two axes of the current plane since only these are involved in the cutting process. The contour program may contain any motions programmed for other axes; their distances to be traversed, however, will not come into effect during the whole cycle.

Only straight line and circle programming with G0, G1, G2 and G3 are permitted as the geometry in the contour. Furthermore, it is also possible to program the commands for rounding and chamfer. If any other motion commands are programmed in the contour, the cycle is aborted with the alarm 10930 "Illegal type of interpolation in the stock removal contour".

The first block with a traversing motion in the current machining plane must contain a motion command G0, G1, G2 or G3; otherwise, the cycle is canceled, and alarm 15800 "Incorrect prerequisites for CONTPRON" is issued. This alarm is also issued if G41/42 is active. The starting point of the contour is the first programmed position in the machining plane.

See the following illustration for contour definition:



If a tool radius compensation is programmed in the contour subroutine with G41/G42, the cycle is canceled, and alarm 10931 "Faulty cutting contour" is issued.

Contour direction

The direction in which the stock removal contour is programmed can be freely selected. In the cycle, the machining direction is defined automatically. In complete machining, the contour is finished in the same direction as machining was carried out when roughing.

When deciding on the machining direction, the first and the last programmed contour points are taken into account. Therefore, both coordinates must always be programmed in the first block of the contour subroutine.

Contour monitoring

The cycle provides contour monitoring with regard to the following:

- Clearance angle of the active tool
- Circular programming of arcs with an arc angle > 180 degrees

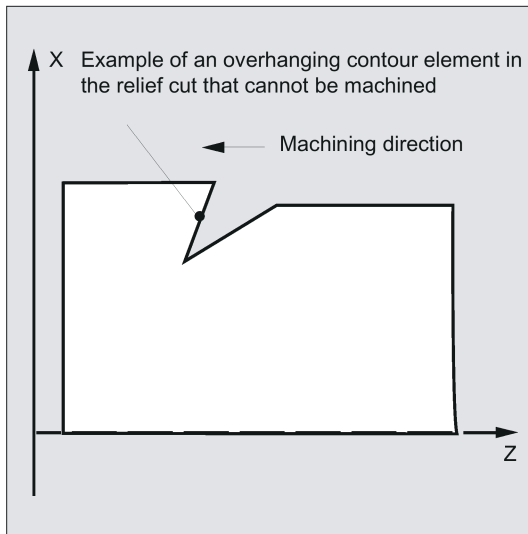
With relief cut elements, the cycle checks whether the machining is possible using the active tool. If the cycle detects that this machining will result in a contour violation, it will be aborted after alarm 61604 "Active tool violates programmed contour" has been output.

If the tool clearance angle is specified with zero in the tool compensation, this monitoring will not be performed.

If too large arcs are found in the compensation, alarm 10931 "Incorrect machining contour" appears.

Overhanging contours cannot be machined by CYCLE95. Contours of this type are not monitored by the cycle and consequently there is no alarm.

See the following illustration for contour monitoring:



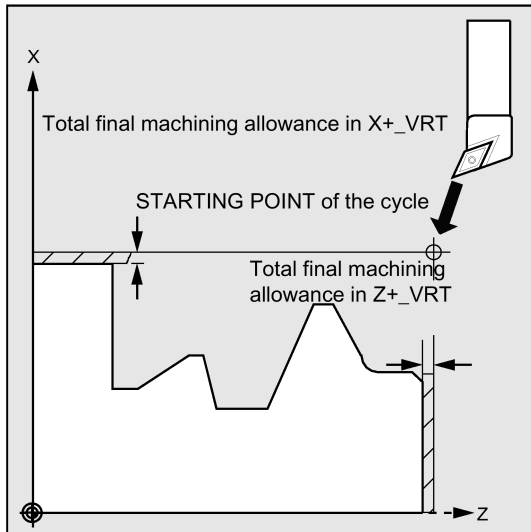
Starting point

The cycle determines the starting point for the machining operation automatically. The starting point is located in the axis in which the depth infeed is carried out, shifted from the contour by the amount of the finishing allowance + lift-off distance (parameter $_VRT$). In the other axis, it is by finishing allowance + $_VRT$ ahead of the contour starting point.

When the starting point is approached, the cutter radius compensation is selected internally in the cycle.

The last point before the cycle is called must therefore be selected such that this approach is possible without collision and space enough is provided to carry out the appropriate compensatory motion.

See the following illustration for starting point:



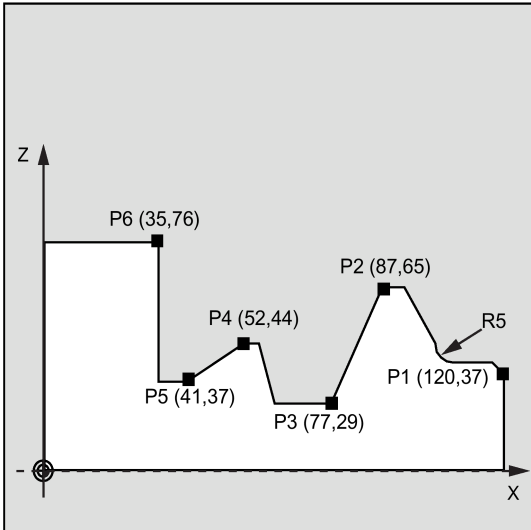
Cycle approach strategy

In roughing, the starting point determined by the cycle is always approached with both axes simultaneously, and in finishing, axis by axis. In finishing, the infeed axis traverses first.

Programming example 1: Stock removal cycle

The contour shown in the illustration to explain the defining parameters is to be machined longitudinally externally by complete machining. Axis-specific finishing allowances are specified. Cutting will not be interrupted when roughing. The maximum infeed is 5 mm.

The contour is stored in a separate program.



The following is the main program:

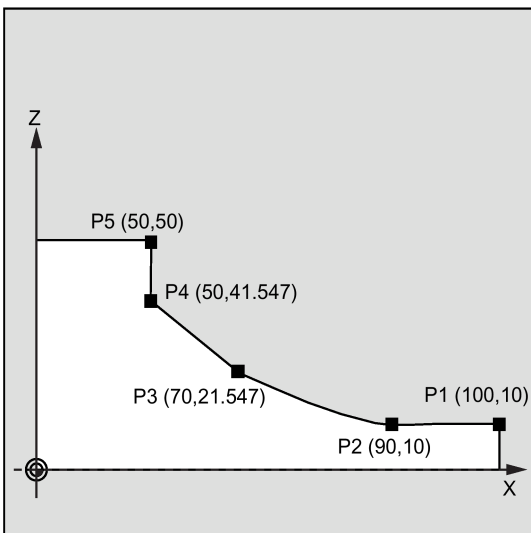
```
N10 T1 D1 G0 G95 S500 M3 Z125 X81 ; Approach position before cycle call
N20 CYCLE95("CONTOUR_1", 5, 1.2, 0.6, , 0.2, 0.1, 0.2, 9, , ; Cycle call
, 0.5)
N30 G0 G90 X81 ;Reapproach starting position
N40 Z125 ; Traverse axis by axis
N50 M2 ; End of program
```

The following is the subprogram:

```
CONTOUR_1.SPF ;Subroutine for contour turning (for
example)
N100 Z120 X37 ; Traverse axis by axis
N110 Z117 X40
N120 Z112 RND=5 ;Rounding with radius 5
N130 Z95 X65 ; Traverse axis by axis
N140 Z87
N150 Z77 X29
N160 Z62
N170 Z58 X44
N180 Z52
N190 Z41 X37
N200 Z35
N210 X76
N220 M02 ; End of subroutine
```

Programming example 2: Stock removal cycle

The stock removal contour is defined in the calling program and is traversed directly after the cycle for finishing has been called.



```

N110 G18 DIAMOF G90 G96 F0.8
N120 S500 M3
N130 T1 D1
N140 G0 X70
N150 Z160
N160 CYCLE95("START:END",2.5,0.8, 0.8,0,0.8,0.75,0.6,1, , , ; Cycle call
)
N170 G0 X70 Z160
N175 M02
START:
N180 G1 X10 Z100 F0.6
N190 Z90
N200 Z70 ANG=150
N210 Z50 ANG=135
N220 Z50 X50
END:
N230 M02

```

Programming example 3

Proceed through the following steps:



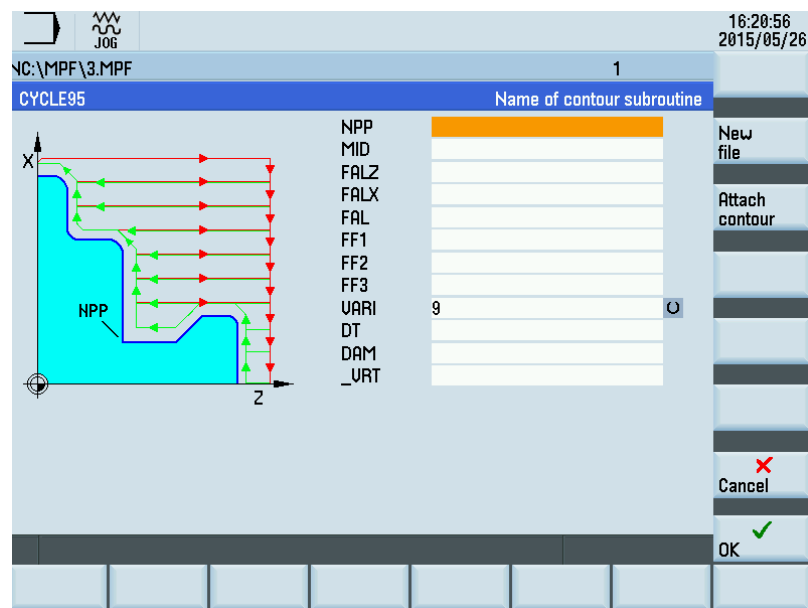
1. Select the program editing operating area.



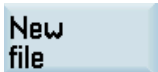
2. Open the vertical softkey bar for available turning cycles.



3. Press this softkey to open the window for CYCLE95. Enter a name in the first input field.



4. Press one of the following two softkeys. The program automatically jumps to the program editor screen form.



If you desire to edit and store the contour in a subroutine, press this softkey.

**Attach
contour**

If you desire to edit and store the contour as a section of a main program, press this softkey.

**Tech
interface**

Note:

For more information about programming in the contour editor, see Section "Free contour programming (Page 266)".

5. Press this softkey to return to the window for CYCLE95 and enter the parameters as desired.

OK

6. Confirm your settings with this softkey. The cycle is then automatically transferred to the program editor.

Note:

The cycle program created as a section of the main program must be stored after the M30 command.

7. If you desire to recompile the cycle, press this softkey.

**Re-
comp.**

Cavity contour

The following example describes how to process a cavity profile and how to define the subroutine of a contour with the contour functionality.

```
G500 G18 G95
G0X50
Z100
T5
M4S1500
G0X50Z0.5
G01X-2F0.15
Z2
G0X50
Z100
T2
G0X50Z10
M4S1500
G1F0.2
CYCLE95( "CON01", 0.50000, 0.20000, 0.20000, 0.20000, 0.20000, 0.20000, 0.10000, 1, , ,1.00000)
G0X55
Z100
M5
T1
M3S1500
G0X50Z10
CYCLE95( "CON02:CON02_E", 0.50000, 0.20000, 0.20000, 0.20000, 0.20000, 0.20000, 0.10000, 5, , ,1.00000)

M30
*****CONTOUR*****
;
```

CON02:

```
;#7__DlGK contour definition begin - Don't change!;*GP*;*RO*;*HD*
G18 G90 DIAMON;*GP*
G0 Z0 X0 ;*GP*
G1 X28 CHR=3 ;*GP*
Z-8.477 RND=2 ;*GP*
G2 Z-45.712 X40 K=AC(-25) l=AC(60) RND=2 ;*GP*
G1 Z-50 RND=3 ;*GP*
Z-55 X45 ;*GP*
;CON,V64,2,0.0000,0,0,MST:1,2,AX:Z,X,K,l;*GP*;*RO*;*HD*
;S,EX:0,EY:0,ASE:0;*GP*;*RO*;*HD*
;LU,EY:28;*GP*;*RO*;*HD*
;F,LFASE:3;*GP*;*RO*;*HD*
;LL;*GP*;*RO*;*HD*
;R,RROUND:2;*GP*;*RO*;*HD*
;ACW,DIA:209/217,EY:40,CX:-25,CY:60,RAD:23;*GP*;*RO*;*HD*
;R,RROUND:2;*GP*;*RO*;*HD*
;LL,EX:-50;*GP*;*RO*;*HD*
;R,RROUND:3;*GP*;*RO*;*HD*
;LA,EX:-55,EY:45;*GP*;*RO*;*HD*
;#End contour definition end - Don't change!;*GP*;*RO*;*HD*
M17
```

CON02_E::***** CONTOUR ENDS *****

12.5.6 Thread undercut - CYCLE96

Programming

CYCLE96 (DIATH, SPL, FORM, VARI)

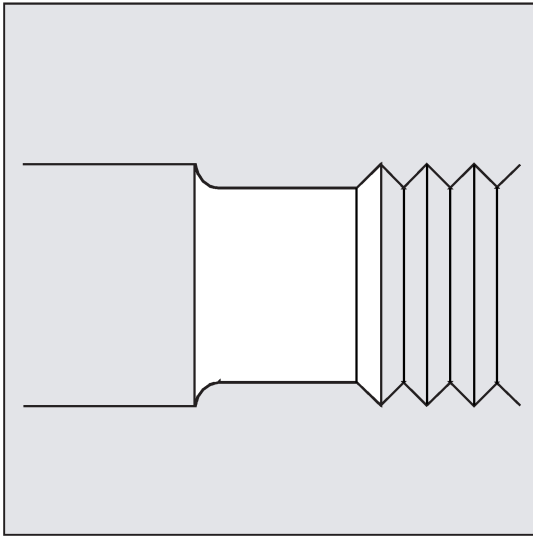
Parameters

Parameter	Data type	Description
DIATH	REAL	Nominal diameter of the thread
SPL	REAL	Starting point of the correction in the longitudinal axis
FORM	CHAR	Definition of the form Values: A (for form A), B (for form B), C (for form C), D (for form D)
VARI	INT	Specification of undercut position Values: 0: According to tool point direction 1...4: Define position

Function

You can use this cycle to perform thread undercuts to DIN76 for parts with a metrical ISO thread.

See the following illustration for CYCLE96:



Sequence

Position reached prior to cycle start:

The starting position is any position from which each thread undercut can be approached without collision.

The cycle creates the following sequence of motions:

- Approach of the starting point determined in the cycle by using G0
- Selection of the tool radius compensation according to the active tool point direction. Traversing along the undercut contour using the feedrate programmed before the cycle was called
- Retraction to the starting point with G0 and deselection of the tool radius compensation with G40

Explanation of the parameters

DIATH (nominal diameter)

Use this cycle to perform thread undercuts for metric threads from M3 through M68.

If the value programmed in DIATH results in a final diameter of <3 mm, the cycle is aborted and alarm:

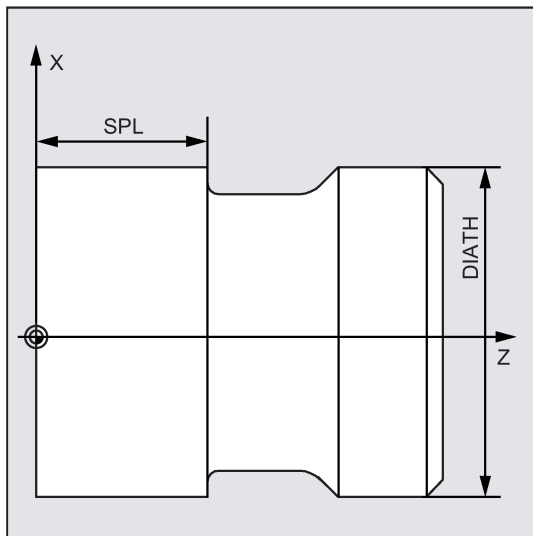
61601 "Finished part diameter too small" is issued.

If the parameter has a value other than specified in DIN76 Part 1, the cycle is also canceled, generating the alarm:

61001 "Thread lead defined incorrectly".

SPL (starting point)

The finished dimension in the longitudinal axis is defined using the parameter SPL.

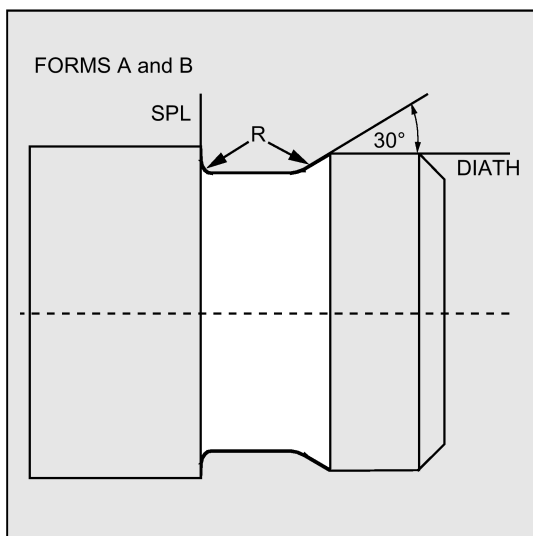


FORM (definition)

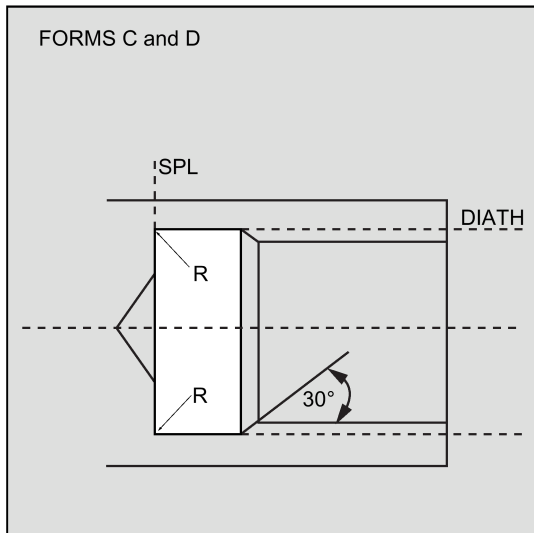
Thread undercuts of the forms A and B are defined for external threads, form A for standard run-outs of threads, and form B for short run-outs of threads.

Thread undercuts of the forms C and D are used for internal threads, form C for a standard run-out of the thread, and form D for a short run-out.

See the following illustration for Forms A and B:



See the following illustration for Forms C and D:



If the parameter has a value other than A ... D, the cycle aborts and creates alarm 61609 "Form defined incorrectly".

Internally in the cycle, the tool radius compensation is selected automatically.

The cycle only uses the tool point directions 1 ... 4. If the cycle detects a tool point direction 5 ... 9 or if the form of the undercut cannot be machined with the selected tool point direction, alarm 61608 "Wrong tool point direction programmed" is issued, and the cycle is canceled.

VARI (undercut position)

The position of the undercut can be either specified directly or derived from the tool point direction with the VARI parameter. See also Section "Undercut (forms E and F to DIN) - CYCLE94 (Page 184)".

The cycle will find the starting point determined by the tool point direction of the active tool and the thread diameter automatically. The position of this starting point referred to the programmed coordinate values is determined by the tool point direction of the active tool.

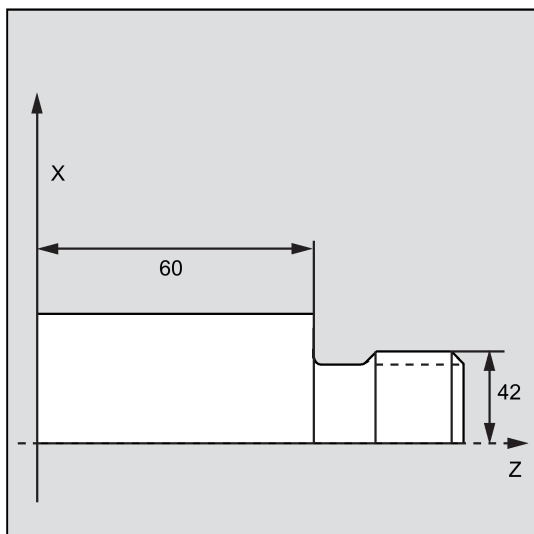
For the forms A and B, the undercut angle of the active tool is monitored in the cycle. If it is detected that the form of the undercut cannot be machined using the selected tool, the message "Changed form of undercut" will be displayed on the control system; the machining, however, is continued.

Note

Before calling the cycle, a tool compensation must be activated; otherwise, the cycle is terminated and the error message 61000 "No tool compensation active" is issued.

Programming example: Thread undercut form A

This program can be used to program a thread undercut of form A.



N10 D3 T1 S300 M3 G95 F0.3	; Specification of technology values
N20 G0 G90 Z100 X50	; Selection of starting position
N30 CYCLE98 (42, 60, "A",)	; Cycle call
N40 G90 G0 X100 Z100	; Approach next position
N50 M2	; End of program

12.5.7 Thread chaining - CYCLE98

Programming

CYCLE98 (PO1, DM1, PO2, DM2, PO3, DM3, PO4, DM4, APP, ROP, TDEP, FAL, IANG, NSP, NRC, NID, PP1, PP2, PP3, VARI, NUMTH, _VRT)

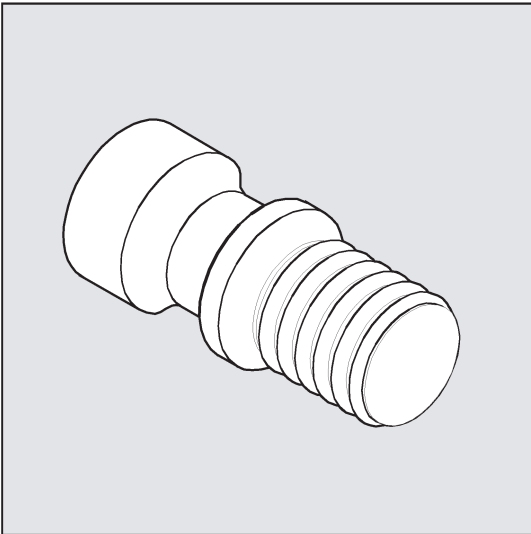
Parameters

Parameter	Data type	Description
PO1	REAL	Thread starting point in the longitudinal axis
DM1	REAL	Thread diameter at the starting point
PO2	REAL	First intermediate point in the longitudinal axis
DM2	REAL	Diameter at the first intermediate point
PO3	REAL	Second intermediate point
DM3	REAL	Diameter at the second intermediate point
PO4	REAL	Thread end point in the longitudinal axis
DM4	REAL	Diameter at the end point
APP	REAL	Run-in path (enter without sign)
ROP	REAL	Run-out path (enter without sign)
TDEP	REAL	Thread depth (enter without sign)
FAL	REAL	Finishing allowance (enter without sign)
IANC	REAL	Infeed angle Range of values: >0: Infeed along the rear flank <0: Infeed along the front flank =0: Infeed at a right angle to the cutting direction
NSP	REAL	Starting point offset for the first thread turn (enter without sign)
NRC	INT	Number of roughing cuts (enter without sign)
NID	INT	Number of idle passes (enter without sign)
PP1	REAL	Thread lead 1 as a value (enter without sign)
PP2	REAL	Thread lead 2 as a value (enter without sign)
PP3	REAL	Thread lead 3 as a value (enter without sign)
VARI	INT	Definition of the machining type for the thread Range of values: 1 ... 4
NUMTH	INT	Number of threads (enter without sign)
_VRT	REAL	Variable retraction path based on initial diameter, incremental (enter without sign)

Function

This cycle can be used to produce several cylindrical or tapered threads in succession. The individual thread sections can have different leads whereby the lead within one and the same thread section must be constant.

See the following illustration for CYCLE97:



Sequence

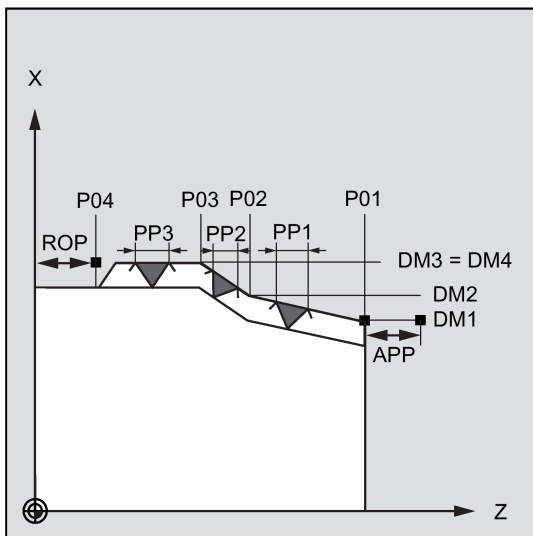
Position reached prior to cycle start:

Starting position is any position from which the programmed thread starting point and run-in path can be approached without collision.

The cycle creates the following sequence of motions:

- Approach of the starting point determined in the cycle at the beginning of the run-in path for the first thread turn with G0
- Infeed for roughing according to the infeed type defined under VARI.
- Thread cutting is repeated according to the programmed number of roughing cuts.
- The finishing allowance is removed in the following step with G33.
- This step is repeated according to the number of idle passes.
- The whole sequence of motions is repeated for each further thread turn.

Explanation of the parameters



PO1 and DM1 (starting point and diameter)

These parameters are used to define the original starting point for the thread series. The starting point determined by the cycle itself and approached at the beginning using G0 is located by the run-in path before the programmed starting point.

PO2, DM2 and PO3, DM3 (intermediate point and diameter)

These parameters are used to define two intermediate points in the thread.

PO4 and DM4 (end point and diameter)

The original end point of the thread is programmed under parameters PO4 and DM4.

With an inside thread, DM1...DM4 corresponds to the tap hole diameter.

Interrelation between APP and ROP (run-in/run-out paths)

The starting point used in the cycle, however, is the starting point brought forward by the run-in path APP, and, correspondingly, the end point is the programmed end point brought back by the run-out path ROP.

In the transversal axis, the starting point defined by the cycle is always by 1 mm above the programmed thread diameter. This lift-off plane is generated within the control system automatically.

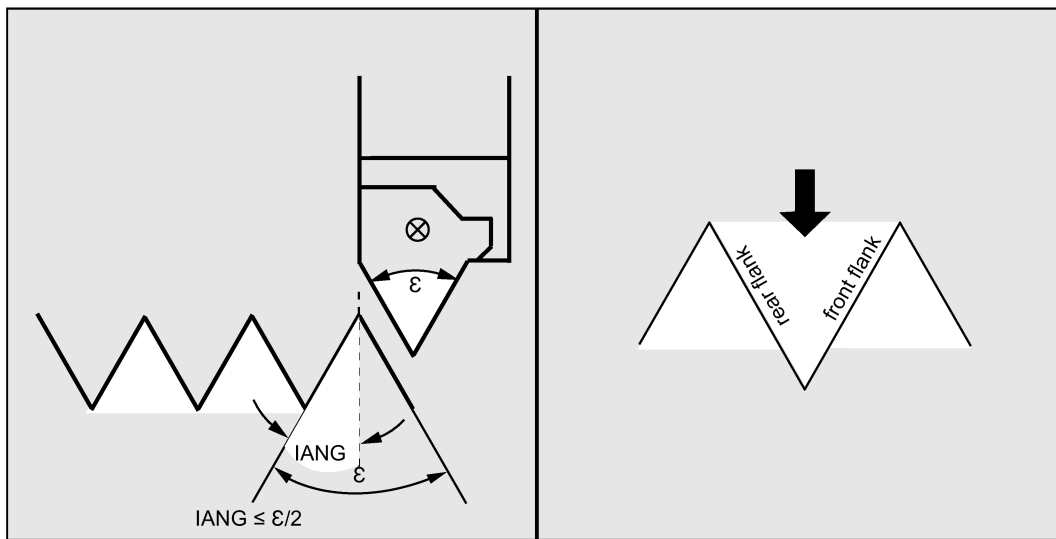
Interrelation between TDEP, FAL, NRC and NID (thread depth, finishing allowance, number of roughing and idle passes)

The programmed finishing allowance acts paraxially and is subtracted from the specified thread depth TDEP; the remainder is divided into roughing cuts. The cycle will calculate the individual infeed depth automatically, depending on the VARI parameter. When the thread depth is divided into infeeds with constant cutting cross-section, the cutting resistance will remain constant over all roughing cuts. In this case, the infeed will be performed using different values for the infeed depth.

A second version is the distribution of the whole thread depth to constant infeed depths. When doing so, the cutting cross-section becomes larger from cut to cut, but with smaller values for the thread depth, this technology can result in better cutting conditions.

The finishing allowance FAL is removed after roughing in one step. Then the idle passes programmed under parameter NID are executed.

LANG (infeed angle)



By using parameter LANG, the angle is defined under which the infeed is carried out in the thread. If you wish to infeed at a right angle to the cutting direction in the thread, the value of this parameter must be set to zero. This means that the parameter can be omitted in the parameter list, as in this case the value is defaulted automatically with zero. If you wish to infeed along the flanks, the absolute value of this parameter may amount maximally to the half of the flank angle of the tool.

The execution of the infeed is defined by the sign of this parameter. With a positive value, infeed is always carried out along the rear flank, and with a negative value, infeed is always carried out along the front flank. If the value of LANG for tapered threads is nonetheless negative, the cycle will carry out a flank infeed along a flank.

NSP (starting point offset)

You can use this parameter to program the angle value defining the point of the first cut of the thread turn at the circumference of the turned part. This involves a starting point offset. The parameter can assume values between 0.0001

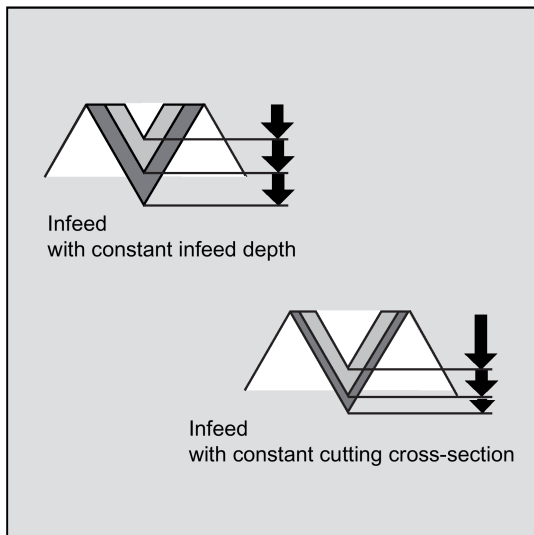
and +359.9999 degrees. If no starting point offset has been specified or the parameter has been omitted from the parameter list, the first thread turn automatically starts at the zero-degree mark.

PP1, PP2 and PP3 (thread lead)

These parameters are used to define the value of the thread lead in the three sections of the thread series. A value without sign must be entered as the lead value.

VARI (machining type)

By using the VARI parameter, it is defined whether external or internal machining will be carried out and which technology will be used with regard to the infeed when roughing. The VARI parameter can assume values between 1 and 4 with the following meaning:



Value	Ext./int.	Const. Infeed/const. cutting cross-section
1	External	Constant infeed
2	Internal	Constant infeed
3	External	Constant cutting cross-section
4	Internal	Constant cutting cross-section

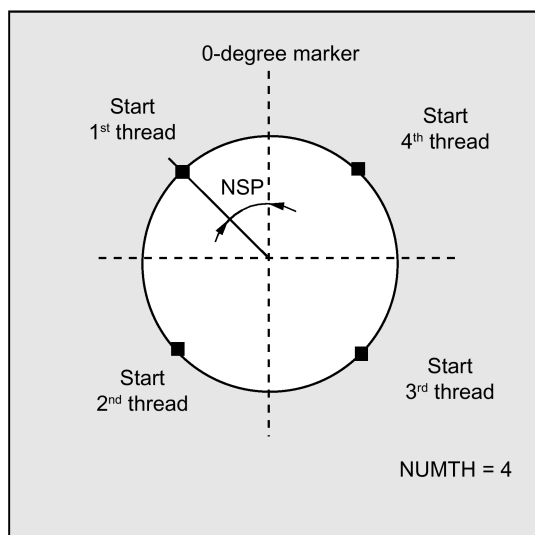
If a different value is programmed for the VARI parameter, the cycle is aborted after output of alarm 61002 "Machining type defined incorrectly".

NUMTH (number of thread turns)

Use the NUMTH parameter to define the number of thread turns with a multiple-turn thread. For a single-turn thread, the parameter must be assigned zero or can be ignored.

The thread turns are distributed equally over the circumference of the turned part; the first thread turn is determined by the NSP parameter.

To produce a multiple-turn thread with an asymmetrical arrangement of the thread turns on the circumference, the cycle for each thread turn must be called when programming the appropriate starting point offset.

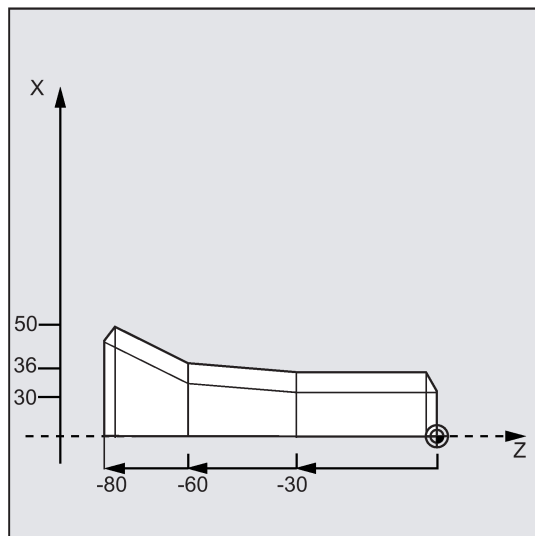


_VRT (variable retraction path)

The retraction path can be programmed on the basis of the initial thread diameter in the _VRT parameter. For _VRT = 0 (parameter not programmed), the retraction path is 1 mm. The retraction path is always measured according to the programmed measuring system, inch or metric.

Programming example: Thread chain

You can use this program to produce a thread chain starting with a cylindrical thread. The infeed is performed vertically to the thread; neither finishing allowance, nor starting point offset are programmed. Five roughing cuts and one noncut are performed. The machining type is defined as longitudinal, external, with constant cross-section of cut.



```

N10 G95 T5 D1 S1000 M4
N20 G0 X40 Z10
N30 CYCLE98 (0, 30, -30, 30, -60, 36, -80, 50, 10, 10,
0.92, , , 5, 1, 1.5, 2, 2, 3, 1,)
N40 G0 X55
N50 Z10
N60 X40
N70 M2

```

```

; Specification of technology values
; Approach starting position
; Cycle call
; Traverse axis by axis
; End of program

```

12.5.8 Thread cutting - CYCLE99

Programming

CYCLE99 (SPL, DM1, FPL, DM2, APP, ROP, TDEP, FAL, IANG, NSP, NRC, NID, PIT, VARI, NUMTH, _VRT, 0, 0, 0, 0, 0, 0, 0, PITA, 0, 0, 0, PSYS)

Parameters

Parameter	Data type	Description
SPL	REAL	Thread starting point in the longitudinal axis
DM1	REAL	Thread diameter at the starting point
FPL	REAL	Thread end point in the longitudinal axis
DM2	REAL	Thread diameter at the end point
APP	REAL	Run-in path (enter without sign)
ROP	REAL	Run-out path (enter without sign)
TDEP	REAL	Thread depth (enter without sign)
FAL	REAL	Finishing allowance (enter without sign)
IANC	REAL	Infeed angle Range of values: >0: Infeed along the rear flank <0: Infeed along the front flank =0: Infeed at a right angle to the cutting direction
NSP	REAL	Starting point offset for the first thread turn (enter without sign)
NRC	INT	Number of roughing cuts (enter without sign)
NID	INT	Number of idle passes (enter without sign)
PIT	REAL	Thread pitch value (enter without sign) Unit defined in parameter PITA
VARI	INT	Definition of the machining type for the thread (values: 300101 = external thread with linear infeed; 300102 = internal thread with linear infeed; 300103 = external thread with degressive infeed; 300104 = internal thread with degressive infeed)
NUMTH	INT	Number of threads (enter without sign)
_VRT	REAL	Variable retraction path for thread cutting, incremental (enter without sign)
PSYS	INT	Internal parameter, only the default value 0 is possible (value: 0)
PSYS	INT	Internal parameter, only the default value 0 is possible (value: 0)
PSYS	INT	Internal parameter, only the default value 0 is possible (value: 0)
PSYS	INT	Internal parameter, only the default value 0 is possible (value: 0)
PSYS	INT	Internal parameter, only the default value 0 is possible (value: 0)
PSYS	INT	Internal parameter, only the default value 0 is possible (value: 0)
PSYS	INT	Internal parameter, only the default value 0 is possible (value: 0)
PITA	INT	Unit of parameter PIT (thread pitch) Values: 1 = pitch in mm/revolution 2 = pitch in threads per inch(TPI)
PSYS	STRING	Internal parameter
PSYS	STRING	Internal parameter
PSYS	STRING	Internal parameter
PSYS	INT	Internal parameter, only the following values are possible (values: 0 = Longitudinal thread; 10 = Face thread; 20 = Taper thread)

Function

The thread cutting cycle consists of three alternatives: longitudinal thread, face thread or taper thread.

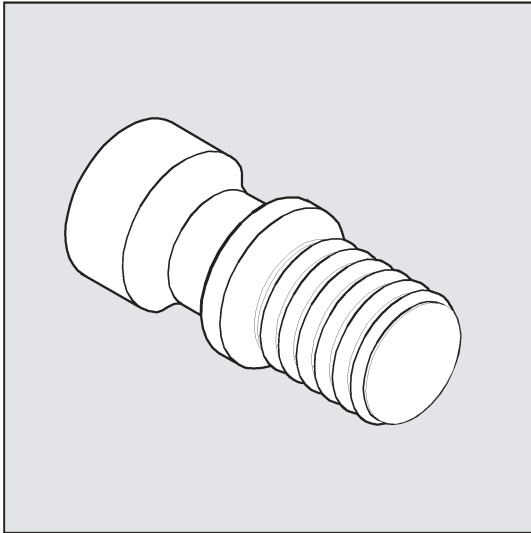
Use this cycle to produce cylindrical and tapered external and internal threads with constant lead in longitudinal and face machining. The thread can be single or multiple. With multiple threads, the individual thread turns are machined one after the other.

For thread infeed, you can carry out the infeed with constant infeed depth or constant cutting cross-section.

Right-hand or left hand thread is determined by the direction of rotation of the spindle which must be programmed prior to the cycle start.

Both feed and spindle override are ineffective in the traversing blocks with thread.

See the following illustration for CYCLE99:



Note

To use this cycle, the spindle must be equipped with an encoder.

Sequence

Position reached prior to cycle start:

Starting position is any position from which the programmed thread starting point and run-in path can be approached without collision.

The cycle creates the following sequence of motions:

- Approach of the starting point determined in the cycle at the beginning of the run-in path for the first thread turn with G0
- Infeed for roughing according to the infeed type defined under VARI.
- Thread cutting is repeated according to the programmed number of roughing cuts.
- The finishing allowance is removed in the following step with G33.
- This step is repeated according to the number of idle passes.
- The whole sequence of motions is repeated for each further thread turn.

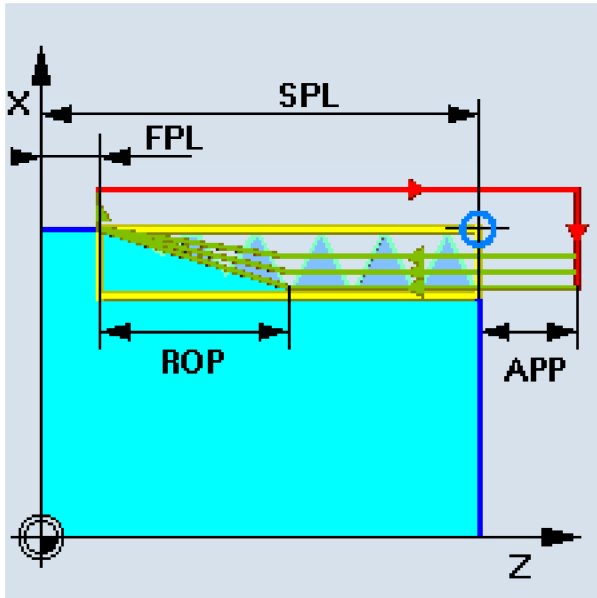
Explanation of the parameters

DM1 and DM2 (diameter)

Use this parameter to define the thread diameter of starting and end point of the thread. In the case of internal threads, this is the tap-hole diameter.

Interrelation SPL, FPL, APP and ROP (starting, end point, run-in and run-out path)

See the following illustration for the parameters for CYCLE99:



The programmed starting point (SPL) or end point (FPL) constitutes the original starting point of the thread. The starting point used in the cycles, however, is the starting point brought forward by the run-in path APP.

The run-out path (Cut out) begins before the programmed end point FPL. It brings the end position of thread forward so that the end of cut out is equal FPL.

Interrelation TDEP, FAL, NRC and NID (thread depth, finishing allowance, number of cuts)

The programmed finishing allowance acts paraxially and is subtracted from the specified thread depth TDEP; the remainder is divided into roughing cuts.

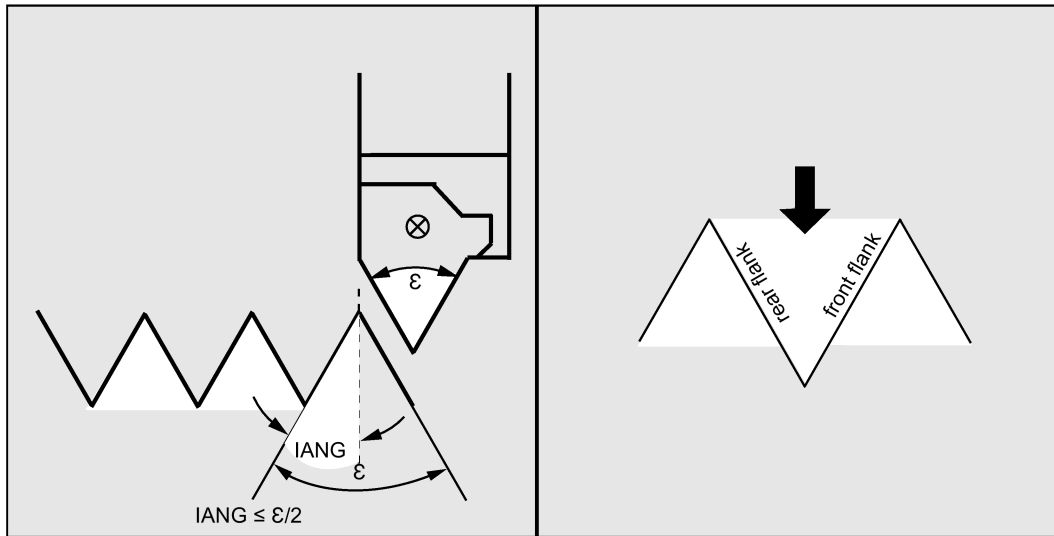
The cycle will calculate the individual infeed depth automatically, depending on the VARI parameter.

When the thread depth is divided into infeeds with constant cutting cross-section, the cutting force will remain constant over all roughing cuts. In this case, the infeed will be performed using different values for the infeed depth.

A second version is the distribution of the whole thread depth to constant infeed depths. When doing so, the cutting cross-section becomes larger from cut to cut, but with smaller values for the thread depth, this technology can result in better cutting conditions.

The finishing allowance FAL is removed after roughing in one step. Then the idle passes programmed under parameter NID are executed.

IANG (infeed angle)

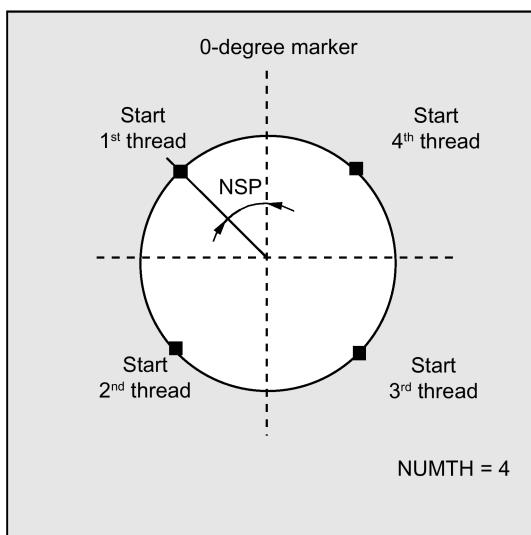


By using parameter IANG, the angle is defined under which the infeed is carried out in the thread. If you wish to infeed at a right angle to the cutting direction in the thread, the value of this parameter must be set to zero. If you wish to infeed along the flanks, the absolute value of this parameter may amount maximally to the half of the flank angle of the tool.

The execution of the infeed is defined by the sign of this parameter. With a positive value, infeed is always carried out along the rear flank, and with a negative value, infeed is always carried out along the front flank. If the value of IANG for tapered threads is nonetheless negative, the cycle will carry out a flank infeed along a flank.

NSP (starting point offset) and NUMTH (number)

You can use this parameter to program the angle value defining the point of the first cut of the thread turn at the circumference of the turned part. This involves a starting point offset. The parameter can assume values between 0 and +359.9999 degrees. If no starting point offset has been specified or the parameter has been omitted from the parameter list, the first thread turn automatically starts at the zero-degree mark.



Use the NUMTH parameter to define the number of thread turns with a multiple-turn thread. For a single-turn thread, the parameter must be assigned zero or ignored directly.

The thread turns are distributed equally over the circumference of the turned part; the first thread turn is determined by the NSP parameter.

To produce a multiple-turn thread with an asymmetrical arrangement of the thread turns on the circumference, the cycle for each thread turn must be called when programming the appropriate starting point offset.

PIT (thread pitch) and PITA (unit of thread pitch)

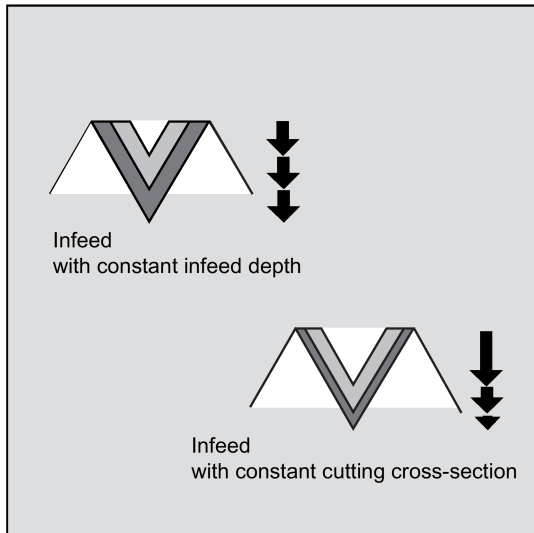
The thread lead is an axis-parallel value and is specified without sign. The unit of it is defined in parameter PITA.

PITA = 1 pitch in mm/revolution

= 2 pitch in threads per inch (TPI)

VARI (machining type)

By using the VARI parameter, it is defined whether external or internal machining will be carried out and which technology will be used with regard to the infeed when roughing. The VARI parameter can assume values between 1 and 4 with the following meaning:



Value	Ext./int.	Const. Infeed/const. cutting cross-section
300101	O	Constant infeed
300102	I	Constant infeed
300103	O	Constant cutting cross-section
300104	I	Constant cutting cross-section

If a different value is programmed for the VARI parameter, the cycle is aborted after output of alarm 61002 "Machining type defined incorrectly".

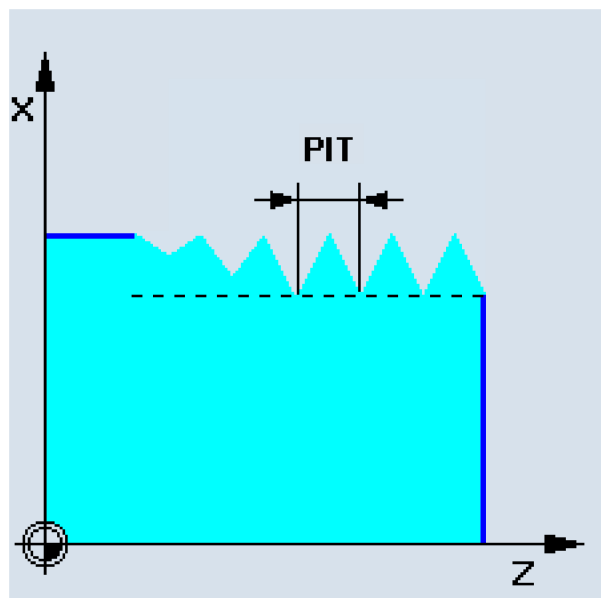
_VRT (variable retraction path)

The retraction path can be programmed on the basis of the initial thread diameter in the _VRT parameter. For _VRT = 0 (parameter not programmed), the retraction path is 1 mm. The retraction path is always measured according to the programmed system of units, inch or metric.

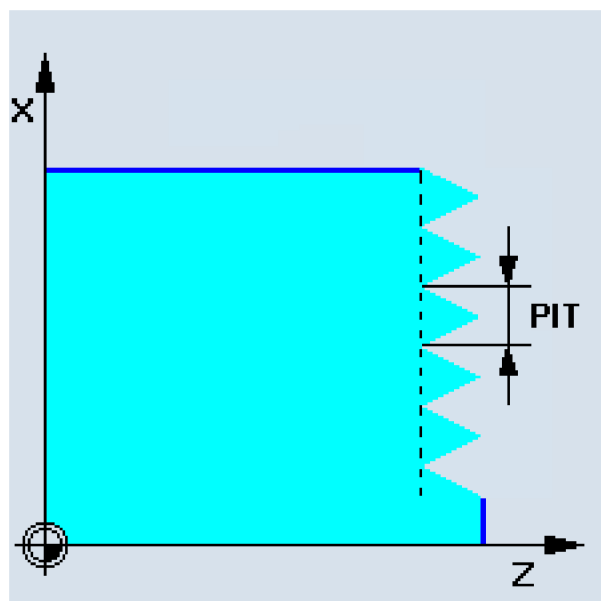
PSYS (the last parameter displayed on the screen)

This parameter specifies the selection of longitudinal thread, face thread or taper thread.

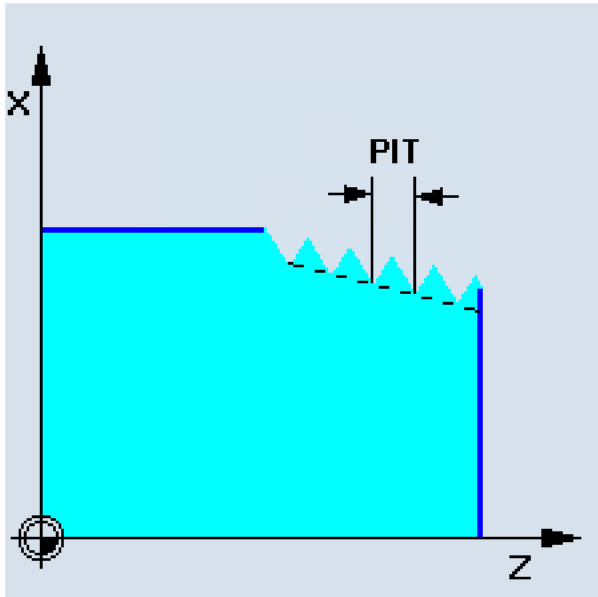
0 = longitudinal thread:



10 = face thread:

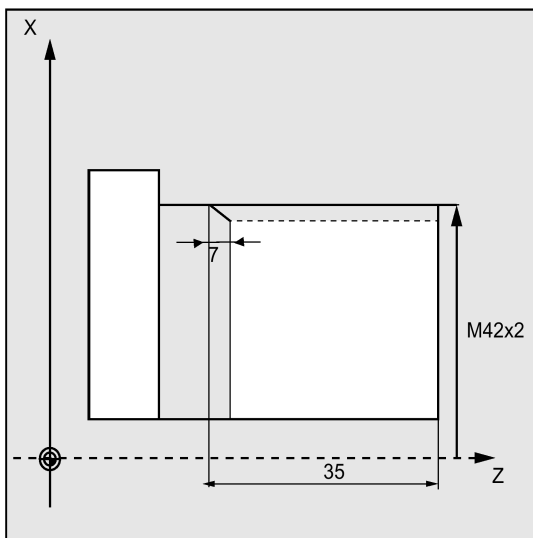


20 = taper thread:



Programming example: Thread cutting

By using this program, you can produce a metric external thread M42x2 with flank infeed. Infeed is carried out with constant cutting cross-section. At the end of thread a cut out of 7 mm is defined. 5 roughing cuts are carried out at a thread depth of 2.76 mm without finishing allowance. At completion of this operation, two idle passes will be carried out.



```

N10 G0 G90 X60 Z100 G95 ; Selection of starting position
N20 T1 D1
N30 M6 ; Tool change
N40 S1000 M4 ; Specification of technology values
N50 CYCLE99(0, 42, -35, 42, 5, 7, 2.76, 0, 0, 0, 5, 2, 4.5, ; Cycle call
300101, 1, 0, 0, 0, 0, 0, 0, 0, 0, 0, 1, , , 0)

N60 G0 G90 X100 Z100 ; Approach next position
N70 M30 ; End of program

```

12.6 Error messages and error handling

12.6.1 General Information

If error conditions are detected in the cycles, an alarm is generated and the execution of the cycle is aborted.

Furthermore, the cycles display their messages in the message line of the control system. These messages will not interrupt the program execution.

The errors with their reactions and the messages in the message line of the control system are described in conjunction with the individual cycles.

12.6.2 Error handling in the cycles

Alarms with numbers between 61000 and 62999 generated in the cycles. This range of numbers, in turn, is divided again with regard to alarm responses and cancel criteria.

The error text that is displayed together with the alarm number gives you more detailed information on the error cause.

Alarm number	Clearing criterion	Alarm Response
61000 ... 61999	NC_RESET	Block preparation in the NC is aborted
62000 ... 62999	Clear key	The block preparation is interrupted; the cycle can be continued the following key on the MCP after the alarm has been cleared: 

12.6.3 Overview of cycle alarms

The error numbers are classified as follows:

6	_	X	_	_
---	---	---	---	---

- X=0 General cycle alarms
- X=1 Alarms generated by the drilling/drilling pattern/milling cycles
- X=6 Alarms generated by the turning cycles

12.6.4 Messages in the cycles

The cycles display their messages in the message line of the control system. These messages will not interrupt the program execution.

Messages provide information with regard to a certain behavior of the cycles and with regard to the progress of machining and are usually kept beyond a machining step or until the end of the cycle. An example of messages is as follows:

"Depth: according to the value for the relative depth" from all drilling cycles.

13 Machining typical turning parts

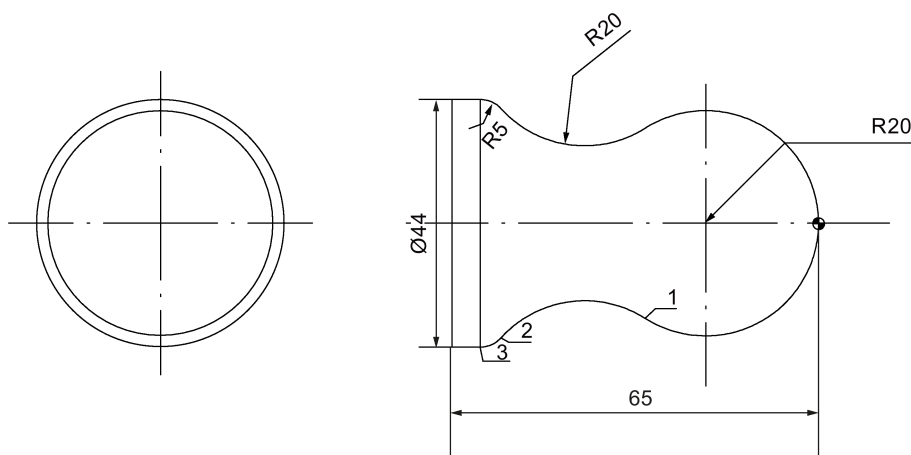
13.1 Operations before programming

Operating sequence

1. Turn on the power supply for the machine tool. Proceed with the operations for the reference point approach, if the machine axis is equipped with an incremental encoder (see Chapter "Switching on and referencing (Page 19)").
2. Create the required tools (see Section "Creating a tool (Page 20)").
3. Activate the tools and the spindle (see Section "Activating the tool and the spindle (Page 23)").
4. Proceed with the handwheel assignment operations if the machine manufacturer has not assigned the handwheel for the control system (see Chapter "Assigning the handwheel (Page 24)").
5. Proceed with the tool setting operations to finish measuring all the tools (see Section "Additional method for measuring the tool (Page 58)").
6. Verify the tool offset result to guarantee the safety and correctness of machining (see Section "Verifying the tool offset result (Page 31)").

13.2 Programming for machining a spherical surface (Example 1)

Workpiece drawing (unit: mm)



Coordinates of key machining points

Point	X coordinate	Z coordinate
1	33.762	-30.726
2	40.704	-56.290
3	44.000	-60.000

Technical requirements

- The arc transition must be smooth, without lapping.
- The cutting trace must be even.
- The sharp edges must be rounded.

Blank data

Blank material: aluminum bar

Blank diameter: 45 mm

Blank length: 100 mm (machining limit along the Z axis: Z-66; three-jaw chuck clamping length: 25 mm)

Required tool

Tool type	Tool number		Roughing		Finishing	
	DIN	ISO	Speed	Feedrate	Speed	Feedrate
35 ° finishing tool	T2D1	T0202	S1000	F0.2	S2000	F0.1

Note

- When programming in ISO mode, you need to set the H number of T2 to 2 in the tool list;
- Cut the workpiece manually after machining is over.

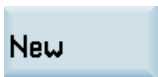
Operating sequence for programming in Siemens mode



1. Select the program management operating area.



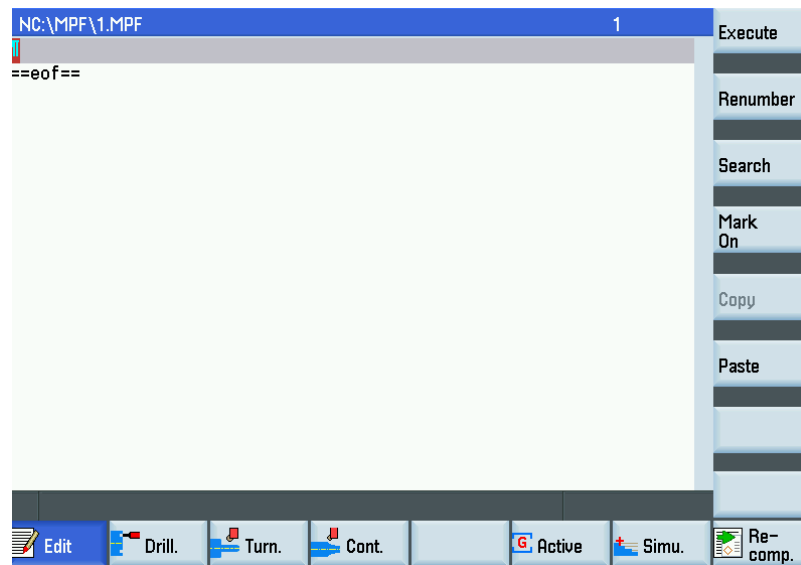
2. Press this softkey to enter the system directory for storing part programs.



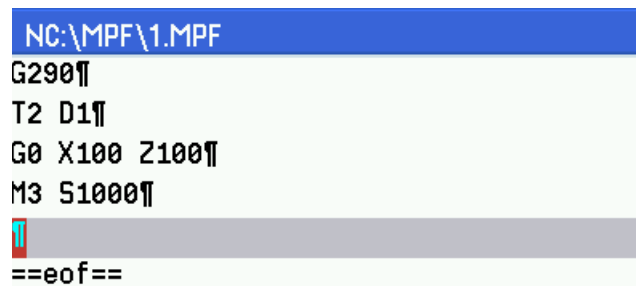
3. Press this softkey and enter the name of the new program.

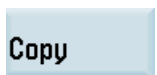


4. Press this softkey to confirm your entry. The part program editor window opens automatically.



5. Enter the following program in the window. The control system saves your editing automatically.





6. Open the vertical softkey bar for available turning cycles.
7. Press this softkey to open the window for CYCLE95 and enter the name of the contour subroutine in "NPP", for example, "RR".
8. Press this softkey to open the program editor window again and enter the program blocks for the external contour:
 G1 X0 Z0
 G3 X33.762 Z-30.726 CR=20
 G2 X40.704 Z-56.29 CR=20
 G2 X44 Z-60 CR=5
 G1 Z-66
 X45

Note:

Make sure you enter the blank diameter in the last line of the program for the external contour (for example, "X45" in the above program). Besides, the program for the external contour has to be positioned after "M30". For detailed information, see the screenshots of the complete program at the end of this example.

9. Press this softkey to return to the window for CYCLE95 and enter the desired parameters as follows:

CYCLE95		Retract. path from contour, incremental
NPP	RR: RR_E	
MID	1.50000	
FALZ		
FALX		
FAL	0.15000	
FF1	0.20000	
FF2	0.10000	
FF3	0.10000	
UARI	1	
DT	0.00000	
DAM	0.00000	
_URT	2.00000	

10. Press this softkey to confirm your input and open the cycle programming window.
11. Enter the following program block:
M3 S2000
12. Press this softkey to return to the program editor window.
13. Move the cursor to "C" in the line for CYCLE95 with the cursor keys and press this softkey to insert a marker.
14. Use the cursor keys to select the entire program block for CYCLE95. Press this softkey to copy the selection to the buffer memory.
15. Place the cursor on the end-of-block character after "M3 S2000" with cursor keys and press this key on the MCP to enter a new line.
16. Paste the content of the buffer memory into the new line.



17. Press this softkey to open the window for CYCLE95 again. Move the cursor to the input field of "VARI".



18. Press this key to change the value to "5".



19. Press this softkey to accept the change.



20. Press this key to enter a new line.
Continue to enter the following program blocks:
G0 X100 Z100
M05
M30
Then you can proceed with the operations for "program simulation and execution (Page 241)".

Screenshots of the complete program

Screen 1:

```
NC:\MPF\1. 1
290
T2 D1
G0 X100 Z100
T3 S1000
CYCLE95( "RR: RR_E", 1.5, , 0.15, 0.2, 0.1, 0.1, 1, 0, 0, 2)
T3 S2000
CYCLE95( "RR: RR_E", 1.5, , 0.15, 0.2, 0.1, 0.1, 5, 0, 0, 2)
G0 X100 Z100
M05
M30
; *****CONTOUR*****
RR:
G1 X0 Z0
G3 X33.762 Z-30.726 CR=20
G2 X40.704 Z-56.29 CR=20
G2 X44 Z-60 CR=5
G1 Z-66
X45
```

Screen 2:

```
NC:\MPF\1. 19
RR_E: ; ***** CONTOUR ENDS *****
;
==eof==
```

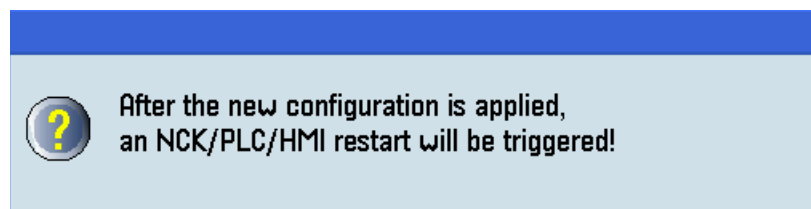
Operating sequence for programming in ISO mode



1. Select the system data operating area.



2. Press this softkey and the system prompts the following window:



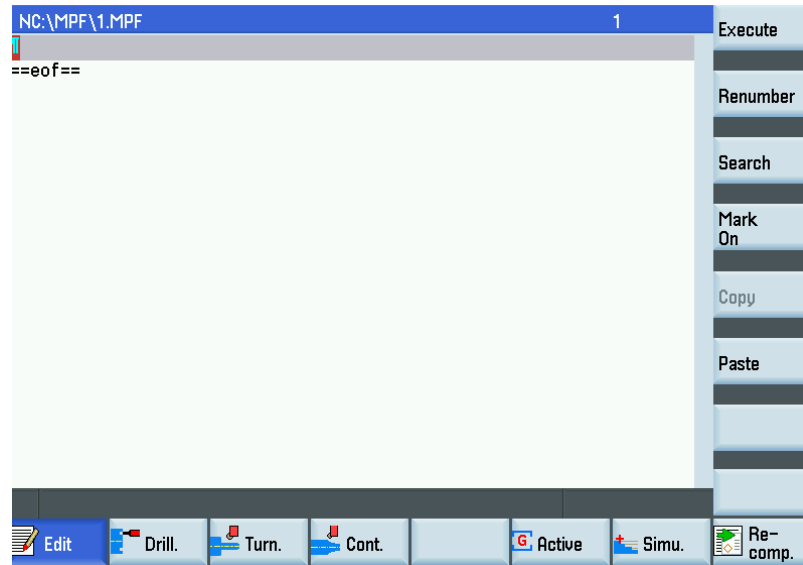
3. Press this softkey and control system automatically starts the mode change from Siemens mode to ISO mode.



4. Select the program management operating area.



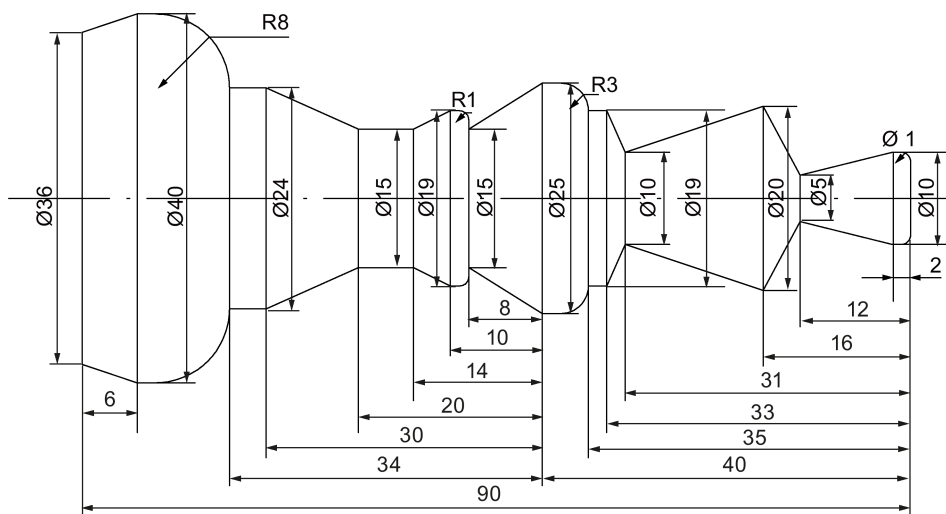
5. Press this softkey to enter the system directory for storing part programs.
6. Press this softkey and enter the name of the new program.
7. Press this softkey to confirm your entry. The part program editor window opens automatically.



8. Enter the following program:
 G291
 T0202
 G0 X100 Z100
 M3 S1000
 X46 Z3
 G71 U1 R1
 G71 P10 Q20 U0.2 W0.1 F0.2
 N10 G0 X0 Z3
 G1Z0
 G3 X33.762 Z-30.725 R20
 G2 X40.704 Z-56.29 R20
 G3 X44 Z-60 R5
 G1 Z-66
 N20 X45
 M3 S2000
 G70 P10 Q20 F0.1
 G0 X100 Z100
 M30
 Then you can proceed with the operations for "program simulation and execution (Page 241)".

13.3 Programming for machining contours (Example 2)

Workpiece drawing (unit: mm)



Blank data

Blank material: aluminum bar

Blank diameter: 45 mm

Blank length: 150 mm (machining limit along the Z axis: Z-95; three-jaw chuck clamping length: 40 mm)

Required tool

Tool type	Tool number		Roughing		Finishing	
	DIN	ISO	Speed	Feedrate	Speed	Feedrate
35 ° finishing tool	T2D1	T0202	S1000	F0.2	S2000	F0.1

Note

- When programming in ISO mode, you need to set the H number of T2 to 2 in the tool list;
- Cut the workpiece manually after machining is over.

Operating sequence for programming in Siemens mode



1. Select the program management operating area.



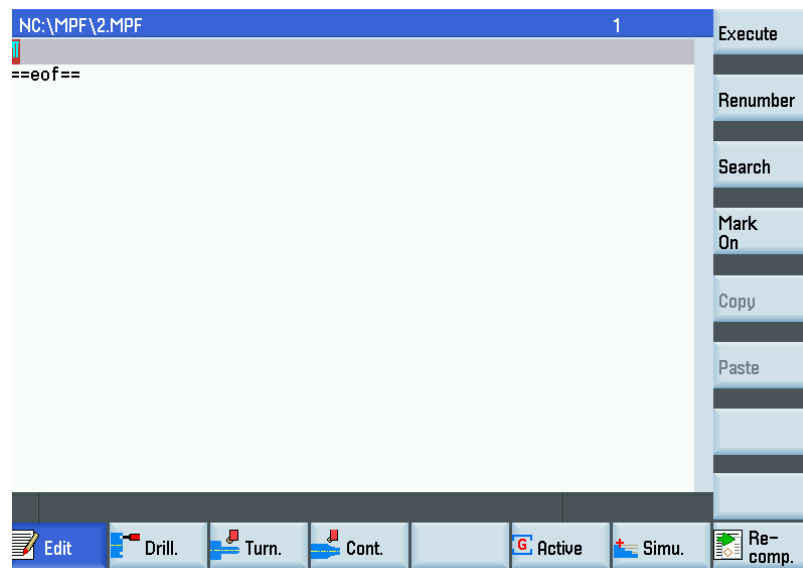
2. Press this softkey to enter the system directory for storing part programs.



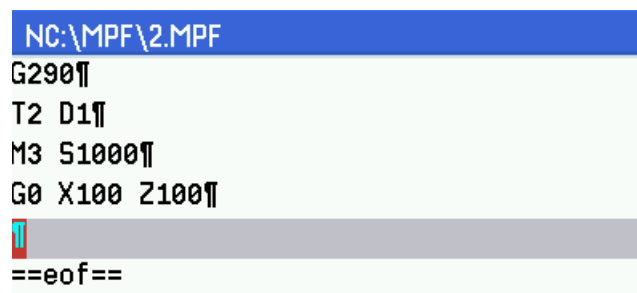
3. Press this softkey and enter the name of the new program.



4. Press this softkey to confirm your entry. The part program editor window opens automatically.



5. Enter the following program in the window. The control system saves your editing automatically.



6. Open the vertical softkey bar for available turning cycles.



7. Press this softkey to open the window for CYCLE95 and enter the name of the contour subroutine in "NPP", for example, "WW".

Attach
contour

8. Press this softkey to open the program editor window again and enter the program blocks for the external contour:

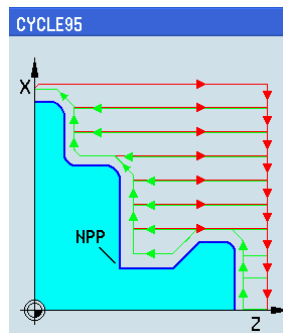
```
G0 Z0 X0
G1 X10 RND=1
Z-2
Z-12 X5
Z-16 X20
Z-31 X10
Z-33 X19
Z-35
X25 RND=3
Z-40
Z-48 X15
X19 RND=1
Z-50
Z-54 X15
Z-60
Z-70 X24
Z-74
X40 RND=8
Z-84
Z-90 X36
Z-95
X45
```

Note:

Make sure you enter the blank diameter in the last line of the program for the external contour (for example, "X45" in the above program). Besides, the program for the external contour has to be positioned after "M30". For detailed information, see the screenshots of the complete program at the end of this example.

9. Press this softkey to return to the window for CYCLE95 and enter the desired parameters as follows:

CYCLE95		Retract. path from contour, incremental
NPP	UW: UW_E	
MID	1.00000	
FALZ	0.00000	
FALX	0.15000	
FAL	0.00000	
FF1	0.20000	
FF2	0.10000	
FF3	0.10000	
UARI	1	
DT	0.00000	
DAM	0.00000	
_URT	2.00000	



10. Press this softkey to confirm your input and open the cycle programming window.

OK

11. Enter the following program block:

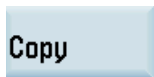
```
M3 S2000
```

12. Press this softkey to return to the program editor window.

Edit

13. Move the cursor to "C" in the line for CYCLE95 with the cursor keys and press this softkey to insert a marker.

Mark
On



14. Use the cursor keys to select the entire program block for CYCLE95. Press this softkey to copy the selection to the buffer memory.



15. Place the cursor on the end-of-block character after "M3 S2000" with cursor keys and press this key on the MCP to enter a new line.



16. Paste the content of the buffer memory into the new line.



17. Press this softkey to open the window for CYCLE95 again. Move the cursor to the input field of "VARI".



18. Press this key to change the value to "5".



19. Press this softkey to accept the change and continue to enter the following program blocks:
G0 X100 Z100
M05
M30

Then you can proceed with the operations for "program simulation and execution (Page 241)".

Screenshots of the complete program

Screen 1:

```
NC:\MPF\2. 1
290
T2 D1
13 S1000
G0 X100 Z100
CYCLE95("UJ:UJ_E", 1, 0, 0.15, 0, 0.2, 0.1, 0.1, 1, 0, 0, 2)
13 S2000
CYCLE95("UJ:UJ_E", 1, 0, 0.15, 0, 0.2, 0.1, 0.1, 5, 0, 0, 2)
G0 X100 Z100
105
130
; *****CONTOUR*****
UJ:
G0 Z0 X0
G1 X10 RND=1
Z-2
Z-12 X5
Z-16 X20
```

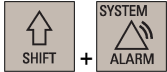
Screen 2:

```
NC:\MPF\2. 19
31 X10
Z-33 X19
Z-35
<25 RND=3
Z-40
Z-48 X15
<19 RND=1
Z-50
Z-54 X15
Z-60
Z-70 X24
Z-74
<40 RND=8
Z-84
Z-90 X36
Z-95
<45
;
```

Screen 3:

```
NC:\MPF\2. 37
UJ_E: ***** CONTOUR ENDS *****
;
==eof==
```

Operating sequence for programming in ISO mode



ISO mode

1. Select the system data operating area.
2. Press this softkey and the system prompts the following window:



After the new configuration is applied,
an NCK/PLC/HMI restart will be triggered!



3. Press this softkey and control system automatically starts the mode change from Siemens mode to ISO mode.



4. Select the program management operating area.



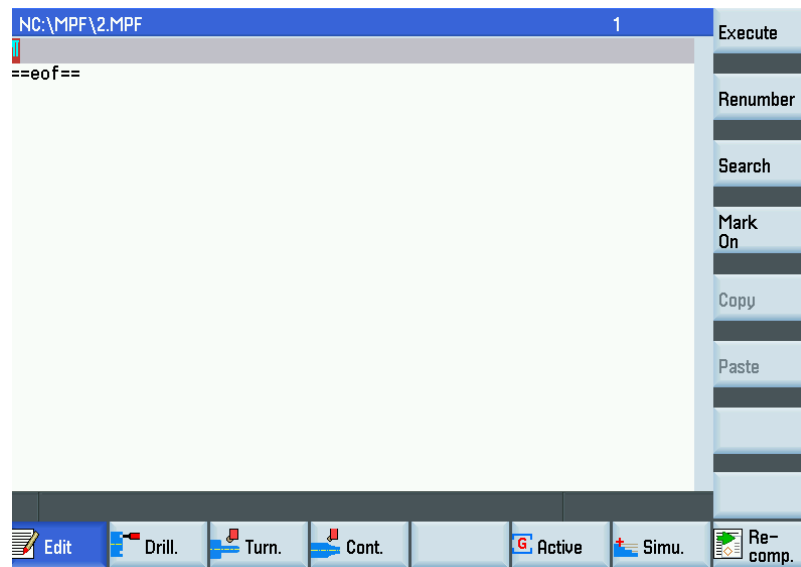
5. Press this softkey to enter the system directory for storing part programs.



6. Press this softkey and enter the name of the new program.



7. Press this softkey to confirm your entry. The part program editor window opens automatically.



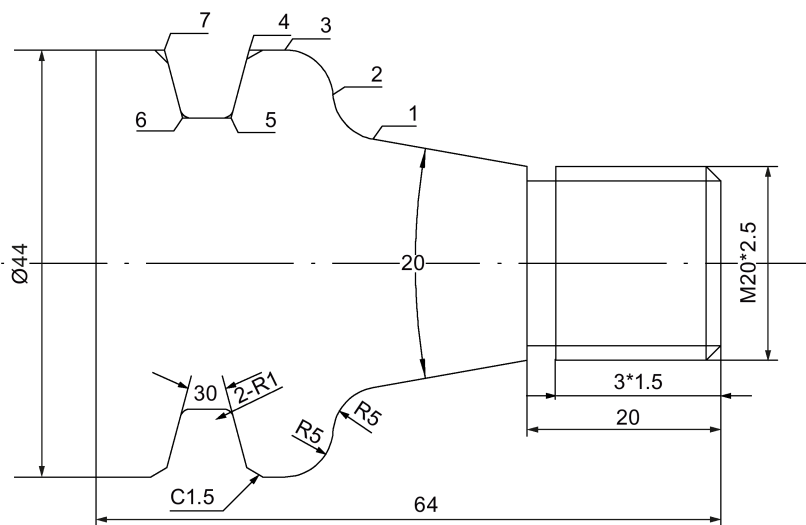
8. Enter the following program:

```
G291
T0202
M3 S1000
G0 X100 Z100
G0 X45 Z3
G71 U2 R1
G71 P10 Q20 U0.15 W0 F0.2
N10 G0 X0 Z3
G1 Z0
X8
G3 X10 Z-1 R1
G1 Z-2
Z-12 X5
Z-16 X20
Z-31 X10
Z-33 X19
Z-35
G3 Z-38 X25 R3
G1 Z-40
Z-48 X15
X17
G3 X19 Z-49 R1
G1 Z-50
Z-54 X15
Z-60
Z-70 X24
Z-74
G3 Z-82 X40 R8
G1 Z-84
Z-90 X36
Z-95
N20 X45
M3 S2000
G70 P10 Q20 F0.1
G0 X100 Z100
M05
M30
```

Then you can proceed with the operations for "program simulation and execution (Page 241)".

13.4 Programming for roughing, grooving and threading (Example 3)

Workpiece drawing (unit: mm)



Coordinates of key machining points

Point	X coordinate	Z coordinate
1	25.596	-35.868
2	34.722	-39.987
3	44.000	-44.974
4	44.000	-48.624
5	30.000	-50.500
6	30.000	-55.500
7	44.000	-57.376

Blank data

Blank material: aluminum bar

Blank diameter: 45 mm

Blank length: 100 mm (machining limit along the Z axis: Z-68; three-jaw chuck clamping length: 25 mm)

Required tools

Tool type	Tool number		Roughing		Finishing	
	DIN	ISO	Speed	Feedrate	Speed	Feedrate
90 ° roughing tool	T1D1	T0101	S1000	F0.2	S2000	F0.1
3 mm grooving tool	T3D1	T0303	S600	F0.1	S600	F0.05
External threading tool	T4D1	T0404	S600	-	S600	-

Note

- When programming in ISO mode, you need to set the H numbers of T1, T3, and T4 to 1, 3, and 4 in the tool list respectively. Three tools are required for the recommended program in ISO mode given later. You need to measure them before machining. M00 is to stop program execution after cutting is over for each tool. To continue executing the program, you need to press the following key on the MCP:



- Cut the workpiece manually after machining is over.

Operating sequence for programming in Siemens mode



1. Select the program management operating area.



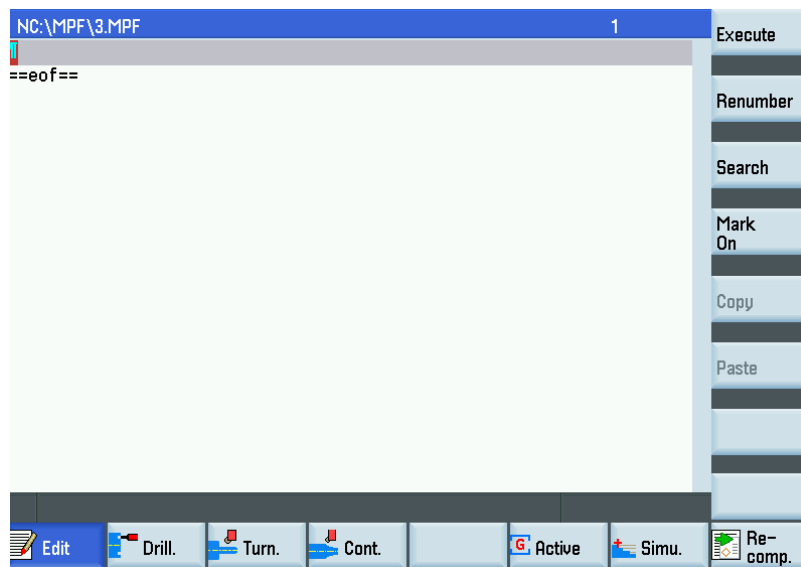
2. Press this softkey to enter the system directory for storing part programs.



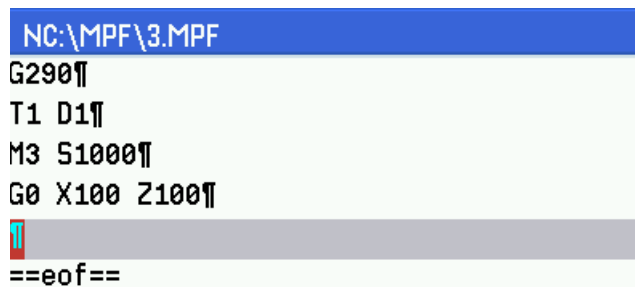
3. Press this softkey and enter the name of the new program.



4. Press this softkey to confirm your entry. The part program editor window opens automatically.



5. Enter the following program in the window. The control system saves your editing automatically.



6. Open the vertical softkey bar for available turning cycles.



7. Press this softkey to open the window for CYCLE95 and enter the name of the contour subroutine in "NPP", for example, "WW".

Attach
contour

8. Press this softkey to open the program editor window again and enter the program blocks for the external contour:

```
G1 X0 Z0
X20 CHF=1.5
Z-20
X25.596 Z-35.868
G2 X34.722 Z-39.987 CR=5
G3 X44 Z-44.974 CR=5
G1 Z-68
X45
```

Note:

Make sure you enter the blank diameter in the last line of the program for the external contour (for example, "X45" in the above program). Besides, the program for the external contour has to be positioned after "M30". For detailed information, see the screenshots of the complete program at the end of this example.

Tech
interface

9. Press this softkey to return to the window for CYCLE95 and enter the desired parameters as follows:

CYCLE95		Retract. path from contour, incremental	
NPP	UW: UW_E		
MID	1.00000		
FALZ	0.00000		
FALX	0.15000		
FAL	0.00000		
FF1	0.20000		
FF2	0.10000		
FF3	0.10000		
VARI	1		
DT	0.00000		
DAM	0.00000		
_URT	2.00000		

10. Press this softkey to confirm your input and open the cycle programming window.

11. Enter the following program block:
M3 S2000

12. Press this softkey to return to the program editor window.

13. Move the cursor to "C" in the line for CYCLE95 with the cursor keys and press this softkey to insert a marker.

14. Use the cursor keys to select the entire program block for CYCLE95. Press this softkey to copy the selection to the buffer memory.

15. Place the cursor on the end-of-block character after "M3 S2000" with cursor keys and press this key on the MCP to enter a new line.

16. Paste the content of the buffer memory into the new line.

17. Press this softkey to open the window for CYCLE95 again. Move the cursor to the input field of "VARI".

18. Press this key to change the value to "5".

OK

Edit

Mark
On

Copy

INPUT

Paste

Re-
comp.

SELECT



19. Press this softkey to accept the change and continue to enter the following program blocks:

```
G0 X100 Z100
M00
T3 D1
M3 S600
G0 X100 Z100
Z-20
X22
G1 X17 F0.1
X20 F0.5
G0 X50
```



20. Open the vertical softkey bar for available turning cycles.



21. Press this softkey to open the window for CYCLE93. Enter the desired parameters as follows:

CYCLE93		Variable retraction path for grooving	
SPD	44.00000		
SPL	-48.62400		
WIDG	5.00000		
DIAG	7.00000		
STA1	0.00000		
ANG1	15.00000		
ANG2	15.00000		
RCO1	-0.50000		
RCO2	-0.50000		
RCI1	0.50000		
RCI2	0.50000		
FAL1	0.10000		
FAL2	0.10000		
IDEP	2.00000		
DTB	0.00000		
VARI	5		
_VRT	1.00000		



22. Press this softkey to confirm your input.

23. Enter the following program blocks:

```
G0 X100 Z100
M00
T4 D1
M3 S600
G0 X100 Z100
```

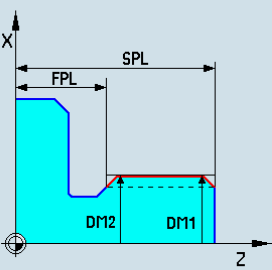


24. Open the vertical softkey bar for thread cutting cycles.

Thread
long.

25. Open the window for CYCLE99. Enter the desired parameters as follows:

CYCLE99		No modification of the internal parameters.	
SPL	0.00000		
DM1	20.00000		
FPL	-17.00000		
DM2	20.00000		
APP	1.00000		
ROP	1.50000		
TDEP	1.40000		
FAL	0.10000		
LANG	0.00000		
NSP	0.00000		
NRC	10		
NID	2		
PIT	2.50000		
UARI	300103		
NUMTH	1		
_URT	2.00000		
PSYS	0		
PSYS	0		



Note:

Set the parameters visible only with the scroll bar to their default values.

26. Continue to enter the following program blocks:

G0 X100 Z100

M05

M30

Then you can proceed with the operations for "program simulation and execution (Page 241)".

Screenshots of the complete program

Screen 1:

```

NC:\MPF\3. 1
O290
T1 D1
S1000
G0 X100 Z100
CYCLE95( "UJ:UJ_E", 1, 0, 0.15, 0, 0.2, 0.1, 0.1, 1, 0, 0, 2)
S2000
CYCLE95( "UJ:UJ_E", 1, 0, 0.15, 0, 0.2, 0.1, 0.1, 5, 0, 0, 2)
G0 X100 Z100
M00
T3 D1
S600
G0 X100 Z100
Z-20
K22
G1 X17 F0.1
K20 F0.5
G0 X50
CYCLE93( 44, -48.624, 5, 7, 0, 15, 15, -0.5, -0.5, 0.5, 0.5, 0.1, 0.

```

Screen 2:

```

NC:\MPF\3. 18
G1, 2, 0, 5, 1)
G0 X100 Z100
M00
T4 D1
S600
G0 X100 Z100
CYCLE99( 0, 20, -17, 20, 1, 1.5, 1.4, 0.1, 0, 0, 10, 2, 2.5, 300103,
1, 2, 0, 0, 0, 0, 0, 0, 1, , , 0)
G0 X100 Z100
M05
M30
; *****CONTOUR*****
JU:
G1 X0 Z0
K20 CHF=1.5
Z-20
K25.596 Z-35.868

```

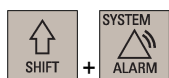
Screen 3:

```

NC:\MPF\3 35
2 X34.722 Z-39.987 CR=5
3 X44 Z-44.974 CR=5
1 Z-68
45
JW_E:; ***** CONTOUR ENDS *****
=eof=

```

Operating sequence for programming in ISO mode



1. Select the system data operating area.



2. Press this softkey and the system prompts the following window:



After the new configuration is applied,
an NCK/PLC/HMI restart will be triggered!



3. Press this softkey and control system automatically starts the mode change from Siemens mode to ISO mode.



4. Select the program management operating area.



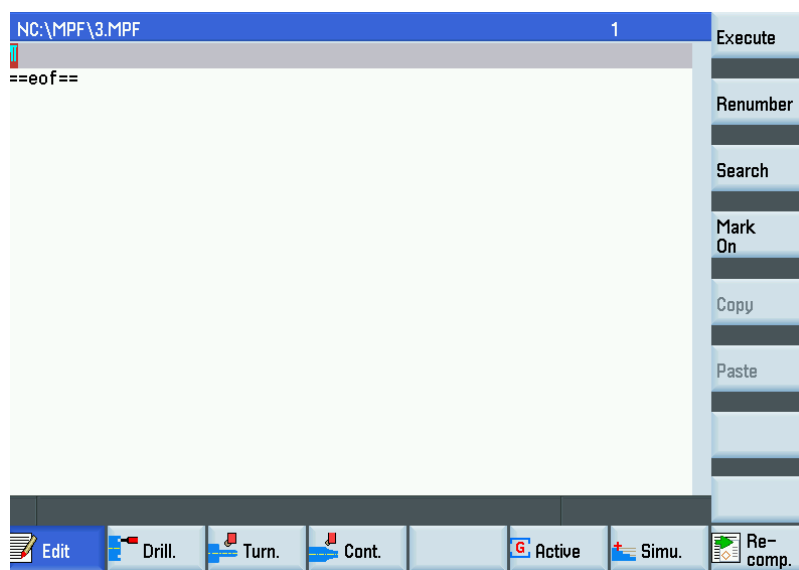
5. Press this softkey to enter the system directory for storing part programs.



6. Press this softkey and enter the name of the new program.



7. Press this softkey to confirm your entry. The part program editor window opens automatically.



8. Enter the following program:
- ```

G291
T0101
M3 S1000
G0 X100 Z100
G0 X46 Z3
G71 U2 R1
G71 P10 Q20 U0.15 W0.1 F0.2
N10 G0 X0 Z3
G1 Z0
X17
X20 Z-1.5
Z-20
X25.596 Z-35.868
G2 X34.722 Z-39.987 R5
G3 X44 Z-44.974 R5
G1 Z-68
N20 X45
M3
S2000
G70 P10 Q20 F0.1
G0 X100 Z100
M00
T0303
M3 S600
G0 X100 Z100
Z-20
X22
G1 X17 F0.1
X20 F0.5

```

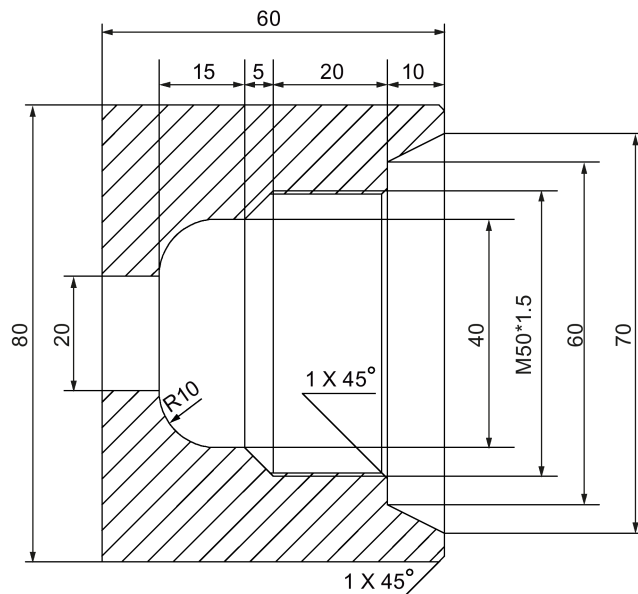
```

G0 X50
Z-54.27
X46
G1 X30 F0.1
G0 X46
Z-54.73
G1 X30 F0.1
G0 X46
Z-58.38
G1 X44 F0.1
X41.67 Z-57.01
X31.48 Z-55.7
G3 X30 Z-54.73 R1
G1 Z-54.7
G0 X46
Z-50.21
G1 X44 F0.1
X42.07 Z-51.88
X31.48 Z-53.3
G2 X30 Z-54.27 R1
G1 Z-54.3
G0 X46
G0 X100 Z100
M00
T0404
M3 S600
G0 X100 Z100
G0 X21 Z3
G92 X20 Z-18 F2.5
X19.5
X19
X18.7
X18.4
X18.1
X17.8
X17.5
X17.3
X17.2
X17.2
G0 X100 Z200
M05
M30
G0 X100 Z100
M05
M30
Then you can proceed with the operations for "program simulation and execution
(Page 241)".

```

### 13.5 Programming for roughing and finishing (Example 4)

**Workpiece drawing (unit: mm)**



### Blank data

Blank material: aluminum bar

Blank diameter: 45 mm

Diameter of the bottom hole: 18 mm

Blank length: 150 mm (machining limit along the Z axis: Z-95; three-jaw chuck clamping length: 40 mm)

## Required tools

| Tool type               | Tool number |       | Roughing |          | Finishing |          |
|-------------------------|-------------|-------|----------|----------|-----------|----------|
|                         | DIN         | ISO   | Speed    | Feedrate | Speed     | Feedrate |
| Turning tool for boring | T1D1        | T0101 | S1000    | F0.2     | S2000     | F0.1     |
| Internal threading tool | T2D1        | T0202 | S1000    | F0.2     | S600      | F1.5     |

### Note

- When programming in ISO mode, you need to set the H numbers of T1 and T2 to 1 and 2 respectively;
- Cut the workpiece manually after machining is over.

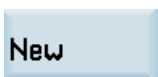
### Operating sequence for programming in Siemens mode



1. Select the program management operating area.



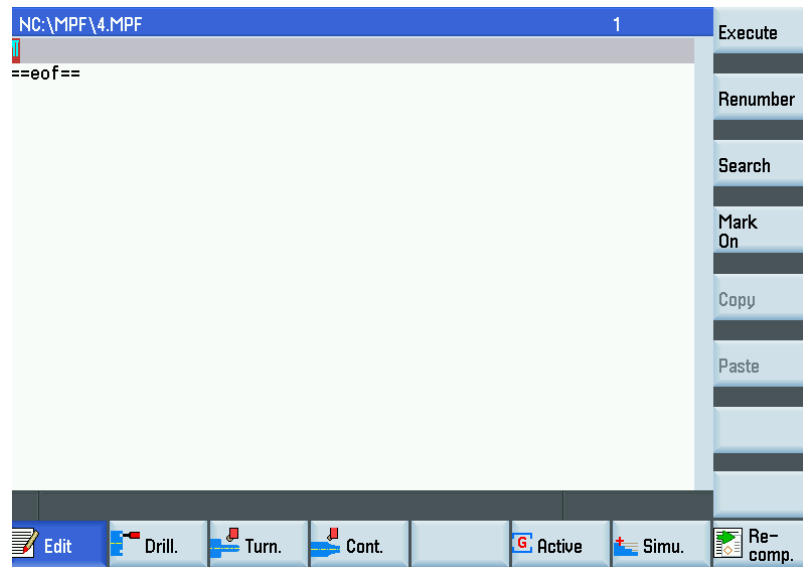
2. Press this softkey to enter the system directory for storing part programs.



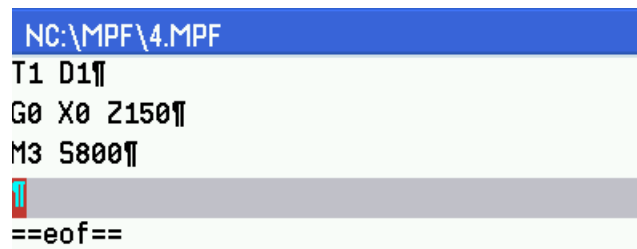
- Press this softkey and enter the name of the new program.



4. Press this softkey to confirm your entry. The part program editor window opens automatically.



5. Enter the following program in the window. The control system saves your editing automatically.



6. Open the vertical softkey bar for available turning cycles.



7. Press this softkey to open the window for CYCLE95 and enter the name of the contour subroutine in "NPP", for example, "LL".



8. Press this softkey to open the program editor window again and enter the program blocks for the external contour:

```
G1 X70 Z0
X60 Z-10
X48.5 CHR=1
Z-30
X40 Z-35
Z-50 RND=10
X20
Z-61
X18
```

**Note:**

Make sure you enter the bottom hole diameter of the blank in the last line of the program for the external contour (for example, "X18" in the above program). Besides, the program for the external contour has to be positioned after "M30". For detailed information, see the screenshots of the complete program at the end of this example.

Tech  
interface

9. Press this softkey to return to the window for CYCLE95 and enter the desired parameters as follows:

| CYCLE95 |          | Retract. path from contour, incremental |  |
|---------|----------|-----------------------------------------|--|
| NPP     | LL: LL_E |                                         |  |
| MID     | 1.00000  |                                         |  |
| FALZ    | 0.05000  |                                         |  |
| FALX    | 0.10000  |                                         |  |
| FAL     | 0.00000  |                                         |  |
| FF1     | 0.20000  |                                         |  |
| FF2     | 0.10000  |                                         |  |
| FF3     | 0.10000  |                                         |  |
| VARI    | 3        |                                         |  |
| DT      | 0.00000  |                                         |  |
| DAM     | 0.00000  |                                         |  |
| _URT    | 1.00000  |                                         |  |

10. Press this softkey to confirm your input and open the cycle programming window.



11. Enter the following program block below the program line for CYCLE95:  
M3 S1000

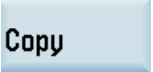
12. Press this softkey to return to the program editor window.



13. Move the cursor to "C" in the line for CYCLE95 with the cursor keys and press this softkey to insert a marker.



14. Use the cursor keys to select the entire program block for CYCLE95. Press this softkey to copy the selection to the buffer memory.



15. Place the cursor on the end-of-block character after "M3 S1000" with cursor keys and press this key on the MCP to enter a new line.



16. Paste the content of the buffer memory into the new line.



17. Press this softkey to open the window for CYCLE95 again. Move the cursor to the input field of "VARI".



18. Press this key to change the value to "7".



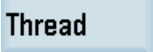
19. Press this softkey to accept the change and continue to enter the following program blocks:  
G0 X0 Z150  
M05  
M00  
T2 D1  
M3 S600



20. Open the vertical softkey bar for available turning cycles.



21. Open the vertical softkey bar for thread cutting cycles.



Thread long.

22. Open the window for CYCLE99. Enter the desired parameters as follows:

| CYCLE99 |           | No modification of the internal parameters. |  |
|---------|-----------|---------------------------------------------|--|
| SPL     | -10.00000 |                                             |  |
| DM1     | 48.50000  |                                             |  |
| FPL     | -29.00000 |                                             |  |
| DM2     | 48.50000  |                                             |  |
| APP     | 2.00000   |                                             |  |
| ROP     | 1.00000   |                                             |  |
| TDEP    | 0.90000   |                                             |  |
| FAL     | 0.05000   |                                             |  |
| IANG    | 0.00000   |                                             |  |
| NSP     | 0.00000   |                                             |  |
| NRC     | 6         |                                             |  |
| NID     | 2         |                                             |  |
| PIT     | 1.50000   |                                             |  |
| UARI    | 300104    |                                             |  |
| NUMTH   | 1         |                                             |  |
| _URT    | 2.00000   |                                             |  |
| PSYS    | 0         |                                             |  |
| PSYS    | 0         |                                             |  |

**Note:**

Set the parameters visible only with the scroll bar to their default values.

23. Continue to enter the following program blocks:

G0 X0 Z150

M05

M30

Then you can proceed with the operations for "program simulation and execution (Page 241)".

**Screenshots of the complete program**

Screen 1:

```

NC:\MPF\4. 1
1 D1
30 X0 Z150
13 S000
CYCLE95("LL: LL_E", 1.00000, 0.05000, 0.10000, 0.00000, 0.20000, 0.1
3000, 0.10000, 3, 0.00000, 0.00000, 1.00000)
13 S1000
CYCLE95("LL: LL_E", 1.00000, 0.05000, 0.10000, 0.00000, 0.20000, 0.1
3000, 0.10000, 7, 0.00000, 0.00000, 1.00000)
30 X0 Z150
105
100
12 D1
13 S000
CYCLE99(-10.00000, 48.50000, -29.00000, 48.50000, 2.00000, 1.00000,
0.90000, 0.05000, 0.00000, 0.00000, 6, 2, 1.50000, 300104, 1, 2.000
00, 0, 0, 0, 0, 0, 0, 0, 1, , , , 0)
30 X0 Z150
105

```

Screen 2:

```

NC:\MPF\4. 15
30
*****CONTOUR*****
LL:
31 X70 Z0
60 Z-10
48.5 CHR=1
2-30
40 Z-35
2-50 RND=10
20
2-61
18
LL_E; ***** CONTOUR ENDS *****
==eof==

```

**Operating sequence for programming in ISO mode**



ISO mode

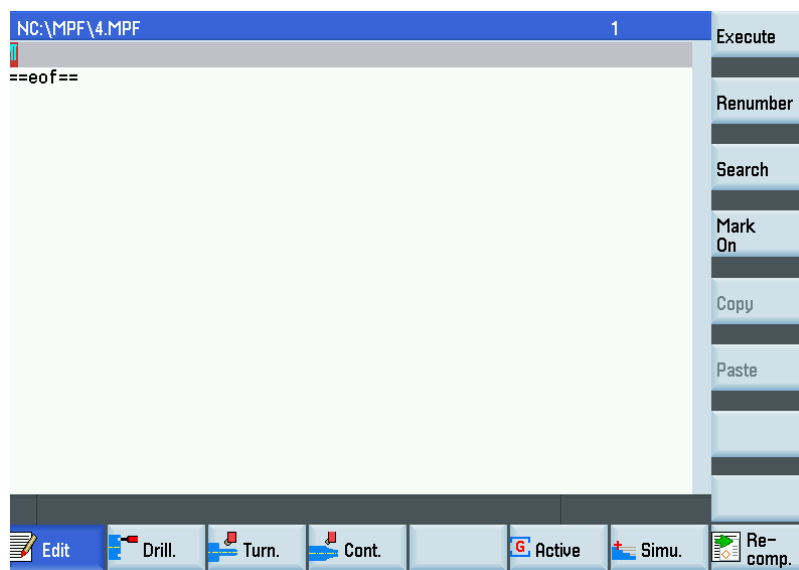
1. Select the system data operating area.
2. Press this softkey and the system prompts the following window:



After the new configuration is applied,  
an NCK/PLC/HMI restart will be triggered!



3. Press this softkey and control system automatically starts the mode change from Siemens mode to ISO mode.
4. Select the program management operating area.
5. Press this softkey to enter the system directory for storing part programs.
6. Press this softkey and enter the name of the new program.
7. Press this softkey to confirm your entry. The part program editor window opens automatically.



8. Enter the following program:
 

```

G291
T0101
M3 S800
G0 X0 Z150
X18 Z3
G71 U2 R1
G71 P10 Q20 U-0.2 W0.05 F0.2
N10 G0 X70 Z3
G1 Z0
X60 Z-10
X50.5
X48.5 Z-11
Z-30
X40 Z-35
Z-40
G3 X20 Z-50 R10
N20 G1 Z-61
M3 S1000

```

```

G70 P10 Q20 F0.1
G0 X0 Z150
M05
M00
T0202
M3 S600
G0 X48
Z-8
G92 X48.5 Z-30 F1.5
X49
X49.3
X49.6
X49.8
X49.9
X50
G0 X0 Z150
M05
M30

```

Then you can proceed with the operations for "program simulation and execution".

## 13.6 Program simulation and execution



1. Switch to "AUTO" mode.



2. Press this softkey to open the program simulation window, and the program control mode PRT is automatically activated.



3. Press this key to start the simulation for the selected part program.



4. After you finish the simulation, you can press this softkey to return to the program editor window.



5. Press this softkey and the system automatically changes to "AUTO" mode in the machining operating area.

6. Make sure the following preconditions are met before start machining:

- These two softkeys have been deactivated.



- The feedrate override is 0%.
- The safety door is closed and the coolant is switched on.

7. Press this key to start workpiece machining.



# A Appendix

## A.1 Operating area overview

### A.1.1 Machining operating area



Pressing this key on the PPU allows you to open the window for the machining operating area. You can perform reference point approach, tool setting operations, as well as program start, stop, control, block search, and real-time simulation, etc. in this operating area.

The machining operating area has different window displays and softkey functions in the following operating modes:

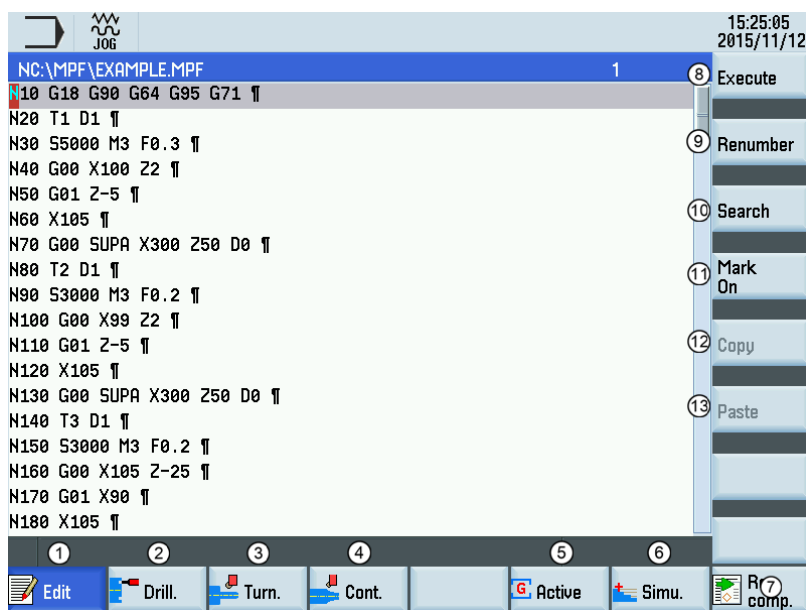
- "JOG" mode (Page 251)
- "AUTO" mode (Page 249)
- "MDA" mode (Page 255)

### A.1.2 Program editing operating area

#### Softkey functions



Pressing this key on the PPU allows you to open the program editor with the last opened part program. If no program is ever opened in the program editor, pressing this key switches to the program management operating area.



- ① Displays and edits the program blocks
- ② Programs drilling cycles and adds to the current program
- ③ Programs turning cycles and adds to the current program
- ④ Opens the contour editor for free contour programming
- ⑤ Calls the dialog box of common programming instructions

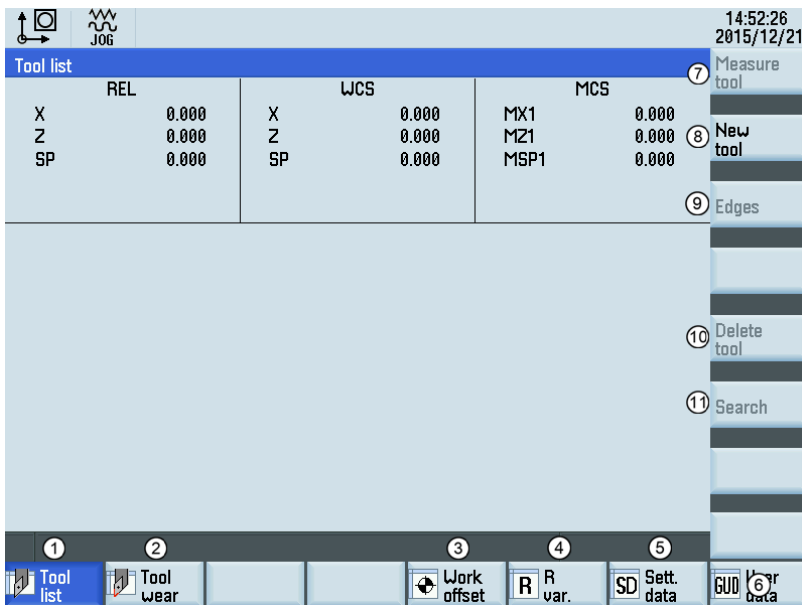
- ⑥ Opens the program simulation window to check the programming results before machining
- ⑦ Recompiles the current cycle or contour selected with cursor by reopening the previous programming window
- ⑧ Executes the current program
- ⑨ Automatically assigns block numbers (Nxx) to all blocks
- ⑩ Opens the block search dialog box
- ⑪ Inserts a marker at the cursor position for copying/deleting the program blocks selected with the cursor
- ⑫ Copies the selected program blocks to the buffer memory
- ⑬ Pastes the copied program blocks at the current cursor position

## A.1.3 Offset operating area

### Softkey functions



Pressing this key on the PPU allows you to open the following window:



| Softkey | Function                                                                                                         | See also                                                                                                                            |
|---------|------------------------------------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------|
| ①       | Displays and modifies the tool offsets                                                                           | <ul style="list-style-type: none"> <li>"Creating a tool (Page 20)"</li> <li>"Creating/changing a cutting edge (Page 22)"</li> </ul> |
| ②       | Displays and modifies the tool wear data                                                                         | "Entering the tool wear offsets (Page 51)"                                                                                          |
| ③       | Displays and modifies the workpiece offsets                                                                      | "Entering/modifying workpiece offsets (Page 57)"                                                                                    |
| ④       | Displays and modifies the R variables                                                                            | "Setting R parameters (Page 63)"                                                                                                    |
| ⑤       | Configures and displays lists of setting data                                                                    | "Entering/modifying the setting data (Page 59)"                                                                                     |
| ⑥       | Displays the defined user data                                                                                   | "Setting user data (Page 63)"                                                                                                       |
| ⑦       | Measures the tool manually (active only in "JOG" mode and when the cursor is on the input field for tool length) | "Measuring the tool manually (with the Y axis) (Page 28)"                                                                           |

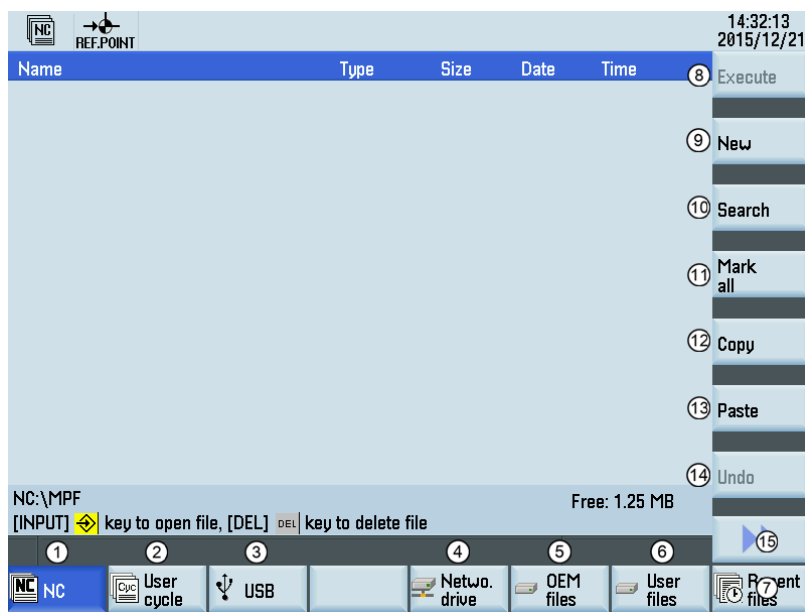
| Softkey | Function                                               | See also                                     |
|---------|--------------------------------------------------------|----------------------------------------------|
| ⑧       | Creates a new tool                                     | "Creating a tool (Page 20)"                  |
| ⑨       | Opens a lower-level menu for cutting edge settings     | "Creating/changing a cutting edge (Page 22)" |
| ⑩       | Removes the currently selected tool from the tool list | -                                            |
| ⑪       | Searches for your desired tool with the tool number    | -                                            |

## A.1.4 Program management operating area

### Softkey functions



Pressing this key on the PPU allows you to open the following window:



- ① Stores the NC programs for subsequent operations
- ② Manages and transfers the manufacturer cycles
- ③ Reads in/out files via the USB drive and executes part programs from the external storage media
- ④ Reads in/out files via the Ethernet interface and executes part programs from a computer
- ⑤ Backs up manufacturer files
- ⑥ Backs up user files
- ⑦ Shows the recently accessed files
- ⑧ Executes the selected file. No editing is allowed in the execution process.
- ⑨ Creates new files or directories
- ⑩ Searches for files
- ⑪ Selects all files for the subsequent operations
- ⑫ Copies the selected file(s) to the clipboard
- ⑬ Pastes the selected file(s) from the clipboard to the current directory

- ⑭ Restores the deleted file(s)
- ⑮ Opens the lower-level menu for more options:
  - Rename the part programs
  - Cut the part programs

Note: Softkeys ② and ⑤ are visible only with the manufacturer password.

### Searching for programs



1. Select the program management operating area.



2. Select the storage directory in which you wish to perform the search.

**Note:**

The following two folders are visible only with the manufacturer password:



3. Press this vertical softkey to open the search window.

4. Enter the complete name with extension of the program file to be searched in the first input field in the search window. To narrow your search, you can enter the desired text in the second field.

5. Use this key to choose whether to include subordinate folders or observe upper/lower case.



6. Press this softkey to start the search.



### Copying, cutting, and pasting programs



1. Select the program management operating area.

2. Open the desired directory.

3. Select the program file that you would like to copy or cut.

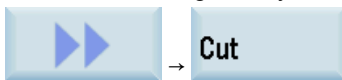


4. Perform either of the following operations as desired:

- Press this softkey to copy the selected file:



- Press the following softkeys to cut the selected file:



5. Select the target directory with the horizontal softkeys.
6. Press this softkey to paste the file from the clipboard to the current directory.



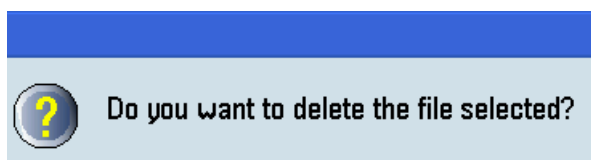
### Deleting/restoring programs



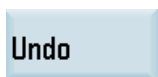
1. Select the program management operating area.
2. Open the desired directory.
3. Select the program file that you would like to delete.



4. Press this key, and the following message appears on the screen:



5. Press this softkey to confirm the deletion.



6. If you want to restore the last deleted file(s), press this softkey.

### Renaming programs



1. Select the program management operating area.
2. Open the desired directory.
3. Select the program file that you would like to rename.



4. Press the extension softkey.



5. Press this vertical softkey to open the window for renaming.

6. Enter a desired new name with the extension in the input field.
7. Press this softkey to confirm your entry.



## Viewing and executing recent programs



1. Select the program management operating area.



2. Press this softkey to open the list of recent files. Note that even the deleted files are also displayed in the list.



3. Select the program file that you would like to execute.



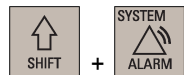
4. Press this vertical softkey to start executing the selected program.



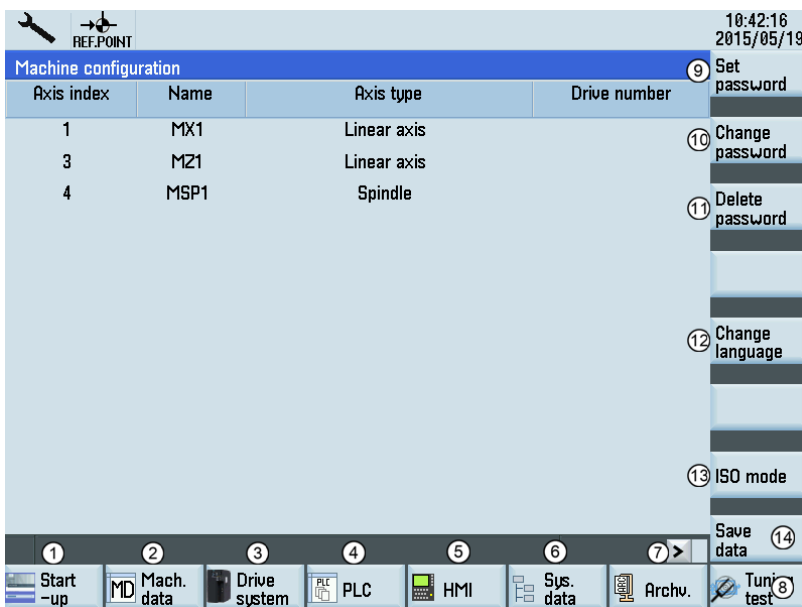
To clear the current file list, press this softkey.

## A.1.5 System data operating area

### Softkey functions



Pressing this key combination allows you to enter the system data operating area. This operating area includes functions required for parameterizing and analyzing the NCK, the PLC, and the drive. The start screen displays the machine configuration data and softkeys available. Depending on the functions selected, the horizontal and the vertical softkey bars vary. The screenshot below uses the turning variant of the control system with PPU161.3 as an example.



- ① Sets the NC, PLC and HMI start up modes
- ② Sets the system machine data
- ③ Configures the connected drives and motors (PPU16x.3 only)
- ④ Provides PLC commissioning and diagnostics
- ⑤ Sets the system date and time and adjusts the brightness of the screen
- ⑥ Backs up and restores system data
- ⑦ Creates and restores startup archives, data archive

- ⑧ Performs the axis optimization (PPU16x.3 only)
- ⑨ Enters the corresponding password (manufacturer password, and end user password) for different access levels
- ⑩ Changes the password as per the corresponding access levels
- ⑪ Deletes the current password
- ⑫ Selects the user interface language. Note that the HMI is automatically restarted when a new language is selected.
- ⑬ Switches to the ISO programming mode
- ⑭ Saves the contents of the volatile memory into a non-volatile memory area

Note: Softkeys ③, ⑧, and ⑬ are visible only with the manufacturer password.



An extended horizontal softkey bar can be accessed via this key on the PPU. Two extended horizontal softkeys are provided:



Views the service information



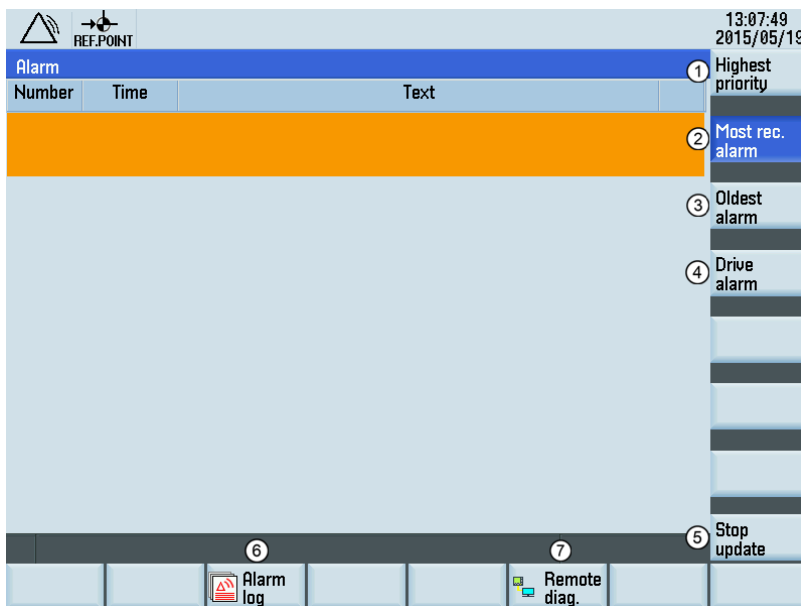
Defines the maintenance planner

## A.1.6 Alarm operating area

### Softkey functions



Press this key on the PPU to enter the alarm operating area. You can check the NC and drive alarms using the softkeys. PLC alarms are not sorted.



- ① Displays all alarms sorted by their priorities. The highest priority alarm is at the beginning of the list.
- ② Displays the alarms sorted by the time of their occurrence. The most recent alarm stands at the beginning of the list.
- ③ Displays the alarms sorted by the time of their occurrence. The oldest alarm stands at the beginning of the list.
- ④ Displays the alarms on the drives. Note that this softkey is available on the PPU16x.3 only.

- ⑤ Stops/starts updating of pending alarms
- ⑥ Views and manages the alarm log
- ⑦ Configures the access right for the remote control through the Ethernet connection. For more information about the softkey function, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Commissioning Manual.

## A.2 Operating mode overview

### A.2.1 "AUTO" mode

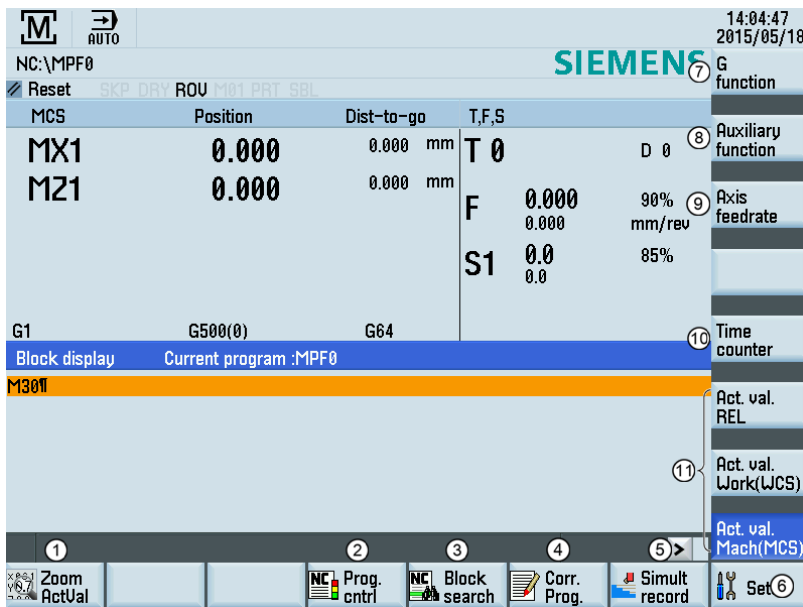
#### Overview

The machine must have been set up for "AUTO" mode according to the specifications of the machine manufacturer. You can perform such operations as program start, stop, control, block search, and real-time simulation and recording, etc.

#### Softkey functions



Pressing this key in the machining operating area switches to "AUTO" mode.



- ① Zooms in the actual value window
- ② Performs the program test, dry run, conditional stop, block skipping, and auxiliary function lock
- ③ Finds the desired block location
- ④ Corrects a wrong program block. Any changes will be stored immediately.
- ⑤ Activates the simulation function
- ⑥ Sets the frequently used setting data
- ⑦ Displays important G functions
- ⑧ Displays currently active auxiliary and M functions
- ⑨ Displays the axis feedrate in the selected coordinate system
- ⑩ Displays the information of part machining time (part timer) and part counter

- ⑪ Switches over the coordinate system in the actual value window



An extended horizontal softkey bar can be accessed via this key on the PPU. The following extended horizontal softkey is provided:

#### Handwheel

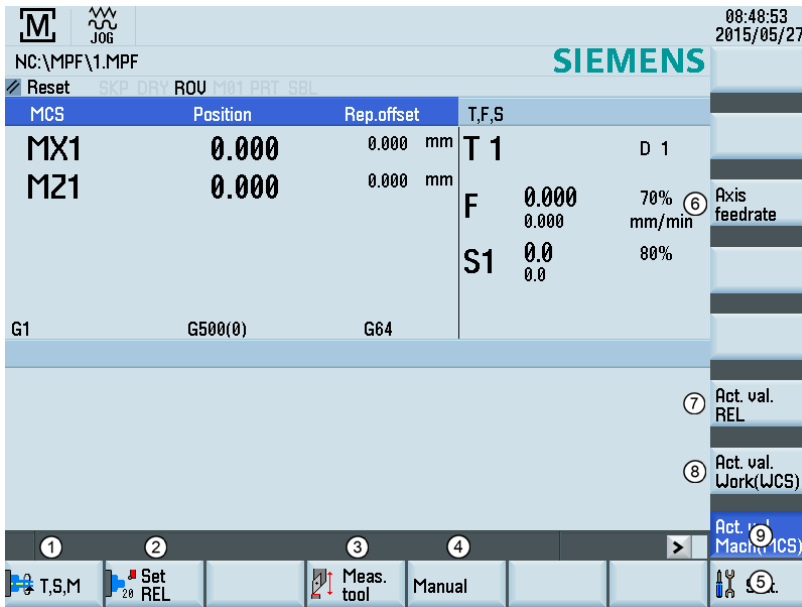
Assigns and activates the handwheel or contour handwheel. For more information, refer to Section "Assigning the handwheel through the MCP (Page 24)" and Appendix "Activating the contour handwheel via the NC program (Page 256)".

#### Parameters

| MCS                                             | Position       | Dist-to-go      |
|-------------------------------------------------|----------------|-----------------|
| <b>MX1</b>                                      | <b>0.000</b>   | <b>0.000 mm</b> |
| <b>MZ1</b>                                      | <b>0.000</b>   | <b>0.000 mm</b> |
| ①                                               | ②              | ③               |
| <b>G1</b>                                       | <b>G500(0)</b> | <b>G64</b>      |
| <b>Block display      Current program :MPF0</b> |                |                 |
| <b>M301</b>                                     |                |                 |
| ④                                               |                |                 |

- ① Displays the axes that exist in the machine coordinate system (MCS), workpiece coordinate system (WCS), or relative coordinate system (REL).
- ② Displays the current position of the axes in the selected coordinate system.
- ③ Displays the remaining distance for the axes to traverse.
- ④ Displays seven subsequent blocks of the currently active part program. The display of one block is limited to the width of the window.

## A.2.2 "JOG" mode



- ① Opens the "T, S, M" window where you can activate tools, set spindle speed and direction (see Section "Activating the tool and the spindle (Page 23)"), and select a G code or other M functions for activating the settable work offset.
- ② Switches the display to the relative coordinate system. You can set the reference point in this coordinate system. For more information, see Section "Setting the relative coordinate system (REL) (Page 254)".
- ③ Opens the tool measurement window where you determine the tool offset data. For more information about this window, see Section "Additional method for measuring the tool (Page 58)".
- ④ Opens the "Manual Machine Plus" user interface. This softkey is visible only if this software option is pre-configured by the machine manufacturer. For more information about this window, see the Manual Machine Plus (MM+), Turning Manual.
- ⑤ Opens the settings window where you can set JOG feedrate and variable increment values. For more information, see Section "Setting the JOG data (Page 254)".
- ⑥ Displays the axis feedrate in the selected coordinate system.
- ⑦ Displays the axis position data in the relative coordinate system.
- ⑧ Displays the axis position data in the workpiece coordinate system.
- ⑨ Displays the axis position data in the machine coordinate system.

### Display of variable increments

The HMI displays the variable increment if it is defined in the standard subroutine program:

|  |                                                               |
|--|---------------------------------------------------------------|
|  | Value ≤ five digits: displays the complete value              |
|  | Value > five digits: displays the first four digits plus ".." |

## Parameters in the "JOG" window

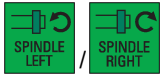
| MCS | Position | Rep.offset | T,F,S                         |
|-----|----------|------------|-------------------------------|
| MX1 | 0.000    | 0.000 mm   | T 1 ④ D 1                     |
| MZ1 | 0.000    | 0.000 mm   | F 0.000 ⑤ 70%<br>0.000 mm/min |
| ①   | ②        | ③          | S1 0.0 ⑥ 80%<br>0.0           |

- ① Displays the axes that exist in the machine coordinate system (MCS), workpiece coordinate system (WCS), or relative coordinate system (REL). If you traverse an axis in the positive (+) or negative (-) direction, a plus or minus sign appears in the relevant field. If the axis is already in the required position, no sign is displayed.
- ② Displays the current position of the axes in the selected coordinate system.
- ③ Displays the distance traversed by each axis in "JOG" mode from the interruption point in the condition of program interruption. For more information about program interruption, see Section "Executing a part program (Page 48)".
- ④ Displays the currently active tool number T with the current tool offset number D.
- ⑤ Displays the actual axis feedrate and the setpoint (mm/min or mm/rev).
- ⑥ Displays the actual value and the setpoint of the spindle speed (r.p.m.).

### A.2.2.1 Running the spindle manually

#### Note

Before running the spindle manually, make sure you have activated the tool.



You can run the spindle manually using these keys on the MCP in "JOG" mode. The spindle speed is selected via MD14512[19].4 as follows:

- MD14512[19].4 = 0 (default): the spindle runs with the speed set in MD32020.
- MD14512[19].4 = 1: the spindle runs with the speed set last time. In this case, if the spindle runs for the first time after the control system powers on, then you must specify a spindle speed in "MDA" mode or the "T, S, M" window.

#### Operating sequence



1. Select the machining operating area.



2. Switch to "JOG" mode.



3. Press this key on the MCP to rotate the spindle counter-clockwisely.

| T,F,S |                 |               |
|-------|-----------------|---------------|
| T 1   |                 | D 1           |
| F     | 0.000<br>0.000  | 80%<br>mm/min |
| S1    | -50.0<br>-100.0 | 50%           |



4. Press this key on the MCP to stop the spindle rotation.

| T,F,S |                |               |
|-------|----------------|---------------|
| T 1   |                | D 1           |
| F     | 0.000<br>0.000 | 80%<br>mm/min |
| S1    | 0.0<br>0.0     | 50%           |



5. Press this key on the MCP to rotate the spindle clockwise.

| T,F,S |                |               |
|-------|----------------|---------------|
| T 1   |                | D 1           |
| F     | 0.000<br>0.000 | 80%<br>mm/min |
| S1    | 50.0<br>100.0  | 50%           |



10. Use this softkey to return to the screen of the machining operating area.

### A.2.2.2 Executing M functions

#### Note

Before executing M functions, make sure all the axes are in safe positions.

#### Operating sequence



1. Select the machining operating area.



2. Switch to "JOG" mode.



3. Open the "T, S, M" window.



4. Press this key to move the cursor to the input field for M functions, and enter the desired value, for example, "8".



5. Press this key on the MCP to activate the coolant function. Now you can see the corresponding status indicator is on, which indicates the coolant supply is switched on.



6. Press this key on the MCP to stop the coolant function.



7. Use this softkey to return to the main screen of the machining operating area.

### A.2.2.3 Setting the relative coordinate system (REL)

#### Operating sequence



1. Select the machining operating area.



2. Switch to "JOG" mode.



3. Press this softkey to switch the display to the relative coordinate system.



4. Use the cursor keys to select the input field, and then enter the new position value of the reference point in the relative coordinate system.

| REL | Position | Rep.offset |
|-----|----------|------------|
| X   | 2.340    | 0.000 mm   |
| Z   | -5.200   | 0.000 mm   |



5. Use this key to activate the values after each entry.

You can use the corresponding vertical softkeys to set the reference point to zero.



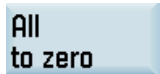
Sets the X axis to zero



Sets the Z axis to zero



Sets the spindle to zero



Sets all axes to zero

### A.2.2.4 Setting the JOG data

#### Operating sequence



1. Select the machining operating area.



2. Switch to "JOG" mode.



3. Press this horizontal softkey to open the following window:

| Settings            |          |        |
|---------------------|----------|--------|
| JOG - Feedrate      | 0.000    | mm/min |
| Variable increment  | 0        | inc    |
| Jog spindle speed   | 0.000    | rpm    |
| Dry run feedrate    | 5000.000 | mm/min |
| Spindle speed       |          |        |
| limitation with G96 | 100.000  | rpm    |



4. Enter values in the input fields and confirm your entries.

Switch  
mm > inch

5. If necessary, press this vertical softkey to switch between the metric and inch dimension systems.



Press this softkey to confirm your changes.



Press this softkey to exit.

### A.2.3 "MDA" mode

#### Basic softkey functions in "MDA" mode



This window displays important G functions whereby each G function is assigned to a group and has a fixed position in the window. To close the window, press this softkey once again.



To display additional G functions, use the corresponding keys.



This window displays the auxiliary and M functions currently active. To close the window, press this softkey once again.



This softkey opens the file saving window where you can specify a name and a storage position for the program displayed in the MDA window. To save your program, either enter a new program name in the input field or select an existing program for overwriting.

**Note:**

If you do not save with this softkey, the program edited in "MDA" mode is actually a temporary file.



Pressing this softkey deletes all the blocks displayed in the MDA window.



This softkey opens a window where you can select an existing program file from a system directory to load into the MDA buffer.

For the explanation of other softkeys in this mode, see Section ""JOG" mode (Page 251)".

#### Notes on part programs in "MDA" mode

Observe the following notes when creating, executing or editing part programs in "MDA" mode:

- **Programs ended with M2/M30/M17**

When a part program which is ended with M2/M30/M17 runs to the end in "MDA" mode, the cursor automatically returns to the beginning of the program and the entire program becomes editable.

- **Programs not ended with M2/M30/M17**

When a part program which is not ended with M2/M30/M17 runs to the end in "MDA" mode, a new empty line automatically appears at the end of the program and the cursor stops at the beginning of the new line. In this case, only this new block line is editable and then executable. All program blocks before the new line, however, are not editable unless you press the following key:



Note that if you do not enter any programming instruction in the new line, pressing the above key deletes the new line and the cursor automatically goes back to the beginning of the program.

- **Programs stopped under other circumstances**

In one of the following cases, the program execution stops and the program is not editable unless you press the above key:

- When a program exception occurs
- When the program runs to M0/M1
- When you press the following key:



## A.3 Activating the contour handwheel via the NC program

The contour handwheel function can help you with the first cutting test after you create a new NC program. When you have activated this function, you can control the feedrate of path and synchronized axes via a handwheel in "AUTO" or "MDA" mode. This section only introduces how to activate the contour handwheel via the NC program. For how to activate this function via the PLC interfaces, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Function Manual.

### Preconditions

- You must not select fixed feedrate, dry-run feedrate, thread cutting, or tapping.

---

#### Note

If you have not programmed the feedrate in the previous blocks, the control system prompts an alarm which indicates the absence of feedrate.

---

- The axes have been referenced. For more information about how to reference the axes, see Chapter "Switching on and referencing (Page 19)".
- The option of "Contour handwheel" is activated with a license key. For more information about how to activate an option, see the SINUMERIK 808D/SINUMERIK 808D ADVANCED Commissioning Manual.
- You have assigned the handwheel to the first geometric axis and activated it. For more information, see Section "Assigning the handwheel through the MCP (Page 24)".

### Activating the contour handwheel in "AUTO" mode

1. Open the desired NC program in the part program editor window. For more information about how to create or edit a part program, see Chapter "Creating part programs (Page 32)".
2. Enter the mandatory instructions "G1" (or "G2"/"G3"), "G94", "G60" and "FD=0" in the blocks. The blocks are saved automatically. See the following example:

```
G1 G60 G94 X20 Z40 F1200
X60 F1200
X200 Z300 FD=0
X0 Z0
M30
==eof==
```

**Note:**

- "FD" and "F" cannot appear in the same NC block; otherwise, the control system prompts an alarm which interrupts the contour handwheel function.
- When the block runs to "FD=0", you can control the feedrate of path and synchronized axes via a handwheel in "AUTO" mode.



3. Press this softkey to execute the program. The system automatically switches to "AUTO" mode in the machining area.

**Activating the contour handwheel in "MDA" mode**



1. Select the machining operating area.



2. Switch to "MDA" mode.

3. Enter the mandatory instructions "G1" (or "G2"/"G3"), "G94", "G60" and "FD=0" in the blocks. The blocks are saved automatically. See the following example:

```
G1 G60 G94 X20 Z40 F1200
X60 F120
X200 Z300 FD=0
X0 Z0
M30
==eof==
```



You can alternatively load an existing part program from a system directory by pressing this softkey if desired and enter "G1" (or "G2"/"G3"), "G94", "G60" and "FD=0" in the blocks.

**Note:**

"FD" and "F" cannot appear in the same NC block; otherwise, the control system prompts an alarm which interrupts the contour handwheel function.



4. Press this key on the MCP. When the block runs to "FD=0", you can control the feedrate of path and synchronized axes via a handwheel in "MDA" mode.

**Traversing direction of axes**

The axis traversing direction depends on the rotation direction of the contour handwheel:

- Clockwise → results in traversing in the programmed direction  
If the block-change criterion is reached, the program advances to the next block (response identical to G60).
- Counterclockwise → results in traversing in a direction opposite to the programmed direction  
Here, the axes can only traverse to the appropriate block start. Pulses are not collected if the handwheel continues to rotate.

---

**Note**

When the axes move against the programmed direction, the program reaches the beginning of each block and no further operation is available.

---

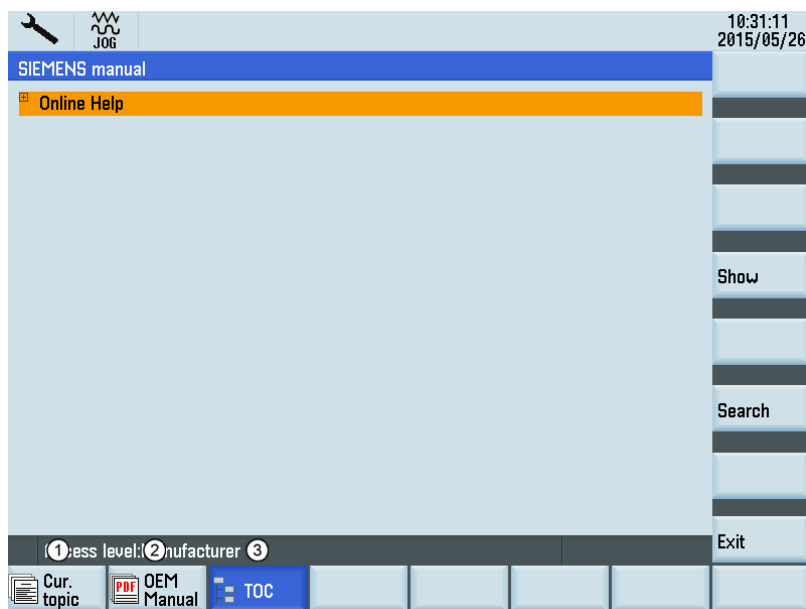
## A.4 The help system

The control system provides comprehensive online help. Whenever necessary, you can call the help system from any operating area.

### The help system



Press this key or the key combination <ALT> + <H> to call the help system from any operating area. If a context-sensitive help exists, Window "①" opens; otherwise, Window "③" opens.



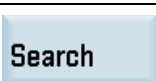
- ① Calls the context-sensitive help for the current topic:
  - Current operating window
  - NC/V70 alarms selected in the alarm specific operation area
  - Machine data or setting data selected
  - V70 data selected
- ② Calls the machine manufacturer-developed PDF manual
- ③ Displays all available help information:
  - Siemens help manuals
  - Machine manufacturer-developed help manuals, if any
  - All available NC/V70 alarms
  - All available V70 parameters
  - All available machine data or setting data

### Softkeys in Window "①"

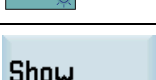
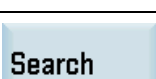
|                 |                                                                                                                                                                                                           |
|-----------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Go to topic     | Use this softkey to select cross references<br>A cross reference is marked by the characters "> ... <".<br><b>Note:</b><br>This softkey is displayed only if the current page contains a cross reference. |
| Search          | Searches for a term in the current topic                                                                                                                                                                  |
| Continue search | Continues search for the next term that matches the search criteria                                                                                                                                       |

|                                                                                   |                       |
|-----------------------------------------------------------------------------------|-----------------------|
|  | Exits the help system |
|-----------------------------------------------------------------------------------|-----------------------|

#### Softkeys in Window "②"

|                                                                                   |                                                                     |
|-----------------------------------------------------------------------------------|---------------------------------------------------------------------|
|  | Zooms in the current view                                           |
|  | Zooms out the current view                                          |
|  | Zooms the current view to page width                                |
|  | Jumps to the desired page                                           |
|  | Searches for a term in the current topic                            |
|  | Continues search for the next term that matches the search criteria |
|  | Exits the help system                                               |

#### Keys for Window "③"

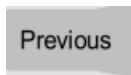
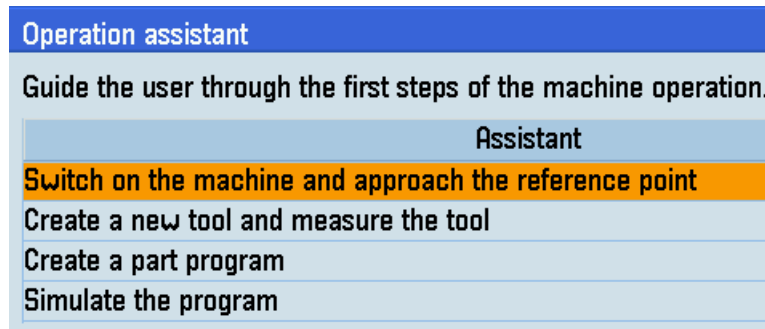
|                                                                                     |                                                                                                                    |
|-------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------|
|  | Expands hierarchical topics                                                                                        |
|  | Collapses hierarchical topics                                                                                      |
|  | Navigates upwards through the hierarchical topics                                                                  |
|  | Navigates downwards through the hierarchical topics                                                                |
|  | Opens the selected topic in the current topic relevant window<br>Functions the same as pressing the following key: |
|                                                                                     |                                 |
|  | Searches for a term in the current topic                                                                           |
|  | Continues search for the next term that matches the search criteria                                                |
|  | Exits the help system                                                                                              |

## A.5 Operation wizard

The operation wizard provides step-by-step guides on basic commissioning and operation procedures.

### Operating sequence

1. Call the operation wizard through the following key operations:



2. Use the cursor keys to select a step of the machining operation.
3. Press this softkey to start the onboard assistant.
4. Press this softkey to enter the next page.
5. Press this softkey to enter the previous page.
6. Press either key to return to the main screen of the operation wizard.
7. Press one of the following five operating area keys to exit the main screen of the operation wizard.

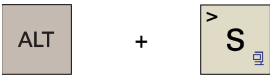


# A.6      Editing Chinese characters

The program editor and PLC alarm text editor both allow you to edit the simplified Chinese characters on the Chinese variant of the HMI.

## Editing simplified Chinese characters

Press the following key combination to switch the editor on or off:



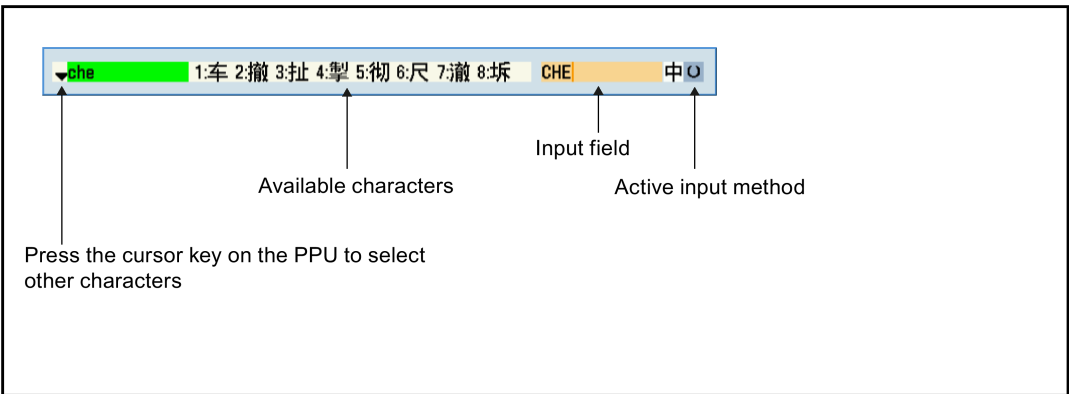
Press this key to toggle between different input methods.

Press the numeric keys (1 to 9) on the PPU to select the desired characters.

## Example of editing simplified Chinese



## Structure of editor



## A.7 Calculating contour elements

### Function

You can use the calculator to calculate the contour elements in the respective input fields.

#### Calculating a point in a circle



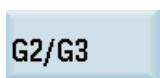
1. Activate the calculator when you position the cursor on the desired input field.



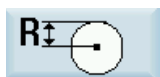
2. Open the lower-level menu for contour elements selection.



3. Select the desired calculation function.



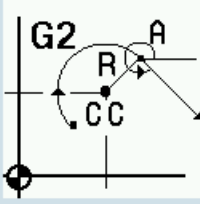
Press this softkey to define the direction of the rotation of the circle.



Press this softkey to switch between the diameter programming and radius programming.

4. Enter the circle center, the angle of the tangent and the circle radius in the following window:

**Calculate a point in a circle**



CC abscissa

CC ordinate

Angle A

Radius R

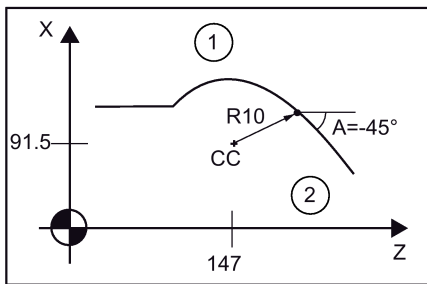
**Diameter programming**



5. Press this softkey to calculate the abscissa and ordinate values of the point.  
The abscissa is the first axis, and the ordinate is the second axis of the plane. The abscissa value is displayed in the input field from which the calculator function has been called, and the value of the ordinate is displayed in the next input field. If the function is called from the part program editor, the coordinates are saved with the axis names of the selected basic plane.

## Example

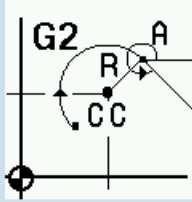
Calculating the point of intersection between the circle sector ① and the straight line ② in plane G18:



### Given:

- Radius: 10
- Circle center point CC: Z=147 X=183 (diameter programming)
- Connection angle for straight lines: -45°

**Calculate a point in a circle**



|             |     |
|-------------|-----|
| CC abscissa | 147 |
| CC ordinate | 183 |
| Angle A     | -45 |
| Radius R    | 10  |

**Diameter programming**

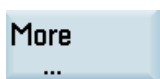
Result: Z = 154.071, X = 190.071

The result appears on the input screen.

### Calculating a point in a plane



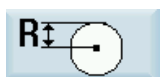
1. Activate the calculator when you position the cursor on the desired input field.



2. Open the lower-level menu for contour elements selection.



3. Select the desired calculation function.



Press this softkey to switch between the diameter programming and radius programming.

4. Enter the following coordinates or angles in the respective input fields:

- Coordinates of the given point (PP)
- Slope angle of the straight line (A1)
- Distance of the new point with reference to PP
- Slope angle of the connecting straight line (A2) with reference to A1



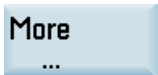
5. Press this softkey to calculate the abscissa and ordinate values of the point.

The abscissa is the first axis, and the ordinate is the second axis of the plane. The abscissa value is displayed in the input field from which the calculator function has been called, and the value of the ordinate is displayed in the next input field. If the function is called from the part program editor, the coordinates are saved with the axis names of the selected basic plane.

### Calculating the Cartesian coordinates



1. Activate the calculator when you position the cursor on the desired input field.

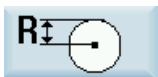


2. Open the lower-level menu for contour elements selection.



3. Select the desired calculation function.

This function converts the given polar coordinates into Cartesian coordinates.



Press this softkey to switch between the diameter programming and radius programming.



4. Enter the reference point, the vector length and the slope angle in the respective input fields.

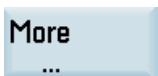
5. Press this softkey to calculate the Cartesian coordinates.

The abscissa value is displayed in the input field from which the calculator function has been called, and the value of the ordinate is displayed in the next input field. If the function is called from the part program editor, the coordinates are saved with the axis names of the selected basic plane.

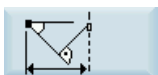
### Calculating the end point



1. Activate the calculator when you position the cursor on the desired input field.

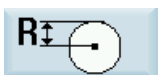


2. Open the lower-level menu for contour elements selection.

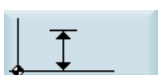


3. Select the desired calculation function.

This function calculates the missing end point of the straight line/straight line contour section whereby the second straight line stands vertically on the first straight line.



Press this softkey to switch between the diameter programming and radius programming.



Press this softkey to define the given end point when the ordinate value is given.



Press this softkey to define the given end point when the abscissa value is given.



Press this softkey to define the second straight line which is rotated counter-clockwise by 90 degrees against the first straight line.



Press this softkey to define the second straight line which is rotated clockwise by 90 degrees against the first straight line.

4. Enter the PP coordinates, angle A, EP abscissa/ordinate, and L length in the respective input fields. The following values of the straight line are known:

Straight line 1: Starting point and slope angle

Straight line 2: Length and one end point in the Cartesian coordinate system

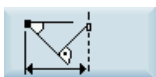
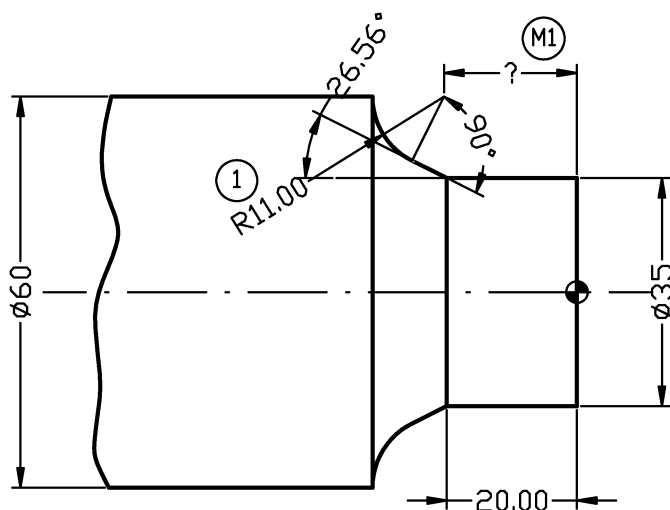
5. Press this softkey to calculate the missing end point.

The abscissa value is displayed in the input field from which the calculator function has been called, and the value of the ordinate is displayed in the next input field. If the function is called from the part program editor, the coordinates are saved with the axis names of the selected basic plane.



### Example

The following drawing must be supplemented by the value of the center circle point in order to be able to calculate the point of intersection between the circle sectors of the straight lines.



The missing center point coordinate is calculated using the calculator function, as the radius at the tangential transition is perpendicular to the straight line.

The radius is located at an angle of 90° clockwise to the straight-line defined by the angle.

Use this softkey to select the appropriate direction of rotation.

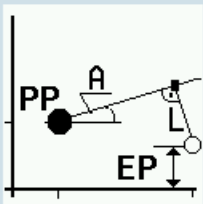


Use this softkey to define the given end point.



Enter the coordinates of the pole, the slope angle of the straight line, the ordinate of the end point, and the circle radius as the length.

Calculate endpoint Pa or Po



|             |        |
|-------------|--------|
| PP abscissa | -20    |
| PP ordinate | 35     |
| PP angle A  | 153.44 |
| EP ordinate | 60     |
| L length    | 11     |

Diameter programming

Result: Z = -19.499, X = 60

## A.8 Free contour programming

### Functionality

Free contour programming enables you to create simple and complex contours.

A contour editor (FKE) calculates any missing parameters for you as soon as they can be obtained from other parameters. You can link together contour elements and transfer to the edited part program.

### Technology

The contour calculator for turning technology provides the following functions for this purpose:

- Toggling between radius/diameter programming (DIAMON, DIAMOF, DIAM90)
- Chamfer/rounding at the start and end of the contour
- Undercuts as transition elements between two axially parallel straight lines, where one runs horizontally and the other vertically (Form E, Form F, thread undercut according to DIN, general undercut)

### Contour editor

Proceed through the following steps to open the contour editor window:



1. Select the program management operating area.



2. Press this softkey to enter the system directory for storing part programs.



3. Select the desired program file, and press this key to open it in the program editor.



4. Press this softkey to open the contour editor window.

5. Initially, you define a contour starting point (see Section "Defining a start point (Page 268)").
6. The contour is then programmed step by step (see Section "Programming example for turning application (Page 278)").

## Recompile

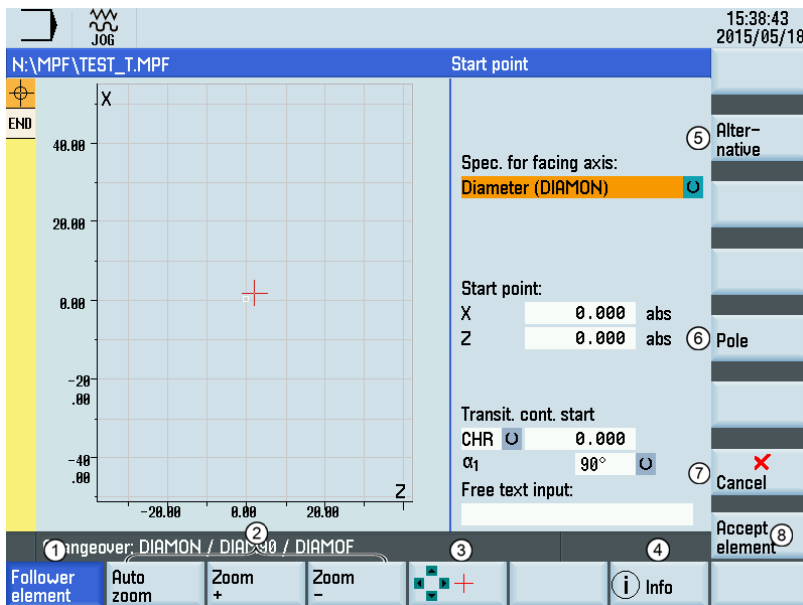


When the program edited in the contour editor is opened in the program editor, if you position the editor cursor in a command line of the contour program and then press this softkey, the main screen of the contour editor opens and you can recompile the existing contour.

## Note

When recompiling, only the contour elements that were generated in the contour editor are created again. Any changes you made directly in the program text are lost; however, you can subsequently insert and edit user-defined texts, which will not be lost.

## Softkey functions



- ① An element was selected using the cursor keys. This softkey enlarges the image section of the selected element.
- ② Zooms the graphic in/out/automatically
- ③ When you select this softkey, you can move the red cross-hair with the cursor keys and choose a picture detail to display. When you deactivate this softkey, the input focus is positioned in the contour chain again.
- ④ If you press this softkey, help graphics are displayed in addition to the relevant parameter. Pressing the softkey again exits the help mode.
- ⑤ Press this softkey to toggle between the selections. This softkey functions the same as pressing the following key:



- ⑥ Defines a pole for contour programming in polar coordinates. The pole can only be entered in absolute Cartesian coordinates.
- ⑦ Exits the contour editor and returns to the program editor window, without transferring the last edited values to the main program
- ⑧ Saves the settings for the start point

## A.8.1 Defining a start point

When entering a contour, begin at a position which you already know and enter it as the starting point.

### Operating sequence



1. Select the program management operating area.



2. Press this softkey to enter the system directory for storing part programs.



3. Select the desired program file, and press this key to open it in the program editor.



4. Press this softkey to open the contour editor window.



5. Use the cursor keys on the PPU to switch between different input fields.



6. Press this softkey to toggle between the selections and enter the desired values as required.



You can alternatively press this key to make your selection.



You can also define a pole for contour programming in polar coordinates by pressing this softkey.

The pole can also be defined or redefined at a later time. The programming of the polar coordinates always refers to the pole that was defined last.



7. Save the settings for the start point.



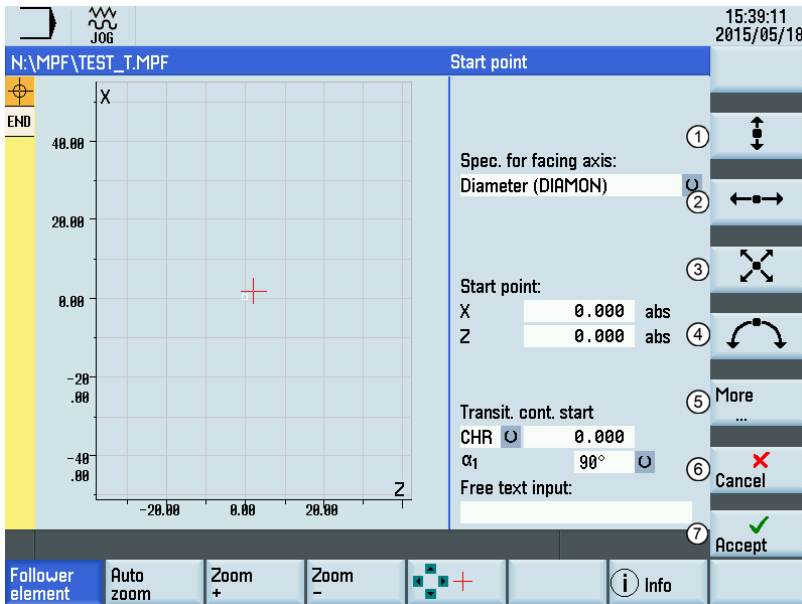
Pressing this softkey cancels the settings and exits the contour editor.

## A.8.2 Programming contour element

### Functionality

**Accept element**

Once you have defined the contour start point, press this softkey and you can begin programming the individual contour elements from the main screen shown below:



- ① Opens the window for programming a vertical straight line (in X direction)
- ② Opens the window for programming a horizontal straight line (in Z direction)
- ③ Opens the window for programming an oblique line in the X/Z direction. The end point of the line is entered using coordinates or an angle.
- ④ Opens the window for programming a circular arc with any direction of rotation
- ⑤ Opens the lower-level menu for more programming options:
  - Specifies a pole
  - Closes the contour
- ⑥ Returns to the program editor without transferring the last edited values to the system
- ⑦ Returns to the program editor with the last edited values transferred to the system

### Further softkey functions

The following softkeys are available in corresponding contour element window for programming the contour elements on the basis of pre-assigned parameters.

#### Tangent to preceding element

**Tangent. trans.**

This softkey presets the angle  $\alpha_2$  to a value of 0. The contour element has a tangential transition to the preceding element, i.e. the angle to the preceding element ( $\alpha_2$ ) is set to 0 degree.

#### Displaying all parameters

**All parameters**

Press this softkey to display a selection list of all the parameters for the selected contour element. If you leave any parameter input fields blank, the control assumes that you do not know the right values and attempts to calculate these from the settings of the other parameters. The contour is always machined in the programmed direction.

### Input switchover

Alter-native

This softkey is displayed only in cases where the cursor is positioned on an input field with several switchover settings.

### Selecting dialog

Dialog select

Some parameter configurations can produce several different contour characteristics. In such cases, you will be asked to select a dialog. By clicking this softkey, you can display the available selection options in the graphic display area. Select this softkey to make the correct selection (green line).

Accept

Confirm your choice with this softkey.

### Changing a selected dialog

Change selection

If you want to change an existing dialog selection, you must select the contour element in which the dialog was originally chosen. Both alternatives are displayed again when you select this softkey.

### Clearing a parameter input field

Delete value

You can delete the value in the selected parameter input field with this softkey or the following key:

DEL

### Saving a contour element

Accept element

If you have entered the available data for a contour element or selected a desired dialog, pressing this softkey allows you to store the contour element and return to the main screen. You can then program the next contour element.

### Appending contour element

Use the cursor keys to select the element in front of the end marker.

Use the softkeys to select the contour element of your choice and enter the values you know in the input screen for that element.

Confirm your inputs with the following softkey:

Accept element

### Selecting contour element

INPUT

Position the cursor on the desired contour element in the contour chain, and select it using this key. The parameters for the selected element will then be displayed. The name of the element appears at the top of the parameterization window.

If the contour element can be represented geometrically, it is highlighted accordingly in the graphic display area, i.e. the color of the contour element changes from white to black.

### Modifying contour element

INPUT

You can use the cursor keys to select a programmed contour element in the contour chain. Press this key to display the parameter input fields. The parameters can now be edited.

### Inserting a contour element

Use the cursor keys in the contour chain to select the contour element in front of the position for the new element.

Then select the contour element to be inserted from the softkey bar.

After you have configured the parameters for the new contour element, confirm the insert operation by pressing the following softkey:

**Accept  
element**

Subsequent contour elements are updated automatically according to the new contour status.

### Deleting contour element

**Delete  
element**

Use the cursor keys to select the element you wish to delete. The selected contour symbol and associated contour element in the programming graphic are highlighted in red. Then press this softkey and confirm the query.

### Closing the contour

**Close  
contour**

By pressing this softkey, you can close the contour from the actual position with a straight line to the starting point.

### Undoing an input

  
**Cancel**

By selecting this softkey you can return to the main screen **without** transferring the last edited values to the system.

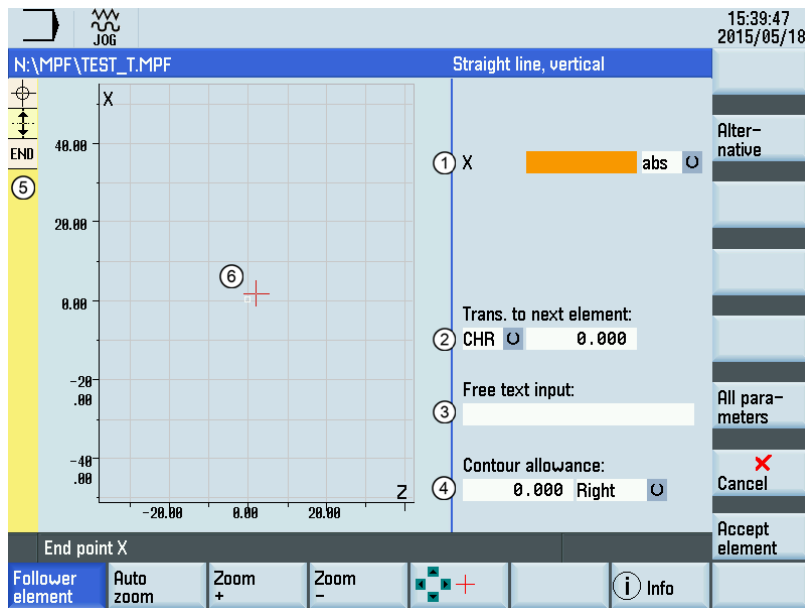
### Contour symbol colors

The meaning of the symbol colors in the contour chain on the left of the main screen is as follows:

| Icon         | Significance                                                                           |
|--------------|----------------------------------------------------------------------------------------|
| Selected     | Symbol color black on a red background → Element is defined geometrically              |
|              | Symbol color black on a light yellow background → Element is not defined geometrically |
| Not selected | Symbol color black on a gray background → Element is defined geometrically             |
|              | Symbol color white on a gray background → Element is not defined geometrically         |

## A.8.3 Parameters for contour elements

### Parameters for programming straight lines



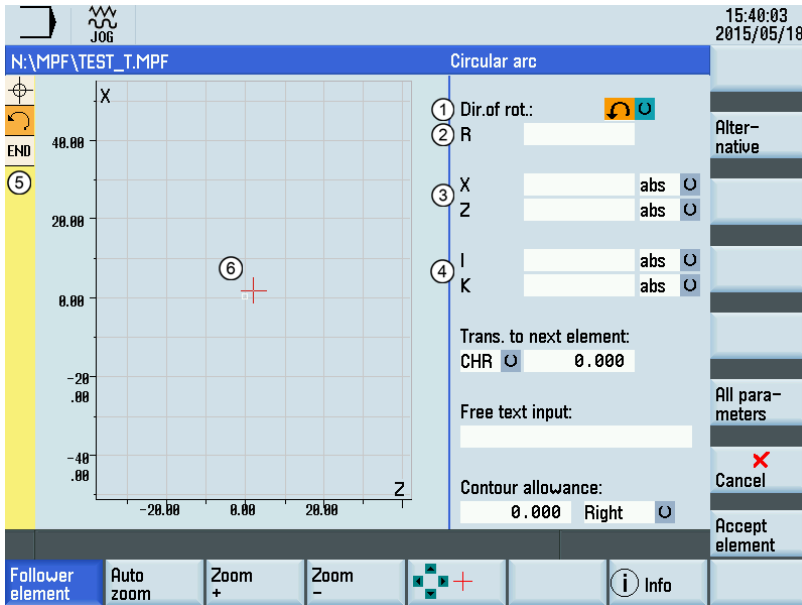
- ① Absolute (abs)/incremental (inc) end position in X or Z direction
- ② Transition element to the next contour is a chamfer (CHR) or a radius (RND). CHR=0 or RND=0 means no transition element.
- ③ Input field for supplementary comments, such as F1000 feedrate values, H or M functions. If comments are entered as text, they must always be started with a semicolon ";".
- ④ You can specify a side-based parallel contour allowance. It is displayed as an allowance in the graphics window.
- ⑤ The contour chain which displays the start point and programmed contour elements. The current position in the chain is color-highlighted.
- ⑥ The graphics window which displays the progress of the contour as you configure the parameters for the contour elements.

#### All parameters

The following additional parameters are displayed after you press this softkey:

| Parameter  | Description                          |
|------------|--------------------------------------|
| L          | Length of the straight line          |
| $\alpha 1$ | Pitch angle with reference to X axis |

## Parameters for programming circular arcs



- ① Direction of rotation of the circular arc: clockwise or counter-clockwise
- ② Radius of circle
- ③ Absolute (abs)/incremental (inc) end positions in X and Z directions
- ④ Absolute (abs)/incremental (inc) positions of circle center point in X (I) and Z (K) directions
- ⑤ The contour chain which displays the start point and programmed contour elements. The current position in the chain is color-highlighted.
- ⑥ The graphics window which displays the progress of the contour as you configure the parameters for the contour elements.

### All parameters

The following additional parameters are displayed after you press this softkey:

| Parameter  | Description                                                     |
|------------|-----------------------------------------------------------------|
| $\alpha 1$ | Starting angle with reference to X axis                         |
| $\alpha 2$ | Angle to preceding element; tangential transition: $\alpha 2=0$ |
| $\beta 1$  | End angle with reference to X axis                              |
| $\beta 2$  | Angle of aperture of circle                                     |

### Transition to next element

A transition element can be used whenever there is a point of intersection between two neighboring elements; this can be calculated from the input values.

You can choose to insert either a radius (RND), a chamfer (CHR) or an undercut as the transition element between any two contour elements. The transition is always appended to the end of a contour element. You select transition elements in the parameter input screen for the relevant contour element.

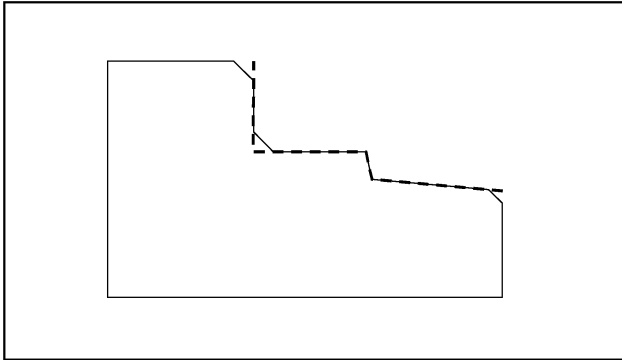
You can access the undercut transition element (see Section "Undercuts for turning technology (Page 275)") with the following softkey:

### Alter-native

### Radius or chamfer at the start or the end of a turning contour:

In simple turning contours a chamfer or radius must often be appended at the start and end of the contour.

A chamfer or radius terminates an axis-parallel contour section on the blank:



You select the direction of transition for the contour start in the starting point screen. You can choose between chamfer and radius. The value is defined in the same manner as for the transition elements.

In addition, four directions can be selected in a single selection field. You select the direction of the transition element for the contour end in the end screen. This selection is always proposed, even if preceding elements were assigned no transition.

### Contour chain

Once you complete or cancel the programming of a contour element, you can navigate around the contour chain (left on the main screen) using the cursor keys. The current position in the chain is color-highlighted.

The elements of the contour and pole, if applicable, are displayed in the sequence in which they were programmed.

You can select an existing contour element with the following key and reassign its parameters:



A new contour element is inserted after the cursor when you select one of the contour elements on the vertical softkey bar; the input focus is then switched to the parameter input on the right of the graphic display. Programming always continues after the element selected in the contour chain.

You can delete the selected element from the chain by selecting the following softkey:



### Graphics window

The graphics window displays the progress of the contour chain as you configure the parameters for the contour elements. The element you have selected is displayed in black in the graphics window.

The contour is displayed to the extent it can be interpreted by the control on the basis of parameter inputs. If the contour is still not displayed in the programming graphic, further values must be entered. Check the contour elements you have already programmed, if required. You may have forgotten to enter all of the known data.

The coordinate system scaling is automatically adapted to changes in the complete contour.

The position of the coordinate system is displayed in the graphics window.

An element was selected using the cursor keys.

Pressing the following softkey allows you to enlarge the image section of the selected element:



## A.8.4 Undercuts for turning technology

### Supplementary conditions

The form E and F undercut and form DIN 76 and general thread undercut functions are only activated when the turning technology is enabled.

Form E and F undercuts as well as thread undercuts are available only if level G18 is set. Undercuts are permitted only on contour edges of the rotational body, which run in the direction of the longitudinal axis (usually parallel to the Z axis). The longitudinal axis is identified by the machine data.

The machine data MD 20100 \$MC\_DIAMETER\_AX\_DEF for turning machines contains the name of the transverse axis (usually X). The other axis in G18 is the longitudinal axis (usually Z). If MD 20100 \$MC\_DIAMETER\_AX\_DEF does not contain a name or contains a name that does not conform to G18, there are no undercuts.

Undercuts only appear on corners between horizontal and vertical straight lines, including any straight lines, which are at 0°, 90°, 180° or 270°. A tolerance of ±3° is required here, so that conical threads are also possible (these undercuts do not meet the standard in this case).

### Selecting an undercut form

When you choose the transition element in the contour programming window, you can use the following softkey to select an undercut as a transition element:

Alter-  
native

You can subsequently define the undercut form by toggling between the selections in the corresponding input fields.

|     |           |   |
|-----|-----------|---|
|     | Undercut  | U |
|     | Form F    | U |
| RxT | 0.1 x 0.1 | U |

In the case of standard thread undercuts, the characteristic size of the thread pitch is P. The depth, length and transition radius of the undercut are calculated according to the DIN standard. The (metric) thread pitches specified in DIN 76 can be used. The entry angle can be freely selected in the 30°-90° range. If the diameter is known when selecting the undercut, an appropriate thread pitch is suggested. Forms DIN 76 A (external control) and DIN 76 C (internal control) are available. The program detects the two forms automatically using their geometry and topology.

Based on the thread undercut according to DIN, you can use the general undercut type to create specific undercuts, e.g. for inch threads.

## A.8.5 Specifying contour elements in polar coordinates

### Functionality

The description given above of defining the coordinates of contour elements applies to the specification of positional data in the Cartesian coordinate system. Alternatively, you have the option to define positions using polar coordinates.

When programming contours, you can define a pole at any time prior to using polar coordinates for the first time. Programmed polar coordinates subsequently refer to this pole. The pole is modal and can be re-defined at any time. It is always entered in absolute Cartesian coordinates. The contour calculator converts values entered as polar coordinates into Cartesian coordinates. Positions can be programmed in polar coordinates only **after** a pole has been specified. The pole input does not generate a code for the NC program.

## Pole

The polar coordinates are valid in the level selected with G17 to G19.



Pole

The pole is a contour element that can be edited, which itself does not contribute to the contour. It can be entered when the starting point of the contour is defined or anywhere within the contour. The pole cannot be created before the starting point of the contour.

This softkey allows you to specify a pole and can only be entered in absolute Cartesian coordinates. This softkey is also present in the starting point screen. This enables the pole to be entered at the start of a contour, so that the first contour element can be entered in polar coordinates.

### Further notes

If the straight line that was generated with close contour is linked to the start element of the contour with a radius or chamfer, the radius or chamfer must be specified explicitly as follows:

- Close contour, enter radius/chamfer, and accept element. The result then corresponds exactly to what would occur if the closing element were to be entered with the radius or chamfer.  
Close contour can only be used for entering contour elements in **polar coordinates** if the starting point of the contour was set to polar and the **same pole** is still valid when the contour is closed.

### Input switchover: Cartesian/polar

The following contour elements can be entered optionally in polar coordinates only after a pole has been defined, whether this was done at the outset or later in the process:

- Circular arcs
- Straight lines (horizontal, vertical, any direction)

To switchover between Cartesian and polar coordinates, additional toggle fields are displayed in the programming windows for the contour elements of oblique lines and circular arcs.

A toggle field is not displayed if no pole exists. Input fields and display fields are then only available for Cartesian values.

### Absolute/incremental input

Absolute and incremental polar coordinates can be input for "polar/Cartesian". The input fields and display fields are labeled **inc** and **abs**.

Absolute polar coordinates are defined by an absolute distance to the pole that is always positive and an angle in the range of  $0^\circ \dots \pm 360^\circ$ . When absolute dimensions are specified, the angular reference is based on a horizontal axis of the working plane, e.g. X axis with G17. The positive direction of rotation runs counter-clockwise.

If there are several input poles, the definitive pole is always the **last pole** before the input or edited element.

Incremental polar coordinates relate to both the definitive pole and the end point of the preceding element.

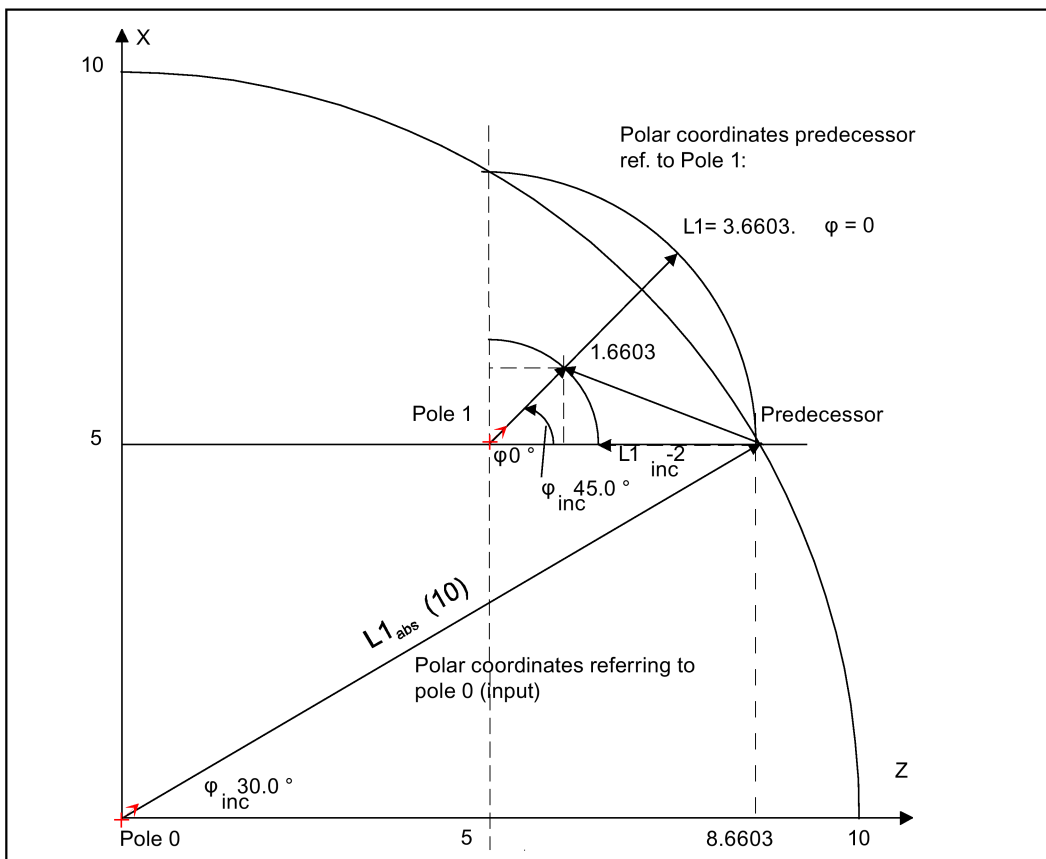
For an incremental input, **the absolute distance** to the pole is calculated using the absolute distance from the end point of the preceding element to the pole plus the length increment that was entered.

The increment can be positive or negative.

**The absolute angle** is calculated accordingly using the absolute polar angle of the preceding element plus the angular increment. Here it is not necessary for the preceding element to have been entered as polar.

In contour programming, the contour calculator converts the Cartesian coordinates of the preceding end point using the definitive pole into polar coordinates. This also applies if the preceding element has been given in polar coordinates, since this could relate to another pole if a pole has been inserted in the meantime.

## Pole change example



**Pole:**  $Z_{pole} = 0.0,$   $X_{pole} = 0.0,$  (Pole 0)

End point:

$L1_{abs} = 10.0$

$\phi_{abs} = 30.0^\circ$

Calculated Cart. Coordinates

$Z_{abs} = 8.6603$

$X_{abs} = 5.0$

**New pole:**

$Z_{pole1} = 5.0$

$X_{pole1} = 5.0$

(Pole 1)

Calculated polar coord. Predecessor

$L1_{abs} = 3.6603$

$\phi_{abs} = 0.0^\circ$

**Next point:**

$L1_{inc} = -2.0$

$\phi_{inc} = 45.0^\circ$

Absolute polar coordinates for current element

$L1_{abs} = 1.6603$

$\phi_{abs} = 45.0^\circ$

Calculate Cartesian coordinates

$Z_{abs} = 1.1740$

$X_{abs} = 1.1740$

## A.8.6 Cycle support

### Functionality

The following technologies are provided with additional support in the form of pre-defined cycles, which then must be parameterized.

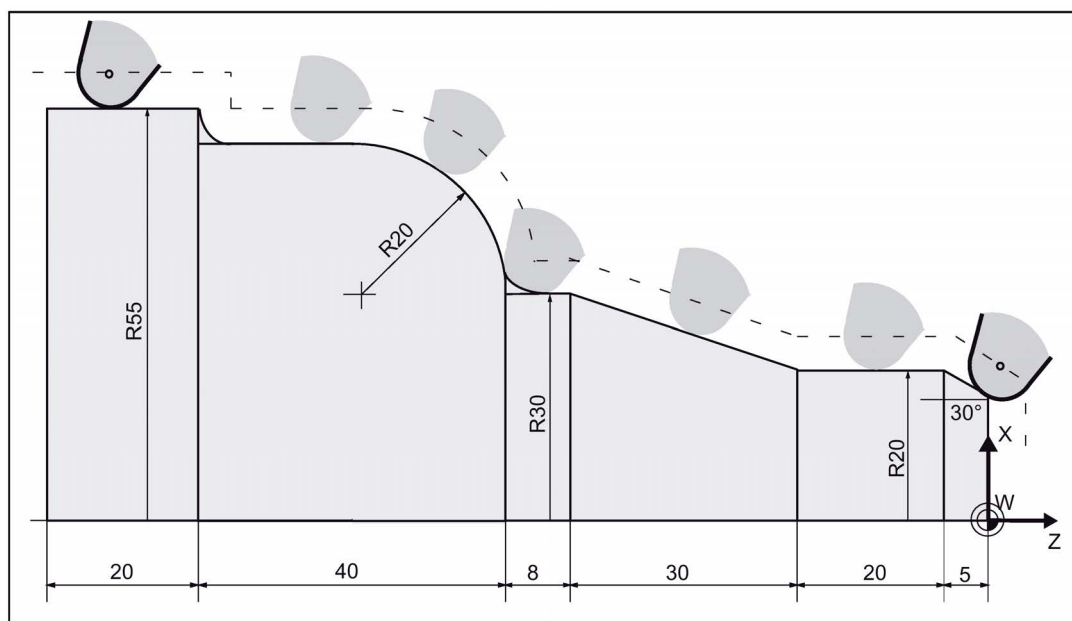
- Drilling
- Turning

For more information, see Chapter "Cycles (Page 149)".

## A.8.7 Programming example for turning application

### Example 1

The following diagram shows a programming example for the "Free contour programming" function.



Operating sequence:



1. Select the program management operating area.



2. Enter the system program directory.



3. Select a program with cursor keys and press this key to open the program in the program editor.



4. Press this softkey to open the contour editor.

Accept  
element



Accept  
element



Accept  
element



Accept  
element



Accept  
element



Dialog  
select

Accept  
element



Accept  
element

5. Define a start point with the following parameters and press this softkey to confirm.
  - Programming mode: DIAMOF
  - Z: 0
  - X: 0
6. Press this softkey to select a contour element of straight vertical line.
7. Enter the parameters for this element and press this softkey to confirm.
  - X: 20 inc.
  - CHR:  $5 \times 1.1223 = 5.6115$
8. Press this softkey to select a contour element of straight horizontal line.
9. Enter the parameters for this element and press this softkey to confirm.
  - Z: -25 inc.
10. Press this softkey to select a contour element of straight line in any direction.
11. Enter the parameters for this element and press this softkey to confirm.
  - X: 10 inc.
  - Z: -30 inc.
12. Press this softkey to select a contour element of straight horizontal line.
13. Enter the parameters for this element and press this softkey to confirm.
  - Z: -8 inc.
  - RND: 2
14. Press this softkey to select a contour element of circular arc.
15. Enter the parameters for this element and press this softkey to select the desired contour characteristics.
  - Direction of rotation: counter-clockwise
  - R: 20
  - X: 20 inc
  - Z: -20 inc
16. Press this softkey to confirm.
17. Press this softkey to select a contour element of straight horizontal line.
18. Enter the parameters for this element and press this softkey to confirm.
  - Z: -20 inc.
  - RND: 2



19. Press this softkey to select a contour element of straight vertical line.



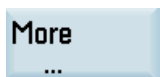
20. Enter the parameters for this element and press this softkey to confirm.
  - X: 5 inc.



21. Press this softkey to select a contour element of straight horizontal line.



22. Enter the parameters for this element and press this softkey to confirm.
  - Z: -25 inc.

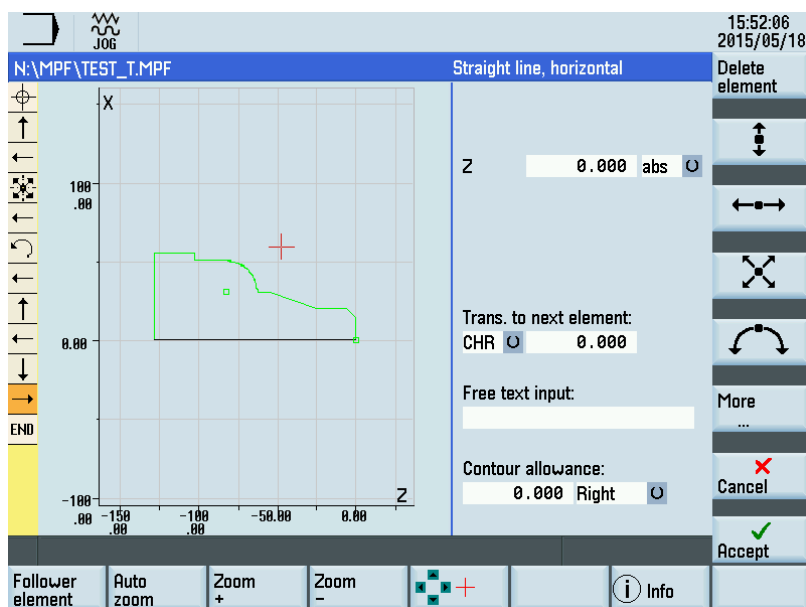


23. Press this softkey to access more options.



24. Press this softkey to close the contour.

Now you can see the programmed contour in the graphics window:



## Example 2

Operating sequence:



1. Select the program management operating area.



2. Enter the system program directory.



3. Select a program with cursor keys and press this key to open the program in the program editor.



4. Press this softkey to open the contour editor.
5. Define a start point with the following parameters and press this softkey to confirm.
  - Programming mode: DIAMON
  - Z: 0
  - X: 0
6. Press this softkey to select a contour element of straight vertical line.
7. Enter the parameters for this element and press this softkey to confirm.
  - X: 48 abs.
  - CHR: 3
8. Press this softkey to select a contour element of straight horizontal line.
9. Enter the parameters for this element and press this softkey to confirm.
  - RND: 4
10. Press this softkey to select a contour element of circular arc.
11. Enter the parameters for this element and press this softkey to select the desired contour characteristics.
  - R: 23
  - X: 60 abs.
  - Z: -20 abs.
12. Press this softkey to confirm.
13. Press this softkey to select a contour element of straight horizontal line.
14. Enter the parameters for this element and press this softkey to confirm.
  - Z: -75 abs.
  - RND: 6
15. Press this softkey to select a contour element of straight line in any direction.
16. Enter the parameters for this element and press this softkey to confirm.
  - X: 90 abs.
  - Z: -80 abs.
  - RND: 4
17. Press this softkey to select a contour element of straight horizontal line.
18. Enter the parameters for this element and press this softkey to confirm.
  - Z: -100 abs.



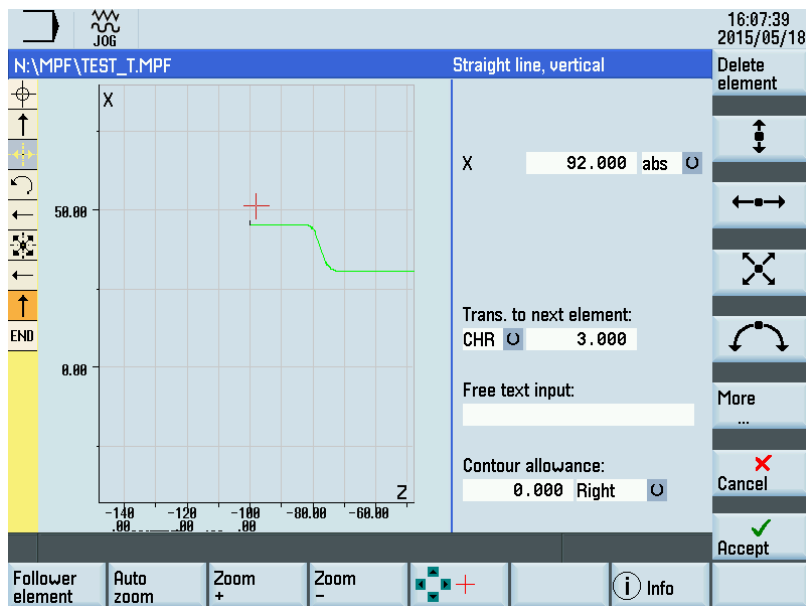
19. Press this softkey to select a contour element of straight vertical line.



20. Enter the parameters for this element and press this softkey to confirm.

- X: 92 abs.
- CHR: 3

Now you can see the programmed contour in the graphics window:



## A.9 Word structure and address

### Functionality/structure

A word is a block element and mainly constitutes a control command. The word consists of the following two parts:

- **address character:** generally a letter
- **numerical value:** a sequence of digits which with certain addresses can be added by a sign put in front of the address, and a decimal point.  
A positive sign (+) can be omitted.

The following picture shows an example of the word structure.

|              |                                    |                                              |                      |
|--------------|------------------------------------|----------------------------------------------|----------------------|
|              | Word                               | Word                                         | Word                 |
|              | Address   Value                    | Address   Value                              | Address   Value      |
| Example:     | G1                                 | X -20.1                                      | F300                 |
| Explanation: | Traverse with Linear interpolation | Path or end-position for the X axis: -20.1mm | Feedrate: 300 mm/min |

### Several address characters

A word can also contain several address letters. In this case, however, the numerical value must be assigned via the intermediate character "=".

Example: **CR=5.23**

Additionally, it is also possible to call G functions using a symbolic name. For more information, see Section "List of instructions (Page 285)".

Example: **SCALE** ; Enable scaling factor

### Extended address

With the following addresses, the address is extended by 1 to 4 digits to obtain a higher number of addresses. In this case, the value must be assigned using an equality sign "=".

|         |                                                              |
|---------|--------------------------------------------------------------|
| R       | Arithmetic parameters                                        |
| H       | H function                                                   |
| I, J, K | Interpolation parameters/intermediate point                  |
| M       | Special function M, affecting the spindle with other options |
| S       | Spindle speed                                                |

Examples: **R10=6.234 H5=12.1 I1=32.67 M2=5 S1=400**

## A.10 Character set

The following characters are used for programming. They are interpreted in accordance with the relevant definitions.

### Letters, digits

A, B, C, D, E, F, G, H, I, J, K, L, M, N, O, P, Q, R, S, T, U, V, W, X, Y, Z  
0, 1, 2, 3, 4, 5, 6, 7, 8, 9

No distinction is made between lowercase and uppercase letters.

### Printable special characters

|   |                              |    |                                 |
|---|------------------------------|----|---------------------------------|
| ( | Open parenthesis             | "  | Inverted commas                 |
| ) | Close parenthesis            | _  | Underscore (belongs to letters) |
| [ | Open square bracket          | .  | Decimal point                   |
| ] | Close square bracket         | ,  | Comma, separator                |
| < | less than                    | ;  | Comment start                   |
| > | greater than                 | %  | Reserved; do not use            |
| : | Main block, end of label     | &  | Reserved; do not use            |
| = | Assignment, part of equation | '  | Reserved; do not use            |
| / | skip                         | \$ | System variable identifiers     |
| * | Multiplication               | ?  | Reserved; do not use            |
| + | Addition and positive sign   | !  | Reserved; do not use            |
| - | Subtraction, minus sign      |    |                                 |

### Non-printable special characters

|               |                                |
|---------------|--------------------------------|
| LF            | End-of-block character         |
| Blank         | Separator between words; blank |
| Tab character | Reserved; do not use           |

## A.11 Block format

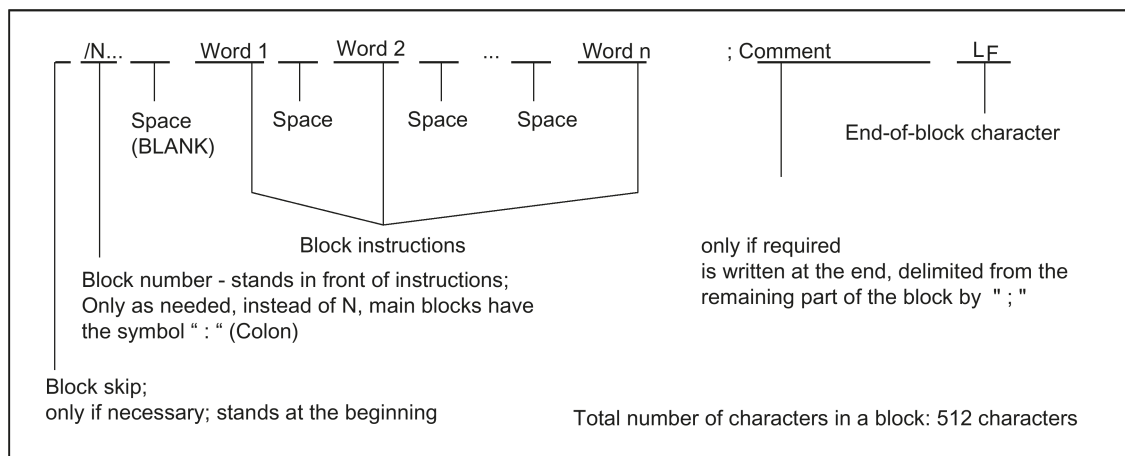
### Functionality

A block should contain all data required to execute a machining step.

Generally, a block consists of several **words** and is always completed with the **end-of-block character** "LF" (Linefeed). When writing a block, this character is automatically generated when pressing the linefeed key on an externally connected keyboard or pressing the following key on the PPU:



See the following block structure diagram:



### Word order

If there are several instructions in a block, the following order is recommended:

**N... G... X... Z... F... S... T... D... M... H...**

### Note regarding block numbers

First select the block numbers in steps of 5 or 10. Thus, you can later insert blocks and nevertheless observe the ascending order of block numbers.

### Block skip

Blocks of a program, which are to be executed not with each program run, can be **marked** by a slash / in front of the block number.

The block skip itself is activated via **Operation** (program control: "SKP") or by the programmable controller (signal). A section can be skipped by several blocks in succession using "/".

If a block must be skipped during program execution, all program blocks marked with "/" are not executed. All instructions contained in the blocks concerned will not be considered. The program is continued with the next block without marking.

### Comment, remark

The instructions in the blocks of a program can be explained using comments (remarks). A comment always starts with a semicolon ";" and ends with end-of-block.

Comments are displayed together with the contents of the remaining block in the current block display.

### Messages

Messages are programmed in a separate block. A message is displayed in a special field and remains active until a block with a new message is executed or until the end of the program is reached. Up to **65** characters can be displayed in message texts.

A message without message text cancels a previous message.

MSG ("THIS IS THE MESSAGE TEXT")

## Programming example

```

N10 ; G&S company, order no. 12A71
N20 ; Pump part 17, drawing no.: 123 677
N30 ; Program created by H. Adam, Dept. TV 4
N40 MSG("DRAWING NO.: 123677")
:50 G54 F4.7 S220 D2 M3 ;Main block
N60 G0 G90 X100 Z200
N70 G1 Z185.6
N80 X112
/N90 X118 Z180 ; Block can be suppressed
N100 X118 Z120
N110 G0 G90 X200
N120 M2 ; End of program

```

## A.12 List of instructions

The functions below marked with an asterisk (\*) are active at the start of the program in the turning variant of the control system, unless otherwise programmed or the machine manufacturer has preserved the default setting for the "turning" technology.

| Address | Significance                                   | Value assignments              | Information                                                                                                                                                                                                                                                                     | Programming                         |
|---------|------------------------------------------------|--------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------------------|
| D       | Tool offset number                             | 0 ... 9, only integer, no sign | Contains compensation data for a particular tool T... ; D0 indicates no compensation for the tool, one tool activates 1~9 numbers, that is, one tool carries at most nine different compensation data at the same time.                                                         | D...                                |
| F       | Feedrate                                       | 0.001 ... 99<br>999.999        | Path velocity of a tool/workpiece; unit: mm/min or mm/revolution depending on G94 or G95                                                                                                                                                                                        | F...                                |
| F       | Dwell time (block with G4)                     | 0.001 ... 99<br>999.999        | Dwell time in seconds                                                                                                                                                                                                                                                           | G4 F...; separate block             |
| F       | Thread lead change (block containing G34, G35) | 0.001 ... 99<br>999.999        | in mm/rev <sup>2</sup>                                                                                                                                                                                                                                                          | See G34, G35                        |
| G       | G function (preparatory function)              | Only integer, specified values | The G functions are divided into G groups. Only one G function of a group can be programmed in a block.<br>A G function can be either modal (until it is canceled by another function of the same group) or only effective for the block in which it is programmed (non-modal). | G...<br>or symbolic name, e.g.: CIP |

| Address  | Significance                                      | Value assignments                                             | Information  | Programming                                                                                                                                                                                                                                                                        |
|----------|---------------------------------------------------|---------------------------------------------------------------|--------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| G group: |                                                   |                                                               |              |                                                                                                                                                                                                                                                                                    |
| G0       | Linear interpolation at rapid traverse rate       | 1: Motion commands (type of interpolation), modally effective | G0 X... Z... |                                                                                                                                                                                                                                                                                    |
| G1 *     | Linear interpolation at feedrate                  |                                                               |              | G1 X...Z... F...                                                                                                                                                                                                                                                                   |
| G2       | Circular interpolation clockwise                  |                                                               |              | G2 X... Z... I... K... F...<br>;End point and center point<br>G2 X... Z... CR=... F...<br>;Radius and end point<br>G2 AR=... I... K... F...<br>;opening angle and center point<br>G2 AR=... X... Z... F...<br>;opening angle and end point                                         |
| G3       | Circular interpolation counter-clockwise          |                                                               |              | G3...; others, as with G2                                                                                                                                                                                                                                                          |
| CIP      | Circular interpolation through intermediate point |                                                               |              | CIP X... Z... I1=... K1=... F... ;I1, K1 is intermediate point                                                                                                                                                                                                                     |
| CT       | Circular interpolation; tangential transition     |                                                               |              | N10 ...<br>N20 CT Z... X... F...<br>;circle, tangential transition to the previous path segment N10                                                                                                                                                                                |
| G33      | Thread cutting with constant lead                 |                                                               |              | ;Constant lead<br>G33 Z... K... SF=...<br>;cylindrical thread<br>G33 X... I... SF=... ;face thread<br>G33 Z... X... K... SF=...<br>;taper thread, in Z axis path larger than in the X axis<br>G33 Z... X... I... SF=...<br>;taper thread, in X axis path larger than in the Z axis |
| G34      | Thread cutting, increasing lead                   |                                                               |              | G33 Z... K... SF=...<br>;cylindrical thread, constant lead<br>G34 Z... K... F17.123 ;<br>lead increasing with ;17.123 mm/rev <sup>2</sup>                                                                                                                                          |
| G35      | Thread cutting, decreasing lead                   |                                                               |              | G33 Z... K... SF=...<br>;cylindrical lead<br>G35 Z... K... F7.321 ;lead decreasing with ;7.321 mm/rev <sup>2</sup>                                                                                                                                                                 |
| G331     | Thread interpolation                              |                                                               |              | N10 SPOS=... ; Spindle in position control<br>N20 G331 Z... K... S...<br>;tapping without compensating chuck e.g. in Z axis<br>;RH or LH thread is defined via the sign of the lead (e.g. K+):<br>+ : as with M3<br>- : as with M4                                                 |

| Address | Significance                                                                              | Value assignments | Information                                    | Programming                                                                                                      |
|---------|-------------------------------------------------------------------------------------------|-------------------|------------------------------------------------|------------------------------------------------------------------------------------------------------------------|
| G332    | Thread interpolation - retraction                                                         |                   |                                                | G332 Z... K... ;tapping without compensating chuck, e.g. in Z axis, retraction motion ; sign of lead as for G331 |
| G4      | Dwell time                                                                                |                   | 2: Special motions, dwell time, non-modal      | G4 F...;separate block, F: Time in seconds<br>or<br>G4 S.... ;separate block, S: in spindle revolutions          |
| G74     | Reference point approach                                                                  |                   |                                                | G74 X=0 Z=0 ;separate block, (machine axis identifier!)                                                          |
| G75     | Fixed point approach                                                                      |                   |                                                | G75 X=0 Z=0 ;separate block, (machine axis identifier!)                                                          |
| TRANS   | Translation, programmable                                                                 |                   | 3: Write memory, non-modal                     | TRANS X... Z... ;separate block                                                                                  |
| SCALE   | Programmable scaling factor                                                               |                   |                                                | SCALE X... Z... ; scaling factor in the direction of the specified axis, separate block                          |
| ROT     | Rotation, programmable                                                                    |                   |                                                | ROT RPL=... ;rotation in the current plane G17 to G19, separate block                                            |
| MIRROR  | Programmable mirroring                                                                    |                   |                                                | MIRROR X0 ; mirror the coordinate axis, separate block                                                           |
| ATRANS  | Additive translation, programming                                                         |                   |                                                | ATRANS X... Z... ; separate block                                                                                |
| ASCALE  | Additive programmable scaling factor                                                      |                   |                                                | ASCALE X... Z... ; scaling factor in the direction of the specified axis, separate block                         |
| AROT    | Additive programmable rotation                                                            |                   |                                                | AROT RPL=... ; rotation in the current plane G17 to G19, separate block                                          |
| AMIRROR | Additive programmable mirroring                                                           |                   |                                                | AMIRROR X0 ;mirror the coordinate axis, separate block                                                           |
| G17     | X/Y plane (when center-drilling, TRANSMIT milling required))                              |                   | 6: Plane selection                             |                                                                                                                  |
| G18 *   | Z/X plane (standard turning)                                                              |                   |                                                |                                                                                                                  |
| G19     | Y/Z plane                                                                                 |                   |                                                |                                                                                                                  |
| G40 *   | Tool radius compensation OFF                                                              |                   | 7: Tool radius compensation, modally effective |                                                                                                                  |
| G41     | Tool radius compensation; along the moving direction of the tool, always left of contour  |                   |                                                |                                                                                                                  |
| G42     | Tool radius compensation; along the moving direction of the tool, always right of contour |                   |                                                |                                                                                                                  |
| G500 *  | Settable work offset OFF                                                                  |                   | 8: Settable work offset, modally effective     |                                                                                                                  |
| G54     | 1. Settable work offset                                                                   |                   |                                                |                                                                                                                  |
| G55     | 2. Settable work offset                                                                   |                   |                                                |                                                                                                                  |
| G56     | 3. Settable work offset                                                                   |                   |                                                |                                                                                                                  |
| G57     | 4. Settable work offset                                                                   |                   |                                                |                                                                                                                  |

| Address                  | Significance                                                                       | Value assignments                                                                                | Information                                                                                                    | Programming                         |
|--------------------------|------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------------------------|-------------------------------------|
| G58                      | 5. Settable work offset                                                            |                                                                                                  |                                                                                                                |                                     |
| G59                      | 6. Settable work offset                                                            |                                                                                                  |                                                                                                                |                                     |
| G53                      | Settable work offset OFF, non-modal                                                |                                                                                                  | 9: Suppressing the settable work offset, non-modal                                                             |                                     |
| G153                     | Settable work offset OFF, non-modal, including base frame                          |                                                                                                  |                                                                                                                |                                     |
| G60 *                    | Exact stop                                                                         |                                                                                                  | 10: Approach behavior, modally effective                                                                       |                                     |
| G64                      | Continuous-path mode                                                               |                                                                                                  |                                                                                                                |                                     |
| G62                      | Corner deceleration at inside corners when tool radius offset is active (G41, G42) |                                                                                                  | Only in conjunction with continuous-path mode                                                                  | G62 Z... G1                         |
| G9                       | Non-modal exact stop                                                               |                                                                                                  | 11: Non-modal exact stop, non-modal                                                                            |                                     |
| G601 *                   | Exact stop window, fine, with G60, G9                                              |                                                                                                  | 12: Exact stop window, modally effective                                                                       |                                     |
| G602                     | Exact stop window, coarse, with G60, G9                                            |                                                                                                  |                                                                                                                |                                     |
| G621                     | Corner deceleration at all corners                                                 |                                                                                                  | Only in conjunction with continuous-path mode                                                                  | G621 AIDS=...                       |
| G70                      | Inch dimension input                                                               |                                                                                                  | 13: Inch/metric dimension data, modally effective                                                              |                                     |
| G71 *                    | Metric dimension data input                                                        |                                                                                                  |                                                                                                                |                                     |
| G700                     | Inch dimension data input; also for feedrate F                                     |                                                                                                  |                                                                                                                |                                     |
| G710                     | Metric dimension data input; also for feedrate F                                   |                                                                                                  |                                                                                                                |                                     |
| G90 *                    | Absolute dimension data input                                                      |                                                                                                  | 14: Absolute/incremental dimension, modally effective                                                          |                                     |
| G91                      | Incremental dimension input                                                        |                                                                                                  |                                                                                                                |                                     |
| G94                      | Feed F in mm/min                                                                   |                                                                                                  | 15: Feedrate/spindle, modally effective                                                                        |                                     |
| G95 *                    | Feedrate F in mm/spindle revolutions                                               |                                                                                                  |                                                                                                                |                                     |
| G96                      | Constant cutting rate ON (F in mm/rev., S in m/min)                                |                                                                                                  |                                                                                                                | G96 S... LIMS=... F...              |
| G97                      | Constant cutting speed OFF                                                         |                                                                                                  |                                                                                                                |                                     |
| G450 *                   | Transition circle                                                                  |                                                                                                  | 18: Behavior at corners when working with tool radius compensation, modally effective                          |                                     |
| G451                     | Point of intersection                                                              |                                                                                                  |                                                                                                                |                                     |
| BRISK *                  | Jerking path acceleration                                                          |                                                                                                  | 21: Acceleration profile, modally effective                                                                    |                                     |
| SOFT                     | Jerk-limited path acceleration                                                     |                                                                                                  |                                                                                                                |                                     |
| FFWOF *                  | Feedforward control OFF                                                            |                                                                                                  | 24: Feedforward control, modally effective                                                                     |                                     |
| FFWON                    | Feedforward control ON                                                             |                                                                                                  |                                                                                                                |                                     |
| DIAMOF                   | Radius dimensioning                                                                |                                                                                                  | 29: Dimension radius/diameter, modally effective                                                               |                                     |
| DIAMON *                 | Diameter dimensioning                                                              |                                                                                                  |                                                                                                                |                                     |
| G290 *                   | SIEMENS mode                                                                       |                                                                                                  | 47: External NC languages, modally effective                                                                   |                                     |
| G291                     | External mode                                                                      |                                                                                                  |                                                                                                                |                                     |
| H<br>H0=<br>to<br>H9999= | H function                                                                         | ± 0.0000001 ... 9999 9999 (8 decimal places) or specified as an exponent: ± (10-300 ... 10+300 ) | Value transfer to the PLC; significance defined by the machine manufacturer                                    | H0=... H9999=...<br>e.g.: H7=23.456 |
| I                        | Interpolation parameters                                                           | ±0.001 ... 99 999.999<br>Thread: 0.001 ... 2000.000                                              | Belongs to the X axis; meaning dependent on G2, G3 -> circle center or G33, G34, G35 G331, G332 -> thread lead | See G2, G3 and G33, G34, G35        |

| Address | Significance                                            | Value assignments                                                | Information                                                                                                                                                                                                                             | Programming                                 |
|---------|---------------------------------------------------------|------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------|
| K       | Interpolation parameters                                | $\pm 0.001 \dots 99999.999$<br>Thread:<br>$0.001 \dots 2000.000$ | Belongs to the Z axis; others, as with I                                                                                                                                                                                                | See G2, G3 and G33, G34, G35                |
| I1=     | Intermediate point for circular interpolation           | $\pm 0.001 \dots 99999.999$                                      | Belongs to the X axis; specification for circular interpolation with CIP                                                                                                                                                                | See CIP                                     |
| K1=     | Intermediate point for circular interpolation           | $\pm 0.001 \dots 99999.999$                                      | Belongs to the Z axis; specification for circular interpolation with CIP                                                                                                                                                                | See CIP                                     |
| L       | Subroutine; name and call                               | 7 decimals; integer only, no sign                                | Instead of a free name, it is also possible to select L1 ...L9999999; this also calls the subroutine (UP) in a separate block, Please note: L0001 is not always equal to L1. The name "LL6" is reserved for the tool change subroutine. | L.... ;separate block                       |
| M       | Additional function                                     | 0 ... 99<br>only integer, no sign                                | For example, for initiating switching actions, such as "coolant ON", maximum five M functions per block.                                                                                                                                | M...                                        |
| M0      | Programmed stop                                         |                                                                  | The machining is stopped at the end of a block containing M0; to continue, press the following key:<br>                                              |                                             |
| M1      | Optional stop                                           |                                                                  | As with M0, but the stop is only performed if a special signal (Program control: "M01") is present.                                                                                                                                     |                                             |
| M2      | End of main program with return to beginning of program |                                                                  | Can be found in the last block of the processing sequence                                                                                                                                                                               |                                             |
| M30     | End of program (as M2)                                  |                                                                  | Can be found in the last block of the processing sequence                                                                                                                                                                               |                                             |
| M17     | End of subroutine                                       |                                                                  | Can be found in the last block of the processing sequence                                                                                                                                                                               |                                             |
| M3      | CW rotation of spindle                                  |                                                                  |                                                                                                                                                                                                                                         |                                             |
| M4      | CCW rotation of spindle                                 |                                                                  |                                                                                                                                                                                                                                         |                                             |
| M5      | Spindle stop                                            |                                                                  |                                                                                                                                                                                                                                         |                                             |
| Mn=3    | CW rotation of spindle                                  |                                                                  | n = 1                                                                                                                                                                                                                                   | M1=3 ; CW rotation stop for spindle 1       |
| Mn=4    | CCW rotation of spindle (for spindle n)                 |                                                                  | n = 1                                                                                                                                                                                                                                   | M1=4 ; CCW rotation stop for spindle 1      |
| Mn=5    | Spindle stop (for spindle n)                            |                                                                  | n = 1                                                                                                                                                                                                                                   | M1=5 ; Spindle stop for spindle 1           |
| M6      | Tool change                                             |                                                                  | Only if activated with M6 via the machine control panel; otherwise, change directly using the T command                                                                                                                                 |                                             |
| M40     | Automatic gear stage switching                          |                                                                  |                                                                                                                                                                                                                                         |                                             |
| Mn=40   | Automatic gear stage switching (for spindle n)          |                                                                  | n = 1                                                                                                                                                                                                                                   | M1=40 ; automatic gear stage; for spindle 1 |

| Address        | Significance                                 | Value assignments                                                                                            | Information                                                                                                                                                                                         | Programming                                                                                  |
|----------------|----------------------------------------------|--------------------------------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------|
| M41 to M45     | Gear stage 1 to gear stage 5                 |                                                                                                              |                                                                                                                                                                                                     |                                                                                              |
| Mn=41 to Mn=45 | Gear stage 1 to gear stage 5 (for spindle n) |                                                                                                              | n = 1                                                                                                                                                                                               | M1=41; 1st gear stage for spindle 1                                                          |
| M19            | Spindle positioned at 0 degree               |                                                                                                              |                                                                                                                                                                                                     |                                                                                              |
| M70            | Spindle switched to the axis mode            |                                                                                                              |                                                                                                                                                                                                     |                                                                                              |
| M...           | Remaining M functions                        |                                                                                                              | Functionality is not defined by the control system and can therefore be used freely by the machine manufacturer                                                                                     |                                                                                              |
| N              | Block number - sub-block                     | 0 ... 9999 9999 only integer, no sign                                                                        | Can be used to identify blocks with a number; is written at the beginning of a block.                                                                                                               | N20                                                                                          |
| :              | Block number of a main block                 | 0 ... 9999 9999 only integer, no sign                                                                        | Special block identification, used instead of N... ; such a block should contain all instructions for a complete subsequent machining step.                                                         | :20                                                                                          |
| P              | Number of subroutine passes                  | 1 ... 9999 only integer, no sign                                                                             | Is used if the subroutine is run several times and is contained in the same block as the call                                                                                                       | L781 P... ;separate block<br>N10 L871 P3 ; three cycles                                      |
| R0 to R299     | Arithmetic parameters                        | $\pm 0.0000001 \dots 9999\ 9999$ (8 decimal places) or specified as an exponent: $\pm (10-300 \dots 10+300)$ |                                                                                                                                                                                                     | R1=7.9431 R2=4<br><br>with specification of an exponent:<br>R1=-1.9876EX9; R1=-1 987 600 000 |
|                | Arithmetic functions                         |                                                                                                              | In addition to the 4 basic arithmetic functions using the operands + - * / , there are the following arithmetic functions:                                                                          |                                                                                              |
| SIN( )         | Sine                                         | Degrees                                                                                                      |                                                                                                                                                                                                     | R1=SIN(17.35)                                                                                |
| COS()          | Cosine                                       | Degrees                                                                                                      |                                                                                                                                                                                                     | R2=COS(R3)                                                                                   |
| TAN()          | Tangent                                      | Degrees                                                                                                      |                                                                                                                                                                                                     | R4=TAN(R5)                                                                                   |
| ASIN()         | Arc sine                                     |                                                                                                              |                                                                                                                                                                                                     | R10=ASIN(0.35) ; R10: 20.487 degrees                                                         |
| ACOS()         | Arc cosine                                   |                                                                                                              |                                                                                                                                                                                                     | R20=ACOS(R2) ; R20: ... Degrees                                                              |
| ATAN2( , )     | Arctangent2                                  |                                                                                                              | The angle of the sum vector is calculated from 2 vectors standing vertically one on another. The 2nd vector specified is always used for angle reference. Result in the range: -180 to +180 degrees | R40=ATAN2(30.5,80.1) ; R40: 20.8455 degrees                                                  |
| SQRT()         | Square root                                  |                                                                                                              |                                                                                                                                                                                                     | R6=SQRT(R7)                                                                                  |
| POT()          | Square                                       |                                                                                                              |                                                                                                                                                                                                     | R12=POT(R13)                                                                                 |
| ABS()          | Absolute value                               |                                                                                                              |                                                                                                                                                                                                     | R8=ABS(R9)                                                                                   |
| TRUNC()        | Truncate to integer                          |                                                                                                              |                                                                                                                                                                                                     | R10=TRUNC(R2)                                                                                |
| LN()           | Natural logarithm                            |                                                                                                              |                                                                                                                                                                                                     | R12=LN(R9)                                                                                   |
| EXP()          | Exponential function                         |                                                                                                              |                                                                                                                                                                                                     | R13=EXP(R1)                                                                                  |
| RET            | Subroutine end                               |                                                                                                              | Used instead of M2 - to maintain the continuous-path mode                                                                                                                                           | RET ;separate block                                                                          |

| Address   | Significance                                                                                | Value assignments                     | Information                                                                                                                                                                                                                   | Programming                                                                                                                           |
|-----------|---------------------------------------------------------------------------------------------|---------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------|
| S...      | Spindle speed                                                                               | 0.001 ... 99<br>999.999               | Unit of measurement of the spindle r.p.m.                                                                                                                                                                                     | S...                                                                                                                                  |
| S1=...    | Spindle speed for spindle 1                                                                 | 0.001 ... 99<br>999.999               | Unit of measurement of the spindle r.p.m.                                                                                                                                                                                     | S1=725 ; speed 725 r.p.m. for spindle 1                                                                                               |
| S         | Cutting rate with G96 active                                                                | 0.001 ... 99<br>999.999               | Cutting rate unit m/min with G96; for spindle only                                                                                                                                                                            | G96<br>S...                                                                                                                           |
| S         | Dwell time in block with G4                                                                 | 0.001 ... 99<br>999.999               | Dwell time in spindle revolutions                                                                                                                                                                                             | G4 S... ;separate block                                                                                                               |
| T         | Tool number                                                                                 | 1 ... 32 000<br>only integer, no sign | The tool change can be performed either directly using the T command or only with M6. This can be set in the machine data.                                                                                                    | T...                                                                                                                                  |
| X         | Axis                                                                                        | ±0.001 ... 99<br>999.999              | Positional data                                                                                                                                                                                                               | X...                                                                                                                                  |
| Y         | Axis                                                                                        | ±0.001 ... 99<br>999.999              | Positional data                                                                                                                                                                                                               | Y...                                                                                                                                  |
| Z         | Axis                                                                                        | ±0.001 ... 99<br>999.999              | Positional data                                                                                                                                                                                                               | Z...                                                                                                                                  |
| AC        | Absolute coordinate                                                                         | -                                     | The dimension can be specified for the end or center point of a certain axis, irrespective of G91.                                                                                                                            | N10 G91 X10 Z=AC(20)<br>;X -incremental dimension,<br>Z - absolute dimension                                                          |
| ACC[axis] | Percentage acceleration override                                                            | 1 ... 200, integer                    | Acceleration override for an axis or spindle; specified as a percentage                                                                                                                                                       | N10 ACC[X]=80 ;for the X axis 80%<br>N20 ACC[S]=50;for the spindle: 50%                                                               |
| ACP       | Absolute coordinate; approach position in the positive direction (for rotary axis, spindle) | -                                     | It is also possible to specify the dimensions for the end point of a rotary axis with ACP(...) irrespective of G90/G91; also applies to spindle positioning                                                                   | N10 A=ACP(45.3)<br>;approach absolute position of the A axis in the positive direction<br>N20 SPOS=ACP(33.1)<br>;position spindle     |
| ACN       | Absolute coordinate; approach position in the negative direction (for rotary axis, spindle) | -                                     | It is also possible to specify the dimensions for the end point of a rotary axis with ACN(...) irrespective of G90/G91; also applies to spindle positioning                                                                   | N10 A=ACN(45.3)<br>;approach absolute position of the A axis in the negative direction<br>N20 SPOS=ACN(33.1)<br>;position spindle     |
| ANG       | Angle for the specification of a straight line for the contour definition                   | ±0.00001 ...<br>359.99999             | Specified in degrees; one possibility of specifying a straight line when using G0 or G1 if only one end-point coordinate of the plane is known or if the complete end point is known with contour ranging over several blocks | N10 G1 X... Z....<br>N11 X... ANG=...<br>or contour over several blocks:<br>N10 G1 X... Z....<br>N11 ANG=...<br>N12 X... Z... ANG=... |
| AR        | Aperture angle for circular interpolation                                                   | 0.00001 ...<br>359.99999              | Specified in degrees; one possibility of defining the circle when using G2/G3                                                                                                                                                 | See G2, G3                                                                                                                            |
| CALL      | Indirect cycle call                                                                         | -                                     | Special form of the cycle call; no parameter transfer; the name of the cycle is stored in a variable; only intended for cycle-internal use                                                                                    | N10 CALL VARNAME ;<br>variable name                                                                                                   |
| CHF       | Chamfer; general use                                                                        | 0.001 ... 99<br>999.999               | Inserts a chamfer of the specified chamfer length between two contour blocks                                                                                                                                                  | N10 X... Z.... CHF=...<br>N11 X... Z....                                                                                              |

| Address  | Significance                                                                       | Value assignments                                                                            | Information                                                                                                                                                                                         | Programming                                                                                                         |
|----------|------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------|
| CHR      | Chamfer;<br>in the contour definition                                              | 0.001 ... 99<br>999.999                                                                      | Inserts a chamfer of the specified leg length between two contour blocks                                                                                                                            | N10 X... Z... CHR=...<br>N11 X... Z...                                                                              |
| CR       | Radius for circular interpolation                                                  | 0.010 ... 99<br>999.999<br>Negative sign - for selecting the circle: greater than semicircle | One possibility of defining a circle when using G2/G3                                                                                                                                               | See G2, G3                                                                                                          |
| CYCLE... | Machining cycle                                                                    | Only specified values                                                                        | The call of the machining cycles requires a separate block; the appropriate transfer parameters must be loaded with values. Special cycle calls are also possible with an additional MCALL or CALL. |                                                                                                                     |
| CYCLE81  | Drilling, centering                                                                |                                                                                              |                                                                                                                                                                                                     | N5 RTP=110 RFP=100<br>.... ; Assign with values<br>N10 CYCLE81(RTP,<br>RFP, ...); separate part<br>program block    |
| CYCLE82  | Drilling, counterboring                                                            |                                                                                              |                                                                                                                                                                                                     | N5 RTP=110 RFP=100<br>.... ;assign with values<br>N10 CYCLE82(RTP,<br>RFP, ...) ;separate block                     |
| CYCLE83  | Deep-hole drilling                                                                 |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE83(110, 100,<br>...) ;or transfer values<br>directly, separate block                                       |
| CYCLE84  | Rigid tapping                                                                      |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE84 (...)<br>;separate block                                                                                |
| CYCLE840 | Tapping with compensating chuck                                                    |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE840 (...)<br>;separate block                                                                               |
| CYCLE85  | Reaming 1                                                                          |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE85(...)<br>;separate block                                                                                 |
| CYCLE86  | Boring                                                                             |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE86(...)<br>;separate block                                                                                 |
| CYCLE92  | Cut-off                                                                            |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE92(...)<br>;separate block                                                                                 |
| CYCLE93  | Recess                                                                             |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE93(...)<br>;separate block                                                                                 |
| CYCLE94  | Undercut DIN76 (forms E and F), finishing                                          |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE94(...)<br>;separate block                                                                                 |
| CYCLE95  | Stock removal with relief cutting                                                  |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE95(...)<br>;separate block                                                                                 |
| CYCLE96  | Thread undercut                                                                    |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE96(...); sepa-<br>rate part program block                                                                  |
| CYCLE98  | Side-by-side thread mounting                                                       |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE98(...); sepa-<br>rate part program block                                                                  |
| CYCLE99  | Thread cutting                                                                     |                                                                                              |                                                                                                                                                                                                     | N10 CYCLE99(...)<br>;separate block                                                                                 |
| DC       | Absolute coordinate;<br>approach position<br>directly(for rotary axis,<br>spindle) | -                                                                                            | It is also possible to specify the dimensions for the end point of a rotary axis with DC(...) irrespective of G90/G91; also applies to spindle positioning                                          | N10 A=DC(45.3)<br>;Approach absolute posi-<br>tion of the A axis directly<br>N20 SPOS=DC(33.1);<br>Position spindle |

| Address | Significance                                      | Value assignments        | Information                                                                                                                                                                                                                                                                                                                                                                   | Programming                                                                     |
|---------|---------------------------------------------------|--------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------|
| DEF     | Definition instruction                            |                          | Defining a local user variable of the type BOOL, CHAR, INT, REAL, directly at the beginning of the program                                                                                                                                                                                                                                                                    | DEF INT VARI1=24, VARI2 ; 2 variables of the type INT ; name defined by user    |
| DITS    | Run-in path with thread G33                       | -1 ... < 0,<br>0,<br>> 0 | Starting with configured axis acceleration;<br>starting with sudden acceleration;<br>run-in path specified, if necessary with axis overload                                                                                                                                                                                                                                   | N10 G33 Z50 K5 DITS=4                                                           |
| DITE    | Run-out path with thread G33                      | -1 ... < 0,<br>0,<br>> 0 | Braking with configured axis acceleration.<br>Braking with sudden acceleration,<br>specification of run-out path, with rounding                                                                                                                                                                                                                                               | N10 G33 Z50 K5 DITE=4                                                           |
| FRC     | Non-modal feedrate for chamfer/rounding           | 0, >0                    | When FRC=0, feedrate F will act                                                                                                                                                                                                                                                                                                                                               | For the unit, see F and G94, G95;<br>for chamfer/rounding, see CHF, CHR, RND    |
| FRCM    | Modal feedrate for chamfer/rounding               | 0, >0                    | When FRCM=0, feedrate F will act                                                                                                                                                                                                                                                                                                                                              | For the unit, see F and G94, G95;<br>for rounding/modal rounding, see RND, RNDM |
| GOTOB   | GoBack instruction                                | -                        | A GoTo operation is performed to a block marked by a label; the jump destination is in the direction of the program start.                                                                                                                                                                                                                                                    | N10 LABEL1: ...<br>...<br>N100 GOTOB LABEL1                                     |
| GOTOF   | GoForward instruction                             | -                        | A GoTo operation is performed to a block marked by a label; the jump destination is in the direction of the end of the program.                                                                                                                                                                                                                                               | N10 GOTOF LABEL2<br>...<br>N130 LABEL2: ...                                     |
| IC      | Coordinate specified using incremental dimensions |                          | The dimension can be specified for the end or center point of a certain axis irrespective of G90.                                                                                                                                                                                                                                                                             | N10 G90 X10 Z=IC(20) ;Z -incremental dimension, X - absolute dimension          |
| IF      | Jump condition                                    | -                        | If the jump condition is fulfilled, the GoTo operation to the block with the following <i>label is performed</i> ; , otherwise, the next instruction/block will follow. In one block, several IF instructions are possible.<br><br>Relational operators:<br>= = equal, <> not equal<br>> greater than, < less than<br>>= greater than or equal to<br><= less than or equal to | N10 IF R1>5 GOTOF LABEL3<br>...<br>N80 LABEL3: ...                              |
| LIMS    | Upper limit speed of the spindle with G96, G97    | 0.001 ... 99<br>999.999  | Limits the spindle speed with the G96 function enabled - constant cutting rate and G97                                                                                                                                                                                                                                                                                        | See G96                                                                         |
| MEAS    | Measurement with deletion of distance-to-go       | +1<br>-1                 | =+1: Measuring input 1, rising edge<br>=-1: Measuring input1, falling edge                                                                                                                                                                                                                                                                                                    | N10 MEAS=-1 G1 X... Z... F...                                                   |
| MEAW    | Measurement without deletion of distance-to-go    | +1<br>-1                 | =+1: Measuring input 1, rising edge<br>=-1: Measuring input1, falling edge                                                                                                                                                                                                                                                                                                    | N10 MEAW=1 G1 X... Z... F...                                                    |

| Address                                              | Significance                                                                                                              | Value assignments                                                               | Information                                                                                                                                                                                                                                     | Programming                                                                                                                                      |
|------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------|
| \$A_DBB[n]<br>\$A_DBW[n]<br>\$A_DBD[n]<br>\$A_DBR[n] | Data byte<br>Data word<br>Data double-word<br>Real data                                                                   |                                                                                 | Reading and writing PLC variables                                                                                                                                                                                                               | N10 \$A_DBR[5]=16.3 ;<br>Write Real variables<br>; with offset position 5<br>; (position, type and<br>meaning are agreed between NC and PLC<br>) |
| \$AA_MM[<br>axis]                                    | Measurement result for an axis in the machine coordinate system                                                           | -                                                                               | Axis: Identifier of an axis (X, Z) traversing when measuring                                                                                                                                                                                    | N10 R1=\$AA_MM[X]                                                                                                                                |
| \$AA_MW[axis]                                        | Measurement result for an axis in the work-piece coordinate system                                                        | -                                                                               | Axis: Identifier of an axis (X, Z) traversing when measuring                                                                                                                                                                                    | N10 R2=\$AA_MW[X]                                                                                                                                |
| \$AC_MEA[1]                                          | Measuring task status                                                                                                     | -                                                                               | Default condition:<br>0: Default condition, probe did not switch<br>1: Probe switched                                                                                                                                                           | N10 IF \$AC_MEAS[1]==1<br>GOTOF .... ; Continue program when probe has switched ...                                                              |
| \$A.....<br>TIME                                     | Timer for runtime:<br>\$AN_SETUP_TIME<br>\$AN_POWERON_TIME<br>\$AC_OPERATING_TIME<br>\$AC_CYCLE_TIME<br>\$AC_CUTTING_TIME | 0.0 ... 10+300<br>min (value read-only)<br>min (value read-only)<br>s<br>s<br>s | System variable:<br>Time since the control system has last booted<br>Time since the control system has last booted normally<br>Total runtime of all NC programs<br>Runtime of the NC program (only of the selected program)<br>Tool action time | N10 IF<br>\$AC_CYCLE_TIME==50.5 ....                                                                                                             |
| \$AC.....<br>PARTS                                   | Workpiece counter:<br>\$AC_TOTAL_PARTS<br>\$AC_REQUIRED_PARTS<br>\$AC_ACTUAL_PARTS<br>\$AC_SPECIAL_PARTS                  | 0 ... 999 999 999, integer                                                      | System variable:<br>Total actual count<br>Set number of workpiece<br><br>Current actual count<br>Count of workpieces - specified by the user                                                                                                    | N10 IF<br>\$AC_ACTUAL_PARTS==15 ....                                                                                                             |
| \$AC_MSNUM                                           | Number of active spindles                                                                                                 |                                                                                 | read-only                                                                                                                                                                                                                                       |                                                                                                                                                  |
| \$P_MSNUM                                            | Number of programmed spindles                                                                                             |                                                                                 | read-only                                                                                                                                                                                                                                       |                                                                                                                                                  |
| \$P_NUM_SPINDLES                                     | Number of configured spindles                                                                                             |                                                                                 | read-only                                                                                                                                                                                                                                       |                                                                                                                                                  |
| \$AA_S[n]                                            | Actual speed of spindle n                                                                                                 |                                                                                 | Spindle number n =1<br>read-only                                                                                                                                                                                                                |                                                                                                                                                  |
| \$P_S[n]                                             | Last programmed speed of spindle n                                                                                        |                                                                                 | Spindle number n =1<br>read-only                                                                                                                                                                                                                |                                                                                                                                                  |
| \$AC_SDIR[n]                                         | Current direction of rotation of spindle n                                                                                |                                                                                 | Spindle number n =1<br>read-only                                                                                                                                                                                                                |                                                                                                                                                  |
| \$P_SDIR[n]                                          | Last programmed direction of rotation of spindle n                                                                        |                                                                                 | Spindle number n =1<br>read-only                                                                                                                                                                                                                |                                                                                                                                                  |
| \$P_TOOLNO                                           | Number of the active tool T                                                                                               | -                                                                               | read-only                                                                                                                                                                                                                                       | N10 IF \$P_TOOLNO==12<br>GOTOF ....                                                                                                              |
| \$P_TOOL                                             | Active D number of the active tool                                                                                        | -                                                                               | read-only                                                                                                                                                                                                                                       | N10 IF \$P_TOOL==1<br>GOTOF ....                                                                                                                 |

| Address             | Significance                                           | Value assignments                | Information                                                                                                                                                                                                                                         | Programming                                                                                                         |
|---------------------|--------------------------------------------------------|----------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------|
| MSG ( )             | Signal                                                 | max. 65 characters               | Message text in inverted commas                                                                                                                                                                                                                     | MSG("MESSAGE TEXT") ; separate block<br>...<br>N150 MSG() ; Clear previous message                                  |
| OFFN                | Dimension specification                                | -                                | Only effective with the tool radius compensation G41, G42 active                                                                                                                                                                                    | N10 OFFN=12.4                                                                                                       |
| RND                 | Rounding                                               | 0.010 ... 99<br>999.999          | Inserts a rounding with the specified radius value tangentially between two contour blocks                                                                                                                                                          | N10 X... Z... RND=...<br>N11 X... Z...                                                                              |
| RNDM                | Modal rounding                                         | 0.010 ... 99<br>999.999<br><br>0 | - Inserts roundings with the specified radius value tangentially at the following contour corners; special feedrate possible: FRCM= ...<br>- Modal rounding OFF                                                                                     | N10 X... Y... RNDM=.7.3 ;modal rounding ON<br>N11 X... Y...<br>...<br>N100 RNDM=.0 ;modal rounding OFF              |
| RPL                 | Angle of rotation with ROT, AROT                       | ±0.00001 ...<br>359.9999         | Specification in degrees; angle for a programmable rotation in the current plane G17 to G19                                                                                                                                                         | See ROT, AROT                                                                                                       |
| SET( , , )<br>REP() | Set values for the variable fields                     |                                  | SET: Various values, from the specified element up to: according to the number of values<br>REP: the same value, from the specified element up to the end of the field                                                                              | DEF REAL<br>VAR2[12]=REP(4.5) ; all elements value 4.5<br>N10<br>R10=SET(1.1,2.3,4.4) ;<br>R10=1.1, R11=2.3, R4=4.4 |
| SETMS(n)            | Define spindle as master spindle                       | n= 1 or n= 2                     | n: Number of the spindle, if only SETMS is set, the default master spindle comes into effect                                                                                                                                                        | N10 SETMS(2) ; separate block, 2nd spindle = master                                                                 |
| SF                  | Thread starting point when using G33                   | 0.001 ... 359.999                | Specified in degrees; the thread starting point with G33 will be offset by the specified value                                                                                                                                                      | See G33                                                                                                             |
| SPI(n)              | Converts the spindle number n into the axis identifier |                                  | n =1,<br>axis identifier: e.g. "SP1" or "C"                                                                                                                                                                                                         |                                                                                                                     |
| SPOS<br><br>SPOS(n) | Spindle position                                       | 0.0000 ...<br>359.9999           | specified in degrees; the spindle stops at the specified position (to achieve this, the spindle must provide the appropriate technical prerequisites: position control)<br>Spindle number n: 1                                                      | N10 SPOS=...<br>N10 SPOS=ACP(...)<br>N10 SPOS=ACN(...)<br>N10 SPOS=IC(...)<br>N10 SPOS=DC(...)                      |
| SPOSA               | Spindle position                                       | 0.0000 ...<br>359.9999           | SPOS and SPOSA have the same functionality but differ in their block change behavior:<br>With SPOS, the NC block is only enabled once the position has been reached.<br>With SPOSA, the block is enabled even if the position has not been reached. | SPOSA=<value> /<br>SPOSA [<n>] = <value>/                                                                           |
| STOPFIFO            | Stops the fast machining step                          |                                  | Special function; filling of the buffer memory until STARTFIFO, "Buffer memory full" or "End of program" is detected.                                                                                                                               | STOPFIFO; separate block, start of filling<br>N10 X...<br>N20 X...                                                  |

| Address   | Significance                      | Value assignments | Information                                                                               | Programming                                                                                 |
|-----------|-----------------------------------|-------------------|-------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------|
| STARTFIFO | Start of fast machining step      |                   | Special function; the buffer memory is filled at the same time.                           | N30 X...<br>STARTFIFO ;separate block, end of filling                                       |
| STOPRE    | Preprocessing stop                |                   | Special function; the next block is only decoded if the block before STOPRE is completed. | STOPRE ; separate block                                                                     |
| TRACYL    | Milling of the peripheral surface |                   | Kinematic transformation (available only if configured accordingly)                       | TRACYL(20.4) ; separate block; cylinder diameter: 20.4 mm<br>TRACYL(20.4,1) ; also possible |
| TRANSMIT  | Milling of the face end           |                   | Kinematic transformation (available only if configured accordingly)                       | TRANSMIT ; separate block<br>TRANSMIT(1) ; also possible                                    |
| TRAFOOF   | Disable TRANSMIT, TRACYL          |                   | Disables all the kinematic transformations                                                | TRAFOOF ; separate block                                                                    |

---

## Trademarks

All names identified by ® are registered trademarks of Siemens AG. The remaining trademarks in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owner.

## Disclaimer of Liability

We have reviewed the contents of this publication to ensure consistency with the hardware and software described. Since variance cannot be precluded entirely, we cannot guarantee full consistency. However, the information in this publication is reviewed regularly and any necessary corrections are included in subsequent editions.

Siemens AG  
Division Digital Factory  
Postfach 48 48  
90026 NÜRNBERG  
GERMANY

Programming and Operating Manual (Turning)  
6FC5398-5DP10-0BA5, 06/2017